

# TurboCAD®

Version 18

## User Guide

IMSI/Design  
25 Leveroni Ct. Suite B  
Novato, CA 94945, USA  
Tel: +1-415-483-8000  
Fax: +1-415-884-9023  
Web Site  
[www.imsidesign.com](http://www.imsidesign.com)  
[www.turbocad.com](http://www.turbocad.com)

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# 1 Getting Started

---

This section covers what you should know before and during TurboCAD installation, and fundamental concepts of using files, including import / export and page setup.

## System Requirements

### Minimum System Requirements:

Pentium® IV Processor; Microsoft® Windows® XP with 512 MB RAM; Windows Vista or Windows 7 with 1024 MB RAM; 300 MB of free hard disk space; 64+ MB of swap space; Super VGA (1024 x 768) display; High Color (16 bit) graphics support; 4X DVD-ROM drive.

### Recommended:

Your experience with TurboCAD will be greatly enhanced with a newer generation, higher speed CPU, 2-4 GB RAM, and larger display resolution. An optional GPU-accelerated wireframe render mode requires a supported graphic accelerator. The latest video drivers are typically required. Newer boards with more power and VRAM generally provide greater performance. The following are recommended: NVIDIA® Quadro FX 1000 & up • NVIDIA® GeForce FX & up • NVIDIA® GeForce 6, 7, 8 & up • AMD-ATI Radeon 9500 & up • AMD-ATI Radeon X, HD 2000, 3000, 4000 • AMD-ATI FireGL X • AMD-ATI FireGL V 3000 & up • INTEL® Integrated graphics series: GMA 3000 & GMA X3000.

## Installing TurboCAD

To install TurboCAD, insert the CD into your CD-ROM. If the installation process does not start automatically, select Start / Run from the Windows taskbar and type D:\setup.exe (where D is the drive letter of the CD-ROM).

After you have installed TurboCAD, the **Setup** program creates a program folder. If you choose the default settings, TurboCAD is installed in the C:\Program Files\IMSI Design\TCWP18 folder (or TCW18 for Deluxe). This folder contains several subfolders that contain TurboCAD program files and related files such as templates, sample drawings, and symbols.

The **Setup** program also creates a program group containing the TurboCAD application icon, as well as shortcuts to the Help and the Readme file.

The program group is accessed through the Start menu.

Before you start the program, please read the Readme file, which contains the latest information on TurboCAD.

To remove TurboCAD from your computer:

1. In Windows, select **Start / Settings / Control Panel**.
2. Double-click **Add/Remove Programs**.
3. Select TurboCAD from the list.
4. Click **Add/Remove** and follow the instructions on the screen.

## Registration

If you do not have Internet access, please call 800-833-8082 to register. Becoming a TurboCAD registered user has many advantages, including technical support, access to extensive online support databases, and program updates.

## Importing and Exporting Files

You can import and export data from other TurboCAD formats, as well as formats of other CAD systems.

To import from another file type:

1. Select **File / Open** or Ctrl+O.
2. Under **Files of type**, select the format you want to import.
3. If you want to specify conversion parameters, click **Setup**.
4. Browse to the file, and double-click it or click **Open**.

To export to another file type:

1. Select **File / Save As**.
2. Under **Save as type**, select the format to which you want to export.
3. If you want to specify conversion parameters, click **Setup**.
4. Browse to the folder and assign a file name.

To import or export only certain components of a file, use **File / Extract From** or **Extract To**. For example, you can choose to import a drawing's layers and blocks, or export only print styles.

To insert the contents of another file as a block, create an external reference (xref). See "External References" on page 303.

### Batch Conversion

*Available in TurboCAD Pro and Platinum only*




---

NOTE: Display the **SDK Samples** toolbar by right-clicking in any toolbar area and selecting **SDK Samples**.

---

Opens the **TurboCAD File Converter** window, in which you can select a source and destination folder for batch conversion. Use the **Files of type** field to select the

conversion format. Click **Run** to convert all files in the selected folder, and place the converted files in the target folder.

### TurboCAD Formats

There are three TurboCAD file formats, TCW, TCT, and TCX, that you can use for saving vector graphics from within TurboCAD.

**TCW:** (TurboCAD for Windows) - a file format for saving vector graphics from within TurboCAD.

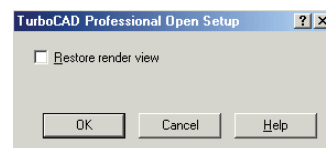
**TCT:** (TurboCAD Template) - a file format for saving TurboCAD drawings as templates. TurboCAD uses a template for starting a new drawing. You can save any drawing as a template, and this drawing can have specific tool properties or styles defined in it. The template file should be placed in the "Template" folder of the TurboCAD root directory. Then when you want to open the template, use **File / New**, and select **New from Template**.

---

NOTE: If you need to save a drawing that is to be opened by an earlier version of TurboCAD, you can use **Save As** and save to one version back i.e. In TurboCAD 15 You can save as TurboCAD 14.

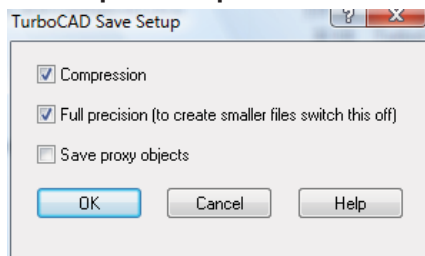
---

### TCW, TCT Import Setup



**Restore render view:** Opens the drawing in render mode. The file must have been saved in render mode.

## TCW, TCT Export Setup



**Compression:** Saves the file in the TCT compressed format. If the speed of operation is more important than the file size, leave this option disabled.

**Full Precision:** TurboCAD stores your objects with the maximum possible accuracy. **Full Precision** enables the maximum depth of mathematical calculations performed to generate and save entities. When disabled, entities will be calculated and saved using only to four decimal points of precision. This results in faster execution and smaller files, but may compromise accuracy.

---

*NOTE: Less than full accuracy can affect the ability of the program to correctly display and edit objects at extremely small scales. Unless you are using a slow machine, and know you do not need extreme accuracy, you should leave **Full Precision** on.*

---

**Save proxy objects:** saves ACIS solid and a number of other TurboCAD Pro object with proxies so that TurboCAD Deluxe or Designer can open the file and see representations of objects that are not supported by those applications. Proxies increase the size of the file, but do allow file to open more quickly.

## Other CAD Formats

The following formats can be read by TurboCAD, either for import, export, or both. Some formats have a **Setup** in which you can set certain parameters for conversion. If you do not specify conversion parameters, the defaults will be used. (If no **Setup** is indicated, the conversion requires no additional input.)

---

*NOTE: Some of these formats are Available in TurboCAD Pro and Platinum only.*

---

**3DS:** Autodesk 3D Studio format. See "SKP File Import Setup" on page 26 or "3DS File Export Setup" on page 27.

**3DV:** VRML Worlds (import only)

**ASAT:** ACIS solid modeling file format for storing graphics as ASCII files. See "SAT Import Setup" on page 33 or "SAT Export Setup" on page 33.

**BMP:** Bitmap format, TurboCAD for Windows (export only). A bitmap is a representation of a graphic image consisting of rows and columns of dots. See "BMP Export Setup" on page 27.

**CGM:** Computer Graphics Metafile. See "CGM Import Setup" on page 27.

**DAE:** Collada files (export only). See "DAE Export Setup" on page 28.

**DC, DCD:** DesignCAD (import only)

**DGN:** Intergraph Standard file format. See "DGN Import Setup" on page 28 or "DGN Export Setup" on page 28.

**DWF:** Drawing Web format. See "DWF Export Setup" on page 28. Import is automatic.

**DWG:** AutoCAD native format for vector graphics. See "DXF (DWG) Import Setup" on page 29 or "DXF (DWG) Export Setup" on page 29.

**DXF:** Drawing eXchange format, used by AutoCAD to interact with other applications.

**EPS:** Encapsulated Post Script format. See "EPS Import Setup" on page 30 or "EPS Export Setup" on page 30

**FCD:** FastCAD DOS format (import only)

**FCW:** FastCAD Windows format (import only). See "FCW Import Setup" on page 30.

**GIF:** Raster graphic format (export only). See "GIF Export Setup" on page 30.

**GEO:** VRML Worlds (import only)

**IGS:** IGES format. See "IGS Import Setup" on page 31.

---

**NOTE:** *IGS is a file format for a public domain called IGES which is actually a neutral file format intended as an international standard for the exchange of data between different CAD/CAM systems. IGES is an ANSI standard maintained by the IGES/PDES Organization (IPO), under the direction of the National Institute of Standards and Technology (NIST). The National Computer Graphics Association (NCGA) acts as the administrator of the standard.*

---

**JPG:** JPEG image compression standard, full 24-color. TurboCAD for Windows (export only). See "JPG Export Setup" on page 31.

**JPG:** JPEG format, SDK sample filter (export only)

**MTX:** MetaStream format. This format enables the creation, delivery, and viewing of scalable 3D graphics. It enables you to view images with Web browsers equipped with 3D plug-in software, and to interact with 3D Web graphics in real time by zooming, panning, or rotating objects within a Web browser. It also scales objects in real time to match the performance of your system. MetaStream compresses 3D geometry and texture information, so that files can be delivered quickly to your machine.

**PDF:** Portable document format (export only). See "PDF Export Setup" on page 32.

**OBJ:** Geometry definition format. See "OBJ Export Setup" on page 31.

**PLT:** Hewlett-Packard Graphics Language. See "PLT Import Setup" on page 32 or "PLT Export Setup" on page 32.

**PNG:** Raster graphic format (export only). See "PNG Export Setup" on page 32.

**SAT:** ACIS solid modeling file format for storing graphics as ASCII (SAT - Save As Text) files. See "SAT Import Setup" on page 33 or "SAT Export Setup" on page 33.

**SHX:** Format that saves selected entities as a single shape, that can be used in a line style (export only). This is not the same as \*.shx font format. See "SHX Export Setup" on page 33.

**SKP:** Google SketchUp format. See "3DS File Import Setup" on page 27, and see "SKP File Export Setup" on page 27

**STL:** Stereo Lithography format. See "STL File Export Setup" on page 33.

**STP, STEP:** Step format.

**SVG:** Web graphic format (export only).

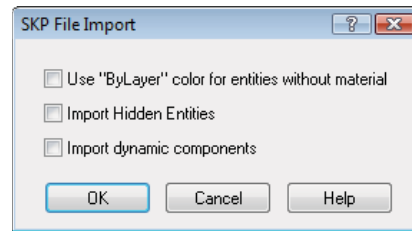
**WMF:** Windows MetaFile format, saves graphics as a series of records that correspond to graphics function calls, such as drawing straight lines, curves, filled areas, and text. "WMF Import Setup" on page 34 or "WMF Export Setup" on page 34

**WRL:** Format for saving graphics as a Virtual Reality Model Language (VRML) description. See "WRL Export Setup" on page 34.

**WRZ:** VRML Worlds

**XLS:** Spreadsheet format (export only)

## SKP File Import Setup

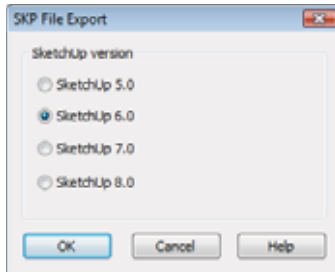


**Use "ByLayer" color for entities without material:** Assign the SketchUp layer color to any entity that does not have an assigned material.

**Import Hidden Entities:** Imports entities which are hidden in the SKP. If this option is off hidden entities will not be imported.

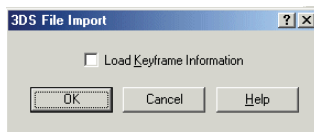
**Import dynamic components:** Dynamic block are imported as PPM objects.

## SKP File Export Setup



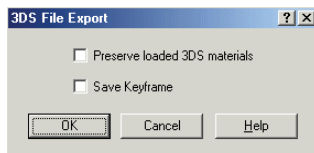
**SketchUp Version:** Specify the version of SketchUp SKP to be used in the file format: 5, 6, 7 or 8.

## 3DS File Import Setup



**Load Keyframe Information:** Accepts or ignores the keyframe 3DS file data.

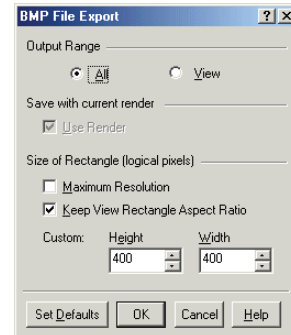
## 3DS File Export Setup



**Preserve loaded 3DS materials:** Saves unique materials from a 3DS file. If not checked, saves TurboCAD materials.

**Save Keyframe:** Saves the hierarchical structure of entities in a file. Do not check this option for 3DS files that you want to use as 3DS symbols in TurboCAD.

## BMP Export Setup



**Output Range:** Creates an export file either out of the entire drawing (**All**) or only the part that is currently in view (**View**).

**Use Render:** Available only when **Render** or **Hidden Line** mode is active. If checked, the exported image will show the render displayed on the screen.

**Size of Rectangle:** Sets up the resolution for the exported graphics. By default, these values are 400 x 400 but you can customize the height and width

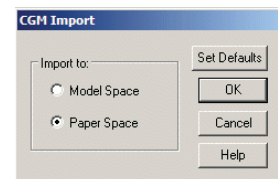
- **Maximum resolution:** The drawing will be defined by a rectangular matrix of 2000 x 2000 logical pixels. If there is at least one image object, this is set to 1000 x 1000.
- **Keep View Rectangle Aspect Ratio:** Keeps the same ratio of length to width.

---

*NOTE: You can determine an object's type by using the Selection Info Palette. See "Selection Info Palette" on page 191.*

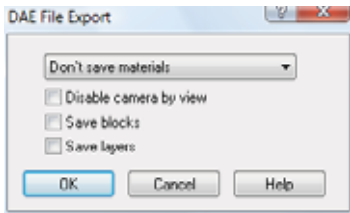
---

## CGM Import Setup



Choose to open the file either in **Model Space** or **Paper Space**.

DAE Export Setup



**Don't save materials:** Objects colors will remain the same.

**Save materials:** For objects that have textures, the imported objects will refer to texture graphics in the TurboCAD folders.

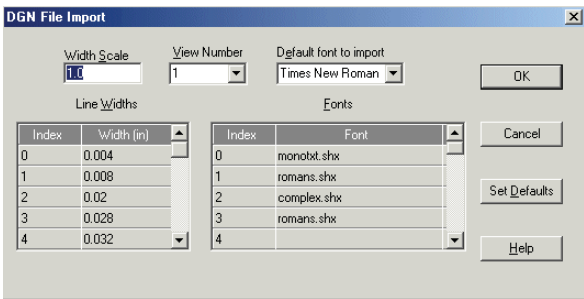
**Save materials and copy textures:** Creates a local copy of texture files into the folder where the exported file is saved.

**Disable camera by view:** Specifies whether a camera is added to the model (the camera is based on the current view).

**Save Blocks:** Block are saved as instances within the Collada file.

**Save Layers::** Layers are saved in the Collada file.

DGN Import Setup



**View Number:** By default, the view number is 1.

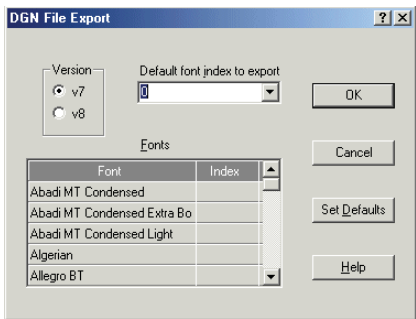
**Line Widths:** You customize a set of drawing line widths identified by the indexes (0, 1, 2, etc.). To do this, use the two-column **Line Width** control ,where the **Index** column provides a reference to a particular line width, and the **Width** column is a data entry field. You can scale the line width values, using the **Width Scale** data entry field.

**Fonts:** As with the drawing line widths, the import file's fonts are indexed. For every index in the **Index** column, you can select a font from the corresponding drop-down list in

the **Font** column. In addition to the options of the **Fonts** control, there is the **Default Font** entry field whose drop-down list lets you select a common font for the ones you haven't changed.

**Set Defaults:** Instead of customizing the imported file options, you can apply the default settings for the DGN file import. To do this, click **Set Defaults**.

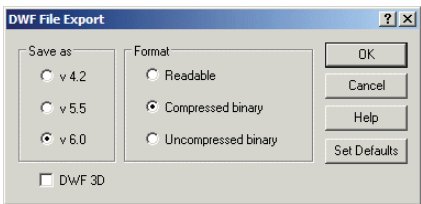
DGN Export Setup



**Version:** Choose between Versions 7 and 8.

**Default font index to export:** The DGN format requires the export file's fonts to be indexed. For every listed font, you can select an index. In addition to the options of the **Fonts** control, there is the **Default Index** entry field whose drop-down list lets you select a common index for the fonts you have not changed.

DWF Export Setup



**Output Format:** If **Readable** is selected, you can later open the exported file to view and/or edit via a text editor.

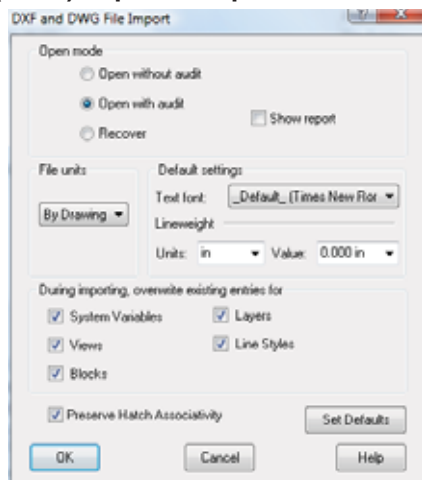
**Skip Invisible Layers:** Invisible layers will not be included in the export file. See "Layers" on page 116.

**DWF 3D:** DWF files of 6.01 format will be created. Files will contain all elements from Model Space, including 3D.



NOTE: TurboCAD does not support importing from this format. DWF files created with this option can be viewed either in Autodesk DWF Viewer (v6.01 and up), or in other applications that support this format.

## DXF (DWG) Import Setup



### Open mode:

- **Open without audit:** Opens the file without checking it. If the file has errors, it will not be opened.
- **Open with audit:** Opens the file and corrects errors when possible. The report is displayed only if errors are found. The file is checked is after it is loaded into memory, though if the file is very corrupt it may not be able to be loaded into memory.
- **Recover:** Opens the file and corrects errors. The report is automatically displayed, regardless of errors. The file is checked before loading it into memory.

**File Units:** Replaces the units of an imported file with the selected units.

**Default text font:** Sets a default import font.

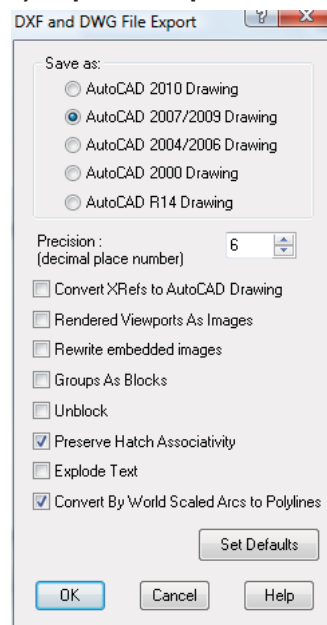
**Lineweight Unit and Value:** Defines how objects using the AutoCAD default line weight are treated on import.

**Overwrite existing entries for:** AutoCAD stores the settings (or values) for its operating environment and some of its commands in system variables. Checking this box

overwrites these settings. In TurboCAD, these settings include World units, the numerical display format, the angular system, etc.

You can overwrite system variables for objects under the following conditions: the DXF/DWG drawing is inserted into an open file, and both files (the imported file and the receiving file), have system variables (views, blocks, layers and/or line styles) of the same names.

## DXF (DWG) Export Setup



**Save As:** Select the AutoCAD version.

**Precision:** The number of decimal places for numerical values of the exported file.

**Convert Xrefs to AutoCAD Drawing:** Converts attached Xrefs into DWG files.

**Rendered Viewports as Images:** All viewports that have renderings will be converted to images, to preserve the appearance of the renderings.

**Rewrite embedded images:** Relevant for files that contain image objects. Choose whether to replace images.

**Groups as Blocks:** Displays objects included in groups as blocks when imported into AutoCAD.

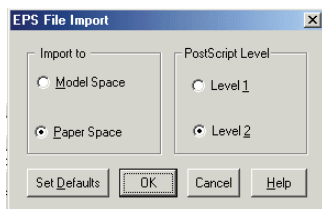
**Unblock:** Explodes blocks into its constituent objects.

**Preserve Hatch Associativity:** Retains hatch association in the resulting DWG file.

**Explode Text:** Explodes any text into its constituent parts.

**Convert By World Scaled Arcs into Polylines:** All arc using the By World setting for line scaling will be converted into arc segments of polylines with a supported AutoCAD scaling method.

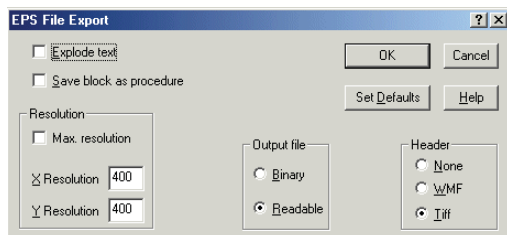
## EPS Import Setup



**Import to:** Choose whether the imported file will open in Model Space or Paper Space.

**PostScript Level:** Level 2 is a more sophisticated version of PostScript. It provides color extensions, support for fonts with many characters, filters (for compression, etc.) and improved image handling (inclusion of JPEG files, etc.).

## EPS Export Setup



**Explode Text:** Explodes any text into its constituent parts.

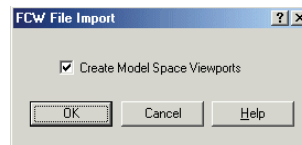
**Save block as procedure:** Saves the drawing's blocks as procedures in the PostScript language.

**Resolution:** Sets the horizontal and vertical resolutions for the two-dimensional presentation of your drawing. Check **Max Resolution** to set the maximum 1600x1600 resolution.

**Output file:** Exports to a binary or readable format.

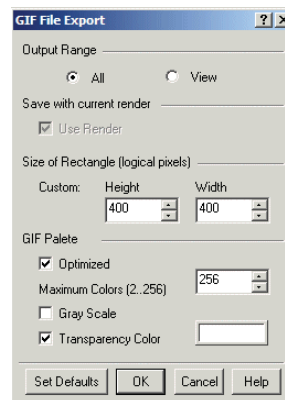
**Header:** Specify whether to include a preview image, and in what format.

## FCW Import Setup



**Create Model Space Viewports:** Specify whether to create viewports for the Model Space presentation of your drawing.

## GIF Export Setup



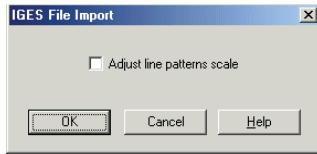
**Optimized:** Color palette optimization. If not checked, the initial color palette will imitate the Halftone palette, in which the color of a pixel is adjusted by mixing colors of adjacent pixels. If checked, the initial color palette will be optimized, containing no more colors than the number specified in **Maximum Colors** field.

**Maximum Colors:** Number of colors in the optimized palette (from 2 to 256).

**Grayscale:** The initial color palette will contain shades of gray.

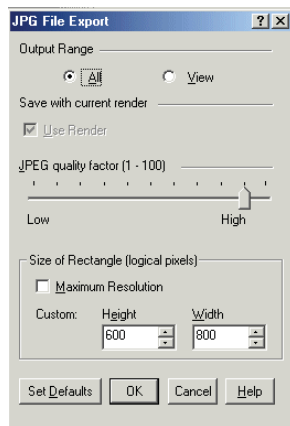
**Transparency Color:** Sets the color that will be transparent. Available for optimized palette.

## IGS Import Setup



**Adjust line pattern scale:** Line style for improving line display. Because IGS files can be created in a variety of ways, either option (on or off) may work to improve precision during import.

## JPG Export Setup



**Output Range:** Creates an export file either out of the entire drawing (**All**) or only the part that is currently in view (**View**).

**Use Render:** Available only when **Render** or **Hidden Line** mode is active. If checked, the exported image will show the render displayed on the screen.

**JPEG quality factor:** Your goal is generally to pick the lowest quality setting, or smallest file size, that decompresses into an image indistinguishable from the original. This setting can vary, but here are some rules of thumb. For good-quality, full-color source images, the default quality setting (Q 75) is very often the best choice. This setting is about the lowest you can go without expecting to see defects in a typical image. Try Q 75 first; if you see defects, then increase. If the image was less than perfect quality to begin with, you might be able to drop down to Q

50 without objectionable degradation. On the other hand, you might need a higher quality setting to avoid further loss. This is often necessary if the image contains dithering or moire patterns. Except for experimental purposes, avoid values above Q 95; Q 100 will produce a file two or three times as large as Q 95, but with negligible quality improvement.

**Size of Rectangle:** Sets the resolution for the exported graphics. By default, these values are 600 x 800 but you can customize the height and width

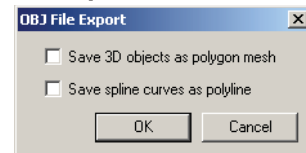
- **Maximum resolution:** The drawing will be defined by a rectangular matrix of 2000 x 2000 logical pixels. If there is at least one image object, this is set to 1000 x 1000.

---

NOTE: You can determine an object's type by using the *Selection Info Palette*. See "Selection Info Palette" on page 191.

---

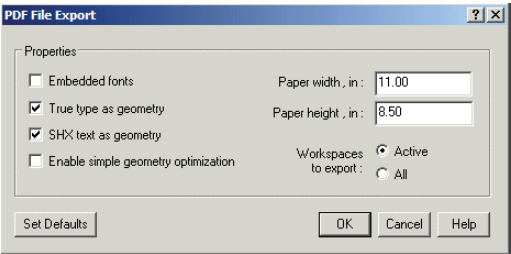
## OBJ Export Setup



**Save 3D objects as polygon mesh:** When 3D objects are converted into a mesh, smoothness may be lost, but exact appearance will be preserved.

**Save spline curves as polyline:** When NURBS curves are converted to polylines, smoothness will be lost, but exact appearance will be preserved.

PDF Export Setup



**Embedded fonts:** Fonts will be embedded in the PDF file, increasing the file size. If not checked, the PDF viewer (such as Adobe Reader) will select a font that exists in the system.

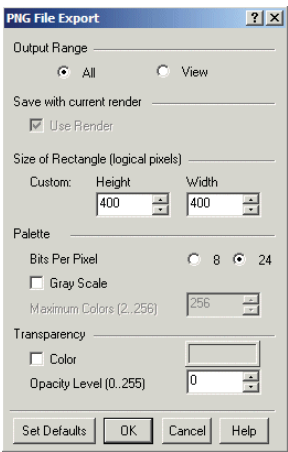
**True type as geometry:** True-type text will be exploded into a collection of lines.

**SHX text as geometry:** SHX text will be exploded into a collection of lines.

**Enable simple geometry optimization:** When checked, smaller PDF files will be created. This may affect the PDF quality.

**Workspaces to Export:** Select Active to export only the active Space, or All to export all Paper Spaces and Model Spaces.

PNG Export Setup



**Bits Per Pixel:** The number of bits per pixel in the saved image. A value of 8 means a 256-color palette; a value of 24 means True Color palette.

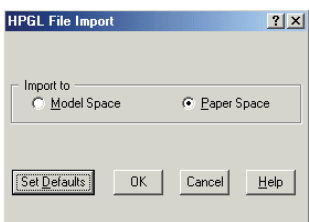
**Gray Scale:** The initial color palette will contains shades of gray.

**Maximum Colors:** Number of colors in the palette (from 2 to 256). Available if **Bits Per Pixel** = 8.

**Color:** Selection of transparency color. When **Bits Per Pixel** = 8, the pixels that have this color will be transparent. When **Bits Per Pixel** = 24, the pixels that have this color will get the level of opacity specified in the **Opacity Level** field. If **Color** is off, the level of opacity will be set for all pixels of an image.

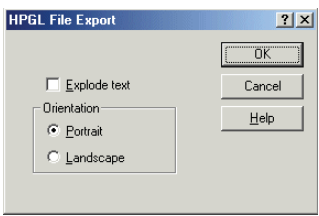
**Opacity Level:** Level of opacity (alpha channel: from 0 to 255). Available if **Bits Per Pixel** = 24.

PLT Import Setup



**Import to:** Choose whether to open the imported file in Model or Paper Space.

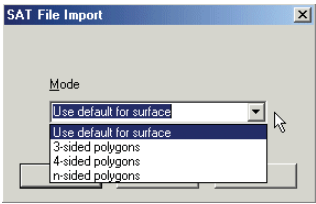
PLT Export Setup



**Explode text:** Explodes any text into its constituent parts.

**Orientation:** Sets the paper orientation to **Portrait** (vertical) or **Landscape** (horizontal).

### SAT Import Setup



**Mode:** Choose one of the modes for representing SAT objects.

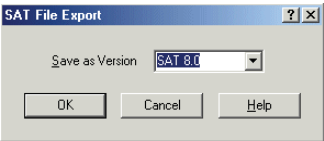
**Default:** The choice of the polygon type is determined by the surface type in the solid object model (planar, cylindrical, spherical, toroidal, etc.)

**3, 4, n-sided polygons:** The number of sides of polygons used for object representation.

If you wish to check what type of polygon is used for the object presentation in an imported SAT file, do the following.

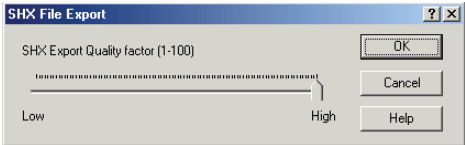
1. Once the SAT file has been imported, select an object in the drawing, make a block of it, and bring this block onto the paper of the TurboCAD desktop.
2. Open the **Properties** window for this block, open the **Block Insertion** page, and assign non-zero values to the rotation coordinates.
3. When a 3D view of your object is displayed, select the object and explode it.
4. After this you can display individual elements (polygons) of the object by selecting the different parts of objects.

### SAT Export Setup



Select the SAT version for export.

### SHX Export Setup



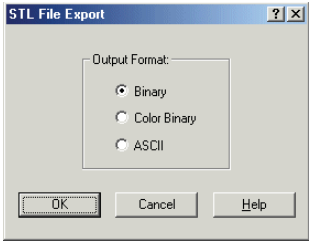
**SHX Export Quality Factor:** A high value creates shapes that more accurately represent the selected geometry. However, large values tend to create shapes that contain more coordinate data and can be a processing burden during regeneration. Low values create less accurate shapes that are more quickly processed. You may need to experiment with different values.

---

*NOTE: SHX export does not create an \*.shx font file. Once created, the results cannot be edited.*

---

### STL File Export Setup

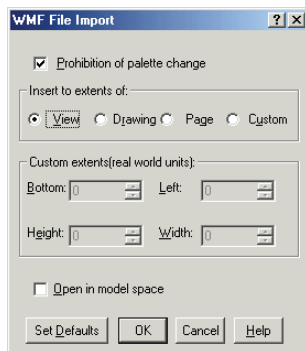


**Binary:** Saves to a binary file.

**Color Binary:** Saves to a color binary file.

**ASCII:** Saves to an ASCII text file.

## WMF Import Setup



**Prohibition of palette change:** Prevents TurboCAD from changing the color palette.

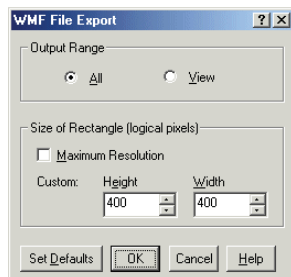
**Insert to extents of:** Sets the size of the imported drawing.

- **View:** The size is based on the current view.
- **Drawing:** The size is based on the entire drawing.
- **Page:** The size is based on the paper sheet.
- **Custom:** Define your own size.

**Custom extents:** Enter coordinates and dimensions of the rectangle that encloses the drawing.

**Open in Model Space:** Opens the imported drawing in Model Space. The drawing is placed on the default **By World** workplane.

## WMF Export Setup



**Output Range:** Creates an export file either out of the entire drawing (**All**) or only the part that is currently in view (**View**).

**Size of Rectangle:** Sets the resolution for the exported graphics. By default, these values are 400 x 400 but you can customize the height and width

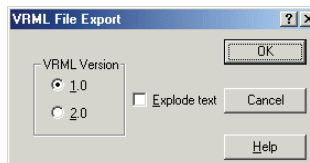
- **Maximum resolution:** The drawing will be defined by a rectangular matrix of 8000 x 8000 logical pixels. If there is at least one image object, this is set to 1000 x 1000.

---

*NOTE: You can determine an object's type by using the Selection Info Palette. See "Selection Info Palette" on page 191.*

---

## WRL Export Setup



Select a version of VRML file to save. You can also explode the drawing text while creating the file for export.

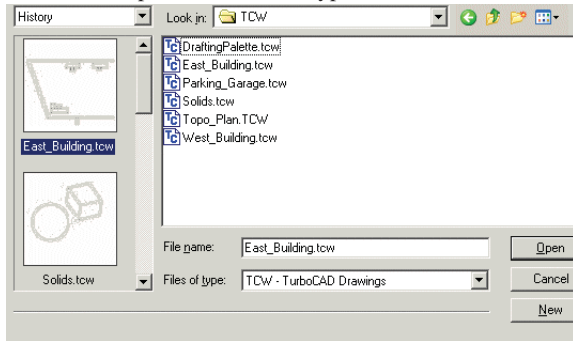
## Starting TurboCAD

If you accepted the default installation settings, TurboCAD can be accessed via the Windows Start / Programs menu.

1. Click the **Start** button on the Windows taskbar.
2. Select **Programs / IMSI Design**, and select the version you have installed.

You can also start TurboCAD by double-clicking the desktop icon.

In the opening screen, you can view a **History** of drawings previously opened, browse to folders containing drawings you want to open, or open a **Template**. You can also use this window to import files of other types.



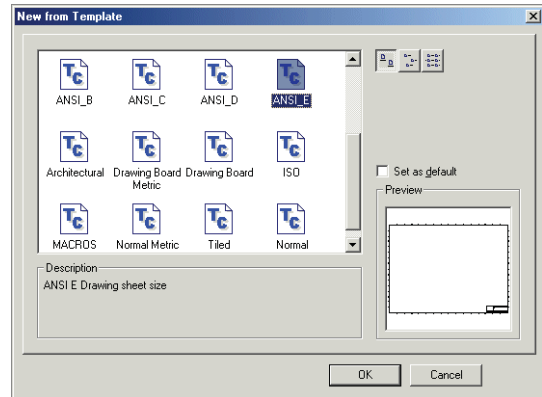
If you want to create a new drawing, click **New**.

You can also close this window by clicking **Cancel**. You can then select **File / New** within TurboCAD to create a new drawing. The **New TurboCAD Drawing** window appears, in which you specify what type of file you want to open.



**New from Scratch:** Creates a new blank drawing with the default settings.

**New from Template:** Enables you to select a predefined template, which has size, units, and layout defined.



**Page Setup Wizard:** Opens the step-by-step guide for setting page size and scale.

**Open Drawing:** Opens an existing drawing. Browse to the desired folder to locate the file. For acceptable file types, see "Importing and Exporting Files" on page 24.

**Open Last Edited:** Opens the last file you worked on.

### Page Setup Wizard

A step-by-step guide to setting page size and drawing scale.

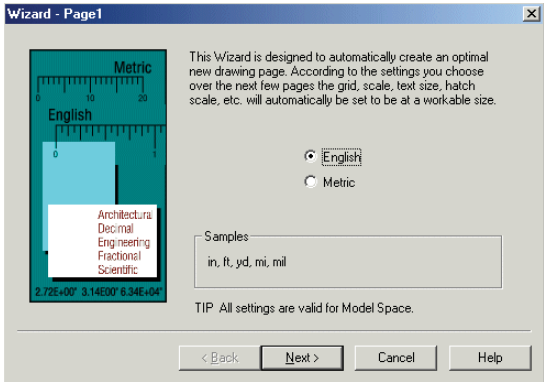
---

*NOTE: This is not exactly the same as the **Page Setup** used for printing, but any relevant parameters you set in this wizard will appear in the **Page Setup**. See "Page Setup" on page 664.*

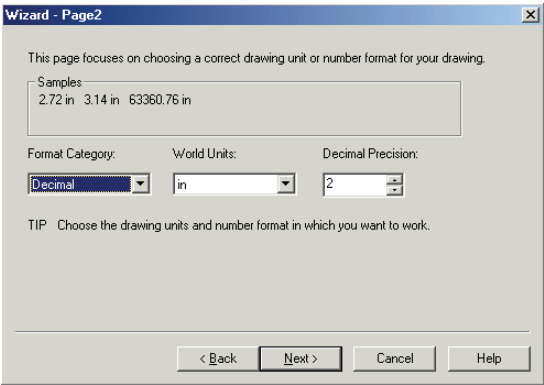
---

If you selected **Page Setup Wizard** in the **New TurboCAD Drawing** window, the wizard opens automatically. You can also access it via **File / Page Setup Wizard**.

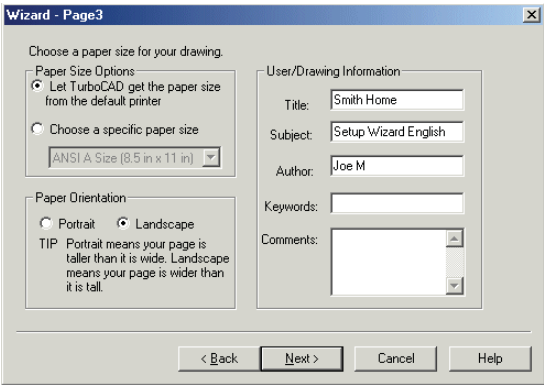
In the first page you can choose between **English** and **Metric** units.



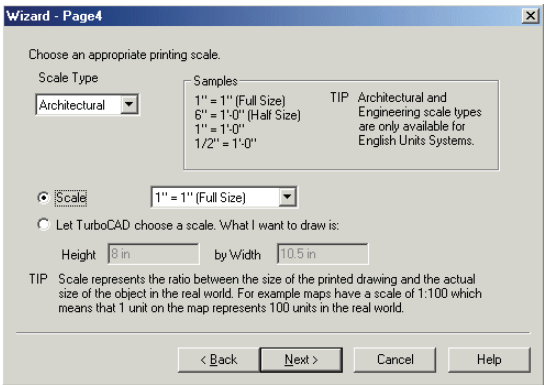
On the second page you can set the type of unit you want to use - architectural (1'-3"), decimal (3.75 ft.), scientific, etc. You can also specify the main unit (inches, feet, yards) and the precision of the units.



The third page contains options for paper size and orientation - these are used when printing. You can also enter drawing information such as title, author, and comments.

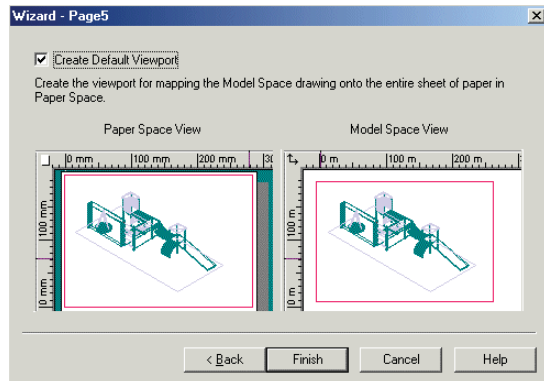


The fourth page enables you to set the printing scale. This is the size of the objects when printed, relative to their actual size specified in the drawing.

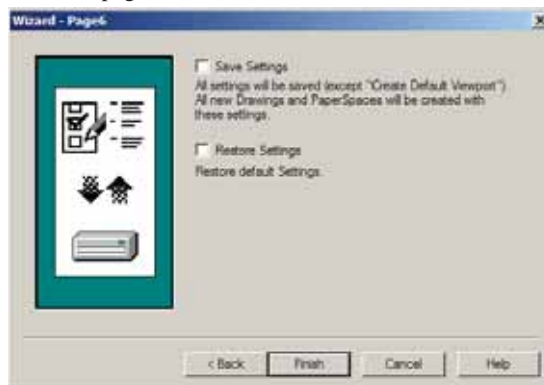




The fifth page enables you to create a default viewport for mapping the Model Space drawings onto the paper sheet as defined in Paper Space. When printing the default viewport from Paper Space, you get a hard copy of the Plan view of the current drawing because the Plan view is the default view for Model Space.



The sixth and final page enables you to save the settings for future drawing or to restore the initial settings. Settings can be edited in the TC Explorer Palette (see "TC Explorer Palette" on page 91.).



## Opening and Saving Files

Within TurboCAD you can choose to open a saved file or start a new file. You can also open new and saved files from within TurboCAD, and there are several options for saving files.

NOTE: The **General** page of the **Program Setup** contains several settings for opening and saving files. See "General Setup" on page 61.

## Opening Files

**Hotkey:** Ctrl+O



**Files of type:** By default, you will open a \*.tcw file. Use this list if you want to open a file of a different format. See "Other CAD Formats" on page 25.

**Open as read-only:** A read-only file can be viewed only; it cannot be edited.

**Description:** Displays text entered in the **Subject** field of the **Summary Info** window. See "Summary Info" on page 38.

**Show Preview:** Displays a thumbnail of the drawing. Only TurboCAD files (\*.tcw and \*.tct) can display previews.

**Setup:** When importing files of other formats, provides access to conversion options.

*NOTE: You can select multiple files from the file window to open at the same time. Simply hold down the Shift key to select multiple files.*

## Saving Files

**Hotkey:** Ctrl+S

Saves the current drawing to disk. If the file was not previously saved, the **Save As** window will appear.



**Save as type:** By default, you will save the file as a \*.tcw file. Use this list if you want to save (export) the file to a different format. See "Other CAD Formats" on page 25.

**Setup:** Enables you to set parameters for the saved \*.tcs file (see "TCW, TCT Export Setup" on page 25). When exporting to another file format, provides access to conversion options.

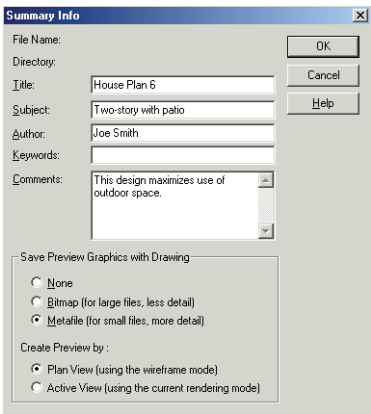
**Save:**

- **Drawing:** Saves the entire drawing.
- **Selection:** Saves the selected objects only.

If **Prompt for Summary Info** is checked in the **General** page of the **Program Setup**, the **Summary Info** window will appear before saving.

## Summary Info

Records general information to be saved with the drawing.



If **Prompt for Summary Info** is checked in the **General** page of the **Program Setup**, the **Summary Info** window will appear when you save the drawing.

**File Name and Directory:** Information for files that have already been saved.

**Keywords:** Text used when searching for this file.

**Subject:** Descriptive text that appears in the **Open** and **Save** windows. This text can also potentially be used by Windows file search utilities.

---

*TIP: If a drawing is being saved as a symbol, the **Title** should be a short description of the symbol, and **Subject** contain a more detailed description. See "Loading an Individual Symbol into the Library" on page 309.*

---

**Save Preview Graphics with Drawing:** Sets the type of graphic image that will be stored with the drawing, so that it can be previewed in TurboCAD and in other Windows programs.

- **None:** No preview graphics will be saved.
- **Bitmap:** Suitable for large files because the bitmap will use the same amount of space no matter how complex the drawing is.

- **Metafile:** Suitable for small files because it displays more detail. Symbol previews are generally stored using this option.

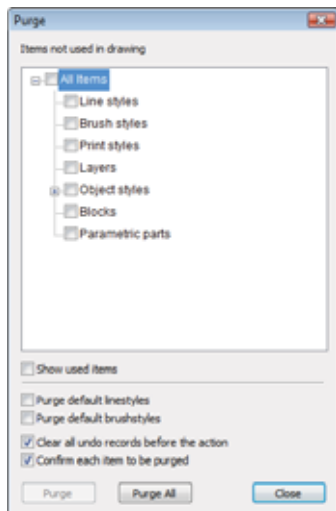
**Create Preview by:** Saves the preview as the plan view or by the current view and render mode.

## Purge

*Available in TurboCAD Pro and Platinum only*

The purge tool is designed to make your files smaller by eliminating un-used elements that are stored in you file. For example you can purge un-used blocks, line styles, or object styles.

1. Select **Purge**. The Purge dialog will open.



2. Select the items you wish to purge. If necessary expand the tree so you can select sub-items. Alternately you can press the Purge All button.
3. After selecting your items press the Purge button.
4. Press Close when you are finished.

## Purge Options

**Show Used Items:** When this option is selected the tree will show the items in the tree. You cannot purge while this option is on. Selecting items in the tree will show purge parameters for that item type. The parameters are displayed in a text box directly below this option checkbox.

**Purge default linestyles:** If this option is turned on un-used default linestyles will be purged from the drawing. This option is off by default. It is recommended to use this option with caution.

**Purge default brushstyles:** If this option is turned on un-used default brushstyles will be purged from the drawing. This option is off by default, it is recommended to use this option with caution.

**Clear all undo records before action:** If this option is turned on all undo buffers will be deleted. This prevents issues with attempt to undo purged items. This option is on by default.

**Confirm each item to be purged:** If this option is on you will be presented with a confirmation dialog for each item that is being purged from the drawing. This option is on by default.



## Getting Help

There are several ways to get help on any TurboCAD topic.

### Online Help

The **Help** menu provides access to the online help, which basically contains the same content as this book.

The help is also context-sensitive, which can be accessed in several ways:

- Click the **Help** button (question mark icon), then click on the tool button, menu item, tab, or palette in question.
- Place the cursor over a button, menu item, or palette, or highlight a menu item, and press F1.
- Press F1 while using a tool or window.
- Press the **Help** button on a window.

To see a tool tip for an icon, hold the cursor over the button for a second or two. The tool tip will appear, showing the name of the tool.

---

*TIP: Look in the **Help** menu to see a list of keyboard shortcuts. Look in the **Help** menu for the **Tip of the Day** for helpful hints.*

---

### Web Tutorials

There are several online demo tutorials that can be accessed from the page that appears in the Internet Palette. You need the Macromedia Flash plug-in to view these tutorials.

These short, animated demos demonstrate some basic concepts about the user interface, and show you how to use some of the TurboCAD tools.

### Help on the Internet

Select **Help / TurboCAD on the Web**. Each item in the submenu opens the Internet Palette to the selected web site.

### Forums

Go to [www.turbocad.com](http://www.turbocad.com) and click User Forums. Choose the forum for the version of TurboCAD you have.

The first time you enter the forums, you will be asked to register. Registration helps maintain your preference settings, and your real name and email address help us all interact with you better.

The TurboCAD Forums are the place to get the absolute best technical support. All the support comes from other users, including professional experts (architects, engineers, and drafters) from all over the world, some using TurboCAD full-time. There are also a few IMSI Design employees and programmers participating. Ours are perhaps the most active forums in the CAD industry. It's free, and everyone is quite eager to help. In fact, forum users help IMSI develop and improve the software!

Feel free to reply to a message, post a new message, or create your own new thread. Introduce yourself. Formalities are waived, and you are welcome regardless of skill level. This is where we get our questions answered and our struggles resolved on a regular basis.

### Technical Support

Go to the support section of <http://www.turbocad.com> for detailed support options.

Technical support is available by calling 415-483-8000.

Technical Support will work closely with you to solve any problems related to our software. Please give our support technicians as much information as possible. Remember that they are not in front of your computer and that they need your help to diagnose the problem.

On occasion, a problem can be traced to hardware, or to another software application. Our technician will supply as much support as possible in these cases, but they are not authorized to support products manufactured or published by another company.

You might find that e-mail is a more convenient way to get technical support. If you do send e-mail, provide as much information as you can about your system and about the problem.

## Technical Support CheckList

You may already have the information you are looking for. Before calling, check this manual thoroughly. To receive the fastest response to your technical questions, please be in front of your computer with TurboCAD running, and be prepared to provide the following information before you call or send e-mail:

- The type of computer and Windows version you are using.
- The name, version number, and other information about your specific version of the product. (To find this information, select **Help / About TurboCAD** from the TurboCAD menu.)
- The exact sequence of events that created the problem. Verify that you can reproduce the problem by following the same series of steps.
- The exact wording of any error messages.
- Steps you've taken to find the answer to your question, including resources used.
- The results of any steps you have undertaken to resolve the problem.



## 2 User Interface

This section provides information on your working environment in TurboCAD. In addition to descriptions of the UI and its components, information is provided about program and drawing setup, customization, and windows.

Object properties, and the various means of setting and changing properties, are also covered.

### Components of the UI

The main area of the TurboCAD screen is the drawing area. When creating and inserting 2D and 3D objects, you typically work in Model Space, and when putting your model on paper, Paper Space is used.

By default, the background color is white, but you can change that by selecting **Options / Background Color** and choosing another color.

Nearly all components of the screen can be customized. See "Customizing the UI" on page 57.

### Drawing Area: Model Space and Paper Space

For creating and laying out your drawing, TurboCAD provides you with two drawing environments: Model Space and Paper Space.

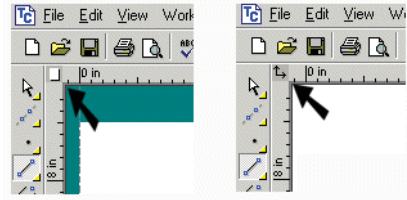
- Model Space is the environment in which you create your drawing, usually called a model. In this area you do drafting and design work, creating two-dimensional drawings or three-dimensional models.
- Paper Space is the environment in which you create the final layout of your drawing for printing or plotting it on paper. In Paper Space you usually arrange the drawing's elements on a sheet of paper.

---

*NOTE: Floating Model Space enables you to use Model Space tools within a viewport in Paper Space. See "Floating Model Space" on page 661.*

---

You can use the small button at the top left corner, at the intersection of the rulers, to toggle between Model and Paper Spaces.

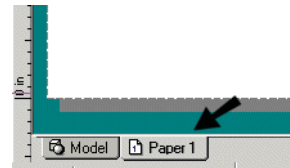


---

*NOTE: If the rulers are turned off, this button will not appear. Go to **Program Setup** to toggle the ruler display.*

---

There are also workspace tabs at the lower left corner of the screen. By default, each file starts with one tab for Paper Space. Click the tabs to switch spaces.



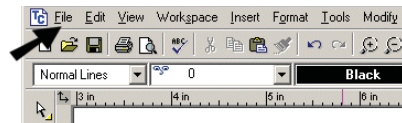
---

*NOTE: If scroll bars are turned off, these tabs will not appear. Go to **Program Setup** to toggle the scroll bar display.*

---

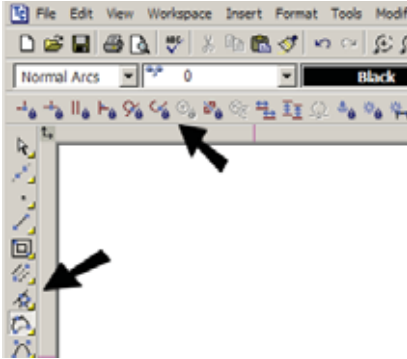
### Menu Bar

All tools and options are available by selecting them from the menus at the top of the screen - **File, Edit, View**, etc. (Most tools are available in toolbar icons as well).



## Toolbars

Toolbars are groups of related icons. They can be located on top and below the screen, and to either side of the screen.



By default, the **Standard** toolbar appears at the top of the screen, and the **Property** toolbar (see "Property Toolbar" on page 80) appears just below it. **Drawing** tools appear along the left side.

Icons that have a small yellow or black triangle in the corner are fly-outs. If you click one and hold the mouse button...



... the entire toolbar will "fly out," and you can select the desired icon. The color of the triangle is dependant upon UI settings.



You can also create tabbed toolbars and popup toolbars that appear when you open the local menu. You can modify a toolbar's icons and create your own toolbars. See "Customize Toolbars" on page 58.

In addition to the default toolbars you see when you first start TurboCAD, there are numerous other toolbars you can display. There are two ways to access additional toolbars:

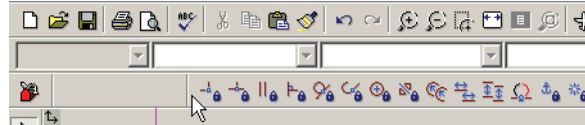
- Open the **Toolbars and Menus** page of **Program Setup**. This opens the **Toolbars** tab of the **Customize** window, which contains a list of all available toolbars; check the ones you want to see.

- Right-click in any toolbar area. The local menu that appears contains the list of all available toolbars; select a toolbar to add or remove from the screen.

## Moving, Docking, and Rolling Toolbars

By default, certain toolbars are docked along the sides of the screen, and when you display another toolbar it appears "floating" on the screen. You can dock and undock all toolbars, and the same applies for palettes as well.

You can drag a toolbar by picking it on its top or bottom edge (for vertical toolbars) or either side edge (for horizontal toolbars).



For palettes, pick and drag it by its title bar.

Moving a toolbar to any edge of the screen will dock it to that edge. Docking to either side creates a vertical toolbar, docking to the top or bottom of the screen creates a horizontal toolbar. You can also move a toolbar onto the drawing area, where it will "float" on the screen.

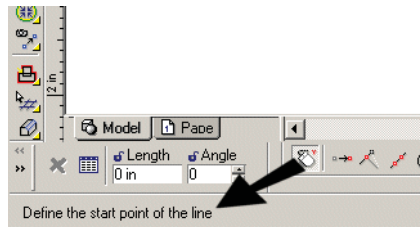
---

*TIP: You can also undock or dock a toolbar by double-clicking any icon on the toolbar.*

---

## Status Bar

Located at the lower left corner of the screen, the Status Bar lets you know the current status of the model.



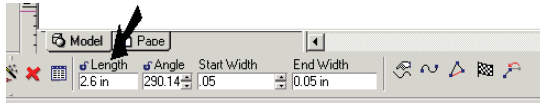
For example, if you have just activated the **Line** tool, the Status Bar will contain the prompt "Define the start point of the line."



## Inspector Bar

Enables you to create objects by defining some or all of its numerical parameters.

By default, the Inspector Bar is located immediately below the drawing window.

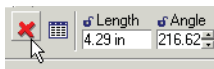


**NOTE:** You can customize the Inspector Bar appearance, by choosing **Classic** or **Custom** in the **Desktop** page of the **Program Setup**. You can also display or hide the Inspector Bar on this page.

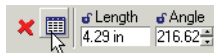
For most tools, the Inspector Bar is divided into three parts.

On the left side are two icons, **Cancel** and **Properties**.

- **Cancel:** Ends an operation without completing it. For some tools, once the operation is complete the tool remains active. You must click **Cancel**, or activate another tools, to exit the tool. The hotkey for **Cancel** is the Esc key.



- **Properties:** Opens the **Properties** window for the object you are creating. See "Object Properties" on page 79.



The middle section of the Inspector Bar contains the numeric fields. For example, when creating a line segment in the Polyline tool, you can set the **Length**, **Angle**, and **Start** and **End Widths**. For a circle, the fields would be **Radius**, **Diameter** and **Circumference**. When moving the cursor to position or size an object, the values in the fields update dynamically.

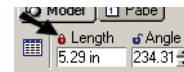
Length	Angle	Start Width	End Width
2.6 in	290.14	.05	0.05 in

You can access the numeric fields via the cursor, or by pressing Tab or Ctrl+E. You can also press Tab to scroll forward to the next field. Return to the drawing by pressing Esc or clicking in the drawing.

**TIP:** If you want to jump from the Coordinate Fields to the Inspector Bar, first press Esc, then Ctrl+E.

If you enter a value and press Enter, Any Ortho or snap modes are ignored. You can also enter the results of mathematic expressions from the **Calculator**. See "Calculator Palette - Variables" on page 52 and "Using Expressions in Data Fields" on page 54.

**Locking:** For some fields you can lock their values by clicking the lock symbol. This means that no matter where you place the cursor, the value will remain fixed.

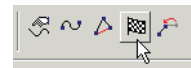


The right side of the Inspector Bar contains options relevant for the tool or mode.

**NOTE:** All options on this section of the Inspector Bar can also be accessed from the local menu, opened by right-clicking.

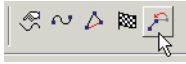
There are two common local menu options that appear for many functions: **Finish** and **One Step Back**.

- **Finish:** Completes the operation and exits the tool. For instance, when creating a polyline, you can click **Finish** after creating the last segment. The hotkey for **Finish** is Alt+F.

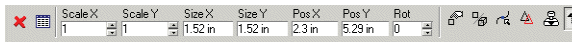


**NOTE:** For many tools, double-clicking when creating the last component of an object is equivalent to clicking **Finish**.

- **One Step Back:** For tools that require multiple steps, this enables you to scroll backward through the steps already completed until you reach the desired start point, then continue working.



When in **Select Edit** mode, the Inspector Bar has a different appearance. The numeric fields enable you to change the size, scale, and rotation of the selected objects, while the local menu options include **Make Copy**, **Selector** properties, etc. **Size** and **Position** fields for 3D objects are turned off by default. See "Select Edit" on page 198.



**NOTE:** The options that appear on the Inspector Bar in **Select Edit** mode depend on whether you are in 2D or 3D Selector mode, and can be changed. See "2D / 3D Selector" on page 181.

## Customizing the Inspector Bar

You can remove any of the fields or local menu option icons by right-clicking anywhere on the Inspector Bar (except the numeric fields). This opens a local menu in which you can deselect any of the controls.

You can also change the width of any of the numeric fields by dragging either side with the mouse.

## Coordinate Fields

When creating or sizing objects, you can define points by entering them directly into the Coordinate Fields. By default, this toolbar is located at the lower right corner of the screen.



When moving the cursor, the values in the fields update dynamically.

The type of coordinates depends on the coordinate system you are using. In the absolute and relative coordinate systems you use X, Y coordinates. In the polar coordinate system you enter an angle and distance. See "Coordinate Systems" on page 105.

You can access the numeric fields via the cursor, or by pressing Shift+Tab or Ctrl+R. You can also press Tab to scroll between Coordinate fields.

**TIP:** If you want to jump from the Inspector Bar to the Coordinate Fields, first press Esc, then Ctrl+R.

If you enter a value and press Enter, Any Ortho or snap modes are ignored. You can also enter the results of mathematic expressions from the **Calculator**. See "Calculator Palette - Variables" on page 52 and "Using Expressions in Data Fields" on page 54.

**Locking:** You can lock a coordinate by clicking the lock box. This means that no matter where you place the cursor, the value will remain fixed.



You can also lock coordinates by using the **Lock** options in the **Modes** menu.

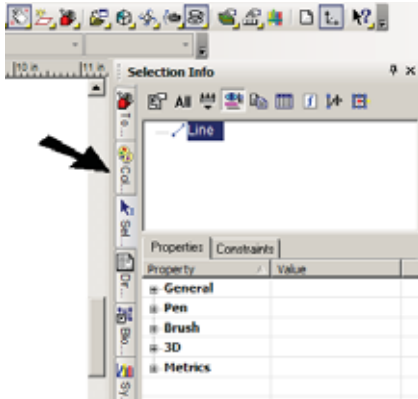
**TIP:** The Coordinate Field locks are particularly useful when you lock just one field. If you use a locked value in one field, you can then set the second coordinate using the mouse. This makes it easy to define a series of points along a specific horizontal or vertical line.

The Coordinate Field also provides access to snap modes. Disable the SNAP button to temporarily turn off any running snaps, and disable the GEO button to temporarily turn off any running geometric aids. Right-clicking on either the SNAP or GEO fields opens the **Drawing Aids** window. See "Snaps" on page 108.



## Palette Area

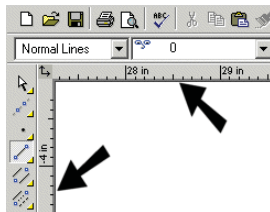
Located by default on the right side of the screen, the tabbed palettes provide convenient access to many features, such as **Blocks**, the **Internet**, **Properties**, etc.



For details on each palette, see "Palettes" on page 48.

## Rulers

You can work with rulers to have a visual representation of the current dimensions.



Rulers are only displayed for orthographic views (top, left, right, etc.) and display the current Model Space or Paper Space units (see "Space Units" on page 71).

---

*TIP: Another handy visual aid, which can be used in tandem with rulers, is the grid. See "Grid" on page 107.*

---

## Scroll Bars



Your drawing can be larger (much larger!) than what you see on the screen. The scroll bars can be used to pan to a different area of your drawing. Click on the arrows at the ends of the scroll bars to move a small amount. Drag the scroll box itself to move faster.

## Local Menu

The local menu, sometimes called the context menu, popup menu or context-sensitive menu, is opened by right-clicking the mouse. The contents of this menu differ depending on where you right-click, and in which tool you are working.

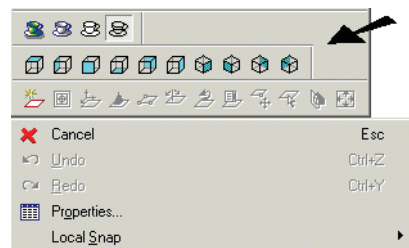
---

*NOTE: You can customize the contents of the local menu via **Tools / Customize**. See "Customize Popup Toolbars" on page 58.*

---

When working in a tool, options on the local menu also appear in the Inspector Bar. See "Inspector Bar" on page 45.

In addition to customizing local menu items, you can add popup toolbars - icons that will appear each time you open the local menu. See "Customize Popup Toolbars" on page 58.



The following items are found on most local menus:

**Cancel:** Cancels the current operation, and exits without completing.

**Undo:** Undoes the last operation and return to the previous condition. The last operation is displayed here.

**Redo:** Reverses Undo.

**Properties:** Opens the **Properties** window. See "Object Properties" on page 79.

**Local Snap:** Sets the snap mode for the next point only. See "Snaps" on page 108 and "Snap Modes" on page 111.

## Palettes

TurboCAD palettes are convenient screen areas used for performing common tasks and for obtaining information. By default, the palettes are displayed on the right side of the screen, and contain tabs for easy display and switching.

The palettes commands are available on the **View** menu, and can be accessed on the flyout toolbar on the **Standard** toolbar.



You can also display the **Palettes** toolbar by right-clicking on any toolbar area and selecting **Palettes**.



To customize how palettes are displayed, see "Customize Palettes" on page 59.

### Docking Palettes

Palettes can be docked just like toolbars (see "Moving, Docking, and Rolling Toolbars" on page 44). In addition, palettes can be docked onto other palettes to create palette groups.

Drag a palette onto the desktop by its named tab (the tab that contains the name of the palette. The palette must actually be displayed before it can be dragged onto another palette group.

To dock a palette to any of the four edges of the TurboCAD window, drag the palette toolbar to the relevant arrow, either displayed at the center of the drawing area, or along the desired edge.

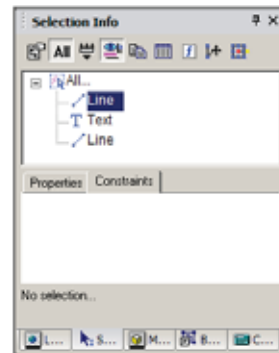



---

*NOTE: These arrows are displayed only if **Docking Stickers** is checked on the **Palettes** page of the **Customize** window (**Tools / Customize**).*

---

To combine palettes into a group, drag more palettes onto a docked palette.



To replace palettes, simply drag their tabs back to the palette area.

### Library Palette



Symbols and parametric parts are groups of objects available for repeated use. While groups and blocks are internal to a drawing, symbols and parts are external files. The **Symbols** palette is used to display and insert symbols found in the various symbol libraries - both those included with TurboCAD installation and those you create yourself. You can also use the palette to save symbols.

See ".Library" on page 304.

## Blocks Palette



A block is a collection of objects combined into a single object. Blocks are useful for storing complex, common drawing objects that you will use more than once. Blocks are stored in the drawing's internal block library and a block reference, and not the actual object, is inserted into the drawing. When a block is edited, all instances of that block in the drawing are updated.

See "Blocks" on page 291.

## Selection Info



Displays information about the currently selected object or objects, such as entity type, measurements, location in the drawing, and physical and engineering properties.

See "Selection Info Palette" on page 191.

You can also use this palette to edit 3D objects. See "Editing 3D Objects using Selection Info" on page 487.

## Measurement Info



Displays measurements calculated by the **Measure** tools. You can measure point coordinates, distance and perimeter, angle, and area. See "Measuring" on page 277.

## Internet Palette



Provides access to the Internet from within the TurboCAD screen. The Internet Palette launches Microsoft Internet Explorer.

By default the TurboCAD home page is opened, but you can enter any URL and use the toolbar icons at the top to navigate the web. The features on this toolbar (**Back**, **Forward**, **Stop**, **Refresh**, etc.) are identical to those used in your conventional browser.

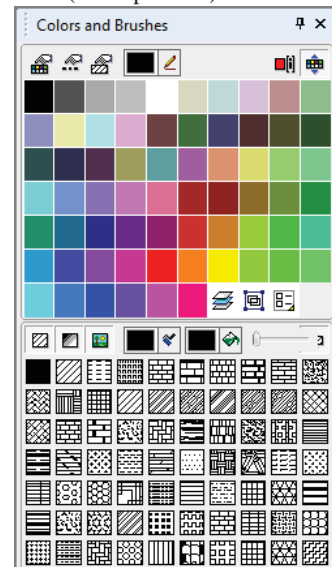
You can insert hyperlinks into your drawing that will open a web page (or a file). See "Hyperlinks" on page 178.

## Colors and Brushes



Displays all colors and brushes currently available. See "Color Palette" on page 65 for details on adding or modifying colors.

By default, the Colors and Brushes palette shows colors on the top and brushes (hatch patterns) on the bottom.



The color selected at the top and the brush pattern selected at the bottom appear in the **Property** toolbar. These properties become the default for the current tool and other tools in the tool group.



For example, if **Line** is active and you set a color, this color will become the default for **Rectangle**, **Polyline**, etc.

The icons at the top of the palette are as follows:

- **Color Palette:**



Opens the palette in which you can create new colors or modify existing ones. See "Color Palette" on page 65.

- **Line Style Editor:**



Opens the **Line Styles** page of the **Drawing Setup**. See "Line Styles" on page 74.

- **Brush Style Editor:**



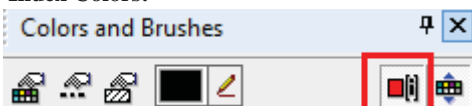
Opens the **Brush Editor**, in which you can create new brushes or modify existing ones. See "Brush Styles" on page 84.

- **Pen Color:**



Click the color swatch to replace the color with one from the palette. If you click the pen icon, you can then click any TurboCAD object to select its color. To pick the color of any other object, i.e. objects in other applications, keep the left mouse button pressed and release it when hovering over the object whose color you want.

- **Index Colors:**



Toggles on the display of indexed colors instead of RGB colors. Indexed colors correspond to the indexing of colors in AutoCAD.

- **Expand Colors:**



Toggle this icon off if you don't want the colors displayed.

The icons at the bottom of the palette are as follows:

- **Show hatches, gradients, or bitmaps:**



Filters what types of brushes are displayed. Hatches are simple repeated line patterns. For details on gradient fills, see "Gradient Patterns" on page 87, and for bitmaps, see "Bitmap Patterns" on page 86.

- **Brush Color and Fill Color:**



Sets the colors of the hatch pattern lines and the empty space between lines. If you click the brush or fill-can icons, you can select color on screen as described for **Pen color** above.

- **Brush Transparency:**



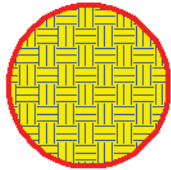
Controls the transparency of the brush pattern and fill.

- **Expand brushes:**

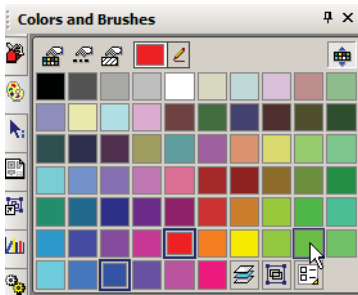


Toggle this icon off if you don't want the colors displayed.

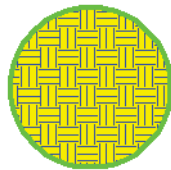
1. To see how Colors and Brushes works, set a color, line weight and brush for the **Circle** tool group, and create a circle.



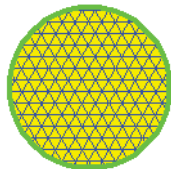
2. Select the circle, and click another color in the upper section of the palette.



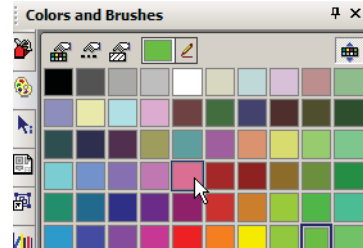
The color of the circle updates.



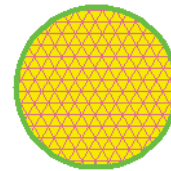
3. Change the brush pattern the same way - first select the circle, then click a different pattern from the lower section of the palette.



4. To easily change the brush color, select the circle and right-click on the color you want for the brush.



The brush pattern (not fill color) updates to the new color.

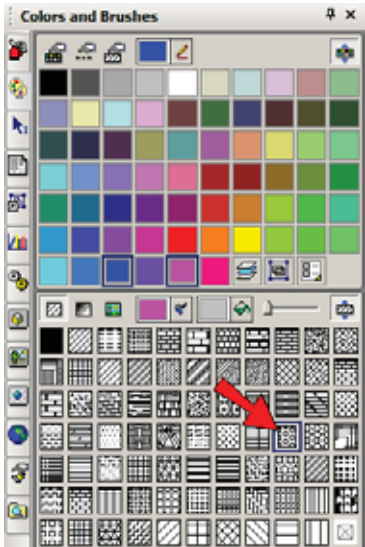



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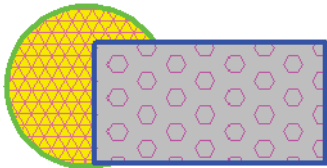
NOTE: The above changes only affect the selected circle. If you create a new circle, or use any other tool in the **Circle/Ellipse** group, the properties you originally set will be used. You can change the default properties simply by activating a tool and setting new properties.

---

5. Activate **Rectangle** (or any other tool in the **Line** group), and set a new pen color, brush, and brush and fill colors.



6. Create a rectangle that overlaps the circle.

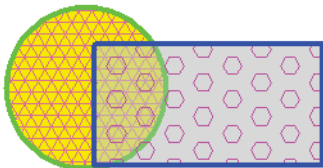


Any tool you use in this group will have these properties.

7. Select the rectangle and lower the **Brush Transparency**.



You can now see through the rectangle.



NOTE: *The rectangle is in front of the circle because it was created after the circle. For details on stacking, see "Stacking Objects" on page 248.*

## Calculator Palette - Variables

Hotkey: F2 (Ctrl+F2 to close)



Enables you to make numerical calculations within TurboCAD, and to define variables.

(20*3)+(10*1.5) = 75			F(.)
Variable	Value	Formula	
Length	5.00	Length=5	
Width	2.50	Width=Length/2	

You can also use this palette to assign constraints to dimensions. See "Constraining Dimensions" on page 269.

## Mathematical Calculations

To perform a calculation, enter the values in the top field of the palette, using parentheses as needed to create inner expressions. No spaces are allowed. The four mathematical operators should be entered as +, -, \* (multiply), and / (divide).

(20*3)+(10*1.5)			F(.)
Variable	Value	Formula	

To calculate the result, press Enter.

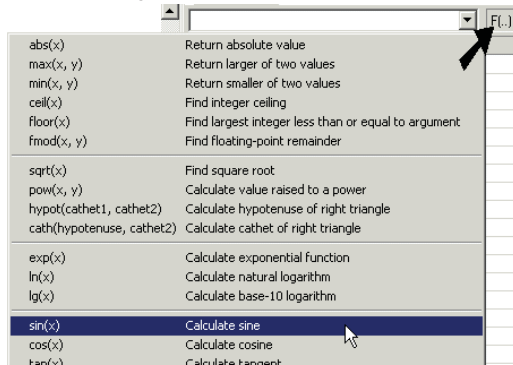
(20*3)+(10*1.5) = 75			F(.)
Variable	Value	Formula	



You can also use these commonly-used values in place of numbers:

- $\pi = 3.1415926$
- $e = 2.71828$

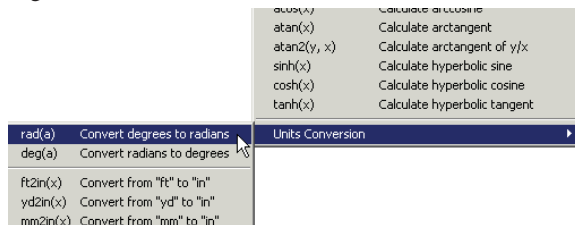
The function (F) button enables you to perform a function on the value in the parentheses. For example, select **sin(x)** to get the sine of an angle.



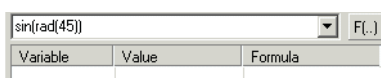
sin(x) appears in the calculator field (you also could have entered this expression yourself manually).



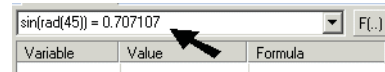
Sine is calculated for radians, so if you have a value in degrees it must be converted. Look in the functions menu under **Units Conversion** and select rad(a), which converts degrees to radians.



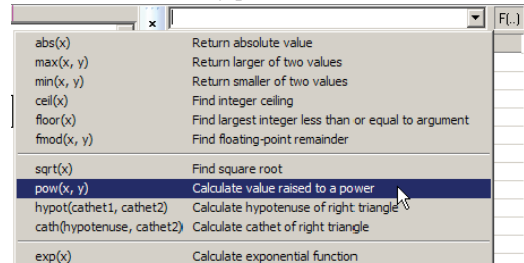
Arrange the expression so that the parentheses are in the correct places, and include the angle in degrees (45 in this case):



Press Enter to obtain the result.



Some functions require two values, such as exponentials. **pow(x, y)** raises x to the y power.



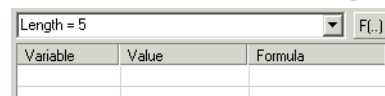
Include the values and press Enter to get the result.



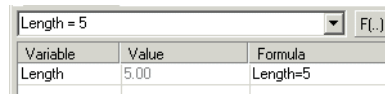
## Defining and Using Variables

In addition to calculating numerical equations, you can use the Calculator Palette to define variables. Subsequent values and variables can then be based on variables you've already created.

In the calculator field, enter the variable definition in the format "variable name=value" (in this example, Length=5). The value can be a number, function, or another variable. Variables are case sensitive and cannot have spaces.



Press Enter, and the variable, value, and formula are listed in the list below.



You can also enter a variable and variable or formula directly in the fields of the list.

The next variable can be based on an existing variable. In the calculator field, enter an equation that uses a variable name, such as “Width=Length/2.”

Width=Length/2			F(.)
Variable	Value	Formula	
Length	5.00	Length=5	

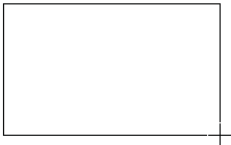
Press Enter, and the new variable appears, with its calculated value.

Width = 2.5			F(.)
Variable	Value	Formula	
Length	5.00	Length=5	
Width	2.50	Width=Length/2	

Using Expressions in Data Fields

Numbers or variables from the Calculator Palette can be used as values in the Inspector Bar or Coordinate fields.

As an example, define **Length** and **Width** variables, and draw a rectangle (see "Rectangle" on page 142). Select the first corner point, but not the second point.



Press Tab to access the first field in the Inspector Bar. The value is highlighted.

Size A	Size B
2.4 in	1.46 in

Press F2 to access the Calculator Palette. In the calculator field, type Length and press Enter.

Length			F(.)
Variable	Value	Formula	
Length	5.00	Length=5	
Width	2.50	Width=Length/2	

NOTE: You could also enter a mathematical expression based on numbers, or an expression that includes one or more variables.

Back in the Inspector Bar, the first field contains the new value.

Size A	Size B
5	1.46 in

For subsequent fields of the Inspector Bar or Coordinate Fields, do not press Tab to scroll. Instead, place the cursor in the desired field, then press F2 to access the Calculator Palette.

Style Manager



Enables you to define styles for commonly-used architectural items, such as windows, doors, and tables.

See "Style Manager" on page 564.

Design Director

Available in TurboCAD Pro and Platinum only



A convenient place to perform commands related to layers, workplanes, views, cameras, and objects or object groups.

See "Design Director" on page 125.

Drafting Palette

Available in TurboCAD Pro and Platinum only



Enables you to place standard views of a model into Paper Space. You can also create sectional views. See "Drafting Palette - Creating Standard Views" on page 643.

## MacroRecorder Palette

Available in TurboCAD Pro and Platinum only



Creates and plays scripts - scenarios of creating and editing objects and manipulating their properties. You can also record object transformation - moving, copying, scaling, and rotating. Scripts can be saved for future play. See "Macro Recorder" on page 671.

## Environments Palette



Render scene environments (see "Environments" on page 530) can be assigned to a drawing to enhance its rendering. Options are available for the model background and foreground. For details on using the palette, see "Environments Palette" on page 532.

## Luminances Palette



Luminances (see "Luminance" on page 518) are light types that are added to an object, as opposed to lights added to the overall model. See "Luminances Palette" on page 521.

## Materials Palette



Materials (see "Materials" on page 522) can be assigned to 3D objects in order to create more realistic rendering. See "Materials Palette" on page 529.

## TC Explorer Palette

Available in TurboCAD Pro and Deluxe only



A multi-purpose palette, enabling you to browse files, view components (layers, blocks, etc.) of all open files, and copy components from one file to another.

See "TC Explorer Palette" on page 91.

## Parametric Part Script Editor



Enables you to write or edit scripts for parametric parts.

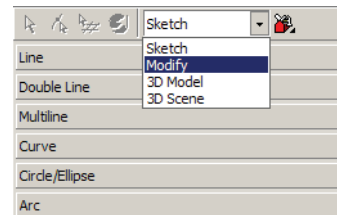
See "Parametric Part Script Editor" on page 323.

## Tools Palette

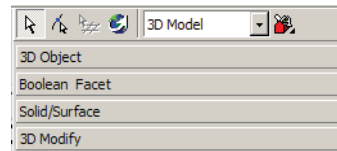


A palette that enables you to view commonly-used sets of tools.

The Tools Palette contains, by default, four templates of tool groups. The default template is Sketch, which contains toolbars for **Line**, **Double Line**, **Curve**, etc.

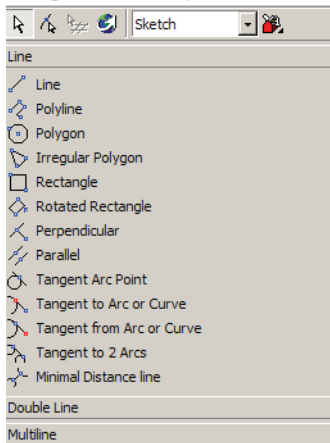


If you switch to another template, such as **3D Model**, you will see tools relating to 3D modeling: **3D Object**, **Boolean Facet**, etc.

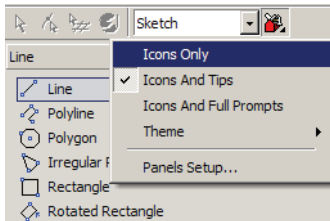


Open a set of tools to see what tools are listed within. These are the same tools you would see in the toolbar of the same name. For example, the **Line** toolbar is available as a

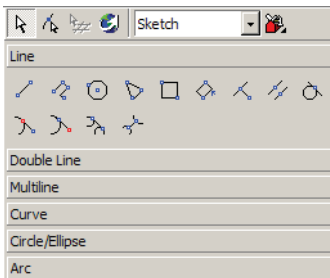
separate toolbar, or as a fly-out toolbar along the left vertical toolbar. But if you use **Line** tools often, it is handy to keep them open in this palette for easy access.



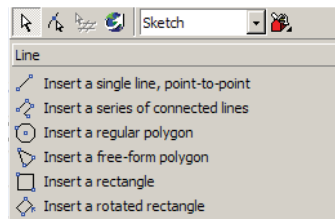
You can also control how these tools are listed in the palette. Click the toolbox drop-down icon on the Tools Palette toolbar. **Icons and Tips** is shown below, each tool has an icon and a tool name listed.



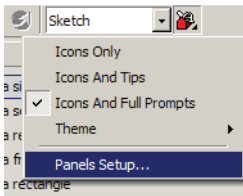
**Icons Only** displays only the icons. This is handy if you are already familiar with the tools and want to save space.



If you are not familiar with each tool, **Icons and Full Prompts** shows the icon along with a complete tool description.



You can also create new templates. Click the toolbox icon and select **Panels Setup**.



*NOTE: If you select **Theme**, you can control the look and colors (skins) of the Tools Palette.*

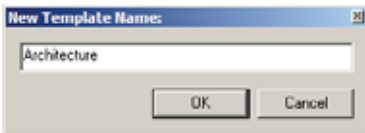
The default templates, or panels, are listed here:



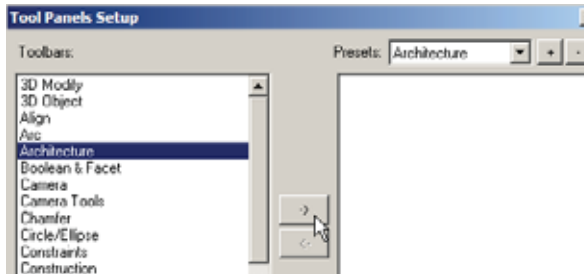
To create a new template, click the + icon.



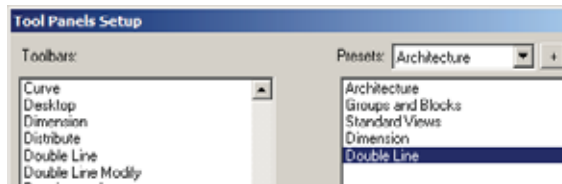
You are then asked for a template name. For example, if you create architectural drawings, name this template "Architecture."



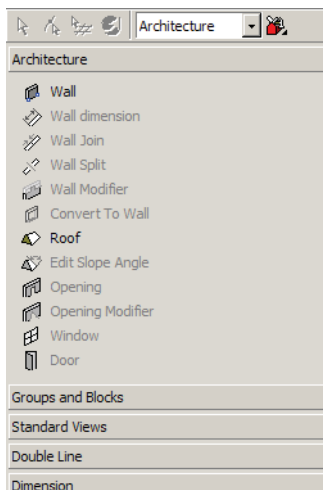
On the left side of the window, find the toolbars you want to add to this template. Click the right-facing arrow to add it to the list. (Or double-click on the toolbar name.)



Continue adding toolbars. You can use the **Move Up** and **Move Down** icons to arrange the order of the template.



Click OK, and the “Architecture” tool group appears in the palette.



## Customizing the UI

You can customize toolbars, menus, keyboard, and your IntelliMouse (if you use one).

You can invoke the **Customize** window at any time by right-clicking in any blank toolbar area.

---

*NOTE: If you prefer an older interface for customizing toolbars and menus, select **UI Themes**, and select the Version 11 interface.*

---

## Customize Options

Enables you to manage the configuration files, control the look of the TurboCAD window, and specify external devices.



**Theme:** Select from several different workspace looks, or “skins.”

**Workspace:** Select from several options that affect what toolbars are displayed and how they are laid out. If you create a workspace you want to use again, click **Save In**.

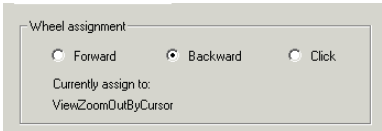
**Other:** Various settings for how icons and tooltips are displayed. **Menu animations** refers to how drop-down menus appear when a menu item is clicked.

**Devices:** Click **Additional Devices Setup** to configure any external devices you use with TurboCAD. By default, only **Intellimouse** is available.

Customize IntelliMouse

Relevant if you have an Intellimouse connected to your computer.

**Wheel assignment:** Shows the command that will be executed when you rotate the mouse wheel forward, back, or click with it. To change it, highlight the desired command, select **Forward**, **Backward**, or **Click**, and click **Assign** to implement the change.



Customize Toolbars

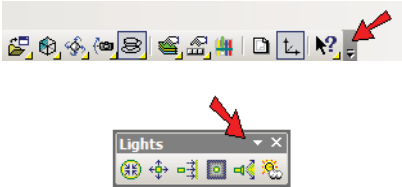
The **Toolbars** page enables you to select which toolbars are displayed.



This page can also be opened by right-clicking in any blank toolbar area.

You cannot delete default toolbars, but you can create new toolbars by clicking the **New** button. Toolbars you create this way can later be deleted.

To customize a new or existing toolbar, first make sure it is displayed. Then click the arrow icon shown; the location of this arrow depends on whether or not the toolbar is docked.

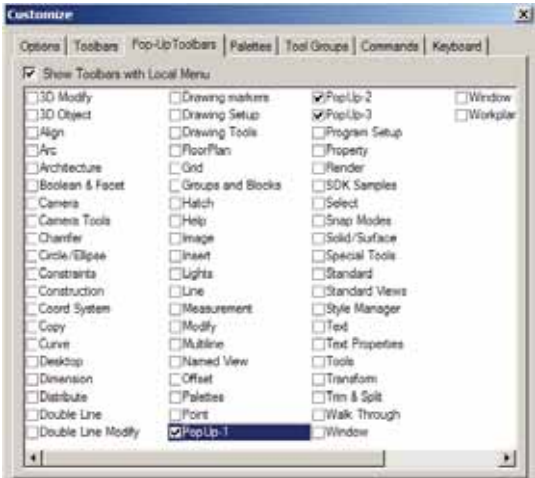


This invokes options which enable you to add or remove icons.

You can also use **Customize Commands** to add or remove icons. See see "Customize Commands" on page 60.

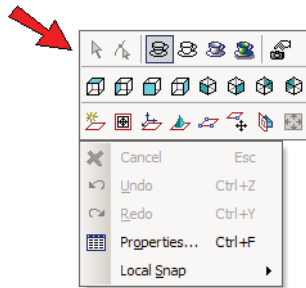
*NOTE: If you prefer an older interface for toolbar customization, select **Tools / UI Themes**, and select the **Version 11** interface.*

Customize Popup Toolbars

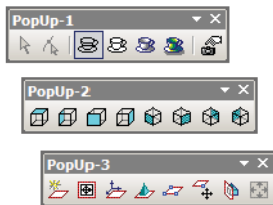


Popup toolbars are icons that appear with the local menu, which is opened by right-clicking. This is useful if you use certain icons frequently and want convenient access to them.

There are three PopUp toolbars defined by default:



If you want to add additional popup toolbars, simply check them on the list. If you want to change the existing popup toolbars, first display them as standalone toolbars. This is done in the **Toolbars** page of the **Customize** window, or by right-clicking in any toolbar area and selecting the popup toolbar you want to display.



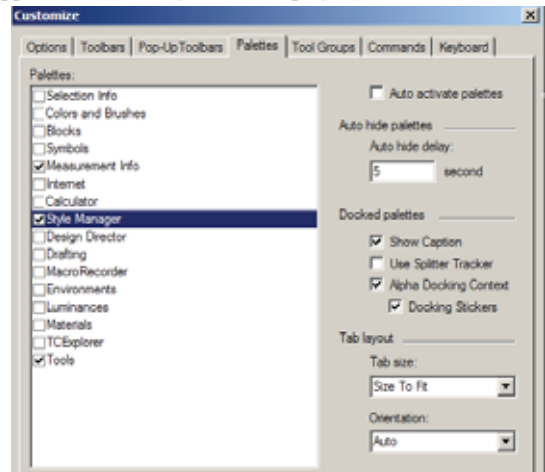
To customize the toolbar, click the arrow icon shown. This invokes options which enable you to add or remove icons.



You can also use **Customize Commands** to add or remove icons. See "Customize Commands" on page 60.

## Customize Palettes

The **Palettes** page enables you to control which palettes appear and how they can be displayed and moved.



**Auto activate palettes:** Opens the palette relevant for the current option. For instance, when using hyperlinks, the Internet palette opens.

**Auto hide palettes:** Hides the palette when not in use.

**Docked palettes:** These options are relevant while moving or docking palettes.

- **Show Caption:** Displays the palette title.
- **Use Splitter Tracker:** If disabled, the palette can be resized dynamically (in real time).
- **Alpha Docking Context:** If disabled, an outline of the palette appears while moving a palette. If on, a transparent blue rectangle appears while moving.
- **Docking Stickers:** Displays the blue controls that appear while moving or docking.

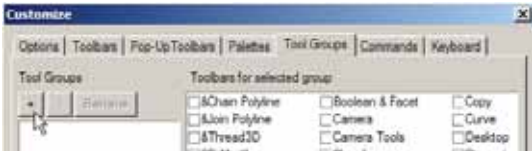


For details on the various palettes available, see "Palettes" on page 48.

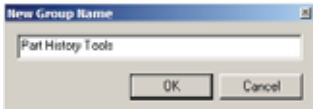
### Customize Tool Groups

Tool groups are used to specify what palettes and popup toolbars appear for certain groups of tools.

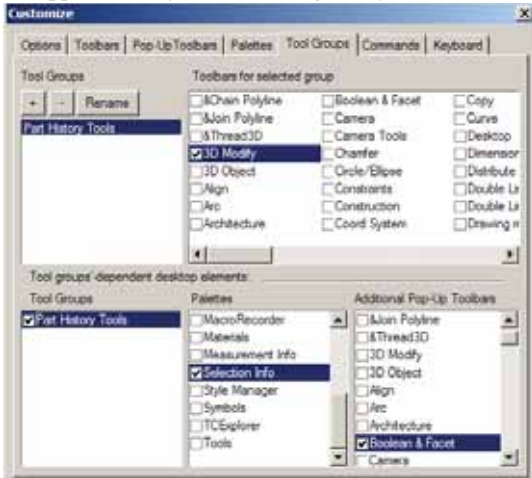
1. To create a tool group, click the “+” icon.



2. Assign a name to the tool group.



3. In the top part of the window, select the toolbars that contain the tools in the group. In the lower part, select the palette (or palettes) and popup toolbars that will appear while you are working in any of these tools.

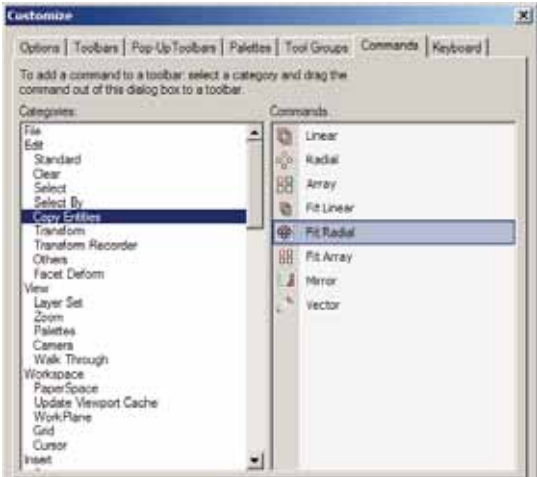


In the above example, the Selection Info palette will be the default open palette when you are working with any **3D Modify** tool. Another example could be to open the Calculator palette when using any Dimension tool.

### Customize Commands

The **Commands** page enables you to customize toolbars by adding and removing icons.

Display the toolbar you want to change, and open the **Commands** page. Locate the commands you want to add to the toolbar, and drag it from the **Commands** list to the toolbar itself.



To remove an icon from a toolbar, drag it away from the toolbar into the **Commands** page.

### Customize Keyboard

The **Keyboard** page enables you to view current keyboard shortcuts (hotkeys), to modify them, or to create new ones.

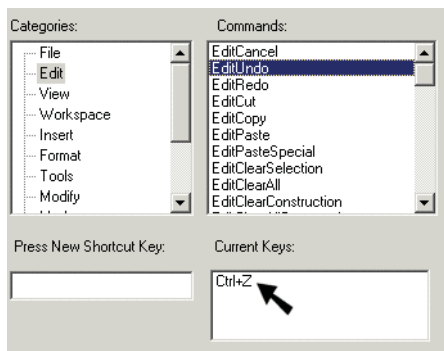
---

**TIP:** Select **Help / Keyboard** for a list of all shortcuts.

---

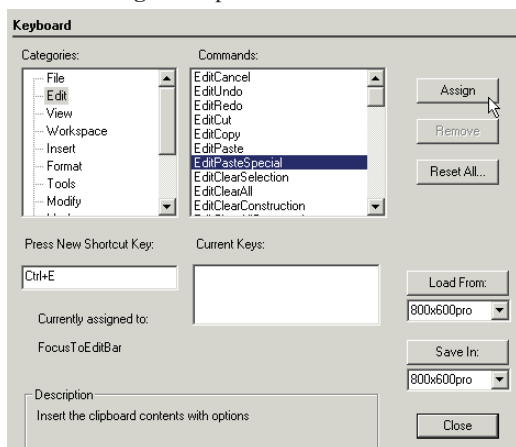


1. Locate the **Category** and then the command for which you want a shortcut.
2. If a shortcut already exists, it will be listed in the **Current Keys** box.



You can click **Remove** to delete any unwanted shortcuts.

3. To assign a shortcut, type it (for instance, press Ctrl+E) and it will appear in the **Press New Shortcut Key** box.
4. Click **Assign** to implement the shortcut.



## Program Setup

Contains pages in which you can control TurboCAD program settings. You can access each page directly from the **Options** menu.

The options that you set in this window are saved when you exit TurboCAD, and will remain in effect the next time you start the program.

The **Program Setup** pages can be accessed from the **Options** menu, or from the flyout icon on the **Standard** toolbar.



You can also display the **Program Setup** toolbar by right-clicking in any toolbar area and selecting **Program Setup**.

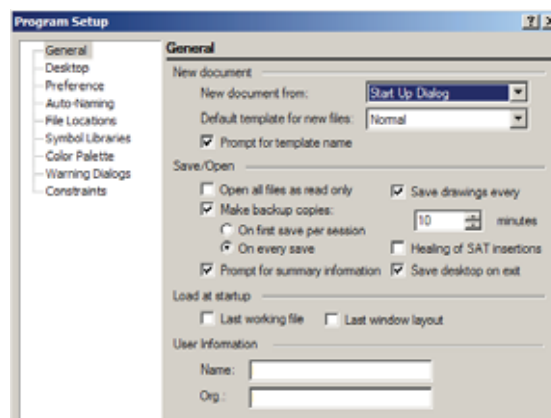
---

*TIP: You can also use the **System Defaults** in the TC Explorer Palette to set program setup options. See "Program Defaults" on page 93.*

---

## General Setup

Sets file opening and saving parameters, as well as user information.



**New document from:** Select how files will be opened when using **File / New**.

**Default template for new files:** If no template is specified when creating a file, a default template will be used. This option sets the default template.

**Prompt for template name:** You will be prompted for a template each time you start a new file. If you want to use the default template without being prompted, clear this box.

**Open all files as read-only:** If checked, files cannot be edited.

**Make backup copies:** If checked, select **On First Save** or **On Every Save** to determine how often backups will be made. Backup files have the extension \*.bak. By default, backup files are saved in the same folder as your primary drawing files, but you can change this on the **File Locations** page.

**Save drawings every:** If checked, autosave copies (\*.asv files) will be made at the specified intervals. After a system crash, TurboCAD will open the autosave version of your drawing as soon as you launch TurboCAD. Autosave files are stored in the \AutoSave folder by default, but you can change this on the **File Locations** page.

**Prompt for summary information:** The **Summary Information** window enables you to record user information connected to your drawing. If checked, this window will be displayed the first time that you save any drawing to disk, and whenever you use **Save As** to save a new copy of the file.

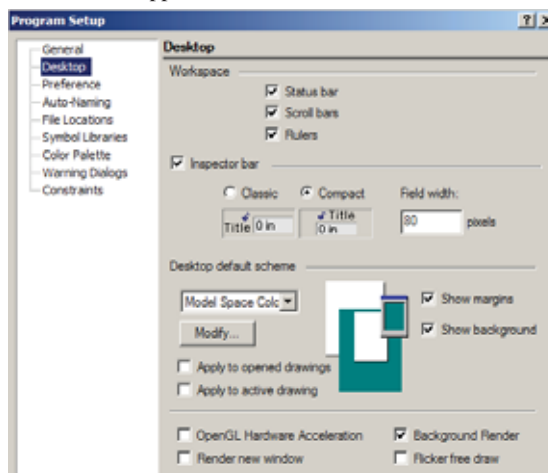
**Healing of SAT insertions:** For imported files, heals gaps between faces and holes within faces.

**Save desktop on exit:** If checked, the desktop settings (toolbars, etc.) will be saved when you exit the program.

**Load at startup:** You can choose to load the last working file and the last window layout (see "Window Layout Templates" on page 78) each time you start TurboCAD.

## Desktop

Controls what appears on the screen.



**Workspace:** Controls the display of the working area

- **Status Bar:** Displays or hides the Status Bar.
- **Scroll bars:** Displays or hides the scroll bars.
- **Rulers:** Displays or hides the rulers.
- **Inspector Bar:** Displays or hides the Inspector Bar. See "Inspector Bar" on page 45.

**Desktop Scheme:** Customizes the color of paper and background, and floating palettes. Click **Modify** to change an element's color.

*TIP: You can also click the different elements in the preview window.*

- **Show margins:** Displays page (printer) margins.
- **Show background:** If checked, the background behind the paper will be displayed. If unchecked, the background will be white.

*TIP: You can switch off the paper until you're ready to print, then use the **Page Setup** (see "Page Setup" on page 664) to place your drawing on the page.*

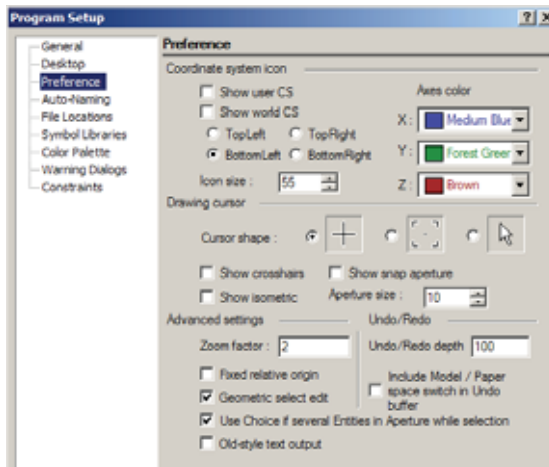
**OpenGL Hardware Acceleration:** Available if your video card is equipped with an accelerator, take this characteristic into account while performing a render.

**Render new window:** When you render your drawing, the render view will open in a new drawing window.

**Background Render:** With this option, TurboCAD tools work faster in Render mode; rendering will “interfere” less with other tools. This setting can only be applied when working with one window.

## Preference

General program options, such as cursor shape, axis color, zoom factor, and display of coordinate system icons.



**Coordinate system icon:** You can choose to display the UCS and/or WCS. For both, you can set the color of the axes and size of the icon. For the WCS, select the corner of the screen where it will be displayed.

### Drawing cursor:

- **Cursor Shape:** Select one of the three shapes provided.
- **Show crosshairs:** Overrides the normal cursor, displaying a screenwide horizontal and vertical line intersecting at the cursor position.
- **Show Isometric:** Overrides the normal cursor, displaying all three axes.

- **Show Snap Aperture:** Displays a circular area around the cursor whose radius is the **Aperture Size**. Aperture is used while snapping, to detect objects.

---

NOTE: You can also use the **Cursor** menu to change the cursor display.

---

**Zoom factor:** Controls how much you zoom in or out of the drawing when you use **Zoom In** or **Zoom Out**. The default zoom step setting is 2, meaning that zooming in doubles the size. See "Zoom In and Zoom Out" on page 99.

**Fixed relative origin:** If checked, the relative origin will be fixed while in relative or polar coordinates. If you need to relocate the origin, it must be done manually. See "Relative Coordinates" on page 106 and "Polar Coordinates" on page 106.

**Geometric select edit:** If checked, objects will be selected by their geometric extents, rather than their cosmetic extents. This permits geometrically accurate scaling of objects like double lines and lines that have nonzero width. See "Geometric and Cosmetic Select Modes" on page 198.

**Use Choice if several entities in aperture while selection:** If more than object lies within the aperture area during a selection, a small window appears from which you can select the desired object.

### Undo/Redo:

- **Undo/Redo depth:** Sets the number of operations stored in the Undo buffer.
- **Include Model / Paper space switch in Undo buffer:** If checked, switching between Model and Paper spaces will be included in the Undo buffer.

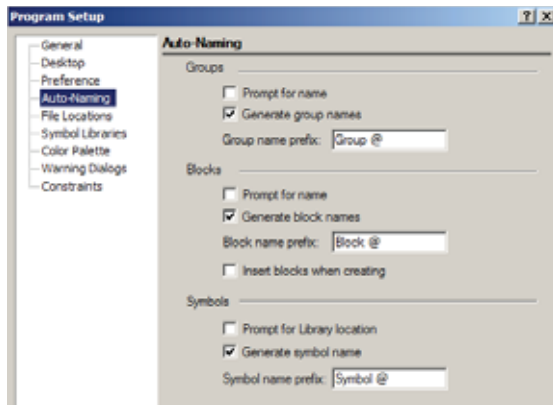
**Old style text output:** Revert to the text display feature of older versions of TurboCAD.

## Toolbars and Menu

Open the **Customize** window to the **Toolbars** page to modify toolbars. See "Customize Toolbars" on page 58.

## Auto-Naming

Controls how groups, blocks, and symbols are named. See "Auto-Naming" on page 289.



**Prompt for name:** You will receive a prompt each time a new item is created.

**Generate names:** Names will be automatically assigned.

**Prefix:** If names are automatically generated, you can enter a string that appears before the item name. The "@" character is a placeholder for the automatic number.

**Insert blocks when creating:** Each block will be inserted into the drawing once it is created.

**Prompt for library location:** If not checked, all saved symbols will be stored in the default folder. See "Loading Symbol Folders into the Library" on page 309.

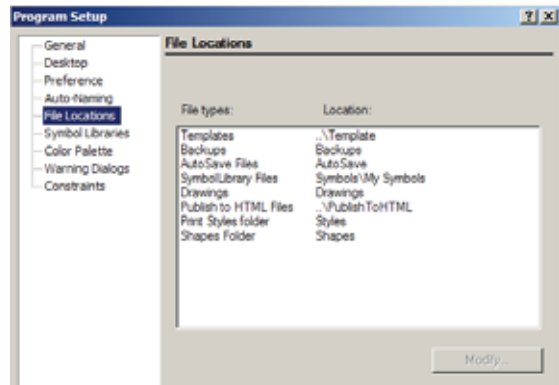
---

**NOTE:** If you turn off both the **Prompt for name** and **Generate names** options, names will not be assigned to groups. Names will be assigned to blocks and symbols, however.

---

## File Locations

Enables you to specify the folders where TurboCAD places several types of program files.

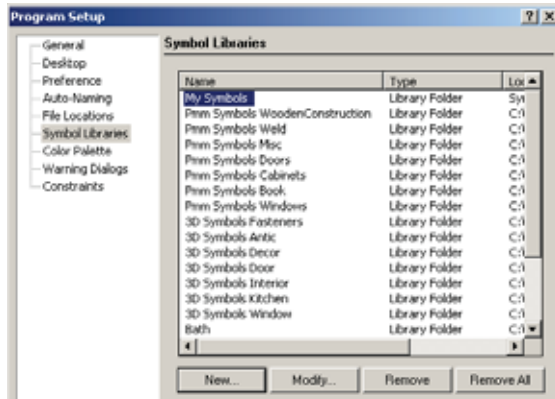


- **Templates (\*.tct)** - Files in which you save settings as well as block libraries, symbol libraries, and scripts, that you can use to start a new drawing.
- **Backups (\*.bak)** - Backups that are created if you check **Make Backup Copies** option in the **General** page.
- **AutoSave Files (\*.asv)** - Files that are saved automatically, if this option is turned on in the **General** page. These files allow you to restore work after a system crash.
- **Symbol Library Files (\*.slw)** - Sets of related symbols, saved to one common \*.slw file, or a set of drawing files saved to a specified folder.
- **Drawings (\*.tcw)** - Drawings and models you create in TurboCAD.
- **Publish to HTML Files** - Files created using **File / Publish to HTML**.
- **Print Styles:** See "Print Style Options" on page 667.
- **Shapes (\*.shx):** Shape files (\*.shx) used in the creating of line styles. (This is not the same as \*.shx font files.) See "Line Style Editor" on page 76.

*TIP: If you have two disk drives (or access to a network drive), it is wise to have your backup files automatically saved to a second drive.*

## Symbol Libraries

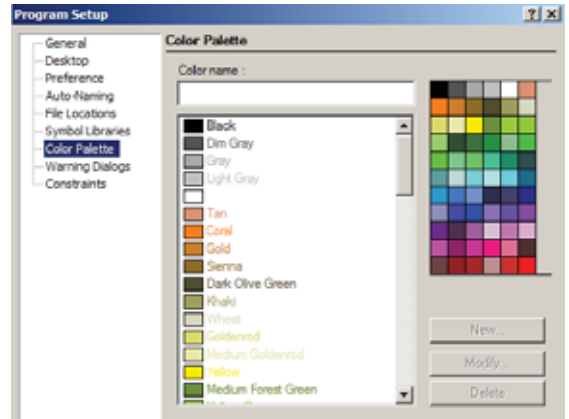
Displays the folders containing files that can be viewed in the Library palette.



Click **New** to add more folders. You can also modify the name of a library folder or file, or delete or modify symbol library files and/or folders. You can also access the symbol libraries from the Library palette. See ".Library" on page 304

## Color Palette

Enables you to add, modify, or delete colors from the TurboCAD palette.



To add a new color, type the name and click **New**. You will then be prompted to select a color from the color wheel.

You can modify a color by selecting it from the list and clicking **Modify**. Delete a color by selecting it and clicking **Delete**.

When you have the colors you want, they can be displayed for easy selection in the Colors and Brushes palette. see "Colors and Brushes" on page 49.

## The Color Wheel

You can set colors by RGB values, or by Hue, Saturation, and Value numbers.

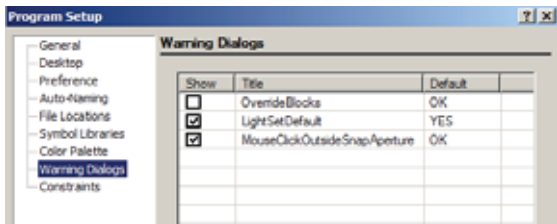
**Red, Green, Blue:** Sets the amount of each color in the light. Values can be a maximum of 255.

**Hue:** The color value, where 0 is red, 60 is yellow, 120 is green, 180 is cyan, 200 is magenta, and 240 is blue. If you change the hue, the values for red, green, and blue will be changed to match.

**Sat:** The saturation level for the color (amount of color), up to a maximum of 240.

**Lum:** The luminosity (brightness) of the color.

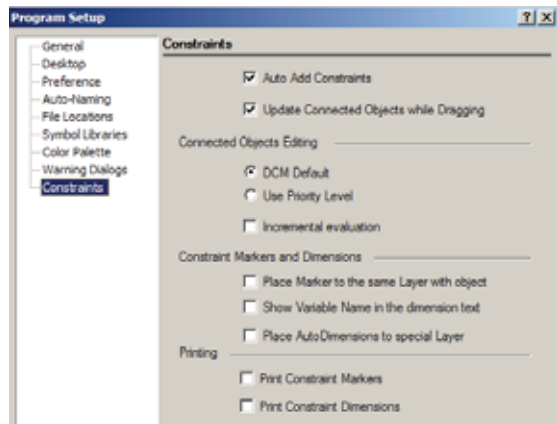
Warning Dialogs



Controls the display of TurboCAD warning messages. If you received a warning message with the “Don't show this message again” checkbox, the warning message will appear in this table. Use the **Show** column to show or hide messages.

Constraints

Available in TurboCAD Pro and Platinum only



Settings and controls for geometric and dimensional constraints.

**Auto Add Constraints:** If checked, the **Auto Add Constraints** tool is enabled. Any constrainable geometry you create while this is active will have constraint automatically assigned. See "Constraining Geometry" on page 251 and "Constraining Dimensions" on page 269.

**Update Connected Objects while Dragging:** Dynamically updates the position, shape and size of constrained objects as you drag them within the **Edit** tool.

**Connected Objects Editing:**

- **DCM Default:** Changes to any part of a set of constrained objects can affect all objects equally.
- **Use Priority Level:** Constraint changes to any part of a set of objects will affect that part first, with minimal changes to the remaining objects.
- **Incremental Evaluation:** Constraint changes are constantly checked and changes are being made. If disabled, the results will be checked only after changes are made. For large scale changes, this option should be enabled.

Constraint Markers and Dimensions:

- **Place marker to the same layer with object:** Constraint markers are placed on the same layer as the object that is constrained. Otherwise they are placed on their own layer.
- **Show variable name in dimension text:** Displays the variable name in parentheses after the dimension value.

**Printing:** Choose whether constraint markers and constrained dimensions will be included when printing.

Drawing Setup

Sets properties related to the current drawing.

The **Drawing Setup** pages can be accessed from the **Options** menu, or from the flyout icon on the **Standard** toolbar.

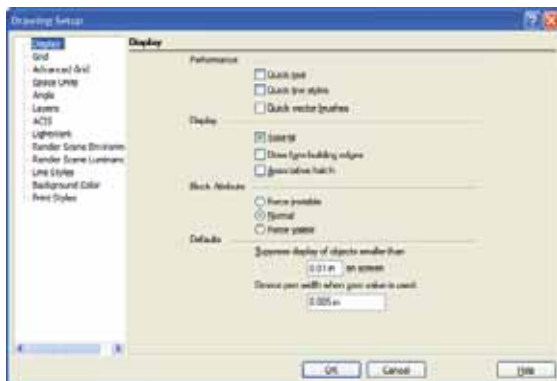


You can also display the **Drawing Setup** toolbar by right-clicking in any toolbar area and selecting **Drawing Setup**.

*TIP: You can also use the **Defaults** for a drawing in the TC Explorer Palette to set drawing setup options. See "Drawing Defaults" on page 94.*

## Display Options

Options for adjusting the quality and redraw speed of the display, and options for displaying block attribute values.



**Performance:** When working with large files or a slow computer, you may be able to increase your display speed by selecting one or more of these options. The display speed can sometimes be slowed by heavy text, styled lines, and fill patterns.

- **Quick text:** Displays all text as small boxes.
- **Quick line styles:** Simplifies the display of line styles.
- **Quick vector brushes:** Simplifies the display of hatch and fill patterns.

### Display:

- **Solid Fill:** A line of a specified width will be drawn as solid.
- **Draw form-building edges:** Draws form-building edges of 3D surfaces. See "Display" on page 359.
- **Associative hatch:** When you modify a hatched object, the hatch pattern will update to fit the new shape.

**Block Attribute:** Options relevant for blocks that contain block attribute definitions.

- **Force invisible:** Hides all attribute values.
- **Normal:** Shows the attribute values as they were defined while creating.

- **Force visible:** Shows all attribute values, even those defined as invisible.

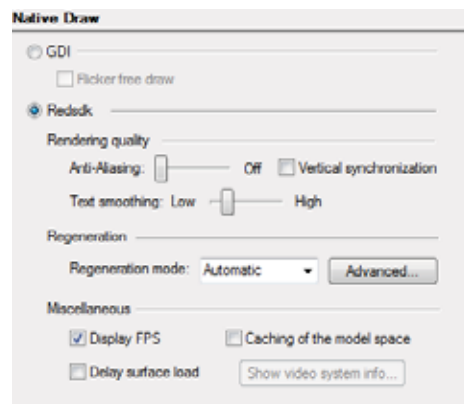
**Suppress display of objects smaller than:** Sets the size at which TurboCAD will draw a simplified representation of objects. This size is measured in device units (the "device" being your computer's display), so that one inch will be equal to approximately one inch of space on your screen. At smaller sizes, TurboCAD will display objects using a simplified representation, increasing display speed. As you zoom in, you get the detail required for precise editing; as you zoom out, you get faster redraw.

**Device pen width when zero value is used:** Controls the printed width of lines set to zero width.

## Native Draw

*Available in TurboCAD Pro and Platinum only*

The Native Draw dialog allows you to specify which display/drawing engine will be used. you can select between standard GDI and Redsdk. GDI uses standard Windows CPU based drawing technology and CPU based OpenGL hardware acceleration. Redsdk is an OpenGL accelerated drawing engine that can substantially accelerate the speed at which entities are drawn.



### GDI Options:

**Flicker Free Draw:** When this option is turned on the redrawing of entities is made smoother, however this can marginally slow down zooming and panning of drawings.



**Redsdk Options:**

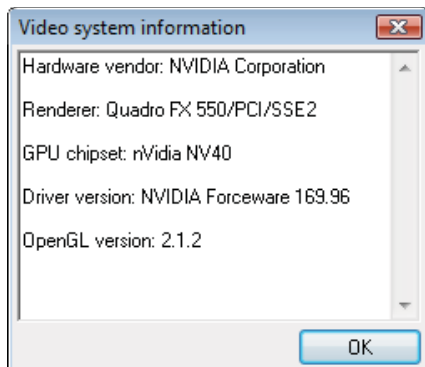
**Display FPS:** If this option is on a display appears in the upper right corner of the drawing space. This display track the Frame Per Second to indicate the speed of the Redsdk drawing engine.

**Anti-Aliasing:** – Creates an anti-aliasing effect on 2D graphic elements. The values available are: Off, 1x, 2x, 4x, 8x, 16x. Turning this on will make the display smoother, but lowers the performance.

**Vertical Synchronization:** Makes the display looking better, while moving, however it may higher memory usage.

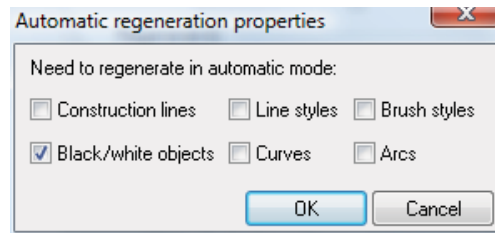
**Text Smoothing:** Turning this on will make the display of text smoother, but lowers the performance.

**Show Video System Info:** Pressing this button will display the current video card setup on your computer. If your current driver for the video card is not the optimal driver for Redsdk you will be prompted to download the correct driver via a link within the window. If you do not download the optimal Redsdk will use a default driver.



**Regeneration:** To optimize performance Redsdk will delay regeneration of some drawn elements during movement such as panning or zooming. You can specify that regeneration be Manual, or automatic. If set to manual all delayed elements will not be regeneration until you press F5 or select Regen or

Redraw from the menu. The Advanced settings for Regeneration allow you to specify which items are optimized and which are not.



**Delay Surface Load:** Delays the loading of surfaces and objects until rendering to optimize memory usage.

**Caching of Model Space:** Store the contents of the model space in cache to optimize presentation.

Redsdk is dependant upon video cards which support OpenGL acceleration, and the features of that card. If you do not have an OpenGL video card you will not be able to use Redsdk.

Switching between GDI mode and Redsdk will require a restart of TurboCAD.

Redsdk conflicts with Aero support for Windows Vista and Window 7. If you switch to Redsdk you will be prompted to allow TurboCAD to turn Aero off.

Redsdk conflicts with GDI's native support for OpenGL hardware acceleration. If you switch to Redsdk you will be prompted to allow TurboCAD to turn native OpenGL hardware acceleration off.

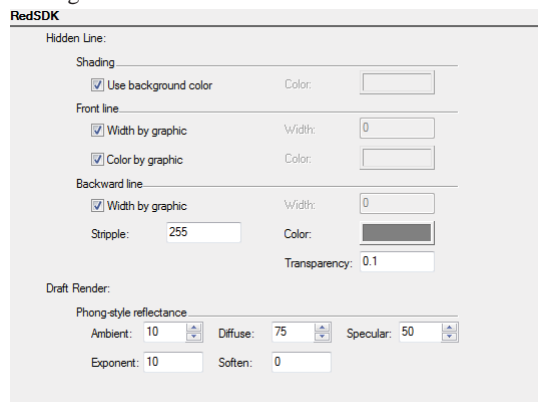
With Redsdk turned on the Camera Properties "Renderable objects" is always disabled, "Suppress Hidden Line" is always enabled for all pages except the Wireframe page in Camera Properties Dialog.



## Redsdk

Available in TurboCAD Pro and Platinum only

The Redsdk settings allow you to control how object appear when using the Redsdk for draft rendering and hidden line rendering.



**Use background color:** When it is turned off (On by default), the option for editing internal color of objects will become available (by default the color is white).

**Width by graphic:** When this option is on the Width of the visible part of graphic's line is taken from the graphic itself. When it is turned off, the field for editing the width of the visible part of object lines will become available (by default the width is = 0).

**Color by graphic:** When this option is on the color of the non-hidden part of an object's lines is taken from the graphic itself. When the option is off the option for editing the color of the non-hidden part of an object's lines will become available (by default the color is white).

**Width by graphic:** When on the Width of the hidden part of an object's line is taken from the graphic itself. When it is turned off, the field for editing the width of the hidden part of object lines will become available (by default the width is = 0).

It should be noted that the width of hidden lines must be less than or equal to the width of visible lines.

### Backward Line Settings

**Color:** This option defines the color of hidden lines (by default it is grey; R=G=B=128)

**Transparency:** This option defines the transparency of hidden lines, with the range of 0 to 1 (by default it is 0.1)

**Stipple:** This option defines the type of dashed line used for hidden lines, with the range of 0 to 255 (by default it is 255)

### Draft Mode Settings

**Ambient:** This field defines the light intensity factor for all light sources of Ambient type (by default it is 10).

**Diffuse:** This field defines the luminance factor (by default it is 75).

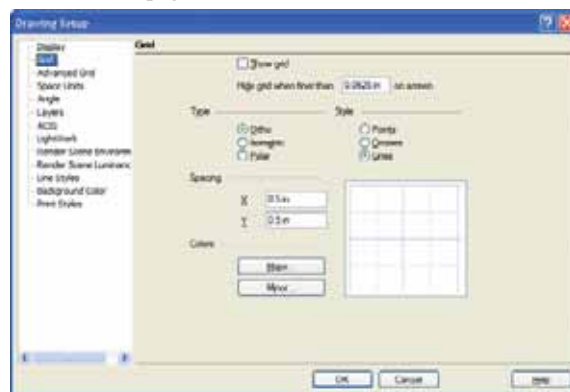
**Specular:** This field defines the glare factor (by default it is 50).

**Exponent:** This field defines the glare attenuation parameter (by default it is 10).

**Soften:** This field defines glare 'softness' (by default it is 0).

## Grid Options

Controls the grid type, size, and display. You can set up different grid properties for Model Space and Paper Space. See "Grid" on page 107.



**Show grid:** Equivalent to displaying the grid using the **Grid** menu or **Grid** toolbar.

**Hide grid when finer than:** This value applies to distances on your screen which are not related to World or Paper units.

**Type:** These grid types are based, by default, on a horizontal baseline. This can be changed on the Advanced Grid page.

- **Ortho:** Orthogonal grid.
- **Isometric:** Aligned along lines measured 30° and 150° from the horizontal axis. This type of grid is used in isometric drawing, which uses perspective drawing techniques to represent 3D objects.
- **Polar:** Points in the polar grid radiate from the origin, and are aligned by their radial distance from the origin and their angular distance from the horizontal axis.

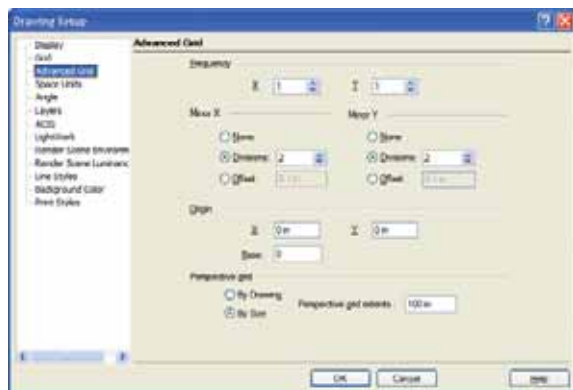
**Style:** Select **Points** (dots at each grid point), **Crosses** (crosses at each main grid point), or **Lines** (graph paper).

**Spacing:** Sets the distance between all grid lines, major and minor, without respect to frequency. Select X and Y spacing, or angular and radial values for a polar grid.

## Advanced Grid Options

Not listed in the **Options** menu; you must select it from within the **Drawing Setup**.

Advanced controls for the frequency and locations of grid lines. See "Grid" on page 107.



**Frequency:** Enables you to alternate visible grid lines with invisible grid lines. For example, if you set the frequency to 4, every fourth grid line will be displayed. You will still be able to snap to invisible grid lines.

**Minor:** Controls for the intermittent (lighter by default) grid lines.

- **None:** No minor grid lines will be displayed.

- **Divisions:** Establishes the number of minor grid lines by setting the number of divisions between each major grid point.
- **Offset:** Creates minor grid points at a fixed distance from each major grid point. Negative values can be used.

**Origin:** Enables you to use a "local" grid. By default, the grid origin is located at the origin of the coordinate system, at (0,0). You can reset the origin point of the grid and change the base angle, aligning the grid to a non-orthogonal object. For example, you can orient the grid of a house to its lot.

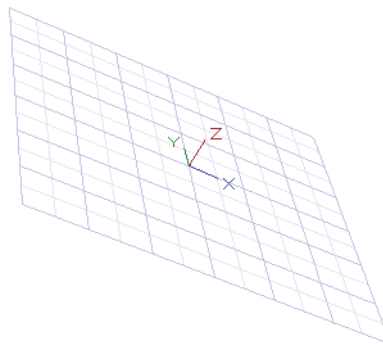
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NOTE: **Set Grid Origin** is also accessible from the **Grid** toolbar.

---

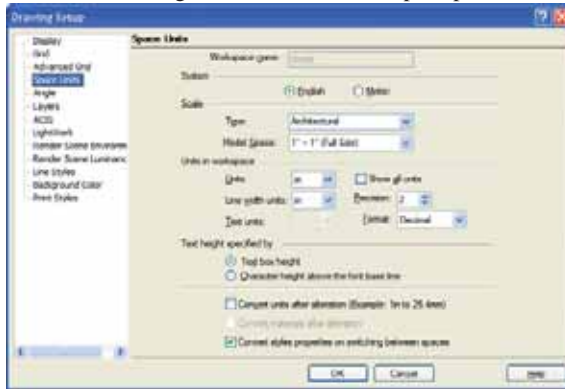
**Perspective grid:** Relevant when the grid is displayed and you are working in **Perspective** mode.

- **By Drawing:** An infinite grid will be displayed.
- **By Size:** The size of the displayed grid is determined by Perspective grid extents. This value sets the grid extents in both directions from the WCS origin, i.e. a value of 3 inches will produce a 6 x 6 grid.



## Space Units

Controls for setting units in Model and Paper Space.



NOTE: *Templates generally have units built in, but you can change units, or create custom units.*

**Workspace name:** Available in Paper Space only.

**System:** Selecting English or Metric will affect controls on the rest of the page.

**Scale:** The ratio of measurements on the page (Paper units) to measurements in the object being modeled (World units).

- **Architectural:**  $3/16" = 1'-0"$
- **Engineering:**  $1" = 200'$
- **Absolute:** 1:500

**Units:** The unit of measure.

**Show all units:** Displays all available units, regardless of whether English or Metric is used.

**Line width units:** The unit for setting and displaying the line width in the **Pen** page of the **Properties** window.

**Precision:** The number of decimal digits. For **Fractional** or **Architectural** units, **Precision** determines the accuracy of the denominator, in powers of two ( $1=1/2"$ ,  $2=1/4"$ ,  $3=1/8"$ , etc.).

**Text units:** Unit for setting and displaying the size of text.

**Format:** Controls the display of numbers:

- Decimal - 3.25
- Scientific -  $1E+01$
- Fractional -  $3 \frac{3}{16}$
- Architectural  $1'-5 \frac{11}{16}"$
- Engineering (scale):  $1" = 200'$

**Text height specified by:** Two methods to adjust text height when the current font is changed:

- **Text box height (default):** In specifying the height value for the text, you in fact define the height of the text bounding box. In this case, changing the font does not affect the height of the bounding box.
- **Character height above the font base line:** The text height value will be used as the actual height of the text. This method preserves the text height (but not the height of the text box) when you change the font.

NOTE: *The value of text height is determined by the vertical size of the uppercase letter "A" in the currently selected font. This value is the sum of four components: **External Leading** is the amount of space that the designer of the font suggests should be added between character rows. **Internal Leading** is the amount of space allowed for an accent mark above a character. **Ascent** is the amount of space allowed for the characters that have neither an accent mark above the character nor a character part below the font base line. **Descent** is the amount of space allowed for the character part below base line. (Lowercase letters such as "g", "p", and "q" have a character part occupying some space below the font base line.)*

**Convert units after alteration:** Forces unit recalculation after changing the units.

- Example 1 - Converting a drawing from feet to inches: if unchecked, 1 foot will become 1 inch. If checked, 1 foot becomes 12 inches.
- Example 2 - Converting a drawing from inches to mm: if unchecked, 1 inch will become 1 mm. If checked, 1 inch will become 25.4mm.

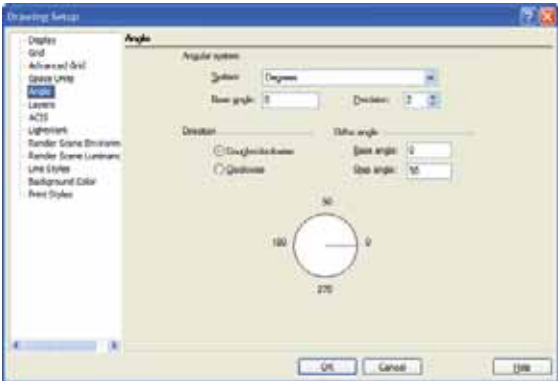
**Convert material after alteration:** Forces conversion of material sizes after changing units.

**Convert styles properties on switching between spaces:**

Activates the conversional mechanism when switching between Model Space and Paper Space.

**Angle Options**

Controls the measurement and display of angles, as well as control over the Ortho angle.



**Angular System:** Select degrees, degrees-minutes-seconds, grads, radians, or the surveyor system.

- **Base angle:** The default base angle is 0 degrees (right quadrant point). You can change this value to start angle measurement from another base angle.
- **Precision:** The number of decimal digits.

**Direction:** Choose whether to measure angles clockwise or counterclockwise.

**Ortho Angle:** By default, Ortho lines are 0 and 90 degrees. You can change this by entering new values here.

- **Base angle:** Sets the angle from which the Ortho angle is measured.
- **Step angle:** The angle to which Ortho constrains lines. If you set the step angle to 15, for example, the line will be constrained to angles 15 degrees apart. The default step angle is 90 degrees.

**Layers**

Enables you to create new layers, set the current drawing layer, set the properties of layers, and delete layers.

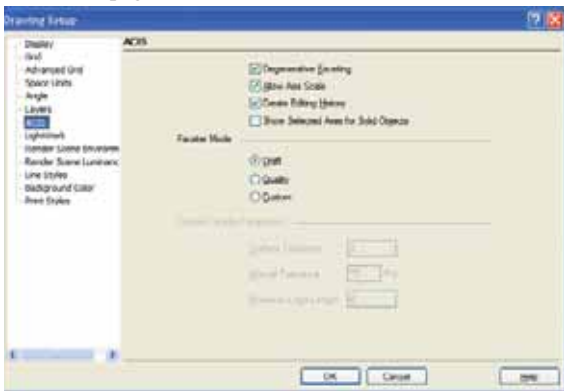


For descriptions of the options and tools in this window, see "Layers" on page 116.

**ACIS Options**

*Available in TurboCAD Pro and Platinum only*

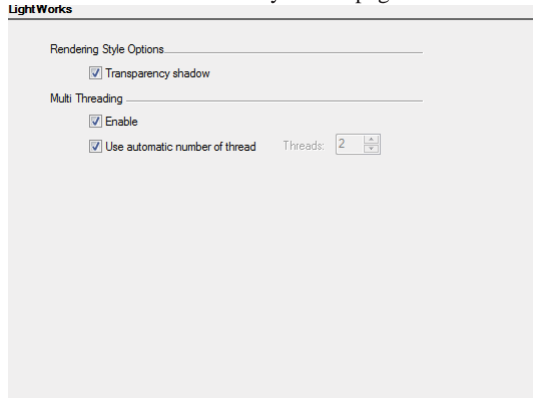
The options on this page are relevant to 3D objects. See "ACIS" on page 359.



## LightWork

Available in TurboCAD Pro and Platinum only

The options on this page are relevant to Render Styles. See “Advanced Render-Render Styles” on page 496.



**Transparency Shadow:** Specifies whether a render created with a render style will support shadows cast by transparent objects.

### Multi Threading Options

**Enable:** Enables multithreading for rendering via the use of multiple CPU cores.

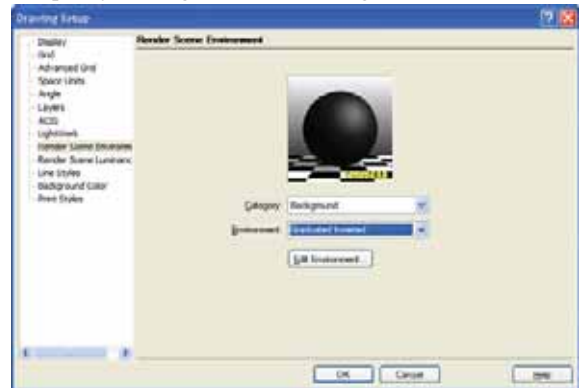
**Use Automatic Number of Threads:** If On the number of threads will be automatically determined.

**Threads:** specifies the number of threads to use if “Use Automatic Number of Threads” is turned off.

## Render Scene Environment

Available in TurboCAD Pro and Platinum only

The options on this page are relevant while rendering. You can specify a background and/or foreground for the model.

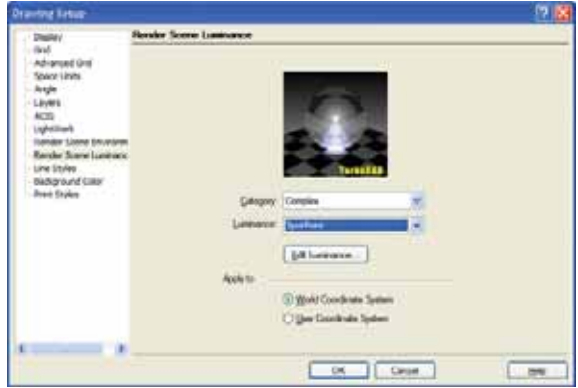


See "Rendering" on page 495 and "Environments" on page 530.

## Render Scene Luminance

Available in TurboCAD Pro and Platinum only

The options on this page are relevant while rendering. You can specify one or more lights to the overall drawing render.



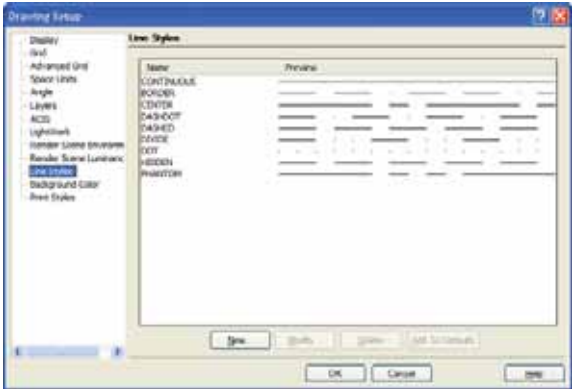
Luminance uses the LightWorks rendering engine, and provides a wider range of effects than standard lights. You can also specify luminance for individual objects (as opposed to the overall drawing) by opening the **Luminance** page of the object's **Properties**.

See "Luminance" on page 518.

Drawing luminance can be applied to the WCS (World Coordinate System) or UCS (User Coordinate System). See "WCS and UCS" on page 105.

## Line Styles

View predefined line styles, modify or delete them, and create new ones.



An object's line style can be set in the **Property** toolbar (see "Property Toolbar" on page 80) or in the **Pen** page of its **Properties** (see "Pen Properties" on page 81).

If you click **New** or **Modify**, you will access the **Line Style Editor**.

**Add to Defaults:** New line styles will be saved so that they can be used in future drawings. Otherwise the line styles will be used only in the current drawing.

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*TIP: You can use the TC Explorer Palette to view line styles defined for any open drawing. See "Line Styles" on page 96.*

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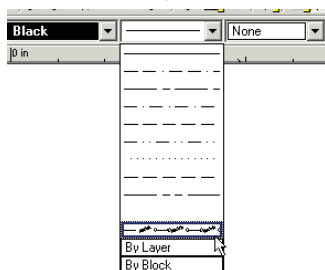
## Applying a Line Style

To apply the line style, you can use the **Property** toolbar. You can also set the line style, and adjust its scale, in the **Pen** page of the object's **Properties**. See "Pen Properties" on page 81.

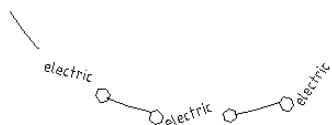
1. Create the object, which appears in the default line style.



2. Select the desired line style from the **Property** toolbar.



The line style is applied.




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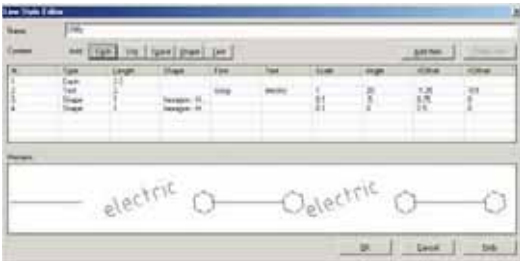
**TIP:** You can use the *TC Explorer Palette* to view line styles defined for any open drawing, and to import line styles from one drawing to another. See "Line Styles" on page 96.

---

Line Style Editor

Available in TurboCAD Pro and Platinum only

Enables you to edit an existing line style or define a new one.

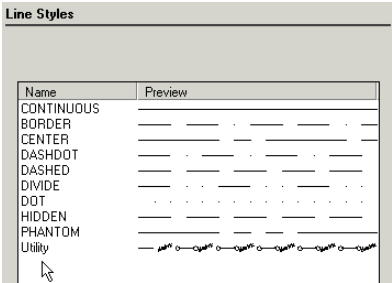


The example shown above (line style “Utility”) has four components: one dash, one text, and two shapes (hexagons). You can also add dots and spaces. Either click **Add Item** to add another component, or click the relevant button (**Dash**, **Dot**, etc.)

For each type of line style component you can set various parameters. For **Shape** you can choose the actual shape and its size and spacing. For **Text**, you can choose the font and angle. The **Preview** window helps you see how the line style will look; use it as a guide when adjusting the parameters.

NOTE: **Shapes** are read from \*.shx files (not to be confused with \*.shx font files), found in TurboCAD’s Shapes folder.

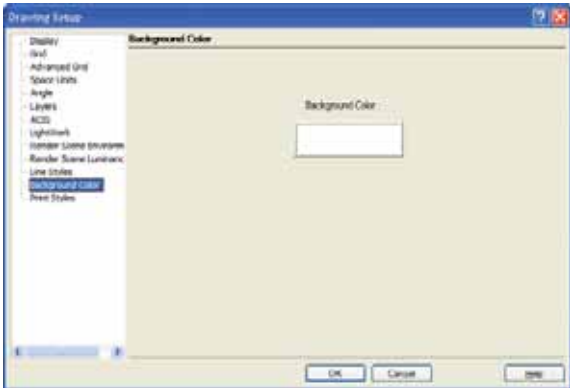
Once the line style has been created, it appears in the **Line Styles** page of the **Drawing Setup**.



The new style also appears on the **Property** toolbar.

Background Color

Changes the background color, which is white by default.



Click in the rectangle on this page to invoke the **Background Color** window, from which you can select the new background color.

Print Style Table

MSelect the print styles you want in your drawing.



Print styles are specific pen and brush settings you can apply to objects when printing. See "Print Style Options" on page 667.



## Working with Multiple Windows

TurboCAD uses a multiple document interface (MDI). This means that the TurboCAD application window can contain multiple windows, each of which contains a TurboCAD drawing. Whenever you open a drawing or start a new drawing, the drawing will appear in its own window within the TurboCAD application window. You can also use multiple windows to display varying views of the same drawing.

One advantage of the multiple document interface is that it allows you to intuitively move and copy objects from one drawing to another.

The window commands can be accessed from **Window** menu or from the **Window** toolbar, opened by right-clicking on any toolbar and selecting **Window**.

### Creating and Displaying Windows

These commands apply to all open windows, whether or not they belong to one drawing or multiple drawings.

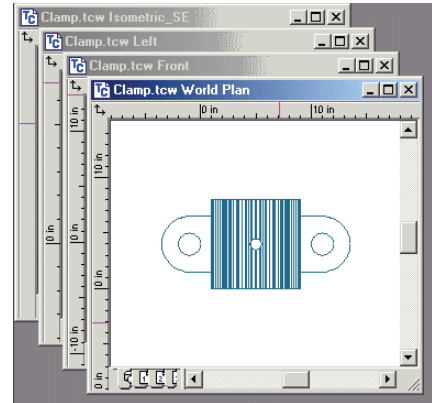
#### New Window

Opens another window for the currently active drawing. The contents of this window are identical to those of the active window, but are displayed in **Zoom All** (zoomed so that all objects are visible in the window).

If the current window is maximized, the new window will also be maximized.

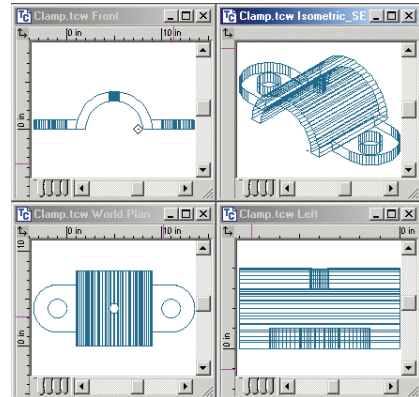
#### Cascade

Resizes and arranges all windows so they overlap, with their title bars showing.



#### Tile

Resizes and arranges all windows so that they are all visible, do not overlap, and are laid out to fill the screen area.



#### Arrange Icons

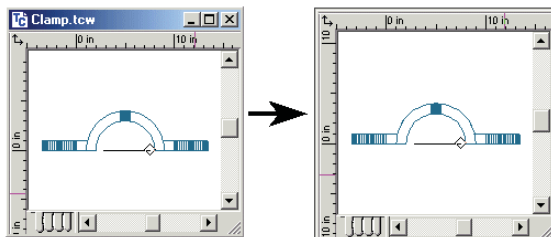
When you minimize a drawing window, Windows shrinks it to an icon. Use **Arrange Icons** to line up these minimized drawing windows along the bottom of the TurboCAD window. (The command does not affect windows that have not been minimized.)

## Close All

Closes all open windows. You will receive a warning if any file has unsaved changes.

## Hide Caption

Removes the title bars from open windows.



## Auto Resize

Preserves the relative proportion of windows when the TurboCAD window is resized.

## Auto Align

You can resize a group of windows if the following is valid for every window: one side of the window lies on a line common to the entire group. This is the line of alignment. The aligned windows can be on the same side of the line of alignment or occupy both sides of it. When placed on opposite sides, the aligned windows may touch each other or not. Occupying the same side of the line of alignment, windows may overlap each other or be separated by space. In all these cases, whenever you catch the side of a window "belonging" to the line of alignment and then drag the window to change its size in the direction perpendicular to the line of alignment, all the windows "belonging" to the same line of alignment will change their size accordingly.

## Multiple Windows of the Same File

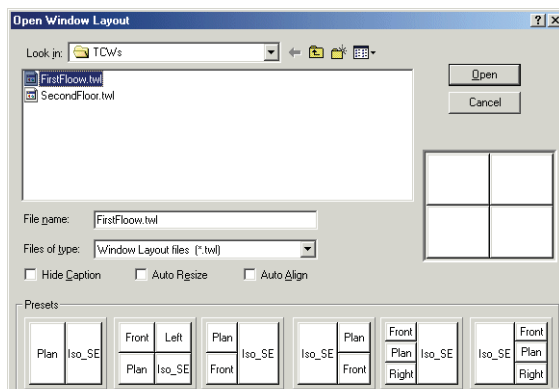
When working on a large and complicated drawing, it is often useful to have simultaneous close-up views of more than one part of the drawing. Not only can you have multiple windows open on the same drawing, you can also start an operation in one window and finish in another.

For example: You may need to draw a line connecting two widely separated objects. Open a second window and tile the windows (**Window / Tile**). Adjust the zoom value in each

window to get a close-up of each item. Then start the line in the first window, and finish it by clicking at the correct location in the second window.

## Window Layout Templates

Enables you to customize the layout of multiple windows of the same drawing.



Several predefined layouts can be selected from the bottom of the window. If you previously saved a template (using **Save Layout Template**), you can browse to select the layout template file.

The **Hide Caption**, **Auto Resize**, and **Auto Align** options can be used to customize some of the window layout properties. These options are also available on the **Window** menu.

You can also start a new session with the last window layout from the last session. Check **Last Window Layout** in the **General** page of the **Program Setup (Options / General)**. See "General Setup" on page 61.

### Saving a Layout to a Template

Saves the current layout as a template you can use for other files. The files are saved with the \*.twl extension.

### Applying a Layout Template

Applies the last template used.

## Moving and Copying Between Different Files

There are two basic methods for copying and moving objects between drawings: copy and paste (or cut and paste), and OLE drag and drop.

### Cut and Paste

You can use the Windows clipboard to transfer objects between drawings.

1. Select the objects in the source document that you want to copy or move.
2. Select **Edit / Copy** (Ctrl+C), or **Edit / Cut** (Ctrl+X). This places a copy of the objects on the Windows clipboard. **Copy** leaves the source objects in place, while **Cut** removes them.
3. Activate the target document by clicking in its window or on its title bar, or by choosing it from the **Window** menu.
4. Select **Edit / Paste** (Ctrl+V). The objects will be pasted from the Windows clipboard into the target document. The pasted objects will appear at the center of the target document window.

### Drag and Drop

OLE drag and drop provides a simple and intuitive way to copy objects between drawings. You can drag objects from one drawing window into another, and you can drag objects stored in block and symbol libraries from their palette into another drawing.

1. Select the objects that you want to drag into another drawing.
2. Place the cursor over the reference point of the selection, and it becomes a double arrow.
3. Drag the selected objects into the window of the target document. An outline of the selection will follow the mouse cursor as you drag. Release the mouse button to place the objects.

---

**WARNING:** *The behavior of snap modes is slightly unusual when you are moving an object using OLE drag and drop. It is therefore not a recommended method for moving objects that require precise placement.*

---

To drag blocks and symbols, simply drag them from their palette into the target drawing window.

## Object Properties

You can set an object's properties *before* or *after* it is created.

### Setting properties for a group of tools (setting properties before creating an object):

Right-click on the tool icon to open the **Properties** window. Or activate a tool, then select **Format / Properties**. Whatever properties you set here will be applied not only to the selected tool, but to *all tools on the same toolbar*. For example, if you set **Line** to be drawn in red, the **Polygon** and **Rectangle** tools will also share this color.

If you change properties of a tool, the new properties will be assigned to all future objects created with the entire set of tools. However, objects already created will not be affected.

---

**NOTE:** *You can save tool properties in template files, so that you don't have to create styles from scratch each time. To do this, set up the properties you want for the tools you commonly use. Then use **File / Save As** to save the file as a \*.tct file (TurboCAD Template). Place the template file in the "Template" folder of the TurboCAD root directory. Then when you want to open the template, use **File / New**, and select **New from Template**.*

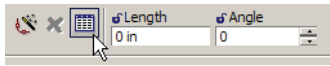
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### Setting properties after creating an object:

You can set certain basic properties, such as color and layer, directly on the **Property** toolbar. For more options, there are several ways to set the remaining parameters:

- With the **Select** tool active, double-click the object.
- Right-click anywhere in the drawing and select **Properties** from the local menu.

- Click the **Properties** icon on the Inspector Bar.



- Open the Selection Info Palette. You can set properties within the table that appears in the palette, or access the object's **Properties** window from within the palette. See "Selection Info Palette" on page 191.

*TIP: You can take properties from one object and assign them to other objects. See "Format Painter" on page 249.*

### Property Toolbar

Provides a quick and convenient way of setting some basic properties of a selected object or group of objects, or for setting the default properties of a group of drawing tools. If the **Property** toolbar is not visible, you can display it using the **View / Toolbars** window.



If no objects are currently selected, the settings on the **Property** toolbar apply to the active drawing tool. For example, if you activate a **Line** tool and then change the settings on the **Property** toolbar, the settings will apply to all objects drawn with any **Line** tool.

If there are objects selected, settings in the **Property** toolbar apply to the selected objects.

*TIP: For properties to appear, the number of selected objects must be less than or equal to **Maximum Multiple Entity Property Count**, which is set in the **Program Defaults** page of the TC Explorer Palette.*

**Entity Style:** Applies predefined property settings to a drawing tool. You can save styles in the **Properties** window. This control can be used only to set the properties of a drawing tool; it is unavailable if you are setting the properties of selected objects.

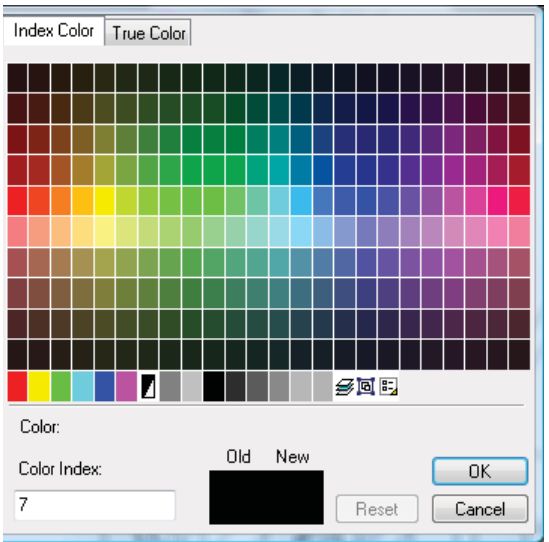
**Layer:** Set the layer on which objects are drawn, or move selected objects to a layer. See "Layers" on page 116.

**Pen Color:** Sets the pen color. There are two types of Pen color available. True Color/RGB and Index color.

For True Colors You can select a predefined color, or define your own using RGB or HSL values. The last three icons in the color thumbnail rows enable you to set the color by layer, by block or open a menu in which you can change how the colors are sorted.



For Indexed colors you can only select the predefined colors.



For more on colors, see "Color Palette" on page 65.

**Pen Style:** Sets the line style of the line. See "Line Styles" on page 74.

**Brush Pattern:** Sets the fill pattern for closed objects. See "Brush Properties" on page 82.

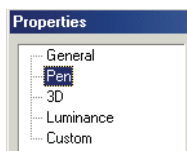
**Pen Width:** Sets the line width.

**Font:** Available for text objects.

**Text Height:** Available for text objects.

## Properties Window

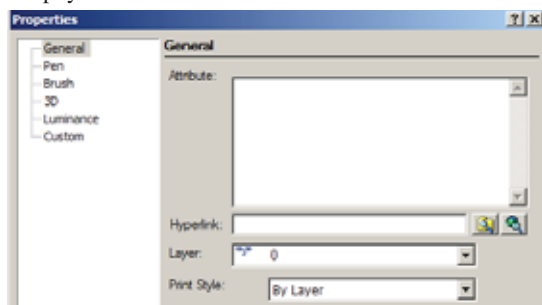
An object's **Properties** window consists of a series of categories, each on its own page. There are categories that are common to all objects: **General**, **Pen**, **3D**, **Luminance**, **Brush** (for 2D objects) and **Custom** - these are all described in this section.



Other categories depend on the selected object. For instance, a sphere has a **Sphere** category, and text objects have a **Text** category.

## General Properties

These are properties generally not related to geometry or other physical characteristics.



**Attribute:** For text and dimension objects, contains the text string. For groups, contains the group name.

**Hyperlink:** Specifies the location of a desired file on your hard disk or on a company's network, or specifies a URL. You can type in the field or browse your system or the web to find the desired path.

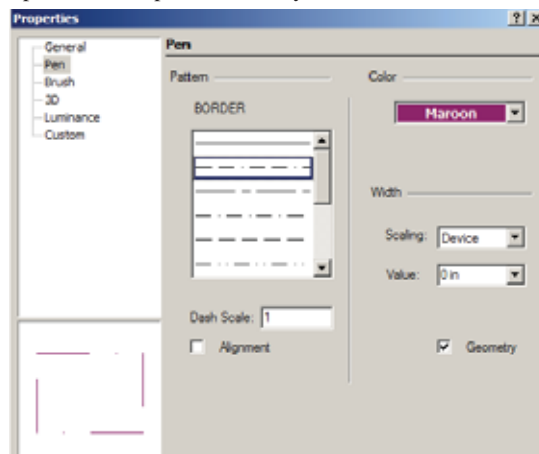
**Layer:** Set the object's layer by selecting from the drop-down box. See "Layers" on page 116.

**Print Styles:** Select the print style you want to use when printing. See "Print Style Options" on page 667.

**Property Value Preset:** A group of properties defined for a group of tools. See "Property Value Presets" on page 84.

## Pen Properties

Properties of the pen and line style.



*TIP: You can also use the Design Director to quickly set the color of an object or group of objects. For single objects, see "Design Director: Graphics" on page 133. For groups of objects, see "Design Director: Categories" on page 134.*

**Pattern:** Select a continuous line or one of many dot-and-dash patterns. If **By Layer** or **By Block** is used, the pattern will depend on the layer or block. There are several predefined line styles to choose from, or you can create your own. See "Line Styles" on page 74.

**Dash Scale:** Scale of the dot-and-dash pattern.

**Alignment:** Align the pattern so that the corners of rectangles and polygons are always solid.

**Width:**

- **Scaling:** Determines whether the width of the line will be scaled, or remain the same size, when you zoom in and out of the drawing.

**Device:** The pen's width and pattern sizes are defined in device units (the monitor or printer). If you zoom the line width and pattern size will not change on the screen.

**World:** The pen's width and pattern sizes are defined in by the drawing spaces units. If you zoom the line width and pattern size will change in accordance with the zoom factor.

**Device Width:** The pen's pattern size is defined by the drawing spaces units, and pen's width is defined in device units (the monitor or printer). If you zoom the line patterns size will change, but the pen width size will not change. **Value:** Pen width. A zero width uses one screen pixel, and will print at one unit of the printer's available resolution (a 300 dots-per-inch printer will print a zero width line at 1/300").

- **Geometry:** Used only when **World** is selected for the Scaling. If checked, TurboCAD creates external and internal "walls" for the object obtained by applying a thickness method to a 2D object. You can render your drawing to see the effects of this option.

## Brush Properties

Fill and hatch properties of objects. A brush pattern can be applied to fill any closed 2D object. Several patterns are provided, but you can use **Brush Styles** (see "Brush Styles" on page 84) to create new hatch patterns, and to also create gradient or bitmap patterns.

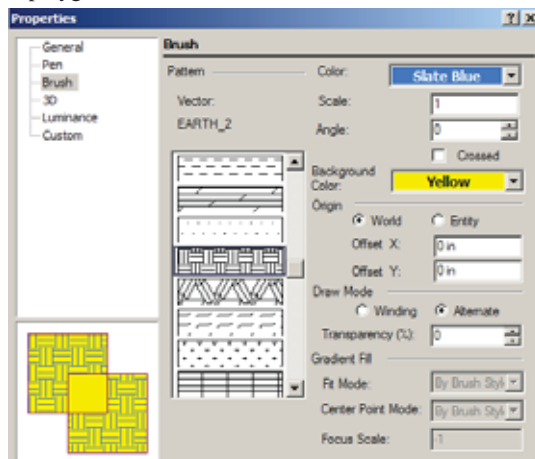
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*TIP: You can use the Colors and Brushes palette to view and all available brushes. See "Colors and Brushes" on page 49. And you can use the TC Explorer Palette to view brush patterns of all open drawings, and to apply brushes. See "Brush" on page 96.*

---

Setting the **Brush** pattern for the group of **Hatch** tools establishes the hatch pattern that will be applied. See "Hatching" on page 356.

These settings are relevant for closed objects such as circles and polygons, and for flexible text.



**Pattern:** Select solid fill, or one of the predefined hatch patterns. To define your own hatch, bitmap, or gradient pattern, see "Brush Styles" on page 84. In the Preview window, the red diagonal line is to help visualize transparency, in the case of a solid or gradient fill.

- **Color:** Set the color of the brush.
- **Scale:** Sets the scale of the hatch pattern. A scale of 2 doubles the size.
- **Angle:** Sets the angle of the pattern.
- **Crossed:** The pattern will be drawn a second time, perpendicular to the first pattern.
- **Background Color:** The color used for empty spaces in the pattern.

**Origin:** By default, hatch patterns use the origin as a point of reference. If you want to modify the placement of hatches without changing their angle, you can modify the point of origin used to place the hatch. **World** uses the WCS, and **Entity** uses the lower left corner of the object being hatched. Use the **Offset** fields to change the reference origin.

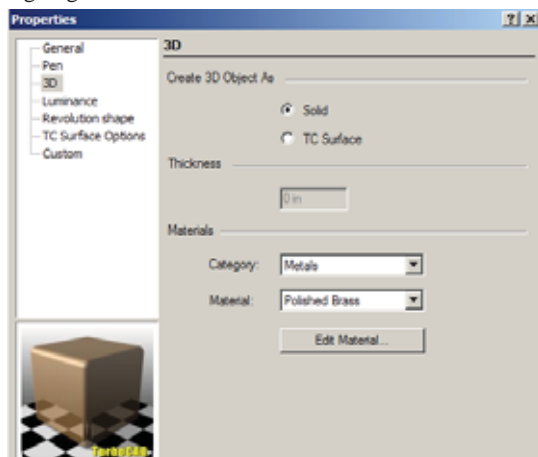
**Draw Mode:** Options for how hatch patterns are drawing over overlapping objects.

- **Winding:** The same hatch pattern will cover areas where filled objects overlap
- **Alternate:** The hatch pattern to be drawn on every other overlapping layer, creating an alternating pattern.
- **Transparency (%):** For solid or gradient fills. A percentage of 0 means the fill is opaque, and 100 means the fill is invisible.

**Gradient Fill:** Relevant for gradient brush styles, which must be defined using **Brush Styles** (see "Gradient Patterns" on page 87). The parameters in this section are the same as the ones already set for the gradient pattern. For **Fit Mode** and **Center Point mode**, use **By Brush Style** to keep the parameters as defined for the pattern. Changing either setting overrides the pattern's settings. For **Focus Scale**, a value of -1 keeps the focus scale the same as the pattern definition. Use a value between 0 and 1 to override it.

### 3D Properties

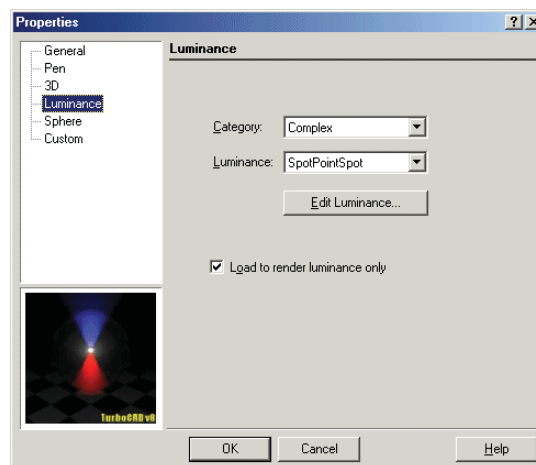
Properties relating to 3D, either for a standard 3D object such as a sphere or box, or for a 2D object made into 3D by assigning it a thickness.



See "3D Properties" on page 371.

### Luminance Properties

Light properties of objects, which is different than the lights contained in the overall model.

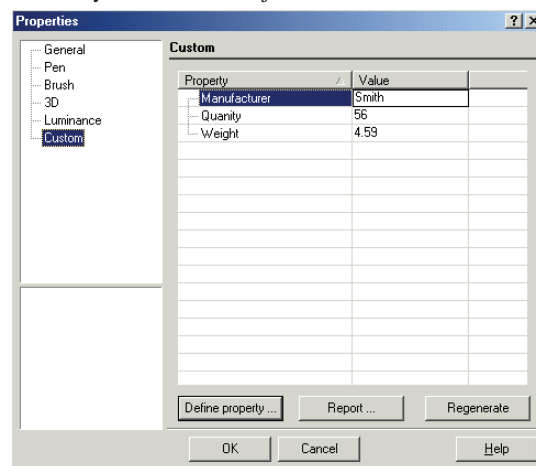


See "Luminance" on page 518.

### Custom Properties

*Available in TurboCAD Pro and Platinum only*

Attributes you can add to objects via the database.



See "Custom Properties, Database, and Reports" on page 624.



## Property Value Presets

A property value preset is a group of properties (**Pen**, **Brush**, etc.) defined for a group of tools. For example, you can have one preset to create blue, dashed lines on Layer 5, and use another preset to create red thick lines on Layer 2. Presets provide a convenient way to quickly assign a set of properties to objects.

Presets can be accessed on the **General** page of an *active* tool's **Properties** window. Presets are only applicable to tools, not objects. You cannot apply a preset to selected objects. You can define a style based on the properties of selected objects by using the Format Painter.

To create a new preset:

1. Right-click on an icon for a group of tools to open the tool's **Properties** window. The tool must be *active* for the **Property Values Presets** field to be editable.

---

*NOTE: Properties set for a tool affect all other tools in its group. For instance, setting **Line** properties affects **Rectangle**, **Polygon**, etc. Presets are separate for different sets of tools.*

---

2. On the **General** page, enter a new preset name and click **New**.
3. Make any changes to the other properties (**Pen**, **Brush**, etc.). The available properties depend on the type of object (for example, the **Properties** window for **Text** tools contains a **Text** page). These properties will be saved to the new preset.

To edit an existing preset, select it from the **Property Values Presets** menu and make any property changes.

## Brush Styles

The **Brush Styles** palette appears by default in the palette area, on the right side of the screen. This tool enables you to modify existing hatch patterns and create new ones. You can also define and edit bitmap and gradient patterns. Brush styles can be used to fill a closed 2D object, either by using an object's **Brush Properties** (see "Brush Properties" on page 82) or by **Hatching** (see "Hatching" on page 356).

---

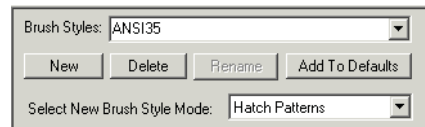
*NOTE: You can use this tool to update existing patterns. Select the hatch, bitmap, or gradient pattern, make the changes, and click **Update Style**. Return to the drawing, and the fill will update in each object where it appears. If the update is not immediate, click the object in **Select** mode.*

---

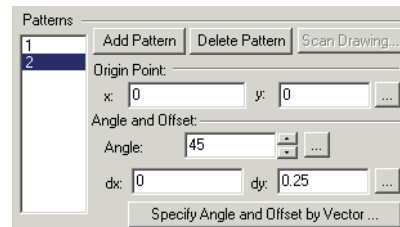
## Hatch Patterns

You can use **Brush Styles** to edit existing hatch patterns and create new ones.

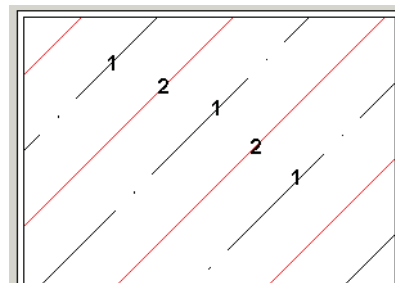
As an example, select **Hatch Patterns** for **Brush Style Mode** and set the **Brush Style** to ANSI35.



This style consists of two Patterns - each pattern is a line that repeats at constant offsets. Highlight each pattern number to see the line highlight in red below, in the Preview area.



In this case, Pattern 1 is the dashed line and Pattern 2 is the unbroken line.

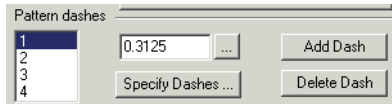


To edit a Pattern line, you can change its **Origin Point** manually, or click the 3-dot icon to select an origin on-screen. You can set the **Angle** and **Offset** values the same way. If you click **Specify Angles and Offset by Vector**, you

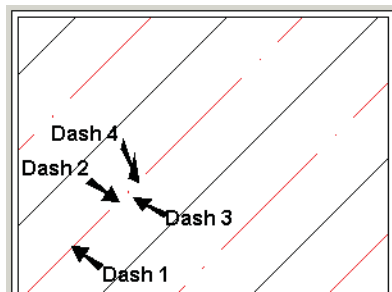


can define two vectors on-screen: the first defines the angle of the line, and the second defines the spacing between repeating lines.

Highlight Pattern 1 (the dashed line), and you can see that it consists of four **Pattern dashes**.



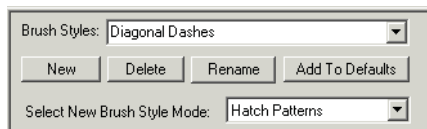
Dashes are always defined in pairs; odd numbers represent the line segments and the even numbers represent the dashes. The length value for dashes is always negative.



**Specify Dashes** is used to create a dashed pattern from an unbroken line. Use **Add Dashes** and **Delete Dashes** to modify the dash pattern. You can update pattern lengths manually or use the 3-dot icon to define length on-screen.

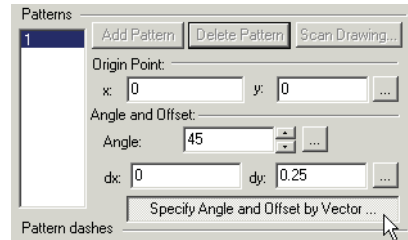
The following example shows how to define a new hatch pattern.

1. At the top of the palette, click **New** and enter a new name for the hatch pattern.

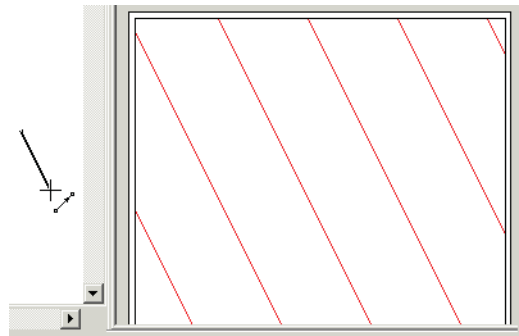


2. The new pattern will be based on whatever pattern was active when the new pattern was created (ANSI35 in this case). Click **Delete Pattern** so that only one pattern line remains.

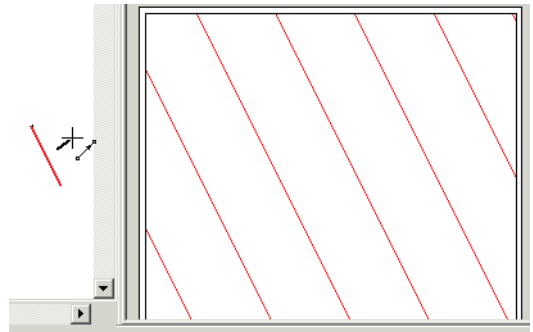
3. Click **Specify Angle and Offset by Vector**.



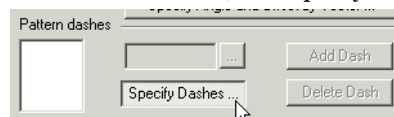
4. Draw a line on-screen that has the angle you want in the pattern. You can snap to existing points or draw a free-form line.



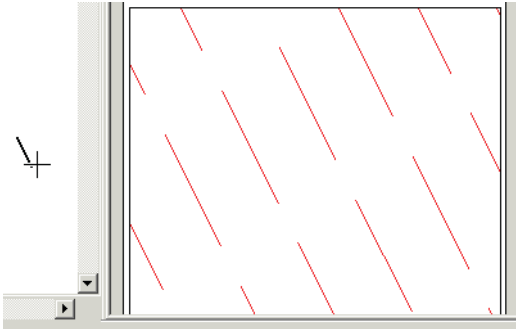
5. Next, draw a line on-screen that defines the offset between repeating lines.



6. To make this a dashes line, click **Specify Dashes**.



7. The first line on-screen defines the length of the line segment of the dashed pattern. Be sure to define this length in the same direction you used to create the original pattern line.

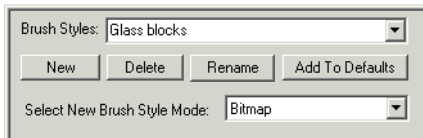


8. The next vector defines the length of the dash, resulting in two dash patterns. If you want more dash patterns, always in pairs, continue defining vectors in the same direction.
9. Create more patterns and dashes as needed to get the entire pattern.

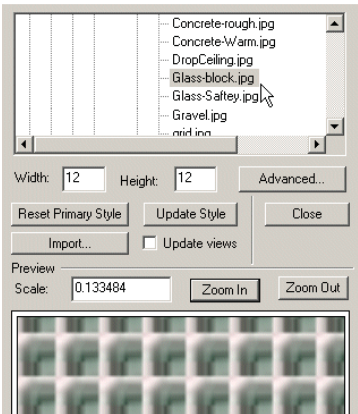
When the style is defined, click **Update Style** to use it in the current drawing, or **Add to Defaults** to be able to use it in future drawings.

**Bitmap Patterns**

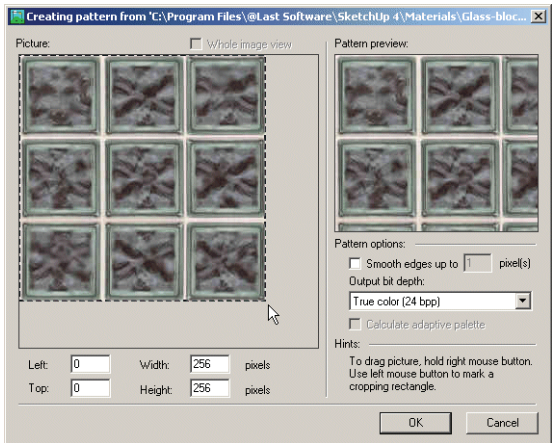
To create a brush style from an image, select **Bitmap** for **Brush Style Mode** and then click **New**. Enter the name of the style at the top of the palette.



In the middle section, use the browser to locate the image file you want to use. A preview of the image appears in the lower section. The size of the repeating image is controlled by the **Width** and **Height** values. You can change these values manually, or click **Advanced** for more control.



When you click **Advanced**, the following window appears. In the left window, use the dotted border to set the section of the image that will repeat. The tiled pattern updates dynamically under **Pattern Preview**. You can also set the border by entering values for **Left**, **Top**, **Width**, and **Height**.

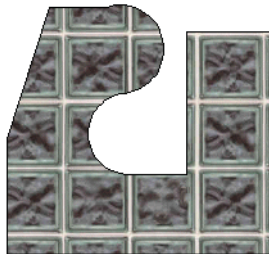
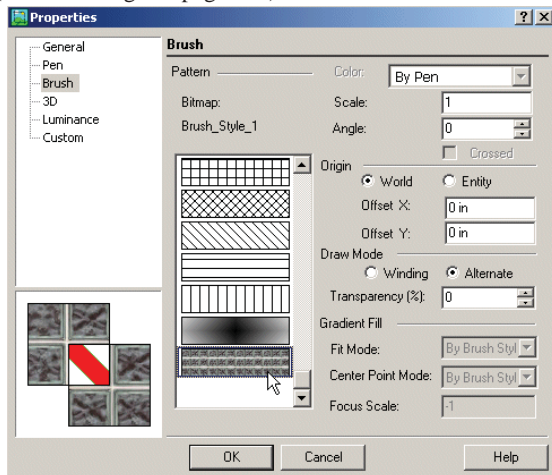


- **Smooth edges up to *n* pixels:** Smooths the transition between bitmap repetitions by blending adjacent edges.

- **Output bit depth:** The number of bit used for the bitmap.
- **Calculate adaptive palette:** Forces the color down-sampling to match the original color palette as closely as possible. Available only if the **Output bit depth** is lower than the value which the original bitmap used.

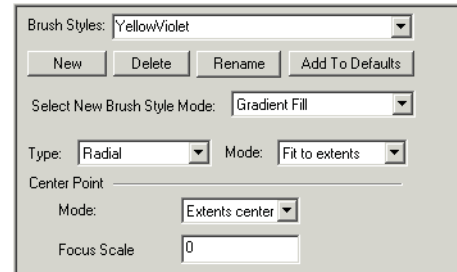
When the style is defined, click **Update Style** to use it in the current drawing, or **Add to Defaults** to be able to use it in future drawings.

Now when you open the **Properties** (see "Properties Window" on page 81) for a 2D closed object, the bitmap style appears in the **Brush** page (see "Brush Properties" on page 82). The style also can be used in the **Hatching** tools (see "Hatching" on page 356)



## Gradient Patterns

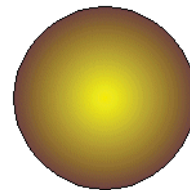
To create a gradient style, in which the fill changes gradually from one color to another, select **Gradient Fill** for **Brush Style Mode** and then click **New**. Enter the name of the style at the top of the palette.



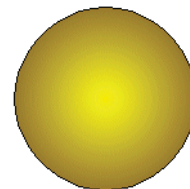
There are four types of gradient fills: **Linear**, **Radial**, **Reflected**, **Diamond**, and **Custom**. Each type is explained later in this section.

**Mode** (called **Fit Mode** in the **Brush Properties**):

- **Exact Fit:** The gradient completely fills the object itself. In this example, the fill completely reaches the second gradient color at the boundary of the circle.

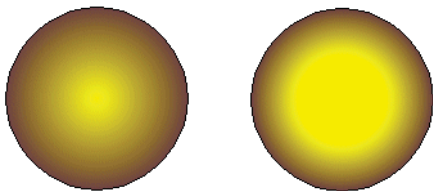


- **Fit to Extents:** The gradient completely fills the extents rectangle that encloses the object. In this example, the second gradient color is only reached at the boundary of the extents rectangle, which is larger than the circle itself. This is why the color at the circle boundary is lighter than in the example above.



Center Point:

- **Modes:** Select **Extents center** to center the fill at the center of the object's extents rectangle. **Reference point** centers the fill at the object's reference point (see "Components of Select Edit Mode" on page 199).
- **Focus Scale:** This must be a value between 0 and 1. Zero means the color interpolation start from the center (left image below). A value of 0.5 will start the interpolation halfway between the center and the edge (right image below).

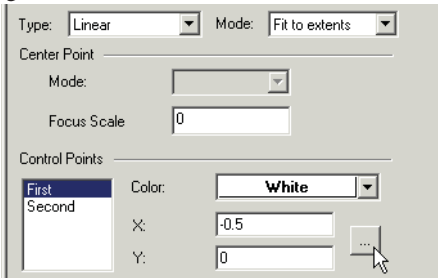


NOTE: Settings for **Fit Mode**, **Center Point**, and **Focus Scale** are set here as the fill defaults. These values can be overridden for filled objects by modifying the object's **Brush Properties** (see "Brush Properties" on page 82).

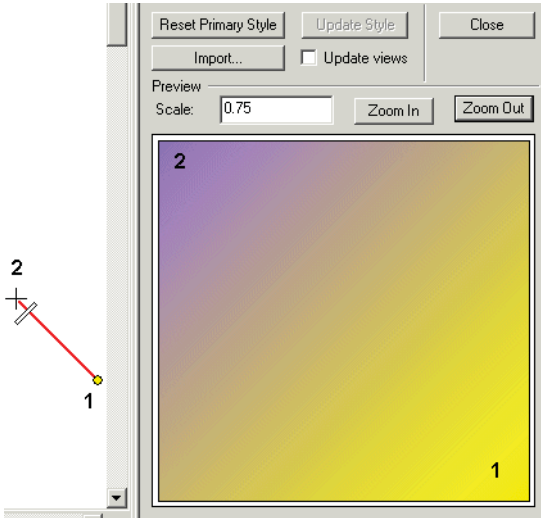
Linear Gradient

Linear fill moves from one color to the other along a straight vector. Under **Control Points**, highlight **First** and set the **Color**. Do the same for **Second**.

By default, the **First** color runs vertically along the left side and changes to the **Second** color along a left-to-right vector. To change this vector, click the icon with the three dots.



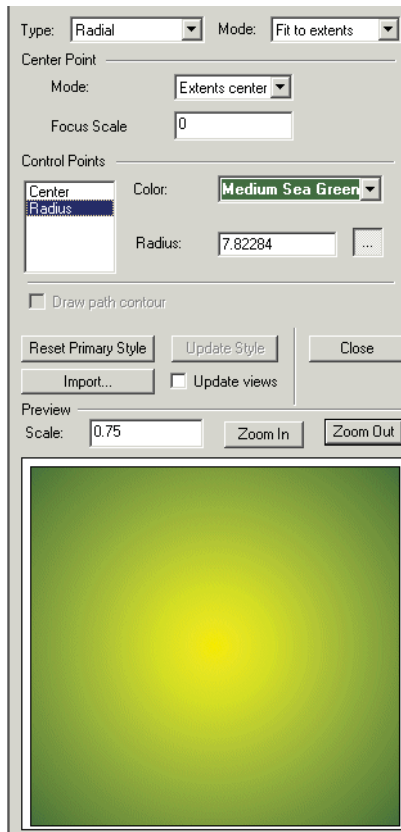
Define the new vector on the screen. The first point of the vector controls where the **First** color appears.



Radial Gradient

Radial fill moves from one color to the other along a radial path. Under **Control Points**, highlight **Center** and set the center color. Highlight **Radius** and set the outer color.

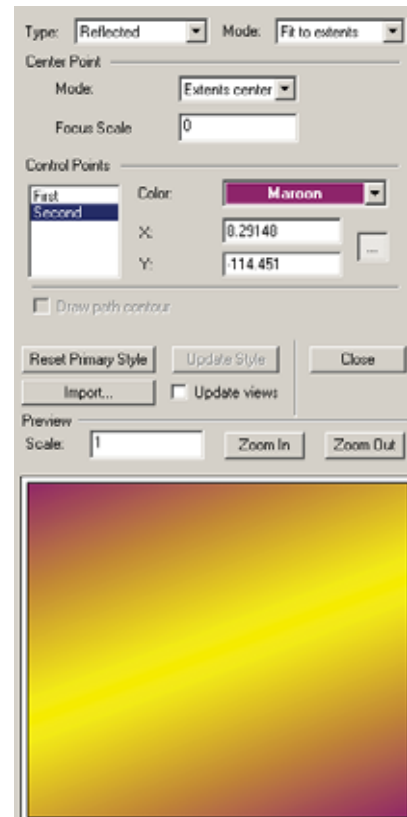
Set the radius manually, or click the 3-dots icon and set the radius on-screen.



## Reflected Gradient

Reflected fill consists of one line of reflected color, fading to another color on either side. moves from one color to the other along a radial path. Under **Control Points**, highlight **First** and set the reflective color. Highlight **Second** and set the fade color.

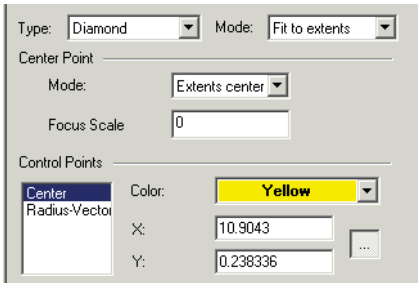
Set the angle of the reflection line manually, or click the 3-dots icon and set the direction on-screen.



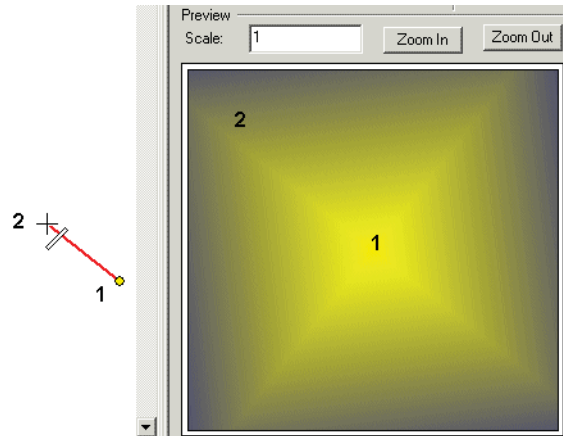
Diamond Gradient

Diamond fill moves from from the center outward in four directions, each separated by 90 degrees. Under **Control Points**, highlight **Center** and set the color of the center and linear patterns. Highlight **Radius-Vector** and set that appears in between the **Center** color lines.

By default, the **Center** color runs vertically and horizontally to either side. To change this vector, click the icon with the three dots.



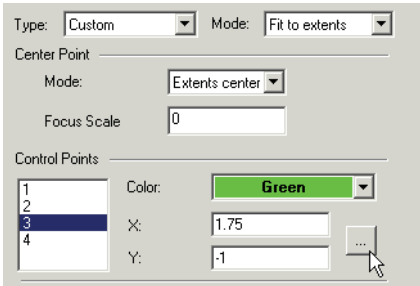
Define the new vector on the screen. The first point of the vector controls where the **Center** color appears. The second point of the vector controls the angle of the four lines.



Custom Gradient

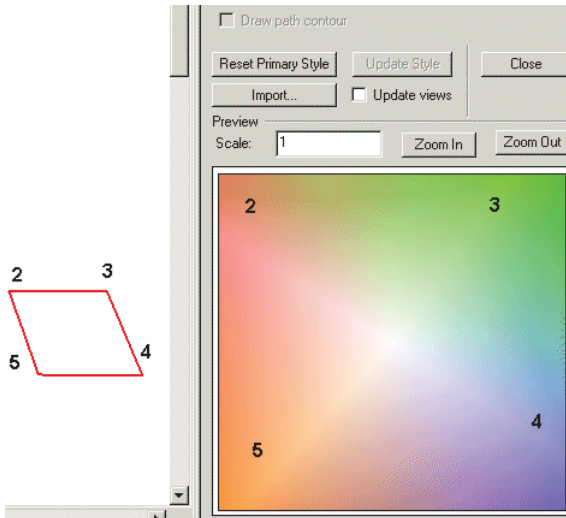
Use this type to create a custom polyline and apply colors to each polyline point. The results will resemble a **Diamond Gradient**, but can have numerous points, each with its own color.

**Color 1** runs from the center outward to each point. By default there are three other points (triangular polyline). To change the polyline, click the icon with the three dots.



Define the polyline on the screen, proceeding in the order of **Color 2**, **Color 3**, and so on. The polyline automatically closes, and cannot intersect itself. Select **Finish** from the local menu to complete the polyline. You can also select an existing polyline from the drawing; click the arrow icon in the Inspector Bar to do this.

If the polyline has more than three points, the number of **Control Points** increases, and you can set a color for each point.



When the style is defined, click **Update Style** to use it in the current drawing, or **Add to Defaults** to be able to use it in future drawings.

## TC Explorer Palette

*Available in TurboCAD Pro and Deluxe only*

This palette enables you to perform many operations in one place. It serves as a file browser with previews, and displays components and editable parameters for all open drawings.

### TC Explorer Toolbar

**File filter:** Enables you to select which files will be listed for a selected folder of the explorer tree. You can define your own file filters in the **Palette Options** (last icon).



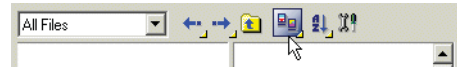
**Back and Forward:** Scrolls backward or forward in the explorer tree.



**Level Up:** Moves up to the higher folder in the explorer tree.



**View:** Determines how the contents of the folder are displayed: icons, thumbnails, detailed list, etc.



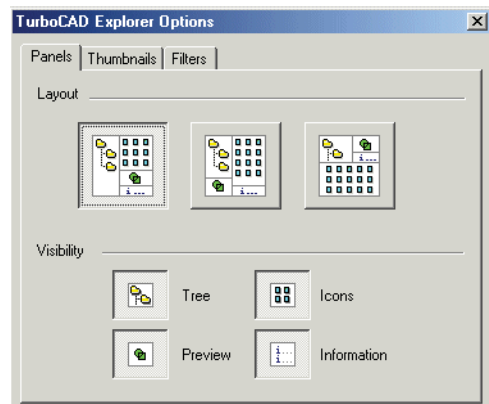
**Sort:** Arranges the contents of the folder by date, size, etc.



**Palette Options:** Sets the palette display and file filters.

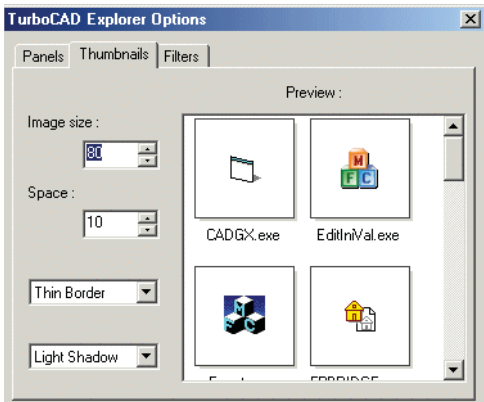


- The **Panel** tab contains choices for panel appearance, and enables you to select what components of the palette are visible.



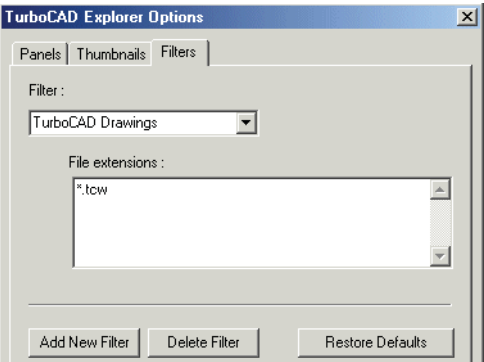


- The **Thumbnails** tab enables you to control the size, shadows, and borders of thumbnails.



**Image Size** and **Space** are both measured in pixels. **Space** refers to pixels around the icon plus its name.

- The **Filters** tab enables you to add, delete, or edit file filters. Filters control which files appear in a folder's contents, and are set according to file extensions.

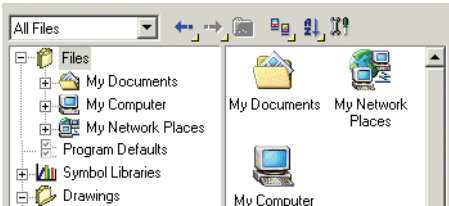


## General Settings

All branches of the explorer tree except the last one (Drawings) enable you to set parameters for the program as a whole.

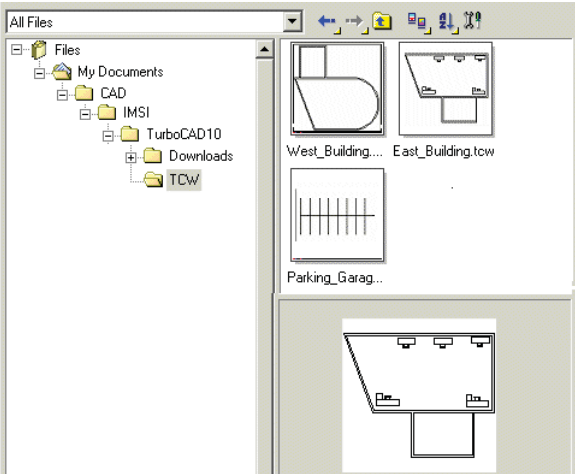
## Files

This branch acts just like Windows Explorer. The files that are listed for a highlighted folder depend on what **Filters** are set. You can list all files, only TurboCAD files, only image files, or you can define your own filter for files of specific extensions.



You can expand a folder by clicking the "+" icon, or by pressing the \* on the numeric keypad.

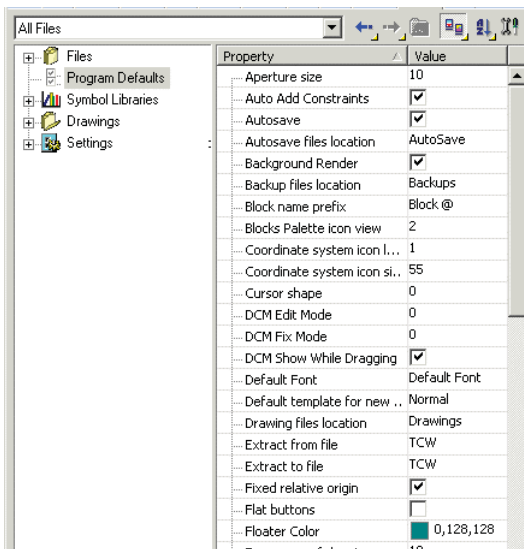
If you highlight a TurboCAD file (\*.tcw) or an image file, a preview of the file will appear.





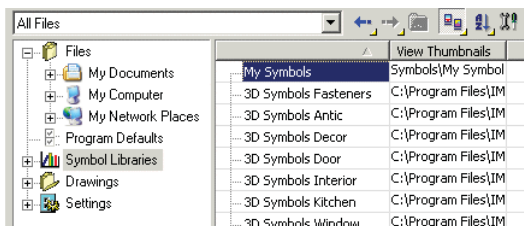
## Program Defaults

Here you can control default program settings - parameters that are active for all files. For example, you can change the aperture size and toggle **Auto Add Constraints** on and off. Most of these parameters can also be set in **Options / Program Setup**.



## Symbol Libraries

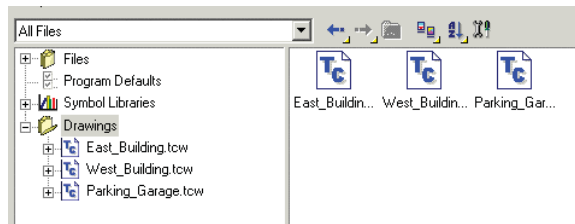
Enables you to change the path for user-defined symbols. You can also drag symbols as you would from the Library palette.



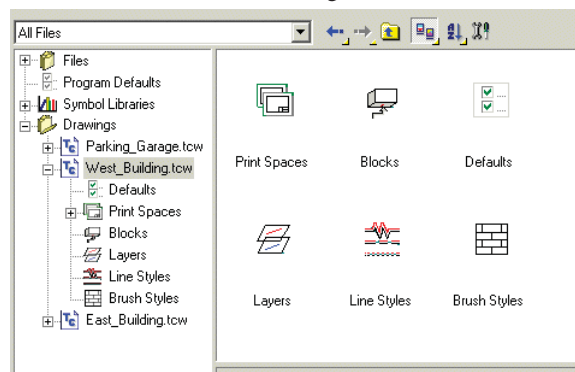
See "Library Folders" on page 305.

## Drawing Settings

The **Drawings** branch of the tree enables you to view components of all open drawings. Click the **Drawings** header to see a list of all open drawings.

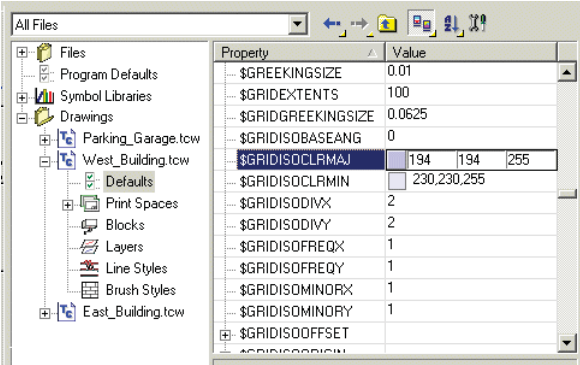


For each drawing there are six components you can view, edit and/or share between drawings.



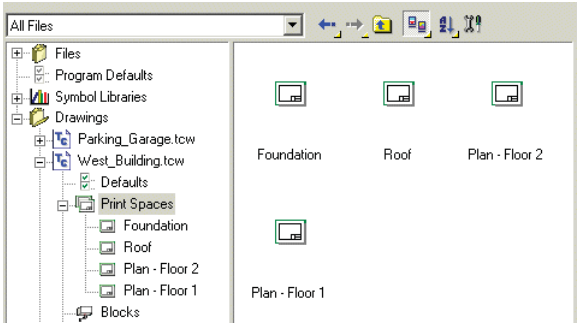
Drawing Defaults

Here you can control default drawing settings - parameters that apply onto to the relevant drawing. For example, you can change the colors of the major and minor grid lines. Nearly all of these parameters can also be set in **Options / Drawing Setup**.

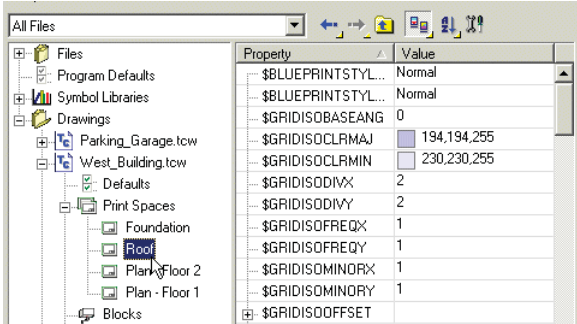


Print Spaces

Lists all paper spaces in the drawing. (See "Paper Space" on page 641.)

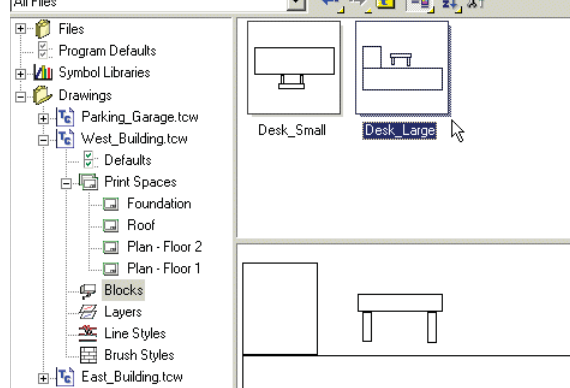


If you highlight a Paper Space, you can edit its properties.



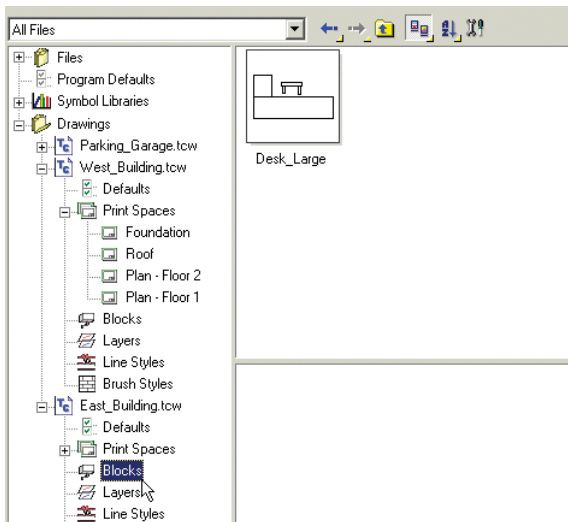
Blocks

Displays all blocks defined in the drawing (see "Blocks" on page 291). Highlight a block's icon or thumbnail to see a preview of the block's contents.



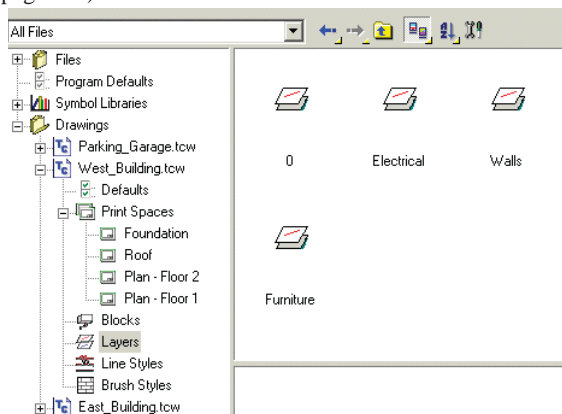
You can use the palette to insert blocks from one open drawing into another open drawing. For example, the West\_Building.tcw file contains two blocks. If you open East\_Building.tcw, drag the Desk\_Large block's icon into

the drawing. The block will then be listed under East\_Building's blocks. It will also appear in that drawing's Blocks palette.



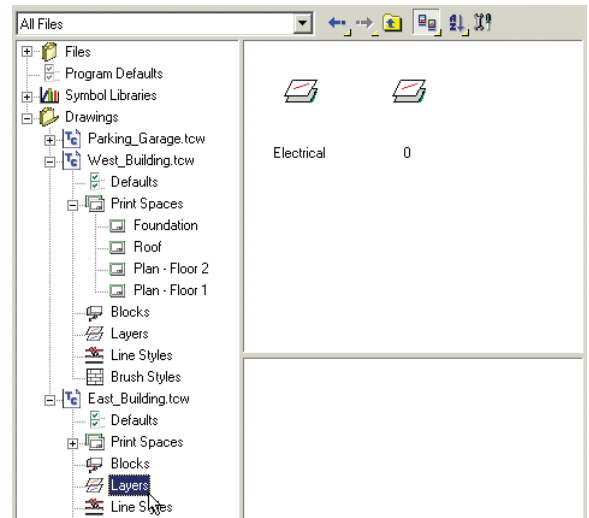
## Layers

Displays all layers defined in the drawing (see "Layers" on page 116).



You can use the palette to import layers from one open drawing into another open drawing. This imports a layer's name, color, style, width, etc., but does **not** import a layer's objects.

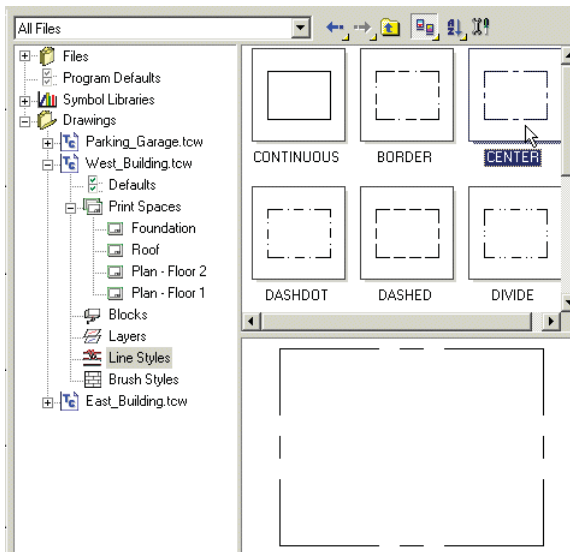
For example, the West\_Building.tcw file contains four layers. If you open East\_Building.tcw, drag the Electrical layer's icon into the drawing. The layer will then be listed under East\_Building's layers, and will appear in **Options / Layers** as well.



To change an object's layer, select the object and then double-click the desired layer's icon. If you double-click a layer belonging to another drawing, that layer will automatically be imported into the current drawing. The active drawing tool will automatically be placed on the double-clicked layer.

## Line Styles

Displays all line styles defined in the drawing (see "Line Styles" on page 74). Highlight a line styles' icon or thumbnail to view it.



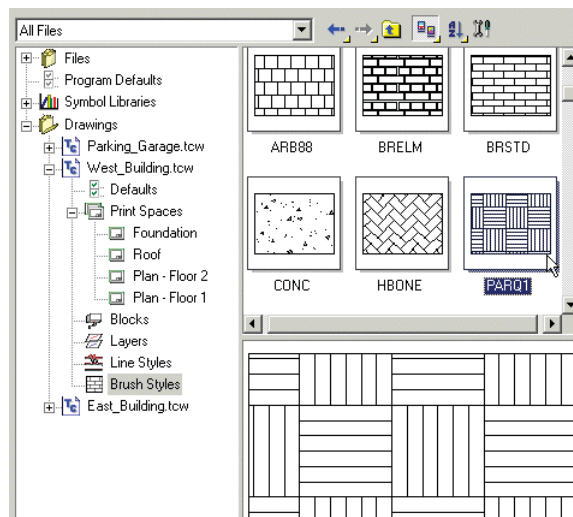
To import a line style from one open drawing into the current drawing, drag and drop or double-click its icon.

To change an object's line style, select the object and then double-click the desired layer's icon. If you double-click a line style belonging to another drawing, that line style will automatically be imported into the current drawing.

You can also make any line style active by double-clicking its icon. All new objects will be created in the current line style. You can also change line styles while creating objects; for example, you can select one corner point of a rectangle, change the line style, and complete the rectangle.

## Brush

Displays all brush (hatch) styles defined in the drawing (see "Brush Properties" on page 82). Highlight a line styles' icon or thumbnail to see how the line style appears.

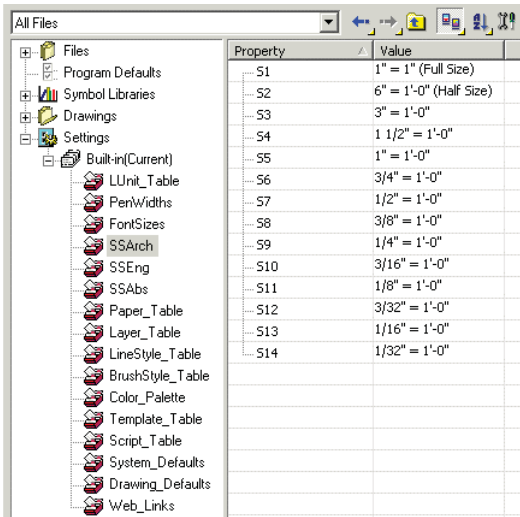


To change an object's brush, select the object and then double-click the desired brush's icon.

You can also make any brush active by double-clicking its icon. All new objects will be created in the current brush. You can also change brushes while creating objects; for example, you can select one corner point of a rectangle, change the brush, and complete the rectangle.

## Settings

Lists parameters such as units, fonts, and tables currently defined in the drawing.



One of the more important of the drawing defaults is the \$MIRRTEXT under Drawing Defaults. This setting controls. When the mirror tool is used block attributes are not flipped if the drawing variable \$MIRRTEXT is set to 0, and are flipped if it is set to 1. \$MIRRTEXT is set to 0 by default.



# 3 Manipulating the 2D Display

This section covers ways to obtain the desired view of your model, and how to save views for future use.

## Views

The view is the area of the drawing space that appears on your screen. As your drawing becomes larger and more complex, it's important to find views that let you work at the right location in your drawing, at an appropriate level of detail.

---

*Tip: Layers are a useful way to change how objects. You can place objects on various layers and then manipulate layer visibility. See "Layers" on page 116.*

---

This section covers views in 2D. When working in 3D, there are additional viewing tools and other ways to examine your model. See "3D Views" on page 361 and "Examining the 3D Model" on page 419.

## Zooming

Zooming means to move "closer to" or "farther from" the drawing space, viewing the drawing at a greater or smaller level of detail.

You can always check the current zoom percentage via the zoom indicator, located, by default, at the lower right corner of the screen.



## Using a Wheel Mouse

Zooming with a wheel mouse is a particularly handy way to get around in a drawing. Move the cursor to where you want to work, and move the wheel backward to zoom out or forward to zoom in.

Some wheel mouse setups require that the scroll bars be turned off for zooming to work.

Some wheel mice are set up by default to scroll instead of zoom. If you would like your wheel to zoom, select **Tools / Customize** and open the **Intellimouse** page. You will be able to assign commands to the wheel functions. This feature works with other wheel mice as well.

## Zoom In and Zoom Out

**Hotkey:** <+> or <-> (on numeric keypad)



Magnifies or shrinks the current view. The zoom factor is equal to two by default, but you can change this in the **Preference** page of the **Program Setup**.

---

*TIP: Change the zoom factor to 1.4 to double the view every two zoom steps.*

---

If you zoom using the <+> and <-> keys of the numeric keypad, the view centers on the cursor. If you use the **View** menu commands or icons, the zoom will center on the current view, regardless of the position of the cursor.

---

*NOTE: As you zoom out, the representation of small objects is simplified, thereby increasing redraw speed. You can set the size at which this occurs using the **Suppress display** field on the **Display** page of the **Drawing Setup**.*

---

## Zoom Window

**Hotkey:** Ctrl+Home



Zooms to an area defined by a rectangle. The rectangle is defined by selecting its two opposite corners, or by dragging the mouse from one corner to the other.

---

*NOTE: Because the dimensions of the zoom rectangle will probably not be exactly proportional to the screen, the closest view that includes the entire rectangle will be used.*

---

## Zoom Extents

**Hotkey:** Ctrl+Backspace



Zooms to an area that completely contains all visible objects (including lights and camera objects). This is the same area enclosed by the selection rectangle if you select all the objects.

Any objects on an invisible layer will not be taken into consideration.

## Zoom Full View

**Hotkey:** Shift+Backspace



Only available in Paper Space, displays the entire drawing sheet. The drawing sheet may consist of multiple printer paper sheets, indicated by dashed lines.

## Zoom Selection

**Hotkey:** Alt+Backspace



Zooms to an area that completely contains all currently selected objects.

---

*TIP: You can also use the Design Director to quickly zoom on an object or group of objects. For a single object, see "Design Director: Graphics" on page 133. For a group of objects, see "Design Director: Categories" on page 134.*

---

## Printed Size

Zooms at approximately the size the drawing will be when printed. In this view, one inch of screen space is approximately equal to one inch on paper. The exact ratio of screen space to Paper Space will vary, depending on your graphics display and screen resolution.

## Uniform for All Views

Used for multiple windows of the same drawing, enables you to zoom all windows simultaneously.

## Panning

Panning means to move, or scroll, the view to another location across the plane of the drawing.

Two simple ways of panning are to use the scroll bars or arrow keys. The scroll bars work the same way that they do in other Windows programs. You can click the scroll bar arrow buttons to scroll one step, click in the body of the scroll bar, or drag the body of the scroll bar itself.

The arrow keys are also simple and intuitive: press the up, down, left, or right arrow key to pan a single step.

---

*TIP: To quickly pan to a specific point, place the cursor at the point, then press Ctrl+End. You will instantly move to a view with the point you chose at the center of the screen.*

---



## Vector Pan



Pans the drawing based on a vector defined by two points.

Select a base point and a destination point. The drawing moves so that the base point moves to the destination point.

## Pan to Point



**Hotkey:** Ctrl+End

Pans the drawing so that the selected point becomes the drawing center.

If you use the **View** menu command, select a point and the point will move to the center of the screen.

To use the hotkey, place the cursor at the desired center and press Ctrl+End.

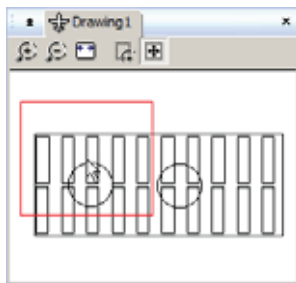
## Aerial View



Opens an additional window to provide a second view of the drawing. You can use **Aerial View** to create a close-up view of one small part of the drawing, or of the entire drawing. **Aerial View** is especially useful if you do not have a large monitor.

You adjust the view in the drawing screen, without changing the aerial view.

Use a rectangle to define the area you want to include in the aerial view. This view appears in the small drawing window.



The red rectangle represents what appears on the main drawing window.

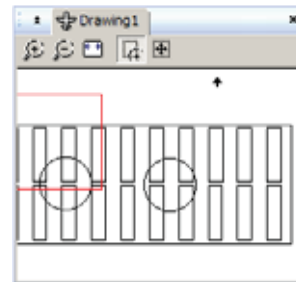
The first three tools in the aerial view window control the view *inside* the aerial view:

- **Aerial Zoom In:** Magnifies the aerial view.
- **Aerial Zoom Out:** Shrinks the aerial view.
- **Aerial Zoom Extents:** Zooms to an aerial view containing all objects in the drawing.

The last two tools are used inside the aerial view window, but affect the overall drawing.

- **Aerial Zoom Window:** Sets a new drawing view by using a selection window in the aerial view.
- **Aerial Pan View:** Sets a new drawing view by moving the red rectangle that appears inside the aerial view window.

You can also pan the aerial view with the cursor. Place the cursor just inside one of the borders of the aerial view window. After a second or two, the cursor will change to a small solid arrow and the aerial view will move in the indicated direction.



Move the cursor away from the edge to end panning.

## Redrawing

**Hotkey:** F5

In most cases, the screen will be redrawn after each change. Occasionally, however, you may find that the screen is not completely "cleaned up" after an operation. You can manually redraw the screen using the **Redraw** command.

## Saving Views

Once you have used the zoom and pan tools to arrive at the exact view of your drawing that you want, you won't want to repeat all that work to get back to the same view later. You can save views and assign them names so that you can easily return to a saved view at any time.

In addition, saved views must be created if you want to create viewports in Paper Space. See "Viewports" on page 657.

---

*NOTE: For 3D views, you can also use camera object to save and display views in separate windows and with specific parameters. See "Camera Objects" on page 422.*

---

The **Named Views** tools are accessible from the **View** menu, or below the **Insert Viewport** icon on the **Standard** toolbar.



You can display the **Named Views** toolbar by right-clicking in any toolbar area and select **Named View**.



---

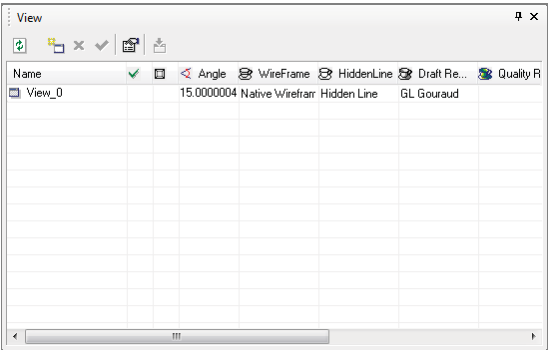
*NOTE: Saved views can also be accessed and manipulated in the Design Director, in which you can also create new views. Render and camera settings can be manipulated here as well. See "Design Director: Views" on page 131.*

---

## Named View



The **Named View** window can be used to create a new saved view, or to display, modify, or delete a saved view.



Named views are saved with your drawing and remain available unless you delete them.

**Draft Rendering:** Sets the rendering type when the view is rendered in **Draft** rendering mode. See "Creating a Rendered View" on page 495 and "Draft Render Properties" on page 499

**Quality Rendering:** Sets the rendering type when the view is rendered in **Quality** rendering mode. See "Quality Render Properties" on page 500

**Perspective:** Activates **Perspective** mode. See "Camera Properties" on page 420.

**Angle:** The view angle for perspective mode.

**Position:** The location of the camera position.

**Target:** The location of the point the camera is facing.

**UP Vector:** A point defining the up direction of the camera.

**Extents:** The boundaries of the view.

## Saving a View

1. Use the zoom and pan tools to arrive at the view you want to save.

---

**NOTE:** If you want to create a view using a specific area of the screen, see "Creating a View" on page 103.

---

2. Select **Named View** from the menu or click the **Named View** icon to open the **Named View** window.
3. Enter a name for the view. The name can be up to 32 characters in length and can include spaces.
4. Click **New** to add the name to the list of named views.

---

**TIP:** Give the view a descriptive name that will help you remember it, such as "Garden Door." If you use names like "View 1" and "View 2," you'll quickly lose track of which is which.

---

## Displaying a Saved View

In the **Named View** window, select the desired view and click **Go To**. This will leave the window open, in case you want to switch to another view.

To immediately go to a named view and close the window, double-click on the view name.

---

**TIP:** If you want to display the previous view, use **View / Zoom / Previous View**.

---

## Modifying a Saved View

Use the zoom and pan tools to arrive at the view with which you want to replace a saved view.

Open the **Named View** window, select the view you want to replace with the current drawing view, and click **Modify**.

## Deleting a Saved View

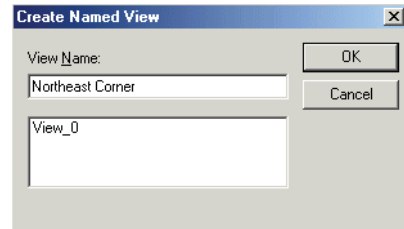
In the **Named View** window, select the view you want to delete, and click **Delete**.

## Creating a View



Saves a defined area of the screen as a named view.

Define the rectangle that surrounds the area you want to save. In the **Create Named View** window, assign a unique name and click OK.




---

**NOTE:** If the drawing is already oriented to the view you want to save (i.e. you do not need to define a view rectangle), you can save the view using the **Named View** tool.

---

## Previous View



Displays the previous view, whether or not it was saved.

---

**NOTE:** If you want to display a saved view other than the previous one, select the view in the **Named View** window and click **Go To**.

---

## 3D View Manipulation

In 3D there are many ways you can change your view. The easiest way is to use one of the standard views - top, bottom, isometric, etc. See "3D Views" on page 361.

. Other tools for view manipulation include:

- **Camera movements:** Incremental controls for adjusting the camera, and therefore your view of the model. See "Camera Movements" on page 421.

- **Walk Through tools:** Used to dynamically move the camera, presenting a moving viewpoint. See "Walk Through Tools" on page 421.
- **Camera objects:** Used to create and save views with specific parameters. The view for each camera object can be displayed in a separate window. See "Camera Objects" on page 422.
- **Rendering:** Enables you to see your 3D model as shaded. Certain render modes also enable you to view texture and materials. See "Rendering" on page 495.

# 4 Drawing Aids

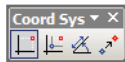
This section covers tools and features of TurboCAD that make your work easier and more organized, such as grid, snaps, layers, and construction geometry.

## Coordinate Systems

There are several types of coordinate systems you can use, and you can switch between them at any time.

For example, when drawing the outer wall of a house, you may want to start the first wall at an absolute location. Each successive wall, however, will be defined by its length and angle relative to the first wall, so you would use polar coordinates for these points. To place walls at an X, Y distance from any other point, you could use relative coordinates.

You can display the **Coord System** toolbar by right-clicking in any toolbar area and selecting **Coord System**.



Coordinates, when entered manually, are entered in the Coordinate Fields, at the lower right corner of the screen. See "Coordinate Fields" on page 46. You can press Shift+Tab to jump to the first field in the Coordinate Fields, then press Tab to scroll through the remaining fields.

*TIP: If you precede a coordinate with a \$ sign, it will be interpreted as an absolute coordinate; if you precede it with an @ sign it will be interpreted as a relative coordinate; if you precede it with a > sign it will be interpreted as a polar coordinate.*

Coordinate systems behave the same way in 2D and in 3D, but in 3D you need to be familiar with the concept of workplanes as well. See "3D Coordinate Systems" on page 362 and "Workplanes" on page 362.

## WCS and UCS

The WCS (World Coordinate System) is an internal, absolute coordinate system. WCS axes can be displayed from **Program Setup**, and they appear at the lower left corner of the screen. The WCS indicator does not represent the origin; it indicates orientation.

The UCS (User Coordinate System) is the system in which you are working. By default, it is located to coincide with the WCS, but it can be moved. UCS axes can be displayed from **Program Setup**, and they appear wherever the origin is located.

When using the 3D Selector, the selection shell local menu and Inspector Bar provide two options that relate selected objects to the UCS (see "2D / 3D Selector" on page 181).

- **Set UCS by Selector:** Moves the UCS origin to the selection reference point.



- **Set Selector by UCS:** Moves the selection to the UCS origin (similar to the **Place on WorkPlane** option for 2D objects).



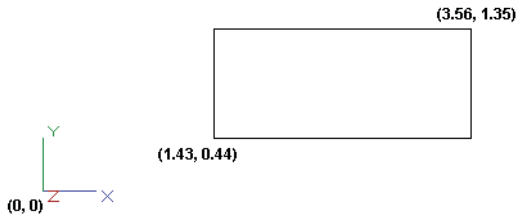
*NOTE: The 2D Selector always moves the UCS (Workplane) to the selection.*

### Absolute Coordinates

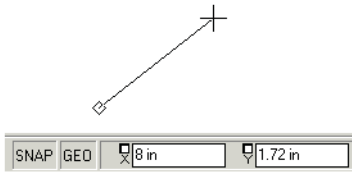
Hotkey: A, Shift+ A



Cartesian (X, Y) coordinates relative to the fixed, absolute (WCS) origin.



You can enter absolute X and Y values in the Coordinate Fields.



### Relative Coordinates

Hotkey: R, Shift+ R

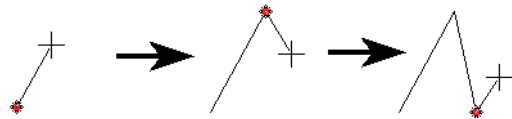


Cartesian (X, Y) coordinates relative to a specified origin.

TIP: If you precede a coordinate with an @ sign it will be interpreted as a relative coordinate.

The relative origin can move as you work, relocating to each selected point, or it can remain fixed. The relative origin is controlled by the **Fixed relative origin** box on the **Preference** page of the **Program Setup**.

The relative origin is marked by a red square. You can use **Relocate Origin** (Shift+L) at any time to move it.

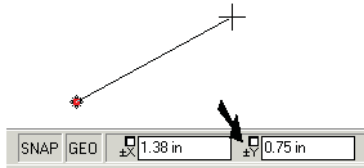


Relative Origin not fixed



Relative Origin fixed

You can enter relative X and Y values in the Coordinate Fields. Note the +/- symbols, indicating that the distance is not absolute.



### Polar Coordinates

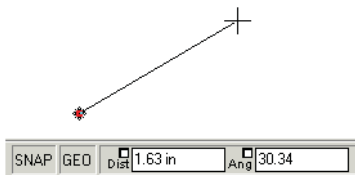
Hotkey: P, Shift+ P



Polar (length and angle) coordinates relative to a specified origin. The relative origin (moving or fixed) behaves the same way as for **Relative** coordinates.

TIP: If you precede a coordinate with a > sign it will be interpreted as a polar coordinate.

You can enter the distance and angle to the selected point in the Coordinate Fields.

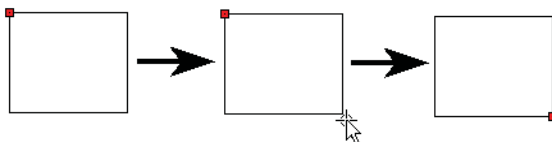


## Relocate Origin

**Hotkey:** L, Shift+ L



For Relative and Polar coordinates, moves the relative origin.



If you want to fix the relative origin in place, open the **Preference** page and check **Fixed relative origin**.

## Locking Coordinates

By default, when you move the mouse around the screen, the values in the Coordinate Fields update dynamically. However, you can enter a value and lock it so that it will not change.

Locking coordinates comes in handy when you need to only lock one field. For example, you can lock an X value and constrain all future points to be placed on the vertical line that passes through the X point.

**WARNING:** Be sure to unlock coordinates when you are finished. As long as a coordinate is locked, you are not free to select the usual range of points.

To lock a value, check the lock box for the relevant field.



You can also use the **Lock** options in the **Modes** menu, or use the X, Y, or Z hotkeys.

## Grid

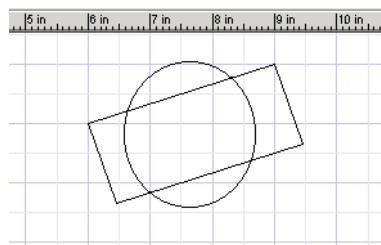
The grid is a visual aid you can use when locating, sizing, and moving objects. The G SEKE (see "SEKE's" on page 110) and **Grid Snap** (see "Grid Snap" on page 112) enable you to use grid points to precisely locate objects.

Grid tools can be accessed from the menu, or from the flyout icon on the **Grid** toolbar:



You can display the **Grid** toolbar by right-clicking in any blank toolbar area and selecting **Grid**.

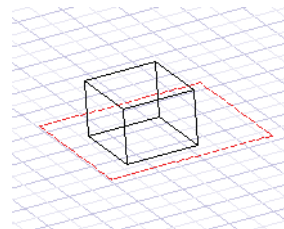
The grid can be a pattern of lines, crosses, or dots that divides the drawing space into Cartesian or polar coordinates.



You can choose whether or not to display the grid, and you can customize how you want the grid to appear.

If you display the rulers while working the grid can be a helpful indicator of orientation within the drawing.

When working in 3D, the grid is displayed on the current workplane. For details, see "Workplanes" on page 362.



Grid properties are set up in the **Grid** and **Advanced Grid** pages of the **Drawing Setup**. See "Grid Options" on page 69 and "Advanced Grid Options" on page 70.

## Displaying and Manipulating the Grid

The tools on the **Grid** toolbar and the **Grid** menu can be used to display the grid, size it, and change its origin.

### Display Grid

**Hotkey:** Alt + G

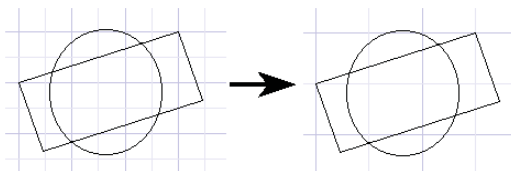


Displays or hides the grid. This command is a toggle function.

### Double Grid



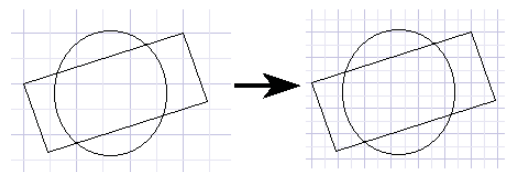
Doubles the spacing of the grid, making it less dense.



### Halve Grid



Divides the spacing of the grid in two, making it more dense.



## Set Grid Origin



Select the new origin for the grid, or enter coordinates in the Coordinate Fields. You can use **Set Origin** to align the grid to a particular point, then use the grid to set future distances.

## Printing the Grid

While printing, you can choose whether or not the grid will be printed as part of your drawing. This option is set in the **Page Setup**. See "Page Setup" on page 664.

## Snap

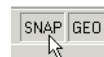
Snap ensure precise placement of points when you use the cursor.

For example, rather than enter coordinates or length to define the endpoint of a line, you can snap to the endpoint or midpoint of an existing line.

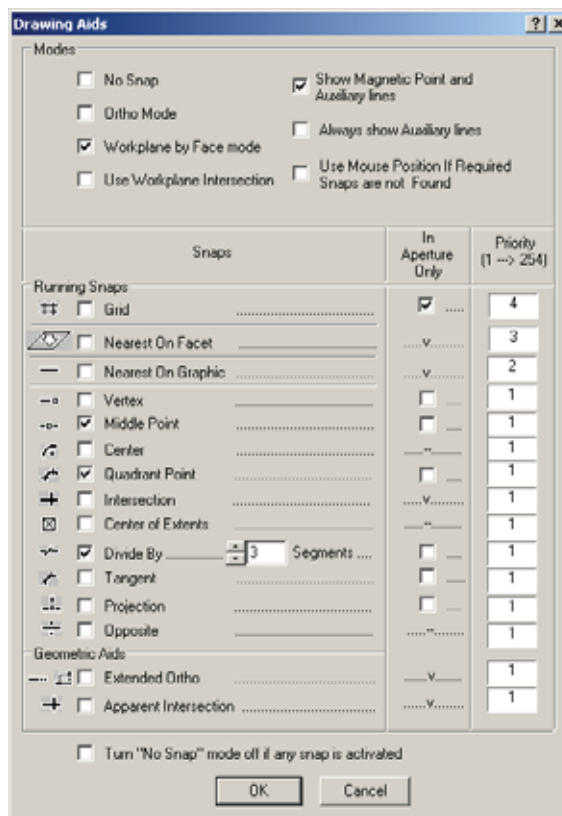
This Snap toolbar displays and controls running snaps. Local snaps can be used regardless of what running snaps are active.

## Snap Settings

The **Drawing Aids** window contains options for setting snaps and their priority levels. Open this window by right-clicking on the SNAP or GEO button next to the Coordinate Fields.







### Modes:

- **No Snap:** No snaps will be permanently active (running).
- **Ortho Mode:** The mouse will move only horizontally and vertically.
- **Workplane by Face mode:** The workplane will be set by the facet of the solid where your cursor is. See "Workplanes" on page 362.
- **Use Workplane Intersection:** Enables you to snap to intersection lines and curves between a 3D object and the current workplane. See "Workplane Intersections Snap" on page 114.

- **Show Magnetic Point and Auxiliary Lines:** A preview point will be displayed when you are within the aperture of a snap point. When relevant (for **Extended Ortho** and **Apparent Intersection**), auxiliary lines will also appear.
- **Always show auxiliary lines:** Once displayed, auxiliary lines will remain onscreen, even when the cursor moves outside the snap aperture.
- **Use Mouse Position if Required Snaps are not Found:** If you click on a point at which there is no snap, the point will be used anyway.

**Running Snaps:** These are snaps that are permanently active, as opposed to snaps that are used only once.

- **In Aperture Only:** The associated snap will function only when the snap point is within the snap aperture. If more than one object is located within the snap aperture, the point will be defined at the closest qualifying snap point, unless **Priority** is set. If not checked, the snap point itself does not need to be located within the snap aperture.

You can set the radius of the snap aperture in the **General** page of the **Program Setup (Options / General)**.

---

*TIP: To see the snap aperture while you are working, set it in the **Preference** page of **Options**.*

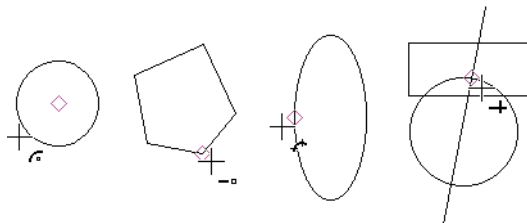
---

- **Priority:** Establishes which snap take precedence if more than one snap is available. A value of 1 has the highest priority.

## Magnetic Point



The magnetic point is the indicator of the snap location. The disadvantage to showing the magnetic point is that for very large files or very slow computers, the magnetic points might take time to display.



## Running and Local Snaps

Snaps can be permanent (until turned off) or can be temporary (used just once). Running snaps are permanent, and those invoked by the local menu and SEKE's are temporary.

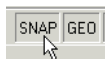
### Running Snaps

Running snaps are snaps that are constantly active, as opposed to one-time snaps invoked by SEKEs and the local menu. You can turn on running snaps by:

- Selecting items from the **Snaps** menu.
- Clicking the icon or combination of icons on the **Snap Modes** toolbar.



- Using the **Drawing Aids** window. Open this window by right-clicking on the SNAP or GEO button next to the Coordinate Fields, and check the desired snaps.



- Use hotkeys. You can view these in the **Snaps** menu. For example, Shift+V turns on **Vertex** snap.

---

**NOTE:** *Be careful with the snap modes you leave running. For example, if only **Vertex** is in use, you will not be able to select any point besides a vertex.*

---

You can temporarily turn off all running snaps by clicking the SNAP button next to the Coordinate Fields. While the button is grayed out, the running snaps are disabled. Click again to activate the snaps.

The GEO button performs the same function for geometric aids (Extended Ortho, Apparent Intersection, and Show Magnetic Point).

It is usually best to turn off both SNAP and GEO at the same time.

## Local Snaps

If you want to use a certain snap once, but do not want to have the particular snap mode constantly in use, you can use local snaps. These are one-time snaps that you can select from the **Local Snap** item of the local menu. You can also used SEKE's (hotkeys) to invoke one-time snaps.

### SEKE's

Single Equivalent Keyboard Entries are keyboard shortcuts, or hotkeys, for quickly performing a function or snapping. SEKE's are affected by the snap aperture settings. If the sought snap point or type is not available within the aperture and **Use snap aperture** is turned on the SEKE will not snap.

To use a SEKE for snapping, place the cursor on the desired object and press the key. For example, place the cursor anywhere on a line and press M to snap to its midpoint.

The following is the list of SEKE snaps:

C - Center (snaps to the center of an arc, circle, or ellipse)

G - Grid (snaps to the nearest grid point).

H - Perpendicular Opposite (snaps to an endpoint so that the new line is bisected by an existing line)

I - Intersection (snaps to the intersection of two objects)

J - Perpendicular (snaps to the perpendicular projection to an object)

M - Midpoint

N - Nearest on Graphic (snaps to the nearest place touched by the cursor)

Q - Quadrant Point (snaps to 0, 90, 180, 270 degree point of a circle, arc, ellipse)

S - No Snap

V - Vertex (snaps to endpoints of lines and arcs)

## Snap Modes

This section lists all available snaps and geometric aids. When accessed from the **Snap Modes** toolbar or from the menu, these are running snaps (permanent until turned off). When accessed by a SEKE or from the local menu, they are one-time snaps.

### No Snap

SEKE: S



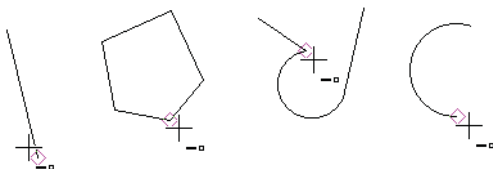
When on, turns off all snap modes. With snap modes off, you define points by simply clicking, or by using the Inspector Bar and Coordinate Fields.

### Vertex Snap

SEKE: V



Snaps to the nearest vertex. A vertex can be an endpoint of a line or line segment, corner of a polygon, or endpoint of an arc or curve.

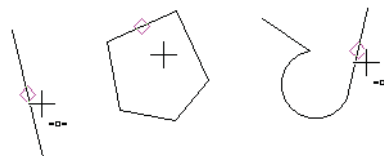


### Middle Point Snap

SEKE: M



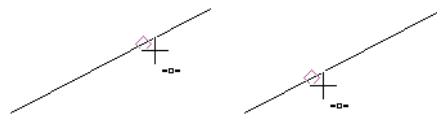
Snaps to the midpoint of a line or line segment.



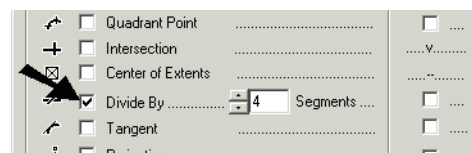
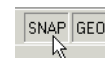
### Divide Point Snap



Snaps to one of the division points of an arc or line segment.



The default number of segments is 3, but you can change this in the **Segments** field of the **Drawing Aids** window. Open this window by right-clicking on the SNAP or GEO button next to the Coordinate Fields.



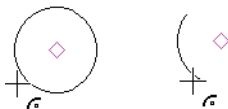
**TIP:** You can also divide arcs and line segments while using the **Edit Tool**, by selecting **Divide Segment** or **Arc Divide** from the local menu. See "Edit Tool" on page 219.

## Center Snap

SEKE: C



Snaps to the center of an arc, circle, or ellipse.

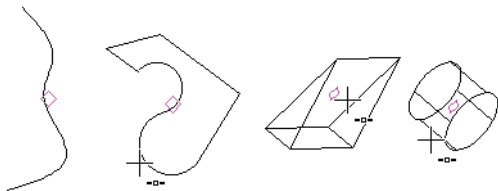


## Center of Extents Snap

SEKE: E



Snaps to the center of extents of a 2D or 3D object. This is the center of the bounding rectangle (2D) or box (3D) that encloses the selected object.



## Nearest on Facet Snap



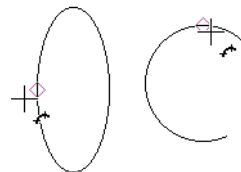
Snaps to the nearest point on a facet, or to the projection of this point onto the current workplane.

## Quadrant Point Snap

SEKE: Q



Snaps to the nearest quadrant point (0, 90, 180, and 270 degrees) on a circle, arc, or ellipse.

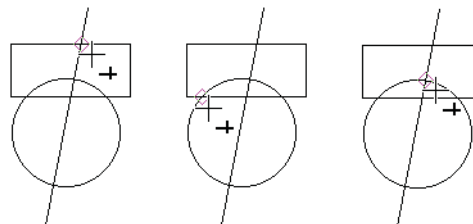


## Intersection Snap

SEKE: I



Snaps to the intersection of two objects.



## Grid Snap

SEKE: G



Snaps to the nearest grid point. See "Grid" on page 107.

For details on grid settings, see "Grid Options" on page 69.

If the **Advanced Grid** option of **Frequency** is set to a figure greater than one, invisible grid lines will also be detected by this snap mode. See "Advanced Grid Options" on page 70.

## Nearest on Graphic Snap

SEKE: N

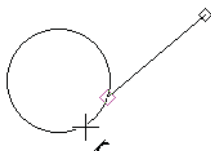


Snap to the point on an object closest to the cursor, within the snap aperture.

## Tangent Snap



Snap to a tangent point on an arc, circle, or ellipse, relative to the previous point.

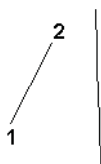


## Projection Snap

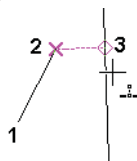


Snap to a point that is the perpendicular projection image of the last point drawn onto a selected object.

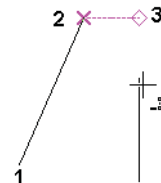
In this example, the line on the right is the projection line. The last point drawn is Point 2, when creating Line 1-2.



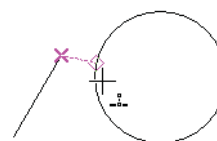
Activate the **Projection** snap and hover over the projection line. Point 3 is the snap point, which is the perpendicular projection of Point 2.



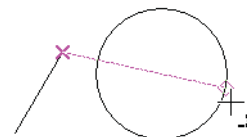
The snap point does not have to lie on the projection line; it can be placed on the extension of the projection line.



In the case of a circle, the projection point is along the radial from the circle center. If the circle is selected on the left side, the snap point will be on that side.



If you select the circle on its right side, the snap point will be on that side.

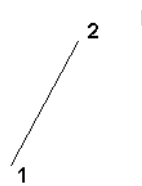


## Opposite (Mirror Point) Snap

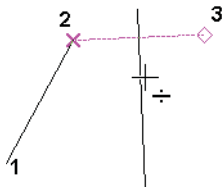


Snap to a point that is the mirror image of the last point drawn, in reference to a selected object.

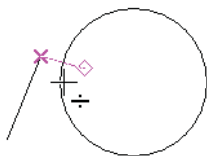
In this example, the line on the right is the mirror line. The last point drawn is Point 2, when creating Line 1-2.



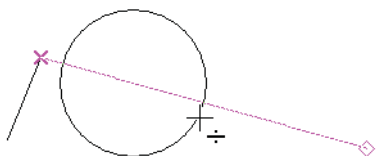
Activate the **Opposite** snap and hover over the mirror line. Point 3 is the snap point.



In some cases it is important where you select the mirror object. If the circle is selected on the left side, the mirror point will be opposite that side.



If you select the circle on its right side, the mirror point will be opposite that side.



## Ortho Snap

**Hotkey:** Shift



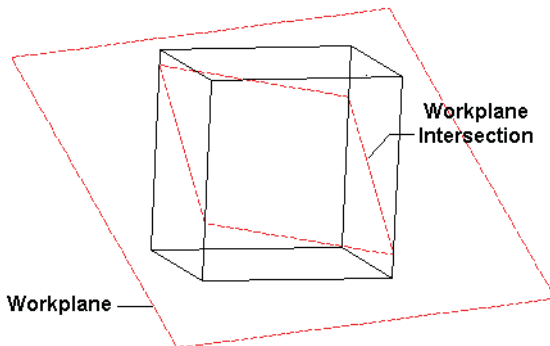
When in use, the cursor can only move horizontally or vertically.

## Workplane Intersections Snap

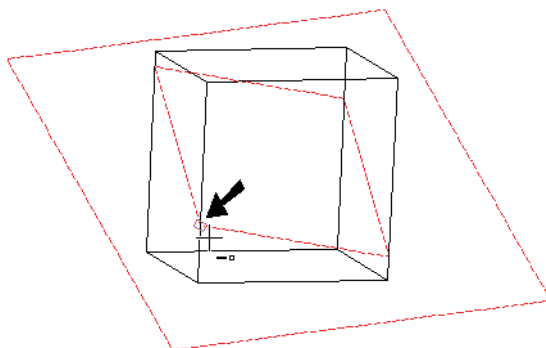


Snap to intersection curves between 3D objects and the current workplane. (For information on workplanes, see "Workplanes" on page 362)

1. The workplane must be displayed (see "Displaying the Workplane" on page 364), and intersection lines must be displayed as well (see "Display Intersections with 3D Objects" on page 370).



2. Activate the **Workplane Intersections** snap along with the snaps you want to use on the intersection lines and curves. In this case **Vertex** snap is active, and you can snap to an endpoint of the intersection line.



## Show Magnetic Point



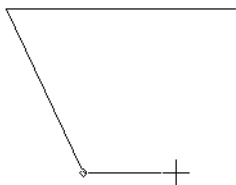
When in use, a diamond-shaped point will be displayed for the locations of all running snaps. See "Magnetic Point" on page 110.

## Extended Ortho Snap

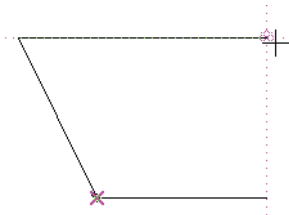


Snap to points located orthogonally from existing points. **Show Magnetic Point** must be turned on.

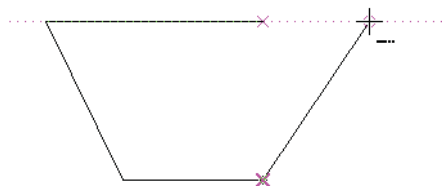
1. Start a polyline with two linear segments, then start the third segment like this:



2. To end the third segment, first hover over the start point. This displays an auxiliary line extending vertically from that point. You can snap to points along this line.



3. Horizontal auxiliary lines can also extend from this point, when the cursor is to the right or left.




---

**NOTE:** If **Always show auxiliary lines** is checked in the **Drawing Aids window**, the auxiliary lines will be displayed, even when you move the cursor away from them.

---

## Apparent Intersection Snap

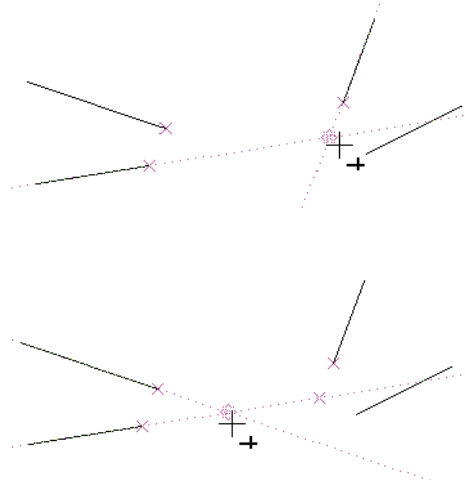


Snap to points where two lines would meet. An auxiliary line is displayed from both lines, extending to this point. **Show Magnetic Point** must be turned on.

1. Start with lines whose extensions will meet (non-parallel). When you pass the cursor over each line, the line's endpoint is marked.



2. When the cursor passes over any apparent intersection point, auxiliary lines will be displayed from the actual lines, and their intersection point is marked.




---

**NOTE:** If **Always show auxiliary lines** is checked in the **Drawing Aids window**, the auxiliary lines will be displayed, even when you move the cursor away from them.

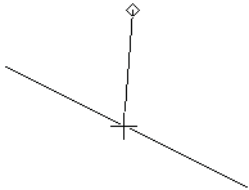
---

## Perpendicular Snap

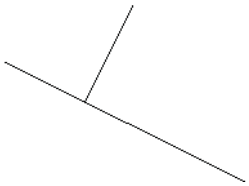
SEKE: J

Connects a line to another line so that the two lines are perpendicular.

1. Select the first endpoint of the line. For the second endpoint, move the cursor over the line to which the new line will be perpendicular.



2. Press J.



## Layers

Layers are virtual levels in a drawing, analogous to acetate tabs (overlays) in traditional drafting. Layers enable you to sort objects in your drawing by type, by creation order, or by any criteria that suits the way you work. You can also use layers to protect certain objects from being edited or deleted.

---

*NOTE: Layers are not related to how objects are stacked in relation to their order of creation. If you change an object's layer, it does not affect its position in the object stack. Setting Up Layers*

---

There are two primary ways of creating and editing layers. The Layer manager, and Design Director which is only available in the Pro product. With regard to layers you can do essentially operations with either.

Open the **Layer** manager by selecting **Format / Layers**, or by clicking the **Layers** icon on the toolbar.



This dialog is used to create layers, to assign properties to each layer, and to organize layers. It is divided into two



frames. The first frame show the tree containing the layer filter, and layer templates. The second frame show the layers. controlled or derive from the item selected in the tree. The top node of the tree controls and shows all layers in the drawing.

Layer 0 is the default layer, and all objects are placed here unless otherwise specified, or unless another layer is created and made active. Layer \$CONSTRUCTION is created when construction geometry is created (see "Construction Geometry" on page 121). Neither of these default layers can be deleted, but you can change their properties.

**Layer Set:** See "Layer Sets" on page 120.

**Columns in Layers window:** Use the horizontal scroll bar to see all columns. You can sort the list by clicking on any column heading; the list will be sorted according to the selected parameter.

- **Visibility (eye icon):** Check to make objects on the layer visible.
- **Read-only (lock icon):** Check to make the layer read-only, so that objects cannot be edited or deleted. You can add objects to a read-only layer via the **Layer** drop-down list on the **TurboCAD Layers** toolbar.



- **Color:** Click on the the color box to open the color dialog, then select a color - The default color is black. Objects will have the layer color if their color is set to **By Layer**.
- **Style:** Select the line style from the drop-down list and double-click to set it. Objects will have the layer style if their style is set to **By Layer**.
- **Order:** Determines the order in which layers are drawn. Order can be used to place certain categories of objects in front of others; objects drawn on higher layers will be “over” those on lower layers. The order numbers are initially set to zero. Layers that have the same order number are sorted alphanumerically. The highest layer number is 32767.

**WARNING:** The **Draw Order** commands will not function as you expect if objects are on different layers and the layers have different **Order** values.

- **Pen Width:** Sets the line width. Objects will have the layer width if their width is set to **By Layer**.
- **Print Style:** Specifies the Print Style to be used by objects on that layer. "Print Style Options" on page 522.

**VP Columns in Layers window:** In addition to the main columns in the Layers window there are five columns with a VP prefix. VP stands for Viewport, and these columns are use to control how objects will appear within a selected viewport.

VP Visible	VP Color	VP Line Style	VP Pen Width	VP Print Style
...	...	...	...	...

The columns are: VP Visible, VP Color, VP Line Style, Pen Width, Print Style.

The VP columns operate in the same way as the main columns, except the settings effect on selected viewports.

To use the VP column settings:

1. Open the Layer Manager.
2. Go to Paper Space.
3. Select the Viewports you wish to configure.

4. Apply the settings in the VP columns of the Layer Manager.

*NOTE: Remember that settings which control object appearance via the Layer Manager (e.g. VP Color) only affect the properties objects which are set to By Layer.*

## Layer Manager Toolbar

**Refresh:** Refresh the Layer manager display.

**New Layer Filter:** Creates a new Layer filter. Assign a name to the filter then setup the filter in the Layer Filters dialog.

**Edit Filter Parameters:** Open the layer filter currently selected in the tree in the Layer Filters dialog.

**Edit Layer Sets:** Opens the Layer Set dialog to create and edit layer sets.

**New Layer Template:** Creates a new layer template from the currently selected layers.

**New Layer:** Creates a new layer. The default name will include a prefix by default, but the name can be changed.

**Delete Layer:** See "Deleting a Layer" on page 118.

**Activate:** Sets the currently selected layer (only one) as the active layer. This will effect any currently active drawing tool or currently selected object.

**Select By:** Selects all objects on the currently selected layers.

**Edit Properties:** Opens that layer page of the Drawing Options.

**Visible/Invisible All:** If any layer is invisible all layers are made visible. If all layers are visible they are all made invisible.

**Invert Visibility:** Turns all visible layers invisible, and all invisible layers visible.

**Exclusive Visible:** Makes only the currently selected layers visible, all other become visible.

**Lock/Unlock All:** If any layer is unlocked all layers are locked. If all layers are locked they are all unlocked.

**Invert Lock:**

**Exclusive Lock:** Makes only the currently selected layers locked, all other become unlocked.

**Layer Prefix:** By default, the layers are named “Layer 1, Layer 2,” etc. You can change or remove the prefix. The @ symbol is a placeholder for the automatic layer number.

---

*Note: Many of the functions available from the Layer toolbar are also available by right clicking and opening the local menu.*

---

## Creating a New Layer

1. Select **Layers**, click **New Layer**, then assign a name for the layer in the **Layer** column (or accept the default name).
2. Adjust the various layer settings, such as color and line style.

## Deleting a Layer

You may delete any layer except Layers 0 and \$CONSTRUCTION. Layers can be deleted even if they contain objects.

If the layer to be deleted is set as the default for a tool (in the **General** page of a tool’s **Properties** window), you will receive a warning message before the layer is deleted.

1. Select **Layers**, and select the layer to be deleted.
2. Click **Delete Layer**. If the layer contains objects, the objects will be deleted. This action can be undone, in case you delete objects inadvertently.

In some cases, objects on deleted layers will be moved to Layer 0, rather than be deleted. If an object exists on Layer 1 in both Paper Space and Model Space, and Layer 1 is deleted from Model Space, in Paper Space the object will be moved to Layer 0. This is due to different **Undo** buffers for Model and Paper Spaces.

## Layer Templates

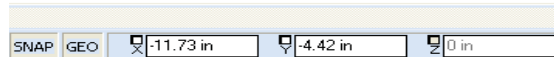
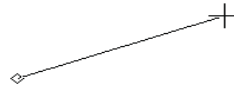
Layer templates allow you to create and save alternate configurations for layers. Layer templates store how layers are setup, but do not store layers themselves. Layer templates are saved in a \*.lrs file which can be stored anywhere in your system directory.

## Manipulating Layers and Properties

By default, new objects are created on Layer 0, unless a layer is specified in the **Properties** window of a specific tool. You can change the layer of objects after they are created, or you can change the default layer of a group of tools.

### Setting the Active Layer

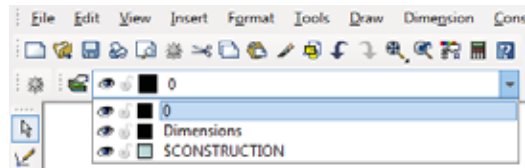
Set the layer you want to be active by selecting it from the **Layer** list in the **TurboCAD Layers** toolbar. You must have a drawing tool selected or an object selected to change the active layer.



Setting the Layer of Selected Objects or a Group of Tools

To quickly change the layer of selected objects:

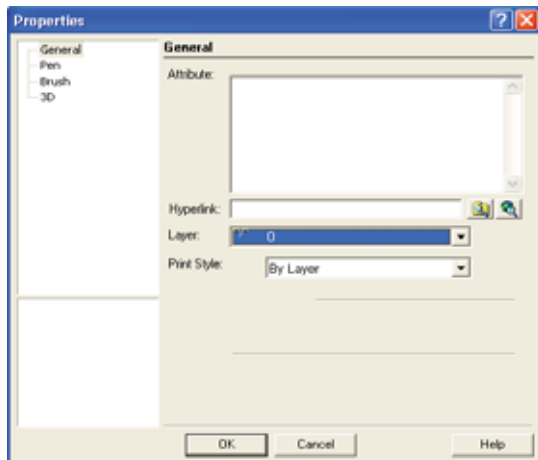
1. Select the objects.
2. Select the layer from the **Layers** list box in the **TurboCAD Layers** toolbar.



To set layers via the **Properties** window:

1. Select the objects and open the **Properties** window. To set the layer for all objects created with a certain group of tools, right-click on one of the tool icons and open the **Properties** window (see "Object Properties" on page 90).

- On the **General** page, select the layer from the drop-down list.



If the layer is set for selected objects, the objects will move to that layer. Future objects, however, will still be created on the tool's default layer. If the layer is set for a group of tools, all objects created with any one of the tools will automatically be placed on the selected layer. For example, setting the layer for the **Line** tool also affects the **Rectangle** and **Polygon** tools.

---

NOTE: You can also select the objects and change the layer on the Selection Info Palette. See "Selection Info Palette" on page 224.

---

### Changing the Layer Visibility via Dropdown

You can change the visibility of a layer using the Layer dropdown by clicking on the eye icon adjacent to the layer in the dropdown.

### Changing the Layer Color via Dropdown

You can change the color of a layer using the Layer dropdown by clicking on the color icon adjacent to the layer in the dropdown.

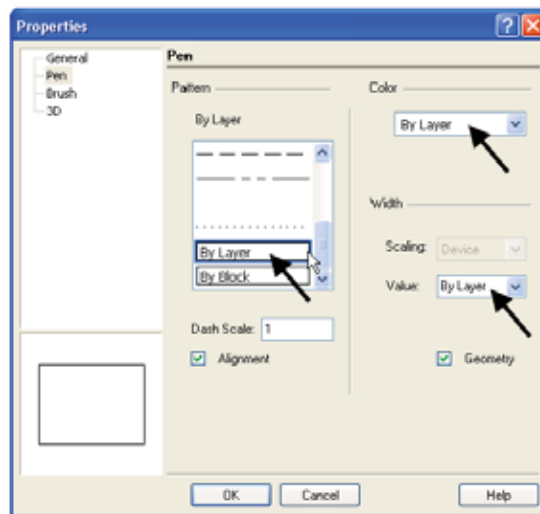
### Changing the Layer Lock via Dropdown

You can change the lock status of a layer using the Layer dropdown by clicking on the lock icon adjacent to the layer

in the dropdown.

### Assigning Layer Properties to Objects

- Open the **Properties** window, either for selected objects or for a group of tools. See "Object Properties" on page 90.
- Open the **Pen** page.
- The properties that can be set to **By Layer** are color, line style, and width. Set any or all of these properties as needed.




---

NOTE: **By Layer** is also available for properties on the **TurboCAD Properties** toolbar.

---

### Layers of Groups and Blocks

If you create a group composed of objects on the same layer of the drawing, the group will reside on that layer. For objects on different layers, the group will be on Layer 0. If you subsequently explode the group, its constituent objects will return to its original layer.

Objects should be moved to Layer 0 before being used to create a block. The block itself, when inserted in the drawing, can be reassigned to the proper layer. Any objects within the block that have a **By Layer** property will take on the assigned layers attributes.

When a block is exploded, all objects originally assigned to Layer 0 will stay on the layer the block was on. Any objects with other layer assignments will retain those layers.

### Layer Sets

A layer set is a group of layers which can be displayed as a group. This is useful for displaying certain aspects of a drawing without changing visibility settings of each layer individually.

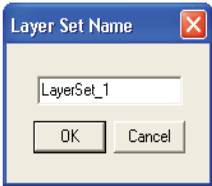
The default layer set is “All Layers,” which appears in the **Format** menu.

### Creating and Manipulating Layer Sets

- 1. In the **Layers** window (**Format / Layers**), click **Edit Layer Sets**.
- 2. When the **Layer Set** dialog opens select **New**.

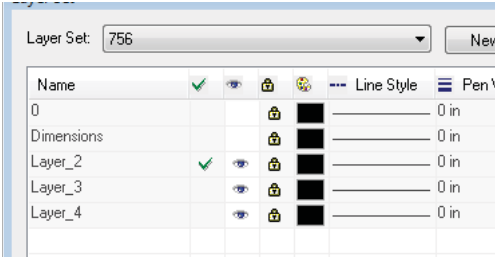


- 3. Assign a name to the set, or accept the default name.



The name appears on the **Layer Set** list.

- 4. On the list of layers, check the visibility of each layer you want included in the layer set.



- 5. To display a layer set, open the **Format** menu.

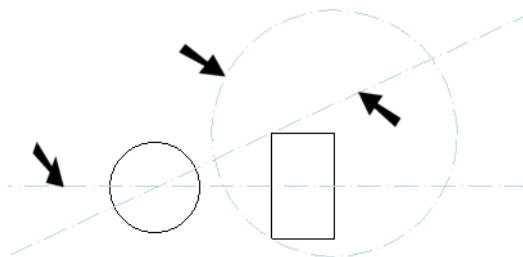
*NOTE: While a layer set is displayed, the properties of each layer are not editable.*

To delete a layer set, select it from the **Layer Set** list and click **Delete**.

To change the layers that appear in a layer set, select it from the **Layer Set** list and change the visibility settings.

## Construction Geometry

Construction geometry tools enable you to place temporary lines and circles in your model. Construction geometry are not drawing objects; they are used as references.

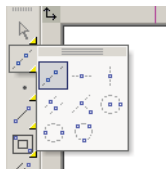


Snaps can be used on construction geometry for creating model geometry, as well as other construction geometry. However, because construction lines are infinite, the **Vertex** and **Middle Point** snaps are not available.

You can display the **Construction** toolbar by right-clicking in any toolbar area and selecting **Construction**.



These tools are also available on the fly-out toolbar from the **Drawing Tools**.



**NOTE:** You can use construction entities with the **Trim** tool to trim other drawing entities, however you cannot trim construction entities themselves.

## Construction Geometry Properties

Construction objects are placed on layer "\$CONSTRUCTION". By default, the layer color is light blue, and the line style is dash-dot.

You can change construction geometry color and line styles via the layer manager.

**NOTE:** It is not recommended to place model geometry on the \$CONSTRUCTION layer. If the color and line style of objects are set to **By Layer**, the objects will appear as construction geometry. It is also not recommended to change the line style to **Continuous**.

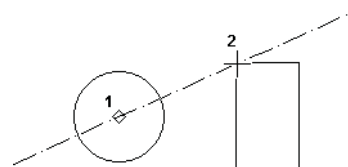
## Creating Construction Geometry

Several types of construction lines and circles are available.

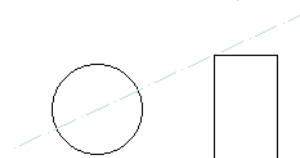
### Construction: Angular Line



1. Select a point through which the construction line will pass. Select a second point, or enter the slant in the Inspector Bar.



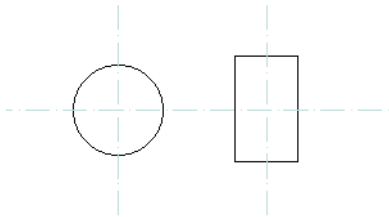
2. The construction line is created, with the properties assigned to the construction layer.



## Construction: Horizontal Line, Vertical Line



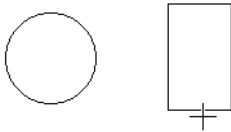
1. Select the point through which the construction line will pass.
2. Continue creating construction lines of the same type, and select **Finish** when done.



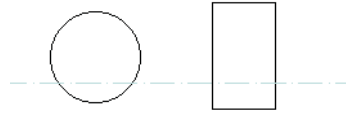
## Construction: Parallel



1. Select the line to which the construction line will be parallel. You can also select a construction line.



2. Select a point through which the construction line will pass, or enter the offset in the Inspector Bar.




---

*TIP: To make multiple construction lines separated by the same distance, lock the **Offset** field in the Inspector Bar, and create parallel lines from each successive line.*

---

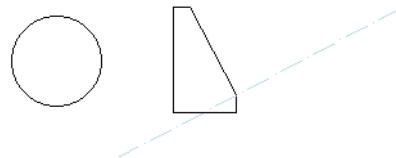
## Construction: Perpendicular



1. Select the line to which the construction line will be perpendicular. You can also select a construction line.



2. Select a point through which the construction line will pass.

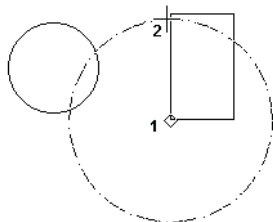


### Construction: Center and Point Circle

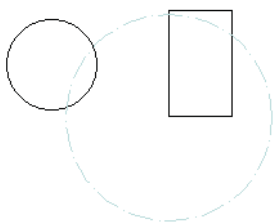


Creates a construction circle by defining its center and a point on its circumference.

1. Select the circle centerpoint. Move the cursor to begin to size the circle.



2. Click to create the circle, or enter the radius, diameter, or circumference in the Inspector Bar.

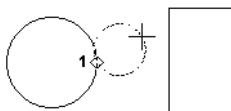


### Construction: Double Point Circle

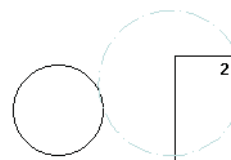


Creates a construction circle by defining two opposite points on its circumference.

1. Select a point on the circle circumference.



2. Select the point on the opposite end of the diameter line, or enter the radius, diameter or circumference, and angle of the diameter line in the Inspector Bar.

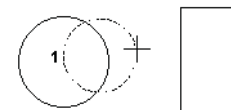


### Construction: Triple Point Circle

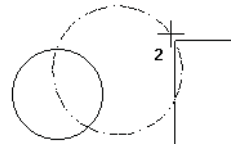


Creates a construction circle that passes through three points.

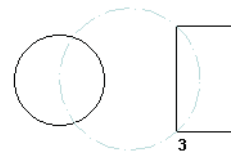
1. Select the first point on the circumference.



2. Select the second point.



3. Select the third point.



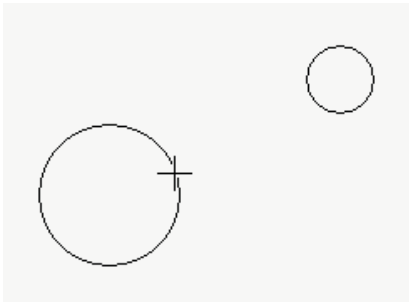
## Construction: Perpendicular by Line



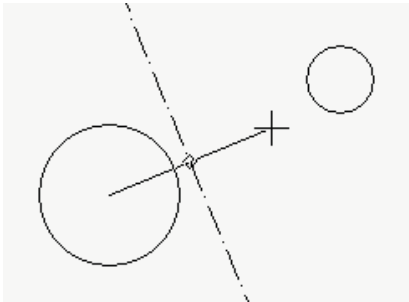
Creates a construction line perpendicular to a line defined by two points.

Start with two circles. We will create a construction line halfway between the centers of the two circles perpendicular to the axis between the two centers.

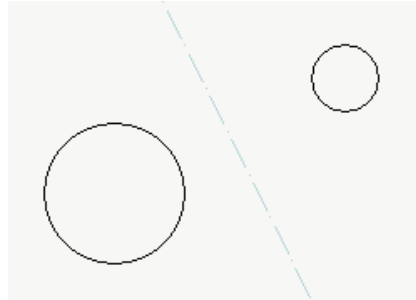
1. Select the tool, then snap to the center of the first circle.



2. Move the cursor. Notice that the preview of the construction line is parallel to the line you are dragging out.



3. Snap to the center of the other circle to set the construction line.



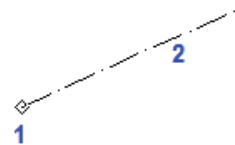
## Rays

Rays are a special type of construction line with only one end point extending infinitely in one direction.

### Angular Ray



1. Select a point for the end of the ray will pass. Select a second point, or enter the slant in the Inspector Bar.



2. The ray is created, with the properties assigned to the construction layer.



Horizontal Line, Vertical Line



1. Select the point through for the end of the ray.
2. Continue creating ray lines of the same type, and select **Finish** when done.

Local menu options:

**Flip:** the Flip option changes the direction of the ray 180 degrees.

Editing Construction Lines

You can use the Edit tool to adjust the nodes of construction lines and rays.

You can split a construction line into two rays by using the Split tool.

Clearing and Hiding Constructions

You can clear construction objects via the **Edit** menu.

- **Edit / Clear / Construction** enables you to delete a single construction object.
- **Edit / Clear / All Constructions** deletes all construction geometry.

To hide constructions, you can hide the \$CONSTRUCTION layer. Open the **Layers** window and uncheck the visibility box for this layer.

Printing Construction Geometry

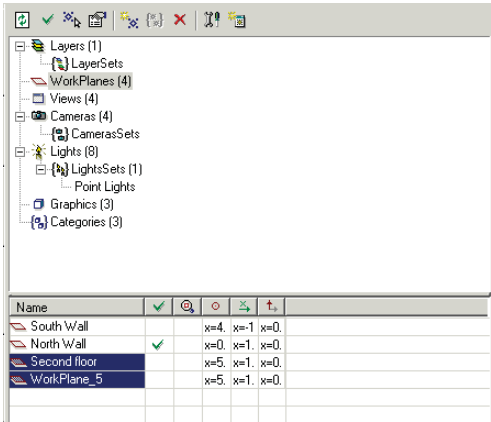
While printing, you can choose whether or not construction geometry will be printed as part of your drawing. This option is set in the **Page Setup**. See "Page Setup" on page 664.

Design Director

Available in TurboCAD Pro and Platinum only

**Hotkey:** F3

The Design Director is a convenient place to perform commands related to layers, workplanes, views, cameras, and graphic objects or object groups.



Design Director Toolbar and Menu

The following options are available on the Design Director toolbar and on the local menu. The available options depending on the item chosen (layer, layer set, workplane, etc.)

- **Refresh** : Redraws the Design Director pane.
- **Activate** (hotkey: Ctrl+A): Applies, or makes active, the selected item.
- **Select By** (hotkey: Ctrl+S): Selects the selected item in the drawing space .
- **Edit Properties** (hotkey: Ctrl+P): Change properties of the selected item.
- **Create New** (hotkey: Ctrl+N): Adds a new item.
- **Create Set:** Creates a set of all or selected items in the table. Relevant for layers, lights, and cameras.
- **Delete** (hotkey: Ctrl+D): Deletes the selected item.
- **Options** : Invokes the **Options** window to customize the Design Director.

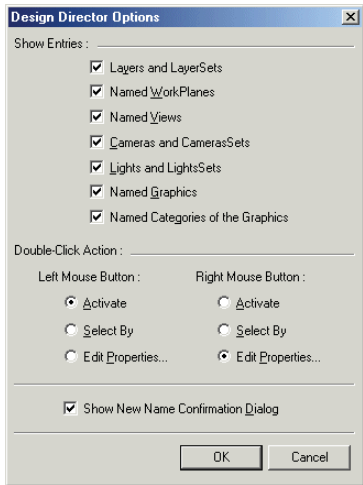
- **Open as Separate Palette** : Open the selected item as separate palette. This is useful if you want to keep multiple sections of the Design Director open at once.

The following options appear on the local menu only:

- **Load From File** (hotkey: Ctrl+Shift+O): Loads an item from the file with the corresponding extension.
- **Save To File** (hotkey: Ctrl+Shift+S): Saves an item to the file with the corresponding extension.
- **Show Toolbar**: Shows or hides the Design Director toolbar.

### Design Director Options

General options for items to display or hide, how the mouse will be used, and confirmation windows.



**Show Entries:** Select the items you want displayed in the Design Director.

**Double-Click Action:** Sets the action performed when double-clicking on either the right or left mouse button.

**Show New Name Confirmation Dialog:** If checked, you will be asked to confirm the name of each new item you create using **Create New**. If not checked, the names will be assigned automatically.

### Design Director: Layers

Enables you to manipulate all the layers and layer sets in the drawing (see "Layers" on page 116 and "Layer Sets" on page 120).

To create a new layer or layer set, select **Create New** in the Design Director toolbar or local menu.

When **Layers** or **Layer Sets** is selected in the upper pane of the Design Director, the lower pane has the following options:

Name	Active	Visible	Lock	Color	Order
0				0.0.0	0
\$CONSTRUCTION				191.216.2.1	
Topo				219.147.1.2	
Subfloor				0.126.255.3	

**Active:** Sets the layer as active, or applies the layer set. Also indicates the layer for a selected object or tool.

**Visible:** Sets the layer visibility.

**Lock:** Use this switch to lock layers, which means the objects on them will be read-only. You can add objects to locked layers, but you cannot edit or delete objects.

**Color:** Sets the layer color.

**Order:** The order number of the layer.

**Line Style:** specifies the Line Style for the layer.

**Pen Width:** Specifies the Pen Width for the layer.

**Print Style:** Specifies the Print Style for the layer.

### Layer Sets

A layer set is a group of layers, which can be handy if you want to set visibility, locking, and other parameters for multiple layers.

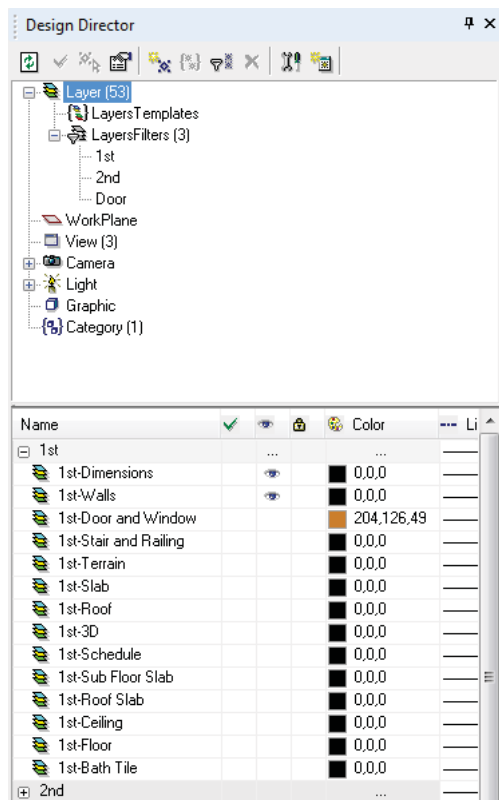
1. Select **Layers** in the upper pane of the Design Director.
2. On the lower pane, select the layers you want to include in the layer set.
3. Select **Create Set** in the Design Director toolbar or local menu.
4. Select **Layer Sets** in the upper pane in order to see the defined layer sets in the lower pane.

5. Select the properties (visibility, locking, etc.) of the layer set. When the layer set is made **Active**, these properties will be automatically applied to all layers in the set.

## Layer Filters

Available in TurboCAD Pro and Platinum only

Layer filters provide you with a powerful tool to organize your work, workflow and drawing. This feature also allows you to import DWG file that have embedded layer filters



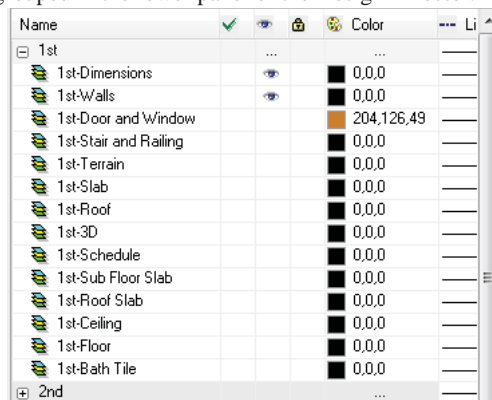
Layer filters will organize your layer by a defined set of parameters.

## Layer Templates

Layer templates allow you to create and save alternate configurations for layers.

Layer templates store how layers are setup, but do not store layers themselves. Layer templates are saved in a \*.lrs file which can be stored anywhere in your system directory.

All layers which meet the parameters defined by the filter will be grouped in the lower panel of the Design Director.



*Note: The Show Filters and Group by Filters options must be turned on in the Design Director options for Filter groups to be displayed.*

You can control all of the settings for the layers in the group. This means that if you change the color adjacent to the filter, the color for all of its layer will be changed to that color. If you change the visibility adjacent to the filter, the visibility of all its layer will be changed to that visibility. If for any setting the layers contained in the filter have mixed settings for an option, the filter will display a blank field or an ellipsis (...) for that setting, however you can still control the settings of the filtered layers.

### To Change Layer Color Properties via the Filter

1. In the Design Director of a filtered drawing:
2. In the lower panel of the Design Director, hover the cursor over the color field adjacent to a filter.
3. Click on the color field.
4. When the Color dialog appears, select a color. The colors of all the filtered layers will be changed to the selected color.

To Create a Filter

- 1. Click the Create Layers Filter button in the toolbar of the Design Director.
- 2. When Prompted assign a name to the filter.
- 3. When the Filter Dialog appears, specify the parameters for the filter.
- 4. Click OK.

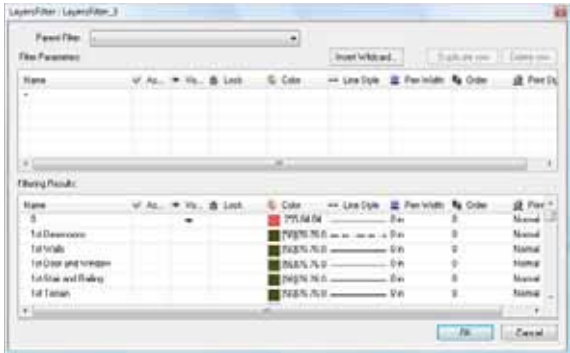
To Edit a Filters Parameters

- 1. Select the Filter in the top Panel of the Design Director.
- 2. Right click and Select Edit Filter Parameters..
- 3. When the Filter Dialog appears, reset the parameters for the filter.
- 4. Click OK.

Layer Filter Dialog

Available in TurboCAD Pro and Platinum only

The Layers filter dialog allows you to specify how your filters work.



The dialog is divided into three sections: Toolbar, Filter Parameters table, Filtering Results table.

Filter Toolbar

- **Parent Filter:** Filter can have sub-filters. This setting allows you to specify a parent filter that will constrain the list of layer that the current filter will apply to.

- **Insert Wildcard:** This button opens the Wildcard dialog which allow you to insert a wildcard into the currently selected field.



The dialog also defines how each of the wildcards function.

- **Duplicate Row:** This button duplicates the currently selected row in the Filter Parameters table.
- **Delete Row:** This button deletes the currently selected row in the Filter Parameters table.

Filter Parameters

The Filter Parameters table is where you specify how the filter will work by entering a set of parameters. Parameters are defined by values and wildcards. Each field in takes specific types of values.

- **Name:** The Name field uses the name of layers to filter. Full names or parts of names can be combined with wildcards to specify what layers you want the filter to catch. The name field is the most versatile in regard to using wildcards.
- **Active:** The Active field uses the Active status of layers to filter. It can take two values TRUE or FALSE. These can be combined with the NOT (~) wildcard to signify the opposite meaning. E.g. ~TRUE means = is not true. Since there can be only one Active layer at anytime, this option will make the filter dynamic as to what layers are captured.
- **Visible:** The Visible field uses the Visible status of layers to filter. It can take two values TRUE or FALSE. These can be combined with the NOT (~) wildcard to signify the opposite meaning. E.g. ~TRUE = is not true.

- **Locked:** The Locked field uses the Locked status of layers to filter. It can take two values TRUE or FALSE. These can be combined with the NOT (~) wildcard to signify the opposite meaning. E.g. ~TRUE = is not true.
  - **Color:** The Color field uses the Colors of layers to filter. It takes values which identify colors, either as RGB (True Color) values expressed as three numbers separated by commas, e.g. 10,20,30, or as Index colors expressed as a single number ranging from 1 to 255. These can be combined with the NOT (~) wildcard to signify the opposite meaning. E.g. ~10 = is not color 10. You cannot embed wildcard within the RGB value.
  - **Line Style:** The Line Style field uses the Line Styles of layers to filter. It takes values which are the names of Line Styles, such as CONTINUOUS or BORDER. These can be combined with the NOT (~) wildcard to signify the opposite meaning. E.g. ~BORDER = is not BORDER.
  - **Pen Width:** The Pen Width field uses the Pen Widths of layers to filter. It takes values which are the numeric values of the pen widths along with the units. For Example 0.5in or 33 mm. These can be combined with the NOT (~) wildcard to signify the opposite meaning. E.g. ~0.5in = is not 0.5in.
  - **Order:** The Order field uses the Order of layers to filter. It takes values which are the numeric values of the Order. For Example 1 or 55. These can be combined with the NOT (~) wildcard to signify the opposite meaning. E.g. ~10 = is not 10.
  - **Print Style:** The Print Style field uses the Print Styles of layers to filter. It takes values which are the names of Print Styles, such as 'Normal' or New Style. These can be combined with the NOT (~) wildcard to signify the opposite meaning. E.g. ~BORDER = is not BORDER.
- Wildcards**
- Wildcards are special characters that are combined with other values to constrain a result. The wildcards that are supported by the filters are:
- **# (pound)** Matches any single numeric digit. For example, if there are 1000 layers named 1 to 1000, you can build the filter to capture all of the layers from 500 to 599 by using 5##, or 550 to 559 by using 55#
  - **@ (at)** Matches any single alphabetic character. For example, if there four layers named EAST, WEST, BEST and TEST, you can build a filter to capture all four by using @@ST, or just the last three layer by using @EST
  - **.** (period) Matches any single non alphanumeric character. For example, if there were four layers named TOP\_VIEW, TOP-VIEW, TOP VIEW and TOP:VIEW, you can build a filter to capture all four by using TOP.VIEW
  - **\*** (asterisk) Matches any string (sequence of characters) and can be used anywhere in the search string, beginning, middle or end. For example, you have dozens of layers that include the term 1ST, you can capture all of them by using \*1ST\*
  - **?** (question mark) Matches any single character; for example, ?BC matches ABC, 3BC, etc. As another example, if there four layers named EAST, 2EST, \_EST and TEST, you can build a filter to capture all four by using ??ST, or just the last three layer by using ?EST
  - **~ (tilde)** Matches anything but the pattern, in other words it equals NOT. For example; ~\*AB\* matches all strings that don't contain AB. As another example, you have dozens of layers that include the term 1ST, you can exclude them all from being captured by using ~\*1ST\*
  - **[ ]** Matches any one of the characters enclosed; for example, [AB]C matches AC and BC. As another example, all of the layers begin with a single number, and you want to capture just those that begin with 5, 7 and 8. You can capture them by using [578]\*
  - **[~]** Matches any character not enclosed; for example, [~AB]C matches XC but not AC. As another example, all of the layers begin with a single number, and you want to exclude from capture just those that begin with 5, 7 and 8. You can do this by using [~578]\*

- **[ - ]** Specifies a range for a single character; for example, [A-G]C matches AC, BC, and so on to GC, but not HC. As another example, all of the text layers have a format like TXT followed by a number followed by more characters e.g. TXT1TOP or TXT2BOTTOM. You want to capture any with the numbers 2, 3, 4, or 5. You can capture all of them using TXT[2-5]\*
- **`** (reverse quote) Reads the next character literally; for example, `~AB matches ~AB. This is especially useful if your layers have characters in their names that otherwise would be interpreted as wildcards.

Important Items to remember:

- All items on the same row of a filter further refine (tighten) the results of the filter. For example, on one row, if under the Name field you have \*1ST\* and in the Color field you have 55, the filter will capture only those items that have both 1ST and are Color 55
- Items on separate rows expand the results of the filter. For example, if on one row in the Name field you have \*1ST\* and on the next row in the Color field you have 55, the filter will capture all layers with 1ST and all layers with Color 55.
- Sub filters are restricted to capturing only layers which are contained within their parent filters.

## Filter Results

The Filtering Results table displays results of the filter as defined in the Filter Parameters table, this gives you a preview of how your filter will operate.

There are several functions which are available if you right click a field in the Filtering Results table.

- **Refine using this value:** If this option is chosen the value in the field will be added to each row of the Filter Parameters table with content. This further restricts the results of the filter.
- **Refine excluding this value:** If this option is chosen the value in the field will be added to each row of the Filter Parameters table with content with a ~ prepending the value. This further restricts the results of the filter.
- **Add all with this value:** If this option is chosen the value in the field will be added to a new row of the Filter Parameters table. This expands the results of the filter.
- **Add all without this value:** If this option is chosen the value in the field will be added to a new row of the Filter Parameters table with a ~ prepending the value. This expands the results of the filter.

## Filters and XREFs

*Available in TurboCAD Pro and Platinum only*

When external references (XREFs) are add to a file all of the layers of the external referenced file are appended. The layers are listed under the XREF name as if the XREF was itself a filter. in add ti on the XREF is listed under the filters in the top panel of the Design Director. Within the Design Director lower panel you can treat the XREF as if it is a filter, including changing the values for the properties of all the layer s by making changes at the filter level. However, you cannot edit the parameters of the XREF as if it were a filter.

Design Director: Workplanes

Enables you to manipulate and create new workplanes. See "Workplanes" on page 362.

To create a new workplane, select **Create New** in the Design Director toolbar or local menu.

When **Workplanes** is selected in the upper pane of the Design Director, all workplanes you saved using **Set Named Workplane** (see "Saving and Recalling WorkPlanes" on page 369) will appear in the lower pane. The lower pane contains the following options:

Name	✓	VI...	Position	X Vector	Up Vector
South Wall			x=4.73542491411	x=-1.00000000000	x=0.00000000000
North Wall	✓		x=0.00000000000	x=1.00000000000	x=0.00000000000
Second floor			x=5.23902936471	x=1.00000000000	x=0.00000000000

**Active:** Sets the workplane as active; all inserted objects will be located or based on this workplane. Also indicates the workplane for a selected object or tool.

**View by WorkPlane:** Sets the view normal to the workplane (workplane is against the screen).

**Position:** The point that represents the UCS origin. See "3D Coordinate Systems" on page 362.

**XVector:** The point that defines the direction of the X-axis in the UCS.

**Up Vector:** The point that defines the direction of the Z-axis of the UCS.

Design Director: Views

Enables you to manipulate and create new views. To create a new view from the current display, select **Create New** in the Design Director toolbar or local menu.

When **Views** is selected in the upper pane of the Design Director, all views you saved (see "Saving Views" on page 102) will appear in the lower pane. The lower pane contains the following options:

Name	✓	VI...	Position	X Vector	Up Vector
View_0			15	x=0.1	x=0.1
From garden			35.0	x=6.1	x=5.0
Closeup of Porch			35.0	x=6.1	x=5.0
Toward pool			35.0	x=4.1	x=5.0

**Active:** Sets the view as active.

**Draft Rendering:** Sets the rendering type when the view is rendered in **Draft** rendering mode. See "Creating a Rendered View" on page 495 and "Draft Render Properties" on page 499

**Quality Rendering:** Sets the rendering type when the view is rendered in **Quality** rendering mode. See "Quality Render Properties" on page 500

**Perspective:** Activates **Perspective** mode. See "Camera Properties" on page 420.

**Angle:** The view angle for perspective mode.

**Position:** The location of the camera position.

**Target:** The location of the point the camera is facing.

**UP Vector:** A point defining the up direction of the camera.

**Extents:** The boundaries of the view.

Design Director: Cameras

Enables you to manipulate all the cameras and camera sets in the drawing (see "Camera Objects" on page 422).

To create a new camera or camera set, select **Create New** in the Design Director toolbar or local menu.

When **Cameras** or **Camera Sets** is selected in the upper pane of the Design Director, the lower pane has the following options:

Name	✓	VI...	Position	X Vector	Up Vector
from NE corner			35.0	x=4.1	x=5.0
from NW corner			35.0	x=6.1	x=5.0
South elevation	✓		1.48	x=5.1	x=4.0
Plan - garden			1.48	x=3.1	x=4.0

**Active:** Displays the view taken by the camera, and attaches the window to the camera.

**Visible:** Displays or hides the camera symbol in the drawing.

**Attached:** Creates a new window associated and synchronized with the camera.

**Draft Rendering:** Sets the rendering type when the view is rendered in **Draft** rendering mode. See "Creating a Rendered View" on page 495 and "Draft Render Properties" on page 499



**Quality Rendering:** Sets the rendering type when the view is rendered in **Quality** rendering mode. See "Quality Render Properties" on page 500

**Perspective:** Activates **Perspective** mode. See "Camera Properties" on page 420.

**Angle:** The view angle for perspective mode.

**Position:** The location of the camera position.

**Target:** The location of the point the camera is facing.

**UP Vector:** A point defining the up direction of the camera.

**Camera Sets**

A camera set is a group of cameras, which can be handy if you want to set rendering, perspective, and visibility parameters for multiple cameras.

- 1. Select **Cameras** in the upper pane of the Design Director.
- 2. On the lower pane, select the cameras you want to include in the camera set.
- 3. Select **Create Set** in the Design Director toolbar or local menu.
- 4. Select **Camera Sets** in the upper pane in order to see the defined camera sets in the lower pane.
- 5. Select the properties (visibility, render type, etc.) of the camera set. When the camera set is made **Active**, these properties will be automatically applied to all cameras in the set.

**Design Director: Lights**

Enables you to manipulate all the lights and light sets in the drawing (see "Lights" on page 513).

To create a new light or light set, select **Create New** in the Design Director toolbar or local menu.

When **Lights** or **Light Sets** is selected in the upper pane of the Design Director, the bottom pane has the following options:

Name				Type	Position	Target
★ Point_1				POINT	x=-0.252923 y=	x=0 y=0 z=0
★ Point_2				POINT	x=10.2217 y=9.	x=0 y=0 z=0
★ Point_3				POINT	x=10.2217 y=3.	x=0 y=0 z=0
★ Point_4				POINT	x=-0.252923 y=	x=0 y=0 z=0
★ Directional_5				DIRECTION	x=32.9166 y=11	x=4.98437 y=5.
★ Ambient_6				AMBIENT	x=0 y=0 z=0	x=0 y=0 z=0
★ Headlight_7				HEADLIGHT	x=0 y=0 z=0	x=0 y=0 z=0
★ Spot_8				SPOT	x=4.98437 y=5.	x=3.23861 y=5.

The lower pane contains the following options:

**On/Off:** Turns the light on and off.

**Visible:** Displays or hides the light indicator in the drawing.

**Color:** Sets the color of the light.

**Type:** Sets the type of light (headlight, spot light, etc.). See "Creating Lights - Light Types" on page 513.

**Position:** The coordinates of the light source, relevant for Point, Spot, and Sky lights.

**Target:** The target point of a directional light, relevant for Directional, Spot, and Sky lights.

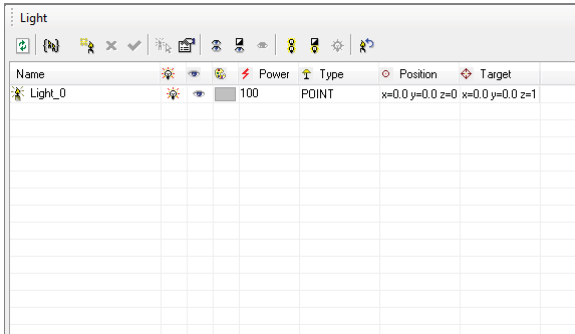
**Light Sets**

A light set is a group of lights, which can be handy if you want to set on/off, visibility, and other parameters for multiple lights.

- 1. Select **Lights** in the upper pane of the Design Director.
- 2. On the lower pane, select the lights you want to include in the camera set.
- 3. Select **Create Set** in the Design Director toolbar or local menu.
- 4. Select **Light Sets** in the upper pane in order to see the defined light sets in the lower pane.
- 5. Select the properties (on/off, visibility, etc.) of the light set.



Lights Dialog



- Refresh:** Refresh the Light manager display.
- New Light Template:** Creates a new light template from the currently selected layers.
- New Light:** Creates a new light. The default name will include a prefix by default, but the name can be changed.
- Delete Light:** Delete the selected lights.
- Activate:** Sets the currently selected layer (only one) as the active layer. This will effect any currently active drawing tool or currently selected object.
- Select By:** Selects the currently selected lights.
- Edit Properties:** Opens the light properties dialog.
- Visible/Invisible All:** If any layer is invisible all lights are made visible. If all lights are visible they are all made invisible.
- Invert Visibility:** Turns all visible lights invisible, and all invisible lights visible.
- Exclusive Visible:** Makes only the currently selected lights visible, all other become visible.
- ON/OFF All:** If any light is off all it is turned on. If all lights are on they are all turned off.
- Invert On:** Lights that are on are turned off, lights that are off are tuned on.
- Exclusive On:** Turns on only the currently selected lights, all others are turned off.

Design Director: Graphics

In the Design Director, “Graphics” refers to objects - 2D, 3D, cameras, etc. You can insert **single** objects into the Design Director in order to have easy access to zooming, visibility, and other properties of the objects.

NOTE: To add **groups** of objects to the Design Director, use **Categories**.

To place a graphic object into the Design Director:

1. Select the object in the drawing space.
2. In the Design Director select **Graphics** in the upper pane and select **Create New** from the toolbar or local menu.
3. Accept the default name or assign one that will help you recognize the object.

All objects you add to the Design Director will appear in the lower pane. The lower pane contains the following options:

Name	ZoomTo	Visible	Color	View by WorkPlane	Get WorkPlane
circle					
Polyline for Revolve					
Revolve solid					

- ZoomTo:** Zooms to the extents of the object.
- Visible:** Displays or hides the object.
- Color:** Sets the object’s pen color.
- View by WorkPlane:** Displays the view by the workplane that is set as **WorkPlane by Entity**. See "Workplane by Entity" on page 366.
- Get WorkPlane:** Sets the current workplane by this object.

### Design Director: Categories

In the Design Director, “Categories” refers to groups of objects - 2D, 3D, cameras, etc. You can insert a category into the Design Director in order to have easy access to zooming, visibility, and other properties of the object group.

NOTE: To add **single** objects to the Design Director, use **Graphics**.

To place a group of objects into the Design Director:

- 1. Select the objects in the drawing space.
- 2. In the Design Director select **Categories** in the upper pane and select **Create New** from the toolbar or local menu.
- 3. Accept the default name or assign one that will help you recognize the object.

All object groups you add to the Design Director will appear in the lower pane. The lower pane contains the following options:

Name					
Cameras			+	-	
Polylines for site			+	-	
House outlines			+	-	

**ZoomTo:** Zooms to the extents of the objects.

**Visible:** Displays or hides the objects.

**Color:** Sets the objects’ pen color.

**Add To Category:** Adds the selected object to the category.

**Remove From Category:** Removes the selected object from the category.

### PDF Underlays

Available in TurboCAD Pro and Platinum only

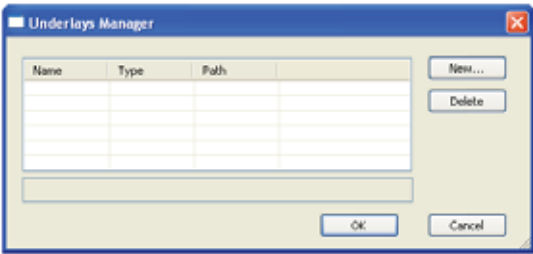
PDF Underlays allow you to insert pages of PDF files into your drawings to use as references. If the PDF contains vector data you will be able to snap to the endpoints and midpoints of lines in the PDF underlay. The underlay will appear as the bitmap data of the PDF. At different zoom levels you may have to use Regen or Redraw to get the maximum resolution for the display of the underlay.

Underlays consist of two parts: the underlay style and the underlay insertion.

Before inserting an underlay in you must create an Underlay style.

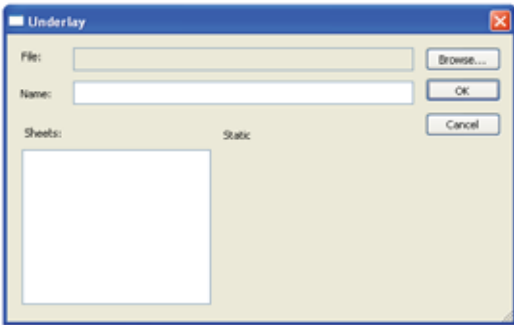
#### Creating an Underlay Style

Open the Underlay Manager.

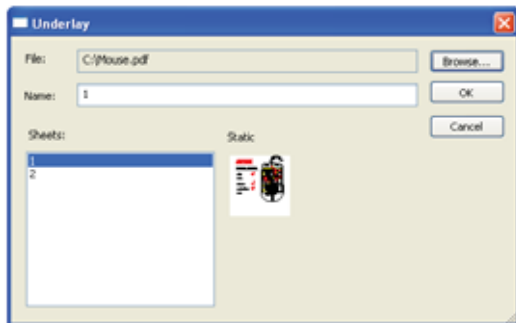


It contains a list of underlay styles currently defined for the drawing.

- 1. Click the New button to create a new Underlay Style. The Underlay dialog opens.



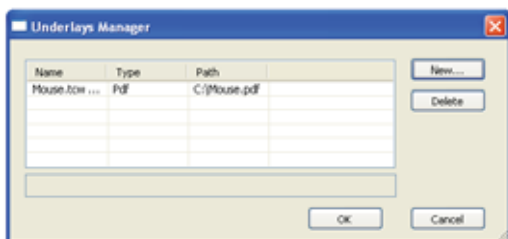
- Click Browse button to display File Open dialog and choose a pdf file.



- Now you can see the list of pages of the selected PDF document and a preview picture for the selected page. By default the page name is used as the Name of the Underlay style but you can change it.



- Click OK to close the Underlay dialog. Now you will see the new Underlay Style is added to the Styles List.

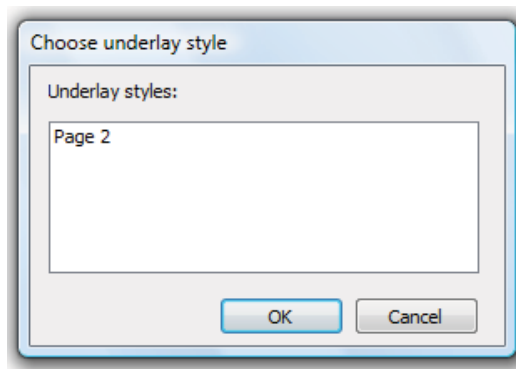


- Click OK to close Underlay Manager dialog

## Inserting an Underlay in a Drawing

### To insert a PDF Underlay:

- Select the Underlay tool.
- You will be prompted to select the desired style. Select the Underlay Style you want then click OK.



- Click to define left bottom corner of the underlay insertion.
- Move the cursor to the desired location and click again to define right top corner of the underlay insertion.





# 5 Inserting Objects

The **Draw** menu contains all of the TurboCAD drawing tools, as well as tools for inserting other types of objects into your drawing, such as files, pictures, and OLE objects.

---

NOTE: The tools covered in this section are 2D. For 3D geometry, see "Creating 3D Objects" on page 371.

---

You can set object properties before they are created, or modify properties of existing objects. See "Object Properties" on page 79.

Before creating 2D objects, it is a good idea to familiarize yourself with the following concepts:

- See "Coordinate Systems" on page 105.
- See "Grid" on page 107.
- See "Snaps" on page 108.
- See "Layers" on page 116.
- See "Construction Geometry" on page 121.

For most objects, you can define size and location by using the cursor, or by specifying exact sizes or dimensions. See "Inspector Bar" on page 45 and "Coordinate Fields" on page 46.

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NOTE: If you are inserting 2D objects while working in 3D, the objects will be placed on the current workplane. See "Workplanes" on page 362.

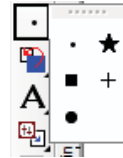
---

## Point

You can display the **Point** toolbar by right-clicking in any toolbar area and selecting **Point**.



These tools are also available on the fly-out toolbar from the **Drawing Tools**.



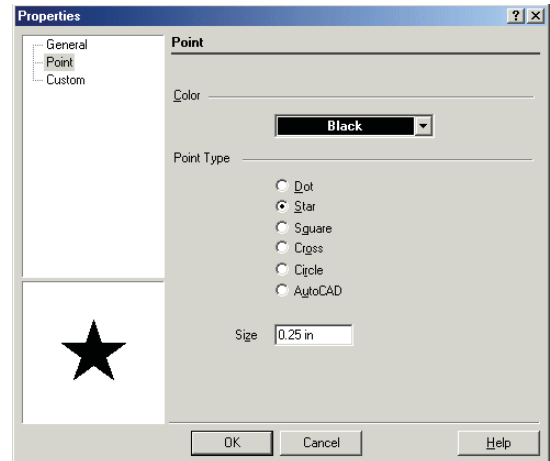
Use **Point** to create several different types of point markers: dot, square, cross, star, or circle.

By default, a point in 2D space. If you want to create a 3D point, select **3D Point** from the Inspector Bar or local menu.



## Point Properties

The **Point** page is included in the **Properties** window when a point is selected, or when the point tool is active.



**Color:** Select the point color.

**Point Type:** Select dot, star, square, cross, circle, or the AutoCAD point.

**Size:** Sets the width and height of the point. This value affects all types of points except **Dot**, which is always displayed at a minimal size.

## Line

Tools for drawing lines and linear objects.

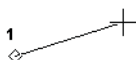
You can display the **Line** toolbar by right-clicking in any toolbar area and selecting **Line**.

### Single Line

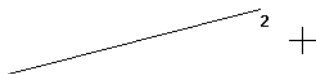


Creates a single line.

1. Define the segment start point.



2. Define the endpoint, or specify the length and angle in the Inspector Bar.



### Polyline



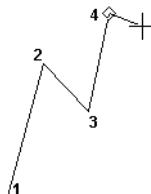
Creates a series of connected straight line or arc segments that comprise one object. The segments can have different and/or tapered line widths.

---

**NOTE:** If you want to create a polyline by joining a series of existing line or arc segments, see "Join Polyline" on page 243 or "Chain Polyline" on page 244.

---

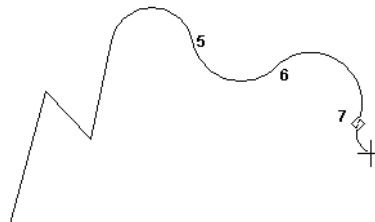
1. By default, line segments are created (as opposed to arcs). Select each endpoint, or enter segment lengths and angles in the Inspector Bar.



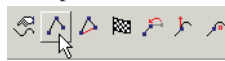
2. To draw arc segments, select **Arc Segment** from the local menu or Inspector Bar.



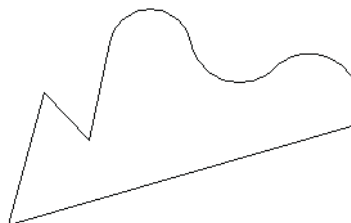
3. By default, each arc segment is tangent to the previous segment, but the tangency can be changed via the local menu options. Select each arc endpoint, or enter parameters in the Inspector Bar.



4. To switch back to lines, select **Line Segment** from the local menu or Inspector Bar.

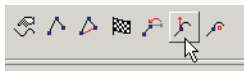


5. When all segments are created, select **Finish** (Alt+F) from the local menu or Inspector Bar, or double-click the last point. If you want to create a line or arc segment connecting the first and last points, select **Close** (Alt+C).

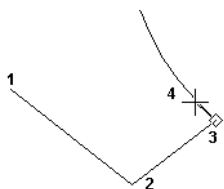


Local menu options (Arcs):

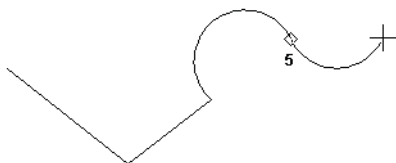
**Direction:** Rather than creating an arc segment tangent to the last segment, you can set the start angle of the arc segment.



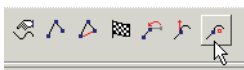
1. For the arc segment, move the mouse to set the start angle, or enter the angle in the Inspector Bar.



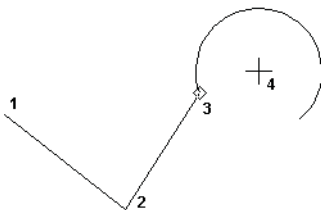
2. Select the arc endpoint, or enter its parameters in the Inspector Bar. The next arc segment you create will revert to the default - tangent to the previous segment.



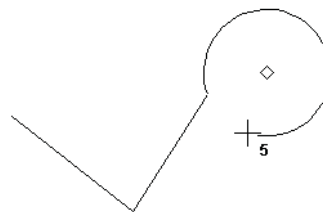
**Center:** Rather than creating the arc tangent to the last segment, you can set the start angle and size by selecting the arc center.



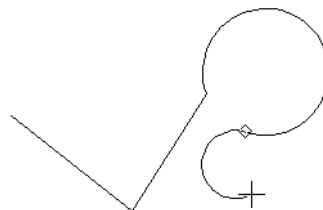
1. For the arc segment, select the arc center, or enter the arc length in the Inspector Bar.



2. Size the arc by selecting its endpoint, or enter its angle or arc length in the Inspector Bar.

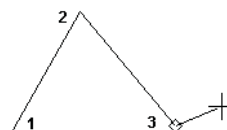


The next arc segment you create will revert to the default - tangent to the previous segment.

**Polyline Line Widths**

There are two additional fields in the Inspector Bar for assigning widths to polyline segments - **Start Width** and **End Width**.

By default, polyline segments have the default line thickness (which can be set in the **Pen Width** box on the **Property** toolbar.)



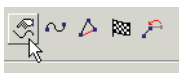
For any segment, you can enter values for **Start** and **End Width**.

Start Width	End Width
0.05 in	0.12 in

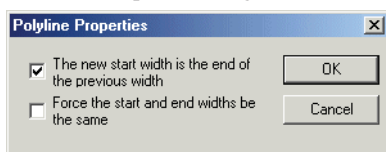
The next segment will have these properties, but the subsequent segments, by default, will revert to the default thickness.



To change the widths of adjacent segments you can open the **Polyline Properties** from the local menu or Inspector Bar.



In this window, you can force start and end widths to be equal, and to set the start width of each segment to be equal to the end width of the previous segment.



Activate both options if you want to apply a uniform thickness to the entire polyline, or to a contiguous group of segments.

---

**NOTE:** If you use **Start** and **End Width** to create a segment with tapering thickness, this taper will be lost when the polyline is exploded, and then the tapered segment is exploded.

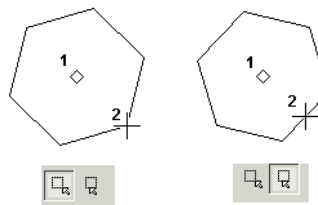
---

## Polygon



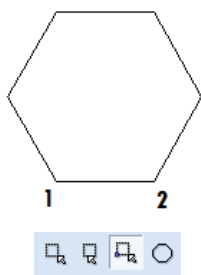
Creates a regular (equal-length sides) polygon.

1. Enter the number of sides in the Inspector Bar. If desired, you can also specify the **Angle** (angle of control point from the center), and **Radius** or **Side** (length of one side).
2. Select the mode: Vertex, Segment or Edge.
3. If you are using Vertex or Segment mode, select the polygon centerpoint.
4. Use the default **Vertex Mode**, which uses one of the vertices for a control point. Or, select **Segment Mode** from the local menu or Inspector Bar to use the midpoint of one segment as a control point.



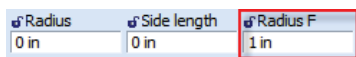


5. Or, select **Edge Mode** from the local menu or Inspector Bar to use the midpoint of one segment as a control point.



### Options

**Fillet Corners:** If this option is turned on the corners of the polygon will be rounded by filleting. A Radius F field appears in the Inspector bar that allows you to specify the radius of the fillets.



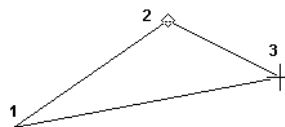
**Create Pattern Constraint:** (*Mechanical Edition Only*) If this option is turned on a pattern constraint will be created from the resulting polygon. Auto Add Constraints must be turned on for this feature to work.

## Irregular Polygon

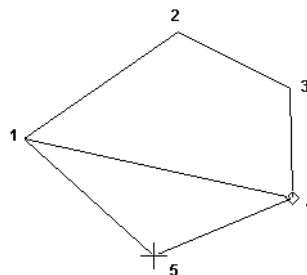
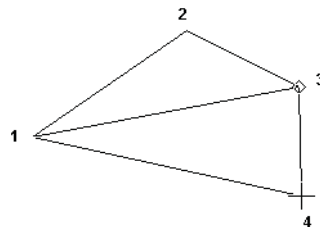


Creates a polygon (closed) with irregular sides and angles. This is equivalent to using the **Polyline** tool to create line segments in “closed” mode.

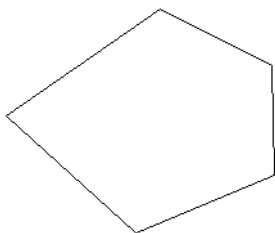
1. Define the first two vertices, creating the first segment. You can also use the Inspector Bar to enter a length and angle.
2. Define the next segment. From this new vertex, a closing segment is created back to the first vertex.



3. Create as many segments as needed. A closing segment is always created from the last vertex.



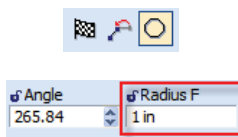
4. Select **Finish** from the local menu or Inspector Bar, or press Alt+F.



**NOTE:** To undo polygon segments in reverse order, select **One Step Back** from the local menu or Inspector Bar (or press Backspace).

### Options

**Fillet Corners:** If this option is turned on at the corners of the polygon will be rounded by filleting. A Radius F field appears in the Inspector bar that allows you to specify the radius of the fillets.

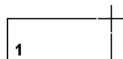


## Rectangle



Creates an orthogonal rectangle by defining two diagonally opposite corners.

1. Select the first corner.

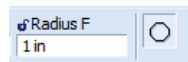


2. Select the diagonally opposite corner. You can also enter the size in the Inspector Bar.



### Options

**Fillet Corners:** If this option is turned on at the corners of the polygon will be rounded by filleting. A Radius F field appears in the Inspector bar that allows you to specify the radius of the fillets.

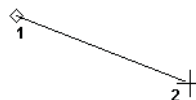


## Rotated Rectangle

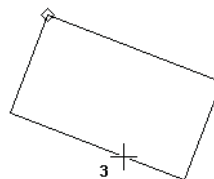


Creates a non-orthogonal rectangle.

1. Select two points to define the base. The angle of this line defines the orientation of the rectangle.



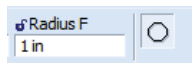
2. Define a third point that specifies the distance from the base to the top.



You can also enter the length of both sides and the orientation angle in the Inspector Bar.

### Options

**Fillet Corners:** If this option is turned on a the corners of the polygon will be rounded by filleting. A Radius F field appears in the Inspector bar that allows you to specify the radius of the fillets.



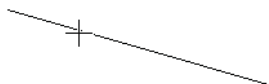
## Perpendicular



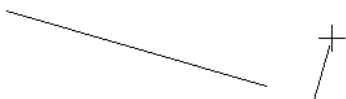
Creates a line perpendicular to an existing line.

**TIP:** This tool creates a line **from** an existing line. To create a perpendicular line **to** a line, you can use the "J" SEKE (see "SEKE's" on page 110).

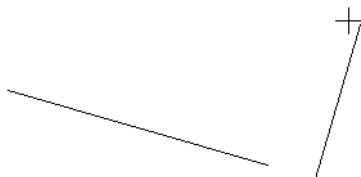
1. Select an existing line.



2. Select the point on the existing line where the perpendicular line will start. You can select a point past the endpoints.

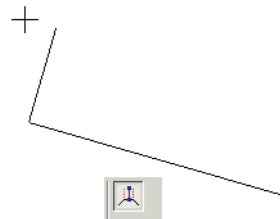


3. Select a third point to set the length of the perpendicular line, or enter the length in the Inspector Bar.



### Local menu option:

**Limit to Segment:** The perpendicular line cannot extend past the endpoints of the existing line.



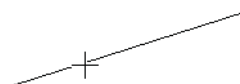
**TIP:** To draw a line perpendicular to an arc or circle, draw a single **Line** from the center of the arc or circle to its destination, then trim the line.

## Parallel

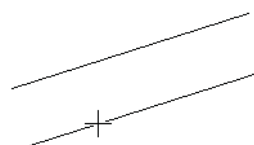


Creates a line parallel to a existing line.

1. Select an existing line.

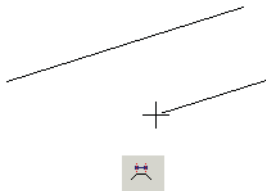


2. Select the location of the parallel line, or enter the offset in the Inspector Bar. By default, the parallel line will have the same length as the original line.



Local menu option:

**Keep Length** (default): If not used, you can create a parallel line with a different length than the original line. You can only adjust one endpoint of the line - the endpoint closest to where you selected the original line. You can enter both the offset and length in the Inspector Bar.



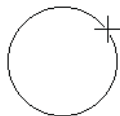
**TIP:** To create multiple parallel lines separated by the same distance, lock the **Offset** field in the Inspector Bar (click the lock symbol above the field). For each parallel line, select a source line and indicate the side where the parallel line will be created. You can also use **Offset** (see "Offset" on page 211).

## Tangent Arc Point

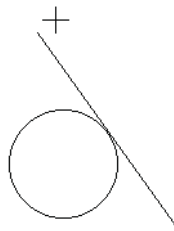


Creates a line tangent to an arc, circle, or ellipse, with the midpoint of the line located at the point of tangency.

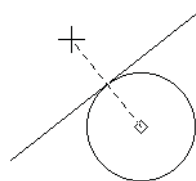
1. Select the tangent arc, circle, or ellipse.



2. Move the mouse to adjust the length of the line, or set the length in the Inspector Bar. The line midpoint always remains at the tangent point.



3. Move the mouse to change the angle of the line and the point of tangency, or set the angle in the Inspector Bar.

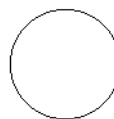


## Tangent to Arc

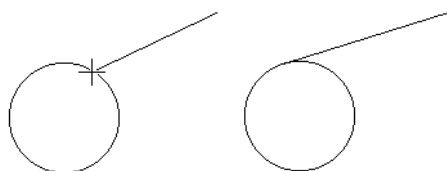


Creates a line tangent to an arc, circle, or ellipse, starting from a selected point.

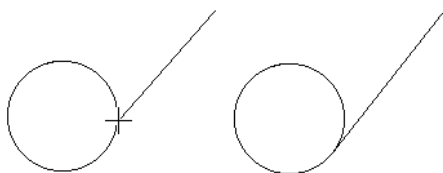
1. Select the start point of the line.



2. Select the tangent arc, circle, or ellipse, close to the point of tangency. The tangent line is created.

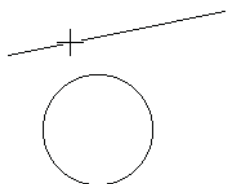


In this example, clicking on the other side of the circle would have created a different tangent line.

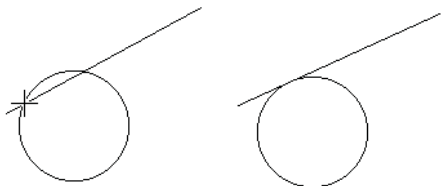


### Tangent Line of Fixed Length

By default, **Tangent to Arc** creates a line from the start point, ending at the tangent object. To fix the length of the line, enter the length in the Inspector Bar and lock it, but do not press Enter. When you move the cursor, the line length remains constant.



Select the tangent object, close to the point of tangency. The fixed-length tangent line is created.

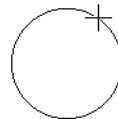


### Tangent from Arc



Creates a tangent line from an arc, circle, or ellipse.

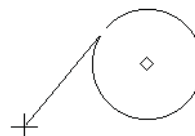
1. Click on the side of the arc or circle from which you want to draw the tangent line.



2. A tangent line appears from the selected point. Click to set the length, or enter the length in the Inspector Bar.



3. Move the cursor to set the line angle, or enter the angle in the Inspector Bar.



To switch the side of the tangent line, pass the cursor through the tangency point.

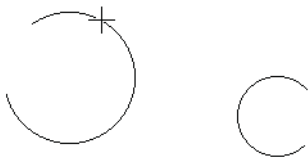


### Tangent to 2 Arcs

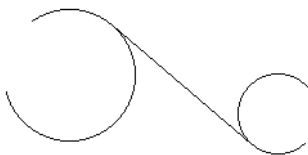
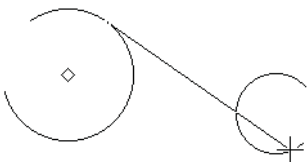


Creates a line tangent to two arcs or circles.

1. Select one of the tangent objects - arc, circle, or ellipse.



2. The tangent line extends from this object. To switch the side of the line, pass the cursor through the point of tangency. Adjust the line so that it is close to the desired point of tangency.
3. Select the second tangent object, close to the point of tangency. The tangent line is created.

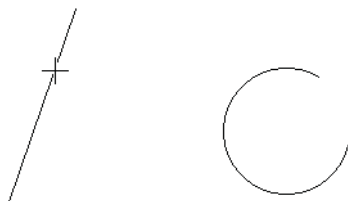


## Minimal Distance

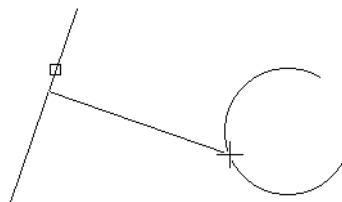


Creates the line representing the shortest distance between two 2D objects. The objects must be on the same workplane.

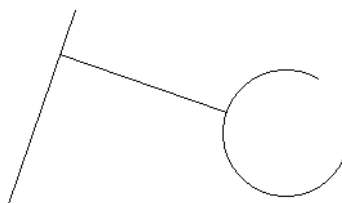
1. Select the first object from which you want to draw the shortest line.



2. Select the second object.

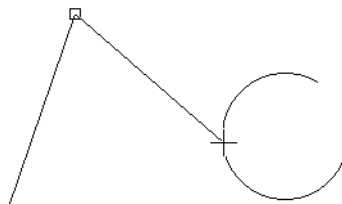
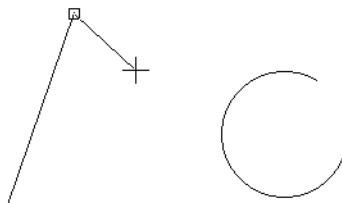


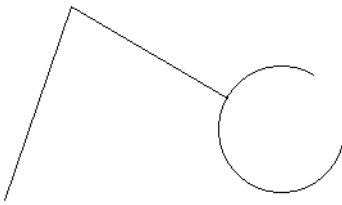
3. The shortest-distance line is created.



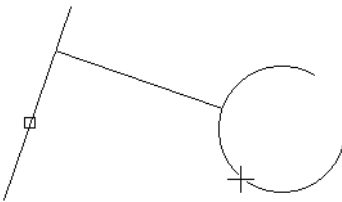
### Local menu options:

**Through Point:** Creates a line passing through a specified point. The through point should only be specified for one of the objects, otherwise a standard single line is created.





**Show beforehand** displays the shortest line to an object when you pass the cursor over it. This allows you to preview the line before creating it.



## Double Line

Tools for drawing double (parallel) lines and linear objects. The **Double Line** tools are essentially similar to the **Line** tools, but there are no tangent tools.

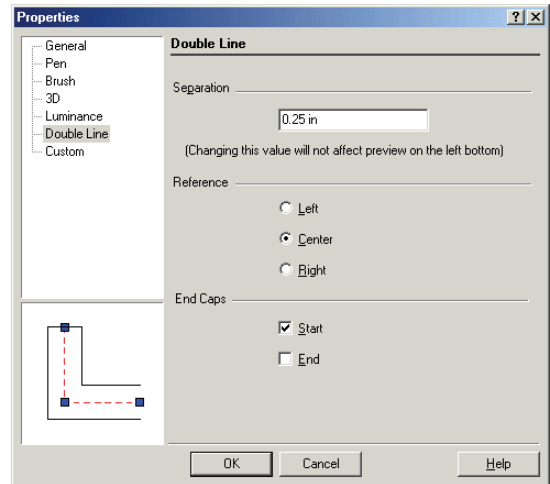
You can display the **Double Line** toolbar by right-clicking in any toolbar area and selecting **Double Line**.



**NOTE:** If you want to use double lines to create representations of walls, there is a special tool for this purpose. See "Wall Tools" on page 535.

## Double Line Properties

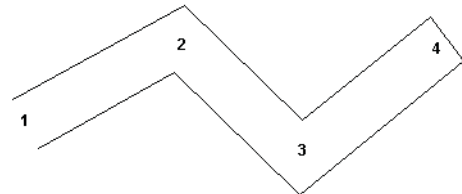
Double Lines have unique properties that you can set in the **Double Line** tab of the **Properties** window.



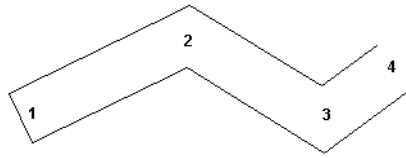
**Separation:** The distance between the lines. If you stretch or scale a double line, this distance does not change.

**Reference:** Choose whether to draw the double line by its center, or left or right side. Right and left sides are determined by "facing" toward the start point. This setting affects how the object is snapped, and where the nodes are located when in the **Edit Tool**.

**End Caps:** Closes the endpoints of the double line.



**End Cap at End**



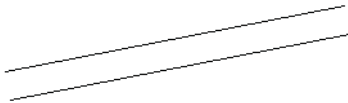
**End Cap at Start**

**TIP:** Another way to create end caps is to use the **Edit Tool** (node editing), right-click on the end node and select **Close Double End**. See "Edit Tool" on page 219.

## Double Line: Single Line



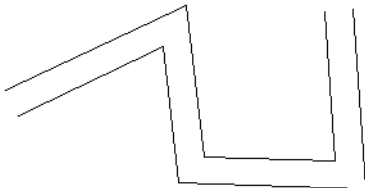
Creates one double line. For the interaction, see "Single Line" on page 138.



## Double Line: Polyline



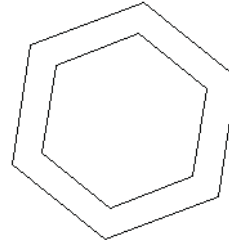
Creates a series of connected straight double line segments (no arc segments) that comprise one object. For the interaction, see "Polyline" on page 138.



## Double Line: Polygon



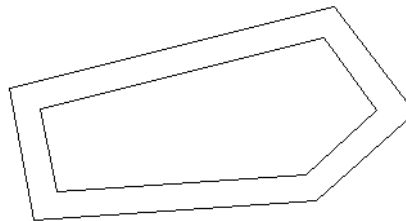
Creates a regular (equal-length sides) double-line polygon. For the interaction, see "Polygon" on page 140.



## Double Line: Irregular Polygon



Creates a double-line polygon (closed) with irregular sides and angles. For the interaction, see "Irregular Polygon" on page 141.

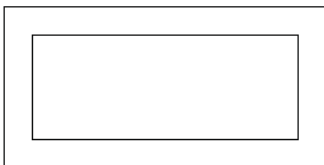




### ***Double Line: Rectangle***



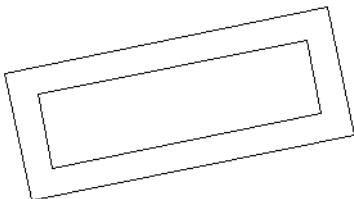
Creates an orthogonal double-line rectangle by defining two diagonally opposite corners. For the interaction, see "Rectangle" on page 142.



### ***Double Line: Rotated Rectangle***



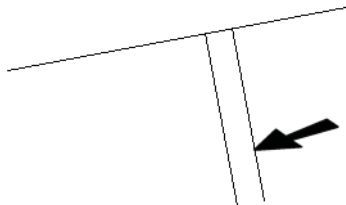
Creates a non-orthogonal double-line rectangle. For the interaction, see "Rotated Rectangle" on page 142.



### ***Double Line: Perpendicular***



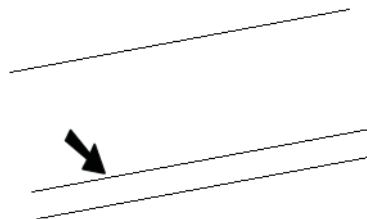
Creates a double line perpendicular to an existing line. For the interaction, see "Perpendicular" on page 143.



### ***Double Line: Parallel***



Creates a double line parallel to an existing line. For the interaction, see "Parallel" on page 143.



## **Multi Line**

Tools for drawing multi lines and linear objects.

Multi Line objects consist of two or more parallel lines, created as one object. The **Multi Line** tools are essentially similar to the **Line** tools, except that there are no tangent tools.

You can display the **Multi Line** toolbar by right-clicking in any toolbar area and selecting **Multi Line**.

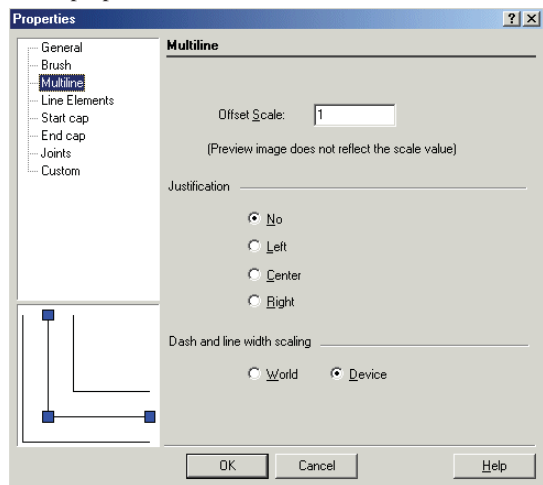


## Multi Line Properties

Five pages of the **Properties** window contain options relevant to Multi Lines.

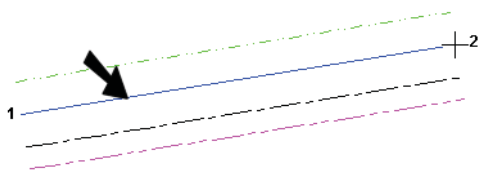
### Multiline Properties

General properties for multi lines:

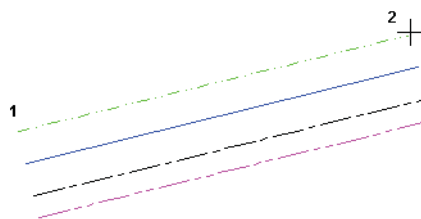


**Offset Scale:** Controls the overall width of the multi line.

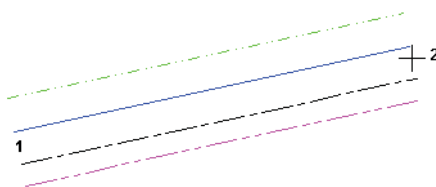
**Justification:** Sets the alignment of the multi line nodes. Left and right are determined by “facing” toward the multi line start point. No justification is the default; the line is aligned to a zero offset, relative to the line offsets defined in the **Line Elements** page.



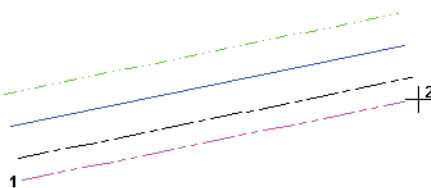
**No Justification**  
(Indicated line has zero offset)



**Left Justification**



**Center Justification**



**Right Justification**

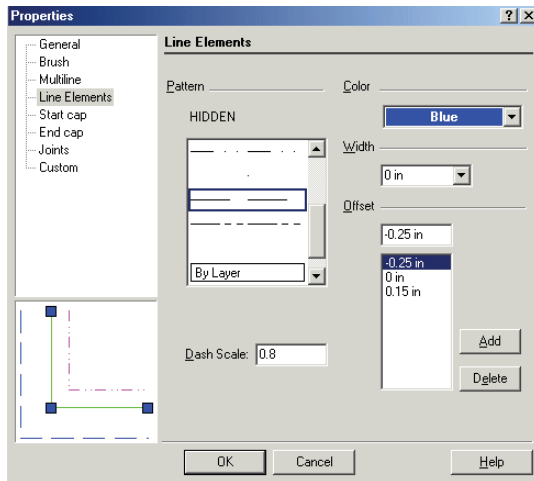
**Dash and line width scaling:** Determines whether the width of the multi lines will be scaled or will remain the same size when zoomed. This setting also applies to the size of the elements in the pen's dot-and-dash pattern.

- **Device:** The pen's width and pattern sizes are defined in device units (the monitor or printer). If you zoom the line width and pattern size will not change on the screen.
- **World:** The pen's width and pattern sizes are defined in by the drawing spaces units. If you zoom the line width and pattern size will change in accordance with the zoom factor.

- **Device Width:** The pen's pattern size is defined by the drawing spaces units, and pen's width is defined in device units (the monitor or printer). If you zoom the line patterns size will change, but the pen width size will not change.

## Line Elements Properties

Properties for each line comprising the multiline.



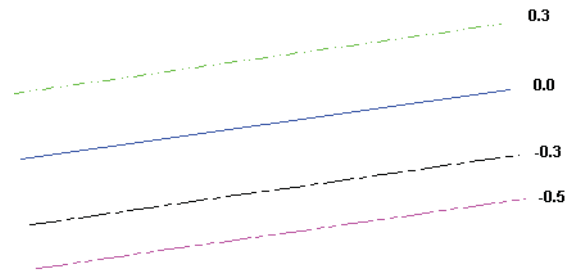
**Pattern:** Select a pattern (continuous, dashed, etc.) for each line, or select **By Layer** or **By Block**. The exact look of each pattern depends on the setting of **Dash scale** and **width** on the **Multiline** page.

**Dash Scale:** The scale of the dot-and-dash pattern.

**Color:** Select a color for each pen. **By Layer** and **By Block** are available.

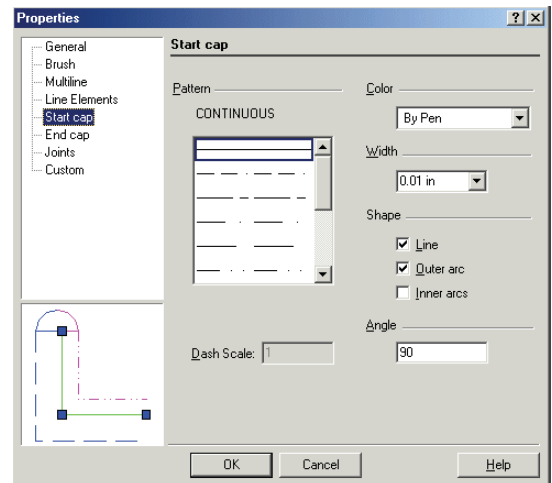
**Width:** Specify the width of each line. Zero width means one screen pixel, and will print at one unit of the printer's available resolution (a 300 dots-per-inch printer will print a zero width line at 1/300".)

**Offset:** For each line, define its offset value. Use **Add** to create new lines, **Delete** to remove lines.



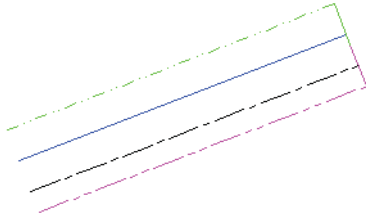
## Start Cap and End Cap Properties

Start and end caps are used to close the ends of the multiline. The options are the same for start caps and end caps.

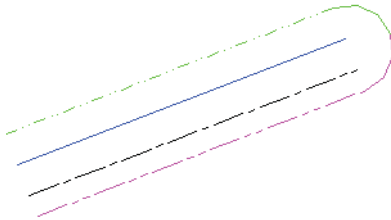


For **Pattern**, **Dash Scale**, **Color**, and **Width**, see "Line Elements Properties" on page 151.

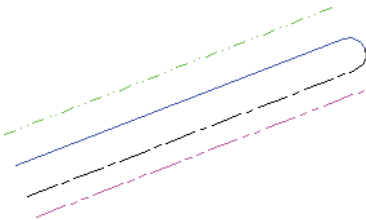
**Shape:** Options for creating the start / end cap. They can be used in combination. **Inner Arc** closes the lines adjacent to the outer lines, if there are four or more total lines.



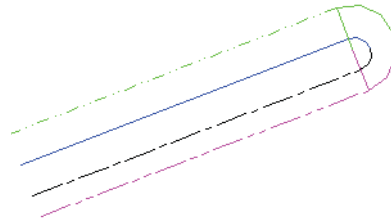
**End Cap - Line**



**End Cap - Outer Arc**

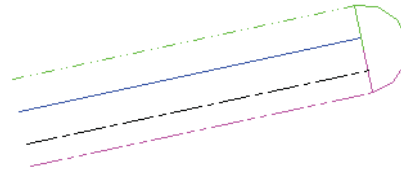


**End Cap - Inner Arc**

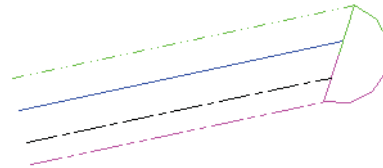


**End Cap - Line, Outer Cap, and Inner Cap**

**Angle:** Sets the angle of the cap, relative to the direction of the multiline. The default value is 90 degrees.



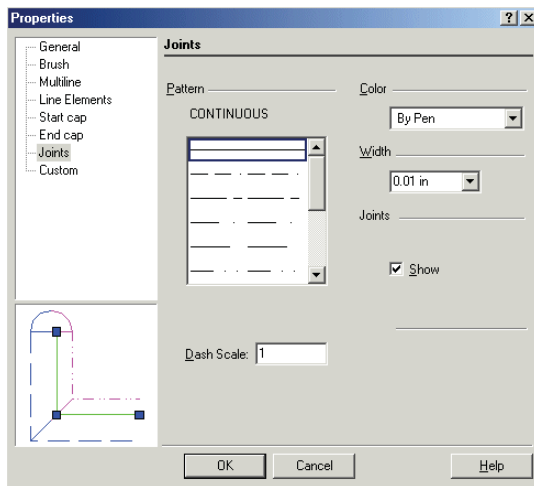
**End Cap - Angle = 90**



**End Cap - Angle = 60**

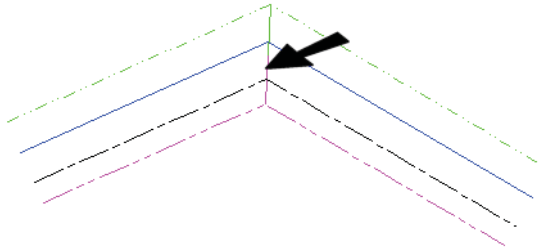
## Joins Properties

Use joints to show breaks at each multi line segment.



For **Pattern**, **Dash Scale**, **Color**, and **Width**, see "Line Elements Properties" on page 151.

**Show:** Choose to show or hide the joints.

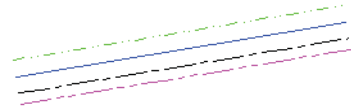


*NOTE: Multilines can be used to trim objects, and can be trimmed themselves. However when trimming multilines you need to click near the center line of the trimmed multiline.*

## Multi Line: Single Line



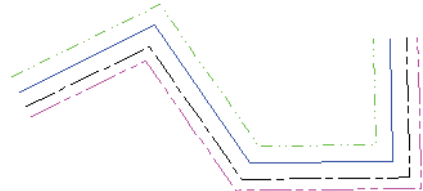
Creates one multi line. For the interaction, see "Single Line" on page 138.



## Multi Line: Polyline



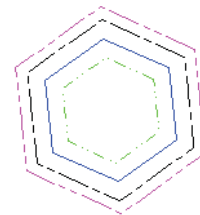
Creates a series of connected straight multi line segments (no arc segments) that comprise one object. For the interaction, see "Polyline" on page 138.



## Multi Line: Polygon



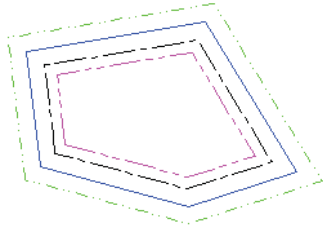
Creates a regular (equal-length sides) multi line polygon. For the interaction, see "Polygon" on page 140.



## ***Multi Line: Irregular Polygon***



Creates a multi line polygon (closed) with irregular sides and angles. For the interaction, see "Irregular Polygon" on page 141.



## ***Multi Line: Rectangle***



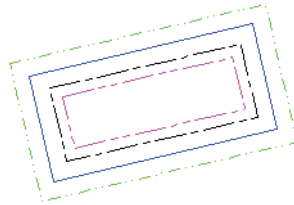
Creates an orthogonal multi line rectangle by defining two diagonally opposite corners. For the interaction, see "Rectangle" on page 142.



## ***Multi Line: Rotated Rectangle***



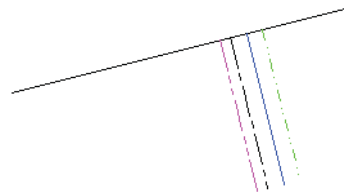
Creates a non-orthogonal multi line rectangle. For the interaction, see "Rotated Rectangle" on page 142.



## ***Multi Line: Perpendicular***



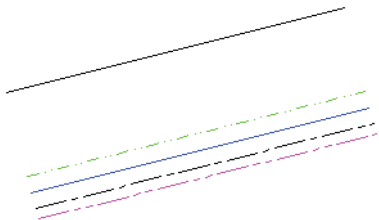
Creates a multi line perpendicular to an existing line. For the interaction, see "Perpendicular" on page 143.



## Multi Line: Parallel



Creates a multi line parallel to a existing line. For the interaction, see "Parallel" on page 143.



## Circle / Ellipse

Tools for drawing circles and ellipses.

You can display the **Circle/Ellipse** toolbar by right-clicking in any toolbar area and selecting **Circle/Ellipse**.



### Circle: Center and Point

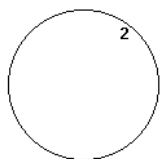


Creates a circle by defining its center and a point on its circumference.

1. Select the circle centerpoint. Move the cursor to size the circle.



2. Click to create the circle, or enter the radius, diameter, or circumference in the Inspector Bar.

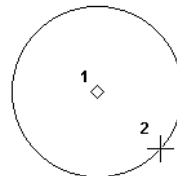


## Circle: Concentric

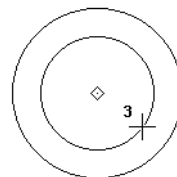


Creates circles that share a centerpoint.

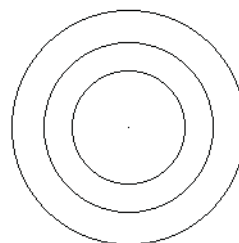
1. Select a centerpoint for the circles.
2. Size the first circle, or enter the radius, diameter, or circumference in the Inspector Bar.



3. Create the second circle the same way.



4. Create more circles as needed.



5. Finish by selecting **Finish** from the local menu or Inspector Bar, or press Alt+F.

## ***Circle: Double Point***

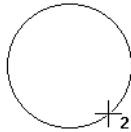


Creates a circle by defining two endpoints of its diameter.

1. Select a point on the circle circumference.



2. Select the point on the opposite end of the diameter line, or enter the radius, diameter or circumference, and angle of the diameter line in the Inspector Bar.

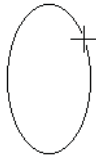


## ***Circle: Tangent to Arc***



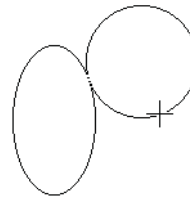
Creates a circle tangent to an arc, circle, or ellipse.

1. Select the existing arc, circle, or ellipse to which you want the circle to be tangent.

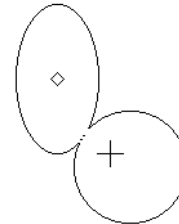


2. Select the centerpoint of the circle, or enter the radius, diameter, or circumference in the Inspector Bar. (If you use the Inspector Bar and then press Enter, you will not be able to reposition the circle.)

The circle will be created from the point you selected on the tangent object.



3. Move the mouse to position the circle, either inside or outside the tangent object.

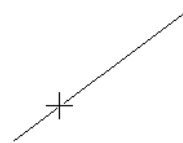


## ***Circle: Tangent to Line***



Creates a circle tangent to a line.

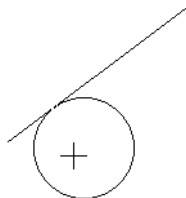
1. Select the existing line to which you want the circle to be tangent.



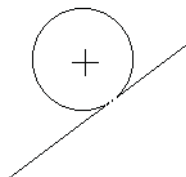
2. Select the centerpoint of the circle, or enter the radius, diameter, or circumference in the Inspector Bar. (If you use the Inspector Bar and then press Enter, you will not be able to reposition the circle.)



The circle will be created from the point you selected on the line.



3. Move the mouse to position the circle, on either side of the line.

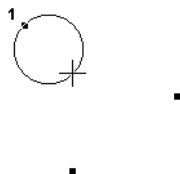


### ***Circle: Triple Point***

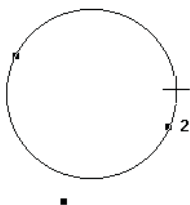


Creates a circle that passes through three points.

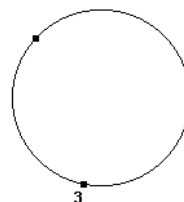
1. Select the first point on the circumference.



2. Select the second point.



3. Select the third point.



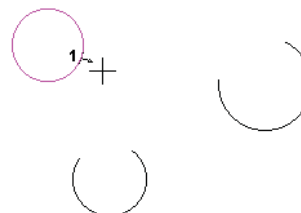
### ***Circle: Tangent to 3 Arcs***



Creates a circle tangent to three arcs or circles. You can specify whether the existing arcs will be inside or outside the new circle.

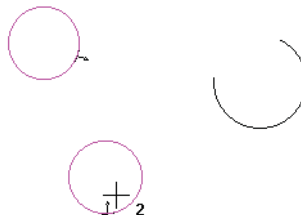
1. Select the first tangent object. Click slightly outside the object to keep it outside the new circle. Click inside if you want the tangent object to be inside the new circle.

If the object is clicked slightly outside, an outward-pointing arrow appears.

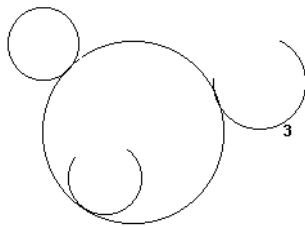


2. Select the second tangent object.

If the object is clicked slightly inside, an inward-pointing arrow appears.



3. Select the third object. The tangent circle is created.

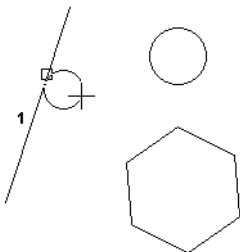


### ***Circle: Tangent to Entities***

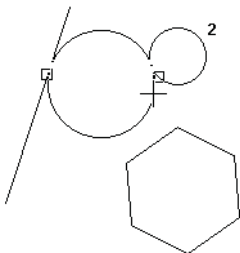


Creates a circle tangent to three objects (rectangles, polygons, arcs, etc.)

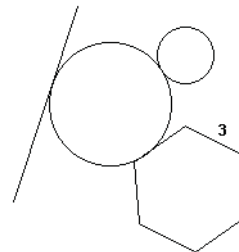
1. Select the first tangent object.



2. Select the second tangent object.

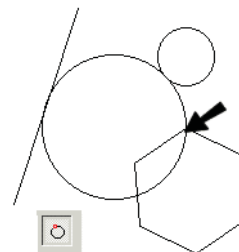


3. Select the third tangent object. The tangent circle is created.



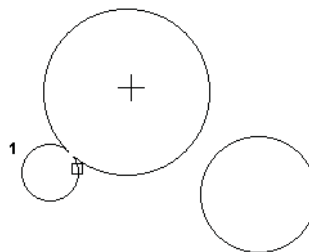
#### Local menu option:

**Through point:** Makes the circle pass through a specific point. In the example below, a vertex of the polygon was selected, rather than the polygon itself.

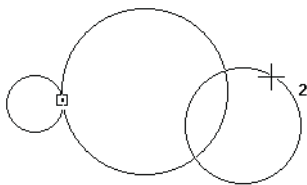


### **Tangent Circle of Fixed Size**

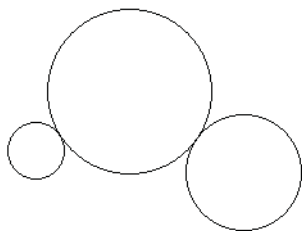
By default, **Tangent to Entities** creates a circle that touches three objects. To fix the size of the circle, enter the radius, diameter, or circumference in the Inspector Bar and lock it, but do not press Enter. When you move the cursor, the circle size remains constant.



Select the second tangent object, close to the point of tangency.



The fixed-sized tangent circle is created.

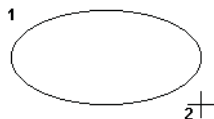


## Ellipse



Creates an ellipse by defining a bounding rectangle for the ellipse. The axes of the ellipse will be orthogonal.

1. Select the first corner of the bounding rectangle.
2. Define the diagonally opposed corner of the bounding rectangle, or enter the major and minor axis lengths in the Inspector Bar.



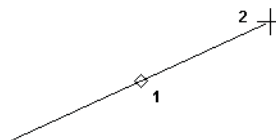
## Rotated Ellipse



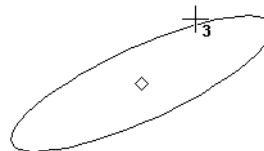
Creates a tilted ellipse.

1. Select the centerpoint of the ellipse.

2. Move the mouse to set the length and angle of the major axis, or enter the major axis length and angle in the Inspector Bar.



3. Set the length of the minor axis, which is always perpendicular to the major axis. You can also enter the axis length in the Inspector Bar.



## Ellipse Fixed Ratio

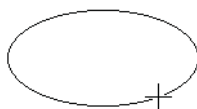


Creates an ellipse by specifying its aspect ratio - the ratio of the major axis to the minor axis. The axes of the ellipse are orthogonal.

1. Enter the aspect ratio in the **a:b** ratio field on the Inspector Bar. (Lock this value if you want to repeat it later, otherwise the field reverts back to the default.)
2. Select the centerpoint of the ellipse.
3. Move the mouse to size the ellipse.



**a:b = 0.5**



a:b = 2.0

TIP: The fixed ratio for an "IsoCircle" (an isometric circle on a cube) is 1.73 (the square root of 3).

## Arc

Most of the **Arc** tools are identical to **Circle/Ellipse** tools. After the initial circle is created, the arc is cut from it by defining the start and end angles.

You can display the **Arc** toolbar by right-clicking in any toolbar area and selecting **Arc**.



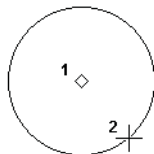
Start and end angles can be changed with the **Edit Tool** (node editing). See "Changing Start and End Angles" on page 226.

### Arc: Center and Radius

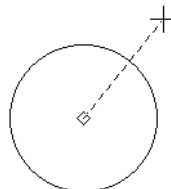


Creates an arc by defining its center, a point on its circumference, and the start and end angles.

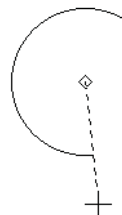
1. Select the arc centerpoint and a point on the circumference of the circle from which the arc will be cut. You can also specify the radius, diameter, or circumference in the Inspector Bar.



2. A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



3. Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.

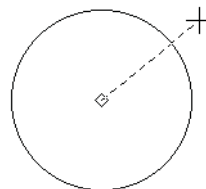


### Arc: Concentric

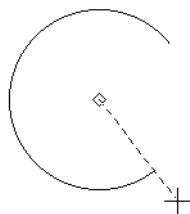


Creates arcs that share the same centerpoint.

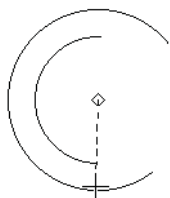
1. Create the first arc by selecting the centerpoint, then moving the mouse and clicking to define its size. You can also enter the radius, diameter, or circumference in the Inspector Bar.
2. A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



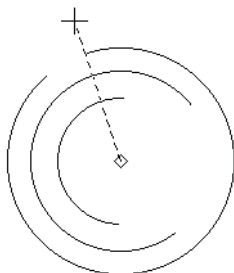
3. Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.



4. Create the next arc the same way. The centerpoint is already defined, you just need to define the size, and start and end angles.



5. Repeat these steps to draw multiple concentric arcs.



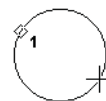
6. Double-click to finish, select **Finish** from the local menu or Inspector Bar, or press Alt+F.

## Arc: Double Point

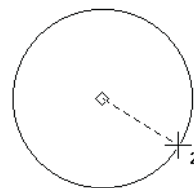


Creates an arc by defining two endpoints of its diameter.

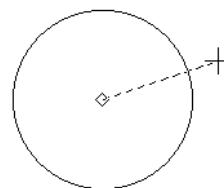
1. Select a point on the circumference of the circle from which the arc will be cut.



2. Select the point on the opposite end of the diameter line, or enter the radius, diameter or circumference, and angle of the diameter line in the Inspector Bar.



3. A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



4. Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.

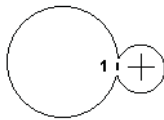


## Arc: Tangent to Arc



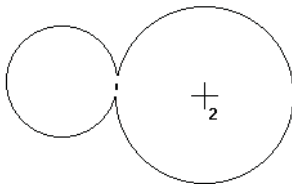
Creates an arc tangent to an arc, circle, or ellipse.

1. Select the existing arc, circle, or ellipse to which you want the arc to be tangent. The tangent circle, from which the arc will be cut, extends from the point you selected.

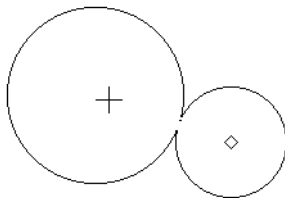


2. Select the centerpoint of the tangent circle, or enter the radius, diameter, or circumference in the Inspector Bar. (If you use the Inspector Bar and then press Enter, you will not be able to reposition the circle.)

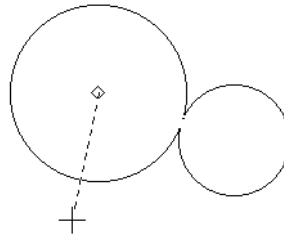
The circle will be created from the point you selected on the tangent object.



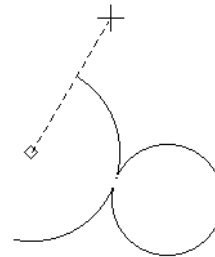
3. Move the mouse to position the circle, either inside or outside the tangent object.



4. A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



5. Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.

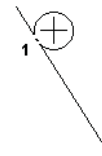


## Arc: Tangent to Line



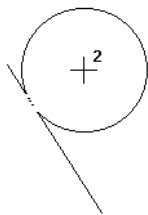
Creates an arc tangent to a line.

1. Select the existing line to which you want the arc to be tangent. The tangent circle, from which the arc will be cut, extends from the point you selected.

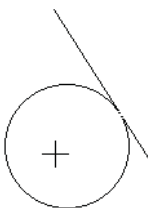


2. Select the centerpoint of the circle, or enter the radius, diameter, or circumference in the Inspector Bar. (If you use the Inspector Bar and then press Enter, you will not be able to reposition the circle.)

The circle will be created from the point you selected on the line.



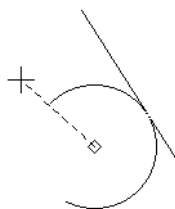
3. Move the mouse to position the circle, on either side of the line.



4. A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



5. Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.



### ***Arc: Start / Included / End***

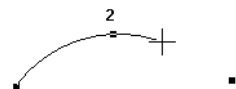


Creates an arc by defining the start point, a point on the arc, and the endpoint.

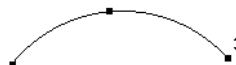
1. Select the start point.



2. Select a point through which the arc will pass.



3. Select the endpoint.



### ***Arc: Start / End / Included***

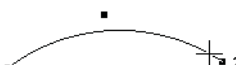


Creates an arc by defining the start point, the endpoint, and a point on the arc.

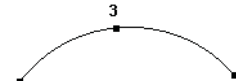
1. Select the start point.



2. Select the endpoint.



3. Select a point through which the arc will pass.



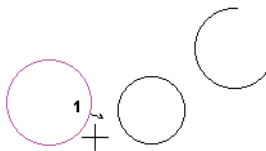
## Arc: Tangent to 3 Arcs



Creates an arc tangent to three arcs or circles. You can specify whether the existing arcs will be inside or outside the new circle.

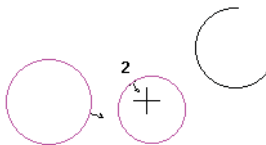
1. Select the first tangent object. Click slightly outside the object to keep it outside the new circle. Click inside if you want the tangent object to be inside the new circle.

If the object is clicked slightly outside, an outward-pointing arrow appears.

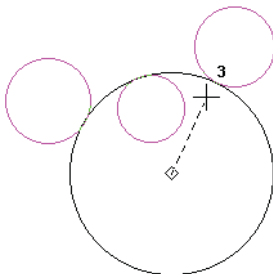


2. Select the second tangent object.

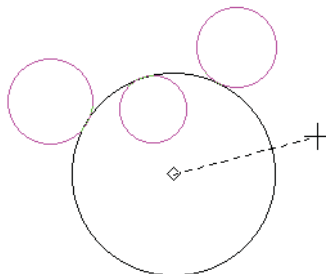
If the object is clicked slightly inside, an inward-pointing arrow appears.



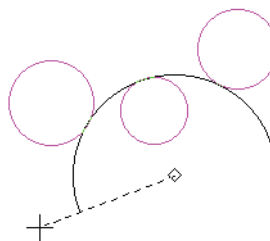
3. Select the third object. The tangent circle is created. The arc will be cut from this circle.



4. A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



5. Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.

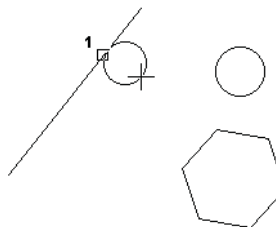


## Arc: Tangent to Entities



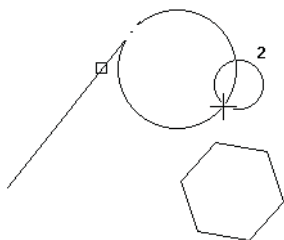
Creates an arc tangent to three objects (rectangles, polygons, arcs, etc.)

1. Select the first tangent object.

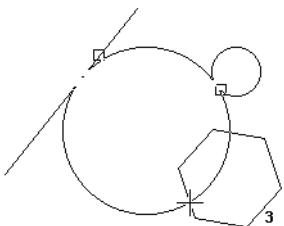




2. Select the second tangent object.

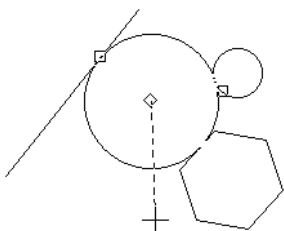


3. Select the third tangent object.

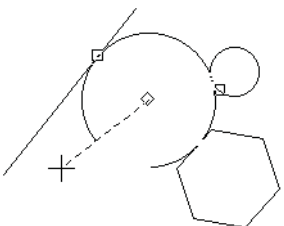


The tangent circle is created. The arc will be cut from this circle.

4. A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.

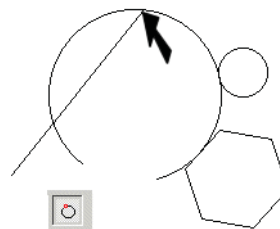


5. Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.



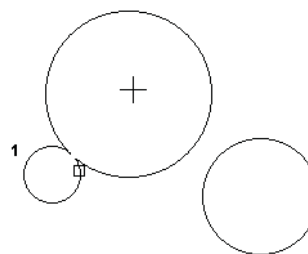
#### Local menu option:

**Through point:** Makes the tangent circle pass through a specific point. In the example below, the line endpoint was selected, rather than the line itself.

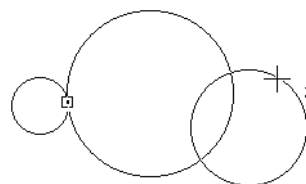


#### **Tangent Arc of Fixed Size**

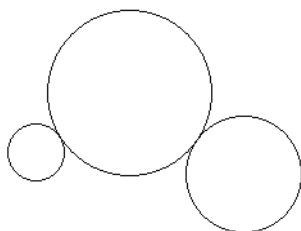
By default, **Tangent to Entities** creates a circle that touches three objects, and the arc is cut from this circle. To fix the size of the circle, enter the radius, diameter, or circumference in the Inspector Bar and lock it, but do not press Enter. When you move the cursor, the circle size remains constant.



Select the second tangent object, close to the point of tangency.



The fixed-sized tangent circle is created.

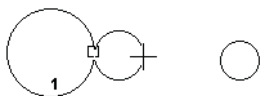


## Arc: Tangent to 2 Entities

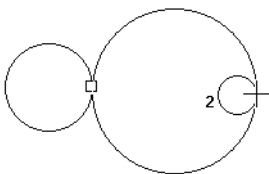


Creates an arc tangent to two objects - lines, arcs, circles, or ellipses. You can select the arc segment that will remain on either side of the tangent points.

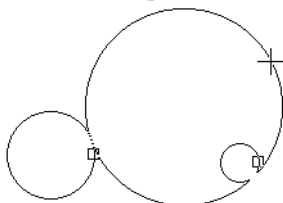
1. Select the first tangent object, close to the point of tangency.



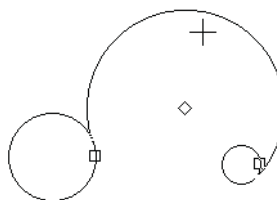
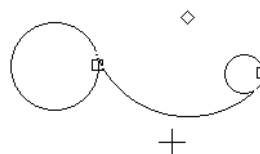
2. Select the second tangent object, close to the point of tangency.



3. Size the circle with the mouse; the arc will be cut from this circle. Or enter the radius, diameter, or circumference in the Inspector Bar.

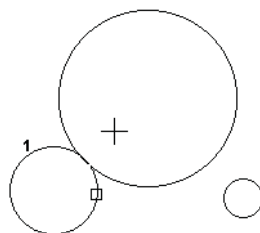


4. Move the cursor to either side of the arc to determine the arc segment that will remain.

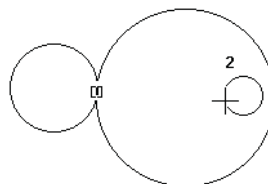


## Tangent Arc of Fixed Size

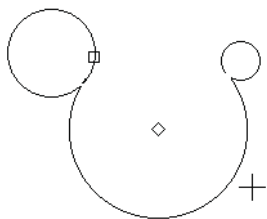
By default, **Tangent to 2 Entities** creates the smallest circle that touches two objects, and the arc is cut from this circle. To fix the size of the circle, enter the radius, diameter, or circumference in the Inspector Bar and lock it, but do not press Enter. When you move the cursor, the circle size remains constant.



Select the second tangent object.



The fixed-sized tangent circle, then arc, is created.

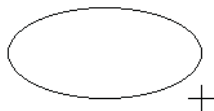


## Arc: Elliptical

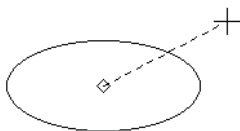


Creates an elliptical arc by defining a bounding rectangle for the ellipse. The axes of the ellipse will be orthogonal.

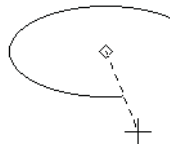
1. Select the first corner of the bounding rectangle.
2. Define the diagonally opposite corner of the bounding rectangle, or enter the major and minor axis lengths in the Inspector Bar.



3. A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



4. Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.

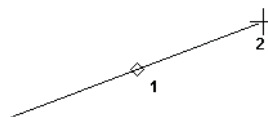


## Arc: Rotated Elliptical

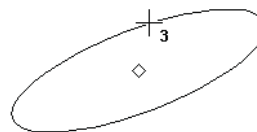


Creates a tilted elliptical arc.

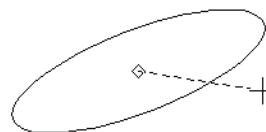
1. Select the centerpoint of the ellipse.
2. Move the mouse to set the length and angle of the major axis, or enter the major axis length and angle in the Inspector Bar.



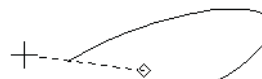
3. Set the length of the minor axis, which is always perpendicular to the major axis. You can also enter the axis length in the Inspector Bar.



4. A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



5. Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.



Arc: Elliptical Fixed Ratio

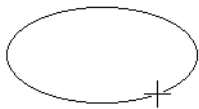


Creates an elliptical arc by specifying its aspect ratio - the ratio of the major axis to the minor axis. The axes of the ellipse are orthogonal.

- 1. Enter the aspect ratio in the **a:b** ratio field on the Inspector Bar. (If you want to repeat this value later then lock it, otherwise the field reverts back to the default.)
- 2. Select the centerpoint of the ellipse.
- 3. Move the mouse to size the ellipse.

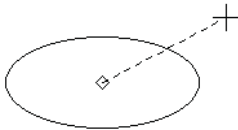


a:b = 0.5

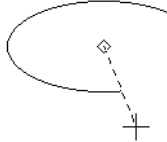


a:b = 2.0

- 4. A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



- 5. Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.



Curves

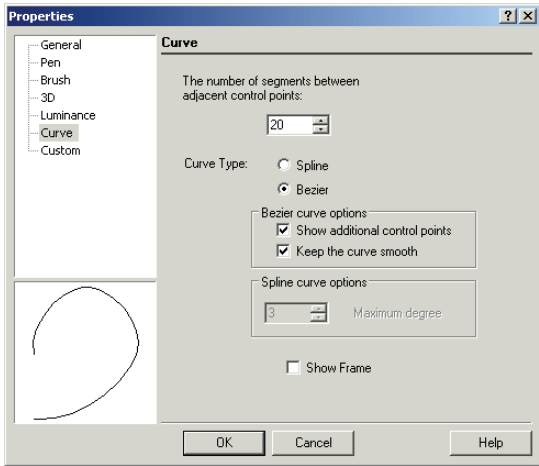
Tools for drawing splines, Bezier curves, sketches, and revision clouds.

You can display the **Curve** toolbar by right-clicking in any toolbar area and selecting **Curve**.



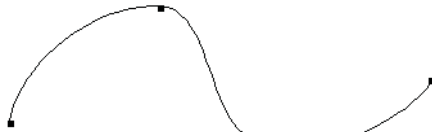
Curve Properties

Curve properties can be set on the **Curves** page of the **Properties** window.

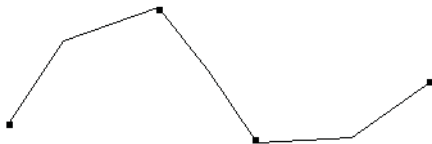


**Number of segments between adjacent control points:**

Curves are composed of many small line segments drawn between control points. A high number of segments will yield a smoother curve; a low number will make the curve appear more jointed.



High number of segments

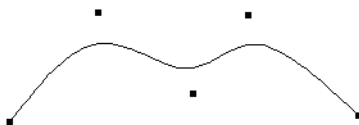


Low number of segments

**Curve Type:** Choose whether the curve is drawn as a **Bezier** curve or **Spline**. Bezier curves intersect the control points; spline curves gravitate toward the control points but do not intersect them (this option creates a **Spline by Control Points** rather than **Spline by Fit Points**).



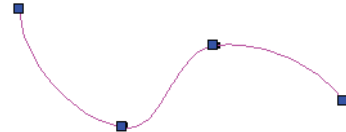
Bezier Curve



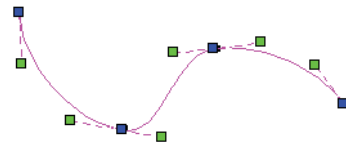
Spline Curve

**Bezier Curve Options:** If the curve is a Bezier curve, the following options are available:

- **Show additional control points:** Additional control points will be displayed when **Edit Node** mode is active (see "Edit Tool" on page 219 and "Editing Splines and Bezier Curves" on page 227). This also enables the **Keep the Curve Smooth** option.

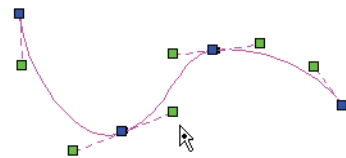


Regular control points in Edit Node mode

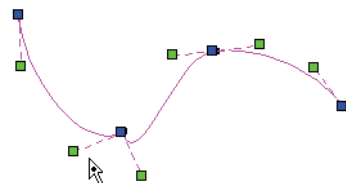


Additional control points in Edit Node mode

- **Keep the curve smooth:** If checked, the additional control points at each node remain linear, so that no sharp corners can be created at a node. If not checked, you can move each additional node independently.



Keep Curves Smooth - additional nodes remain linear

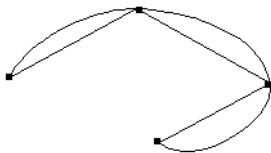


Keep Curves Smooth disabled

**WARNING:** *Modifying tools such as **Split** or **Trim** may alter a curve drastically, because the curve loses the influence of deleted control points.*

**Spline Curve Options:** The degree value (n) creates a spline of n-th order, which affects the spline smoothness.

**Show Frame:** If checked, a polyline frame will be displayed connecting the curve's control points. This provides visual feedback about how the curve is drawn.



## Splines and Bezier Curves

These three tools each use a series of points to create a curve.

### Spline by Control Points



### Spline by Fit Points



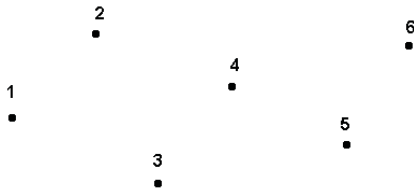
### Bezier



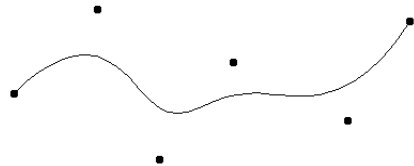
For **Spline by Control Points**, the points act as a guide for the curve; the spline does not actually pass through all of the points. For **Spline by Fit Points** and **Bezier**, the curve does pass through each point. These two tools produce similar results; the main difference is the algorithm behind them, and how they are edited.

**NOTE:** *To create splines in 3D, see "3D Spline by Control Points" on page 386 and "3D Spline by Fit Points" on page 387.*

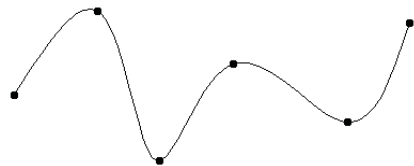
The three curves will be shown for this series of points:



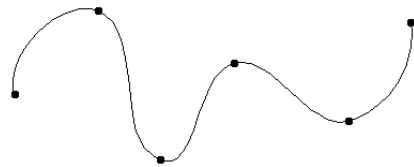
Select the control points in the desired order. You can also enter the length and angle between points in the Inspector Bar. After selecting the last point, select **Finish** from the local menu or press Alt+F. You can also double-click the last point.



Spline by Control Points

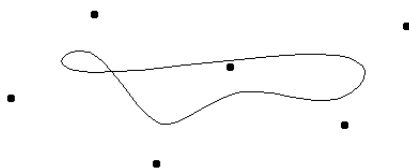


Spline by Fit Points

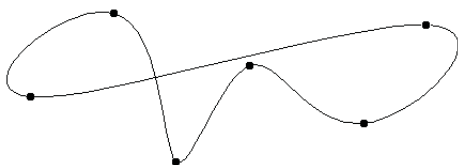


Bezier

If you want to close the curve, select **Close** from the local menu instead of **Finish**.



**Closed Spline by Control Points**



**Closed Spline by Fit Points**

Once created, you can use the **Edit Tool** to change the shape of a spline and add knots. See "Editing Splines and Bezier Curves" on page 227.

---

*TIP: You can change a spline into a Bezier curve, and vice-versa, by opening the **Properties** window and editing the **Curves** page. If you change a Bezier curve into a spline, it will be a **Spline by Control Points**. You can also convert 2D objects into a Bezier curve - see "Convert to Curve" on page 250.*

---

## Sketch

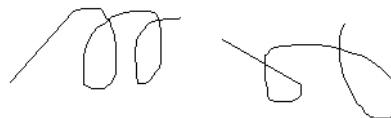


Creates a freehand drawing.

1. Press and hold the mouse button to draw a freehand curve. Release the mouse when finished.



2. As long as the tool is still active, you can continue drawing new curves.



## Revision Cloud



Creates uniform revision clouds. These are most commonly used in Paper Space, but the tool is available in Model Space as well.

1. Before selecting the start point, select **Number of segments** from the local menu. The default number is 1, and each segment consists of two arcs.
2. Select the start and endpoints of the first segment.



3. Continue selecting points for more segments.



4. When finished, select **Finish** (Alt+F) from the local menu, or select **Close** (Alt+C) to join the endpoints.



## Gear Contour

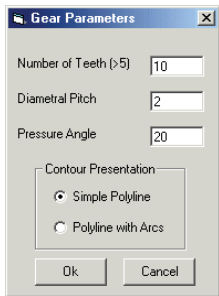
Available in TurboCAD Pro and Platinum only



NOTE: You can display the **Special Tools** toolbar by right-clicking in any toolbar area and selecting **Special Tools**.

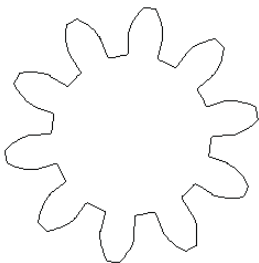
Creates the outline for a toothed gear.

When the tool is invoked, the **Gear Parameters** window appears:



Enter the number of teeth and other physical parameters. You can create the contour as one polyline, or as separate polylines.

Click OK to create the contour.



## Inserting Objects from Other Files

You can insert data from external files into your TurboCAD drawing. For example, you can insert a Word document or a \*.gif picture file, or even data from another TurboCAD (or other CAD format) file.

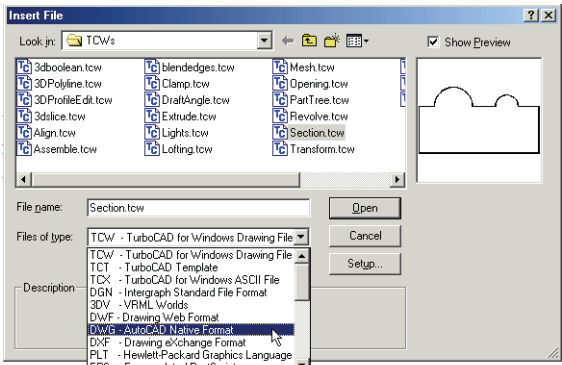
You can insert files, pictures, and OLE objects. These tools can be accessed from the **Insert** menu, or from the **Insert** toolbar, displayed by right-clicking in any toolbar area and selecting **Insert**.



## Inserting a File



Inserts a TurboCAD (or other CAD format) file into the current drawing.



Select the type of file you want to import from the **List Files of the Type** drop-down list. See "Importing and Exporting Files" on page 24.

The complete contents of the file will be inserted into your drawing, alongside any existing objects. If the inserted file contains blocks, the **Add Blocks** window will appear (see "Inserting Blocks from Another File" on page 293.)

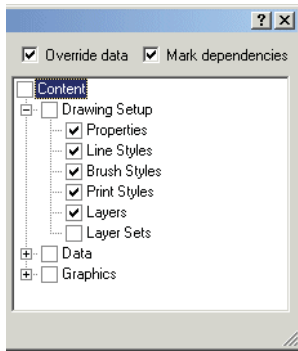
NOTE: The contents of the inserted file are embedded, not linked. See "Embedded and Linked OLE Objects" on page 177.

## Inserting Partial Data from a File

Using **Insert / File** will insert all data from the selected CAD file into your drawing. If you want to insert only selected components of a file (layers, UCS, blocks, etc.), use **File / Extract From**.



The right side of the **Open** window contains a box in which you can select what you want to add from the selected file.



For example, you can insert all objects ("graphics") but omit their properties or line style. Or you could insert a file's layers without the objects that they contain.

You can use **File / Extract To** to save only selected components in your drawing to a \*.tcw file. This is useful for creating file templates.

**Override data:** If any items with the same name are found in the target file, they will be replaced by the items in the source file.

**Mark dependencies:** Automatically selects items on which other items depend. For example, if you select **Layers**, then **Print Styles**, **Brush Styles**, **Line Styles**, and **Properties** will also be selected, because these are all items defined for a specific layer. Similarly, if you select **Print Styles**, then **Properties** will also be selected.

## Inserting a Picture

*Available in TurboCAD Pro and Platinum only*



You can insert a picture from a file, from a list of images, or as an OLE object. Inserting pictures is useful, for example, if you have a picture of a site plan and want to create buildings directly on it.

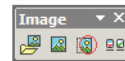
---

**NOTE:** *To create a tracing of an inserted picture, see "Tracing" on page 283.*

---

As with inserting any object, the picture will be inserted "on top" of existing objects. To adjust the object stack, you can use the **Bring to Front** and **Send to Back** tools. See "Stacking Objects" on page 248.

The **Image** toolbar can be opened by right-clicking on any toolbar area and selecting **Image**.




---

**NOTE:** *Nearly all graphic formats are raster images, meaning they are composed of dots. Even when reading vector formats, such as \*.wmf, the image is converted into a raster format for TurboCAD purposes.*

---

## Image Manager



Enables you to manipulate all the raster images in the current drawing.

You can insert files directly from this window using **From Image List**.



The toolbar at the top of the window can be used to change the image list as thumbnails, list, or detailed list.

**Image List:** Lists all images inserted into the drawing. The list can be sorted by name, status, or path. To edit a name, select it and then click it again. You can then enter the new name.

**New:** Adds a new image file to the list.

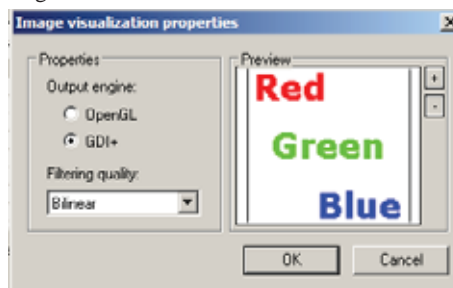
**Delete:** Deletes an image file from the list.

**Embed:** Saves the image file in the drawing as an embedded object, and not as a reference to an external file.

**Unload:** Unloads image data from work memory.

**Reload:** Reloads image into work memory.

**Properties:** Opens the **Image Visualization Properties** window, which is useful for setting the image quality for large images.



- **OpenGL:** Limits image output to the maximum OpenGL texture size (1024x1024 is supported by most hardware), and resizes image to a square view. For example, a 400 x 300 image will be resized to 512 x 512, thereby losing accuracy. As you increase the zoom, the image may become blurry.
- **GDI+:** Supports large images without resizing, maintaining high accuracy. Not all hardware can render this way efficiently, so this method may be slower than OpenGL.
- **Filtering quality:** Controls scaled or rotated images.

**Nearest point:** The pixel with coordinates nearest to the desired pixel value is used, resulting in a set of large squares when viewed closely.

**Bilinear:** A weighted average of a  $2 \times 2$  area of pixels surrounding the desired pixel is used. This is the most common filtering algorithm, adding smoothness in large scales.

**Trilinear:** A more exact extension of **Bilinear** and is practically indiscernible. But it is unsupported with OpenGL.

---

NOTE: These parameters are not file-specific; they affect visualization of all images in all TurboCAD files.

---

## Inserting a Picture from a File



Using this tool you can insert any type of graphic - \*.gif, \*.jpg, etc. The file is embedded and the picture itself cannot be edited. You can, however, select it and perform any of the **Select Edit** commands (see "Select Edit" on page 198).

All inserted files are listed in the **Image Manager**.

1. Change the workplane, if necessary. The picture will be inserted onto the current workplane (though it can be moved later).
2. Select **Insert / Picture / From File** and browse to select the desired picture.
3. Select two points to determine the size of the inserted picture. Use the local menu option **Keep Aspect Ratio** if you want to maintain the picture's proportional size.




---

*NOTE: The first point you select corresponds to the upper-left corner of the picture.*

---

4. You can move, rotate, or scale the picture using any of the **Select Edit** tools.

## Inserting a Picture from a List of Images



Invokes the **Image Manager**, from which you can select a picture to insert. See "Image Manager" on page 174.

## Inserting a Picture as an OLE Object

Inserts a picture as a linked OLE object. This means that the picture can be edited in its source application and all changes will be reflected in the TurboCAD drawing as well. File types that are allowed for OLE insertion are \*.wmf (Windows metafile) and \*.dib (device-independent bitmap).

1. Select **Insert / Picture / as Object** and browse to select the desired picture.

---

*TIP: You can also drag the file into the drawing from Windows explorer.*

---

2. The picture is inserted in its original size. You can resize, move, or rotate it by using the **Select Edit** tools.
3. To edit the picture, you can double-click on it to open it in its source application.

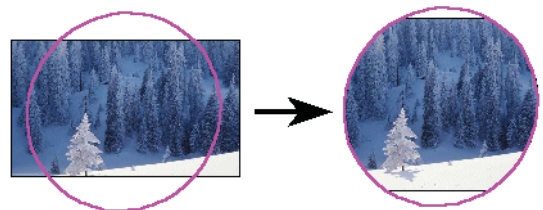
To select a previously inserted picture, click along an edge instead of inside the picture.

## Clipping Images

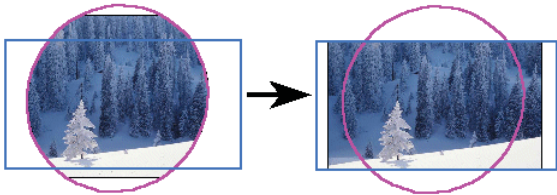


Enables you to use a border to clip a raster image.

1. Select the image you want to clip.
2. Select the new image outline. The image will be clipped to the outline.

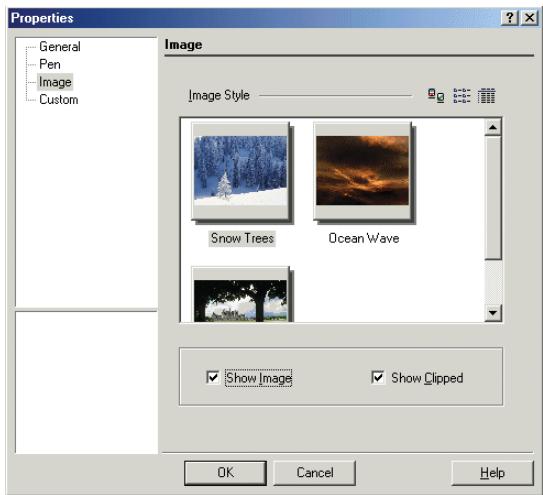


The image still retains its original size, even if it is not fully displayed. Therefore, you can clip it again using larger borders.



### Image Properties

You can view certain properties of all currently inserted picture by selecting an image and opening its **Properties** (see "Object Properties" on page 79) to the **Image** page.



The toolbar at the top of the window can be used to change the image list as thumbnails, list, or detailed list.

**Show Image:** Shows or hides the images.

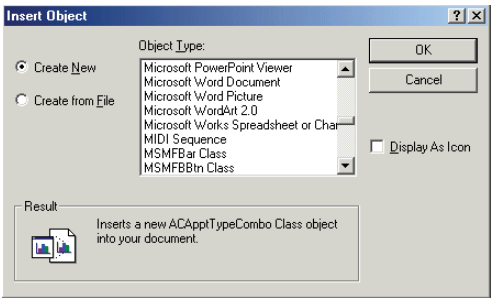
**Show Clipped:** Uncheck to hide the clipping contour and show the original image. Check the box to show a clipped image.

### Inserting an OLE Object



Enables you to insert OLE objects from other Windows applications. You can edit an OLE object from within TurboCAD using the features and commands of the Windows application in which the object was created (the source application). For example, you can insert a Microsoft Word application into your drawing and edit the text or graphics using Microsoft Word tools.

1. Select **Insert / Object**, and the **Insert Object** window appears.



**Create New:** Inserts an OLE object from another application. Select the application from the list of all applications found on your system. The object will appear in the drawing. The object is editable in its source application.

**Create from File:** Inserts an object from another application that has already been created in its source application. Enter the path and name of the file, or click **Browse** to find it. Click **Link** if you want it linked to its original file (see "Embedded and Linked OLE Objects" on page 177).

**Display as Icon:** Places an icon representing the object into the drawing.

---

*TIP: You can also drag the file into the drawing via Windows Explorer.*

---

## Embedded and Linked OLE Objects

OLE objects that you insert can be either linked or embedded.

- **Embedded** objects become an integral part of your file, and take up more space in the file than a linked object. Embedded objects will move with the file if you transfer it. If you make changes to the OLE object's source file, these changes will **not** affect the embedded object.
- **Linked** objects are references to the file on which they were based, and they retain their connection to that file. If you make changes to the OLE object's source file, the changes **will** affect the linked object.

If you plan to use multiple copies of an OLE object, using links will reduce your file size.

Linked objects are referenced by their path. If you move the TurboCAD file, you will need to move all linked objects as well. If you move the source file of a linked object to another folder, Windows will not be able to locate the file, and you will need to edit the link.

---

**WARNING:** *Norton AntiVirus Script Blocking can cause problems with OLE. For example, Microsoft Word documents are inserted as a picture, not a Word document.*

---

## Paste Special

Enables you to choose how to insert (paste) into your drawing an object that was previously cut (Ctrl+X) or copied (Ctrl+C) to the clipboard. The object on the clipboard can be from TurboCAD, or it can come from another application.

The options available in the **Paste Special** window depend on the type of object in the clipboard, and the object's source application. For example, if there is a Microsoft Word document on the clipboard, you can paste it as:

- **Word Document:** Inserts as an OLE object which you can edit within TurboCAD using the Microsoft Word application.
- **Picture:** Inserts as an image
- **Text:** Inserts as a TurboCAD text object.

When inserting an object as OLE, select **Paste Link** to link the object to its source file or **Paste** to embed the object (break the link with the source file). Check **Display As Icon** to insert the OLE object as an icon. You can then click **Change Icon** to select another icon.

---

**NOTE:** *Only two types of data can be inserted into TurboCAD as an editable TurboCAD drawing object: text (inserted as a TurboCAD text object), and TurboCAD drawing objects that have been copied to the clipboard from TurboCAD. All other clipboard objects will be pasted either as OLE objects or as pictures (either bitmaps or metafiles).*

---

## Editing Inserted OLE Objects

Edits an OLE object in its source application.

1. Select an OLE object in your drawing. This object may appear as a document or a bitmap, or it may appear as an icon, depending on the options you chose when you inserted the OLE object.
2. Double-click on the object to open it for editing.

## Editing Links to OLE Objects

You can set each linked object so that the update happens automatically, or so that the update must be done manually.

**Links box:** contains all the links in the active drawing. You can highlight multiple links by pressing Shift or Ctrl.

**Update Option:** Sets whether the selected links are updated automatically or manually.

**Update Now:** Updates the current link. This is how to update links that are set to update manually.

**Open Source:** Opens the file in its source application. This also enables you to compare the linked object to the source file.

**Change Source:** Enables you to choose a new source for the object. Once the object is linked to the new file, its contents will change to match the new file.

**Break Link:** Terminates the link between the object and its source file (embeds the object). It is still an OLE object, but changes to it no longer affect the source file and vice-versa. Breaking the link also affects the size (in bytes) of your

drawing, since the embedded object contains its own copy of the data in the object, while the linked object contains only a reference to the object.

## Hyperlinks

**Hotkey:** Ctrl+H

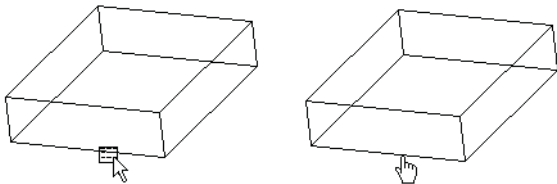


A hyperlink is attached to an object in your drawing, and enables you to jump to a specified file (for example, a drawing or text document) on your hard disk or company network, or to open a URL.

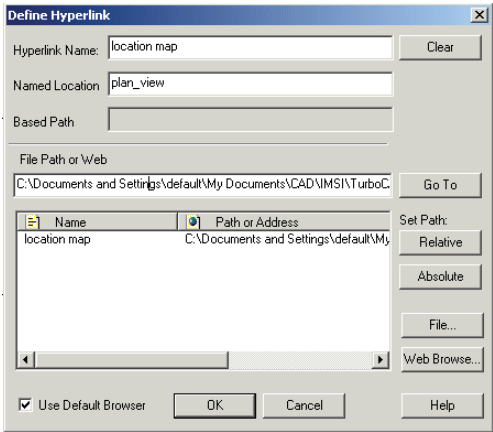
Another example of hyperlink usage is to display pictures of certain items, such as a bracket or fastener. You can create a hyperlink to a \*.jpg or \*.gif file that will appear in the Internet Palette when clicked with the **Pick Hyperlink** tool. You can also display text or spreadsheet files to provide descriptions of the clicked object.

*NOTE: You can also assign and edit a hyperlink via the **General** tab of an object's **Properties** (see "General Properties" on page 81).*

1. Place the cursor over the desired object. The cursor will be shaped like a screen. (If a hyperlink has already been defined for the object, the cursor will be shaped like a hand.).



2. The **Define Hyperlink** appears.



**Hyperlink Name:** Assigns a name (alias) to the hyperlink.

**Named Location:** Opens the hyperlinked file to a specific location, such as a named view. If you use a named view as the hyperlink, that view will be shown when the hyperlink is opened.

**Based Path:** Displays the default base path for all relative path hyperlinks in the current drawing.

**File Path or Web:** The location of a desired file on your disk or company network, or a URL. You can browse for either files or websites.

**Go To:** Invokes the target referenced by the hyperlink. If the target is a TurboCAD file, that file will open in another window. A \*.doc file will open in Microsoft Word, and a URL will open in the Internet Palette.

*NOTE: The Internet Palette opens automatically if **Auto Activate** is checked on the **Palettes** page of the **Customize** window.*

**Set Path: Relative** and **Absolute** are available if a file path or web address is provided. A relative path is relative to the path to the active drawing.

**Name and Path or Address:** Displays a list of hyperlinks used in the current drawing. To use any of these hyperlinks for the selected object, double-click the hyperlink's **Name** field.

**Use Default Browser:** Uses the default Internet browser installed on your computer.

#### Local menu options:

**Edit the Hyperlink:** Use this option on an object that already has a defined hyperlink. This opens the **Define Hyperlink** window, in which you can change any parameters of the link.



**Open the Hyperlink:** Brings up the defined hyperlink.



**Use Default Browser:** Uses the default web browser installed on your computer.



## Deleting Objects

The simplest way to delete objects is to select them and press the Delete key.

The **Clear** menu also contains options for clearing (deleting) objects:

**Clear Selection:** Deletes all selected objects.

**Clear All:** Deletes all objects in the active Model Space or Paper Space. Other spaces are not affected, but using **Clear All** in Model Space will leave viewpoints in Paper Space empty.

---

**WARNING:** **Clear All** deletes objects even on layers that are locked and not visible. To delete only visible, unlocked objects, use **Select All** and the Delete key.

---

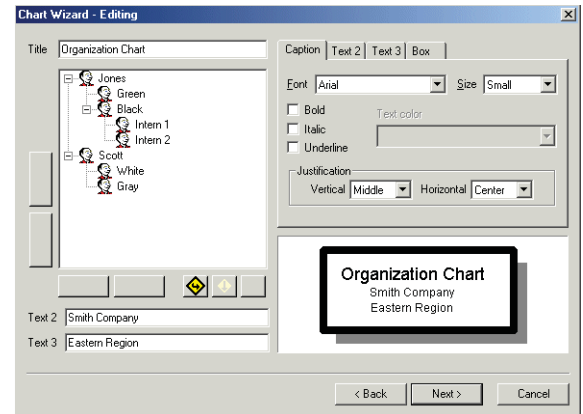
**Clear Constructions:** Deletes all constructions or a selected construction from the drawing. In Paper Space, only constructions added in Paper Space will be deleted. See "Construction Geometry" on page 121.

## Organizational Chart

*Available in TurboCAD Pro and Platinum only*

The Organizational Chart wizard is a separate module of TurboCAD that allows you to create organizational charts simply by entering text into an outline. The chart is created as a \*.csv (comma-separated value) file, which can be read as a spreadsheet.

1. In the first page of the wizard, choose whether to open an existing \*.csv file or to create a new one.
2. The second page of the wizard is where you enter or edit the chart data. The title and each item in the chart can have up to three lines of text (**Text 2**, and **Text 3** fields and tabs), and for each entry in the chart you can specify text alignment, box shape, etc.




---

**TIP:** The format of each item is taken by default from the format of the title (but can be changed). If you want the entire chart to have a particular format, change the settings while the **Title** field is still activated. Similarly, the format of a child item takes the format of its parent.

---

- Use these icons to add a new item directly below (**Create Sibling**) or in one level (**Create Child**) from the selected item. The “trash can” icon deletes a selected item.



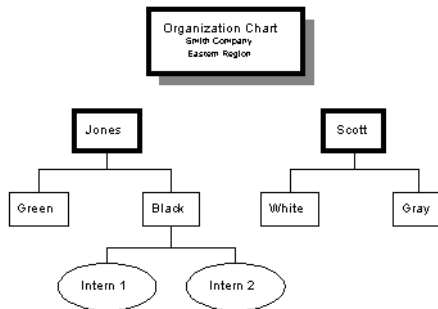
- Use these arrows to adjust the hierarchical levels of existing items. You can also drag an item over another item to create a parent-child relationship.



- Use these arrows to change the order of items.



3. In the last page of the wizard, set the chart orientation and separator line properties.
4. Finally, choose whether to insert the chart into the drawing, and whether to save it.




---

**WARNING:** *If you do not save your chart, you will not be able to edit it later in the wizard.*

---

The inserted chart is not one single object, rather it is composed of standard TurboCAD drawing objects. Each item is a group that can be exploded into its component parts. You can edit and redesign your chart using any of the geometric editing tools.



# 6 Selecting and Transforming Objects

This section covers selecting, moving, copying, rotating, and scaling objects, both 2D and 3D.

## Selecting Objects

To select objects, you must be in **Select** mode. You can access this by selecting **Edit / Select**, or by clicking the **Select** icon.



---

*TIP: You can also access **Select** mode by pressing the Space Bar.*

---

- To select a single object, click on it.
- To select multiple objects, keep the Shift key pressed while selecting.
- To deselect an object from a selected group, Shift-select it again.
- To select all objects, use **Select All** or use Ctrl+A.
- When a group of objects is selected, press Ctrl to select one of the objects in the group. You can edit just this one object, and when you are finished, click outside the group. The previously selected group is once again selected.

Four hotkeys are also available for selecting:

- **F6** selects the first object created. Pressing F6 repeatedly will select subsequent objects in the order they were created.
- **F7** selects the last object created. Pressing F7 repeatedly will select subsequent objects in reverse creation order.
- **Shift+F6** selects multiple objects. Starting with one object, subsequently created objects are added to the selection set.
- **Shift+F7** selects multiple objects. Starting with one object, previously created objects are added to the selection set.

For any selected object or group of objects, you can view and access some properties in the Selection Info Palette. See "Selection Info Palette" on page 191.

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*TIP: You can use the Design Director to select all objects that reside on a particular layer or sit on a particular workplane, select lights, etc. See "Design Director" on page 125.*

---

## 2D / 3D Selector

The way objects appear when selected, and the options for editing them, depend on whether they are selected as 2D or 3D objects. This is controlled by the **2D** or **3D Selector**.

Usage of the **2D Selector** can be set in the **Selector Properties** window. This can be opened by right-clicking anywhere while in **Select** mode, and selecting **Selector 2D Properties** from the local menu. (If the 3D Selector is active, the local menu item will be **Selector 3D Properties**.)

You can also click the icon on the Inspector Bar to open the **Selector Properties**.



While working, you can easily switch between **2D** and **3D Selector Mode**, without having to open the **Selector Properties** window. Simply click the **Toggle 2D/3D** icon on the Inspector Bar.



General

Controls how 2D and 3D objects are selected, and what transformation fields appear in the Inspector Bar.



**2D Mode for Model and Paper Space:** Treats all objects as if they are 2D, and only uses the 2D selection rectangle. The Inspector Bar contains transformation fields in X and Y only.

**3D Mode for Model Space:** Treats all objects as if they are 3D, and only uses the 3D selection box. The Inspector Bar contains transformation fields in X, Y, and Z.

**2D/3D Depending on Selected Object and Space Mode:** Treats objects differently; 2D objects are treated as 2D objects and 3D objects are treated as 3D objects.

**Edit Properties:** If this option is set double clicking on any object with the selector will open it Properties dialog.

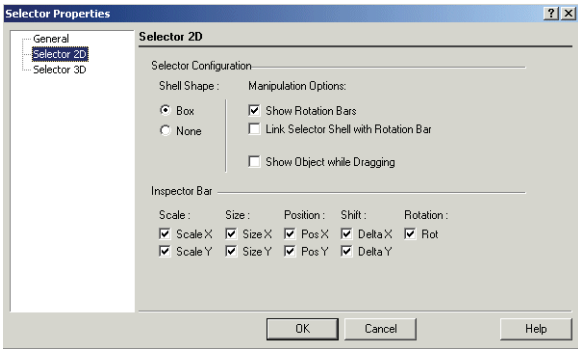
**Edit Content (Geometry):** If this option is set double clicking on any object with the selector will set the object for editing. For most objects the will be the equivalent of activating the edit tool. For text double clicking will allow you to edit the text content.

**Customize:** Allows you to specify how double clicking will act for each class of objects.



Selector 2D

Options for 2D selection and fields in the Inspector Bar.



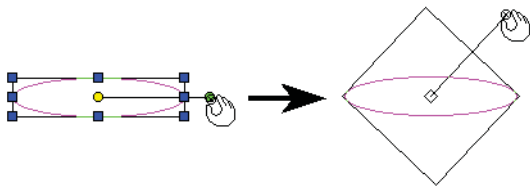
**Shell Shape:** Select **None** to make the selector shell (bounding rectangle) invisible.

Manipulation Options:

- **Show Rotation Bars:** If unchecked, the rotation bar is not shown and is not accessible.
- **Link Selector Shell with Rotation Bar:** If checked, when you press Ctrl and move the rotation handle, the selector shape will change accordingly.



Link Selector Shell not checked



Link Selector Shell checked

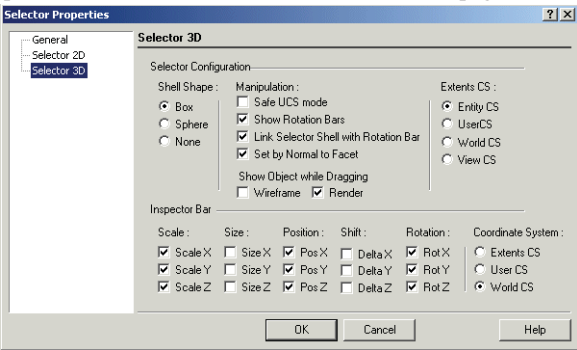
- **Show Object while Dragging:** Displays the selected objects dynamically as they are transformed. If not checked, only the selection shell is visible.

**Inspector Bar:** Controls the 2D transformation fields that appear on the Inspector Bar. See "Inspector Bar" on page 45.

*NOTE: The 2D Selector always moves the UCS (workplane) to the selection.*

### Selector 3D

Options for 3D selection, containing additional options for selection boxes and Inspector Bar fields. Many of these parameters are the same as for the **Selector 2D** page.



**Shell Shape:** **Box** is the default. The **Sphere** shell is useful for visualizing rotation, but does not allow drag and drop scaling. Select **None** to make the selector shell invisible.

**Manipulation:** Options for manipulating the display.

- **Safe UCS Mode:** Prevents changes to the current workplane when you manipulate objects with the 3D Selector. If unchecked, you will get a warning message when using 3D Selector for 2D object replacement.

*WARNING: Use this option with caution. Simply moving an object along the same workplane will not change the workplane, but rotating an object will change the workplane.*

**Extents CS:** Sets the coordinate system (workplane) used by the selector shell. Because these settings work in tandem with other settings, their performance can become complex. Careful study of the **Extents CS** and **Coordinate system** parameters will help you understand this powerful tool.

- **Entity CS:** Selector shell CS equals the CS of the selected objects.
- **User CS:** Selector shell CS equals the CS of the current workplane.
- **World CS:** Selector shell CS equals the world CS.
- **View CS:** Selector shell CS equals the CS of the plane of the current view.

**Inspector Bar:** Sets the fields that appear in the Inspector Bar (see "Inspector Bar" on page 45.)

- **Scale and Size:** X, Y, and Z fields.
- **Position:** Current location.
- **Shift:** Change in position.
- **Rotation:** Rotation from position
- **Coordinate System:** Sets the CS references in the **Position in Space** fields.

When using the 3D Selector, the local menu and Inspector Bar provide three options that do not appear when working in 2D.

- **Set UCS by Selector:** Moves the UCS origin to the selection reference point.



- **Set Selector by UCS:** Moves the selection to the UCS origin (similar to the **Format / Place on WorkPlane** option for 2D objects).



*NOTE: The 2D Selector always moves the UCS (Workplane) to the selection.*

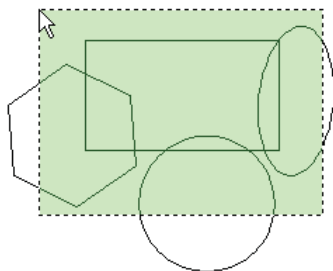
- **Lock/Unlock Axis:** Locks or unlocks a rotation bar. Move the cursor to the end of the rotation bar you wish to lock or unlock, and select **Lock** or **Unlock Axis** from the local menu.



- **Auto select compound profile:** When selecting curves for a geometry tool, this option will enable you to select smoothly connected chains of curves.

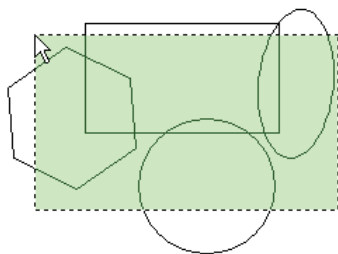
### Using a Selection Window

You can select a group of objects by dragging a rectangle around them. Click the first corner of the rectangle, keeping the mouse button pressed, and drag the rectangle to the opposite corner.



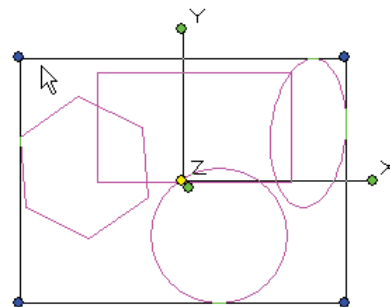
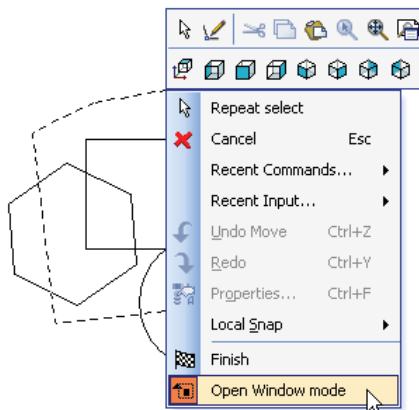
If you need to select objects using a shape other than a rectangle, see "Select by Fence" on page 187.

If you draw the selection rectangle from right to left, all objects completely or partially inside the fence will be selected.

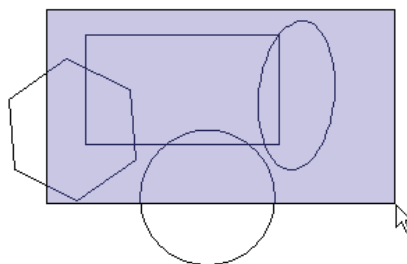


Alternatively, same can be performed by clicking **Edit / Select By / Fence**.

Draw any shape for selecting the objects. Turn the **Open Window Mode 'On'** from local menu if it is 'Off'. Then click on 'Finish'.

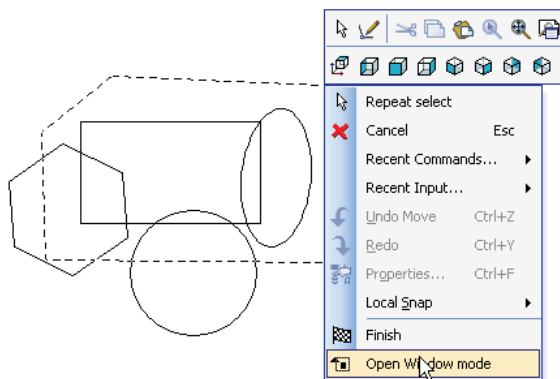


If you draw the selection rectangle from left to right, only objects completely inside the fence will be selected.

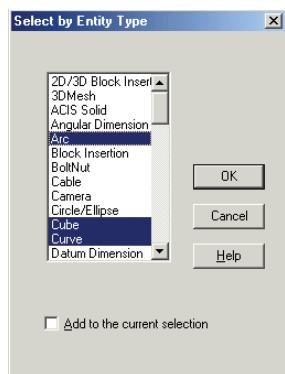
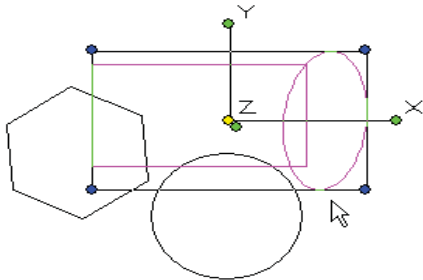


Alternatively, same can be performed by using **Fence**.

Draw any shape for selecting the objects. Turn the **Open Window Mode 'Off'** from local menu if it is 'On'. Then click on 'Finish'



## Select by Entity Type



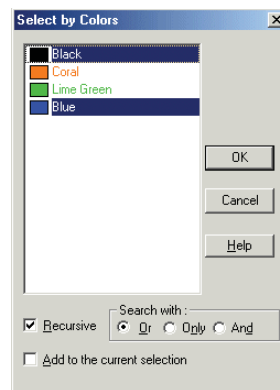
Selects objects of specified types.

Press Shift to select a range of types from the list. Press Ctrl select or deselect individual types.

**Add to current selection:** All objects selected by the match will be added to any current selection in the drawing.

## Select by Color

Hotkey: Ctrl+K



Selects objects of specified colors.

By default, the color list contains all colors used by visible objects in the drawing. The default list does not include colors used by objects that are contained within groups and blocks. See "Color Palette" on page 65.

Press Shift to select a range of colors from the list. Press Ctrl select or deselect individual colors.

**Recursive:** Colors of objects contained within groups and blocks are added to the color list. Because the color list needs to be rebuilt, select this option before selecting colors.

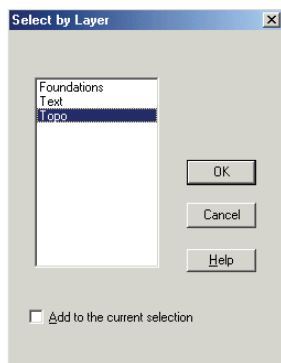
Search with:

- **Or:** Objects that contain at least one of the highlighted colors will be selected.
- **Only:** Objects that contain all of the highlighted colors, and only those colors, will be selected.
- **And:** Objects that contain all of the highlighted colors will be selected.

**Add to current selection:** All objects selected by the color match will be added to any current selection in the drawing.

## Select by Layer

**Hotkey:** Ctrl+L



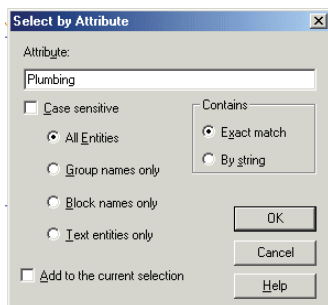
Selects objects that lie on specific layers (see "Layers" on page 116).

Press Shift to select a range of layers from the list. Press Ctrl select or deselect individual layers.

**Add to current selection:** All objects selected by the layer match will be added to any current selection in the drawing.

## Select by Attribute

**Hotkey:** Ctrl+I



Selects objects that share the same text in their **Attribute** fields. This field is on the **General** page of their **Properties** window.

Type text in the **Attribute** field that exactly matches the contents of the **Attribute** field of a set of objects, then click OK to select the objects.

The attribute text can contain the wildcard symbol - \*. For example, the attribute "Wall\*" will select "Wall1," "Wall+Window" and WallBrick." All characters that follow the \* symbol will be ignored.

**Case sensitive:** The search will distinguish between uppercase and lowercase letters.

**All Entities:** Selects all objects that share the specified text attribute.

**Group names only:** Selects only groups that share the specified text attribute.

**Block names only:** Selects only blocks that share the specified block name - the name that is assigned to a block while creating. The name of an inserted block is listed in the **Refers to** field of the **Properties** window, **Block Insertion** page.

**Text entities only:** Select only text objects that share the specified text attribute.

---

NOTE: *The attribute of a text object is the text string itself.*

---

**Exact match:** Selects all attributes that exactly contain the text in the **Attribute** field.

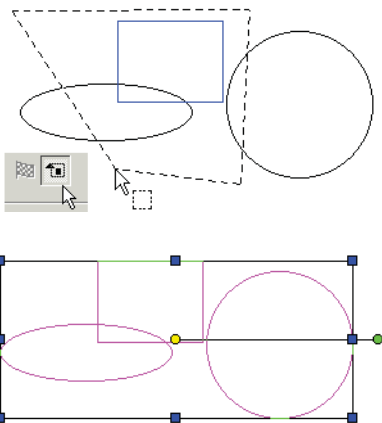
**By string:** Selects all attributes that contain the string in the **Attribute** field.

**Add to current selection:** All objects selected by the attribute match will be added to any current selection in the drawing.

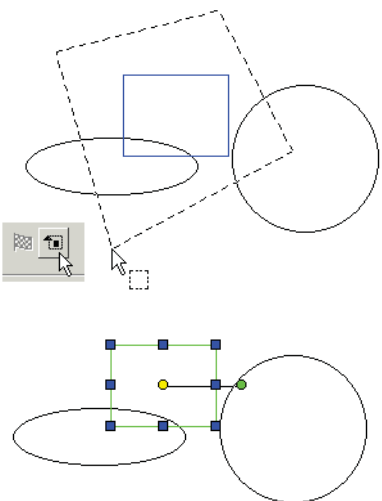
## Select by Fence

Selects a set of objects by drawing a closed polygonal “fence” around them. See “Irregular Polygon” on page 141 for details on how to create the fence.

If **Open Window Mode** is activated, all objects completely or partially inside the fence will be selected.



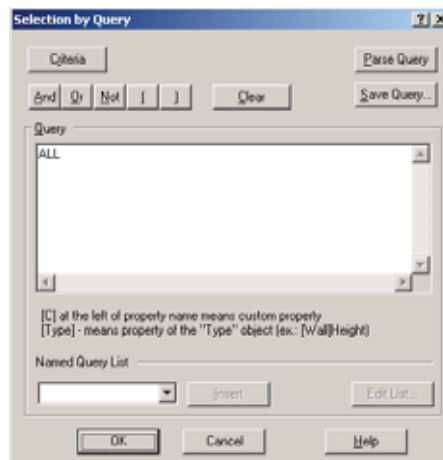
If **Open Window Mode** is not activated, only objects completely inside the fence will be selected.



## Select by Query

Available in TurboCAD Pro and Platinum only

**Hotkey:** Ctrl+Q



Selects objects meeting a complex set of criteria. For example, you can select all blue arcs or yellow dashed curves on Layers 3 and 5. Query selection can be useful in large or complicated drawings in which you need to filter through numerous colors, layers, entity types, etc.

You can write a query manually in the edit box, if you know the syntax rules. But it is recommended to use the tools in the **Selection by Query** window.

---

**NOTE:** **Query** will not select individual objects that are members of groups or blocks.

---

**Query window:** Contains the expression used to filter the objects for selection. You can assemble the expressions via the **Criteria** button, or you can enter them manually.

---

**NOTE:** The query text does not automatically wrap. Press **Enter** to separate lines. Also, new criteria are inserted at the end of the expression, regardless of cursor position. You can cut and past text to correct the order.

---

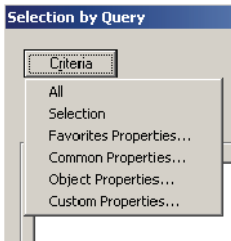
**And:** Selects objects meeting all criteria.

- Or:** Selects objects meeting any criteria.
- Not:** Selects objects that do not meet the specified criteria.
- Parenthesis:** Used for grouping elements of the expression.
- Clear:** Erases the current expression.
- Parse Query:** Tests the syntax of the query expression. If there is an error in logic or format, an error message will appear. You will also be notified if the query is sound.

NOTE: ***Parse Query** only tests the logic of the query expression. It does not test whether objects will be selected, nor does it implement the query.*

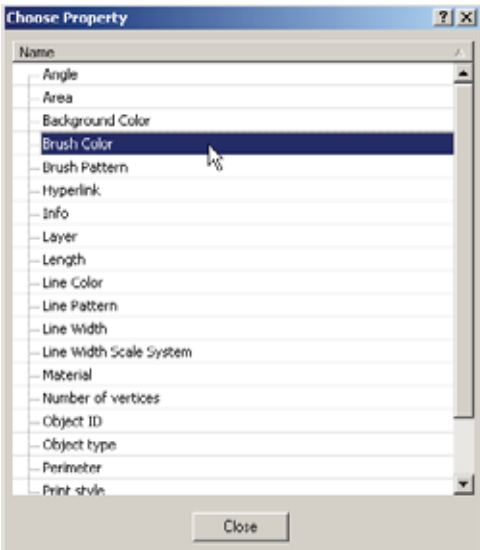
Query Criteria

The main elements of a query are names of object properties, values of properties, and logical operations. Click the **Criteria** button to start your query.



- All:** Runs the query on all objects in the file.
- Selection:** Runs the query only on selected objects.

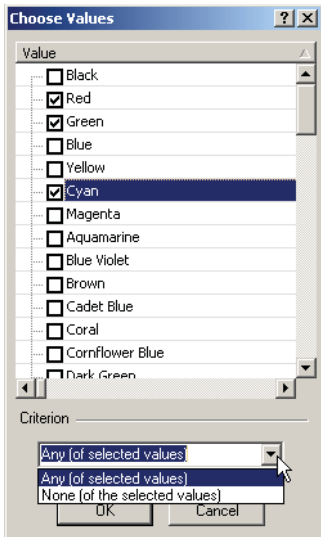
**Common Properties:** Enables you to select from a list of commonly-used properties. For example, you can select objects that have certain brush colors, line widths, layers, etc.



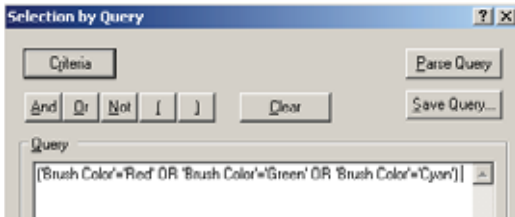
For example, If you click **Brush Color**, the **Choose Values** window will have the list of all brush colors. Check all the colors you want to include in the query.



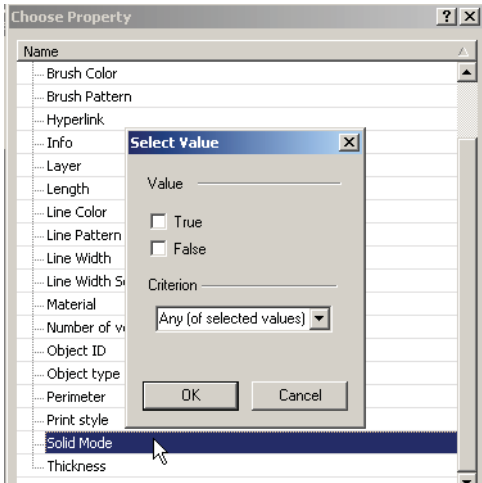
At the bottom of the window, you can choose **Any** to select all objects with these colors. If you choose **None**, objects that have other colors will be selected.



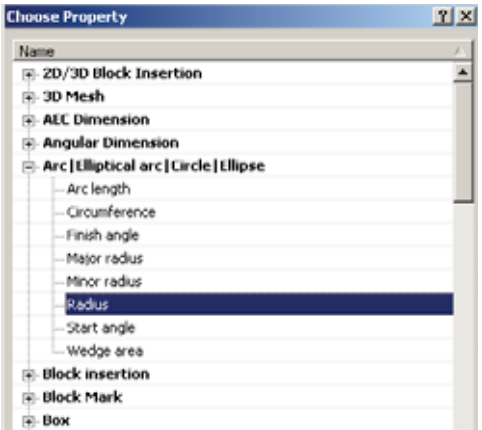
This is how the brush color query looks in the **Selection by Query** window.



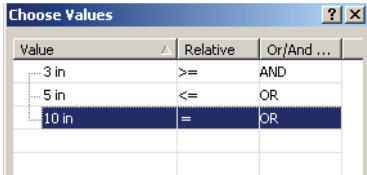
NOTE: Some properties do not have specific values or a list of choices. For example, **Solid Mode** is a **True / False** choice.



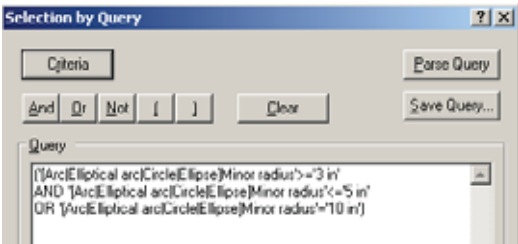
**Object Properties:** Enables you to select objects that have certain physical properties such as length or radius. For example, arcs and circles can be selected by arc length, circumference, radius, etc. Click **Radius**.



Enter a **Value** and its **Relative** (equal, less than, greater or equal, etc.). In this example, the query will select arcs and circles with radii between 3 and 5 inches, or equal to 10 inches.



This is how the radius value query looks in the **Selection by Query** window.



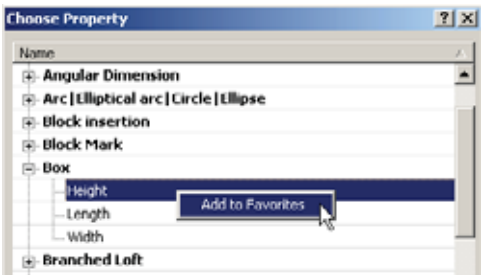
**Custom Properties:** See "Custom Properties, Database, and Reports" on page 624.

**Favorite Properties:** Enables you to choose from a list of queries you use often. You can add common properties and object properties to the list of favorites.

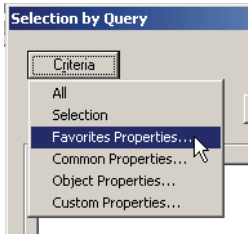
For example, open the **Common Properties**, and right-click on **Brush Color**. Select **Add to Favorites**.



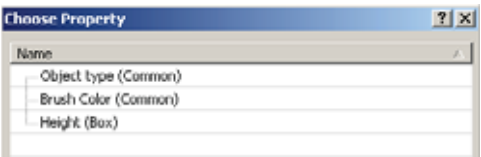
Then open the **Common Properties**, and right-click on **Height** under **Box**. Select **Add to Favorites**.



To see where these favorites are listed, open **Favorite Properties**.



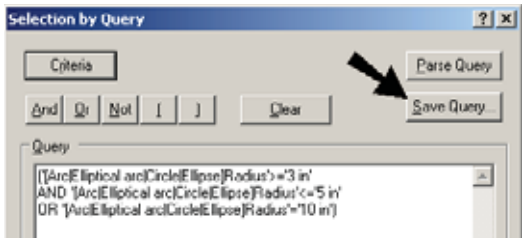
The brush color and height properties are listed here.



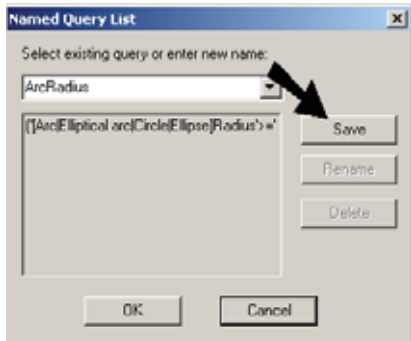
To remove a property from the favorites list, right-click on it and select **Remove**.

### Saving and Loading a Query

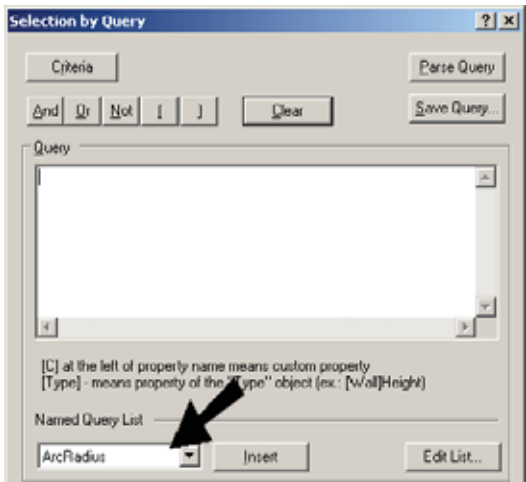
If you want to use a query later, you can save it. Click **Save Query**.



Enter a name for the query and click **Save**.



Then in the **Selection by Query** window, find the query name in the **Named Query List** and click **Insert**.



To change saved queries, click **Edit List**.

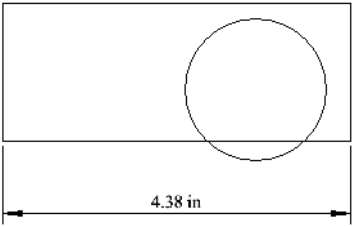
### Selection Info Palette

Displays information about the currently selected object or objects, such as **Properties** (entity type, measurements, location in the drawing), or **Constraints**.

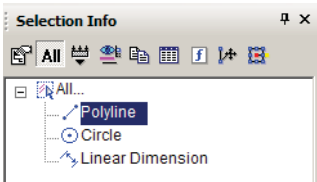
*NOTE: For details on using the Selection Info palette for 3D objects, and to see part history, see "Editing 3D Objects using Selection Info" on page 487.*

### Selection Info: Properties

This example consists of a rectangle, a circle, and a linear dimension.

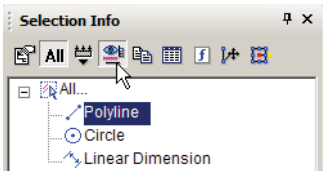


1. Open the Selection Info palette to see the three objects listed.

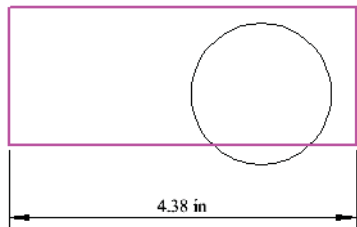


*TIP: The number of selected objects must be less than or equal to **Maximum Selection Info Entities**, which is set in the **Program Defaults** page of the TC Explorer Palette.*

2. In the Selection Info toolbar, click **Highlighting**, and make sure Polyline is selected.



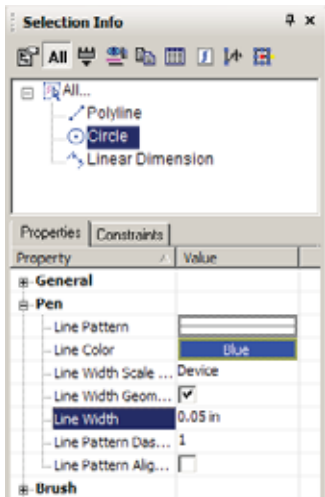
This highlights the rectangle in the drawing window.



Conversely, if you select objects in the drawing area, they will be highlighted in the Selection Info palette.

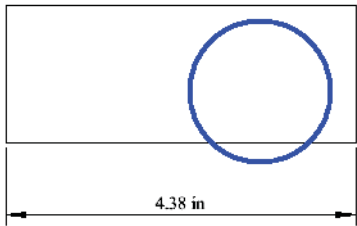
NOTE: If a group is selected, its objects will be listed under the group name in a similar tree format.

- 3. You can use this palette to change an object’s properties. Highlight the circle, and the lower section of the palette contains the categories of properties for the circle. In this example, the color and line weight were changed in the **Pen** category.

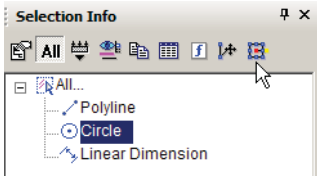


NOTE: For 3D ACIS objects, physical properties are not automatically listed. Click the **Physical Metrics** icon in the palette toolbar if you want to see engineering properties (volume, moments of inertia, etc.).

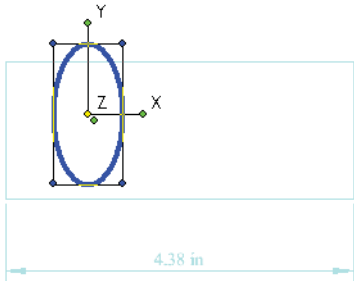
The new properties appear in the drawing area. (You may have to turn off the **Highlight** icon in the toolbar to see the changes.)



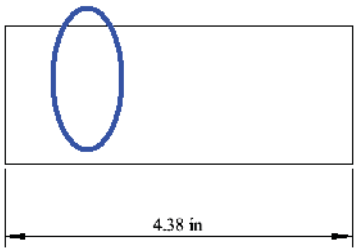
- 4. To make some changes to the circle, click **Select**.



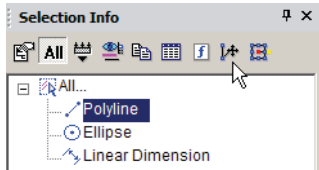
This invokes **Select Edit** mode, and all objects other than the circle are faded. Perform a change like moving and scaling the circle. (For details on editing this way, see "Select Edit" on page 198.) Note that this changes “Circle” to “Ellipse” in the palette.



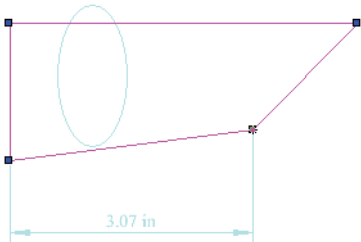
5. Select **Cancel** to exit **Select Edit** mode.



6. You can also invoke the Edit Tool. Highlight the Polyline and click **Edit Tool**. This enables you to reshape an object by moving its nodes. See "Edit Tool" on page 219.

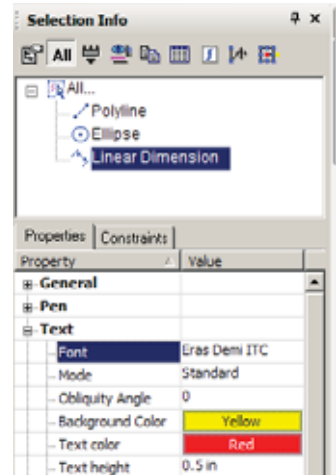


7. Move one or more nodes to change the shape.

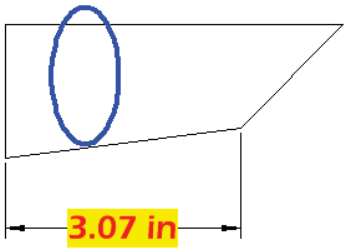


8. To exit the **Edit Tool**, select **Cancel** twice.

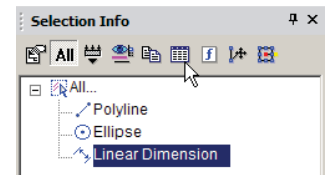
9. To change the dimension, highlight and make changes in the **Text** category.



The changed dimension appears in the drawing area.



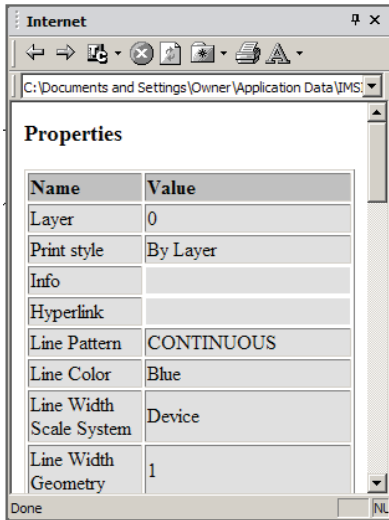
**NOTE:** You can also edit an object's properties by clicking the **Properties** icon in the palette toolbar. This opens the **Properties** window for the object (see "Object Properties" on page 79.)



10. The **Copy** icon is used to view an object’s properties in HTML format.



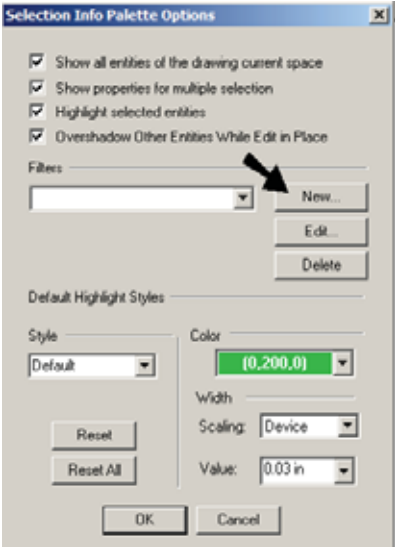
You can view these properties in the Internet Palette. The HTML file is stored in the ...\\Programs\\XML folder.



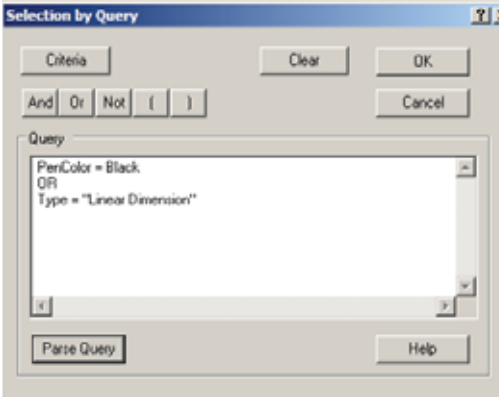
11. To set options for the Selection Info palette, click **Options**.



12. The options at the top control highlighting and fading for objects that are not being edited. To set a filter for what objects are displayed in the palette, click **New**.

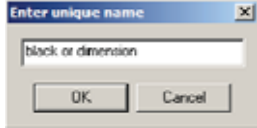


This options the **Selection by Query** window, in which you can specify types of objects you want to display. In this example, all objects that are either black or dimensions will be displayed in the palette.

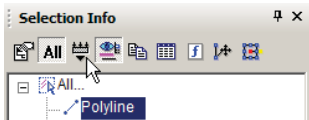


NOTE: For details on selecting by query, see "Select by Query" on page 187.

13. Each filter must be assigned a name.



14. To apply this filter, click **Filter** in the toolbar.

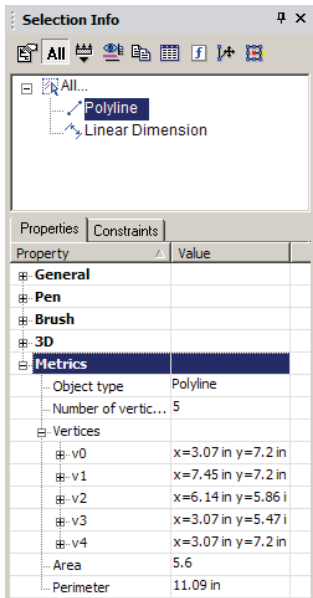


15. Select the filter you just created.



Now only the polyline and dimension appear in the palette.

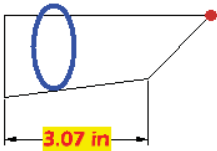
16. You can change geometry by modifying geometry in the **Metrics** category. Open **Metrics** for the Polyline - the coordinates for each vertex are listed.



17. Open the branch for a vertex to see its X and Y coordinates. These values can be changed.

# 3D	
Metrics	
Object type	Polyline
Number of vertic...	5
Vertices	
v0	x=3.07 in y=7.2 in
v1	x=7.45 in y=7.2 in
x	7.45 in
y	7.2 in
v2	x=6.14 in y=5.86 i
v3	x=3.07 in y=5.47 i
v4	x=3.07 in y=7.2 in

The vertex highlighted in the palette is also highlighted in the drawing area.



Some values cannot be edited, such as Area and Perimeter. These values depend on the coordinates of the vertices, and are grayed out.

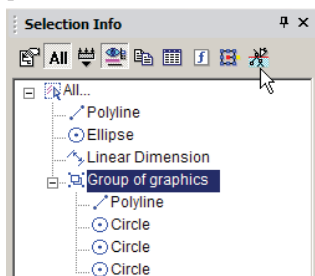
# Metrics	
Object type	Polyline
Number of vertic...	5
Vertices	
v0	x=3.07 in y=7.2 in
v1	x=7.45 in y=7.2 in
v2	x=6.14 in y=5.86 i
v3	x=3.07 in y=5.47 i
v4	x=3.07 in y=7.2 in
Area	5.6
Perimeter	11.09 in

18. To see options for another type of object, create several objects and group them (select them, then select **Format / Create Group**).

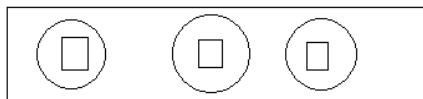


NOTE: For details on groups, see "Groups" on page 289.

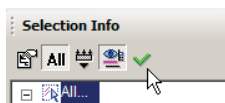
19. This object is a “Group of Graphics,” and you can open it to see each object that comprises the group. To edit the group, click **Edit Content**.



20. The group's contents fill the screen. Make some change, such as adding or removing objects from it.



21. When finished, click **Finish Edit Content**.



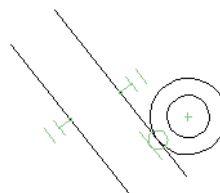
## Selection Info: Constraints

*Available in TurboCAD Pro and Platinum only*

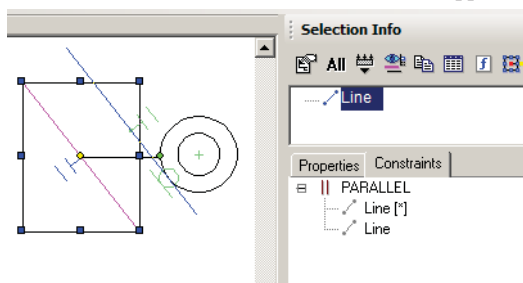
The **Constraints** tab of the Selection Info palette shows what constraints have been applied, and to which objects they are applied.

For details on constraints, see "Constraining Geometry" on page 251.

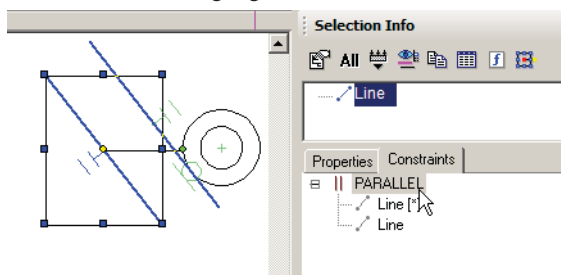
This example consists of two lines and two circles. Three constraints were applied: the two lines are constrained to be parallel, the circles are constrained to be concentric, and a tangent constraint was applied to the outer circle and adjacent line.



1. Make sure the **All** icon is disabled at the top of the Selection Info palette. Select the outer line, which only has one constraint (parallel). In the **Constraints** tab, the PARALLEL constraint is listed, below which are listed the two lines to which the constraint applies.

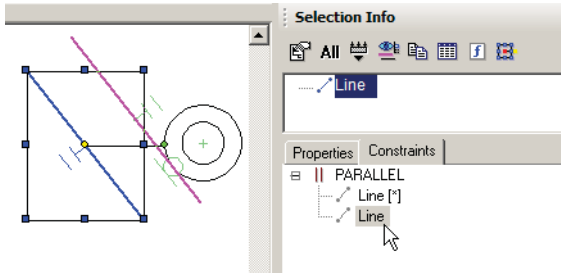


2. Move the cursor over the PARALLEL constraint in the **Constraints** tab, and the two lines that have this constraint are highlighted.

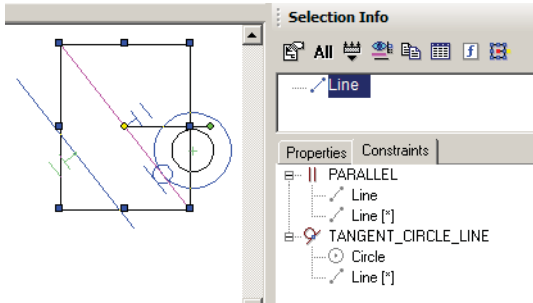




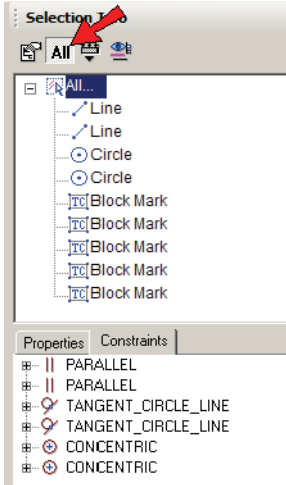
3. Move the cursor over either line below PARALLEL. The relevant line is highlighted.



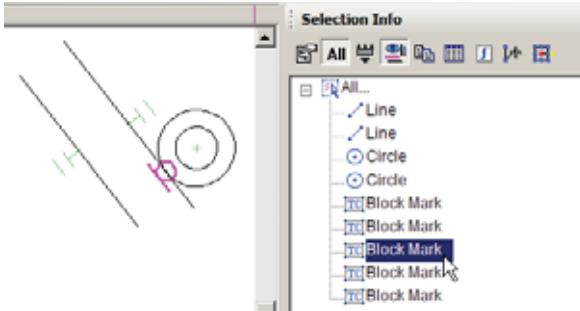
4. Now select the inner line, which has two constraints applied to it: PARALLEL and TANGENT. These two constraints are listed:



5. Now click **All**. The four objects (two lines, two circles) are listed, along with several blockmarks. These are the constraint markers themselves, which can be selected and deleted if you want to remove the constraint. In the **Constraints** tab, two items are listed for each constraint - one for each of the constrained objects.



6. Select any blockmark in the palette to see it highlighted onscreen (**Highlighting** must be enabled).



## Select Edit

(Editing by 2D Selector, Editing in **Select** mode)

This section covers ways you can edit objects in the current selection set.

### Geometric and Cosmetic Select Modes

There are two distinct modes for geometric selection: **Geometric** and **Cosmetic**. You can switch between the two modes by clicking **Geometric Select Edit**, located in the **Preference** property tab of the **Program Setup (Options / Preference)**.

**Geometric** extents are based on the distances between object vertices (the three corners points of a triangle or the two endpoints of a line). **Cosmetic** extents are based on distances between the outermost points on the object that will be drawn with the minimum (zero) pen width. Therefore, the difference between the two modes is relevant for objects such as double lines, multi lines, and objects that have a nonzero pen width. The select mode will come into play when using snaps and assigning dimensions.

For most applications, and for greater accuracy, use **Geometric** select mode. There are two main advantages to using **Geometric** select mode:

- When you scale using geometric extents, it is easy to see exactly what is going to be scaled. For example, if you scale a double-line object, the double-line separation distance and pen width are not scaled. **Geometric** select mode makes this visually obvious.
- When you scale an orthogonal linear object using geometric extents, you get accurate results. For example, if you scale a 1-inch horizontal line segment 200 percent along the X axis, the line segment will always be 2 inches long. If you were to use cosmetic extents, the results would vary depending on the pen width.

---

*TIP: If you want to change the width (separation distance) or pen width of a double line, you can enter new values in the **Properties** window.*

---

The disadvantage of geometric selection is that if you snap the endpoint of a wide line to another object, the cosmetic end of the line will overlap the object. You can work around this problem by selecting a line using cosmetic extents and using a snap mode to position the cosmetic end of the line.

---

**WARNING:** ***Geometric** select mode is **not** the installation default. If you choose to use **Cosmetic** select mode, zoom in closely and be careful where you snap. If you set **Geometric** select mode, it will remain active the next time you open TurboCAD.*

---

### Select Edit in 3D

(Editing by 3D Selector)

The **Select Edit** tools (see "Select Edit" on page 198) work the same way in 3D as for 2D.

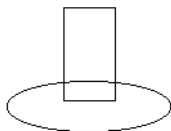
As with **Select Edit** for 2D objects, you can move, rotate, scale, and copy objects when they are selected. The only difference is that with 3D space there are additional options for movement. Whereas in 2D, you can only rotate in the XY plane, 3D objects can be rotated in the XZ and YZ planes as well. Movement can be in the Z direction, in addition to X and Y.

You need to activate the **3D Selector** in order to have all editing tools available. See "2D / 3D Selector" on page 181.

## Components of Select Edit Mode

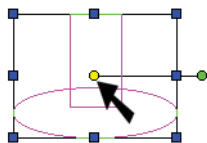
(Reference point, rotation handle, drag handles)

For this section, consider the following example of two 2D objects.

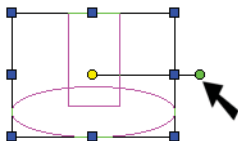


When they are both selected in **Select** mode, the selection set appears in magenta, with several small circles and squares.

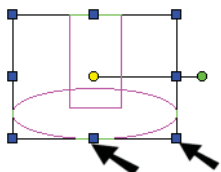
The yellow circle is the **reference point**. By default, it is located at the center of extents of all selected objects, but it can be moved. It is used for moving, and as a reference for rotating and scaling.



The green circle is the **rotation handle**. You can click and drag this point to rotate the objects around the reference point.



The blue squares are **drag handles**. You can click and drag these handles to scale the objects. Click an interior handle to resize one dimension of the rectangle. Click on a corner handle to resize both dimensions.



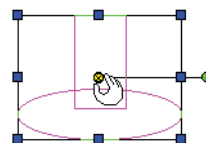
## Changing the Reference Point

To move the reference point, you must first select (or pick) it. There are several ways to do this:

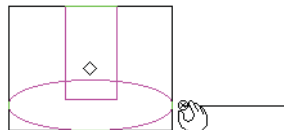
- Press D
- Press Ctrl and click the reference point
- Select **Edit Reference Point** from the local menu
- Click the icon in the Inspector Bar



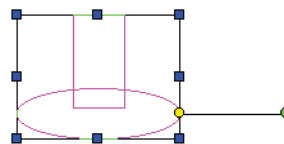
The cursor changes to a hand symbol.



Move the reference point to its new location, in this case the quadrant point of the ellipse.



Now any moving, scaling, or rotating will be done relative to this new point.



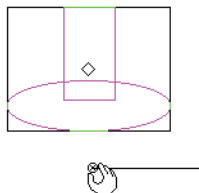
To return the rotation bar to its default position, select **Default Reference Point** from the local menu, or click the icon on the Inspector Bar.



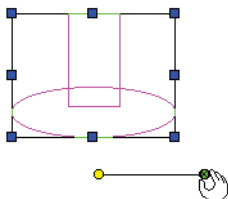
## Changing the Rotation Bar

For both ends of the rotation bar, you can select the points by pressing Ctrl and clicking. (The yellow point is the reference point and has additional ways to be selected.)

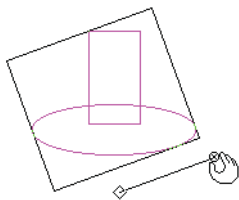
The cursor changes to a hand symbol. In this example, the reference point is moved first.



The green rotation handle is selected and moved the same way.



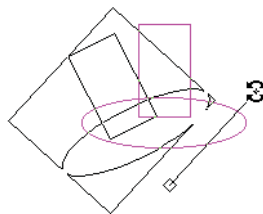
When you move the rotation bar, whether the selection rectangle changes depends on the **Link Selector Shell** option in the **Selector 2D** page of the **Selector** properties. See "Selector 2D" on page 182.



To return the rotation bar to its default position, select **Default Rotation Bars** from the local menu, or click the icon on the Inspector Bar.



Now any rotating will be done relative to the new rotation handle and reference point.

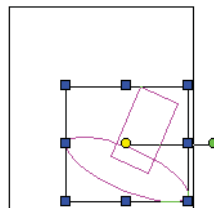


## Moving Objects in Select Edit

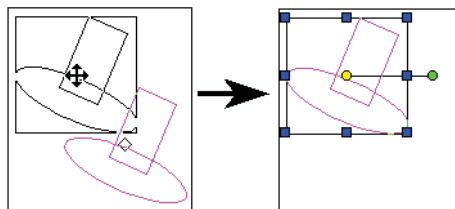
The easiest way to move objects is to select the objects, then click and move their reference point. You can also specify the move distances in the coordinate fields.

*NOTE: To move (or copy) objects relative to specific points or objects, see "Assembling" on page 475 and "Transforming" on page 212.*

1. Select the objects you want to move. (See "Changing the Reference Point" on page 199 if you want the reference point in a different place.)



2. Click the reference point, and the cursor changes to a four-directional arrow. Move the reference point to the new location, or enter the new position in the Inspector Bar.

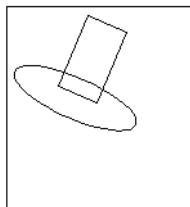


---

NOTE: You can make a copy by clicking the reference point, pressing **Ctrl**, and then locating the new reference point.

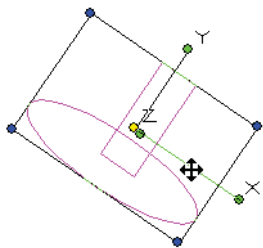
---

- When the objects are moved, click anywhere or press **Esc** to exit **Select** mode.



### Moving along One Axis

You can move or drag an object along the X or Y axis (or Z for a 3D object) by clicking and moving the axis handle itself.



### Simple Dragging

If you place the cursor anywhere inside a selection other than on the reference point, you can simply drag the selection (keeping the mouse pressed) to another location within the drawing. With simple dragging, you cannot use snaps or the Coordinate Fields, but it is convenient if you do not need accurate placement.

### OLE Drag and Drop

This method is essentially the same as moving objects in **Select** mode (clicking and moving the reference point.) However, for OLE drag and drop, the object is dragged - the mouse is kept pressed while moving.

The main advantage of OLE drag and drop is you can copy the selection from one drawing into another, or even from a drawing into another application. You may also use this

method to create symbols or blocks by dragging into the Library palette or Block palette. See "Blocks" on page 291 and ".Library" on page 304.

---

WARNING: The behavior of snap modes is slightly unusual when you are moving an object using OLE drag and drop. It is therefore not a recommended method for moving objects that require precise placement.

---

### Rotating Objects in Select Edit

The easiest way to rotate objects is to select the objects, then click and move their rotation handle. You can also enter rotation values in the Inspector Bar.

---

NOTE: To rotate (or copy) objects relative to specific points or objects, see "Assembling" on page 475 and "Transforming" on page 212.

---

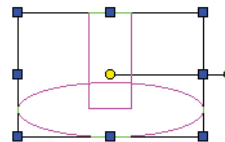


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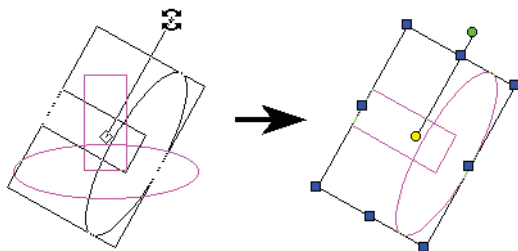
TIP: If you are using the 3D Selector, you can lock a rotation axis. Right-click on the rotation handle and select **Lock Axis** from the local menu. See "2D / 3D Selector" on page 181.

---

- Select the objects you want to rotate. (See "Changing the Rotation Bar" on page 200 if you want the rotation bar in a different place.)



2. Click the rotation handle, and the cursor changes to a rotation symbol. Rotate to the new position, or enter the angle in the Inspector Bar.

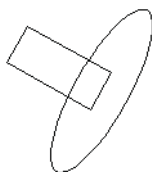



---

NOTE: You can make a copy by clicking the rotation handle, pressing Ctrl, and then rotating.

---

3. When the objects are rotated, click anywhere or press Esc to exit **Select** mode.



## Scaling Objects in Select Edit

The easiest way to scale objects is to select the objects, then click and move drag handles. You can also enter scale values for relevant axes in the Inspector Bar.

---

NOTE: To scale (or copy) objects relative to specific points or objects, see "Transforming" on page 212.

---

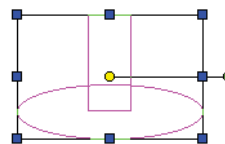


---

TIP: If you are scaling an ACIS solid object, you can specify uniform scaling or enable scaling in all axes - see "ACIS" on page 359.)

---

1. Select the objects you want to scale.



2. If you want to maintain the aspect ratio (the ratio between X, Y, and Z), select **Keep Aspect Ratio** from the local menu or click the icon on the Inspector Bar.



You can also maintain the aspect ratio by clicking a drag handle and pressing Shift while resizing.

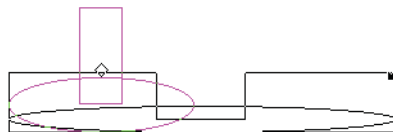
3. Click one of the drag handles to resize, or enter the scale and size in the Inspector Bar.

---

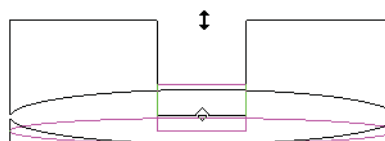
NOTE: You can make a copy by clicking the drag handle, pressing Ctrl, and then resizing.

---

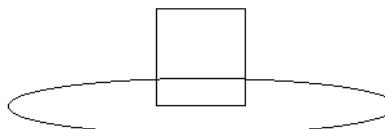
Click on a corner handle to scale in more than one dimension.



Click on an interior handle to scale in one dimension only.



4. When the objects are resized, click anywhere or press Esc to exit **Select** mode.



**NOTE:** If the object you are scaling contains double lines or lines drawn with a pen width greater than 0, the results of scaling a selection will vary, depending on whether you are using **Geometric** or **Cosmetic** selection. For accurate results, use the default **Geometric** select mode. See "Geometric and Cosmetic Select Modes" on page 198.

## Copying Objects in Select Edit

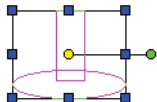
This section covers ways you can copy objects while in **Select** mode.

While these methods are generally the most convenient for making quick copies, there are other tools you can use for more complex and detailed copying. See "Copying Objects" on page 206. For other tools you can use to copy objects relative to specific points, see "Transforming" on page 212.

### Make Copy

Creates one copy of the selected objects.

1. Select the objects you want to copy.

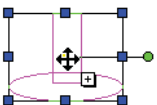


**WARNING:** If you want to change the reference point, do not do so while **Make Copy** is active, or you will create a copy on top of the original. Change the reference point first, then activate **Make Copy**. See "Changing the Reference Point" on page 199.

2. Select **Make Copy** from the local menu, or click the icon on the Inspector Bar.



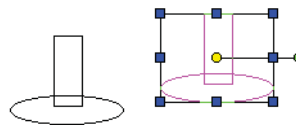
3. Click the reference point, and the cursor changes to a four-directional arrow with a plus sign.



4. Move the reference point to the new location, or enter the new position in the Inspector Bar.



5. The copied objects are now the ones selected. If you want to make another copy, you must click the reference point again.



6. When the objects are copied, click anywhere or press Esc to exit **Select** mode.



7. Turn off **Make Copy** unless it is still needed.

### Simple Dragging to Copy

If you place the cursor anywhere inside a selection other than on the reference point, you can simply drag the selection (keeping the mouse pressed) to another location within the drawing. If **Make Copy** is active, the objects will be copied instead of moved.

To activate **Make Copy**, select it from the local menu, or click the icon on the Inspector Bar.



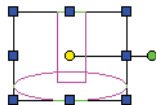
With simple dragging, you cannot use snaps or the Coordinate Fields, but it is convenient if you do not need accurate placement.

Turn off **Make Copy** when it is no longer needed.

## Rubber Stamp

Creates multiple copies of the selected objects.

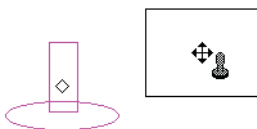
1. Select the objects you want to copy.



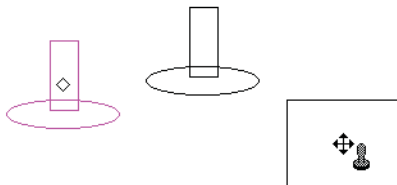
2. Select **Rubber Stamp** from the local menu, or click the icon on the Inspector Bar.



3. A rubber stamp icon appears, and you can click to locate the first copy.

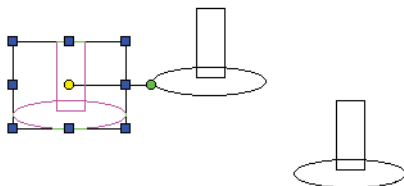


4. Create as many copies as you need.

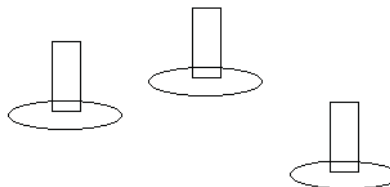


5. To finish press Esc, or select **Cancel** from the local menu or Inspector Bar.

The original objects are still selected.



6. Click anywhere or press Esc to exit **Select** mode.

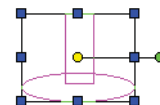


7. Turn off **Make Copy** unless it is still needed.

## Making a Rotated Copy

Creates one rotated copy of the selected objects.

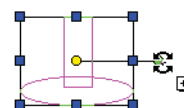
1. Select the objects you want to copy. (See "Changing the Rotation Bar" on page 200 if you want the rotation bar in a different place.)



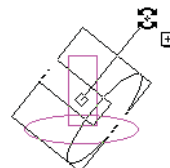
2. Select **Make Copy** from the local menu, or click the icon on the Inspector Bar.



3. Click the rotation handle, and the cursor changes to a rotation arrow with a plus sign.

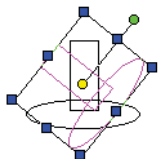


4. Move the rotation handle to the new location, or enter the angle in the Inspector Bar.





5. The copied objects are now the ones selected. If you want to make another copy, you must click the reference point again.



6. When the objects are copied, click anywhere or press Esc to exit **Select** mode.

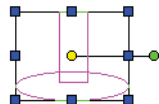


7. Turn off **Make Copy** unless it is still needed.

### Making a Scaled Copy

Creates one scaled copy of the selected objects.

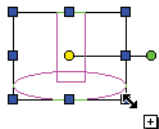
1. Select the objects you want to copy.



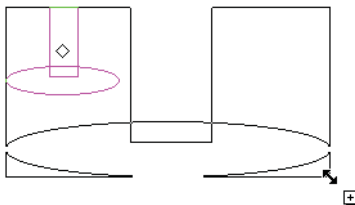
2. Select **Make Copy** from the local menu, or click the icon on the Inspector Bar.



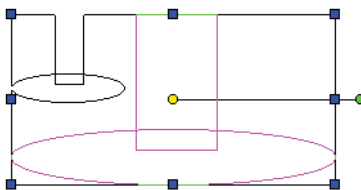
3. Click one of the drag handles, and the cursor changes to a resize symbol with a plus sign.



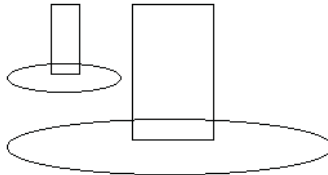
4. Drag the handle to the new location, or enter the new scale or size in the Inspector Bar.



5. The copied objects are now the ones selected. If you want to make another copy, you must click the reference point again.



6. When the objects are copied, click anywhere or press Esc to exit **Select** mode.



7. Turn off **Make Copy** unless it is still needed.

## Using Ctrl to Copy

An easy way of creating one copy of selected objects is to use the Ctrl key. The procedure is basically the same for moving, rotating, or scaling objects.

- To create a copy, select the objects and start moving by clicking the yellow reference point. Press Ctrl, and complete the move.
- To create a rotated copy, select the objects and start rotating by clicking the green rotation handle. Press Ctrl, and complete the rotation.
- To create a scaled copy, select the objects and start dragging one of the blue rotation handles. Press Ctrl, and complete the scaling.

## Copying Objects

The **Copy Entities** tools enable you to make single or multiple copies of selected objects, while precisely controlling the placement of the copies.

You can display the **Copy** toolbar by right-clicking in any toolbar area and selecting **Copy**.

---

**NOTE:** You can also make simple copies of objects in **Select Edit** (see "Copying Objects in Select Edit" on page 203). For other tools you can use to copy objects relative to specific points, see "Transforming" on page 212.

---

The **Copy Entities** tools can be used on 2D and 3D objects. For the objects you want to use, be sure the **Selector** is set correctly (**2D**, **3D**, or both). See "2D / 3D Selector" on page 181.

These tools are typically invoked when objects are already selected. You can also invoke them even if no objects are currently selected, as long as the **Select** tool is active. In this case, the local menu option **Select** can be used to select objects to copy. Close the **Select** option when all objects are selected.

### Local menu options:

For all of the **Copy** tools (except Copy in Place), the following local menu options are available:

**Select:** Enables you to add or remove objects from the selection set. Once the objects have been selected, deactivate this object to continue with the operation.



**Repeat:** Enables you to copy the selected objects again. Otherwise, the operation ends once the first operation is complete.



In *TurboCAD Platinum*, you can use **Auto Constraints** and to control the spacing of pattern-copied objects. See "Constraining Pattern - Copied Objects" on page 273.

---

**TIP:** When using the **Copy Entities** tools, you will probably need to switch often between the *Inspector Bar* and the drawing area. You can press **Tab** to enter the *Inspector Bar*, and click once in the drawing or press **Esc** to return to the drawing.

---

## Copy In Place

Available in *TurboCAD Pro*, *Platinum*, and *Deluxe* only

Copy in Place creates a copy of all selected objects in exactly the same place as they are.

1. Select the objects you want to copy and select **Copy in Place**.
2. The resulting copied objects will now be the selected objects.

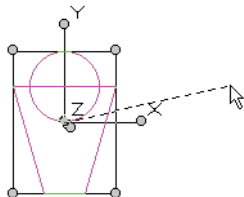
## Linear Copy



Creates copies of 2D or 3D objects along a straight line, in which each copy is separated by a specified distance.

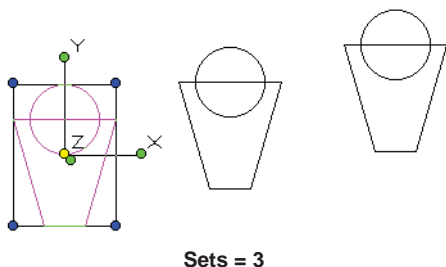
1. Select the objects you want to copy and activate **Linear Copy**.
2. In the *Inspector Bar*, enter the number of **Sets** - the total number of objects that will result.

- Set the distance between adjacent copies by using the mouse to define the copy vector, or enter **Step** values in the Inspector Bar. The copy vector is defined from the reference point of the selection set.

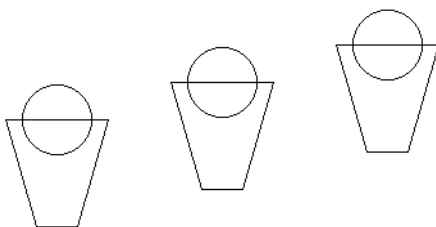


NOTE: To make a single copy using a distance vector that does **not** start at the reference point of the selected objects, see "Vector Copy" on page 211.

- The copies are made after the vector has been defined.



- Click anywhere to exit **Select** mode, or press Esc.



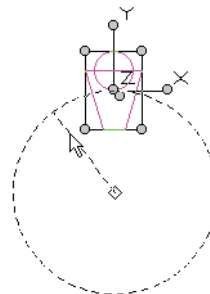
## Radial Copy



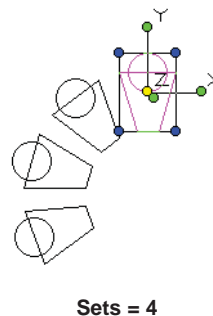
Creates copies of 2D or 3D objects along an arc, in which each copy is separated by a specified angle and rotation (optional).

- Select the objects you want to copy and activate **Radial Copy**.
- In the Inspector Bar, enter the number of **Sets** - the total number of objects that will result.
- Set the angle between adjacent copies by using the mouse to define the center of the copy arc and the angle, or enter the angle in the Inspector Bar. You can enter a rotation value to set the rotation of each copy relative to the previous one. If **Rotation** = 0, the copies will not rotate at all. By default, **Rotation** = **Angle**.

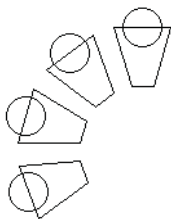
The copy arc is defined from the reference point of the selection set.



- The copies are made after the arc has been defined.



- Click anywhere to exit **Select** mode, or press Esc.



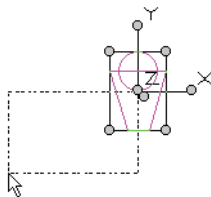
NOTE: For copying in 3D, the copy arc is positioned on the current workplane. Each copy is rotated around the axis perpendicular to the workplane, passing through the reference point of the selected objects.

## Array Copy

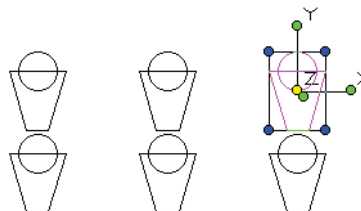


Creates copies of 2D or 3D objects into a 2D or 3D linear array, in which each copy is separated by a specified distance.

- Select the objects you want to copy and activate **Array Copy**.
- In the Inspector Bar, enter the *total* number of **Rows**, **Column**, and **Levels** (number of copies in Z) that will result.
- Set the distance between adjacent copies by using the mouse to define the copy vector, or enter **Step** values in the Inspector Bar. The copy vector is defined from the reference point of the selection set.

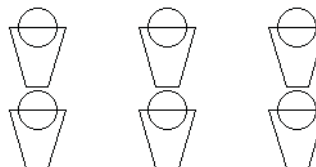


- The copies are made after the vector has been defined.



**Rows = 2, Columns = 3**

- Click anywhere to exit **Select** mode, or press Esc.

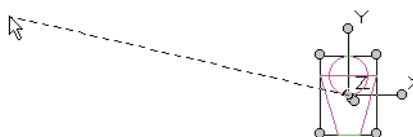


## Fit Linear Copy

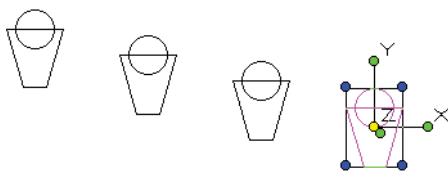


Creates copies of 2D or 3D objects along a straight line, along which a specified number of copies are evenly spaced.

- Select the objects you want to copy and activate **Fit Linear Copy**.
- In the Inspector Bar, enter the number of **Sets** - the *total* number of objects that will result.
- Set the distance between the first and last copies by using the mouse to define the copy vector, or enter size values in the Inspector Bar. The copy vector is defined from the reference point of the selection set.

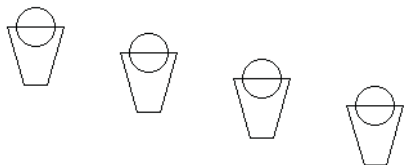


- The copies are made after the vector has been defined.



Sets = 4

- Click anywhere to exit **Select** mode, or press Esc.



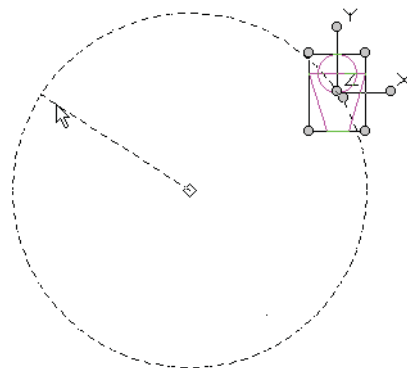
### Fit Radial Copy



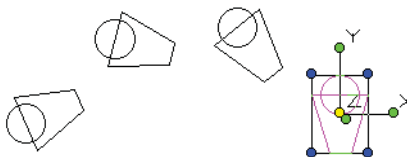
Creates copies of 2D or 3D objects along an arc, along which a specified number of copies are evenly spaced and rotated.

- Select the objects you want to copy and activate **Fit Radial Copy**.
- In the Inspector Bar, enter the number of **Sets** - the total number of objects that will result.
- Set the angle the first and last copies by using the mouse to define the center of the copy arc and the angle, or enter the angle in the Inspector Bar. You can enter a rotation value to set the rotation of each copy relative to the previous one. If **Rotation** = 0, the copies will not rotate at all.

The copy arc is defined from the reference point of the selection set.

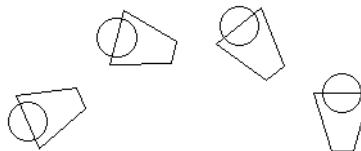


- The copies are made after the arc has been defined.



Sets = 4

- Click anywhere to exit **Select** mode, or press Esc.



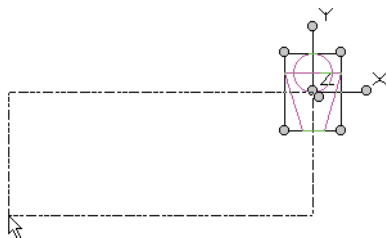
**NOTE:** For copying in 3D, the copy arc is positioned on the current workplane. Each copy is rotated around the axis perpendicular to the workplane, passing through the reference point of the selected objects.

## Fit Array Copy

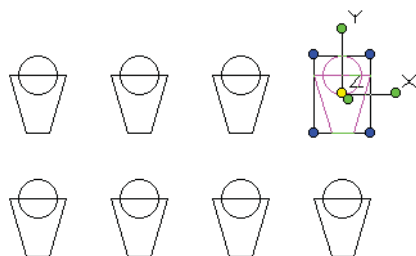


Creates copies of 2D or 3D objects into a 2D or 3D linear array, in which a specified number of copies are evenly spaced.

1. Select the objects you want to copy and activate **Fit Array Copy**.
2. In the Inspector Bar, enter the *total* number of rows, columns, and levels (number of copies in Z) that will result.
3. Set the distance between first and last copies in each direction by using the mouse to define the entire copy array boundary, or enter **Bound** values in the Inspector Bar. The copy array is defined from the reference point of the selection set.

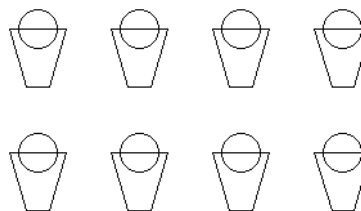


4. The copies are made after the array boundary has been defined.



Rows = 2, Columns = 4

5. Click anywhere to exit **Select** mode, or press Esc.

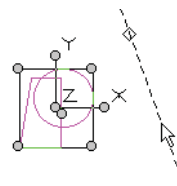


## Mirror Copy

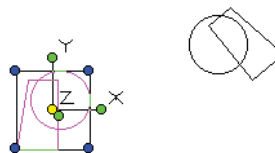


Creates a mirror image of an object by defining a mirror line.

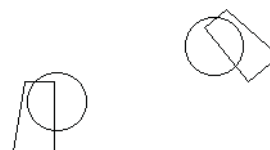
1. Select the objects you want to mirror and activate **Mirror Copy**.
2. Define the mirror line by selecting two points, or by selecting one point and entering an angle in the Inspector Bar.



3. The mirror copy is made after the mirror line has been defined.



4. Click anywhere to exit **Select** mode, or press Esc.



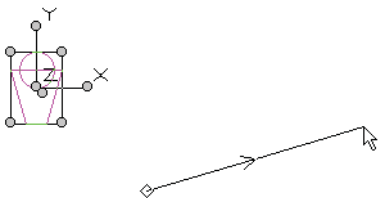
NOTE: For text to be mirrored, it must be **Flexible**. Otherwise the text will read the same way on both sides of the mirror line. See "Text Properties" on page 325.

## Vector Copy



Creates a single copy of an object at a specified angle and distance from the reference point of the original.

1. Select the objects you want to copy and activate **Vector Copy**.
2. Define the copy vector by selecting two points, or by selecting one point and entering the length and angle in the Inspector Bar. The copy vector can be anywhere, but it is applied to the reference point of the selection set.

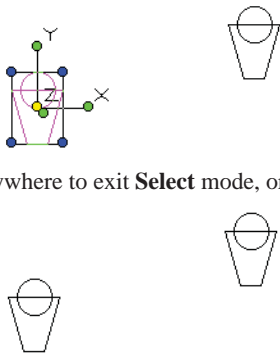



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*NOTE: To make one or more copies using a distance vector that starts at the reference point of the selected objects, see "Fit Linear Copy" on page 208.*

---

3. The copy is made after the vector has been defined.



4. Click anywhere to exit **Select** mode, or press Esc.

## Offset



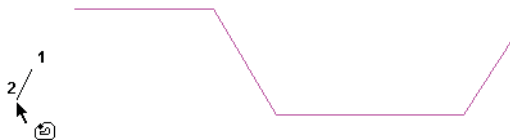
Creates one or more offset copies of a single object. The object can be open or closed. For closed objects, the offsets will also be closed.

The offsets are created so that linear segments remain parallel. When possible, arc start and end angles remain the same as well.

1. Activate the function, and select the object (line, arc, polyline, polygon, etc.) you want to offset.



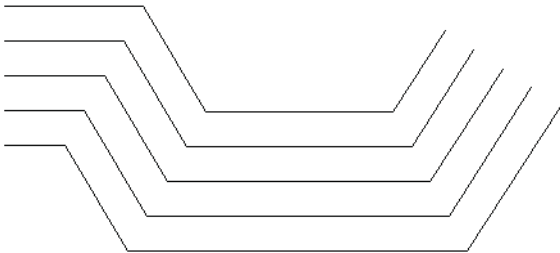
2. Set the offset distance either by entering it in the Inspector Bar or by selecting two points. The angle of this line does not matter; its absolute length will be used.



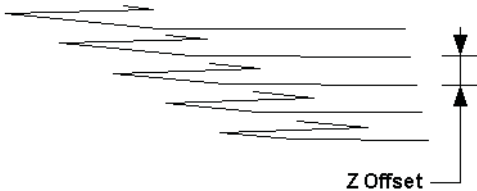
3. Enter the number of **Sets** in the Inspector Bar. This is the total number of copies, and does not include the original.
4. Select the side of the object where you want to place the copies.



The offset copies are made on the selected side.



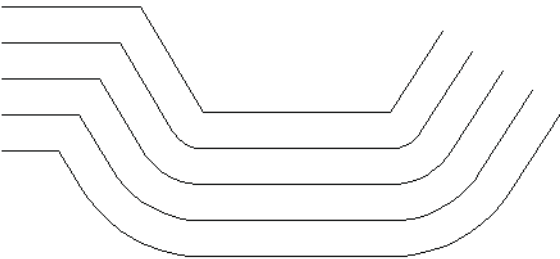
5. If you enter a **Z Offset**, each offset curve will be separated from the adjacent one by this distance.



6. You can use the same parameters to offset other objects, you can change the parameters, or select **Cancel** from the local menu or Inspector Bar to exit the tool.

Local menu option:

**Rounded Corners:** The exterior corners (relative to the offset direction) will be rounded.



# Transforming

*Available in TurboCAD Pro, Platinum and Deluxe only*

The **Transform** tools enable you to quickly move, rotate, or scale objects. You can optionally leave the original object in place, creating a copy. In addition, you can record the transform operations for use on other objects.

You can display the **Transform** toolbar by right-clicking in any toolbar area and selecting **Transform**.



The **Transform** tools can be used on 2D and 3D objects. For the objects you want to use, be sure the Selector is set correctly (2D, 3D, or both). See "2D / 3D Selector" on page 181.

*NOTE: The **Assemble** tool also enable you to move and rotate objects, relative to other objects or locations. **Assemble** tools require more accurate input than **Transform** tools, however, and do not create copies. See "Assembling" on page 475.*

The **Transform** tools are typically invoked when objects are already selected. You can also invoke them even if no objects are currently selected, as long as the **Select** tool is active. In this case, the local menu option **Select** can be used to select objects to copy. Close the **Select** option when all objects are selected.

For each of the transform tool (**Move**, **Rotate**, **Scale**, and **Generic**), the following local menu options are available:

**Select:** Enables you to add or remove objects from the selection set. When the correct objects have been selected, deactivate this object to continue with the operation.



**Repeat:** Enables you to transform or copy the selected objects again. Otherwise, the function ends once the first operation is complete.





**Keep Original Object:** Activated by default, makes a copy of the selected objects. Otherwise the object is simply transformed.



**Finish:** Complete the transform operation.

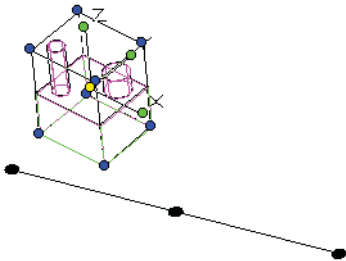


## Move

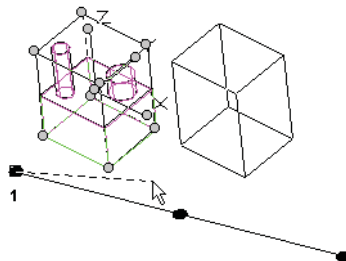


Moves or copies objects relative to two defined points.

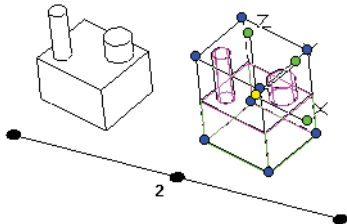
1. Select the objects you want to move or copy, and activate **Move**.



2. Select the source point - the first of two points that defines the transform vector.

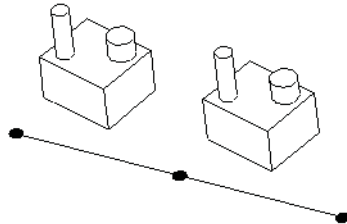


3. Select the destination point. The objects are moved along the transform vector from their original location.

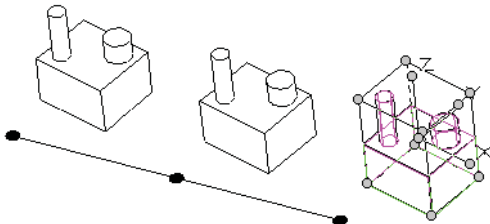


**NOTE:** By default, the original objects remain in place. If you do not want to create a copy, deselect the local menu option **Keep Original Object**.

4. The new or moved object is now the one selected. Click outside to deselect it.



If you use the local menu option **Repeat**, you can transform the selected objects again. Otherwise the operation ends.

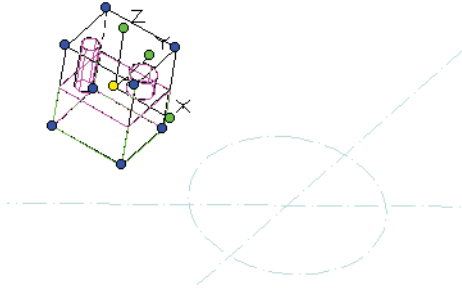


## Rotate



Rotates and/or copies objects relative to three defined points.

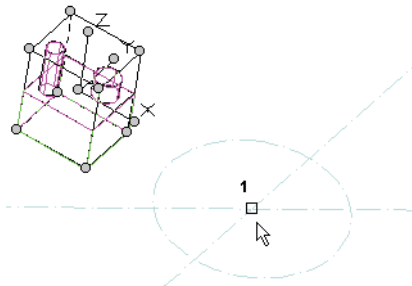
1. Select the objects you want to rotate or copy, and activate **Rotate**.



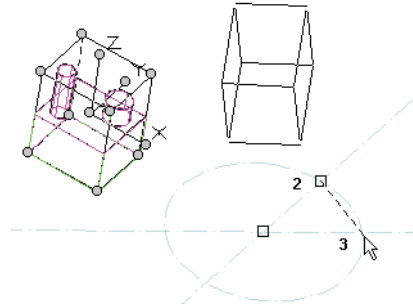
2. From the local menu or Inspector Bar, select the axis of rotation (X, Y, or Z).



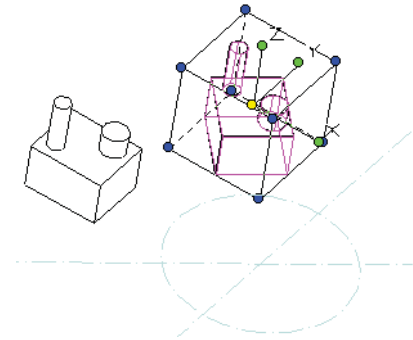
3. Select the center of rotation.



4. Select the source and destination points - these establish the transform angle relative to the rotation center.



5. The objects are moved along the transform angle from their original location.

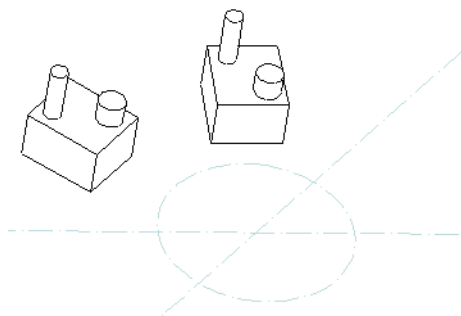



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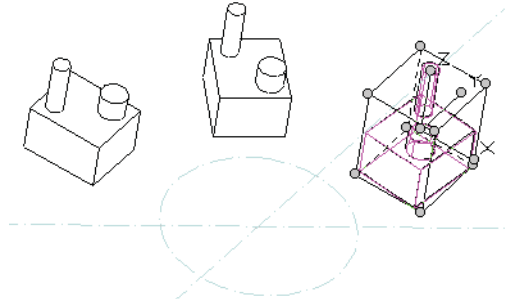
NOTE: By default, the original objects remain in place. If you do not want to create a copy, deselect the local menu option **Keep Original Object**.

---

6. The new or moved object is now the one selected. Click outside to deselect it.



If you use the local menu option **Repeat**, you can transform the selected objects again. Otherwise the operation ends.

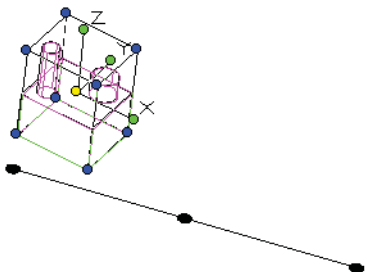


## Scale

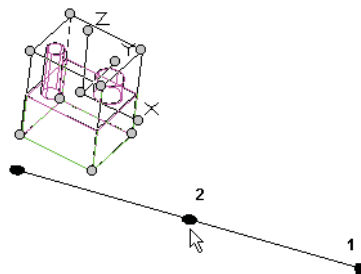


Scales and/or copies objects relative to three defined points.

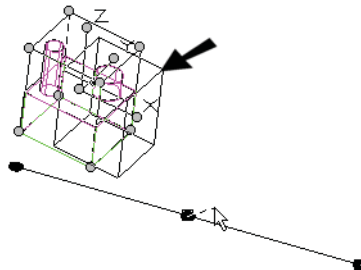
1. Select the objects you want to scale or copy, and activate **Scale**.



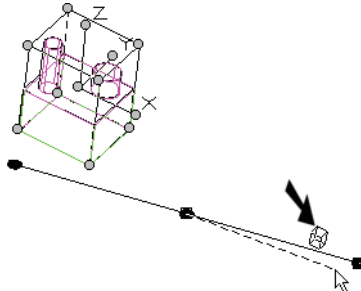
2. Select the scaling start point and end point.



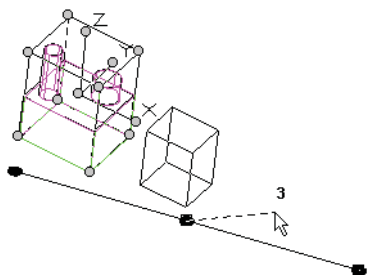
When selecting the destination point, the closer to the scaling end point, the closer the scale approaches 1:1.



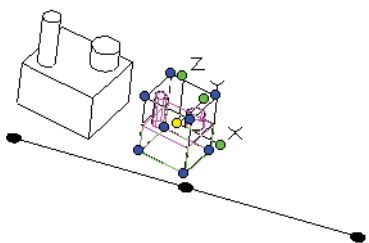
The closer to the scaling start point, the scale approaches zero.



3. Select the scaling destination point.



4. The objects are scaled, relative to their original location.

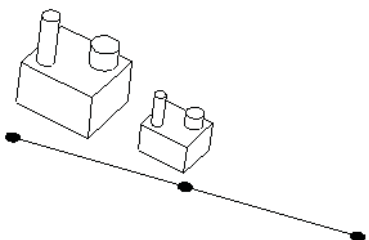



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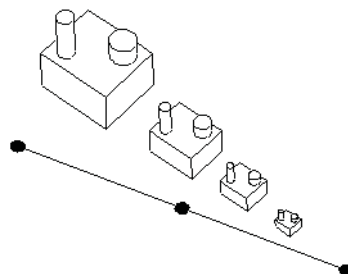
**NOTE:** By default, the original objects remain in place. If you do not want to create a copy, deselect the local menu option **Keep Original Object**.

---

5. The new or moved object is now the one selected. Click outside to deselect it.



If you use the local menu option **Repeat**, you can transform the selected objects again. Otherwise the operation ends.

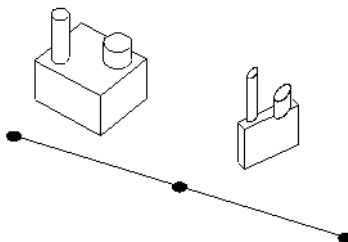


Additional local menu option:

**Keep Aspect Ratio:** By default, the objects are scaled so that all dimensions remain proportional to the original.



Deactivate this option if you want to be able to change the aspect ratio.



## Generic



Transforms a 3D object by selecting six points.

The first two points define the movement distance, similar to using **Move**.

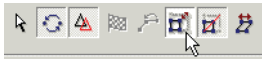
The third and fourth points enable you to rotate the object in the current workplane.

The fifth and sixth points enable you to flip the object.

*NOTE: The Generic Transform tool is also available from the local menu of the selector tool.*

Local menu options:

**Scaling:** Enables you to change the object's scale while transforming.



**Keep Aspect Ratio:** If **Scaling** is used, choose whether to maintain the object's aspect ratio.



**Skew:** If **Scaling** is used and **Keep Aspect Ratio** is not used, you can scale the object while transforming.

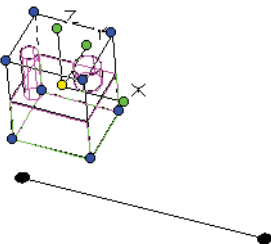


## Transform Recorder

*Available in TurboCAD Pro and Platinum only*

Enables you to record a transformation and apply it to other objects.

1. Select the objects whose transformation you want to record.

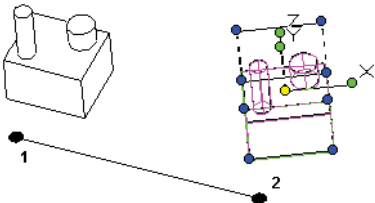


2. Click **Start Transform Recorder**.



3. Perform the transformation, using the **Move**, **Rotate**, **Scale**, and **Generic** tools as needed.

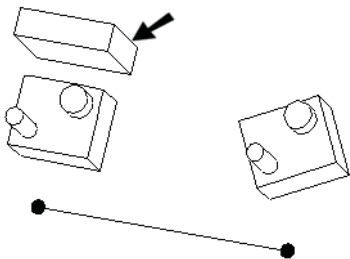
In this example, **Move** with **Keep Original Object** was used to copy the object from Point 1 to Point 2. Then **Rotate** was used on the copied object, with **Keep Original Object** turned off.



4. When the transformation is complete, click **Stop Transform Recorder**.



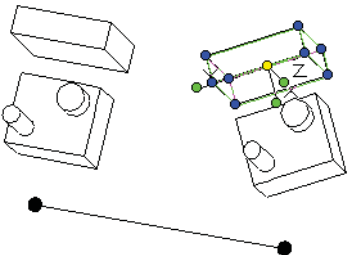
5. Add a new object or group of objects to which to apply the recorded transformation.



6. Select this object and click **Apply Stored Transformation**.



7. The **Move** and **Rotate** commands are applied to the selected object.



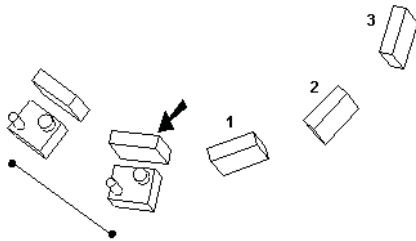
8. If you want to apply the transformation with different settings, click **Player**.



9. In the Inspector Bar you can specify the number of sets (the number of *additional* copies) to create. Set any additional parameters in the local menu or Inspector Bar, and click **Play**.



In this example, the transformation was applied to the previous copy, and three additional copies were made.



It is important to note that **Play (Apply Stored Transformation)** uses settings from the **Player (Customize and Apply Stored Transformation)**. **Player** options remain in effect until changed, and will affect subsequent transformations, even if they are re-recorded.

---

**NOTE:** *If you specify multiple **Sets** and **Keep Original Objects** is turned off, the selected objects will be transformed by the distance times the number of sets. No copies will be made.*

---

#### Local menu options:

Only one option is unique to **Player**.

**Toggle CS:** If not selected, the object will be rotated around the center of the original object. If selected, the reference point of the new object will be used as the center of rotation.

# 7 Editing and Modifying 2D Objects

This chapter focuses on tools used to change geometry and formats of objects, as well as measuring tools.

You can also perform editing commands in **Select** mode, including moving, rotating, scaling, and copying. See "Select Edit" on page 198.

## Edit Tool



Nodes are hidden markers identifying elements of geometric objects. The **Edit Tool** enables you to reshape objects by editing their nodes. You can edit a single node or multiple nodes - even a group of nodes belonging to different objects.

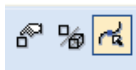
*NOTE: This section focuses on editing nodes of 2D objects. For 3D objects, see "Edit Tool in 3D" on page 427. For editing tables, see "Changing Rows and Columns, Merging Cells" on page 623. And for editing walls, see "Editing Walls" on page 539.*

There are several ways to activate the **Edit Tool**:

- Click **Edit Tool** on the **Select** toolbar, then select the object to edit.



- Select **Edit Tool**, then select the object to edit.
- Select the object that you wish to edit, then select **Edit Tool** from the local menu. You can also click the icon on the Inspector Bar.



- Select the object to edit, then press Ctrl and select it again.
- Access the **Edit Tool** from the Selection Info Palette. See "Selection Info Palette" on page 191.

In **Edit Tool** mode, the cursor becomes an arrow with a dot.



When the node editing is finished, you can return to **Select Edit** mode by selecting it from the local menu, by clicking the icon on the Inspector Bar, or by pressing Esc.

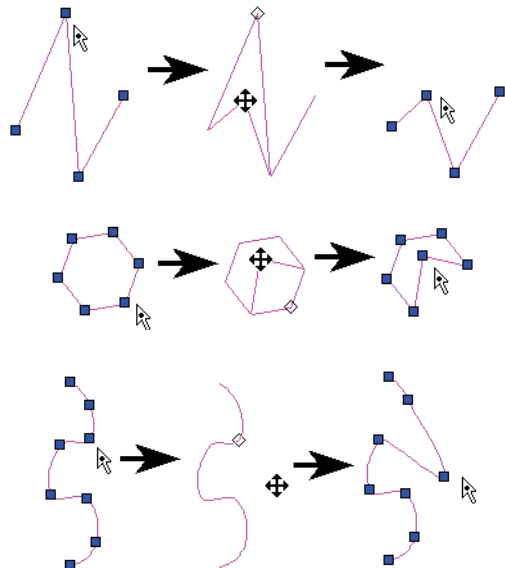


You can then exit **Select** mode by clicking outside the selection set, or by pressing Esc again.

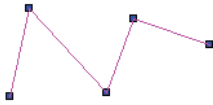
*NOTE: If you need to node edit an object that is part of a group, explode the group first (**Format / Explode**). For an object that is part of a block, you can node edit the object in **Edit Content** mode. See "Editing a Block" on page 295.*

## Moving a Node

The most basic method of node editing is to move nodes. Simply click on any node and drag it to its new location.



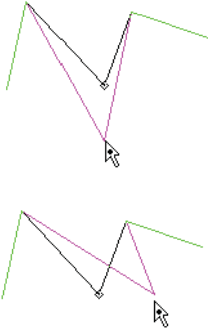
If you want to move nodes orthogonally, you can press Shift while dragging. This example will use a polyline:



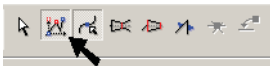
In the local menu or Inspector Bar, if **Ortho Origin by Entity** is disabled, the nodes will be moved orthogonally according to the active UCS.



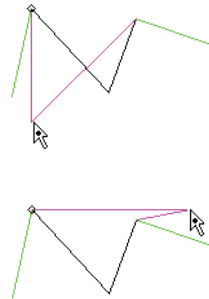
In this mode, you can move the node directly above, below, left, or right of its current position.



If **Ortho Origin by Entity** is enabled, the node will move so that the segment itself will become orthogonal.



In this mode, the segment before the node will become either vertical or horizontal.

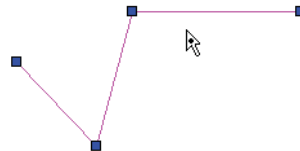


For Bezier curves, you can choose to have additional control while node editing. By checking **Show additional control points** in the curve's **Properties**, you can add two more control nodes at every node. You can also control the tangency at each node. See "Curve Properties" on page 168.

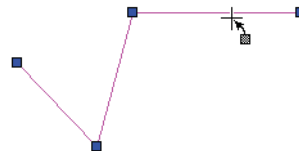
## Adding a Node

Adds nodes to lines and line segments, including objects created using double lines and multi lines. You can also add nodes to Bezier curves, splines, and sketches.

1. Use the **Edit Tool** on the object to which you want to add a node.

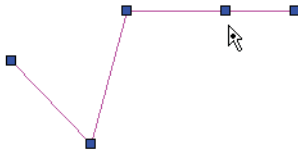


2. Press Ctrl while placing the cursor where you want to add the node. The cursor has a + symbol.



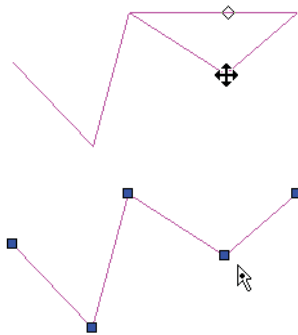


- Click to add the node.

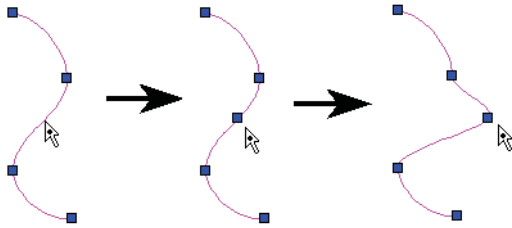


(You could also place the cursor where you want to add the node, then select **Add Node** from the local menu.)

- You can now drag the new node to create a new line segment.



The same method can be used to add nodes to curves.



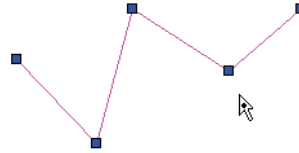
## Deleting Nodes

Deletes nodes from lines and line segments, including objects created with double lines and multi lines. You can also delete nodes from Bezier curves, splines, and sketches.

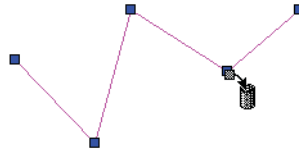
You cannot delete a node that is needed to define an object as a line segment or polygon. This means that you cannot delete the endpoint of a single line segment, and you cannot delete

any node of a triangle. If you delete the endpoint of a polyline, you will delete the entire line segment that terminates at that point.

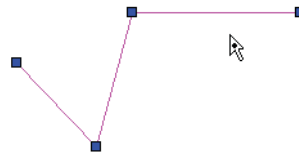
- To delete a single node, use the **Edit Tool** on the object from which you want to delete a node.



- Press Ctrl while placing the cursor where you want to delete the node. The cursor becomes a “trash can” symbol.



- Click to delete the node. The adjacent nodes will be connected to form a new segment.



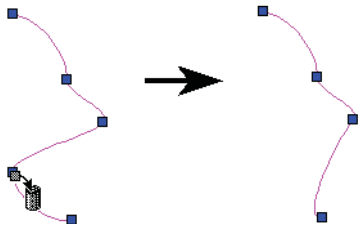
You can also use the local menu to delete one or more nodes. Within the **Edit Tool**, drag a selection window around the node or nodes you want to delete. Selected nodes turn from blue to magenta. Then select **Delete Node(s)** from the local menu.

---

**NOTE:** If you remove a node from a polygon, the polygon will remain closed, but will become an irregular polygon.

---

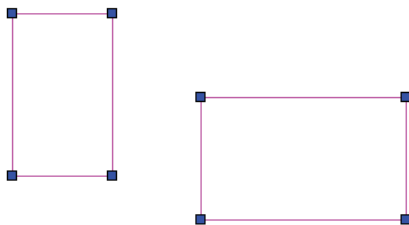
You can also delete nodes from Bezier curves, splines, and sketches.



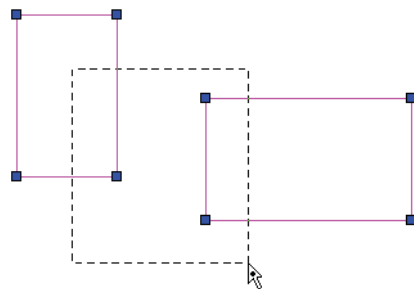
## Editing Multiple Nodes

You can use a selection window to edit a group of nodes, even if the nodes belong to different objects.

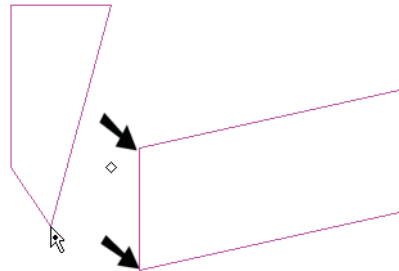
1. Activate the **Edit Tool**. To select multiple objects use the Shift key. You can use Ctrl+A to select all objects for node editing.



2. Use a selection window to enclose all the nodes you want to edit.



3. The selected nodes turn from blue to magenta. When you move any of the selected nodes, all nodes move as a group.

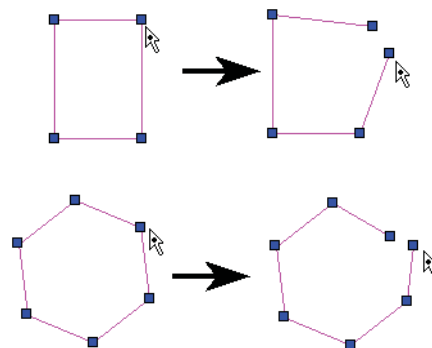


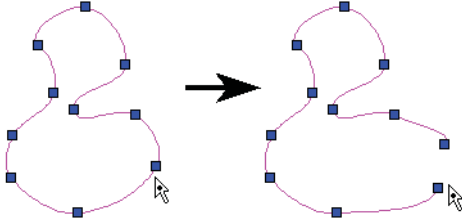
4. To deselect nodes, draw an “empty” selection window (one that does not encompass any nodes).

## Breaking (Opening) Closed Objects

Breaks a closed object (polygon, closed polyline, rectangle, closed curve, etc.), turning it into a polyline.

1. Use the **Edit Tool** on the closed object you want to break.
2. Right-click on the node at which you want the break, and select **Break** from the local menu.
3. The object is now an open polyline, with two end nodes at the node where you broke the object.

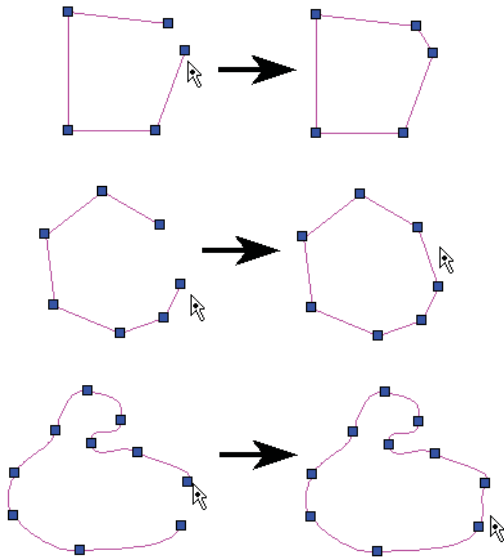




## Closing Open Objects

Closes an open object (polyline, open curve), turning it into a closed polyline, curve, or polygon.

1. Use the **Edit Tool** on the open object you want to close.
2. Right-click on the node at which you want to close the object, and select **Close** from the local menu.
3. A line segment is created between the node you selected and the other endpoint.



**NOTE:** If the two endpoints of the polyline share the same location, the object will simply be closed geometrically.

If you simply snap the endpoints of a polyline together, the resulting object will only appear to be closed, but is still considered open. This means, for example, that the object cannot be filled with a hatch pattern. In order to geometrically close the object, you must use this feature.

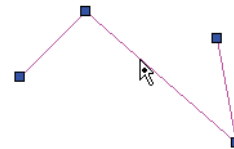
## Editing Linear Objects

In addition to adding, deleting, or moving nodes, there are several **Edit Tool** features unique to linear objects.

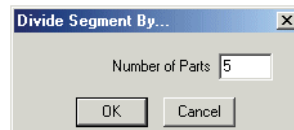
### Dividing a Line Segment

Adds nodes to a line or line segment, breaking it into a polyline composed of segments of equal length. Double lines and multi lines can also be divided.

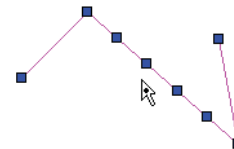
1. Use the **Edit Tool** on the object you want to divide.



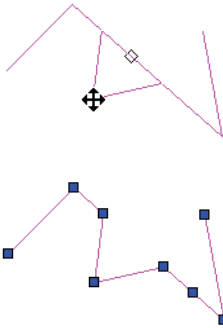
2. Right-click over the segment you want to divide, and select **Divide Segment** from the local menu. Enter the number of segments.



The selected segment is divided into the specified number of segments.



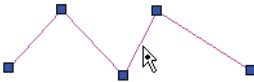
3. You can now drag any of the new nodes to create new line segments.



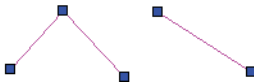
## Hiding or Showing a Line Segment

Applies only to single-line segments.

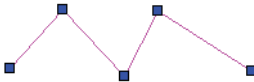
1. Use the **Edit Tool** on the object whose segment you want to hide.



2. Right-click over the segment you want to hide, and select **Hide Segment** from the local menu. The segment is no longer displayed.



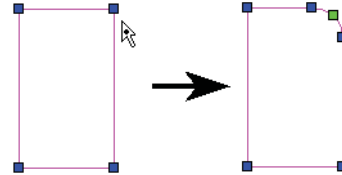
3. To show the segment again, select **Show Segment** from the local menu.



## Filletting Two Line Segments

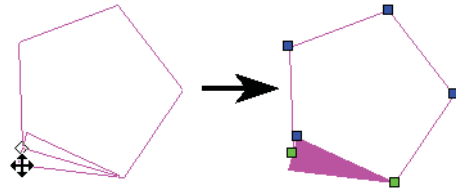
Adds an arc connecting two line segments, creating a fillet.

1. Use the **Edit Tool** on the multi-segmented object you want to fillet (polyline, rectangle, polygon, etc.).
2. Enter the fillet radius in the Inspector Bar.
3. Right-click on the node to fillet and select **Insert Arc**.

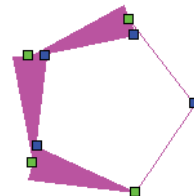


## Modifying Line Widths

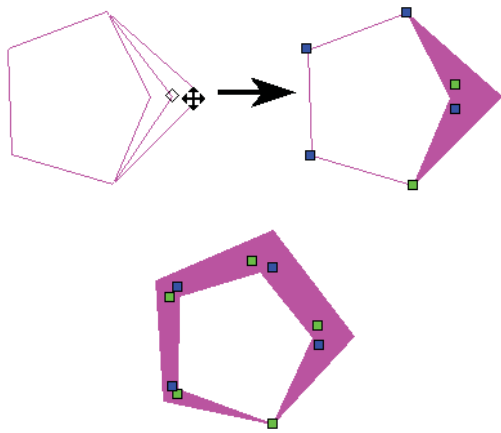
1. Use the **Edit Tool** on the line object (line, rectangle, polygon, etc.) whose width you want to change. Arc segments of polylines can be edited as well.
2. Right-click and select **Edit Widths**. Drag an endpoint to widen it, or enter the width in the Inspector Bar.



With this method, each endpoint will be sized individually, with no relation to any adjacent segments.



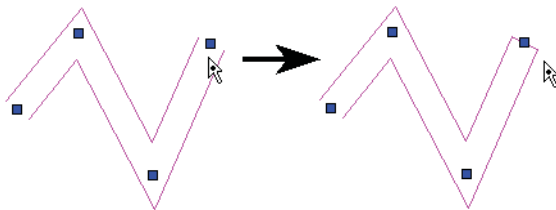
To modify the width of all adjacent segments, right-click and select **Edit Linked Widths**.



### Closing and Opening Endpoints of Double Lines

Creates or removes a start or end cap of a double line. This can also be controlled in the double line's **Properties** - see "Double Line Properties" on page 147.

1. Use the **Edit Tool** on the double line object whose endpoint you want to close.
2. Right-click on the end node, and select **Close Double End**.



3. To open the endpoint, select **Open Double End**.

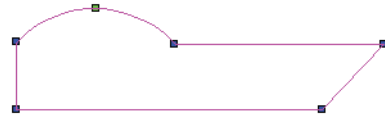
### Modifying Double Line Intersections

Right-click on the intersection node you want to modify. For the options, see "Double Line Options" on page 245.

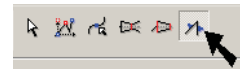
### Editing Segments

For objects with multiple segments, you can move segments so that they remain parallel to their original location. Arc segments can also be edited.

1. Use the **Edit Tool** on a multi-segmented object like a polyline, rectangle, polygon, etc.



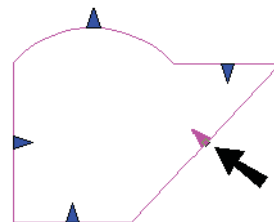
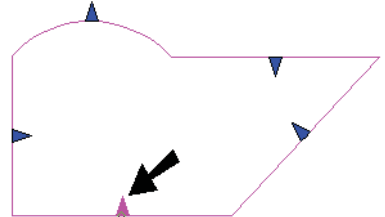
2. From the local menu or Inspector Bar, select **Edit Segments**.



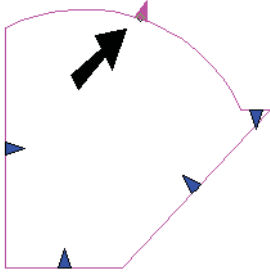
The nodes are no longer marked, and segments are marked with triangles.



3. Click and drag one of the segments. It remains parallel to its original location, and its length updates to meet adjacent segments.



4. If you edit an arc segment, its radius will change and its center will remain in place.



NOTE: This also works for walls, but each wall can be node edited separately when using **Edit Segments**. See "Editing Walls" on page 539.

## Editing Circles, Ellipses, and Arcs

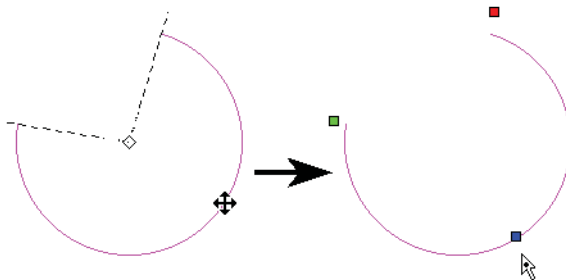
In addition to adding, deleting, or moving nodes, there are several **Edit Tool** features unique to circular and elliptical objects.

### Changing the Radius of a Circle or Arc

1. Use the **Edit Tool** on the circle or arc whose radius you want to change.



2. Click and move the blue handle to resize the object. You can also enter the new radius in the Inspector Bar.

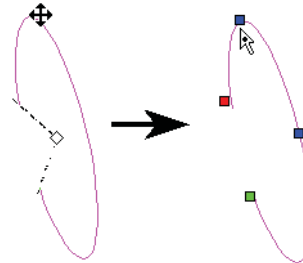


### Changing the Axes of an Ellipse or Elliptical Arc

1. Use the **Edit Tool** on the ellipse or elliptical arc whose axes you want to change.
2. There are two blue handles, one for the major axis and one for the minor.



3. Click and move the relevant blue handle to resize the axis. You can also enter the new axis length in the Inspector Bar.



To scale the object while maintaining its aspect ratio (major axis to minor axis), hold the Shift key while dragging either blue handle.

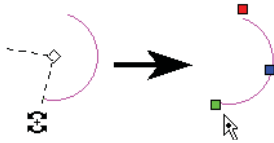
### Changing Start and End Angles

Circles, arcs, ellipses, and elliptical arcs all have start and end angles. For circles and ellipses the start and end angles are equal, but you can change this with the **Edit Tool**.

1. Use the **Edit Tool** on the object whose start and end angles you want to change.
2. The green handle is used to change the start angle, and the red handle is for the end angle.



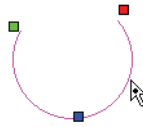
- Click and move either angle handle to resize the angles. You can also enter the new angles in the Inspector Bar.



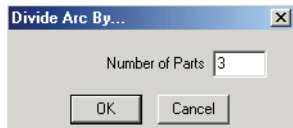
### Dividing an Arc

Adds nodes to an arc or circle, breaking it into separate arcs. The resulting arcs can be edited just like any arc.

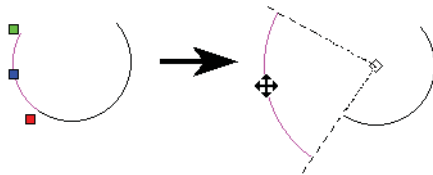
- Use the **Edit Tool** on the arc or circle you want to divide.



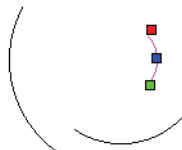
- Right-click over the arc you want to divide, and select **Arc Divide** from the local menu. Enter the number of arcs.



The arc is divided into the specified number of arcs. Each resultant arc can be node edited separately.



- To node edit a different arc, simple click on it while still in the **Edit Tool**.



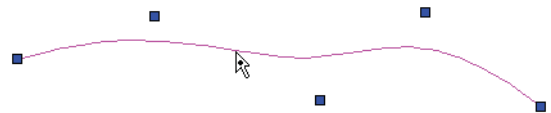
**TIP:** To divide an arc segment that is part of a polyline, right-click on the green node and select **Insert Arc**. This divides the arc into two equal segments.

### Editing Splines and Bezier Curves

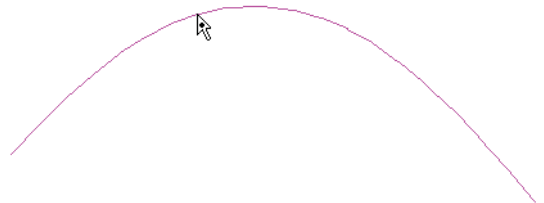
In addition to adding, deleting, or moving nodes, there are several **Edit Tool** features unique to splines and Bezier curves.

#### Changing the Shape of a Spline or Bezier Curve

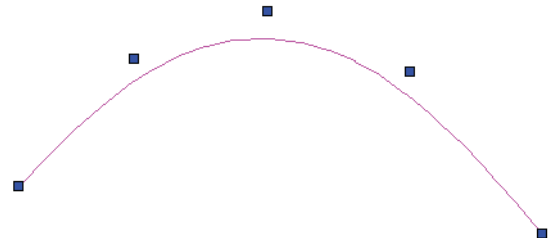
- Use the **Edit Tool** on the curve you want to edit.



- Click and drag any point along the curve.



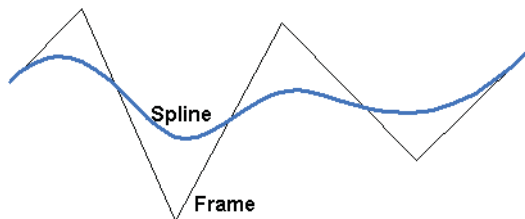
The control points update accordingly.



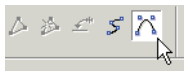
## Changing Control Points and Fit Points

Splines are created by specifying either a series of fit points (points through which the spline passes) or control points (guide points). For either type of spline, you can view and move both types of points.

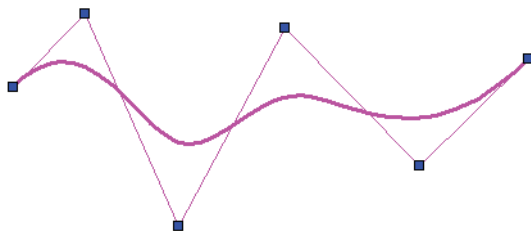
It is easier to understand the types of points when the spline is displayed with its frame. Open the spline's **Properties** to the **Curves** page and check **Show Frame**.



1. Use the **Edit Tool** on the curve you want to edit. By default, **Edit Control Points** is active in the Inspector Bar and local menu.



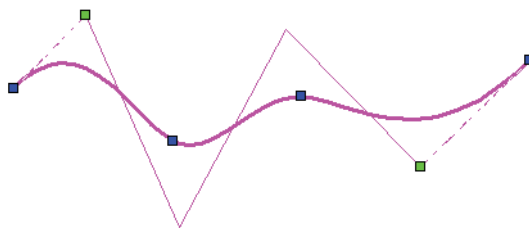
2. The control points are highlighted at the ends of frame segments. Each point can be clicked and dragged manually or assigned values in the Inspector Bar or Coordinate Fields.



3. Switch to **Edit Fit Points**.



4. Now the points through which the spline passes are highlighted. If the spline is open, you can use the green points to change the slope at the spline ends.

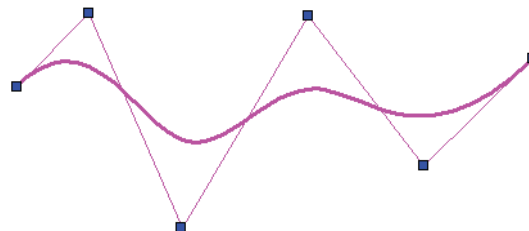


**NOTE:** Control points and fit points are not available for Bezier curves. For details on node-editing Bezier curves, see "Curve Properties" on page 168.

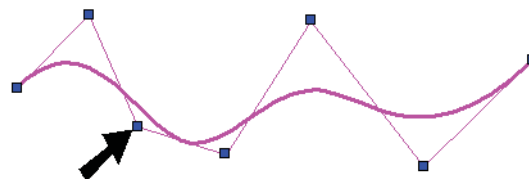
## Adding Nodes and Knots

A knot is basically a control point. You can add knots to increase the editing detail of a spline. A node is a point through which the spline will pass (fit point).

1. Use the **Edit Tool** on the spline you want to edit. In this example, the frame of the spline is displayed (check **Show Frame** on the **Curve** page of the spline's **Properties**).

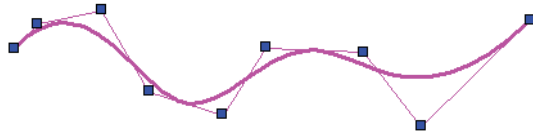


2. It does not matter whether fit points or control points are highlighted. To add a control point (knot), right-click on the spline (not on the frame) where you want to add a knot and select **Add Knot**. The control point is added along the frame.

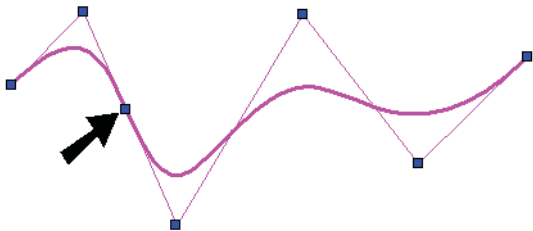




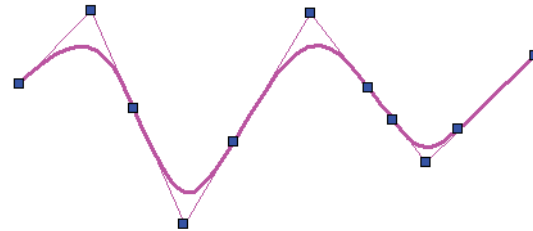
3. If you continue adding knots, you can “tighten” the frame to the spline, providing more control points for editing.



4. To add a fit point (node), right-click on the spline, near where it passes through the frame. Select **Add Node**. The fit point is added along the frame.



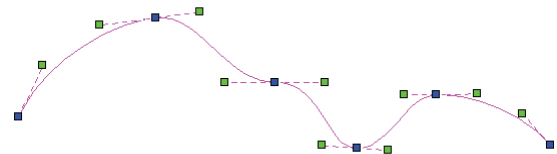
5. If you continue adding nodes, you can “tighten” the spline to the frame, providing more fit points for editing.



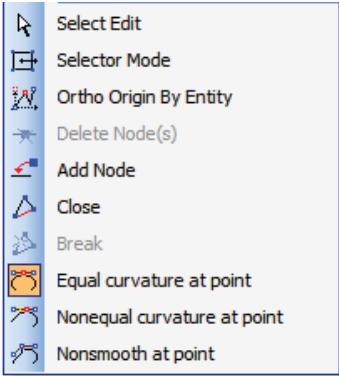
*NOTE: Nodes can be added to Bezier curves as well, but knots can be added only to splines.*

### Changing Node Curvature of Beziers

Bezier curve use weighted nodes to control the curvature at each node point.



You can control how each node controls the curvature by selecting a node, and right clicking to open the local menu. In the local menu there are three options for controlling the curvature of a node.

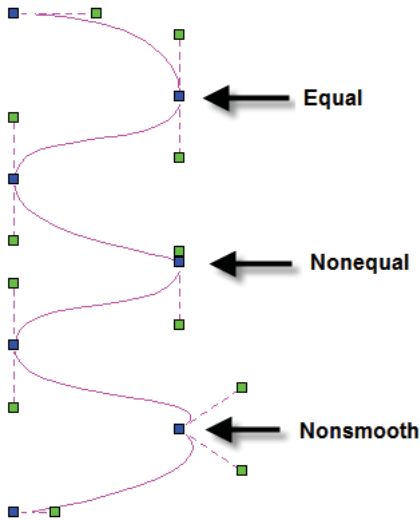


**Equal Curvature at point** — This is the default. When selected, adjusting one of the green control handles causes the paired handle to change an equal distance, but the handles remain locked to a common control line. This option makes the curve smooth and equal on both sides of the node.

**Nonequal Curvature at point** — When this option is selected, adjusting one of the green control handles does not affect the distance of the paired handle, but the nodes remain locked to a common control line. This option makes the curve smooth but unequal on either side of the node.

**Nonsmooth at point** — When this option is selected, adjusting one of the green control handles does not affect the the paired handle at all, the nodes have separate control lines. This this option allows you to make a sharp point in the curve.

NOTE: These controls will have no effect if the “keep Curve Smooth” option is turned off in the curve properties. “Curve Properties” on page 168.



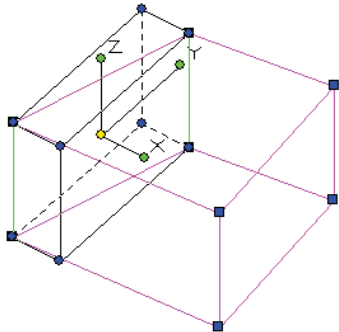
The effect of selecting a different curvature only appears after you begin moving an affected handle.

**Selector Mode in Edit Tool**

You can use the Selection tool functions to edit nodes by turning on the Selection Mode. Selection mode is available from the local menu or from the Inspector bar.



While in this mode you can select multiple nodes and move them, rotate them, and scale their collective size just as you would any other object using the selector.

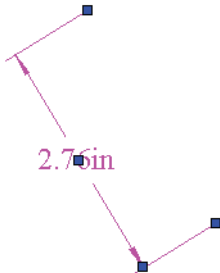


NOTE: The selector type (2D or 3D) that will be used in Edit tool is dependant upon the type that is current for the selector.

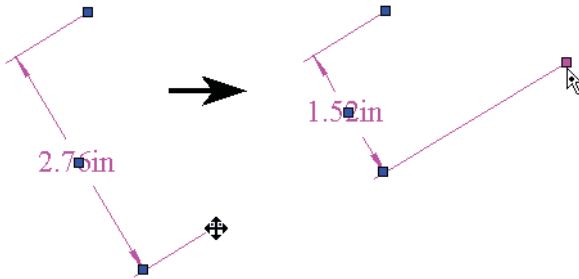
**Editing Dimensions**

For details on creating dimensions, see “Dimensions” on page 331.

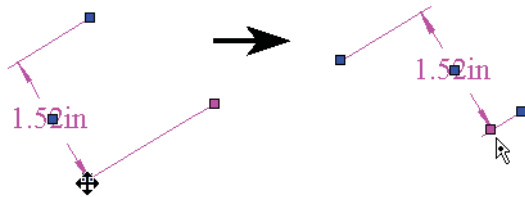
When using the **Edit Tool** on a dimension, there are several nodes for editing: one at each extension line, one on the dimension line itself, and one on the text. Each can be moved to change the dimension.



Move the node on an extension line to change its length and/or position. If this change leads to a new dimension value, the new value will appear after the node is moved.

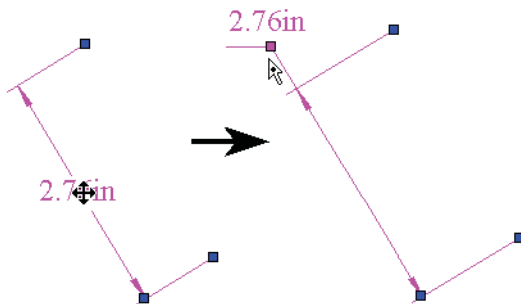


Move the node on the dimension line to change the position of the dimension text.

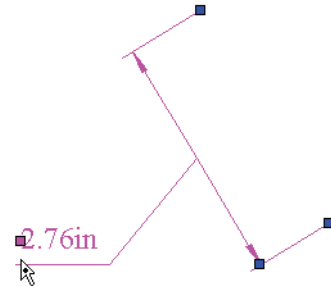


Move the node on the dimension text to move the text. How the text is moved depends on the **Text Movement** option set on the **Format** page of the dimension's **Properties** (see "Dimension Properties" on page 332.)

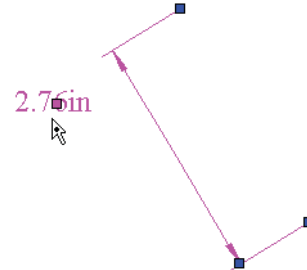
- **Keep Dimension Line with Text:** The dimension line will move with the text.



- **Move Text, Add Leader:** The moved dimension text will be attached to the existing dimension line by a leader.



- **Move Text, No Leader:** Only the dimension text will move.



## Modifying Geometry

The **Modify** tools are generally used to change the shapes of existing objects. These tools do not change physical properties such as layers or line widths; rather they modify the actual geometry.

You can display the **Modify** toolbar by right-clicking in any toolbar area and selecting **Modify**.



Some **Modify** tools require input of two or more objects. If you are working in 3D, all 2D objects selected for modification must lie on the same workplane (see "Workplanes" on page 362). To change an object's workplane, see "Place on WorkPlane" on page 371.

## Object Trim



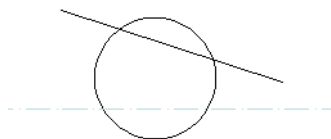
Uses one or more cutting edges to trim objects.

---

**NOTE:** *Complex objects such as dimensions and text must be exploded before they can be trimmed. Text must be exploded twice - first into groups of polylines, then into polylines.*

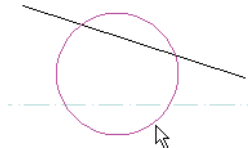
---

The example for **Object Trim** contains three objects: a circle, line, and construction line.



### Using One Cutting Edge

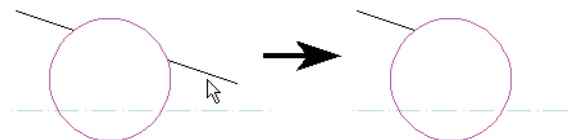
1. Select the object to be used as the cutting edge.



2. Select the object to trim, clicking on the portion you want deleted.



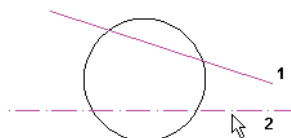
3. Continue selecting objects to trim, if needed.



4. Press Esc if you want to select another cutting edge. Select **Cancel** from the local menu or Inspector Bar to exit.

### Using Multiple Cutting Edges

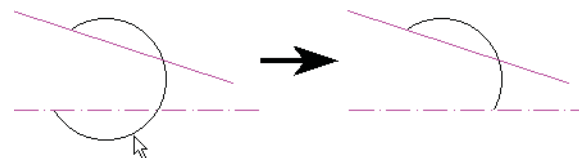
1. Select the objects to be used as the cutting edge. Use Shift to select multiple objects.



2. Select the object to trim, clicking on the portion you want deleted. One or both cutting edges will be used, depending on what is selected.



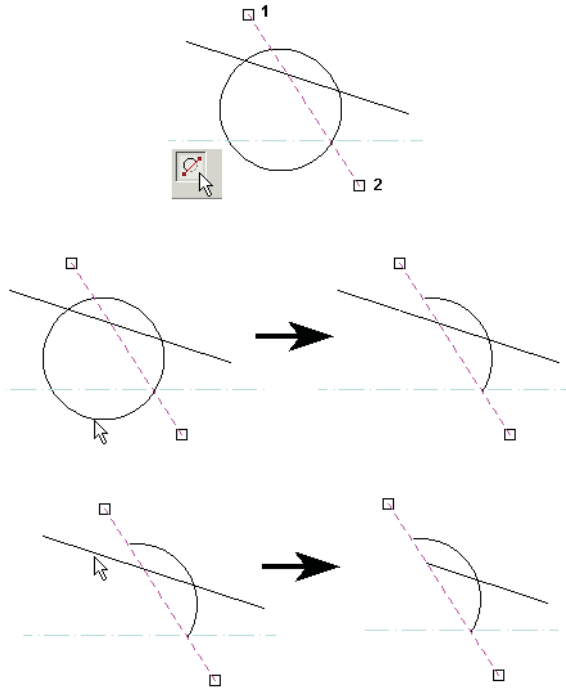
3. Continue selecting objects to trim, if needed.



4. Press Esc if you want to select another cutting edge. Select **Cancel** from the local menu or Inspector Bar to exit.

Local menu option:

**Trim by Line:** Enables you to define the cutting edge by selecting two points.



## Stretch

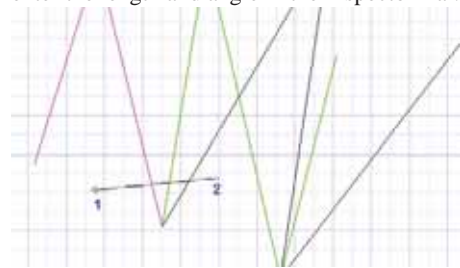


Moves a group of nodes of one or more objects by defining a movement vector.

1. Drag a selection window from right to left around the elements you want to stretch. Be careful to only select nodes that you want to move. If you select an entire segment (line or arc) or an entire object it will be moved and not stretched..



2. Right click and select Finish
3. Select two points to define the movement vector, or enter the length and angle in the Inspector Bar.



The selected nodes move by the defined vector.

Local menu option:

**Multiple:** If this option is selected the stretch tool will continue to operate until you press ESC or select another tool.

**PickAdd:** If this option is on you can add objects to the selection simply by clicking on them, and you have to hold down Shift to deselect objects. If this option is off you have to hold down the Shift key or de-select to select additional objects.

**Displacement:** If this option is selected the node or object will be displaced by the UCS X and Y value of the point you select. For example if you click on the coordinate X= -10 and Y= 5 the displacement will be -10 in the X axis and 5 in the Y axis.

## Split

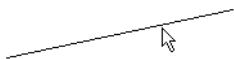


Divides an object into two separate objects. This tool works on any single-line or double-line object, as well as on circles, arcs, and curves.

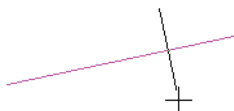
You can split by using the cursor, or by specifying a ratio in the Inspector Bar. The ratio is the length of the first resulting segment to the length of the second resulting segment. The first segment starts at the start point of the object, which depends on how the object was created.

### Splitting Lines and Double Lines

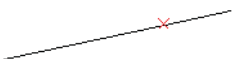
1. Select the line or line segment you want to split.



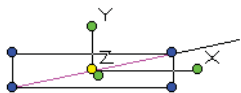
2. Position the split indicator where you want the split, or enter the split ratio in the Inspector Bar.



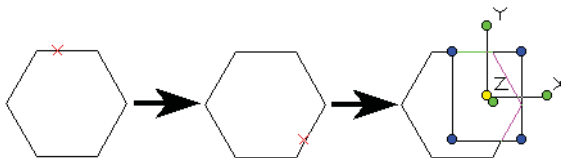
3. Click to locate the split. A temporary red X indicates the location.



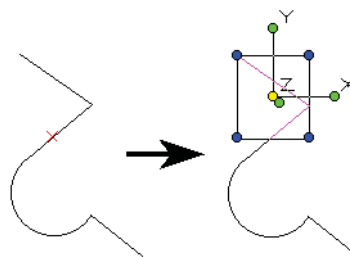
You can verify that the line is now two separate objects by selecting one of the lines.



To split a closed object such as a rectangle or polygon, you need to create at least two splits. For each one, click on the segment you want to split.

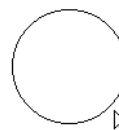


When splitting a polyline, you place the split on one segment (line or arc), but the entire polyline is split at the selected point.

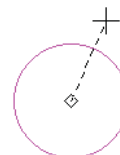


### Splitting Arcs and Circles

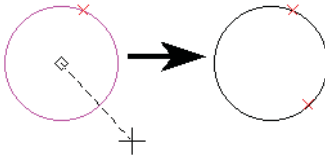
1. Select the circle, arc, or arc segment you want to split.



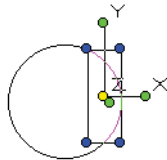
2. Position the split indicator where you want the split, or enter the split ratio in the Inspector Bar.



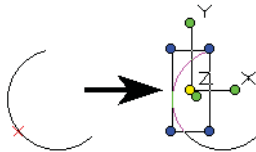
- Click to locate the split. A temporary red X indicates the location. For a circle, indicate a second split point.



You can verify that the circle is now two separate objects by selecting one of the arcs.



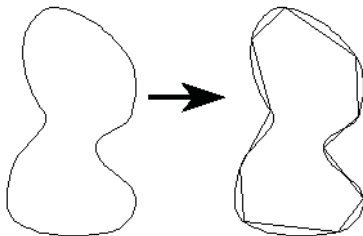
For arcs, you only need to define one split point.



## Splitting Curves

It is helpful to display the curve's frames before splitting it. This is because the split actually takes place on the line segments of the frame.

- To display the frame, open the **Properties** window to the **Curve** page and check **Show Frame**.



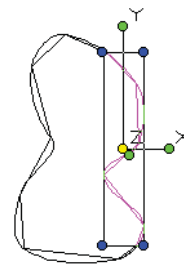
- Select the curve. Position the split indicator where you want the split, or enter the split ratio in the Inspector Bar.



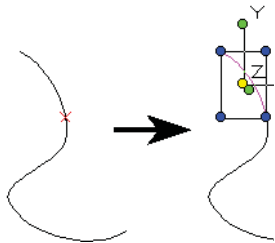
- Click to locate the split. A temporary red X indicates the location. For a closed curve, select a second split point.



You can verify that the curve is now two separate objects by selecting one of the segments.



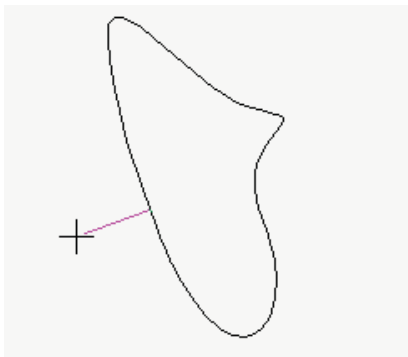
For open curves, you only need to define one split point.



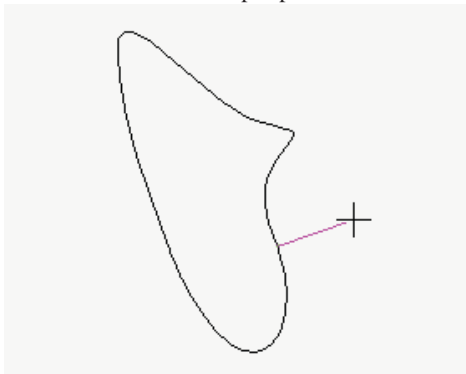
### Splitting 3D Curves

It is helpful to display the curve's frames before splitting it. This is because the split actually takes place on the line segments of the frame.

1. Select the curve. Position the split indicator where you want the split, or enter the split ratio in the Inspector Bar.



2. Click to locate the split. For a closed curve, You may want to select a second split point.



3. You can then move the split off section.

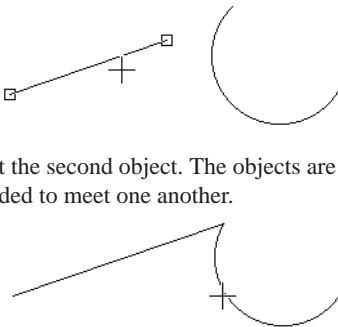


### Meet 2 Lines

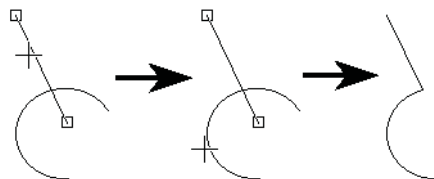


Extends two lines, double lines, walls or arcs so that their endpoints meet.

1. Select the first object.
2. Select the second object. The objects are trimmed or extended to meet one another.



If an object is to be trimmed, select the portion of the object you want to remain.



Local menu options:



The local menu options apply to double lines. See "Double Line Options" on page 245.

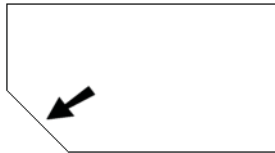
---

*NOTE: You can use Meet 2 Lines on splines and Beziers, however the extension will be straight element of the curve, not curved continuations as they are with arcs. 3D Polylines, and splines can also be used, but the terminal resulting geometry will be placed on the concurrent workplane of the two objects used.*

---

## Chamfer

Connects two lines (single or double lines) with a beveled corner. The selected objects do not have to intersect, and they can overlap.

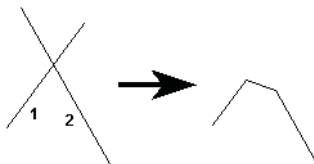



---

*NOTE: The resulting objects are separate line segments, unless the **Polyline** option is used, in which case the result is a polyline.*

---

If the objects overlap, they will be trimmed to create the chamfer. Be sure to select the side of the object you want to remain.



There are three methods for creating chamfers: **Distance / Distance**, **Distance / Angle**, and **Length / Angle**. For all methods, the same local menu options are available.

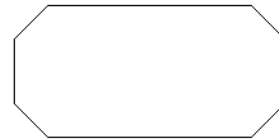
### Local menu options:

**Polyline:** Chamfers all corners of a polygon or polyline in one step.

1. Enter the chamfer distances, angles, or length in the Inspector Bar.
2. Select the multi-segmented object you want to chamfer.
3. Select **Polyline** from the local menu or Inspector Bar.



The chamfer is applied to all corners. The resultant object is one polyline.




---

**WARNING:** Be careful when using the **Polyline** option with different chamfer distances. The results will not be symmetric.

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The other local menu options apply to double lines. See "Double Line Options" on page 245.

Chamfer Distance / Distance

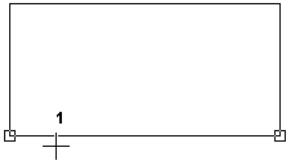


The chamfer is created by specifying its length along each line.

- 1. Enter both chamfer distances in the Inspector Bar. These are the distances along the lines that are to be chamfered. Distance A will be applied to the first line you select.

Dist.A	Dist.B
1 in	0.5

- 2. Select the first line to chamfer.



- 3. Select the adjacent line. The corner is chamfered.



Chamfer Distance / Angle

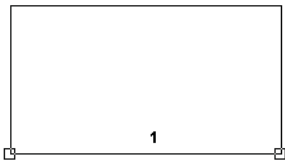


The chamfer is created by specifying its distance along one line and its angle from the second line.

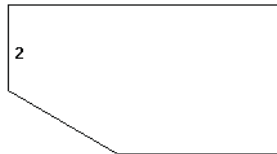
- 1. Enter the distance and angle in the Inspector Bar. The distance is measured along the first line you select.

Distance	Angle
1 in	30

- 2. Select the first line to chamfer.



- 3. Select the adjacent line. The corner is chamfered.



Chamfer Length / Angle

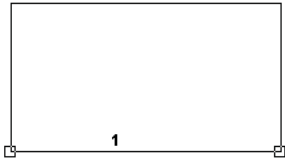


The chamfer is created by specifying the total chamfer length (as opposed to the distance along the lines) and chamfer angle.

- 1. Enter the length and angle in the Inspector Bar. The length is the total length of the diagonal chamfer line.

Length	Angle
1 in	60

- 2. Select the first line to chamfer.



- 3. Select the adjacent line. The corner is chamfered.



## Line Length

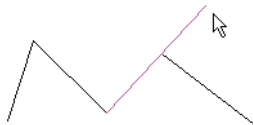


Shrinks or extends a line or line segment from either endpoint. Double lines and multi lines can also be used. If you use this tool on a polyline or polygon, any adjacent segment will move to accommodate the new endpoint.

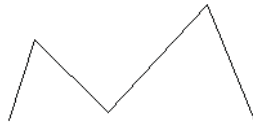
1. Select the line or line segment close to the endpoint you want to move.



2. Move the cursor to set the new length, or enter the length or delta in the Inspector Bar. Negative values can be used, thereby extending the line in the opposite direction.



If another segment meets the moved endpoint, it is changed as well.




---

**NOTE:** You can use *Line Length* on splines and Beziers, however the extension will be straight element of the curve, not curved continuations as they are with arcs. 3D Polylines, and splines can also be used, but the resulting geometry will be derived relative to the objects workplane.

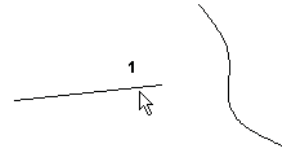
---

## Shrink / Extend Line

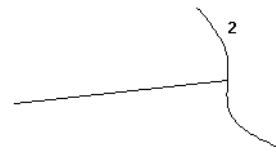


Trims or lengthens a line so that it meets another object. Double lines and multi lines can also be used.

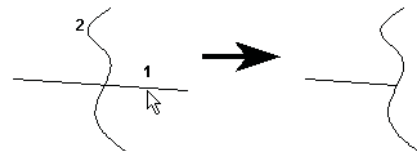
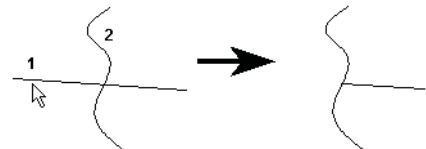
1. Select the line you want to shrink or extend. Click near the endpoint you want to move.



2. Select the object that the line is to meet. The endpoint of the line meets the selected object.



When shrinking a line, be sure to click near the endpoint that will move, otherwise the wrong part of the line will be deleted.



Local menu option:

**Cleanup:** Relevant for double lines, makes the cutlines invisible, forming a clean intersection. See "Double Line Options" on page 245.

---

NOTE: You can use *Shrink / Extend Line* on *splines and Beziers*, however the extension will be straight element of the curve, not curved continuations as they are with arcs.

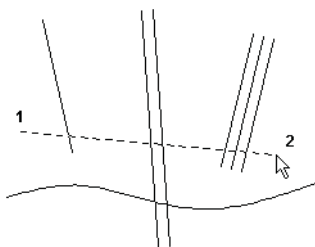
---

## Multi Shrink / Extend Line

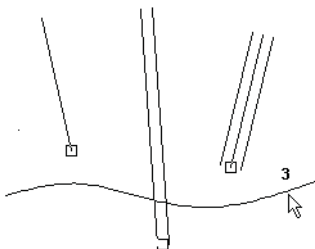


Similar to **Shrink / Extend Line**, trims or lengthens a group of line so that they meet another object. Double lines and multi lines can also be used.

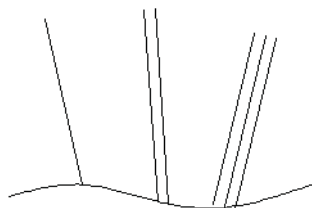
1. Select two points to draw a temporary “fence” that passes over all of the lines you want to shrink or extend. Be sure the fence intersects each line closest to the endpoints you want to move.



2. The endpoints that will move are indicated by small squares. Select the object that the lines are to meet.



3. The endpoints of the lines meet the selected object.




---

NOTE: You can use *Multi Shrink / Extend Line* on *splines and Beziers*, however the extension will be straight element of the curve, not curved continuations as they are with arcs.

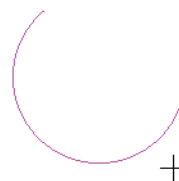
---

## Arc Complement



Reverses the start and end angles of an arc. In other words, the arc will become the portion of the original circle that was cut.

1. Select the arc you want to modify.



2. Click anywhere to create the arc complement.




---

TIP: When using this tool on arcs that are the result of exploding a polyline, or created as the result of using the **Arc Divide** option of the **Edit Tool**, this tool may select more than one arc at a time. If this happens, move the arcs from each other.

---

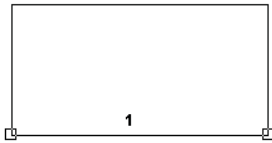
## Fillet



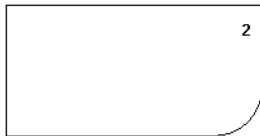
Connects two lines (either single or double lines) or arcs (except elliptical arcs) with a smoothly fitted arc. The filleted objects do not have to intersect, and they can overlap. The result is a rounded corner.

NOTE: You can also fillet corners with the **Edit Tool** (see "Filleting Two Line Segments" on page 224). To apply fillets to a 3D polyline, see "Bolt" on page 387.

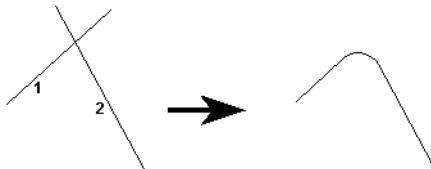
1. Enter the fillet radius in the Inspector Bar.
2. Select the first line or arc to fillet.



3. Select the adjacent line or arc. The corner is rounded.



If the objects overlap, they will be trimmed to create the fillet. Be sure to select the side of the object you want to remain.



### Local menu options:

**Polyline:** Fillets all corners of a polygon or polyline in one step.

1. Enter the fillet radius in the Inspector Bar.
2. Select the multi-segmented object you want to fillet.
3. Select **Polyline** from the local menu or Inspector Bar.



The fillet is applied to all corners.



**To Lines and Arcs:** The resulting object is broken into lines and arcs. (The default, **To Polyline**, leaves the filleted object as a single object, even if the objects were originally separate.)



In **Select** mode, you can verify that the fillets are separate objects.



The other local menu options apply to double lines. See "Double Line Options" on page 245.

NOTE: You can use **Fillet** on splines and Beziers, however the extension will be straight element of the curve, not curved continuations as they are with arcs.

---

**NOTE:** You can use *Fillet* to change an arc on a polyline, but only on arcs that are between two straight segments. Simply select the two straight segments and the arc will be adjusted to the radius of the fillet.

---

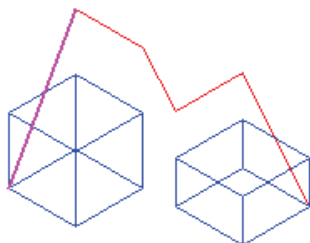
## Fillet3D

Available in TurboCAD Pro, and Platinum Only

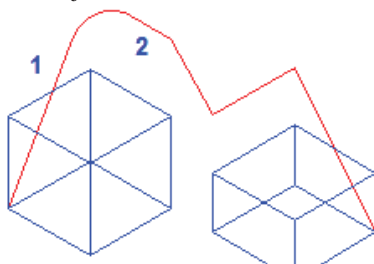


Creates a fillet on a 3D polyline

1. Enter the fillet radius in the Inspector Bar.
2. Select the first line or arc to fillet.



3. Select the adjacent line. The corner is rounded.




---

**NOTE:** You can re-fillet to the same two segments with a different radius and the fillet will be adjusted.

---

Local menu options:

**Polyline:** Fillets all corners of a polygon or polyline in one step.

1. Enter the fillet radius in the Inspector Bar.
2. Select the multi-segmented object you want to fillet.
3. Select **Polyline** from the local menu or Inspector Bar.



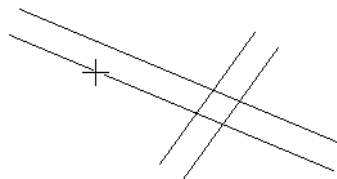
The fillet is applied to all corners.

## T-Meet 2 Double Lines

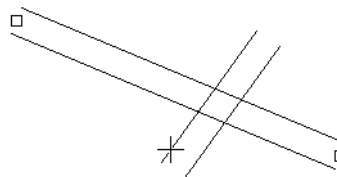


Forms a T intersection of two double lines. The first double line selected will shrink or extend to meet the intersection.

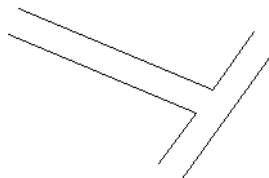
1. Select the double line that will be the stem of the T. Click on the portion you want to remain.



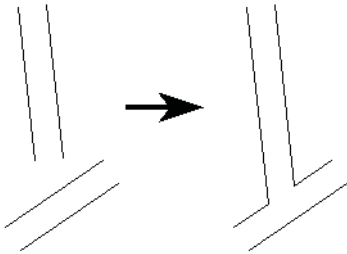
2. Select the line that will be the top of the T.



The T intersection is created.



If the stem of the T does not meet the other line, it will be extended to create the intersection.



#### Local menu options:

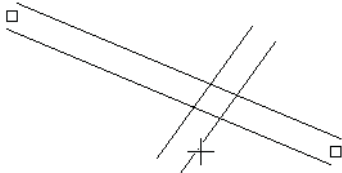
See "Double Line Options" on page 245.

### Intersect 2 Double Lines

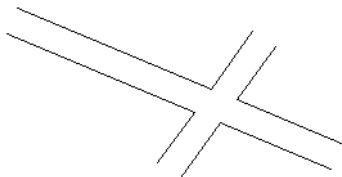


Forms an X intersection of two double lines, thereby cleaning up the intersection.

1. Select both lines whose intersection you want to create.



2. The X intersection is created.



#### Local menu options:

See "Double Line Options" on page 245.

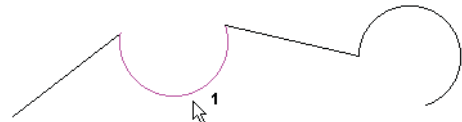
### Join Polyline



Connects two or more separate objects, connected at their endpoints, into a single-object polyline. Objects that overlap cannot be joined.

**NOTE:** If you want to create a polyline from a **partial** chain of segments, or if you have overlapping objects, see "Chain Polyline" on page 244.

1. Select the first object you wish to include in the polyline. Selection order is not important.

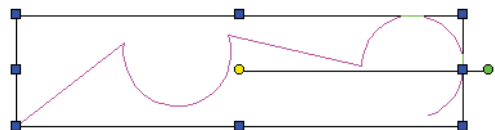


2. One by one, select the remaining objects to include.



3. Select **Finish** from the local menu or Inspector Bar.

The objects now comprise one object, as you can see in **Select** mode.



You can also first select the objects, then activate **Join Polyline** and immediately select **Finish**.

#### Local menu options:

**Delete Original Objects:** Removes the original segments, leaving only the polyline.



**Auto joining:** Automatically selects all objects connected to the selected object. This option must be active before selecting any segments. You can select any object in the chain, not only the first or last object. If more than one segment branches from an endpoint, the chain will stop.



**3D Polyline:** Enables you to include objects located on different workplanes.



## Chain Polyline



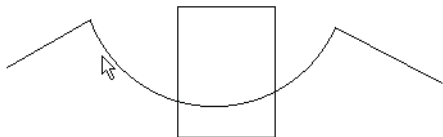
Connects intersecting objects or portions of objects into a single polyline. A chain of connected objects is created, starting from a selected object and proceeding in a specified direction.

---

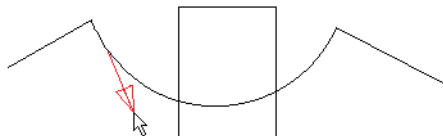
**NOTE:** The difference between this tool and the **Auto Joining** option of **Join Polyline** (see "Join Polyline" on page 243) is that **Chain Polyline** enables you to create a partial chain, and to work with overlapping objects.

---

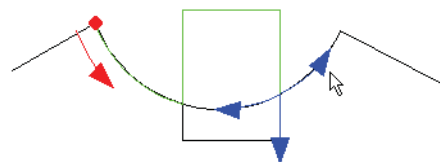
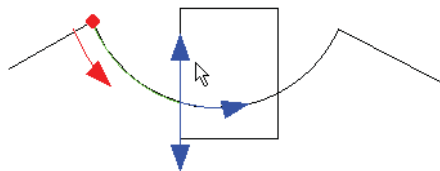
1. Select the object from which the chain will start.



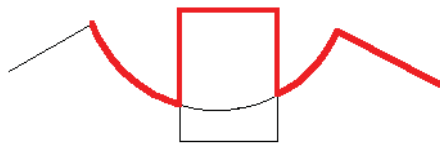
2. Set the direction in which the chain will proceed.



3. If overlaps are encountered, you must specify the path to continue the chain.



4. The chain proceeds until it reaches the last object, and the polyline is created.



If you do not want the chain to proceed all the way to the last object, you can select **Finish** from the local menu or Inspector Bar at any time.

---

**NOTE:** The original objects remain in the drawing.

---

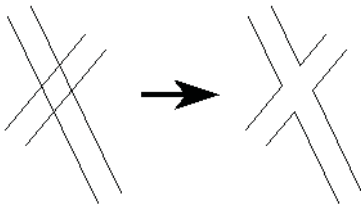


## Double Line Options

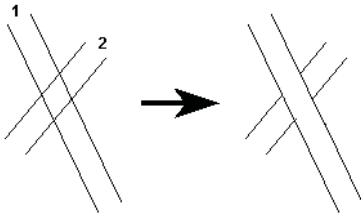
You can specify how to create intersections of double lines, and whether cut lines and gaps will occur at the intersections.

For example: if you form a clean T-intersection of two double lines at an angle other than 90 degrees, the end of one of the first line will be cut at an angle, and the second double line will have a gap to accommodate the first double line.

**Cleanup:** Forms a clean intersection.



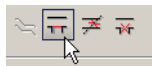
If you deselect **Cleanup**, the first line selected will remain unbroken.



**Drop CutLines:** Removes the cut lines in the first line, to even out the end of the line.



**Drop Gaps:** Fills in the gap in the second line.



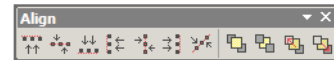
## Formatting Objects

This section covers tools for setting the appearance and format of drawing objects and other objects.

### Aligning Objects

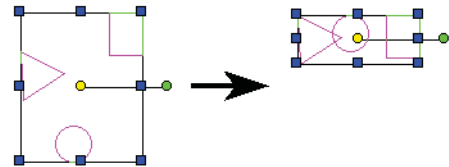
Aligns all currently selected objects, relative to the bounding box of the selection.

You can display the **Align** toolbar by right-clicking in any toolbar area and selecting **Align**.

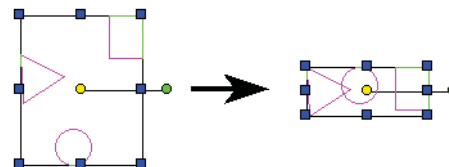


1. Select the objects you want to align.
2. From the **Align** menu (or the **Align** toolbar), select the type of alignment.

For example, **Top** moves the objects to the top of the bounding box.



**Middle** aligns the objects horizontally along the middle of the bounding box (as opposed to **Center**, which aligns vertically).



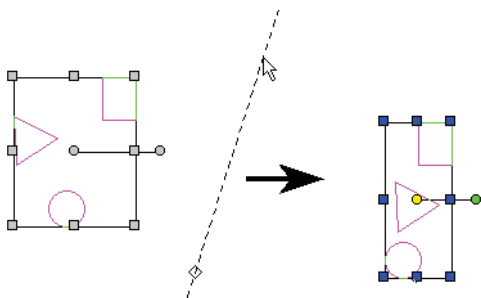
## Align Along Line

Align objects along a line.

1. Select the objects you want to align.
2. Select **Along Line**, or click the icon in the **Align** toolbar.



3. Define the location of the alignment line by selecting one point, then select a second point to set the angle. You can also enter the angle in the Inspector Bar.

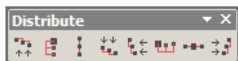


## Distributing Objects

Distributes all currently selected objects, relative to a specified location on the objects (top, left, center, etc.). There must be at least three objects selected for this function to be available.

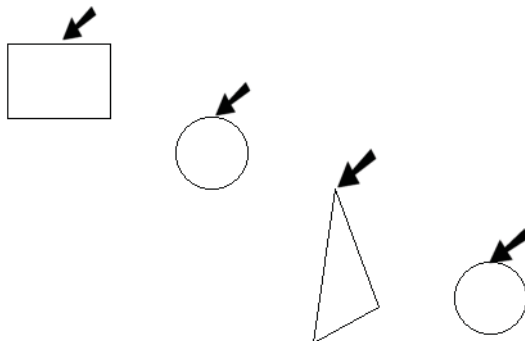
Objects are distributed between the two most extreme objects, i.e. the farthest left and the farthest right, or highest and lowest. Object order is maintained (top to bottom or left to right).

You can display the **Distribute** toolbar by right-clicking in any toolbar area and selecting **Distribute**.

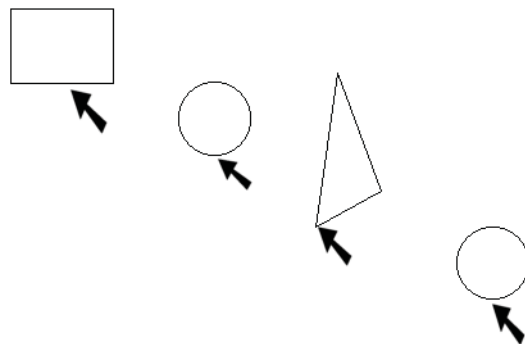


1. Select the objects you want to distribute.
2. From the **Distribute** menu (or the **Distribute** toolbar), select the type of distribution.

For example, **Top** arranges the objects so that the tops of their bounding boxes are evenly spaced vertically.

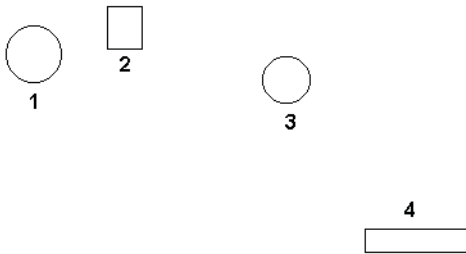


**Bottom** arranges the objects so that the bottoms of their bounding boxes are evenly spaced vertically.

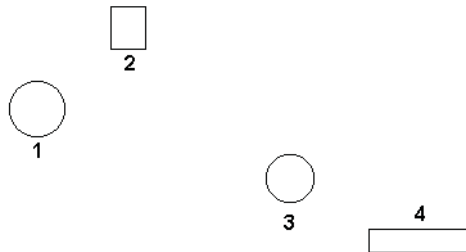


For vertical distribution, **Middle** means equal spacing between reference points, and **Space** means equal spacing between object bounding boxes. For horizontal distribution, **Center** means equal spacing between reference points, and **Distance** means equal spacing between object bounding boxes.

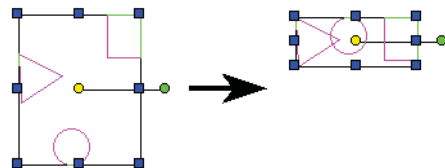
In this example, Object 2 is the highest, followed by 1, 3, and 4.



After vertical distribution, the highest and lowest objects (Objects 2 and 4) remain in place. Objects 1 and 3 are moved so that top-to-bottom order remains the same: 2, 1, 3, 4.



## Exploding Objects



**Hotkey:** Alt+Shift+E



Breaks an object, group, or block into its constituent parts. Each explosion is one level deep, to preserve the hierarchical structure of objects. Objects must be selected first in order to be exploded.

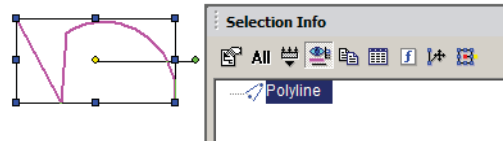
**NOTE:** The opposite of **Explode** is to join objects into a group (**Format / Create Group**). See "Groups" on page 289.

You can also use **Explode** to split solids that are difficult to split using 3D editing tools. Different types of objects are exploded differently, and use different explosion levels. Blocks and groups are first broken into their individual objects, then each object can then in turn be exploded. Polylines are broken into individual segments. Dimensions are exploded into lines, arrows, and text, then each of these can also be exploded. Double-line objects are exploded into single lines. You can explode a solid object twice to turn it into an exploded (node-editable) surface object.

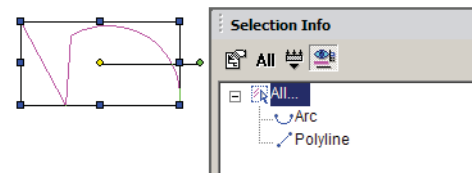
When using **Explode**, it can be helpful to open the Selection Info Palette (see "Selection Info Palette" on page 191), so that you can see the start condition and end results.

## Exploding Example - Polyline

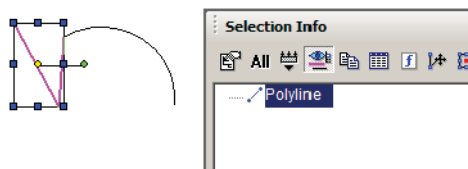
1. Create a multi-segmented polyline (with **Auto Add Constraints** off) and select it. In the Selection Info Palette, the object type is indicated as "Polyline."



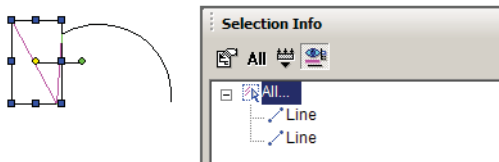
2. Select **Explode**, or click the **Explode** icon. The polyline is exploded into one arc and one small polyline.



3. Select the small polyline.



4. Explode it into its constituent parts - two lines.

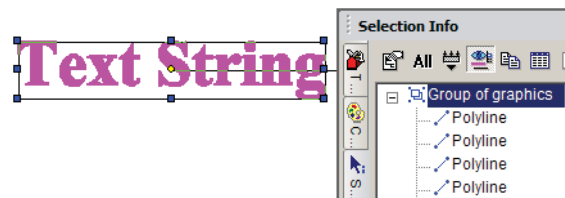


## Exploding Example - Text

1. Use the **Text** tool (see "Inserting Text" on page 326) to create a text object, and select it.



2. Select **Explode**, or click the **Explode** icon. The text is exploded into a Group of Graphics.



3. **Explode** again, and now the group consists of one polyline representing each character.

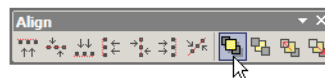


4. Select any character; it is identified as a polyline.



## Stacking Objects

These are also available on the **Align** toolbar, opened by right-clicking in any toolbar area and selecting **Align**.



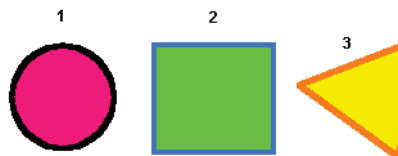
Objects are "stacked" when created - each object is created "on top" of the previous object. This sometimes has no visual effect, but stacking order can matter in the case of images and filled or hatched objects.

---

NOTE: *Stacking is not related to layers, and changing an object's layer does not effect its position in the object stack. See "Layers" on page 116.*

---

This example contains three filled objects that were created in the indicated order:

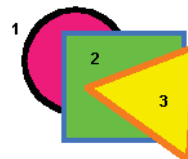



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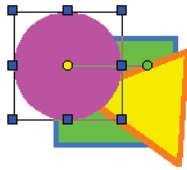
TIP: *You can press F6 to select the first object you created. Keep pressing F6 to scroll through the objects in their creation order. Press F7 to select the last object created, and repeat F7 to scroll backwards.*

---

If you move the objects so that they overlap, you can see that the first object is at the bottom of the stack; the last object is at the top.



1. Select the circle and select **Format / Bring to Front**.



2. The circle is now at the top. This, in effect, changes the creation order - the circle is now considered the most recent object. (You can verify this by pressing F7.)



3. Select the circle again and select **Format / Back One**. Rather than sending the circle to the bottom of the stack, it moves just one level down.



If you change the stacking order of multiple selected objects, the selection set will move as a whole. The relative stacking order of objects within the selection set is maintained.

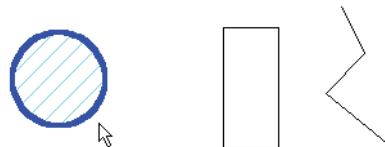
## Format Painter

*Available in TurboCAD Pro and Platinum only*

**Hotkey:** Ctrl+Shift+P

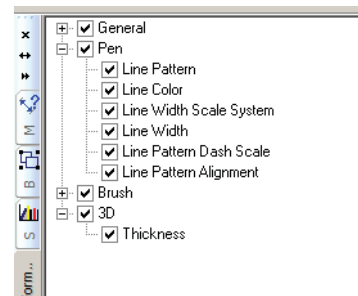
Takes properties from one object and applies (“paints”) them onto other objects. You can paint all properties or only selected ones.

1. Select the source object - the object with the properties you want to apply to other objects.

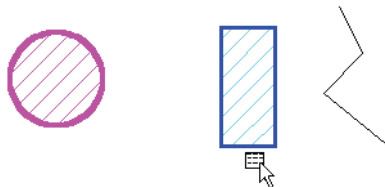


2. The Format Painter Palette appears in the palette area, listing all the properties that can be painted.

The list of properties depends on the selected object. For instance, a text object will have a category for **Text** properties; a sphere will have a category for **Sphere** properties.



3. By default, all properties are selected. If you want to remove any properties, uncheck the relevant box.
4. Select the object to which you want to paint the properties.



5. Select additional objects, if needed. Only the relevant properties will be applied. For example, the circle's **Brush** properties are not applied to the polyline (open objects cannot be filled).



6. Select **Cancel** from the local menu or Inspector Bar.



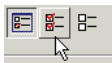
#### Local menu options:

**Make property values preset:** If this is selected the Make Property Value Presets dialog will appear. You can specify a name and type of preset that you want to create. The parameters of the preset will be generated from the object you selected, and the properties you specified in the Format Painter palette.

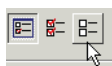
**Use Palette:** Displays or hides the Format Painter Palette.



**Mark All:** Selects all properties for painting. If the Format Painter Palette is not displayed, the properties will appear in a separate window.



**Unmark All:** Clears all properties for painting. If the Format Painter Palette is not displayed, the properties will appear in a separate window.



## **Regions**

*Available in TurboCAD Pro and Platinum only*



Combines 2D closed objects to form one object, called a region. Any overlap between selected objects is removed. The results are identical to using **2D Add**, except that this tool requires that the objects be selected first. When using **2D Add**, you select the objects during the operation.

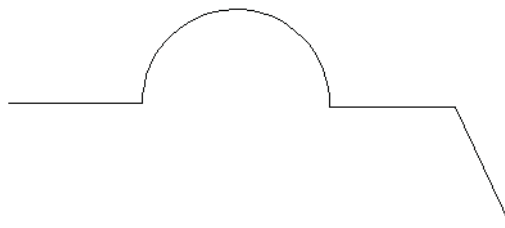
## **Convert to Curve**

**Menu:** Format / Convert to Curve

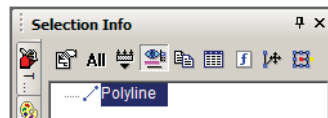


Converts a 2D object into a Bezier curve. Any single-line 2D object can be converted. Objects like double lines, dimensions, and text must be exploded first before converting.

1. Start with a 2D object such as a polyline. This example uses a polyline consisting of both linear and arc segments.

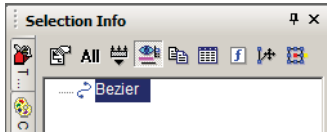


If you select the polyline and view the Selection Info Palette, you can see the object's type.

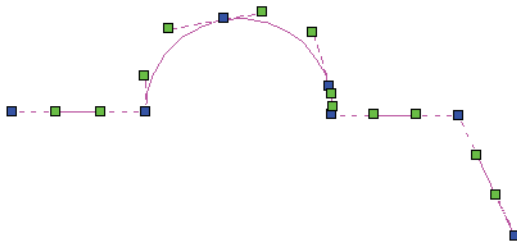


- To convert the polyline, select it and activate the **Convert to Curve** function. The **Tolerance** value sets the precision of the conversion; a small tolerance means high precision, and therefore more control points available for node editing.

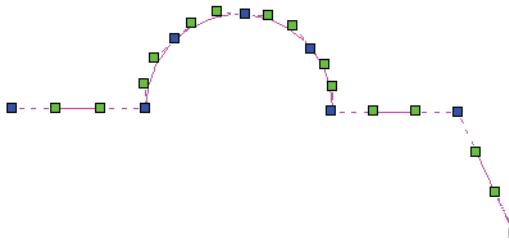
The polyline is replaced by a Bezier curve. This can be verified in the Selection Info Palette.



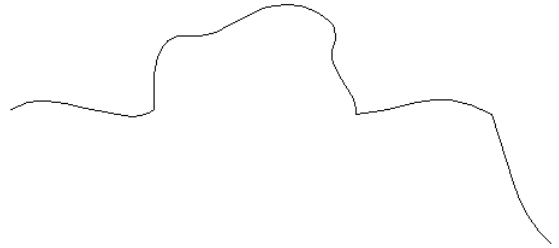
If you edit the curve with the **Edit Tool**, you can see the control points available for changing the curve's shape. (For details on Bezier options, see "Curve Properties" on page 168.) This curve used a low precision (Tolerance = 0.1).



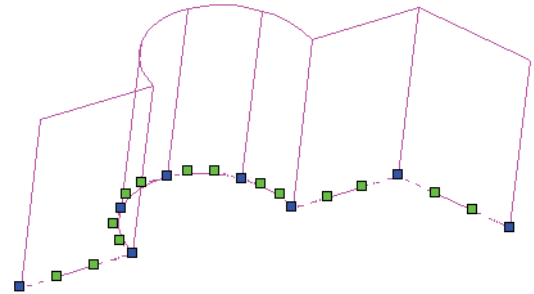
This curve used a higher precision (Tolerance = 0.001).



Nodes and control points can be moved to change the curve's shape.



**Convert to curve** also works on 2D objects that are converted into 3D by entering a thickness in the **3D** page of the **Properties**.



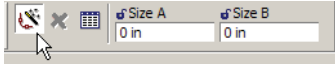
Sharp corners will remain sharp when two adjacent line segments cannot be approximated by an arc within the specified tolerance. This could happen when the vertex is too "sharp" which is defined by vertex angle, line lengths, and tolerance.

## Constraining Geometry

*Available in TurboCAD Pro and Platinum only*

Geometric constraints create positional relationships between 2D sketch objects. When used in conjunction with dimensional constraints (see "Constraining Dimensions" on page 269), you can easily control and update objects and dimensions.

NOTE: These tools are used for constraining geometry **after** it has been created. If you want to automatically constrain geometry **while** it is created, make sure **Auto Add Constraints** is active in the Inspector Bar.

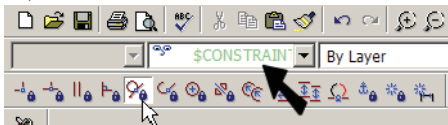


You can display the **Constraints** toolbar by right-clicking in any toolbar area and selecting **Constraints**.



Options for constraints can be found in **Program Setup**; see "Constraints" on page 66.

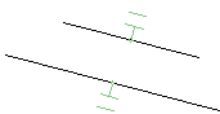
When you activate a constraint, the active layer switches to "CONSTRAINTS." (For details on layers, see "Layers" on page 116.)



In the Inspector Bar, the **Illuminate Suitable Entity** option is active by default. This means that only those objects that can be selected for the constraint type will be highlighted when the cursor passes over them. For example, if you are using the **Concentric** constraint, only arcs, circles, and ellipses will be highlighted and selectable.



When constraints are created, a constraint marker is attached to the constrained objects. This marker is on the CONSTRAINTS layer, in the layer's color. This example shows two lines that are constrained to be parallel.



Constraint markers can be selected like other objects. To remove a constraint, simply select and delete its marker.

NOTE: Constraints of selected objects can be viewed in the Selection Info palette. See "Selection Info: Constraints" on page 196.

## Horizontal / Vertical Constraint

Available in TurboCAD Pro and Platinum only



Makes a line horizontal or vertical.

1. Start with a line.
2. Activate the **Horizontal** or **Vertical** constraint.
3. Click the line, which becomes horizontal or vertical.

## Points Coincident Constraint

Available in TurboCAD Pro and Platinum only

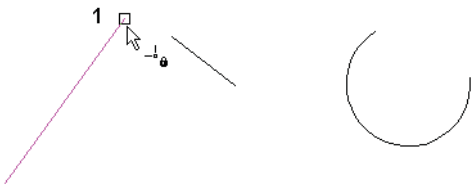


Moves a point or endpoint to another point.

1. Start with two lines and an arc.

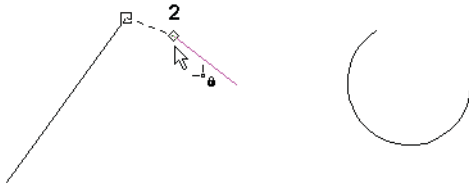


2. Activate the **Points Coincident** constraint, and click an endpoint of the first line.

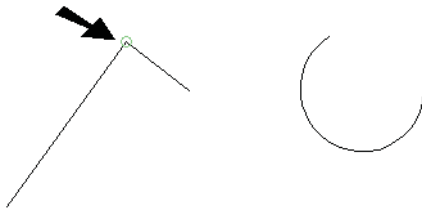




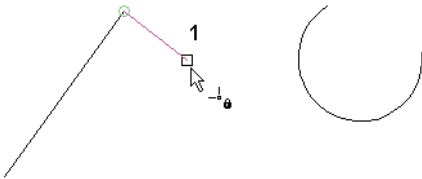
- Click an endpoint of the second line.



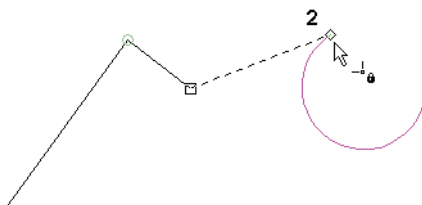
The first line is moved so that the two points are coincident. The length and orientation of the line is maintained, and a constraint symbol appears at the coincident point.



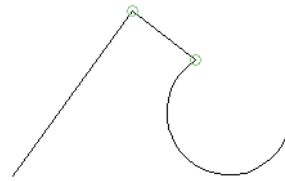
- Create another constraint between Point 1 . . .



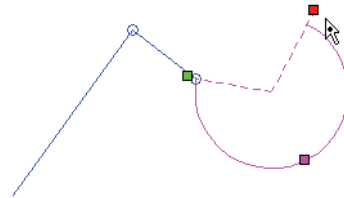
- . . . and Point 2 - an endpoint of the arc.



Both lines move, and their lengths and orientations are maintained.



- If you edit any of the objects, such as the arc in this case, the coincident constraint will be maintained, and the other objects will keep their size and orientation (when possible).



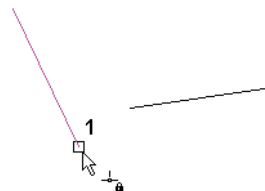
## Coincident Constraint

Available in TurboCAD Pro and Platinum only

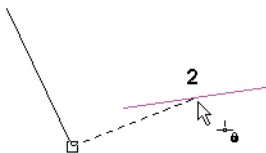


Makes a vertex lie on another vertex, or on another object.

- Start with two lines. Activate the **Coincident** constraint and select the vertex you want to constrain.



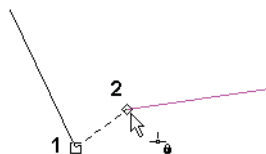
2. Select the line which you want the vertex to lie on.



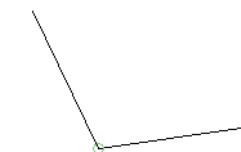
The vertex moves to the line, or to a point on the line's theoretical extension.



If you constrain the vertex to another vertex . . .



. . . the two vertices will meet.



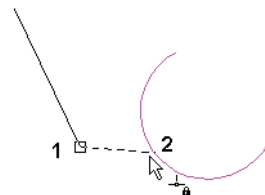
1. This constraint is similar when using arcs and circles. Start with a line and a circle, and constrain the line endpoint to the arc endpoint.



The endpoints now meet.



2. If you constrain the vertex to the arc itself . . .



. . . the line joins the arc at the tangent point.



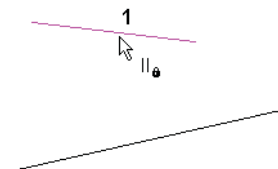
## Parallel Constraint

*Available in TurboCAD Pro and Platinum only*

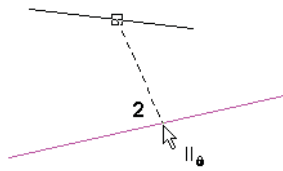


Makes the first selected line parallel to the second selected line.

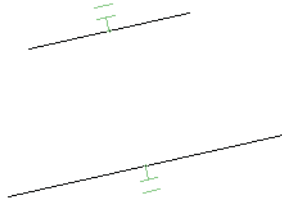
1. Start with two lines. Activate the **Parallel** constraint and select the line you want to be made parallel.



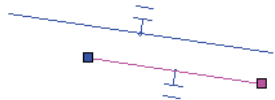
2. Select the line to which you want the first line to be parallel.



The lines are parallel, and the parallel constraint symbol is added to both lines.



3. If you edit either line, the parallel constraint will be maintained.



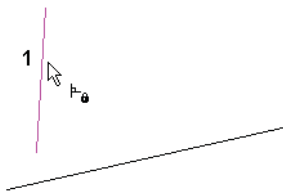
## Perpendicular Constraint

Available in TurboCAD Pro and Platinum only

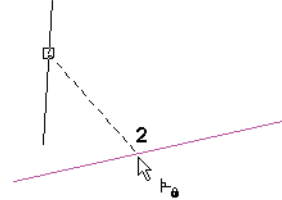


Makes the first selected line perpendicular to the second selected line.

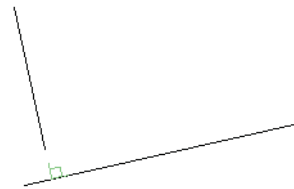
1. Start with two lines. Activate the **Perpendicular** constraint and select the line you want to be made perpendicular.



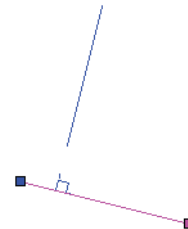
2. Select the line to which you want the first line to be perpendicular.



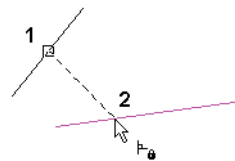
The lines are perpendicular, and the perpendicular constraint symbol is added at the point where the lines, or their extensions, meet.



3. If you edit either line, the perpendicular constraint will be maintained.



If you apply this constraint to lines that do not intersect . . .



... the perpendicular constraint symbol will appear along the extension of one or both lines.



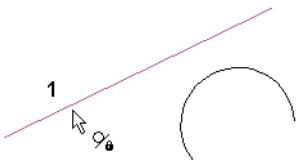
## Tangent Constraint

Available in TurboCAD Pro and Platinum only

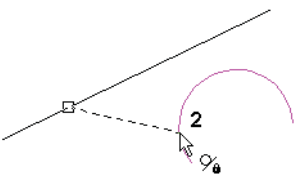


Makes an arc or circle tangent to another arc or circle, or to a line.

1. Start with a line and an arc/circle. Activate the **Tangent** constraint and select either object (when one object is a line, the selection order does not matter).



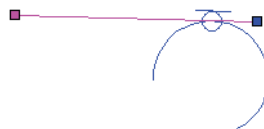
2. Select the other object.



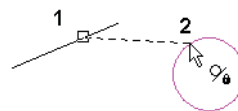
The line is moved, in an offset direction, so that it is tangent to the arc/circle. The tangent constraint symbol is added at the point of tangency.



3. If you edit either object, the tangent constraint will be maintained.



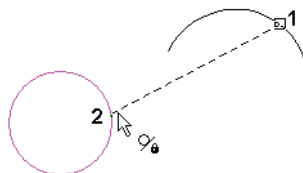
If you apply this constraint to a line whose offset will not meet the arc/circle ...



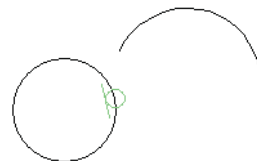
... the line is moved so that its extension is tangent to the circle.



You can also make two arcs/circles tangent to one another. In this case, the first selected object will move to meet the second selected object. If the arc does not meet the other arc/circle ...



... the arc is moved so that its extension is tangent to the arc/circle.



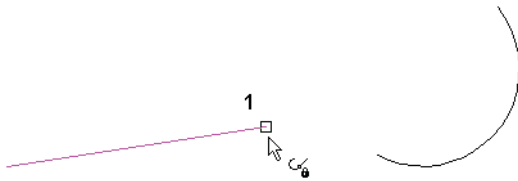
## Connect Constraint

Available in TurboCAD Pro and Platinum only

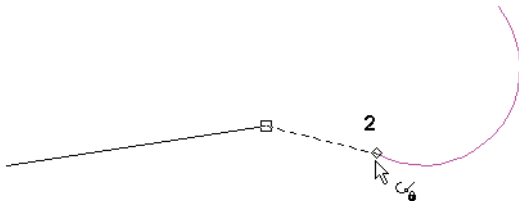


Moves, rotates, and trims arcs or lines so that the two selected endpoints will be coincident, and the objects will be tangent. If two lines are selected, they are connected with no regard to tangency, equivalent to the **Points Coincident** constraint.

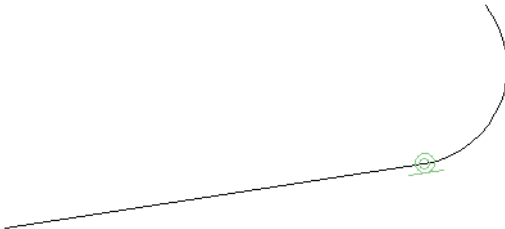
1. Start with a line and an arc/circle. Activate the **Connect** constraint and select the endpoint of the line you want to connect to the arc.



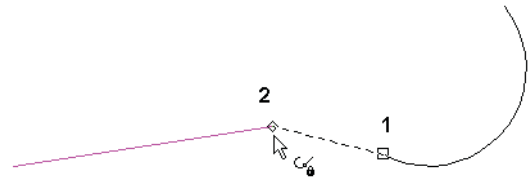
2. Select the endpoint of the arc to which the line will be connected.



The line is moved, in an offset direction, so that it is tangent to the arc/circle. The arc is trimmed as needed. The connect constraint symbol is added at the point of tangency.



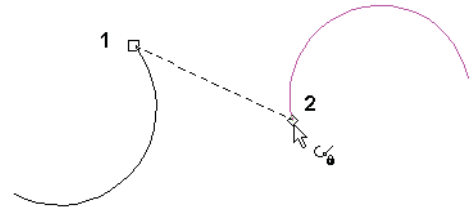
If you select the arc/circle first, and then the line . . .



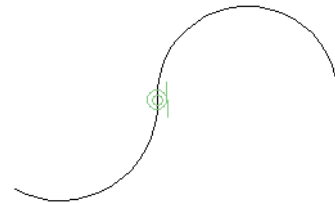
. . . the arc moves and the unselected line endpoint remains in place.



You can also select two circle/arcs. Select the arc you wish to move first . . .



. . .and the first arc moves to connect to the second arc.



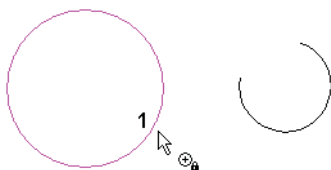
## Concentric Constraint

Available in TurboCAD Pro and Platinum only

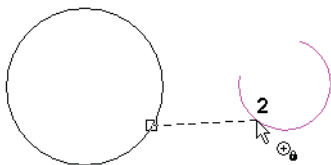


Makes two arcs/circles concentric - sharing the same center point. You can also use this tool to place a point or endpoint at the center of a circle or arc.

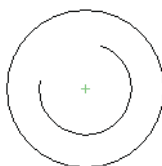
1. Start with two arcs/circles. Activate the **Concentric** constraint and select the arc/circle you want to make concentric. This object will move to the second.



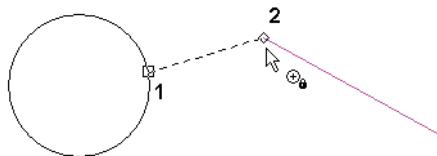
2. Select the second arc/circle.



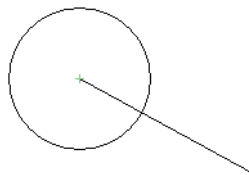
The first arc/circle moves, keeping its orientation, so that it is concentric with the second. The concentric constraint symbol (a plus sign) is added at the point of tangency.



You can also use this tool on a point or line endpoint. Select a circle/arc and the point or endpoint. . .



. . . and the circle/arc moves so that its center meets the endpoint or point.



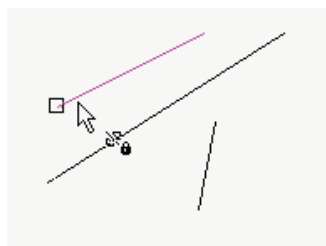
## Symmetric Constraint

Available in TurboCAD Pro and Platinum only

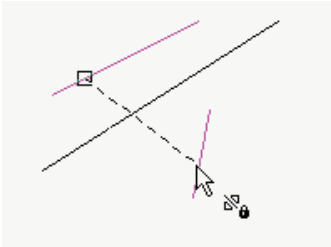


Makes two objects, or object endpoints, symmetric with respect to a symmetry axis.

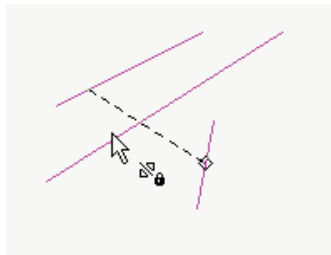
1. Start with three lines - one of which is the symmetry line. In some cases the symmetry line will move as a result of this constraint, so if you want it to remain in place, fix it with a constraint such as **Fix Geometry**.
2. Activate the **Symmetric** constraint and select the first line.



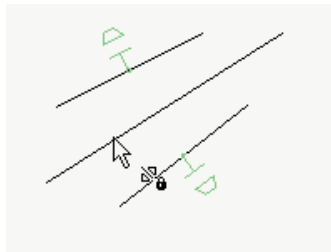
3. Select the second line.



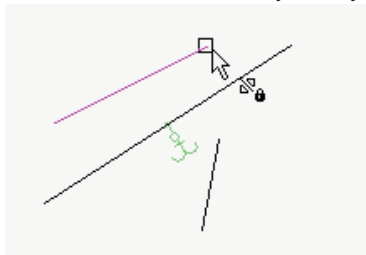
4. Finally, select the symmetry line.



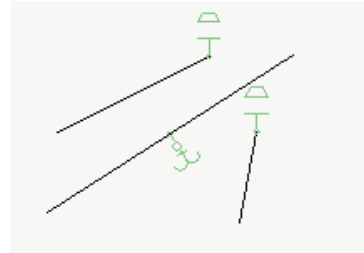
The first two lines are now symmetric with respect to the third.



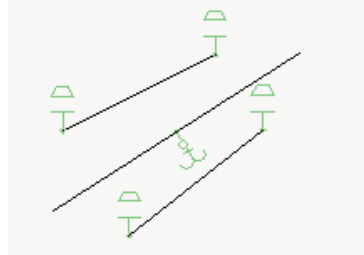
5. You can also use this constraint on endpoints. Select Vertices 1 and 2, then select the symmetry line.



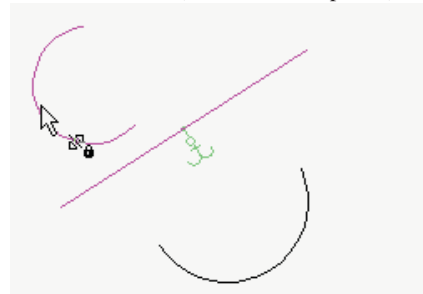
The lines themselves keep their orientation, but their endpoints are symmetric.



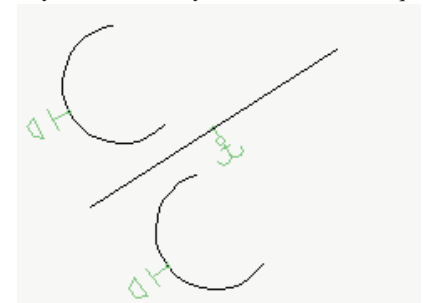
6. To make both lines completely symmetric, apply the **Symmetric** constraint to the other two endpoints.



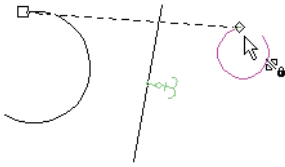
7. This constraint can also be used on circles and arcs. If you select two arcs (not at their endpoints) . . .



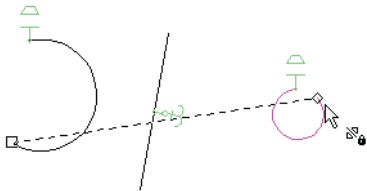
... they will become symmetric and have equal radii.



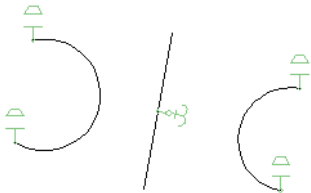
8. Like with lines, you can apply the **Symmetry** constraint at both sets of endpoints. First the upper endpoints . . .



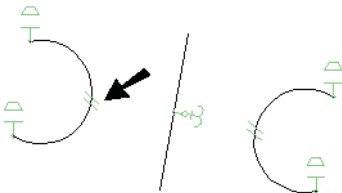
9. . . . then the lower endpoints.



The result is that both sets of endpoints are symmetric, but the orientations of the arcs are still different, because they have different radii.



10. You can make the arcs completely symmetric by adding an **Equal Radius** constraint.




---

NOTE: You could get the same results if you apply three **Symmetric** constraints - one to the arcs themselves, and one for both sets of endpoints.

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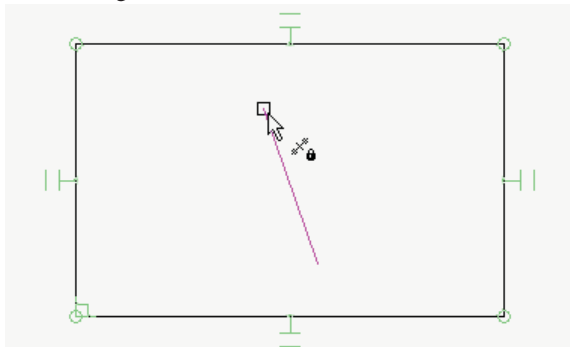
## Midpoint Constraint

Available in TurboCAD Pro and Platinum only

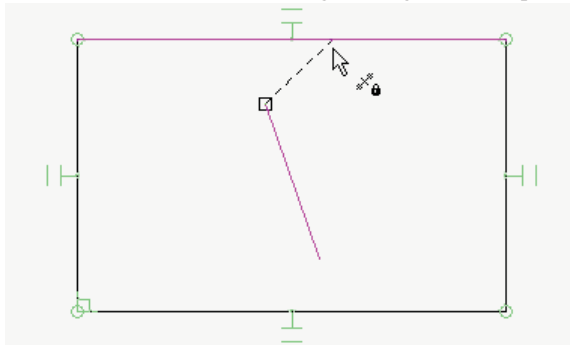


Align a point or the end of a line with the midpoint of a selected line.

1. Start with a line and a constrained rectangle. Activate the **Midpoint** constraint and select the end of the line line you want to align to the midpoint of the side of the rectangle.

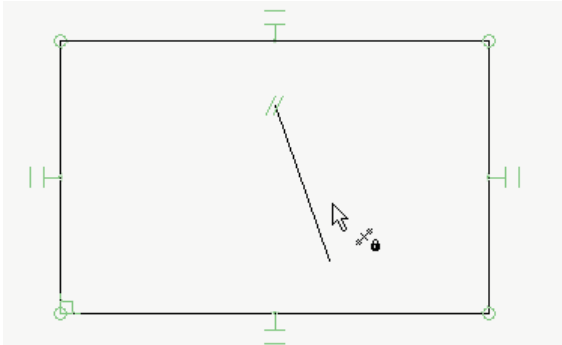


2. Select the side of the rectangle to align to its midpoint.

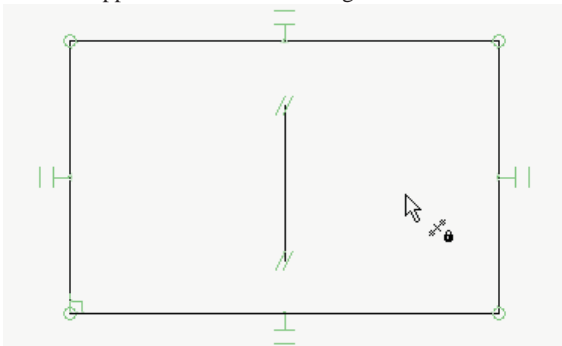




The end of the line aligns with the midpoint of the rectangle's side.



3. Repeat the procedure with the other end of the line and the opposite side of the rectangle.



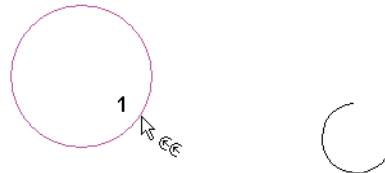
## Equal Radius Constraint

Available in TurboCAD Pro and Platinum only

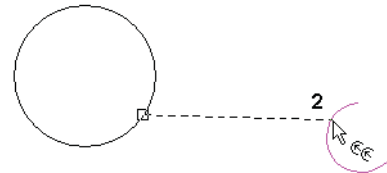


Makes the first selected circle/arc have the same radius as the second circle/arc.

1. Start with two circles/arcs. Activate the **Equal Radius** constraint and select the arc whose radius you want to change.



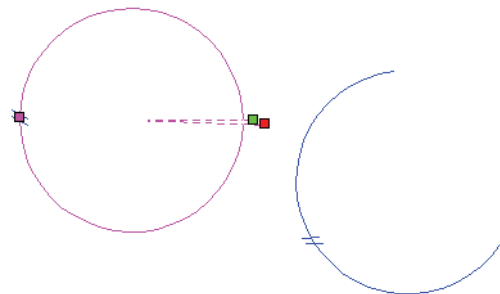
2. Select the arc whose radius you want to assign to the first arc.



The first arc's radius changes to match the second. The equal constraint (an equal sign) appears.



3. If you edit either arc, the equal radius constraint will be maintained.



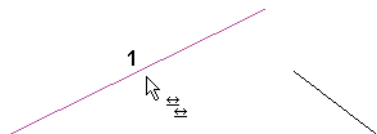
## Equal Length Constraint

Available in TurboCAD Pro and Platinum only

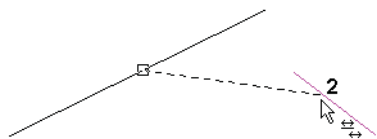


Makes the first selected line have the same length as the second line.

1. Start with two lines. Activate the **Equal Length** constraint and select the line whose length you want to change.



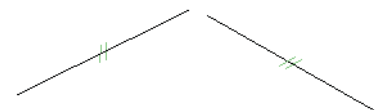
2. Select the line whose length you want to assign to the first line.



The first line's length changes to match the second. The equal constraint (an equal sign) appears.



3. If you edit either line, the equal length constraint will be maintained.



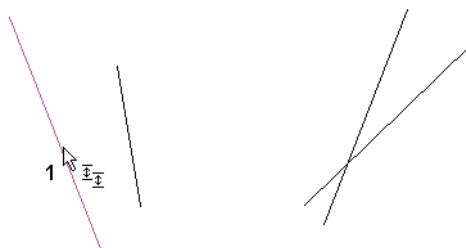
## Equal Distance Constraint

Available in TurboCAD Pro and Platinum only

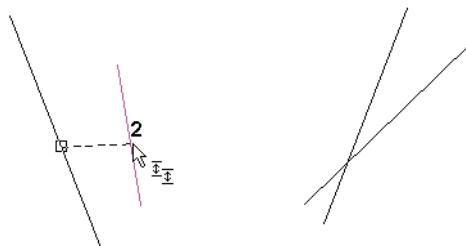


Sets the distance between two lines or points to the same value defined between two other lines or points. Sets of lines are also made parallel.

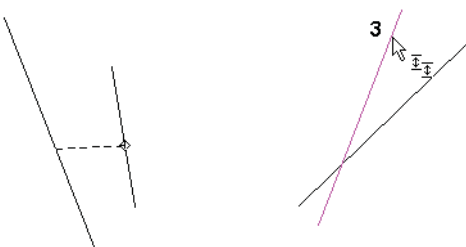
1. Start with four lines. Activate the **Equal Distance** constraint and select the first line in the first set.



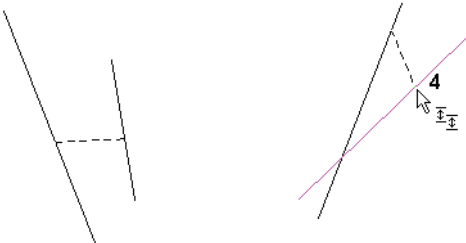
2. Select the second line in the first set. This establishes the distance for the next set.



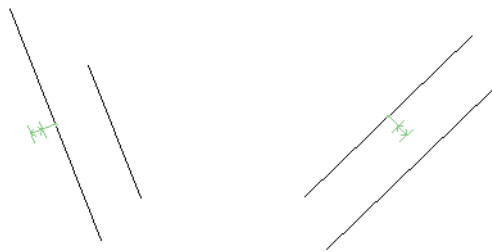
3. Select the first line in the second set.



4. Select the second line in the second set.



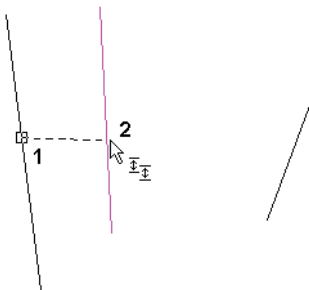
The first set and second set of lines are made parallel. The distance between the lines in the second set matches the distance in the first set.



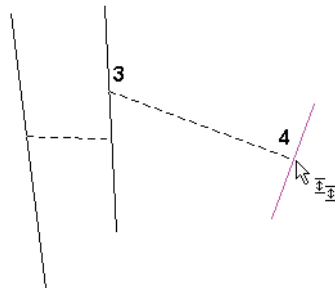
5. If you edit any line, the equal length constraint will be maintained.



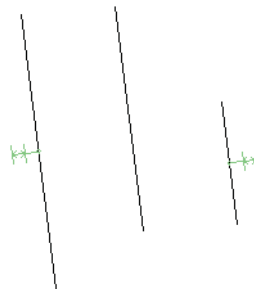
6. You can also apply this constraint to a row of lines or points. Select the first and second lines.



7. The third line is the same as the second, then select the fourth.



The result is three lines, parallel and separated by the same distance.



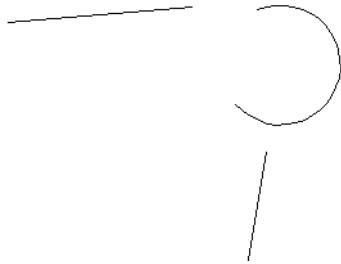
### Change Chirality Constraint

Available in TurboCAD Pro and Platinum only

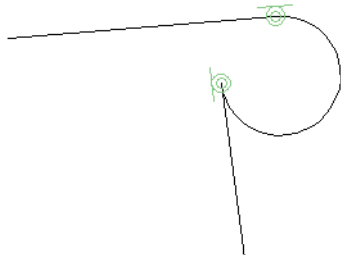


This constraint tool provides two options for correcting tangency of arcs. This tool is useful if arc tangency becomes skewed when applying other constraint, or when editing causes an arc to flip.

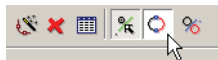
1. Start with two lines and an arc like these.



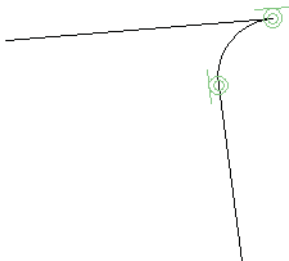
2. Apply a **Connect** constraint (see "Connect Constraint" on page 257) at both ends of the arc so that it is tangent to, and trimmed to, both lines. If you want a filleted corner, this result does not have the desired arc tangency.



3. The first option is **Flip Arc**, which changes the arc to its complement. Select this option . . .



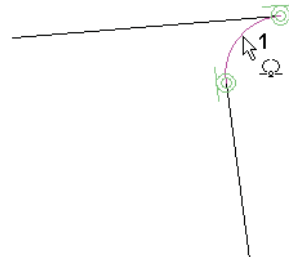
4. . . . and select the arc. The tangency along the vertical line is fixed, but is still incorrect along the horizontal line.



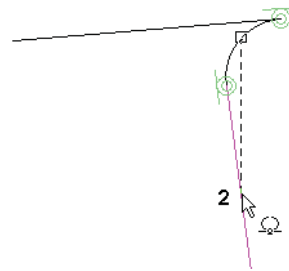
5. The other option is **Change Chirality**, which changes the arc orientation. Select this option . . .



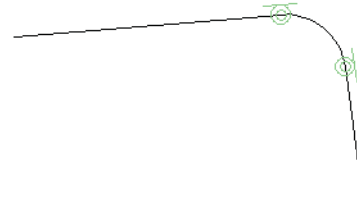
6. . . . and select the arc.



7. Then select the line whose tangency you want to keep - the vertical line.



This is the result - a filleted corner.



## Fix Geometry Constraint

Available in TurboCAD Pro and Platinum only



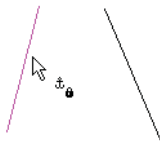
Locks an object or endpoint at its current coordinate, preventing the object or point from moving when constraints are applied.

---

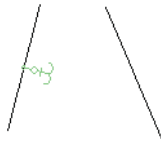
NOTE: The fixed object or point can be moved while editing geometry, but will not move when constraints are placed upon it.

---

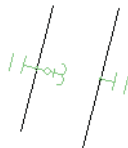
1. To fix an object, activate the **Fix Geometry** constraint and select the object you want to fix.



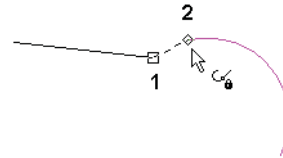
The anchor symbol indicates that the object is fixed.



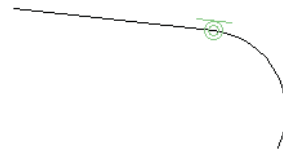
2. Now if you apply a constraint, such as **Parallel** in this case, the non-fixed line will move, regardless of the selection order.



3. For another example of this tool, start with a line and arc and apply a **Connect** constraint (see "Connect Constraint" on page 257).



4. The result is that the line maintains its orientation and is tangent and trimmed to the arc.



5. Undo this constraint, and apply a **Fix Geometry** constraint to the line's endpoint.



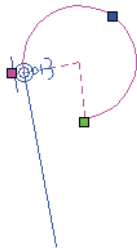
Now the anchor symbol is attached to the endpoint.



6. Now apply the **Connect** constraint. The endpoint stays fixed, and the orientations of the line and arc are modified.



7. If you edit the line or arc, such as changing start or end angles, the fixed point will not move, and the other objects will move or rotate accordingly.



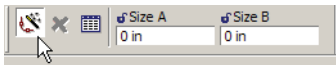
Auto Constraint

Available in TurboCAD Pro and Platinum only

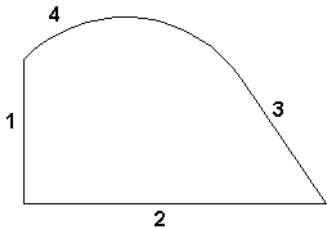


Automatically applies multiple constraints to a selected set of objects. You can use this tool on compound objects like polygons and polylines; these objects are exploded before constraint are applied. This is a useful tool for applying a fast set of constraints on files created in previous versions of TurboCAD.

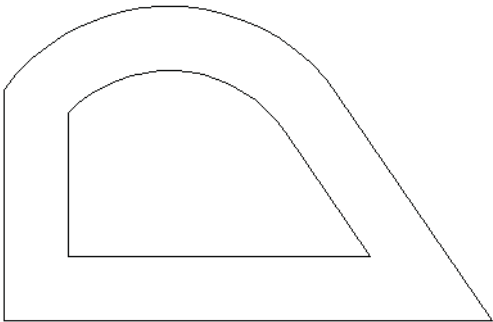
NOTE: If you want to constrain objects while they are created, and not after they are created, make sure **Auto Add Constraints** is active in the Inspector Bar.



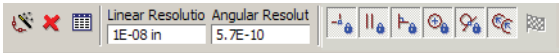
1. Start with a polyline (see "Double Line: Polyline" on page 148). Segment 1 is vertical, Segment 2 is horizontal. After Segment 3, switch to arc segments. Arc 4 is tangent to Segment 3, then close the polyline.



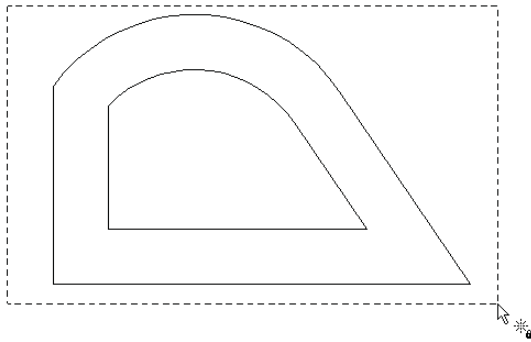
2. Use **Offset** (see "Offset" on page 211) to make an outward copy of the polyline.



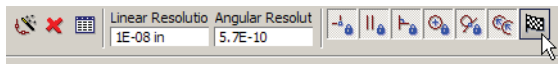
3. Activate **Auto Constraint**, and in the Inspector Bar you can select which constraints you want to apply. The available constraints are **Points Coincident**, **Parallel**, **Perpendicular**, **Concentric**, **Tangent**, and **Equal Radius**. In this example, all are selected.



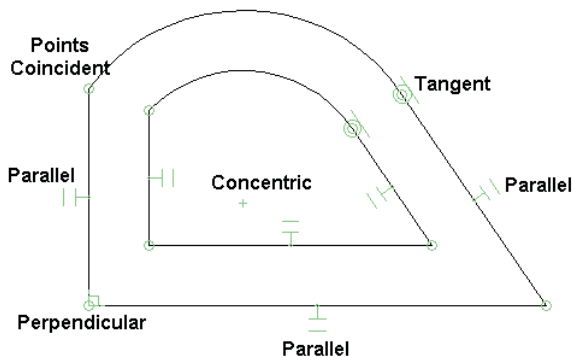
4. Drag a selection box around all objects you want to constrain.



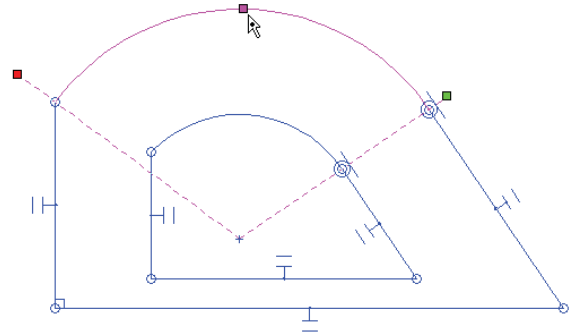
5. Click **Finish**, or select it from the local menu.



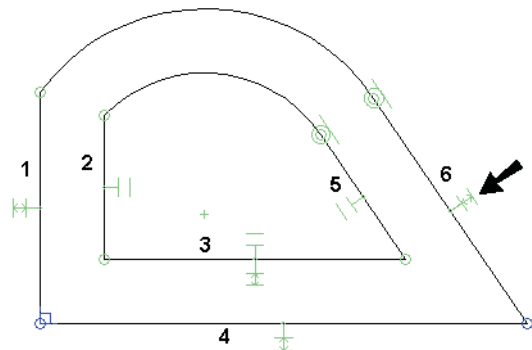
Constraint symbols are displayed for every identified condition.



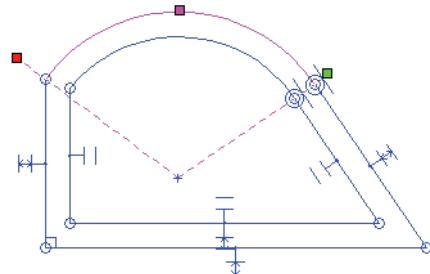
6. You can test the constraints by editing one of the arcs. Use the **Edit Tool** to increase the radius. All constraints are maintained, but the offset is no longer uniform.



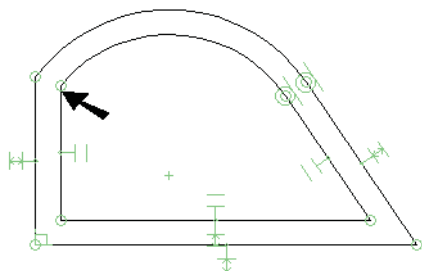
7. Undo, and apply constraints to maintain the offset distance. Use **Equal Distance** (see "Equal Distance Constraint" on page 262) to make the distance from Segment 3-4 equal to Segment 1-2. Do the same for Segment 5-6.



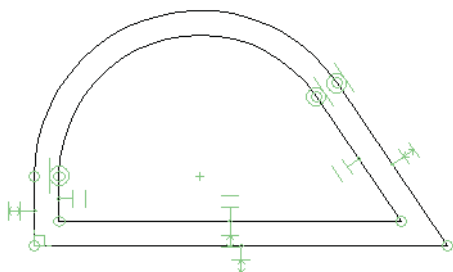
8. Now if you edit either arc, the uniform offset is maintained.



- For another update, add a **Connect** constraint at the corner shown.



This changes not only the specified corner, but its offset corner as well.



## Auto Dimension

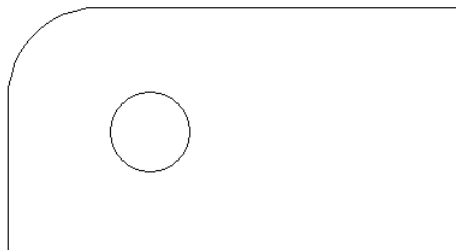
Available in TurboCAD Pro and Platinum only



Automatically applies dimensions to a sketch.

NOTE: For detailed explanations of dimension tools, see "Dimensions" on page 331. **Auto Dimension** is included in this section because it works best in conjunction with geometric constraints.

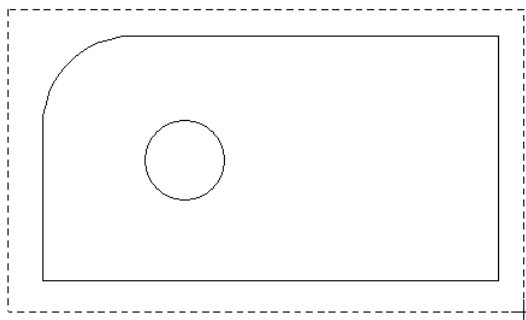
- Start with a rectangle and draw a small circle inside it. Fillet one corner (see "Fillet" on page 241).



- Activate **Auto Dimension**, and in the Inspector Bar you can select which dimensions you want to identify and label. The available dimensions are **Radius**, **Distance**, **Angular**, and **Parallel**. In this example, all are selected.



- Drag a selection box around all objects you want to dimension.

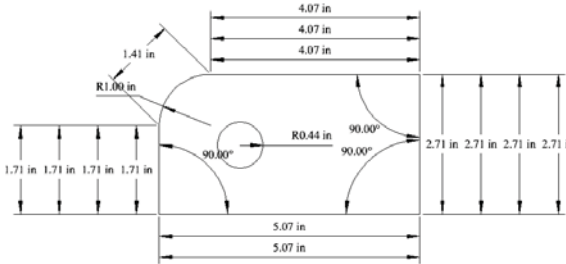


- Click **Finish** or select it from the local menu.

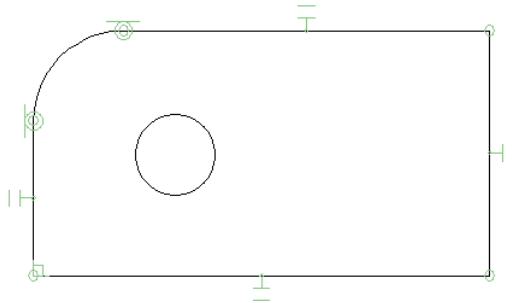




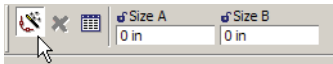
5. This is the result - dimensions placed between all identified endpoints. This occurs when objects are not constrained. In particular, **Points Coincident** constraints are needed to reduce the number of auto dimensions.



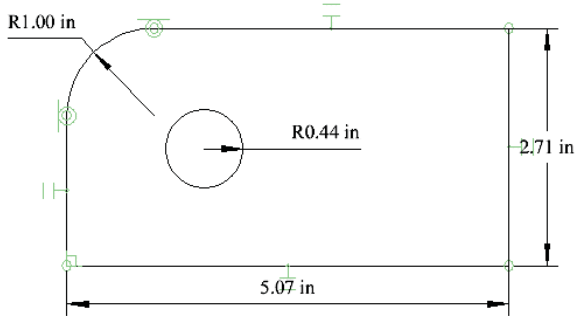
6. Undo the dimensions. The easiest way to constrain the sketch is to use **Auto Constraint** (see "Auto Constraint" on page 266). The result in this case includes **Tangent**, **Parallel**, and **Points Coincident** constraints.



NOTE: You could also get the same constraints if you keep **Auto Add Constraints** active in the toolbar while you are creating geometry (rather than applying constraints after the geometry has been created).



7. Use the same steps as before to apply auto dimensions. This is the result.



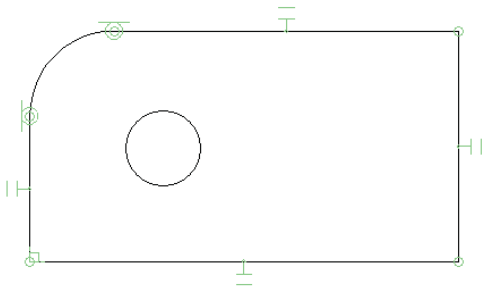
Dimensions created this way are assigned variables, which can be seen and edited in the dimensions's **Properties (Format page)**, and in the Calculator Palette (see "Calculator Palette - Variables" on page 52).

## Constraining Dimensions

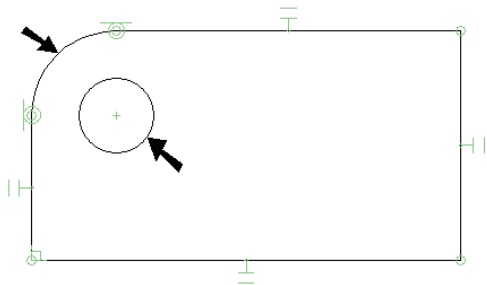
Available in TurboCAD Pro and Platinum only

You can use the Calculator Palette to assign constraints to dimensions, making them dependent on other dimensions or values.

1. Start with the same shape used to demonstrate **Auto Dimension** (see "Auto Dimension" on page 268), and apply **Auto Constraints**.



2. Add one more constraint: make the fillet and the circle **Concentric**.

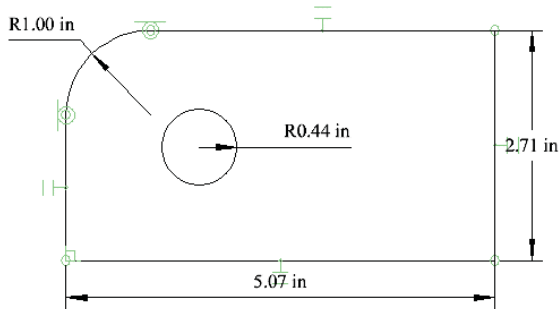


3. In the Inspector Bar, make sure **Auto Add Constraints** is active. With this active, all dimensions you create will be placed as variables in the Calculator Palette. Otherwise, dimensions will be created but they cannot be constrained, or used as constraints for other dimensions.



The dimension types that can be constrained are **Orthogonal**, **Parallel**, **Distance**, **Angular**, **Radius**, and **Diameter**.

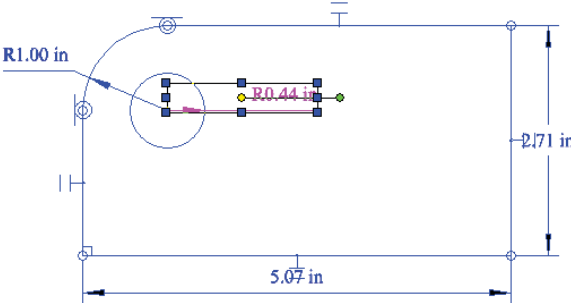
4. Apply **Auto Dimension**, and you should get the following four dimensions:



5. Make sure the Calculator Palette is open (see "Calculator Palette - Variables" on page 52). The four dimensions you created are listed here, with a variable name assigned to each.

Calculator		
Variable	Value	Formula
Var1	1.00	
Var2	0.44	
Var3	2.71	
Var4	5.07	

6. **Select** one of the dimensions, and the corresponding item is highlighted in the list.



Variable	Value	Formula
Var1	1.00	
Var2	0.44	
Var3	2.71	
Var4	5.07	

NOTE: The dimension's variable can also be seen and edited in the **Format** page of its **Properties**.

7. Click the variable for the hole radius dimension, and change its name to something meaningful, like "HoleRad." Avoid using spaces in variable names.

Var1	1.00	
HoleRad	0.44	
Var3	2.71	
Var4	5.07	

8. Change the remaining variable names.

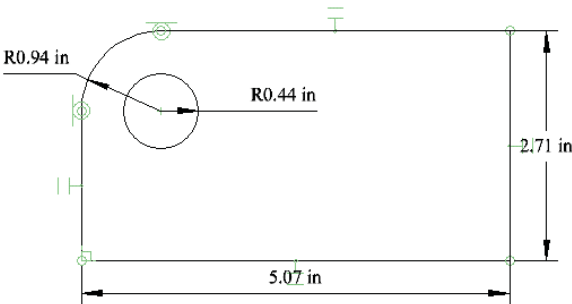
Variable	Value	Formula
FilletRad	1.00	
HoleRad	0.44	
Width	2.71	
Length	5.07	

9. One way to constrain a dimension is to base it on another dimension. For the fillet radius, click inside the **Formula** field and enter a formula that makes the fillet radius a set amount (0.5" in this example) larger than the hole radius.

Variable	Value	Formula
FilletRad	0.49	HoleRad+0.5
HoleRad	0.44	
Width	2.71	
Length	5.07	

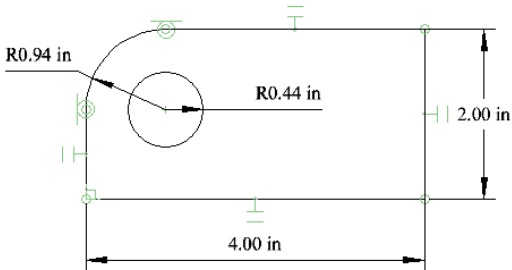
10. Press Enter and the fillet radius updates based on the current value of the hole radius. The drawing updates as well.

Variable	Value	Formula
FilletRad	0.94	HoleRad+0.5
HoleRad	0.44	
Width	2.71	
Length	5.07	



11. You can also constrain dimensions to have an exact value, such as the value below for “Length.” Width can also be constrained to be a constant factor (such as 1/2) of the length.

Variable	Value	Formula
FilletRad	0.94	HoleRad+0.5
HoleRad	0.44	
Width	2.71	Length/2
Length	4.00	

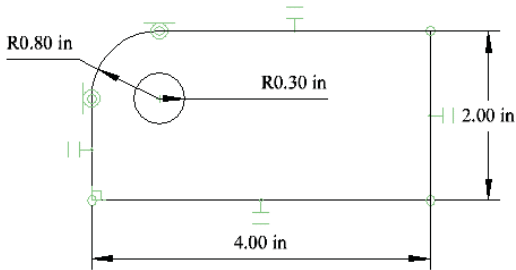


12. You can also define a new variable independent of any of the current dimensions. “PinRad” is the radius of the pin that fits in the hole, and is assigned a numerical value.

Variable	Value	Formula
FilletRad	0.94	HoleRad+0.5
HoleRad	0.44	
Width	2.00	Length/2
Length	4.00	
PinRad	0.25	

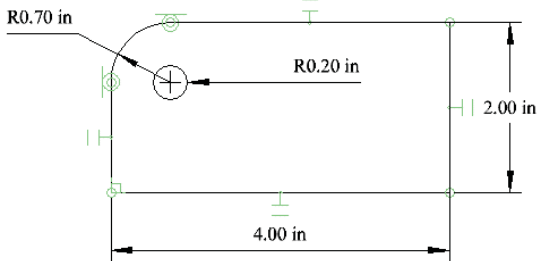
13. Once “PinRad” is defined, the hole radius can be made to a set amount larger than the pin.

Variable	Value	Formula
FilletRad	0.80	HoleRad+0.5
HoleRad	0.30	PinRad+0.05
Width	2.00	Length/2
Length	4.00	
PinRad	0.25	

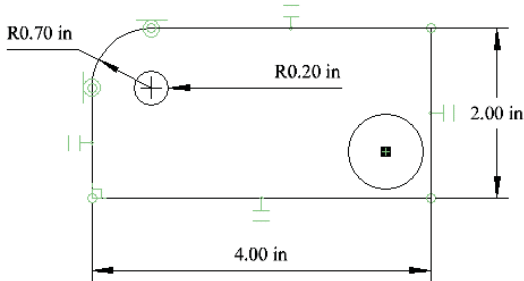


14. If “PinRad” is updated, then both “HoleRad” and “FilletRad” are automatically updated.

Variable	Value	Formula
FilletRad	0.70	HoleRad+0.5
HoleRad	0.20	PinRad+0.05
Width	2.00	Length/2
Length	4.00	
PinRad	0.15	

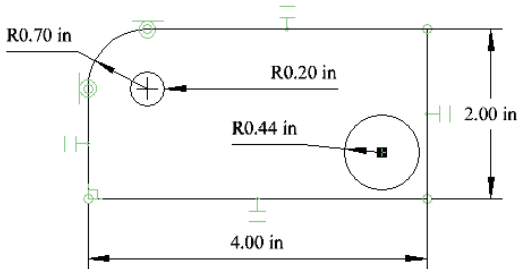


15. You can also constrain new dimensions, as long as **Auto Add Constraints** is active. Create another circle, and inside this circle add a point (see “Point” on page 137). Make the point and the circle **Concentric**.



NOTE: The point is added because a circle’s center point is not identified as an object; a physical point must be placed there.

16. Add a **Radius** dimension to the new circle, and its variable appears in the constraint list.



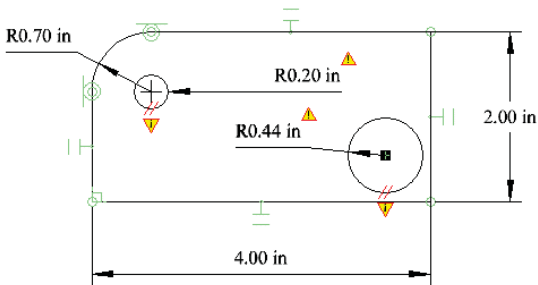
Variable	Value	Formula
FilletRad	0.70	HoleRad+0.5
HoleRad	0.20	PinRad+0.05
Width	2.00	Length/2
Length	4.00	
PinRad	0.15	
Var6	0.44	

17. If you try placing an **Equal Radius** constraint on the two circles, you will receive an error message:

**TurboCAD Warning**

There are overdefined or inconsistent constraints.

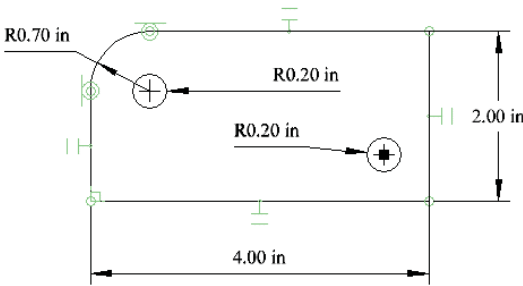
18. When you click OK, the problem dimensions are indicated.



This occurs because the second circle is considered to already have a set radius, once it is dimensioned. So constraining it to be equal to the first circle is contradictory. You could have assigned the **Equal Radius** constraint before assigning the dimension.

19. Undo, and use a dimensional constraint to set the radius, making it equal to "HoleRad."

Variable	Value	Formula
FilletRad	0.70	HoleRad+0.5
HoleRad	0.20	PinRad+0.05
Width	2.00	Length/2
Length	4.00	
PinRad	0.15	
HoleRad2	0.20	HoleRad



### Constraining Pattern - Copied Objects

Available in TurboCAD Platinum only

This section refers to objects copied in patterns (lines, linear arrays, radial arrays), using the **Copy Entities** tools (see "Copying Objects" on page 206).

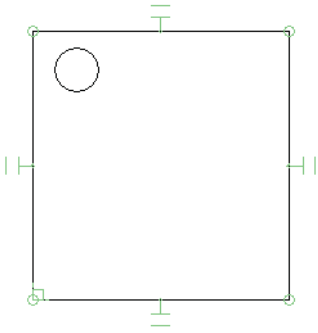
When you copy objects in a pattern, you can use **Auto Constraints** and dimension variables to control spacing, angles, etc.

This example uses a circle copied into a linear array.

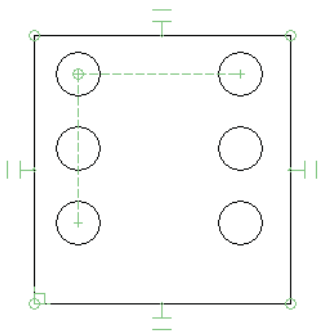
1. Before you begin, make sure **Auto Add Constraints** is active in the Inspector Bar.



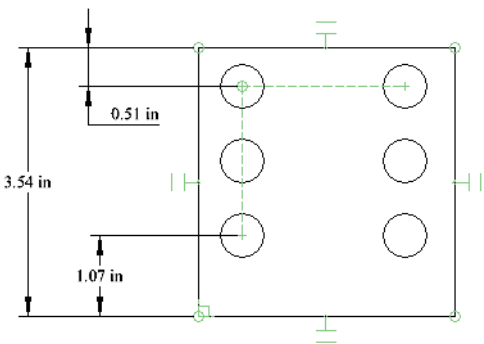
2. Create a rectangle and add a small circle. The rectangle should have parallel and coincident constraints assigned automatically.



3. Select the circle and use **Fit Linear Copy** to copy the circle into a grid. (See "Fit Linear Copy" on page 208)



4. Assign some **Orthogonal** dimensions as shown. (See "Orthogonal Dimension" on page 340).



5. Open the Calculator palette (F2) and the dimensions should be shown, identified by a variable. In this example, the variables were assigned names (L, E1, E2).

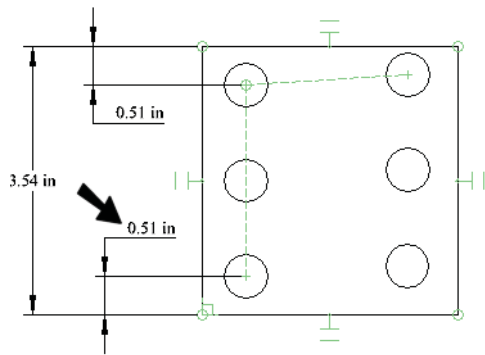
Variable	Value	Formula
L	3.54	
E1	0.51	
E2	1.07	

NOTE: For more information on working with variables, see "Calculator Palette - Variables" on page 52

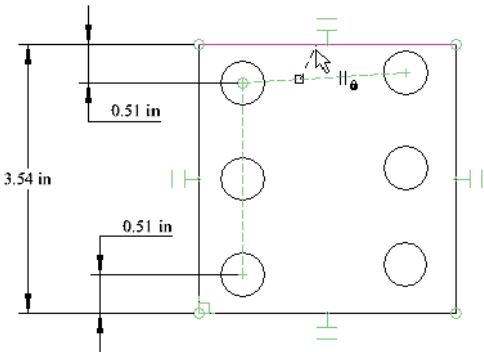
6. Create a formula so that the two edge dimensions will be equal. In this case, the variable E1 was used as the formula for E2.

Variable	Value	Formula
L	3.54	
E1	0.51	
E2	0.51	E1

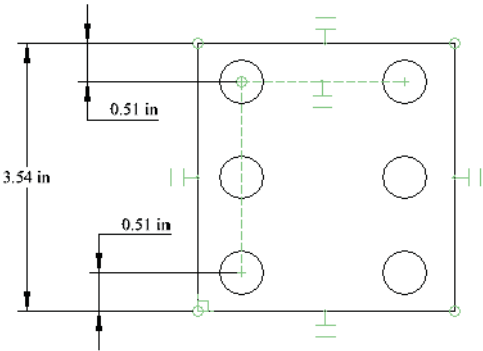
The dimensions update, and the spacing between copied objects remains constant. However, in this case, the linear array does not have a 90 degree angle.



7. To fix this, activate the **Parallel** constraint and click first the array constraint line, then the top line of the rectangle.



Now the array is perpendicular again.



## 2D Boolean Operations

Boolean operations use two or more existing 2D objects to create a new object. Objects can be combined, subtracted, and intersected.

You can display the **Boolean and Facet** toolbar by right-clicking in any toolbar area and selecting **Boolean and Facet**.




---

NOTE: 3D objects can also be combined using Boolean operations. See "3D Boolean Operations" on page 432

---

The object created as a result of a 2D Boolean operation is considered a **region**. You can use the Selection Info Palette to determine the type of any object or objects you select. See "Selection Info Palette" on page 191.

Selection order can be important, because the properties (color, layer, etc.) of the resulting region are taken from the **first** object selected.

### 2D Add



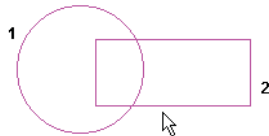
Combines 2D closed objects to form one region. Any overlap between selected objects is removed.

---

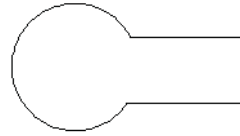
NOTE: The results of **2D Add** are identical to using **Format / Create Region**.

---

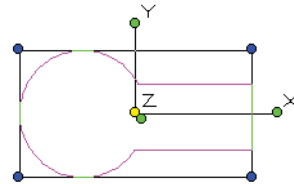
1. Select two or more closed objects to combine.



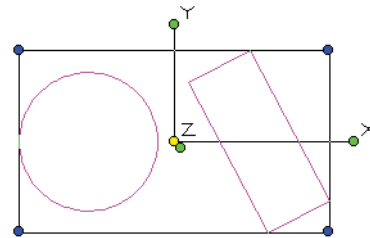
2. Select **Finish Selecting** from the local menu or Inspector Bar.



The resulting object is one region. You can verify this by selecting the region.



The selected objects do not have to overlap. If you combine non-overlapping objects, they will still be combined into one region. You can verify this by selecting the region.

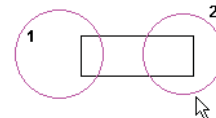


### 2D Subtract



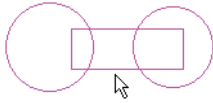
Subtracts one or more objects from one or more objects, to form one region.

1. Select one or more objects to subtract area from.

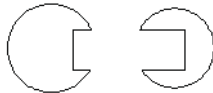


2. Select **Finish Selecting** from the local menu or Inspector Bar.

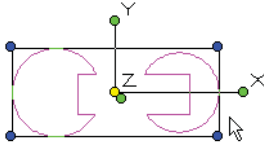
3. Select one or more objects to subtract from the previously selected objects.



4. Select **Finish** from the local menu or Inspector Bar. The second group of objects is removed from the first group.



The resulting object is one region. You can verify this by selecting the region.

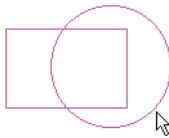


## 2D Intersect



Produces the intersection of two or more closed 2D objects, to form a region.

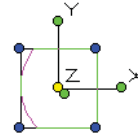
1. Select two or more objects whose intersection you want to obtain.



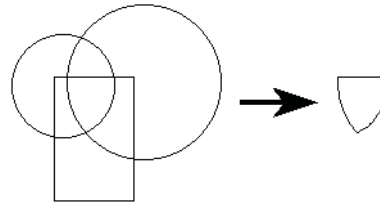
2. Select **Finish Selecting** from the local menu or Inspector Bar.



The resulting object is one region. You can verify this by selecting the region.



You can select more than two objects, but they must all intersect one another.




---

*NOTE: If you select objects that do not overlap, the result will be no objects (original objects deleted).*

---



# Measuring

You can also display the **Measurement** toolbar by right-clicking in any toolbar area and selecting **Measurement**.

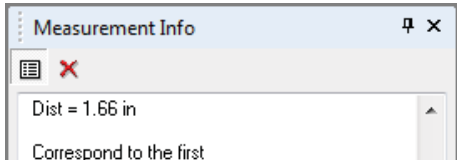


**Measurement** tools are constrained to the current workplane. If you snap to a point that does not lie on the workplane, that point will be projected onto the workplane. Make sure the workplane is set properly for the objects you wish to measure, or the measured values will not be accurate. See "Workplanes" on page 362

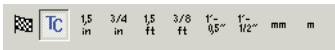
Results of measuring are displayed in the Measurement Info Palette, which is opened automatically when using a measuring tool.

*TIP: To copy the contents of the Measurement Info Palette to the Windows clipboard, use the Copy command from the local menu. You can then paste the information into any word processor, text editor or other program that will accept text.*

The Measurement Info Palette can be set to keep a history of all the measurements you take. Use the Keep History button at the top of the palette to retain measurement history. Use the Clear Content button to delete the content of the palette.



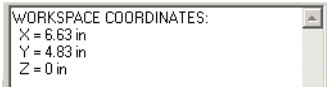
The default coordinate units are those set in the **Space Units** page of the **Drawing Setup** (see "Space Units" on page 71.) You can adjust the units by selecting an option from the Inspector Bar or local menu. English units are available in fraction and decimal formats, and metric units can be displayed as well.



# Coordinates of a Point



Select the point, and its coordinates are displayed in the Measurement Info Palette.

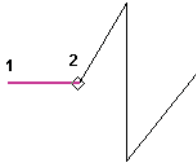


# Measuring Distance

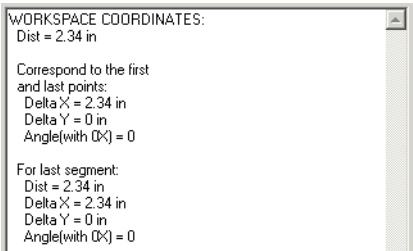


Measures the distance between two points, the cumulative length between additional points, or the length of entire objects (perimeter).

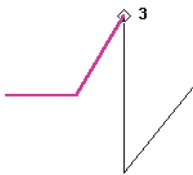
1. Select two points to define the distance you want to measure.



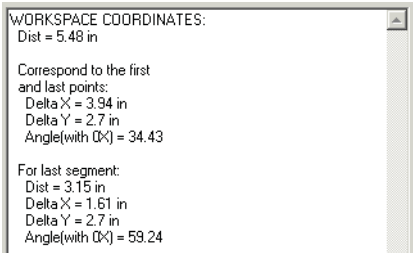
The distance and deltas in X and Y are listed in the Measurement Info Palette.



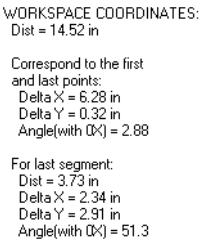
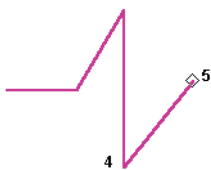
- 2. If needed, continue selecting points. The outline of selected segments is indicated by a thick line.



The first listed measurement is the total length of segments. The second group of measurements relate to the first and last points - in this case, Point 1 and Point 3. The last group applies to the last segment only (Point 2 to Point 3).



- 3. Continue selecting points, in order, to determine the perimeter of the object.



*NOTE: If you want to obtain the measurements of curved objects, you need to use the local menu options.*

Local menu options:

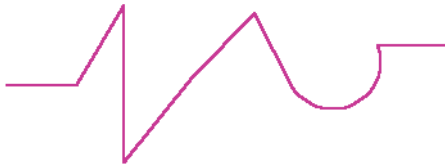
**By Entity:** Enables you to select two or more objects to determine their total length.



- 1. Select the first object.



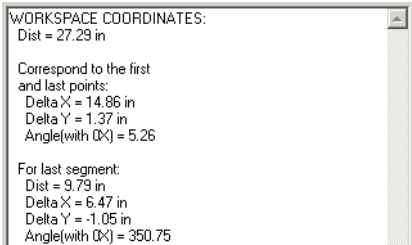
- 2. Select the second object. The total length is calculated based on the shortest line connecting the two endpoints.



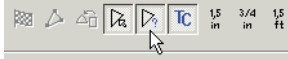
*NOTE: When selecting objects, be sure to click close to the desired start point. If you select close to the wrong end, you may obtain an incorrect measurement.*



- 3. Select additional objects if needed. The total distance, deltas between the first and last points, and data for the last segment are listed in the Measurement Info Palette.



**Single Entity:** Measures the distance of a single object.



1. Select the desired object.



The total distance and data deltas between the first and last points are listed in the Measurement Info Palette.

WORKSPACE COORDINATES:  
Dist = 14.52 in  
  
Correspond to the first  
and last points:  
Delta X = 6.28 in  
Delta Y = 0.32 in  
Angle(with X) = 2.88

2. If you select another object, the data from the first object disappears, and is replaced by that for the new object.



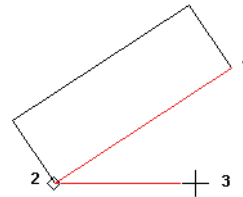
WORKSPACE COORDINATES:  
Dist = 9.79 in  
  
Correspond to the first  
and last points:  
Delta X = 6.47 in  
Delta Y = -1.05 in  
Angle(with X) = 350.75

## Measuring Angles

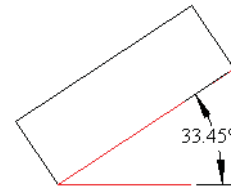


Measures the angle between three points, or between two lines.

1. Select three points that define the angle you want to measure.



The angle is measured and displayed. It is calculated according to the order of selected points.

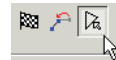


The angle is also displayed in the Measurement Info Palette.

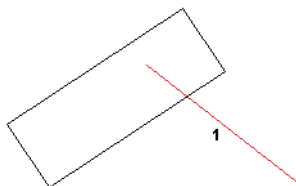
WORKSPACE COORDINATES:  
Angle = 33.45

### Local menu option:

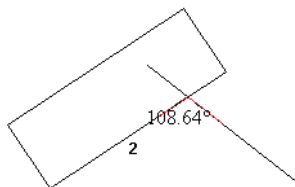
**By Entity:** Measures the angle between two lines. The lines must intersect (this tool does not infer where extensions of lines will meet).



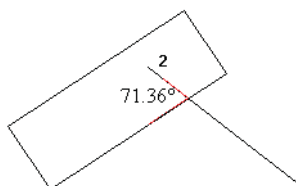
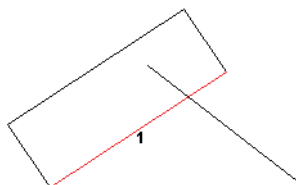
1. Select the first line. The point you select on the line defines the angle that will be measured.



2. Select the second line, and the angle is displayed.



In this example, if you selected the lines in a different location, you will measure a different angle.

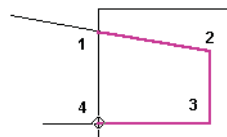


## Measuring Area

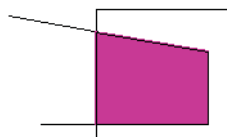


Measures the area of a closed defined boundary.

1. Select points to define segments of the boundary. The points do not have to lie on existing objects; they can be anywhere. A thick line indicates the progress of the boundary.



2. Select the first point again to close the boundary, or select **Close** from the local menu or Inspector Bar. The defined area is filled.



3. The calculated area is displayed in the Measurement Info Palette.



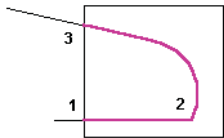
### Local menu option:

**By Entity:** Enables you to select the entire perimeter of an object.

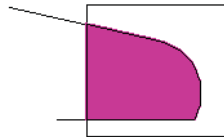


You can also use this option to define a chain along a portion of an object. This is useful when you need to measure the area of curved objects.

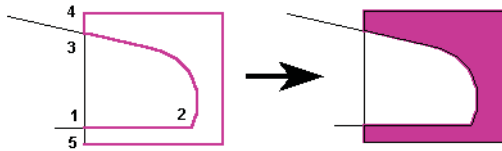
1. Select a closed object, or select points along a portion of an object.



2. Select the first point again to close the boundary, or select **Close** from the local menu or Inspector Bar. The defined area is filled.



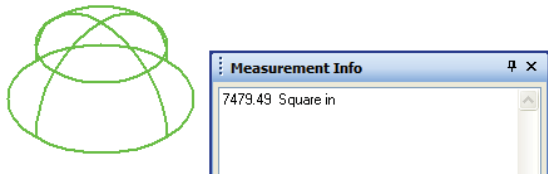
You can also use **By Entity** to define a boundary along more than one object.



## Measuring Surface Area



Select a surface. Its surface area is displayed in the Measurement Info Palette.

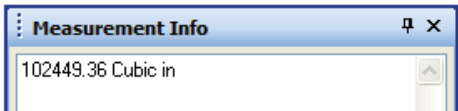
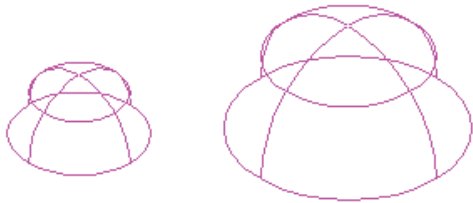


NOTE: To select multiple surfaces hold down the Shift key.

## Measuring Volume



Select the object. Its volume is displayed in the Measurement Info Palette.

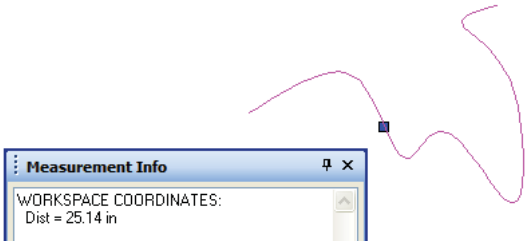


NOTE: To select multiple surfaces hold down the Shift key.

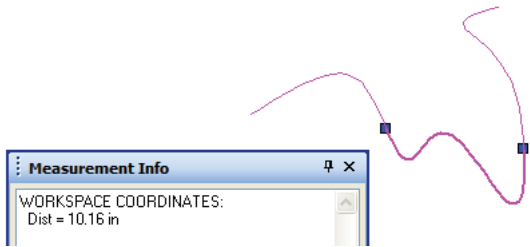
## Curve Length



This tool works exactly the same as the Measure Distance. Click on a 2D or 3D curve. Its length is displayed in the Measurement Info Palette.



Select a second point on the curve to measure a partial length of the curve.



Geometric Parameters

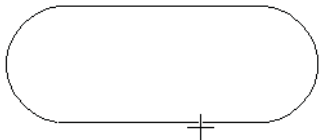
Available in TurboCAD Pro and Platinum only



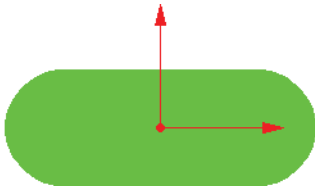
NOTE: You can display the **Special Tools** toolbar by right-clicking in any toolbar area and selecting **Special Tools**.

Calculates engineering properties (area, center of gravity, moments of inertia, etc.) for a 2D closed object.

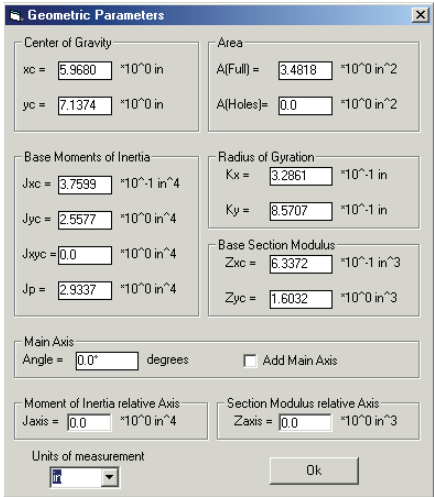
- 1. Select the closed area.



- 2. The area is highlighted, and its axes are indicated in red.



The parameters of the closed area appear in the **Geometric Parameters** window.



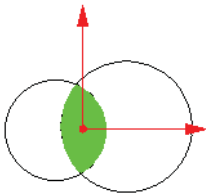
Click OK to close the window, and select **Finish** from the local menu or Inspector Bar to exit the tool.

Local menu options:

**By Closed Graphic** (default): Selects a single closed object.



**By Closed Area:** Click inside a closed area to obtain the properties for only that area.



**Relative Axis:** Enables you to define another axis for the calculation of the properties.



## Tracing

Available in TurboCAD Pro and Platinum only



The **Trace** tools originate from the stand-alone ScanPro application, which has now been incorporated into TurboCAD Pro. They enable you to create 2D raster-to-vector tracings of inserted pictures (see "Inserting a Picture" on page 173) or of geometric objects.

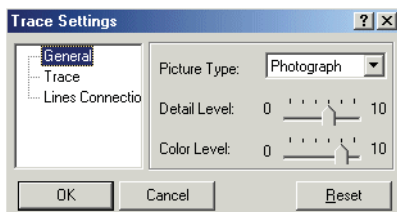
**Trace** tools are available on the **Tools** toolbar, which you can display by right-clicking in any toolbar area and selecting **Tools**.

Before creating a tracing, it's important to understand the local menu options, since these greatly affect the appearance and quality of the trace.

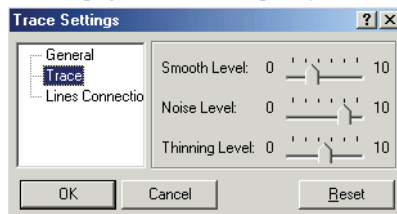
**Trace Options:** Settings for trace quality.



The **General** page provides options for the **Picture Type**. The **Detail Level** and **Color Level** values update with the picture type.



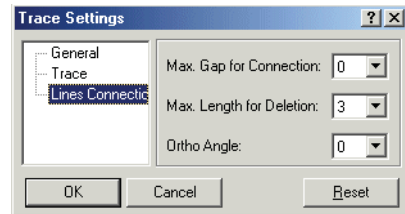
The **Trace** page controls the quality of the tracing.



- **Smooth Level:** Reduces jags and discontinuities in lines and curves.
- **Noise Level:** Reduces spots.

- **Thinning Level:** Reduces line widths.

The **Lines Connection** page contains parameters relevant when **Connect Lines** is activated.



- **Max Gap for Connection:** The largest gap which will be closed when connecting lines.
- **Max Length for Deletion:** The largest segment that can be deleted, i.e., breaking the lines on either side.
- **Ortho Angle:** The angle at which line segments will be joined into polylines.

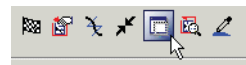
**Curve Recognition:** Traced polylines will be created as curves. Otherwise they will be created as lines and polylines.



**Connect Lines:** Disconnected lines will be merged into a single line. The tolerances for connected lines are set in the **Lines Connection** page of the **Trace Options**.



**Grab the Window:** If on, then the tracing is made directly behind the traced objects. If off, the tracing is saved to the buffer.



The advantage to grabbing the window is speed, but some artifacts may be left on the image (grid, flyout controls, etc.). When not used, there are no artifacts, and the image size is limited only by system memory. However, the tracing takes longer, especially in render mode.

**Preview Mode:** A trace preview will appear in magenta over the image.



**Tracing Colors:** By default, all colors will be used for tracing. This option is used when you want to trace around only selected colors.



The following two options are relevant only for **Trace by Point**:

**Show / Hide Trace Rectangle:** Toggles the display of the tracing rectangle, defined by its aperture size.



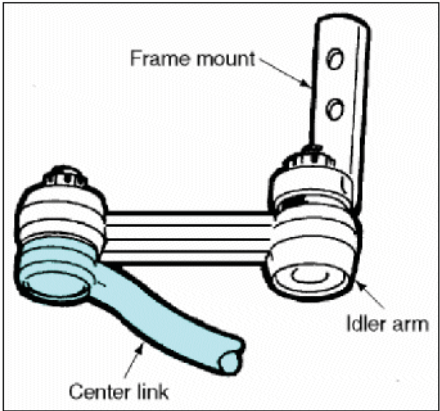
**Nearest Graphic Only:** Traces only the object closest to the tracing rectangle's center, identified by crosshairs.



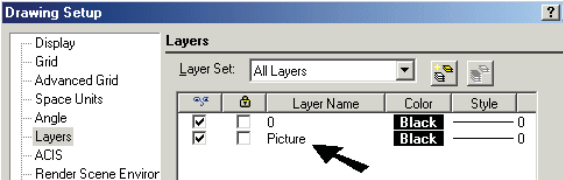
### Trace by Rectangle



Creates a tracing of all objects within a selection rectangle. This example uses the following inserted picture:

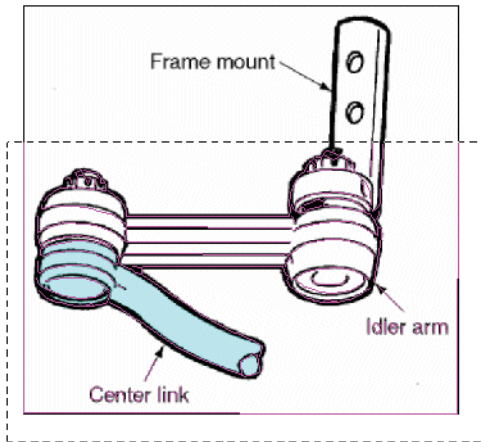


To easily see the results of the trace, it is good practice to place the inserted picture on its own layer (see "Layers" on page 116), which can then be blanked.

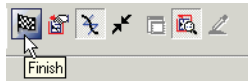




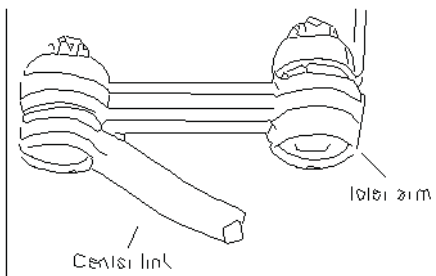
1. Activate **Trace by Rectangle** and define the selection rectangle. (You can also use fields in the Inspector Bar to control the rectangle size.) If **Preview Mode** is on, the traced lines will appear in magenta.



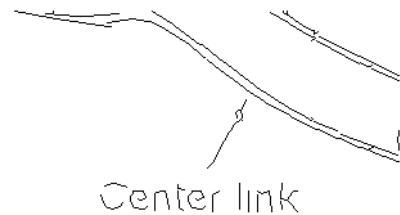
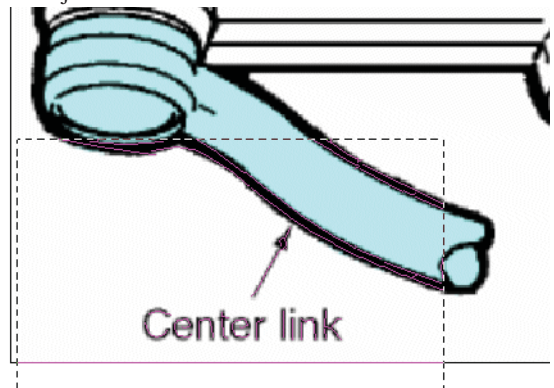
2. Select **Finish** from the Inspector Bar or the local menu.



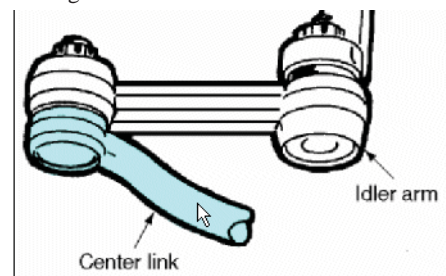
3. Move or delete the picture, or simply blank its layer, to see the tracing.



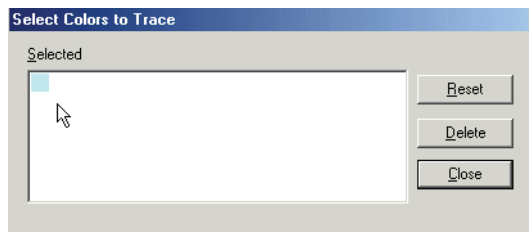
In the above example, the text lines did not trace completely. To improve trace quality, you can increase the zoom of the traced objects.



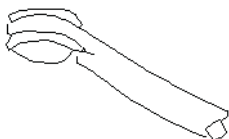
The previous example traced around all colors. If you only wanted to trace around the shaded area of the center link, you would use the **Tracing Colors** option, which must be set before starting the trace.



When the **Select Colors to Trace** window opens, click on the colors you want to include. The colors then appear as blocks in the window.



The result is a trace only around the center link.




---

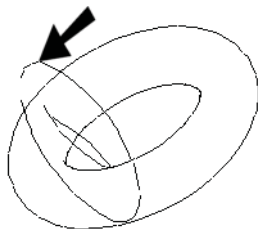
NOTE: Trace colors remain active for future traces, so it is important to click **Reset** (to zero colors) if necessary.

---

In addition to pictures, you can also trace around geometric objects. This example is a solid torus, rotated and displayed in Hidden Line mode.



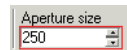
The trace is created in the current viewing plane. The result is shown below, rotated so that the 2D trace can be seen:



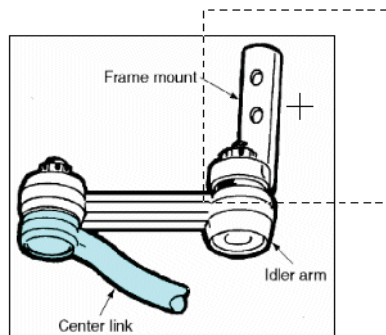
## Trace by Point



This tool is basically the same as **Trace by Rectangle**, with a different way of defining the trace area. The selection area is sized by the **Aperture** field, which is the length of each side of a square whose center is represented by crosshairs.



Using the same example picture as for **Trace by Rectangle**, place the cursor at the center of the desired selection area.



The results are the same - all objects within the area will be traced.

If **Nearest Graphic Only** is active, then only the object closest to the crosshairs will be traced.

## Detail Section

*Available in TurboCAD Pro and Deluxe only*



The section detail tool allows you to create a 2D detail section of 2D elements for part of your drawing. The Detail Section tool ignores 3D objects entirely.

1. Select the Detail Section tool.
2. In the Inspector bar specify the scale for the detail section.

3. Select an existing, closed 2D shape (polyline, curve, circle) to act as the cutting contour for the section.
4. Drag the detail to the desired location.

---

*NOTE: Dimensions and text that are not entirely within the cutting contour will be ignored/excluded from the resulting detail section. Hatches within the detail area are not scaled in the resulting detail section.*

---

Local menu option:

**Trim by circle:** If you select this option you define the cutting contour drawing a cutting circle. Simply select the center point of the circle, then the radius. After that you just drag the resulting detail to its new location.



# 8 Groups, Blocks, and the Library

Groups, blocks, and symbols are features that enable you to combine and store multiple objects for easy access, and for additional and future use.

## Auto-Naming

By default, each new group, block, or symbol is assigned a name automatically. These names are numbered, and the number increases by one whenever you add a new group, block, or symbol to the drawing, ensuring that each has a unique name. By default, the generated names are "Group 1", "Group 2", "Block 1", "Block 2", "Symbol 1", "Symbol 2," and so forth.

You can change the default naming settings in the **Auto-Naming** page of the **Program Setup**. See "Auto-Naming" on page 64.

The "@" symbol is a placeholder where the actual number will be inserted. For example, if you type "Part @" in the **Group name prefix** field, the first group you create will automatically be named "Part 1", and the next group will be named "Part 2". If the **Group name prefix** is "#@ gear," the groups will be named "#1 gear," "#2 gear," etc.

---

*NOTE: The first '@' that you include in the name is a placeholder for the automatic number. Any '@' characters that occur after the first one will be part of the actual name.*

---

The name of a group or symbol (not a block) is listed on the **General** page of the **Properties** window, in the **Attribute** field. You can use this field to change the name (or path, for a symbol).

## Groups

Two or more objects can be combined as a group. A group is treated as a single object for purposes of selecting and editing.

Each group contains its own drawing data. This means that making a copy of a group increases the size of your model. If you need to create several groups of numerous or complex objects, you may want to use blocks instead. Blocks are stored in a library, and each instance of the block refers to this external source, without significantly increasing the file size. See "Blocks" on page 291.

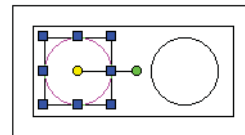
Because groups can contain groups and blocks, they can be complex hierarchical structures. The structure of a group can be seen in the Selection Info Palette (**View / Selection Info**), which can also be used to edit a group's contents. See "Selection Info Palette" on page 191.

## Creating a Group



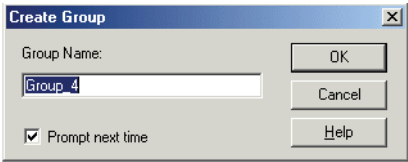
Combines all selected objects into a group. This tool is available on the Groups and Blocks toolbar, or on the icon in the **Drawing Tools**.

1. Create the objects you want to combine into a group. Each object is initially separate, as you can see in **Select** mode.

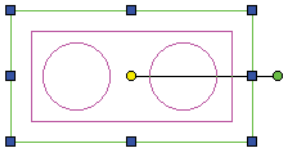


2. Select all the objects you want to include.
3. Click the **Create Group** tool or select **Create Group**.

4. If **Auto-naming** is not used, you must assign the group name.

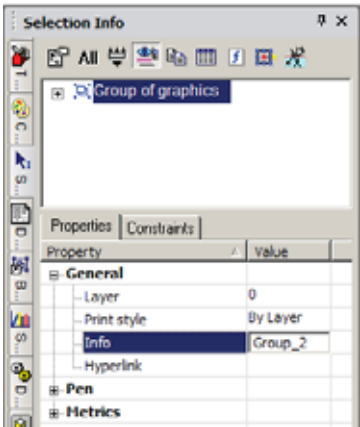


The group is created, and the objects are now selected as one object.

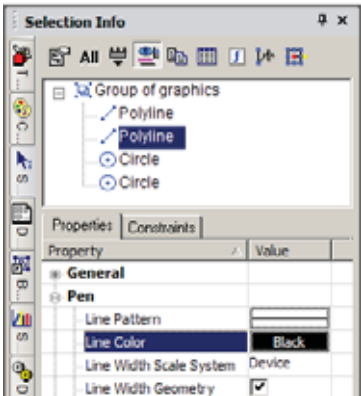


NOTE: The new group will be placed on Layer 0 if its components are originally on different layers. See "Layers" on page 116. If you explode a group, the objects will return to their original layers. Layer 0 should always be left visible, or newly created groups will instantly "disappear."

If you open the Selection Info Palette, you can view the contents and properties of a selected group.

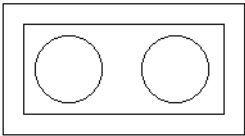


You can highlight any component of the group to view its individual properties as well.

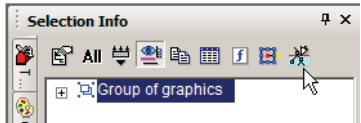


### Editing a Group

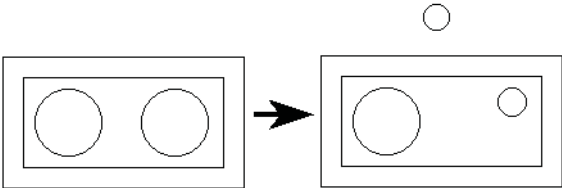
This example will use the following group, consisting of four objects:



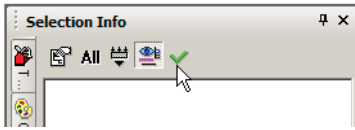
1. Select the group, and select **Edit Group Content**. If the Selection Info Palette is open (see "Selection Info Palette" on page 191), you can also click the **Edit Content** button on the palette toolbar.



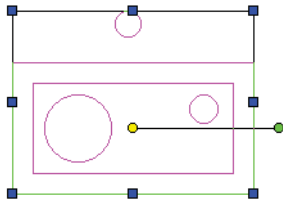
2. You are now in edit mode, and the screen contains only the group's objects. You can add objects, delete objects, or edit existing objects. Whatever appears on the screen will become part of the edited group.



- When finished, select **Finish to Edit Group**, or click **Finish Edit Content** in the Selection Info Palette.



The group now contains the edited and new objects.



## Exploding a Group

To break a group into its constituent parts, select it, then select **Explode**, or click the **Explode** icon.



If you explode a group that contains nested groups or blocks, the nested groups will remain intact. Each sub-group must be exploded separately. See "Exploding Objects" on page 247.

## Blocks

One or more objects can be combined and stored as a block. A block is treated as a single object for purposes of selecting and editing.

Each block is stored in the drawing's internal library, and each instance of the block is a reference to this source. This means that numerous instances of a block can be added to the model without significantly increasing the file size. Groups are similar, but they are not linked to sources; each group contains its own drawing data. See "Groups" on page 289.

**NOTE:** A drawing's block library is internal to the drawing, and is stored with the file. Symbol libraries are similar but are stored separately, and can be accessed while in any drawing. If you need to create a group of objects that will be used in multiple drawings, create a symbol. See "Loading Symbol Folders into the Library" on page 309.

If you want to import the entire contents of another file (TurboCAD or other format) as a block, see "External References" on page 303.

Because blocks can contain individual objects, groups, and other blocks, they can be complex hierarchical structures. For block manipulation, use the Blocks Palette (**View / Blocks**).

**TIP:** You can use the TC Explorer Palette to view blocks of any open drawing, and to drag blocks to and from drawings. See "Blocks" on page 94.

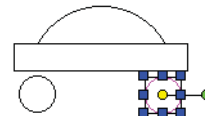
## Creating a Block



Combines all selected objects into a block. Objects can include drawing objects, images, OLE objects, groups, and other blocks. Blocks can be comprised of 2D and 3D objects.

**WARNING:** You should move all objects to be used in a block to Layer 0, before creating the block. Many features will not work properly otherwise.

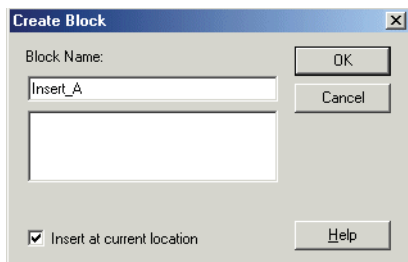
- Create the objects you want to combine into a block. Each object is separate, as you can see in **Select** mode.



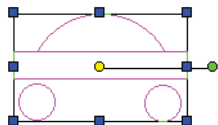
You can set the reference point of the objects before creating the block (see "Changing the Reference Point" on page 199), or you can change it later.

2. Select the objects you want to include in the block.
3. Click **Create Block**. Or see "Creating a Block by Drag and Drop" on page 292.
4. If auto-naming is not used, you must assign the block name. (See "Auto-Naming" on page 289.)

Check **Insert at current location** to create the block in place. Otherwise, the block will be removed from the drawing and stored in the library for future insertion.

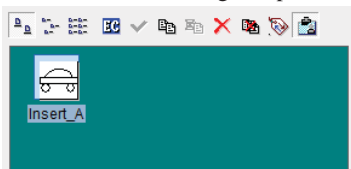


5. The block is created. If the block is inserted in the drawing, you can select it as one object.



Blocks are created on Layer 0, even if their components are on other layers. To insert the block, see "Inserting a Block" on page 292.

If you open the Blocks Palette, you can see each block you have created. You can also add blocks to the drawing, edit blocks and create new blocks using this palette.

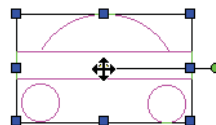


**WARNING:** Do not include lights in a block. If you do, the lights will remain in their original locations regardless of where you place the block in your drawing.

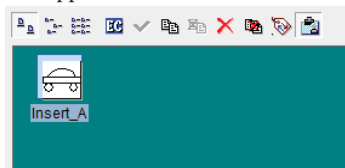
## Creating a Block by Drag and Drop

To create a block this way, the Blocks Palette must be open. Select **Blocks palette**, or the Blocks Palette tab.

1. Select the objects that you want to combine into a block. Click and hold the reference point (the yellow circle). The cursor turns into a double arrow.



2. Drag the objects into the Blocks Palette. If auto-naming is not used, assign a block name.
3. The block appears in the Blocks Palette.



## Inserting a Block

To insert a block into the drawing, simply drag it out of the Blocks Palette and drop it into your drawing. The inserted block will still be selected after you place it, so that you can move, scale, or rotate it. See "Select Edit" on page 198.

Blocks are placed on Layer 0, even if their components are on other layers. Layer 0 should always be left visible, or blocks will instantly "disappear."

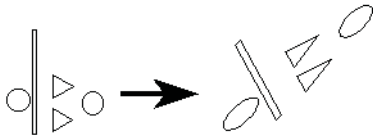
**TIP:** You can use the TC Explorer Palette to drag blocks to and from drawings. See "Blocks" on page 94.

## Block Insertion Properties

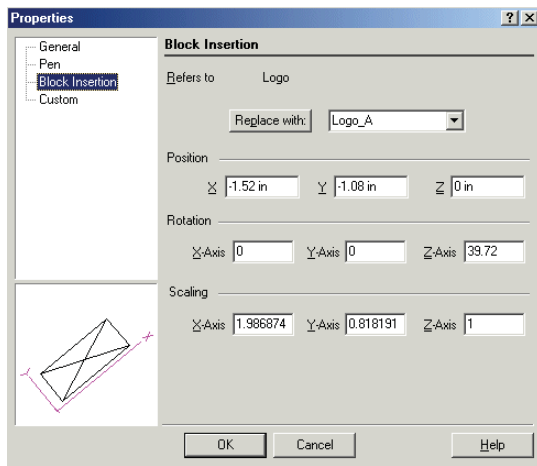
These properties can be used if you want to change any aspect of the block instance - its location, scale, angle, or the block reference itself. For any block, open the **Properties** window and open the **Block Insertion** page.



For example, a block was inserted, then moved, rotated, and resized.



Its **Block Insertion** page contains the current values for **Position**, **Rotation**, and **Scale**.



You can change the values in this window, or use the **Select Edit** tools and see the updated values in these fields.

To replace a selected block with another block, select the replacement block from the list and click **Replace with**. Click **OK** to implement the change.

## Inserting Blocks into Another File or Application

You can also use the drag-and-drop technique to insert blocks into another open file.

Dragging a block into another drawing accomplishes two things: it inserts the block into the target drawing, and it places the block into the library of the target document. The target drawing must be open and its window must be visible on the screen. (Use **Window / Tile** to see all open windows.) After you drag the block, the target file becomes the active window.

Drag-and-drop can also be used to place blocks, symbols, or any selected objects into other Windows applications, such as Microsoft Word or graphics programs.

---

*NOTE: You can also use **File / Extract To** to export all blocks into another file.*

---

## Inserting Blocks from Another File

The **Insert / File** tool can be used to insert some or all blocks from another file into the current drawing (see "Inserting a File" on page 172). If both drawing have blocks with identical names, you can choose whether to ignore or replace them.

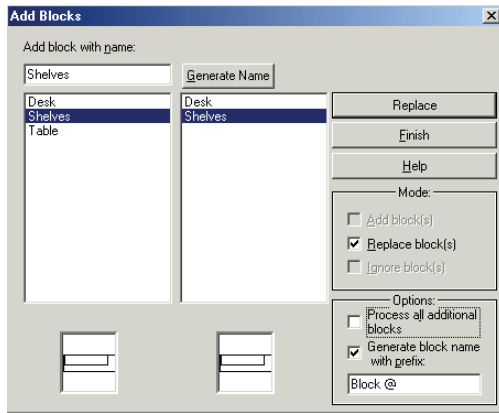
---

*TIP: You can also use **File / Extract From** to insert selected components like blocks (or layers or other settings) from another file into your drawing. However, this method will insert **all** blocks, without enabling you to pick and choose.*

---

1. Select **Auto Naming** and make sure that **Prompt for Name** is checked for **Blocks**.
2. Select **Insert / File** and choose a file containing one or more blocks you wish to insert.

3. Use the **Add Blocks** window to select the blocks to import:



The left panel displays the blocks found in the selected file, and the right panel displays any blocks that exist in the current drawing. Select the mode (**Add**, **Replace**, or **Ignore**) and click the relevant button at the top right (**Add**, **Add / Replace All**, etc.) to generate the blocks. If you want to pick and choose the blocks to add, make sure **Process all additional blocks** is not checked.

**Generate name:** Assigns a new name to a block you wish to add.

**Modes:** The options here depend on the selected block, and whether a block with the same name already exists in the current drawing.

- **Add block(s):** Adds the selected block.
- **Replace block(s):** The blocks from the external file will replace those in the current drawing.
- **Ignore block(s):** Click **Ignore All** and the blocks will not be added.

#### Options:

- **Process all additional blocks:** Adds and/or replaces all blocks found in the source file.
- **Generate block name with prefix:** Assigns a name automatically, with the specified prefix, to the inserted blocks.

Using **Insert / File** also adds all drawing objects found in the source file. However, you can press **Undo** (Ctrl+Z) immediately after using **Add Blocks** to clear the imported objects, leaving only the imported blocks. You may have to undo twice, to remove objects both in Model Space and Paper Space. Other source file components like layers, lights, and views will also be inserted, but they can be deleted manually if needed.

---

**WARNING:** *If the source file and current drawing have layers or other components with identical names, the layers will be replaced with those of the inserted file.*

---

There are other ways to import blocks from another drawing, without importing other components:

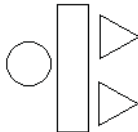
- Open both the source file and new file, and select **Windows / Tile** so that you can see both drawing windows. Use the Blocks Palette to drag blocks from the source file to the new file. This method imports the blocks only.
- In the source file, select the blocks you want to export (select in the drawing area, not in the Blocks Palette). Copy the blocks (Ctrl+C or Edit / Copy), and paste them (Ctrl+V) into the destination file. The **Add Blocks** window will appear. This method imports both the blocks and the layers the blocks are on.

### Editing a Block

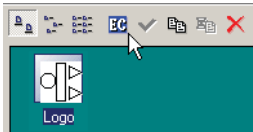
(Changing a block's contents)

You can use any editing or node editing tool to modify, move, copy, add, etc. objects that comprise a block.

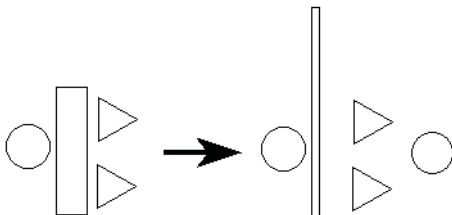
This example use the block shown below, consisting of four objects.



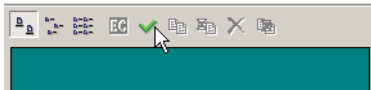
1. In the Blocks Palette, select the block, and click **Edit Content**.



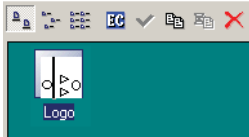
2. In edit mode, the screen contains only the block components, which are available for editing. You can add, delete, or edit objects. Whatever appears on the screen will become part of the edited block.



3. When finished, right click and select **Finish block/group editing**, or click **Finish Edit Content** in the Blocks Palette.



The icon for the block is updated to reflect the edited components. Any instances of the block already in the drawing are updated, including resized and scaled blocks.



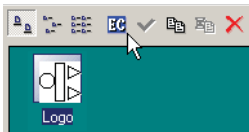
### Relocating a Block Reference Point

**Hotkey:** Ctrl+Shift+R

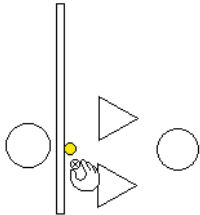


A block is inserted by placing its reference point. By default, the reference point is at the center of extents of the entire block, but this location can be changed.

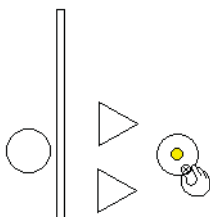
1. In the Blocks Palette, select the block, and click **Edit Content**.



2. Activate **Relocate Block Ref Point**. The default reference point appears, as a yellow circle.



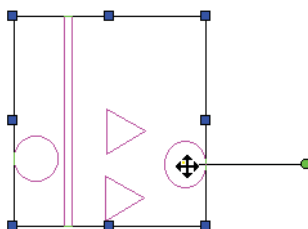
- Click to specify the new location for the reference point.



- Select **Finish to Edit Block**, or click **Finish Edit Content** in the Blocks Palette.



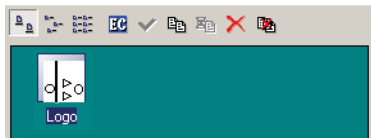
Now when you insert the block by dragging, you will drag it by the new reference point.



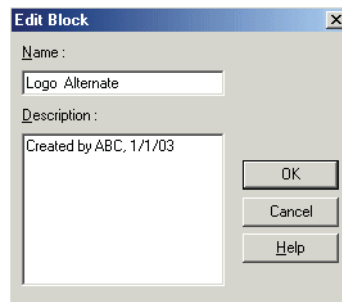
**NOTE:** For any instance of a block you can move its reference point and rotation bar. (See "Changing the Reference Point" on page 199 or "Changing the Rotation Bar" on page 200.) However, this change only affects the block itself and any copies of it. New instances of the block will use the block's defined reference point.

## Changing a Block Name and Description

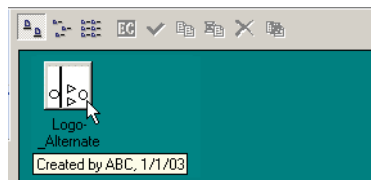
- To change a block name or add a description, double-click the block icon in the Blocks Palette.



- Make the necessary changes in the **Edit Block** window.



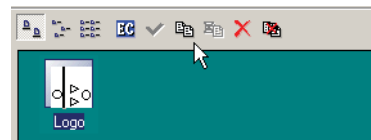
- After the changes are made, any description you add will be displayed as a tool tip on the block icon.



## Copying a Block

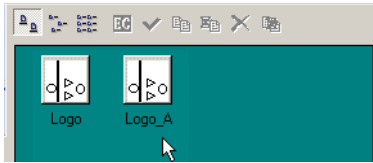
This section is on copying a block to create a new block. To create copies of the same block in the drawing, you can use any of the standard editing tools. See "Copying Objects in Select Edit" on page 203 or "Copying Objects" on page 206.

- To copy a block, select it in the Blocks Palette and click **Copy**.



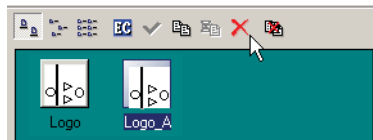
- Depending on the **Auto-Naming** settings, a copy will be created with a generated name, or you will be prompted for a name. See "Auto-Naming" on page 289.

3. The new block appears in the Blocks Palette, and can be edited like any other block. See "Editing a Block" on page 295.



## Deleting a Block

1. To delete a block, select it in the Blocks Palette and click **Delete**.

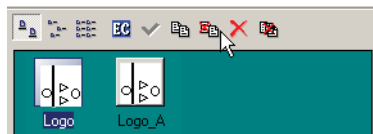


2. In the **Delete Block** window, click one of the following:
  - **Remove all references:** Deletes all instances of the block but leaves the block in the palette for future use.
  - **Remove block with all references:** Deletes the block and all instances. Be careful when using this option, because it cannot be undone.

## Replacing Blocks

One convenient feature of blocks is that they can easily be replaced by other blocks. You can replace all blocks in the drawing, or just selected blocks.

1. To replace instances of a block, select the block in the Blocks Palette and click **Replace References**.



2. In the **Replace Block Reference** window, select the replacement block. You can also choose to replace all instances in the entire drawing, or just those in the active space (visible).
3. Click OK to implement the change.

To replace an instance of a single block, open its **Properties** to the **Block Insertion** page. Use the **Replace with** button to replace it with another block.

## Exploding a Block

To break a block into its constituent parts, select it, then select **Explode**, or click the **Explode** icon.



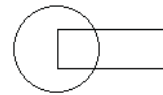
If you explode a block that contains nested groups or blocks, the nested groups will remain intact. Each sub-block must be exploded separately. See "Exploding Objects" on page 247.

## Block Attributes



A block attribute is AutoCAD- informational text associated with a block, that you can enter whenever you insert a block. TurboCAD reads and displays block attributes from AutoCAD drawings (DWG) and DXF files.

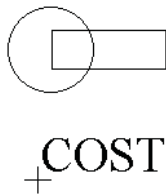
1. Create the objects that comprise the blocks.



(You can also add a block attribute after a block has been created, in **Edit** mode. This is done the same way as adding another geometric object. See "Editing a Block" on page 295.)

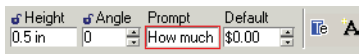
2. Select **Block Attribute Definition**. Select the start point for the text, preferably on or near the block objects.

3. Type the “tag” name for the block attribute, such as “COST.” This name is used to uniquely identify the attribute within the block, since more than one attribute can be created. If the drawing will be sent to AutoCAD, do not use spaces (use underscores instead).



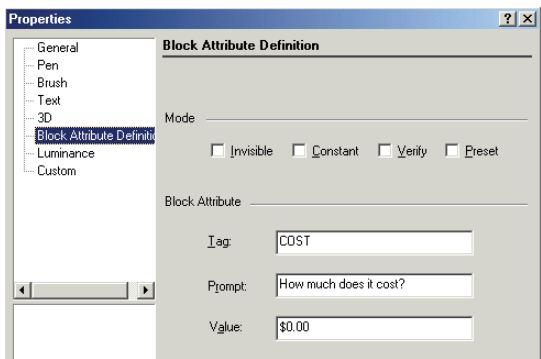
NOTE: This tool works like the text tool, in terms of alignment and local menu options. See "Inserting Text" on page 326.

4. Enter the prompt and default value in the Inspector Bar, or you can enter these properties later. For example, the **Prompt** can be “How much does it cost?” and **Default** can be \$0.00.



5. Press Enter to finish the definition. You can create multiple attributes, such as Part Number, Owner, etc.

Once the block attribute is created, you can open its **Properties** to add or change the **Tag**, **Prompt** and **Default** values, as well as **Mode**.



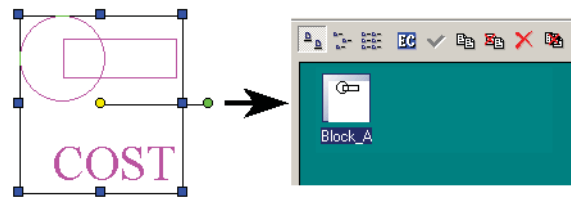
**Mode:** Affects how the dialog appears when the block is inserted:

- **Invisible:** The text of the block attribute is not displayed in the drawing. You can see the information in the Selection Info Palette or edit it by displaying the block's properties.

WARNING: When you explode a block containing invisible block attributes, the information is lost.

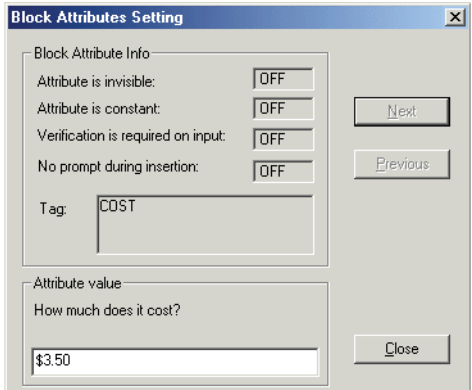
- **Constant:** The attribute value is fixed and unchangeable. It is shown to you during block insertion. Exploding the block will turn the block attribute into text that can then be edited.
- **Verify:** Prompts you to verify that the attribute value is correct when you insert the block. In AutoCAD, the attribute value is shown for your verification during block insertion if the AutoCAD variable ATTDIA is set to 0. This flag has no effect in TurboCAD as it takes place in AutoCAD when the variable ATTDIA is set to 1.
- **Preset:** Sets the attribute to its default value when you insert a block containing a preset attribute. In AutoCAD, you are not prompted for a value if the AutoCAD variable ATTDIA is set to 0. This flag has no effect in TurboCAD as it takes place in AutoCAD when the variable ATTDIA is set to 1.

When the block attributes are defined, simply include them in the selection of objects that will make up the new block.



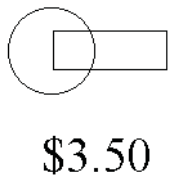
### Setting Block Attributes

When you insert a block that has one or more attributes, a window will appear that gives the prompt and offers the default value. For example, the prompt says "What does it cost?" the default value is \$0.00. The value can be changed to another value, in this case, \$3.50.

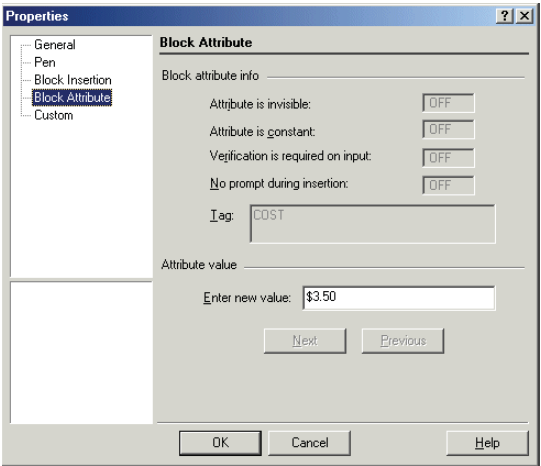


For multiple attributes, use the **Next** and **Previous** buttons to set values for all of them.

When the block is inserted, the value is included with the objects.



When a block has attributes defined, its Properties will contain an additional page - **Block Attribute**. You can edit each attribute value in this window, using the **Next** and **Previous** buttons to scroll through multiple attributes.



### Sync Attributes

Sometimes it is necessary to add attributes to blocks after many instances of the blocks have already been inserted into the drawing. In this case the new attributes are not automatically added to the prior insertions. The Sync Attributes button will add the new attributes to the older blocks.



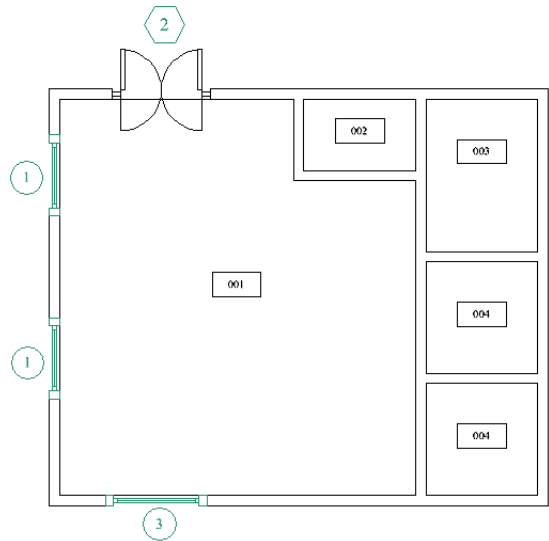
The values for these newly synced attributes within the older blocks will be the default value you assign to the attribute when you create it.

### Extracting Block Attributes

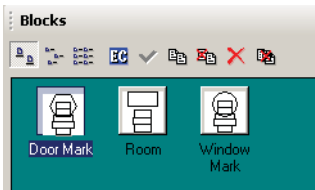
If you attach attributes to blocks, you can extract these attributes and display them in a table. You can also export the attributes to a report in an external file.

NOTE: If you want to create a schedule for architectural elements in your file (walls, slabs, windows, or doors), without defining block attributes, see "Schedule Styles" on page 592.

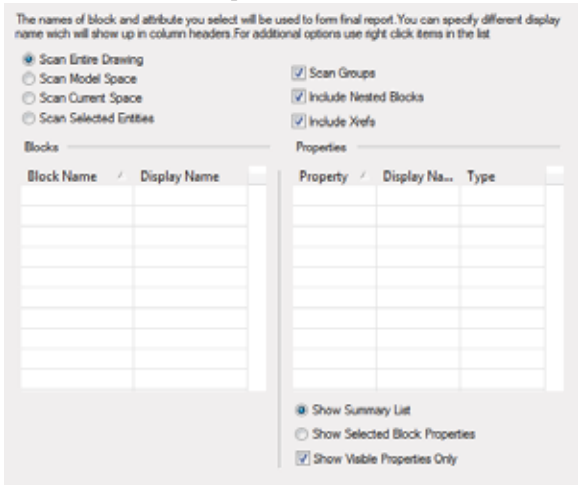
This example has three blocks used to mark windows, doors, and slabs.



Here are the three blocks in the Blocks Palette.



1. Select **Extract Attributes**. In this window you can select the blocks and attributes that will be included in the schedule or report.

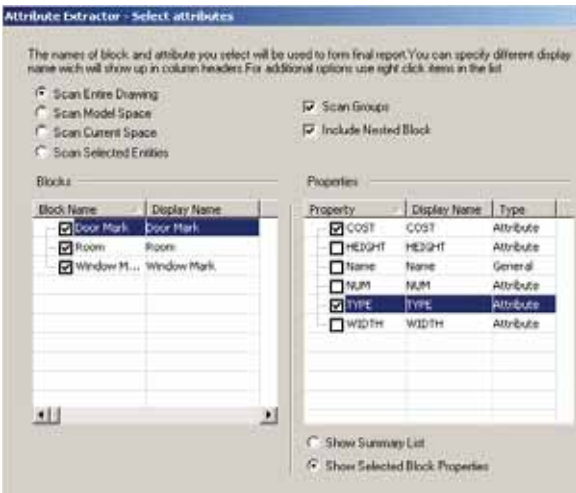


NOTE: You can re-order a a column by dragging its header to the new location.

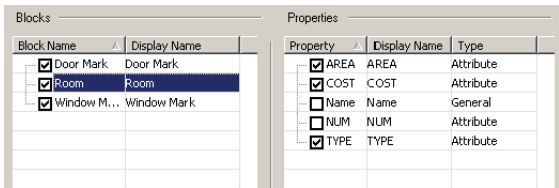
- **Scan Entire Drawing:** Attributes will be extracted from all paper spaces and model space.
- **Scan Model Space:** Attributes will only be extracted from model space.
- **Scan Current Space:** Attributes will be extracted from the current model space or paper space.
- **Scan Selected Entities:** Attributes will be extracted only from currently selected objects.
- **Scan Groups:** If any groups contain blocks, these blocks will be scanned for attributes.
- **Scan Nested Block:** If blocks contain nested blocks, these nested blocks will be scanned for attributes.
- **Include Xrefs:** The content of Xrefs will also be scanned.
- The **Blocks** list contains all blocks that have attribute definitions. The **Properties** list all attributes found for the blocks checked in the **Blocks** list.



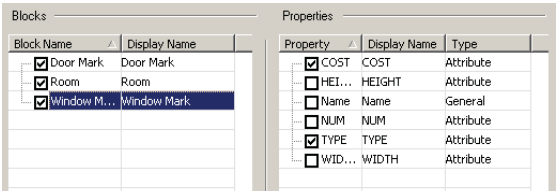
- **Show Summary List:** The **Properties** list contains all attributes for all blocks checked in the **Blocks** list.
  - **Show Selected Block Properties:** The **Properties** list contains attributes only for the block that is currently checked in the **Blocks** list.
  - **Show Visible Properties Only:** If selected only attributes that are visible will be shown.
2. You can select attributes for each block that will be included. For example, click **Show Selected Block Properties** at the bottom, and select the “Door Mark” block. Check only the “COST” and “TYPE” attributes.



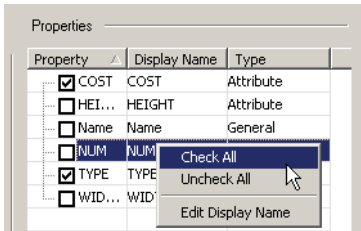
3. Select the “Room” block and check “AREA,” “COST,” and “TYPE.”



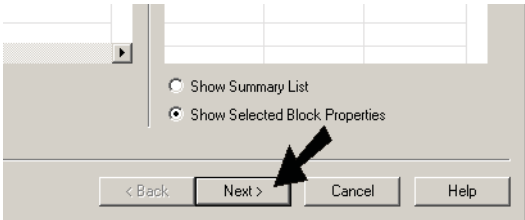
4. Select the “Window Mark” block and check “COST” and “TYPE.”



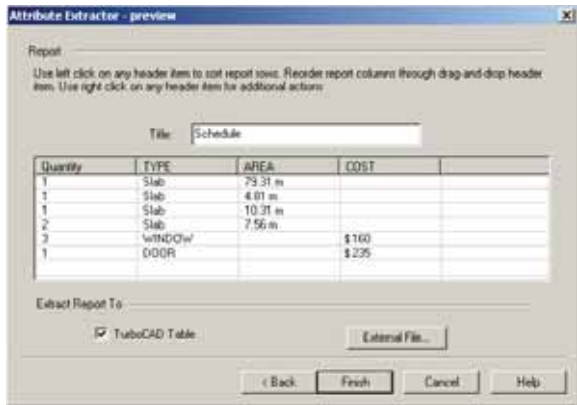
You can right-click on any field under **Blocks** or **Properties** to get a popup menu in which you can check or uncheck all, or change the display name.



5. When the blocks and properties are defined, click **Next**.

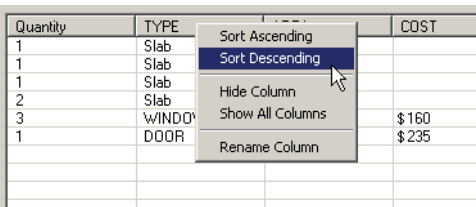


TurboCAD scans the file, and the **Preview** window displays the results.

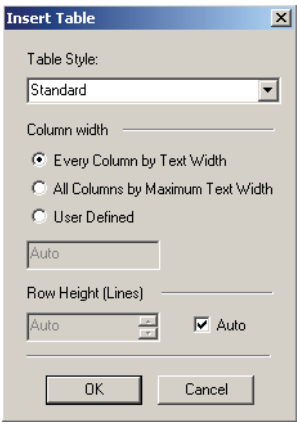


If **TurboCAD Table** is checked, the report will be inserted into the file. If you want to export the results, click **External File**.

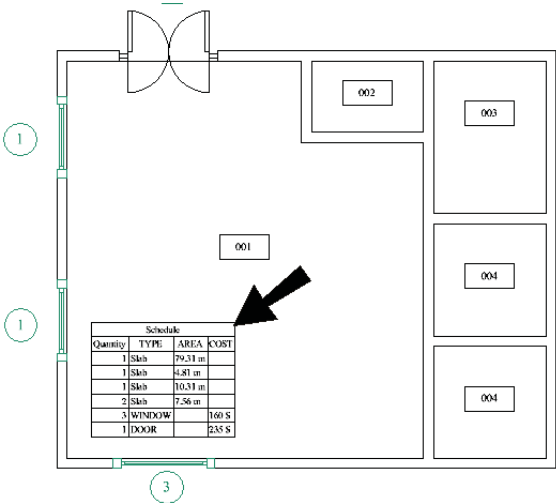
You can click on any column header to change the sorting order, or hide or rename a column.



6. Click **Finish**. If the table is to be inserted into TurboCAD, you will see the **Insert Table** window. Here you can define the column and row sizes.



7. Click OK, and then click where you want to place the table.

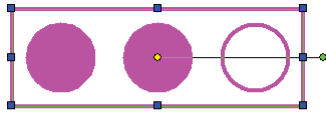


You can make changes to the table formatting in the Selection Info palette.

## In Place Editing of Groups and Blocks

You can edit groups or blocks in place within the drawing.

1. Select the block or group.



2. Right click to open the local menu and select Edit tool. All other elements in the drawing will fade-out, while the selected entity remains clear.
3. Proceed by making you desired changes to the object. This can include changing properties, moving geometry, adding geometry, editing geometry, and deleting geometry.
4. If you are editing a block, other instances of the block will show the changes you are making simultaneously.



5. To finish, right click and select **Finish block/group editing**, from the local menu.

Note that Block attributes that are edited, added or deleted will not be updated in exiting block insertions, including the one that you selected. Only the “Original” block in the palette will reflect attribute changes.

## External References

*Available in TurboCAD Pro and Deluxe only*

An external reference (xref) is a kind of a block in that it is stored in the current drawing's block library. However, unlike a block, the objects associated with an xref definition are not stored in the current drawing; they are stored in another drawing file. When you create an xref, the entire contents of this other file are imported as a block. See "Blocks" on page 291.

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**NOTE:** You can also access An external reference via Block palette.

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Xrefs are usually used to display the geometry of a common base drawing in the current drawing, such as a frame. They can be taken from files any formats readable by TurboCAD.

Only files that have objects in Model Space can be added as Xref's.

1. To import another drawing as a block (xref), select **Format / Create External Reference**.
2. In the **External Reference File Location** window, select a file type and locate the desired file.
3. If you want the path to the referenced drawing to be relative to the current drawing (as opposed to an absolute path), check **Return Path as Relative**.
4. The selected file is added to the block library of the current file. You can view it and insert it using the Blocks Palette. However, you cannot edit an xref in the Blocks Palette - you must change the original file.

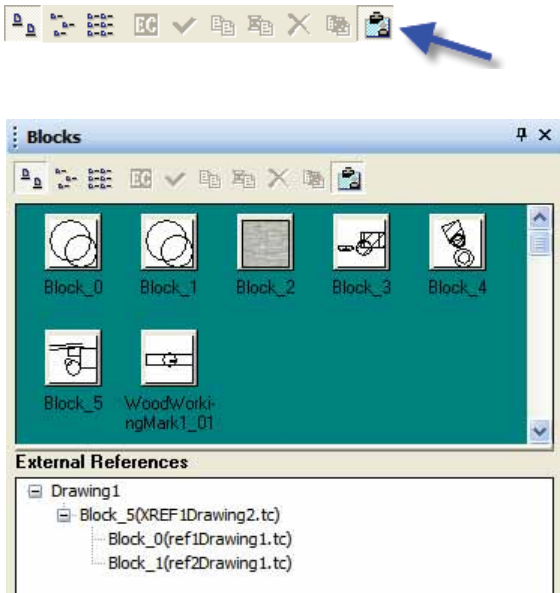
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**NOTE:** If you edit the original file from which the xref was created, the block in the current library will not change. You will need to recreate the xref.

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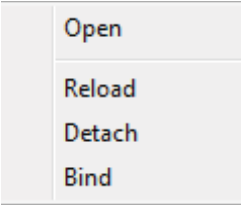
External References Panel

If you click on the **External References** button at the top of the Block palette you will see the **External References** panel at the bottom of the palette.



When xref's are nested in other drawings they are shown in a tree format listed below that referenced drawing.

Right-clicking on any of the xrefs listed in the external References panel will open the following local menu:



- Open:** will open the xref in TurboCAD.
- Reload:** will reload the reference file, including any updates.
- Detach:** will detach the referenced file, and any insertions of that file in the drawing will be deleted. You cannot detach nested xrefs. You must open the file to which they are attached to remove them.

**Bind:** Bind embeds the selected XREF as a Block in the drawing. All attachment to the external drawing is lost.

**VISRETAIN:** Through the Design Director it is possible to edit the various properties of the layers within an XREF. These changes do not affect the original drawing. Even if the external referenced drawing is altered these layer changes will be retained. However, if the XREF is reloaded the changes will be lost. It is possible to disable this feature by changing the \$VIZRETAIN variable through the DCExplorer Palette.

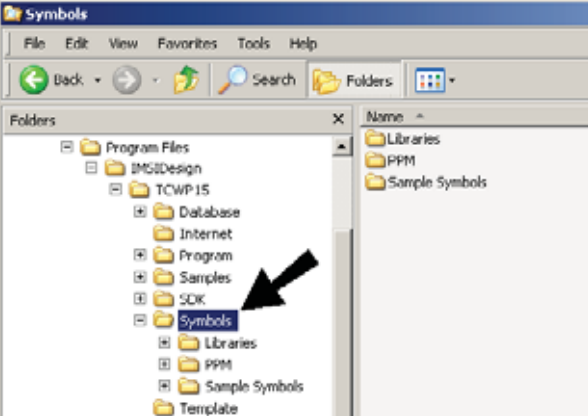
**Exploding XREFs:** When an XREF has been bound to a drawing it becomes a block. Instances of that block in the drawing can be exploded so that you can edit the geometry directly in the drawing

.Library

The TurboCAD library is where you can access symbols and parts. Symbols and parts, like blocks and groups, are objects available for repeated use. But while groups and blocks are internal to a drawing, library items are external files. Generally, each symbol or part is stored as a file, and libraries of files are stored in specified library folders. Any Windows folder can be recognized as a library folder, and any vector drawing file (not just TurboCAD files) can be used as a symbol.

A symbol or part is placed into a drawing as a group; it has a unique name and is selected and manipulated as a single object. Because they are placed as groups, each instance of a part or symbol creates additional drawing data, and inserting numerous library items can increase your drawing size significantly. If you need to use many instances of a particular symbol or part, you can reduce your file size by making it into a block. A block is stored internally in a drawing, and references to it are inserted into the drawing rather than the objects themselves. See "Blocks" on page 291.

TurboCAD provides several symbols and parts, organized in folders by category. Each symbol is a separate \*.tcw file, and parts are \*.ppm files. These files are installed, by default, to the “Symbols” folder in the TurboCAD installation folder.

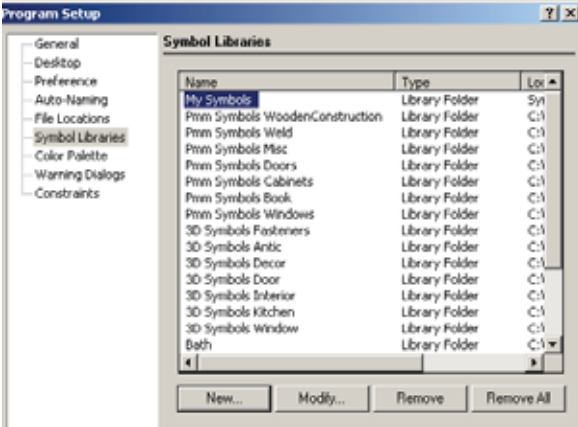


You can modify these symbols (though it is preferable to modify copies and leave the originals intact), and also create and save your own symbols.

Symbol manipulation is done via the Library Palette.

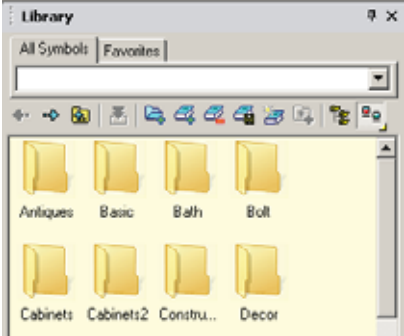
### Library Folders

1. To see which folders will appear in the Library palette, look in the **Symbol Libraries** page of the **Program Setup**.

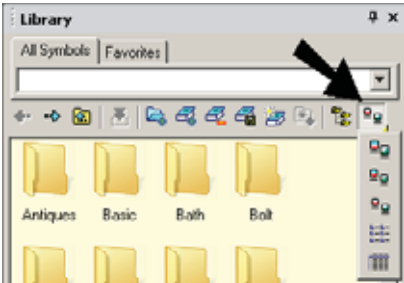


Folders that do not appear here can be added by clicking **New**, or you can load a file from within the Library palette.

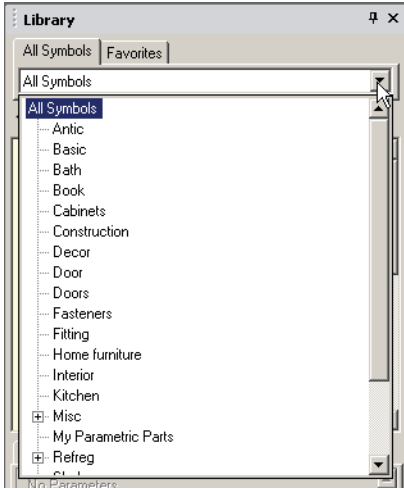
2. Open the Library palette to see the folders it contains.



3. If you want to change how the folders are displayed (icon size, or detailed list), click this icon:

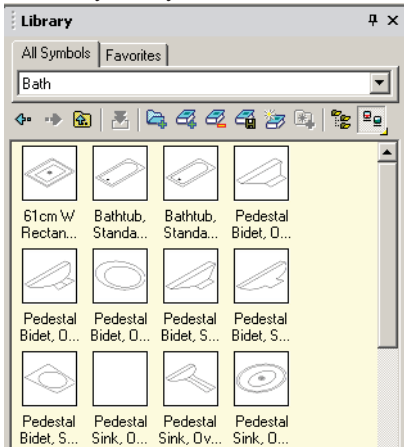


4. You can also view the folder structure in tree form by clicking the drop-down arrow for **All Symbols**.



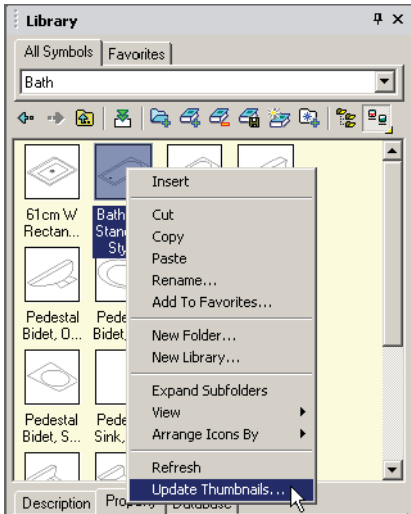
NOTE: The other tab is **Favorites**, where you can store symbols and parts you use often. See "Favorites" on page 308.

5. The library contains symbols as well as parametric parts. Symbols can be 2D or 3D, and are objects saved as their own \*.TCW file. For examples of symbols, open the "Bath" folder. (Your thumbnails may look different, or you may not see thumbnails at all.)

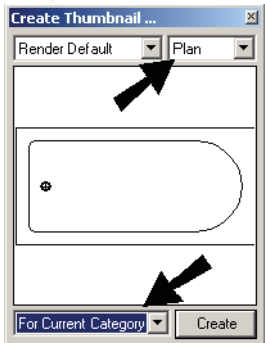


NOTE: If a symbol is not in \*.TCW format, its thumbnail will be a generic icon.

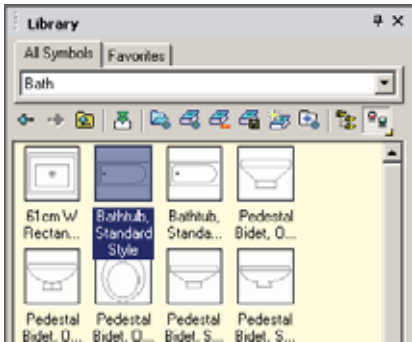
6. To change the way thumbnails are displayed, right-click on one them and select **Update Thumbnails**.



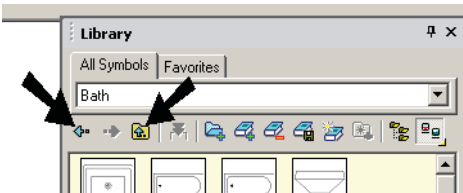
7. The top right field defines the standard view (Plan, ISO, etc.). The bottom left field determines if the thumbnail properties will be applied to only the current thumbnail, all items in the current folder, or all items in the entire library.



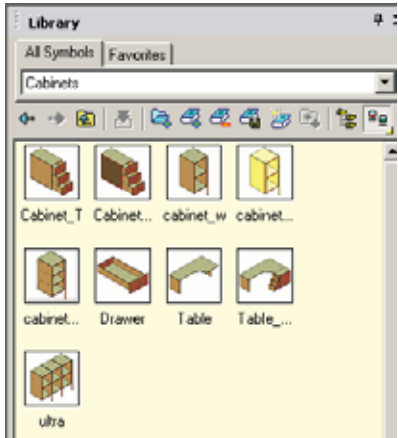
In this example, all thumbnails in “Bath” now are now shown in plan view.



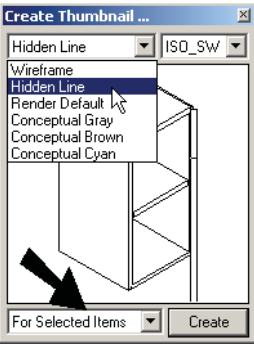
8. To maneuver within the Library’s folder structure, you can use the **Back** and **Forward** arrows, or the **Up** arrow. Go back to root library folder.



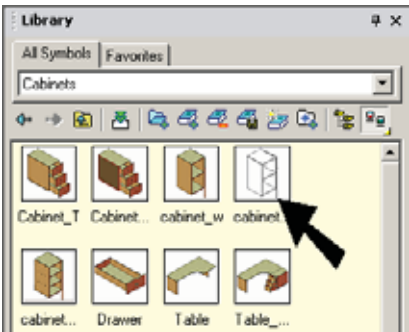
9. Parametric parts and 3D symbols have more thumbnail options. Open the “Cabinets” folder - these are parametric parts created from scripts.



10. Right-click on one of these thumbnails. On the top left field, you can choose Wireframe, Hidden Line, or Default. In addition, you can select one of the “Conceptual” renderings; three colors are available.



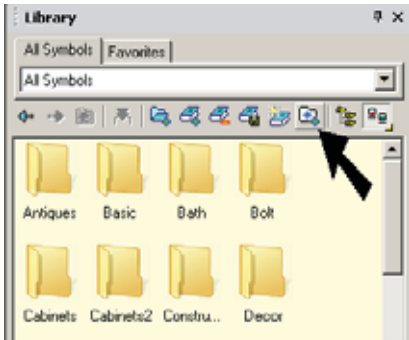
In this example, Hidden Line display is now applied only to the selected thumbnail.



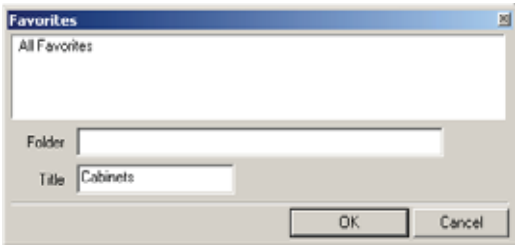
## Favorites

The **Favorites** tab of the Library palette is where you can store symbols and parts you use often.

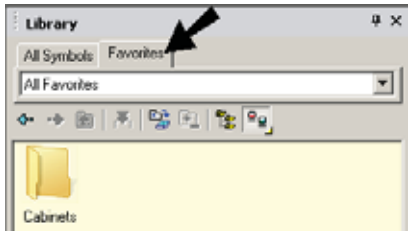
1. You can place an entire folder in **Favorites**. To do this, click the folder, then click **Add to Favorites**.



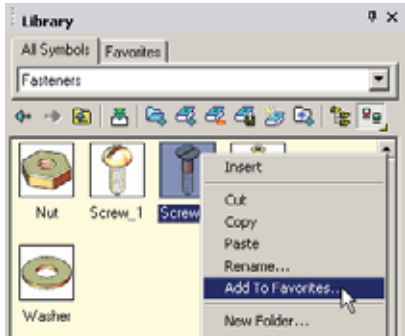
2. If there is no folder currently in **Favorites**, the folder will be placed in the main folder.



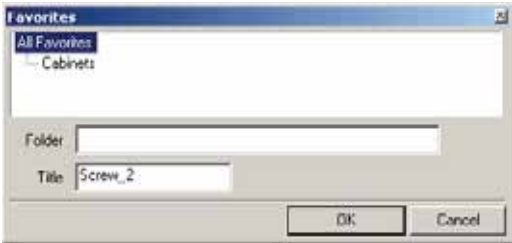
3. Open the **Favorites** tab, and the folder is listed.



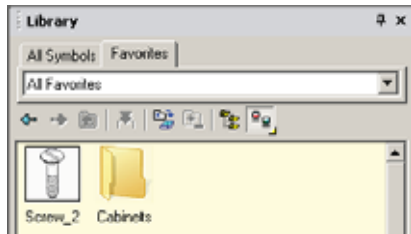
4. You can also add individual symbols or parts to **Favorites**. Return to **All Symbols** and locate the symbols you want to add. Another way to add a symbol is to right-click on it and select **Add to Favorites**.



5. You can choose to add the symbol to the main folder, or to an existing folder within **Favorites**. In this example, the symbol will be placed in the main folder.

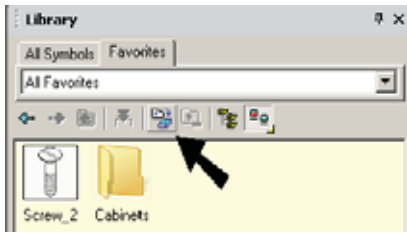


Now **Favorites** contains the single symbol and the folder.





- If you want to save the **Favorites** as a library, click **Save Favorites as New Library**.



## Loading an Individual Symbol into the Library

There are two ways to save an individual symbol - saving the entire file or dragging selected objects directly into the Library.

### Saving a File as a Symbol

If you create a symbol as its own file, simply save the file in a folder already identified as a library folder. Or you can save it to any folder, then load that folder into the Library palette.

For example:

- Create a new door and save the file as “FrenchDoor.tcw” in the “Symbols\Sample Symbols\3D Symbols\Door” folder.
- Open a new file, open the Library Palette, and open the “Door” folder.
- The “French Door” symbol appears in the palette, and you can drag it into the drawing.

### Saving Selected Objects as a Symbol

To save objects as a symbol, select them and create a group.



(It is not essential to make a group, but the symbol will be easier to work with later.)

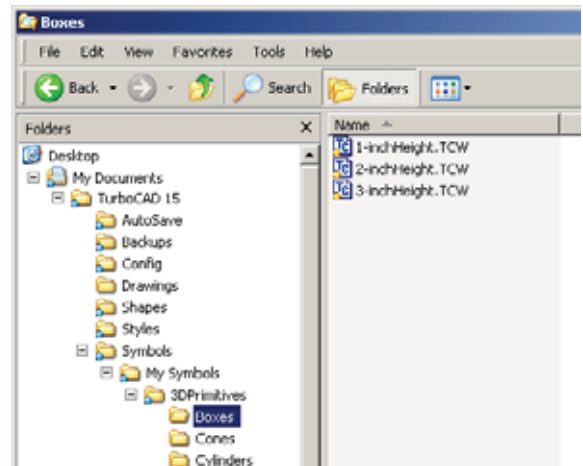
Open the Library palette, open a folder, and drag the objects by their reference point (the yellow circle) into the Palette. You will be prompted for a symbol name.

## Loading Symbol Folders into the Library

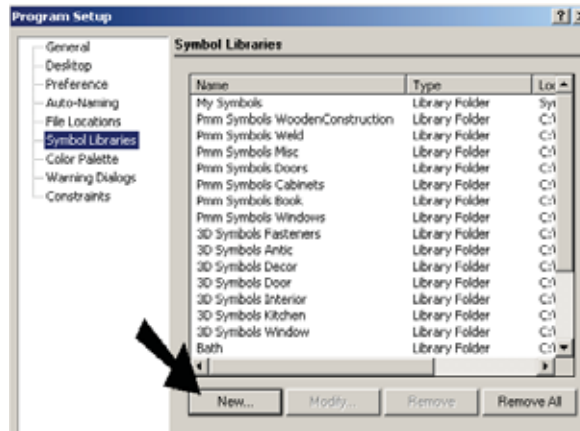
This section explains how take folders of symbols you’ve already saved, and load these folders into the Library.

*NOTE: For loading parametric parts into the Library, see “Loading a Parametric Part into the Library” on page 321.*

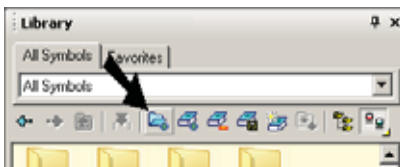
- In this example, a few boxes, cones, and cylinders were created in TurboCAD, and each saved as its own \*.TCW file.
- Here is the folder structure for these symbols. The folder “3D Primitives” was created in “My Documents\TurboCAD\Symbols\My Symbols.” “3D Primitives” has three sub-folders for boxes, cones, and cylinders. Each of these folders has a few \*.TCW files.



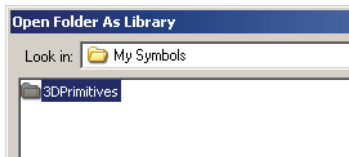
- There are two ways to add “3D Primitives” to the library. One way is to open **Symbol Libraries**, click **New**, and browse to “3D Primitives.”



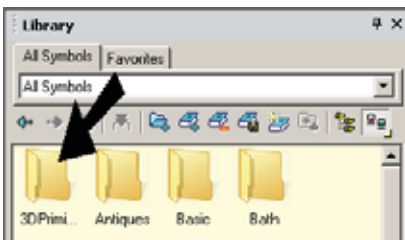
- Another way to load this folder is within the library. Click **Load Folder**.



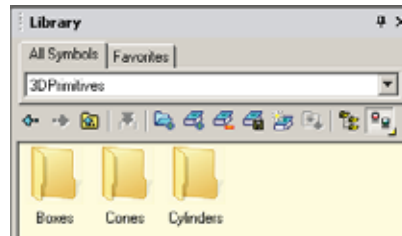
- If you use **Load Folder**, browse to where you saved “3D Primitives” and load it.



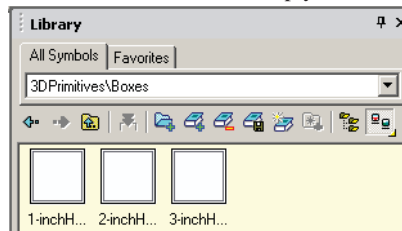
The “3D Primitives” folder now appears in the library.



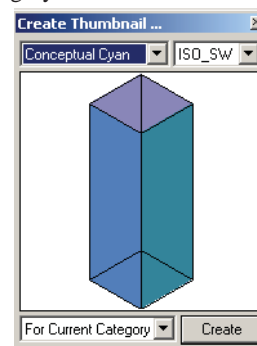
- Double-click “3D Primitives” to see the three sub-folders inside.



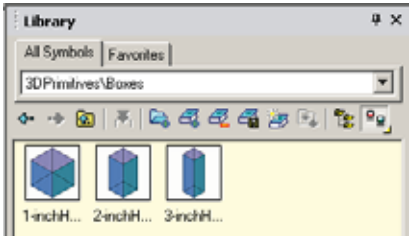
When you first open a new folder that contains symbols, the thumbnails are empty.



- Right-click on one of the thumbnails and select **Update Thumbnails**. In this example, an ISO view and “Conceptual Cyan” are used, and applied to the entire category.



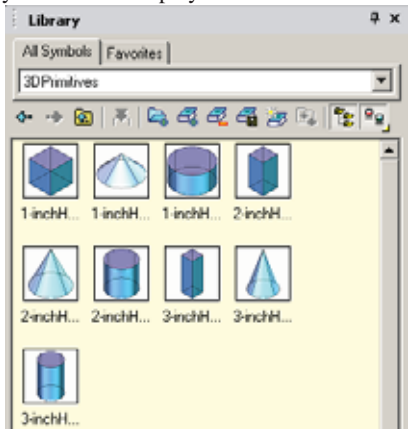
This updates the thumbnails.



8. Use **Up** or **Back** to return to the three sub-folders. If you want to see all of the symbols contained in all three folders, click **Expand Subfolders**.

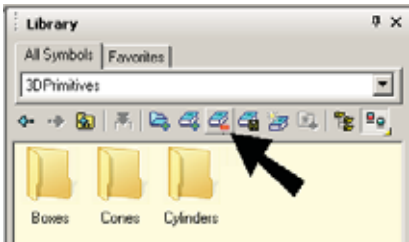


Now all symbols from “Boxes,” “Cones,” and “Cylinders” are displayed.

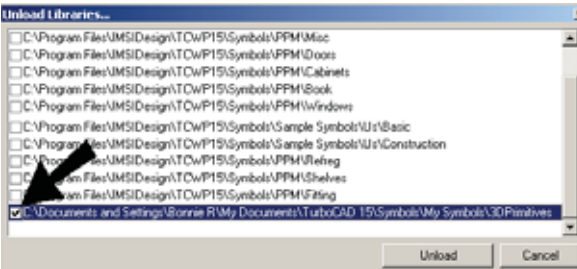


9. To return to the folder structure, click **Expand Subfolders** again.

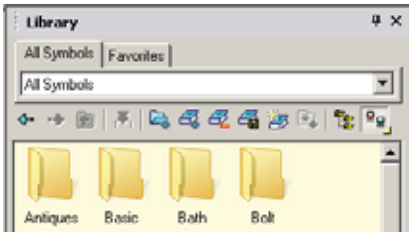
10. To unload a folder, click **Unload Libraries**.



11. Find “3D Primitives” on the list and check the box. Then click **Unload**.



“3D Primitives” is no longer displayed.

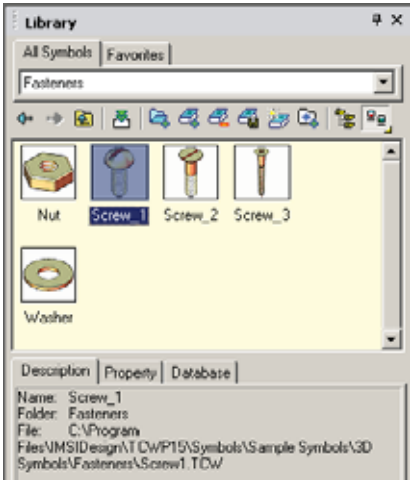


## Inserting a Symbol from the Library into the Drawing

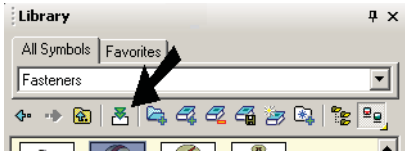
This section explains inserting symbols, which are standard TurboCAD objects saved as their own files.

NOTE: For inserting parametric parts, see "Inserting a Parametric Part from the Library" on page 312.

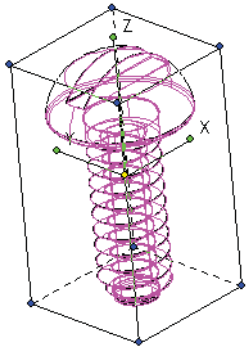
1. For an example of a symbol, open the "Fasteners" folder. Click on one of the symbols, and the **Description** tab at the bottom shows the name and location of the symbol's file.



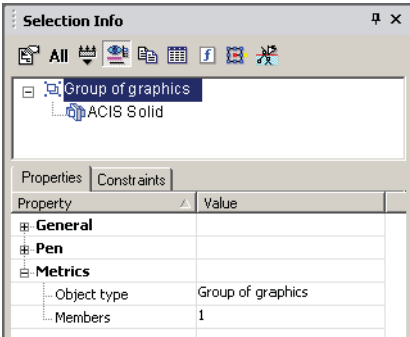
2. There are three ways to insert a symbol. You can click and drag the symbol from the palette to the drawing. Or you can double-click the symbol thumbnail. Or you can click the **Insert Symbol** icon.



The symbol appears in the drawing, in Select Edit mode.



3. In the Selection Info palette, you can see that the symbol is grouped. If you want to edit the symbol, you first need to **Explode** it.



## Parametric Parts

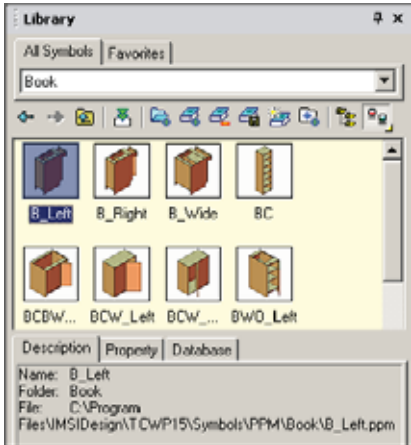
A parametric part is a group of objects that have parameters you can define before or after the part is inserted. For example, you can insert a bookcase whose height and shelf spacing can be defined.

### Inserting a Parametric Part from the Library

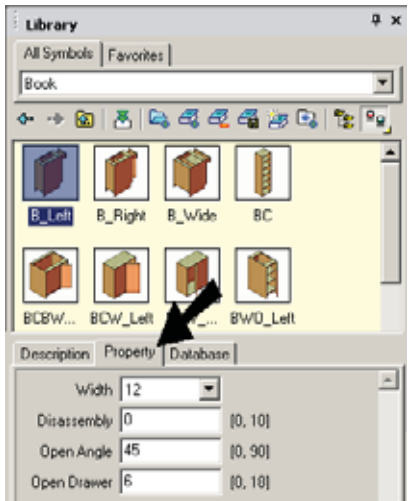
The parametric parts included in the TurboCAD installation were created using scripts. (If you want to view these scripts, see "Parametric Part Script Editor" on page 323.)

NOTE: For details on how to save and insert parametric parts you create yourself, see "Loading a Parametric Part into the Library" on page 321.

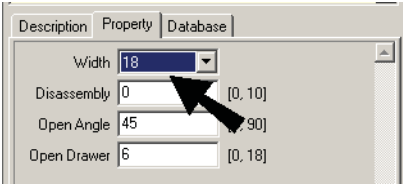
1. For an example of a parametric part, open the "Books" category. Click on one of the parts, and the **Description** tab at the bottom shows the name and location of the part's file.



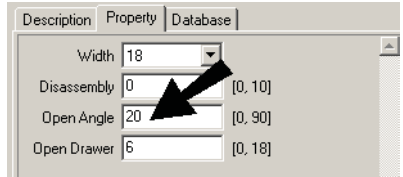
2. Open the **Property** tab. These are the editable fields defined in the part's script.



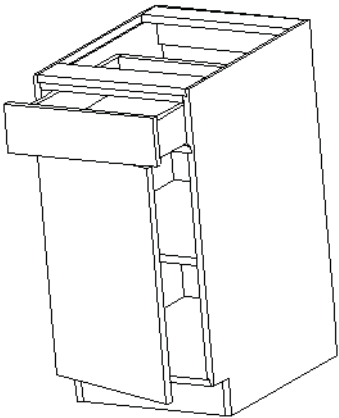
3. The properties with a drop-down arrow contain a list of pre-set, selectable values.



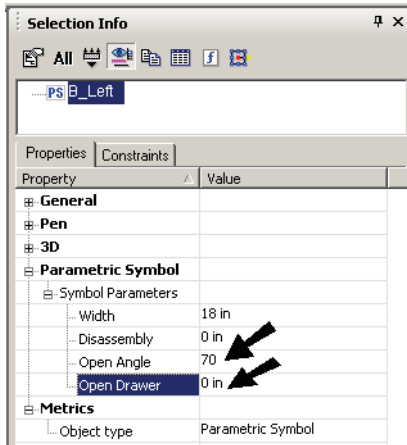
4. Fields that are not pre-set have a defined range of values you can enter. For example, **Open Angle** below must be between 0 and 90.



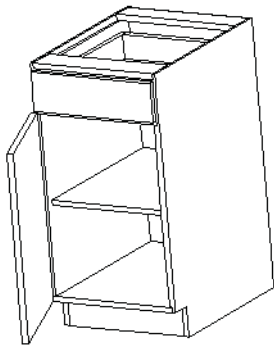
5. There are three ways to insert a part. You can click and drag the part from the palette to the drawing. Or you can double-click the part thumbnail. Or you can click the **Insert Symbol** icon. The part is inserted with the properties set in the palette.



6. You can make changes to the part in the Selection Info palette. This palette shows the part's name, as well as fields for each of the properties shown in the Library palette. In this example, **Open Angle** was increased to 70 degrees, and **Open Drawer** was changed to zero.



Now the drawer is closed, and the door is open wider.



## Creating a Parametric Part

Available in TurboCAD Pro and Platinum only

This section explains how to create a parametric part from within TurboCAD.

NOTE: You can also create a parametric part using a script. For details on creating scripts, check the "Docs" folder of the TurboCAD installation disk.

There are four basic steps in this process:

1. creating the object(s) themselves
2. saving the objects as a part and defining parameters
3. defining relationships between parameters
4. inserting the part

### Step 1: Creating the Object(s)

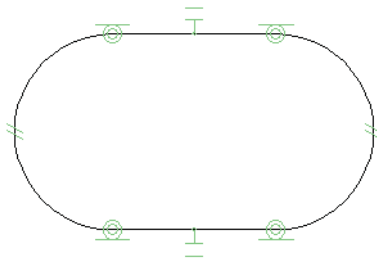
In this example, a part will be created from a polyline with holes, made 3D by extruding it. The objects will be created using different sizes than the actual part that will be saved, to show that parameters can be used for accurate sizing.

1. In this example, **Auto Constraints** is turned on. This is so that dimensions can be used as parameters, and so that geometric relationships between objects will be maintained when parameters are changed.

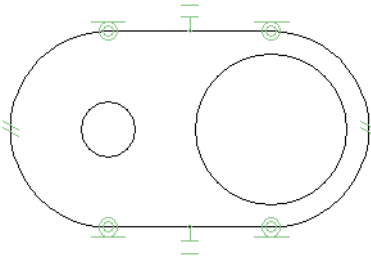


NOTE: For details on constraining objects, see "Auto Constraint" on page 266.

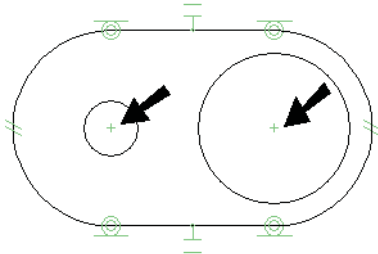
2. Create a polyline using two identical linear segments and two tangent arc segments.



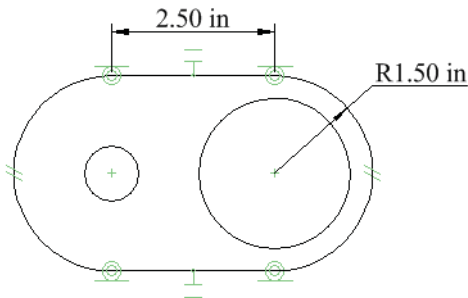
3. Create two circles concentric with the polyline arcs. The circles can be different sizes now, but in the eventual parametric part, the circles will be identical.



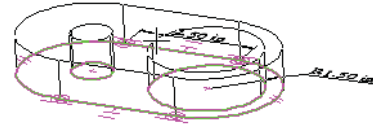
4. Apply **Concentric** constraints so that the circles will remain concentric with the polyline arcs.



5. Add a **Linear Dimension** and a **Radial Dimension** to measure the polyline.



6. Activate **Simple Extrude**, and make sure **Use Compound Profile** is active. Select the polyline, then press Shift and select two circles. Click to define the height, or enter an exact value.

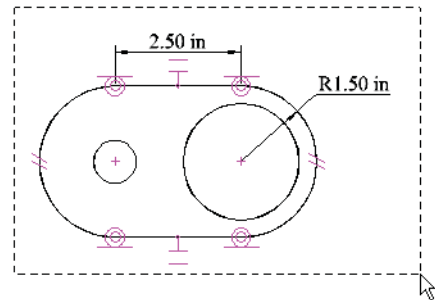


## Step 2: Defining the Part and its Parameters

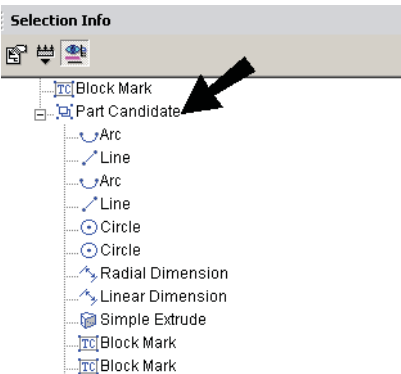
Now a part will be created from this object. There will be five parameters defined for this part:

- H1 = radius of one internal circle
- H2 = radius of the other internal circle
- R = radius of the polyline arc
- W = length of the linear polyline segment
- D = extrude depth

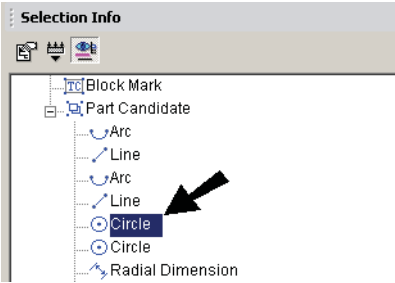
1. Select **Parametric Part Manager**.
2. Use a selection window to select all of the objects.



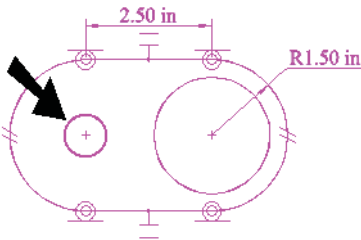
3. The Selection Info palette opens, and the part is temporarily called “Part Candidate.” Expand this item to see what the part contains.



4. The parameters H1 and H2 are for the two internal circles. Highlight the first Circle in the palette.



The circle is also highlighted on the model.



5. The radius of this circle is found under “Metrics,” at the bottom of the palette. Right-click on the **Radius** field.

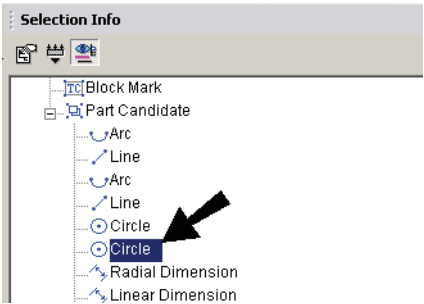
Property	Value
Brush	
3D	
Metrics	
Object type	Circle
Area	0.56
Center	x=2.39 in, y=1.38 in
Radius	0.42 in
Circumference	2.65 in

6. Enter the parameter name “H1,” then click OK.

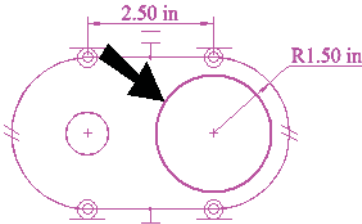


NOTE: You are not restricted to defining only geometric parameters. You can also define material, pen color, line width, etc. as parameters.

7. Next, highlight the other Circle.



The other circle is now highlighted on the model.





8. Right-click on the **Radius** field.

Property	Value
Brush	
3D	
Metrics	
Object type	Circle
Area	4.17
Center	x=4.89 in y=1.38 in
Radius	1.15 in
Circumference	7.24 in
Major radius	1.15 in

9. Enter the parameter name “H2.”

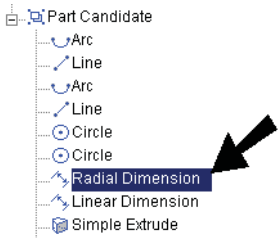
Set Part Parameter

Enter unique name:

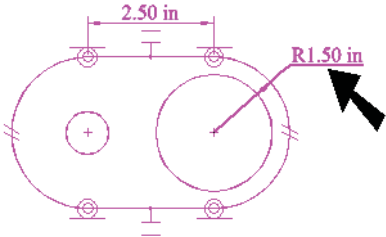
H2

OK Cancel

10. For the next parameter, highlight the Radial Dimension.



The dimension of the polyline arc is now highlighted.



11. The value of this dimension is found under “Constraints.” Right-click on this field.

Property	Value
Alternate Uni...	
Tolerances	
Constraints	
Value	1.5
Variable Name	Var1

12. Name this parameter “R.”

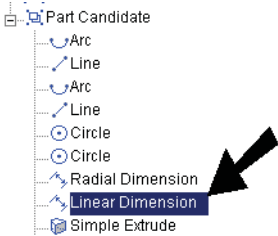
Set Part Parameter

Enter unique name:

R

OK Cancel

13. Next is the Linear Dimension.



14. This value is also found under “Constraints.” Name its parameter “W.”

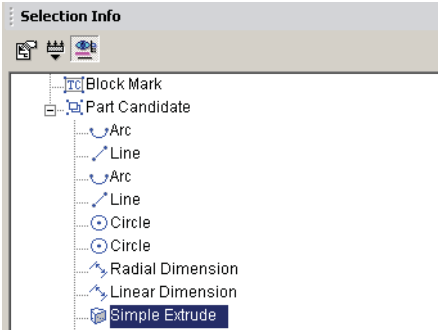
Set Part Parameter

Enter unique name:

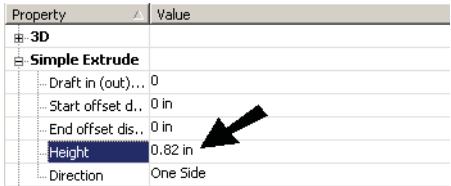
W

OK Cancel

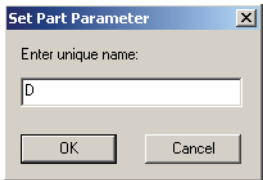
15. The last parameter is the depth of the part. Highlight “Simple Extrude.”



16. This parameter is called “Height” and is found under “Simple Extrude.” Right-click on this field.



17. Name this last parameter “D.”



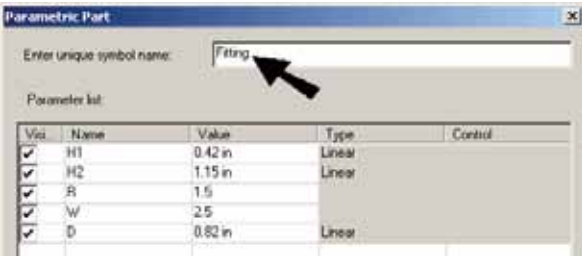
18. Now that all five parameters are defined, select **Finish** from the local menu or Inspector Bar.



NOTE: There are other options in the local menu and Inspector Bar. **Relocate Reference Point** enables you to define the point by which the part will be inserted. For circular parts, **Set Assembly Axis** can be used to set an axis for assembly using **Assemble by Axis** (see “Assemble by Axis” on page 480).

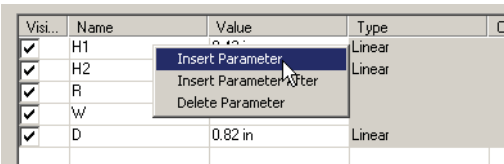
### Step 3: Defining Relationships between Parameters

1. The **Parametric Part** window is now open, in which you can define relationships between the parameters you’ve defined. First, enter a name for the part.

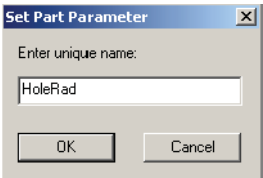


The eventual part will be defined by only three parameters: the hole radius, distance between hole centers, and depth. All other parameters will be functions of these three.

2. The names of the parameters currently listed don’t tell the user exactly what they are for, so we can add parameters with more obvious names. To do this, right-click on the first parameter on the list, and select **Insert Parameter**.



3. Name this parameter “HoleRad.”



4. “HoleRad” is now first parameter on the list. Under “Value,” enter the default radius that will be used when you first insert the part.

Parameter list:				
Visi...	Name	Value	Type	Co
<input checked="" type="checkbox"/>	HoleRad	0.25		
<input checked="" type="checkbox"/>	H1	0.42 in	Linear	
<input checked="" type="checkbox"/>	H2	1.15 in	Linear	
<input checked="" type="checkbox"/>	R	1.5		
<input checked="" type="checkbox"/>	W	2.5		
<input checked="" type="checkbox"/>	D	0.82 in	Linear	

5. Under “Type,” specify “Linear.”

Parameter list:				
Visi...	Name	Value	Type	Control
<input checked="" type="checkbox"/>	HoleRad	0.25	Linear	
<input checked="" type="checkbox"/>	H1	0.42 in	Linear	
<input checked="" type="checkbox"/>	H2	1.15 in	Linear	
<input checked="" type="checkbox"/>	R	1.5		
<input checked="" type="checkbox"/>	W	2.5		
<input checked="" type="checkbox"/>	D	0.82 in	Linear	

6. Create another parameter for the center-to-center distance between the holes. This parameter can be named “CenToCen.” Specify an initial “Value” and “Type.”

Parameter list:				
Visi...	Name	Value	Type	Control
<input checked="" type="checkbox"/>	HoleRad	0.25	Linear	
<input checked="" type="checkbox"/>	CenToCen	3.0 in	Linear	
<input checked="" type="checkbox"/>	H1	0.42 in	Linear	
<input checked="" type="checkbox"/>	H2	1.15 in	Linear	
<input checked="" type="checkbox"/>	R	1.5		
<input checked="" type="checkbox"/>	W	2.5		
<input checked="" type="checkbox"/>	D	0.82 in	Linear	

7. “H1” and “H2” will be have the same value as “HoleRad,” and therefore do not need to be displayed as parameters you can edit when inserting the part. So for “H1,” uncheck the box under “Visible.”

Parameter list:				
Visi...	Name	Value	Type	Control
<input checked="" type="checkbox"/>	HoleRad	0.25	Linear	
<input checked="" type="checkbox"/>	CenToCen	3.0 in	Linear	
<input type="checkbox"/>	H1	0.42 in	Linear	
<input checked="" type="checkbox"/>	H2	1.15 in	Linear	
<input checked="" type="checkbox"/>	R	1.5		
<input checked="" type="checkbox"/>	W	2.5		
<input checked="" type="checkbox"/>	D	0.82 in	Linear	

8. Under “Value” next to “H1,” replace the number with “HoleRad.”

Parameter list:				
Visi...	Name	Value	Type	Co
<input checked="" type="checkbox"/>	HoleRad	0.25	Linear	
<input checked="" type="checkbox"/>	CenToCen	3.0 in	Linear	
<input checked="" type="checkbox"/>	H1	HoleRad	Linear	
<input checked="" type="checkbox"/>	H2	1.15 in	Linear	
<input checked="" type="checkbox"/>	R	1.5		
<input checked="" type="checkbox"/>	W	2.5		
<input checked="" type="checkbox"/>	D	0.82 in	Linear	

9. Do the same for “H2.”

Parameter list:				
Visi...	Name	Value	Type	Contr
<input checked="" type="checkbox"/>	HoleRad	0.25	Number	
<input checked="" type="checkbox"/>	CenToCen	3.0 in	Number	
<input type="checkbox"/>	H1	HoleRad	Linear	
<input type="checkbox"/>	H2	HoleRad	Linear	
<input checked="" type="checkbox"/>	R	1.5		
<input checked="" type="checkbox"/>	W	2.5		
<input checked="" type="checkbox"/>	D	0.82 in	Linear	

10. Make “R” invisible as well, and define its value as twice the hole radius.

Parameter list:				
Visi...	Name	Value	Type	Control
<input checked="" type="checkbox"/>	HoleRad	0.25	Linear	
<input checked="" type="checkbox"/>	CenToCen	3.0 in	Linear	
<input type="checkbox"/>	H1	HoleRad	Linear	
<input type="checkbox"/>	H2	HoleRad	Linear	
<input type="checkbox"/>	R	2*HoleRad		
<input checked="" type="checkbox"/>	W	2.5		
<input checked="" type="checkbox"/>	D	0.82 in	Linear	

11. Make “W” invisible, and set its value equal to “CenToCen.”

Parameter list:				
Visi...	Name	Value	Type	Contr
<input checked="" type="checkbox"/>	HoleRad	0.25	Linear	
<input checked="" type="checkbox"/>	CenToCen	3.0 in	Linear	
<input type="checkbox"/>	H1	HoleRad	Linear	
<input type="checkbox"/>	H2	HoleRad	Linear	
<input type="checkbox"/>	R	2*HoleRad		
<input type="checkbox"/>	W	CenToCen		
<input checked="" type="checkbox"/>	D	0.82 in	Linear	

12. Now the three parameters needed for the part are defined. They are “HoleRad,” “CenToCen,” and “D.” Click OK to close the **Parametric Part** window.

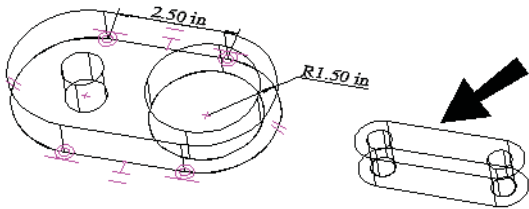
Step 4: Inserting the Part

- 1. Continue in the file in which you defined the part. Select **Insert / Parametric Part**. Select the part you just defined.

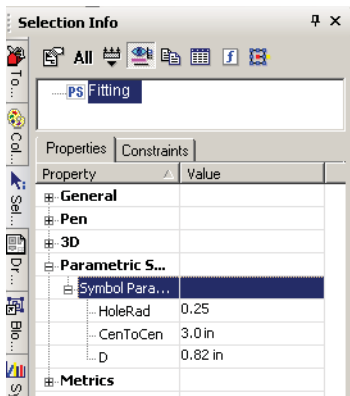


NOTE: **Load from File** can be used to insert parametric parts saved in other files.

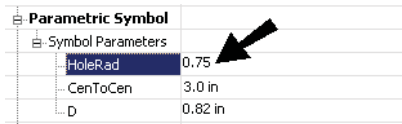
- 2. Click to insert the part. The inserted part is shown to the right of the original geometry. Note that the two holes are equal, and that the polyline arc's radius is twice as large as the hole radius.



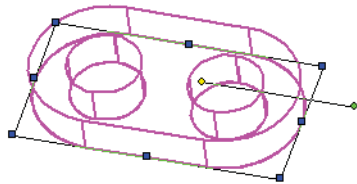
- 3. Select the part, and it appears in the Selection Info palette, with its three parameters available for editing. The initial values are the ones you set in the **Parametric Part** window.



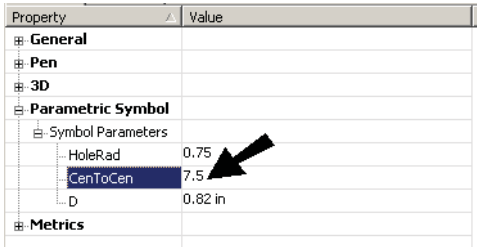
- 4. Increase “HoleRad.”



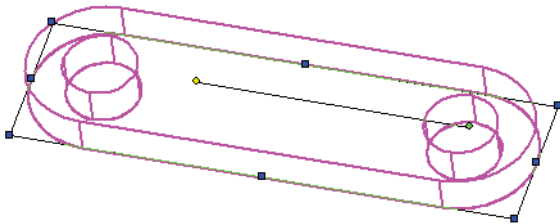
The internal holes updates, as well as the overall radius of the part, which was set to be twice as large as the hole radius.



- 5. Increase the “CenToCen” parameter value.



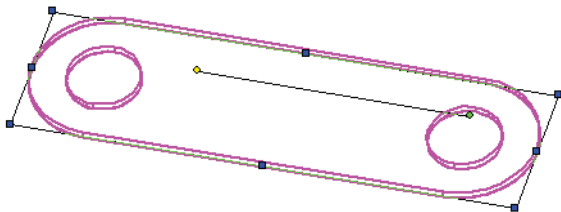
The distance between holes updates, while the radius values remain the same.



6. The last value to change is “D.”

Property	Value
General	
Pen	
3D	
Parametric Symbol	
Symbol Parameters	
HoleRad	0.75
CenToCen	7.5
D	0.15 in
Metrics	

This changes the depth of the extruded part.



### Editing a Parametric Symbol

Before loading the part, you can still edit it. Click the local menu option **Edit Existing Symbol**.



You can edit either the content or the parameters. When finished, click OK and the part is saved with the changes.

### Loading a Parametric Part into the Library

This section explains how to save, and then insert, a parametric part created within TurboCAD. For details on creating parametric parts, see "Creating a Parametric Part" on page 314.

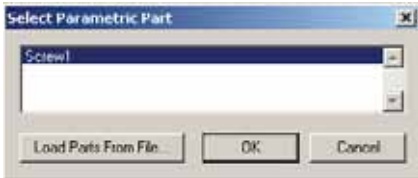
1. In this example, two cylinders are used to create a parametric part representing a simplified screw.



2. Use **Parametric Part Manager** to create a part from these two objects. There are four parameters defined: radius and height for both cylinders

Parametric Part			
Enter unique symbol name:		Screw1	
Parameter list:			
Visi...	Name	Value	Type
<input checked="" type="checkbox"/>	R1	1.28 in	Linear
<input checked="" type="checkbox"/>	H1	0.39 in	Linear
<input checked="" type="checkbox"/>	R2	0.30 in	Linear
<input checked="" type="checkbox"/>	H2	2.49 in	Linear

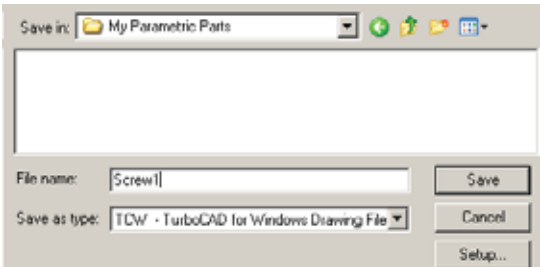
3. When the part definition is completed, erase all objects in the drawing.  
4. Select **Parametric Part** to insert the screw.



The part, and nothing else, is now in the drawing.



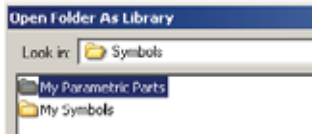
5. Now the file can be saved. In this example, the file is saved as “Screw1.tcw” in the folder “My Parametric Parts.”



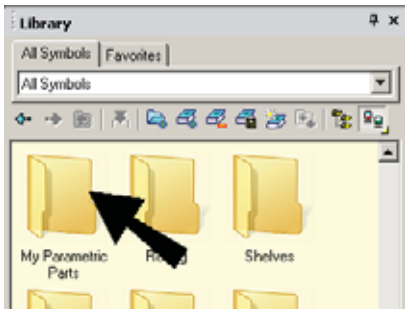
6. Start a new file, and open the Library palette. To load the new folder, click **Load Folder**.



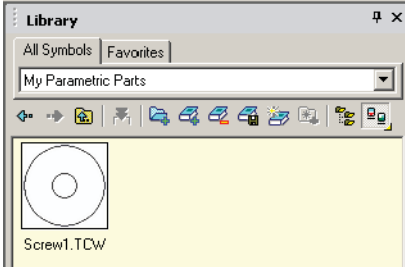
7. Browse to the “My Parametric Parts” folder and load it.



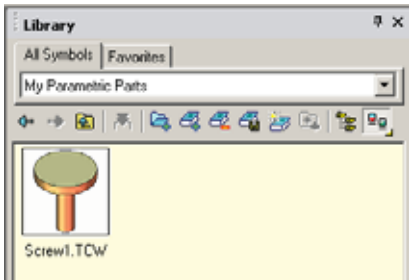
The folder now appears in the library.



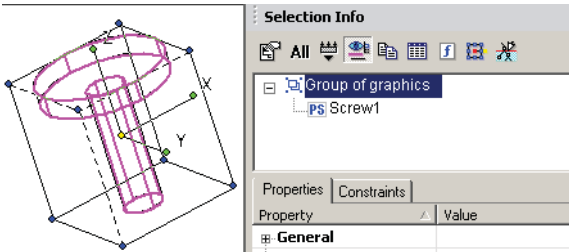
8. Open “My Parametric Parts” to see the “Screw1” part.



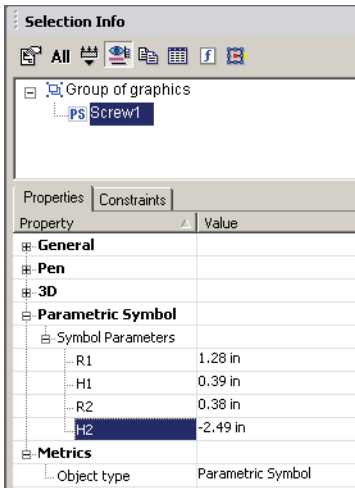
9. Right-click on the thumbnail and update it, if you want.



10. Drag the part from the library into the drawing. Open the Selection Info palette, and you can see that the part is a group.



11. To edit the part, you can **Explode** it, or just highlight its name in the palette and edit its parameters.



NOTE: If you add more parts to the same folder, they will not automatically appear in the library. You need to use **Unload Library** to remove the folder, then use **Load Folder** to load it again.

## Parametric Part Script Editor

Available in TurboCAD Pro and Platinum only

Other than creating a parametric part from within TurboCAD, the other way to create a part is to write a script. The parts provided in folders such as “Cabinet” are created this way.

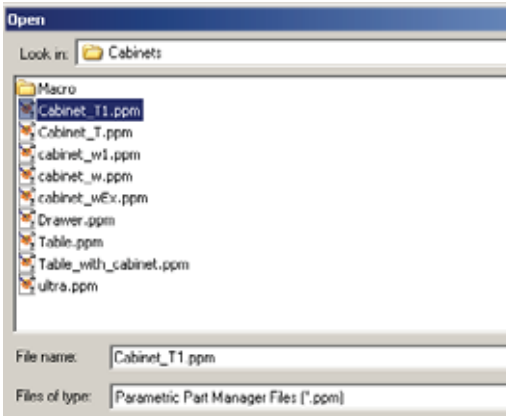
For details on creating scripts, check the “Docs” folder of the TurboCAD installation disk.

If you want to write your own script, or edit an existing script, you can use the Parametric Part Script Editor.

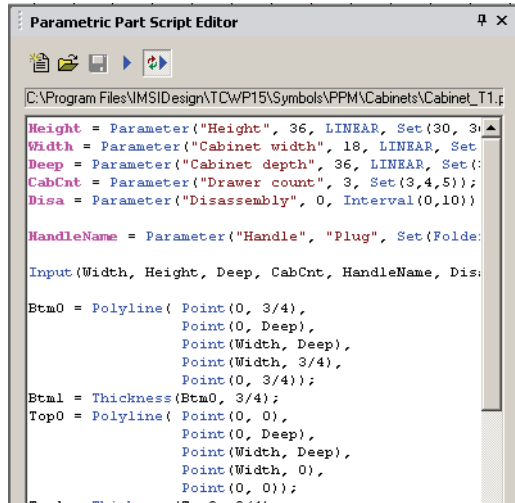
1. To load an existing script into the editor, click **Open**.



2. Locate one of the scripts in the “Cabinet” folder.

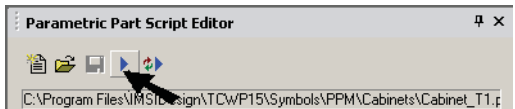


3. The script appears in the editor.



NOTE: Scripts can be edited in any text editor as well.

4. A scripted part can be loaded into the drawing by clicking **Insert PPM Symbol**.



Note: For Details on creating Parametric Part Script see “Customized Programming” on page 671



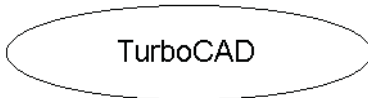


# 9 Annotation

Annotation tools enable you to enhance your drawing with non-geometric objects, including text, dimensions, and hatching.

## Text

These tools enable you to add strings of letters and other characters into your model.



The text tools are available on the **Insert** menu, and can be accessed on the flyout toolbar on the **Drawing Tools** toolbar.



You can also display the **Text** toolbar by right-clicking on any toolbar area and selecting **Text**.



**NOTE:** With this tool you can add single straight lines of text. To add multiple lines in paragraph format, see "Multi Text" on page 328. To create text that follows a curve, see "Text Along Curve" on page 331.

## Text Properties

Certain parameters of text (color, font, and text height) can be set on the **Property** toolbar.



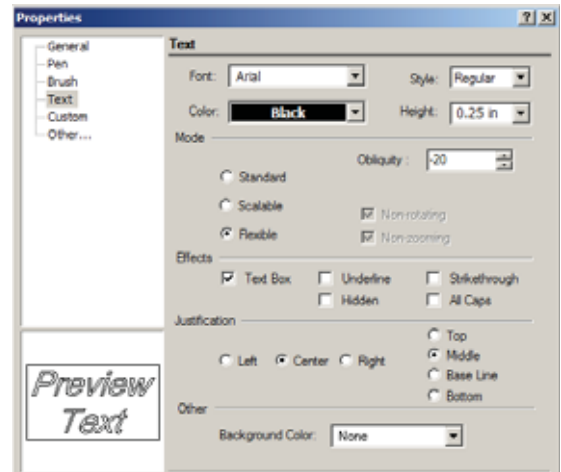
Other parameters (bold, italics, etc.) can be set in the **Text Properties** toolbar, which can be displayed by right-clicking in any toolbar area and selecting **Text Properties**.



**NOTE:** You can use the Manager palette to create multiple styles of text, which set certain basic text properties such as font and height. See "Text Styles" on page 566.

The **Text Properties** toolbar can be used while creating text (i.e. when **Text** is active), and it can be used to edit selected existing text.

You can access all properties of the **Text** tool by right-clicking on the tool icon. For details on changing properties of selected text only, see "Object Properties" on page 79.



**Font:** Fonts can be True Type or \*.shx (AutoCAD compatible). The available TrueType fonts depend on your Windows setup. AutoCAD (\*.shx) fonts are installed by TurboCAD, and are located in the Program\Fonts folder.

: Choose Regular, Italic, Bold, or Bold Italic. These four styles are not available for every TrueType font.

**Height:** You can set the height in either World or Paper units, depending on the current mode - Model Space or Paper Space. Text can be measured in points only if you are using Paper units.

NOTE: There are also options for setting text height in the **Space Units** page of the **Drawing Setup (Options / Space Units)**.

**Mode:**

- **Standard:** Always proportional even if resized.
- **Scalable:** Can be resized non-proportionally.
- **Flexible:** Pen, brush and slant options are available.
- **Non-rotating, Non-zooming:** Text will not change position or be scaled when rotating or zooming. (These will not work if **Old- text output** is checked in **Options / Preference**.)

**Obliquity:** Degree of text slant (slant of each character), available only for **Flexible** mode. A positive value slants to the left.

**Effects:** Turn on any of the following: **Text Box** (rectangular border around the text), **Underline**, **Hidden** (used for notes that you do not want to display), **Strikethrough**, and **All Caps**.

**Justification:** Determines both the text justification and text spill. The options for text spill are **Top**, **Middle**, **Base Line** (base line of font layout) and **Bottom**. Left-justified text is typed to the right of the insertion point; right-justified text is typed to the left of the insertion point, and center-justified text is centered on the insertion point.

TIP: You can get more precise control of the placement of text relative to the insertion point by using **Align**, accessed by right-clicking in the drawing area while the **Text** tool is active. This feature can only be applied to text as it is being drawn; you cannot use it to format existing text.

**Background Color:** The color that appears behind each character.

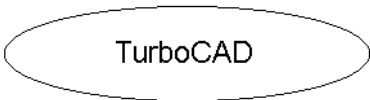
**Inserting Text**

A

1. Set the desired font and other text parameters. See "Text Properties" on page 325.
2. Click on the point where you want to place your text.



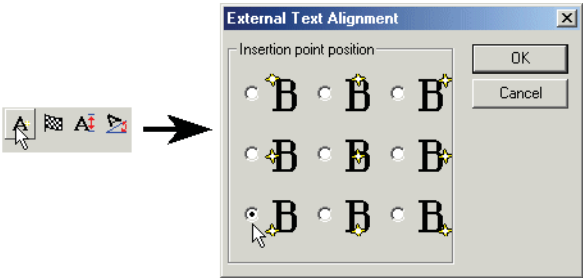
3. Type the text, using the Backspace key to make corrections. Press Enter to add a new text line. To finish, click on the drawing, press Shift+Enter, or select **Finish** from the local menu.



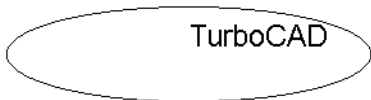
NOTE: By default, the text is centered at the insertion point. You can change this, however, via the **Properties** window, or by using the **Align** local menu option.

Local menu options:

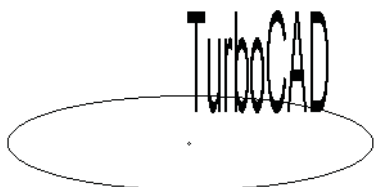
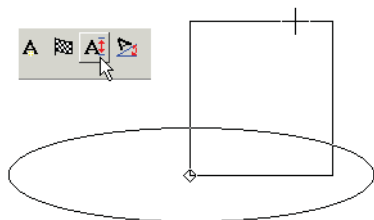
**Align:** Changes the text placement relative to the insertion point. By default, the text is centered horizontally and vertically, but click any option to adjust the placement.



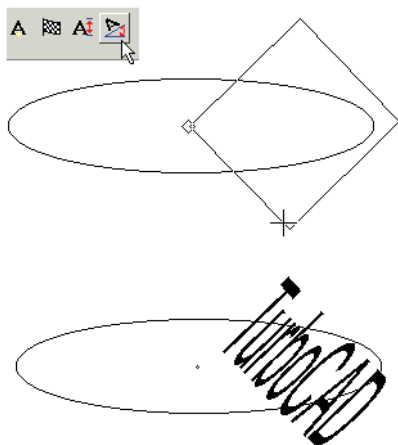
Click OK, and the text placement is updated.



**Height:** Changes the text height. Move the mouse to adjust the height rectangle, or enter a height in the Inspector Bar.

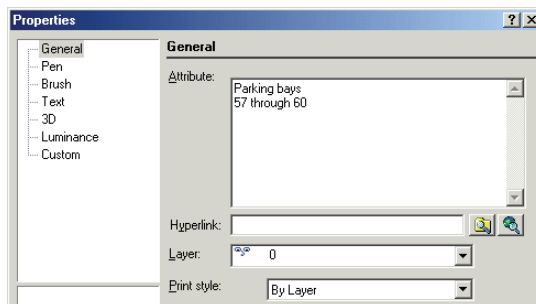


**Angle:** Adjust the angle of the text line (not the text slant). Move the mouse to rotate the text rectangle, or enter the angle in the Inspector Bar.



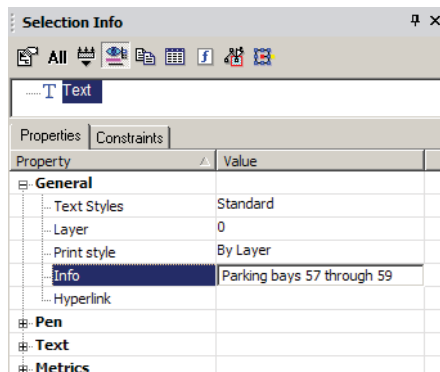
## Editing Text

You can edit existing text by accessing its **Properties** (see "Object Properties" on page 79). Open the **General** page, and edit the text in the **Attribute** field.



You can also edit the content of the text by clicking on the text with the Text tool, or, if in the Selector General properties the Double Click Action is set to Edit content, you can edit the text content by double clicking on the text with the Selector tool.

If you use the Selection Info Palette, you can use the **Edit in Place** tool to edit text directly on the screen. See "Selection Info Palette" on page 191. In this palette, the text is also listed (and can be edited) in the **General** category, next to **Info**.



Exploding Text

You can use **Explode** to explode text into individual characters, and explode characters into polygons and polylines, and explode these into lines. This can be useful if you need to change a very small aspect of text, or if you need a text-shaped polyline to extrude.

See "Exploding Objects" on page 247.

Spelling Check

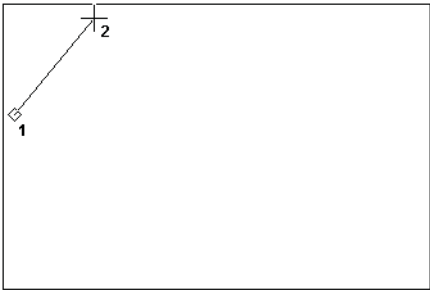
You can check the spelling of selected text only, or you can choose to check all text in the drawing. Set spelling check options by selecting **Tools / Spell Options**.

Multi Text



Creates multi-line paragraphs that fit within a paragraph width boundary. Each multiline text object is a single object, regardless of the number of lines it contains.

- 1. Select the first point of the text, then a second point to define the paragraph width and angle. You can also use the Inspector Bar to enter the width and angle.



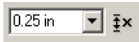
- 2. The **Character** window opens, in which you can define the text properties that will be active for multi text created from now on. (You can change these properties mid-text.) These properties are explained below.



**Font:** Sets the color and font type. The “B” icon creates **bold text**, the “I” icon creates *italics*, and the “U” icon underlines the text.



**Text Height:** Sets the height of the tallest letter. The “X” icon enables you to set the height as a factor of the current height.



Width and Spacing values: **Oblique Angle** is the angle of text slant (the slant of each character). **Width Factor** is the width multiplier of each character.



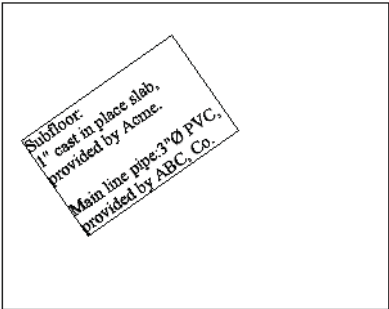
**Symbols:** While entering the text, you can insert a degree, plus/minus, or “phi” (angle) symbol. The last icon is used to insert a fraction. The center three icons are used for top, center, or bottom **Justification**.



**Import / Save:** Once text is created, you can **Save** it as a .txt file. This text can later be inserted elsewhere by clicking **Import**.



- 3. To create the text, simply start typing. The text will start where you defined the first text point. You can enter line breaks manually, and breaks will be created by word-wrapping, according to the paragraph width you set.



If you want to change text properties (font, height, etc.) while typing, make the change in the **Character** window and continue entering text. You can also enter symbols by clicking the relevant icon in the **Character** window.

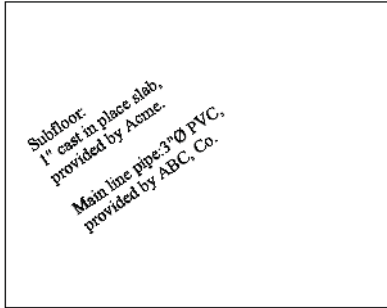
While entering text, you can use the Backspace key or the **Undo** and **Redo** icons.



4. In the Inspector Bar or local menu, select **Finish** when the text is complete.



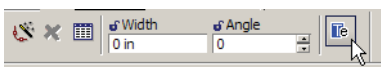
The text is complete, and the preview box disappears.



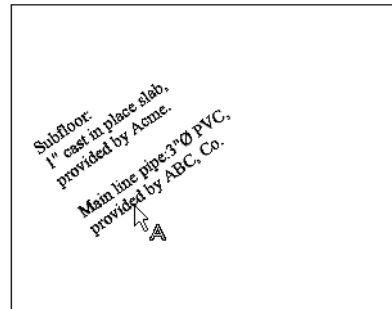
## Editing Multi Text

You can edit text before or after it is created by editing it on-screen:

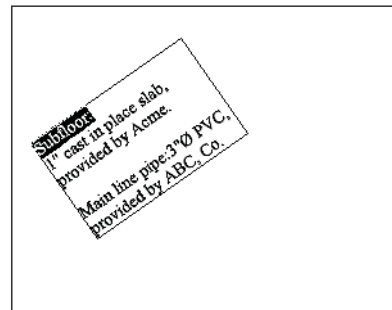
1. You can also edit the content of the multi-text by clicking on the multi-text with the Multi Text tool, or, if in the Selector General properties the Double Click Action is set to Edit Content, you can edit the multi text content by double clicking on the multi text with the Selector tool.
2. You can also use the old mode of editing Multi line text. Make sure you are in **Multiline Text** mode (**Draw / Text / Multi Text**) and select **Edit Text** from the local menu, command line, or Inspector Bar.



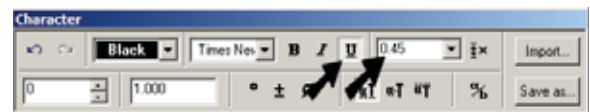
3. The cursor appears as the letter “A”; select the text you want to edit.



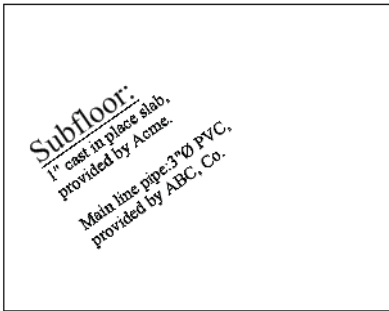
4. Edit the text as you would in any text editor. To make a change to a specific string, such as the first word in this example, highlight that string.



5. Set the properties for that string in the **Character** window. In this example, the text was underlined and its height was increased.

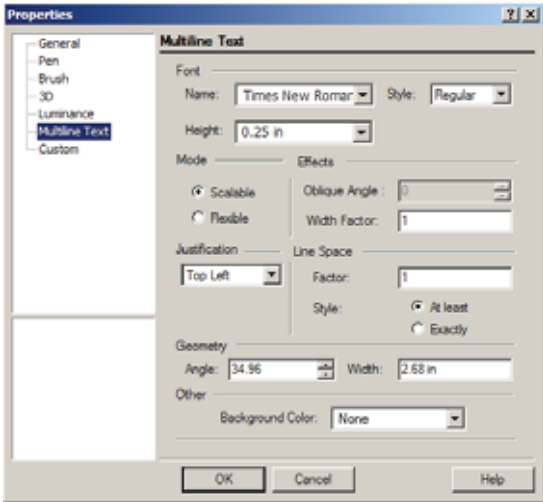


6. Select **Finish** to implement the change.

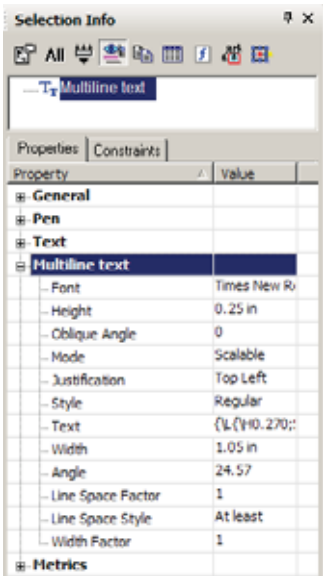


Multi Text Properties

If you open a multiline text object's **Properties** (see "Object Properties" on page 79), there are certain parameters of the text you can edit. To edit the text itself, and for all properties not available on this window, see "Editing Multi Text" on page 329.



The properties also appear in the Selection Info palette, when the multiline text is selected. All parameters can be edited, except for the text string itself.



- Font:** Sets the font name, style, and height.
- Mode:** Select **Scalable** for text that can be re-sized non-proportionally or **Flexible** if you want pen, brush and slant options available.
- Oblique Angle:** Available for **Flexible** mode only, sets the slant of the text (slant of each character).
- Width Factor:** Sets the width of each character.
- Justification:** Adjusts the text relative to its paragraph box.
- Factor:** Sets the distance between lines.
- Style:** Select **At Least** to automatically set the line size relative to the largest character of a line. Select **Exactly** to keep all lines the same size.
- Angle:** Sets the angle of the paragraph.
- Width:** Sets the width of the paragraph.

## Text Along Curve

Available in TurboCAD Pro and Platinum only



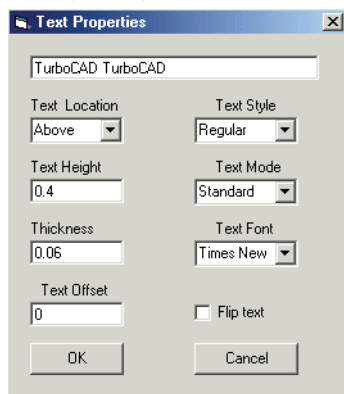
NOTE: You can display the **Special Tools** toolbar by right-clicking in any toolbar area and selecting **Special Tools**.

Creates text that follows a curve.

1. Select the curve, picking the center of the desired text.



2. In the **Text Properties** window, type the text string and set other properties such as font, height, and thickness. The **Text Location** field enables you to place the height above, below, or centered on the curve.



NOTE: For definitions of other properties, see "Text Properties" on page 325.

3. Click OK, and the text is created.



To edit curved text, open its **Properties** window (see "Object Properties" on page 79). Open the **Other** page, click **Text Properties**, and click **Go To Page**.

This opens the **Text Properties** window, in which you can change any parameters, or the text string itself. If the text was created upside-down, click **Flip Text** to right it.

## Dimensions

You can display the **Dimensions** toolbar by right-clicking in any toolbar area and selecting **Dimensions**.



TurboCAD contains a variety of dimension tools that you can use to display the measurements of lines and angles.

Dimensions consist of three basic components:

- Dimension lines, with arrows pointing to either end of the dimension. A linear dimension can have an interior dimension line or two exterior dimension lines. Exterior dimension lines can be supplemented with an optional interior line.
- Extension lines, which connect the dimension line to the object being dimensioned. Extension lines have optional line segments that continue the extension outward beyond the dimension line (extensions to the extension).
- Dimension text, typically displaying the distance being dimensioned in World units.

NOTE: For information on editing dimensions, see "Editing Dimensions" on page 230. To create a group of dimensions automatically, see "Auto Dimension" on page 268. To constrain dimensions, see "Constraining Dimensions" on page 269.

## Dimension Properties

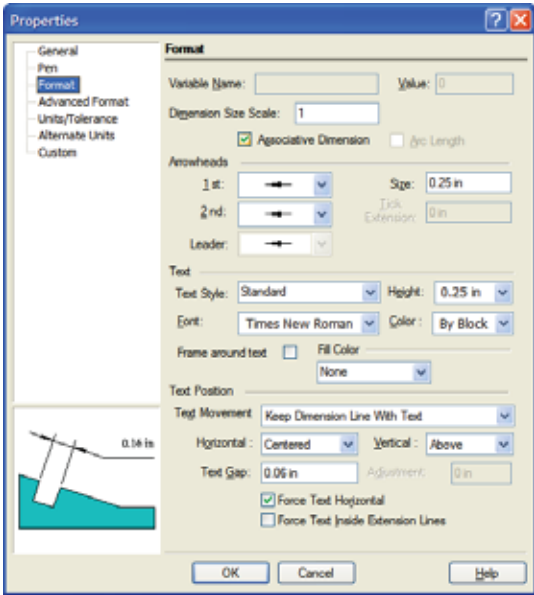
Controls the format of dimensions. You can set the properties for all dimensions, or for just a single dimension. See "Object Properties" on page 79.

Text

There is no **Text** page in dimension **Properties**, but parameters can be set in the Selection Info palette. For details on **Text** parameters, see "Text Properties" on page 325.

Format

Controls the shape of dimension arrows and the position of text relative to the dimension line.



**Variable Name and Value:** If the dimension is produced as a result of **Auto Dimensions** (most commonly used together with **Auto Constraints**), the dimension will have a variable name and value assigned to it. This value appears in **Properties** / **Format** window, as well as the Calculator Palette.

NOTE: See "Auto Dimension" on page 268 "Auto Constraint" on page 266 and "Calculator Palette - Variables" on page 52.

**Dimension Size Scale:** The scaling factor for displaying the dimension.

**Arc Length:** Available for **Angular** dimension, displays the arc length rather than degree measurement.

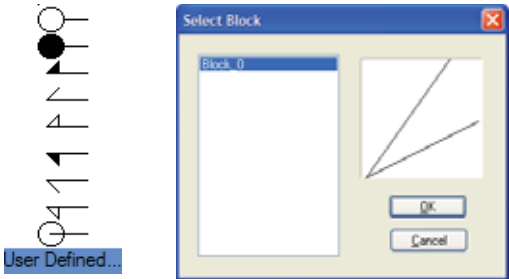
**Associative Dimension:** Checked by default. Associative dimensions retain their positions and the dimension text will update when their associated object changes. To tell whether a dimension is associative, select the object. Any dimension associated with the object will be colored blue. See "Associative Dimensions" on page 338.

NOTE: This parameter is only available when setting the default properties of dimension tools (see "Object Properties" on page 79). You will not see it in the **Properties** window for a selected dimension. This parameter must be set before the dimension is created.

Arrowheads:

- **1st and 2nd:** Select arrow shapes for the start and end of the dimension.
- **Leader:** Select the arrow shape for leader dimensions.
- **Size:** Length of the arrowhead or diameter of the dot.
- **Tick Extension:** If the arrowhead is set to **Tick**, specify the length of the tick extension line.

You can specify a custom user defined arrowhead. User defined arrow heads are based by selecting a block.



Text:

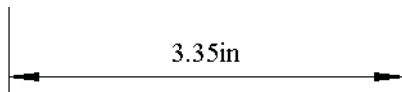
- **Text Style:** Select the Text style for the dimension. Text styles are specified in the Style Manager.
- **Height:** Sets the height of the dimension text.
- **Font:** Specifies the font use in the dimension text.
- **Color:** Sets the color of the dimension text.



- **Frame around text:** Draws a rectangular frame around the dimension text.
- **Fill Color:** Specifies a color which will fill the text frame. The options are: None, Background and Custom. Background fills the frame with the background color. Custom allows you to select a color.

**Text Position:** Controls the dimension text position:

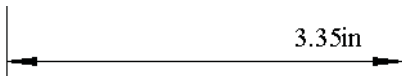
- **Text Movement:** These options control how the dimension text is moved when editing the dimension. See "Editing Dimensions" on page 230.
- **Horizontal:** Select the text alignment with respect to witness lines. The preview window displays each option. Options vary for dimension and leader text.



Dimensions - Centered



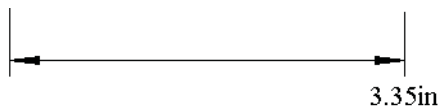
Dimensions - First Extension Line



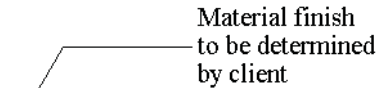
Dimensions - Second Extension Line



Dimensions - Over First Extension Line

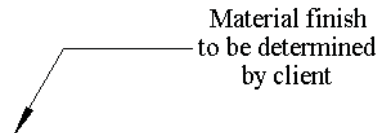


Dimensions - Over Second Extension Line

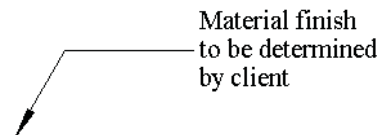


Material finish  
to be determined  
by client

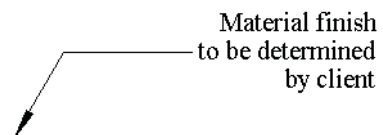
**Leaders - Default Justification  
(Depends on the leader orientation)**



**Leaders - Center Justification**

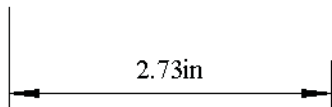


**Leaders - Left Justification**

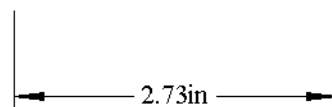


**Leaders - Right Justification**

- **Vertical:** Select the text alignment with respect to the dimension/leader line. **Outside** places the text on the side of the dimension/leader line opposite the selected dimension/leader points.



Above



On Line



Outside

- **Text Gap:** The distance between the dimension text and the dimension line.
- **Adjustment:** Available for **Vertical On Line** position, The distance of the text above or below the dimension line. A value of zero (default) will place the text on the same level as the dimension line. Higher values move the text above the line; lower (negative) values move the text below the line.
- **Force Text Horizontal:** Makes the dimension text horizontal no matter how the dimension is rotated.
- **Force Text Inside Extension Line:** Keep dimension text between the extension lines, regardless of the distance between the extension lines. If unchecked, the system will decide where to place text.

## Advanced Format

Options for drawing and scaling the dimension and extension lines.

The **Advanced Format** dialog box is divided into three main sections: **Dimension Line**, **Extension Lines**, and **Leader Text Frame**.

- Dimension Line:**
  - Force Interior Line:** Checked (indicated by a green checkmark).
  - Baseline Increment:** 0.38 in
  - Color:** By Block
  - Line width:** By Block
  - Line type:** By Block
  - Do not draw:**
    - ☐ First Line
    - ☐ Second Line
    - ☐ Outside Extension
- Extension Lines:**
  - Draw As Spline:** Unchecked
  - Extension:** 0.13 in
  - Offset:** 0.06 in
  - Color:** By Block
  - Line type:**
    - 1st line: By Block
    - 2nd line: By Block
  - Do not draw:**
    - ☐ First Line
    - ☐ Second Line
- Leader Text Frame:**
  - Size:** (empty field)
  - ☐ Fit to Text

**Dimension Line:** The line over which the dimension text is located.

- **Force Interior Line:** If checked, the dimension will have an interior dimension line even if the text is outside the extension lines. If unchecked, the interior dimension line will be drawn only if the dimension text is inside the extension lines.
- **Baseline Increment:** The distance between each dimension in a series of baseline dimensions.
- **Do not Draw:** Options for omitting parts of the dimension line.
- **Color:** Sets the color of the dimension line.
- **Line Width:** Sets the Line width for the dimension line.
- **Line Type:** Specifies the line type for the dimension line.

**Extension Line:** Lines that connect the dimension line to the object being dimensioned.

- **Draw as Spline:** If checked, the **Leader** tool will use a spline rather than a line to attach the dimension text to the corresponding drawing location. See "Spline by Control Points" on page 170.
- **Extension:** The length of the extension line segments that extend outward beyond the dimension lines.
- **Offset:** The distance between the extension lines and the dimensioned object.
- **Do not Draw:** Options for omitting parts of the extension lines.
- **Color:** Sets the color of the extension lines.
- **Line Width:** Sets the line width for the extension lines.
- **Line Type:** Specifies the line type for the extension lines.

**Leader Text Frame:** Available only for **Leader** dimensions. Select a shape for the closed line bounding the dimension text.

- **Size:** When **Circle** or **Quadrant** is selected for the text box, define the box size.
- **Fit to Text:** Check if you want the text box to fit the dimension text.

## Units / Tolerance

Parameters for formatting the appearance of dimension text, and for controlling the display of tolerance - the allowable deviation from the dimension.

The screenshot shows the 'Units / Tolerance' dialog box. It has two main sections: 'Primary Units' and 'Tolerance'.  
**Primary Units section:**  
 - **Append Units:** ☒  
 - **Units:** By Draw (dropdown)  
 - **Format:** Decimal (dropdown)  
 - **Prefix:** (empty text box)  
 - **Suffix:** (empty text box)  
 - **Scale:** 1 (text box)  
 - **Round Off:** 0.01 (text box)  
 - **Special Symbol:** None (dropdown)  
 - **Precision:** 2 (spin box)  
 - **Fraction format:** Not stacked (dropdown)  
 - **Trailing Zeros:** ☒  
 - **Leading Zeros:** ☒  
 - **Zero Feet:** ☒  
 - **Zero Inches:** ☒  
**Tolerance section:**  
 - **Append:** ☐  
 - **As Limits:** ☐  
 - **Precision:** 3 (spin box)  
 - **Vertical Position:** Middle (dropdown)  
 - **Relative Height:** 1 (text box)  
 - **Upper:** 0 (text box)  
 - **Lower:** 0 (text box)  
 - **Trailing Tolerance Zeros:** ☒  
 - **Leading Tolerance Zeros:** ☒  
 - **Zero Feet:** ☐  
 - **Zero Inches:** ☐

### Primary Units:

- **Append Units:** If checked, the dimension text will display its unit.
- **Units:** If **Append Units** is checked, select the desired units.
- **Format:** Options for how the dimension text is displayed (decimal, feet, radians, surveyor, etc.).
- **Prefix, Suffix:** Add a prefix and/or a suffix to the dimension text. (Not available for **Leader** dimensions.)
- **Scale:** Change the scale of the value displayed by the dimension text relative to World units. For example, if you enter a value of 0.1, the dimension will display a value of 0.1 inch when dimensioning a distance of 1 inch in World units. The default value for the linear measurement scale is 1.

---

NOTE: You will probably want to leave Scale unchanged, unless you have an inset in your drawing that uses a different scale than the rest of your drawing.

---

- **Round Off:** -The decimal place to which the dimension text will round off. If you type a value of 0.1, for example, the text will be rounded off to tenths.
- **Special Symbol:** Prepends a symbol to the dimension text. The options include: None, Diameter, Degree, and Plus Minus.
- **Precision:** The level of accuracy, represented by a number between 1 and 10.
- **Trailing Zeros:** If checked, trailing zeros will appear in the dimension text. For example, if a dimensioned line is exactly 2.34 units long, **Precision (Options / Space Units)** is set to 4, and **Round Off** is set to 4, the dimension text will read 2.3400.

---

NOTE: *Trailing Zeros, Round Off, and Precision are interrelated, and need to be considered as a group when establishing dimension settings. If Round Off is less than Precision, dimension text may not accurately reflect the exact measurement. If Round Off is greater than Precision, then Trailing Zeros will show the additional available precision if Trailing Zeros is set to a value equal to Precision. For most applications, Precision and Round Off should be set to the same level (a Precision of 4 is the same as a Round Off of 0.0001). Trailing Zeros should then be used if necessary to display the level of precision in use. Trailing Zeros is not applicable when fractions are used.*

---

- **Leading Zeros:** For dimensions of less than one unit, a zero will appear at the beginning of the dimension. For example: 0.5 feet, as opposed to .5 feet.
- **Zero Feet, Zero Inches:** Relevant for architectural and engineering units. For dimensions less than one foot or one inch, the zero will appear as a placeholder. For example: 0'-0 1/4" as opposed to 1/4".

#### Tolerance:

- **Append:** If checked, the dimension text will include a tolerance.

- **As Limits:** If checked, the dimension will be shown as a pair of values defining the limits of the dimension value.
- **Relative Height:** The height of the tolerance relative to the rest of the dimension text.
- **Upper, Lower:** Values for positive and negative tolerance.

The remaining tolerance parameters are the same as for **Primary Units**.

### Alternate Units

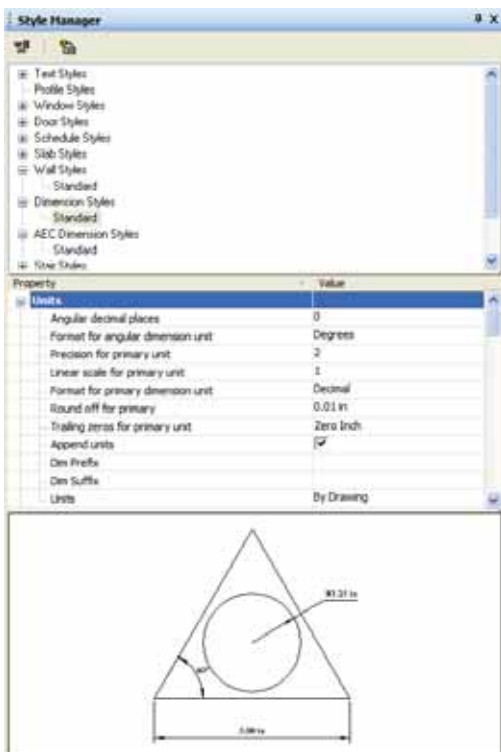
Alternate units are a secondary set of units for each dimension, such as 1" [25.4mm].

**Use alternate units:** If checked, the dimension and/or tolerance will be displayed with a value in alternate units, in square brackets following the primary dimension. The parameters are the same as on the **Units / Tolerance** page.

## Dimension Styles

You can manage and create dimension styles using the Style Manager. For more on the Style Manager see "Style Manager" on page 564.

In the Style Manager, there is one style, "Standard," listed under "Dimension styles."

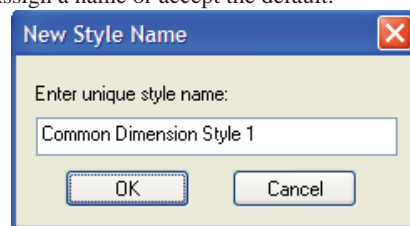


*NOTE: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window.*

- You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted, then click **Create New**.

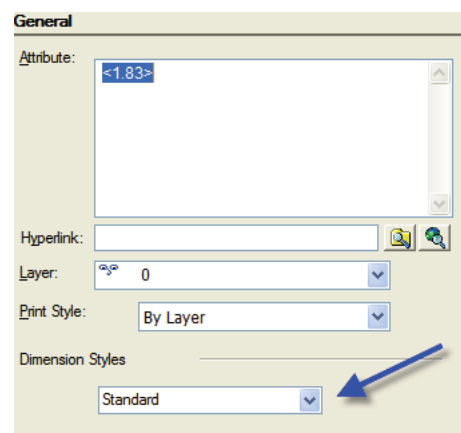


- Assign a name or accept the default.



This creates a new which is a copy of "Standard."

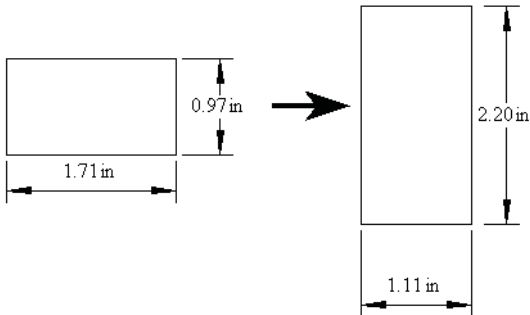
- To change the current wall dimension to the new style, open its **Properties** to the **General** page.



Dimension styles can control all of the properties of your dimensions. They are also automatically imported from DWG files.

## Associative Dimensions

When a dimension is associated with objects, you can freely move, rotate, or resize the associated objects and the dimension will retain its position relative to the objects, and the dimension text will change automatically to reflect changes.




---

NOTE: To make dimensions associative to other dimensions, see "Constraining Dimensions" on page 269.

---

Before creating associative dimensions, associativity needs to be activated in the default **Properties** of the dimension tools. Associativity cannot be set in the **Properties** window of a selected dimension; this parameter must be set before the dimension is created. Right-click on any dimension tool to bring up the **Properties** window, and open the **Format** page. Make sure that **Associative Dimension** is checked.

---

TIP: To tell whether a dimension is associative, select the object. Any dimensions associated with that object will be colored blue.

---

There are the following limitations on associativity:

- Dimension cannot be associative if you override the automatic dimension text, either in the Inspector Bar or in the **Attribute** field of the **General** page of a dimension's **Properties**.
- Datum dimensions cannot be associative. However, grouping datum dimensions with their objects can achieve a similar effect to associative dimensioning. Be sure to explode to edit objects later.
- The following dimensions will only be associative if originally created using **Snap to Vertex** or **Snap to Center**, or if created using **Segment** or **Entity Dimensioning**: Baseline, Continuous, Incremental, Leader, Orthogonal, Parallel.
- When dimensioning over a Viewport in Paper Space, the dimensions will associate with the viewport itself, but not with the corresponding objects in Model Space. See "Viewports" on page 657.

To break the associativity between a dimension and its objects, select both the object(s) and the dimension. Right-click and select **Drop Link** from the local menu.



## Segment and Entity Dimensioning

For linear dimensions (orthogonal, parallel, and rotated), you can define the dimension by selecting two points (manually), or you can select a segment or entire object to dimension.

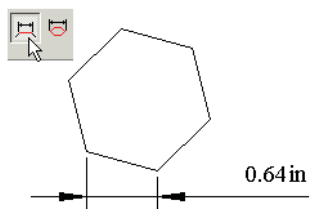
**Segment** and **Entity** modes are options on the local menu, as well as on the Inspector Bar while using a linear dimension tool. Once you have selected a dimensioning mode, the mode remains in effect until changed.

---

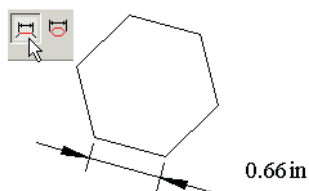
TIP: If you generally prefer one of the dimensioning modes over the others, you can save it in a drawing template. To do this, use **File / Save As** to save the file as a \*.tct file (TurboCAD Template). Place the template file in the "Template" folder of the TurboCAD root directory. Then when you want to open the template, use **File / New**, and select **New from Template**.

---

In **Segment** mode, select any line segment, and the dimension between endpoints is created.

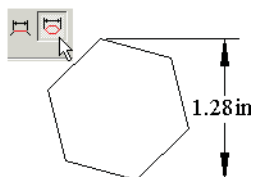


**Segment mode - orthogonal dimension**

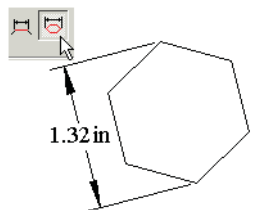


**Segment mode - parallel dimension**

In **Entity** mode, select an object, and the largest dimension in the specified direction is created. This is useful for dimensioning objects like polylines, polygons, rotated rectangles, etc.



**Entity mode - orthogonal dimension**

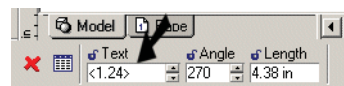


**Entity mode - parallel dimension**

## Creating Dimensions

Creating dimensions involves two basic steps: selecting the objects to be measured, and locating the dimension. By default the magnitude of the dimension is calculated automatically and is recorded as the dimension text, measured in current World units. (Leader dimensions, which contain text labels, are the exception to this rule.)

You can override the default text of a dimension by specifying the text in the Inspector Bar prior to finishing the dimension.



You can also change the dimension text in the dimension's **Properties**, by changing the **Attributes** field of the **General** page.

---

**NOTE:** To create several types of dimensions automatically in one step, see "Auto Dimension" on page 268.

---

When **Auto Add Constraints** is active, any dimensions you assign are created as variables that appear in the Calculator Palette.



Dimension variables can be constrained to other dimensions, or to other variables or numbers. This is particularly powerful when used in conjunction with **Auto Constraints**. See "Constraining Dimensions" on page 269.

## Orthogonal Dimension



Creates a horizontal or vertical dimension.

---

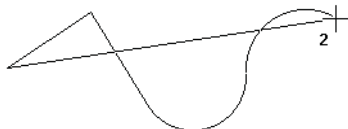
NOTE: To create orthogonal dimensions automatically, see "Auto Dimension" on page 268.

---

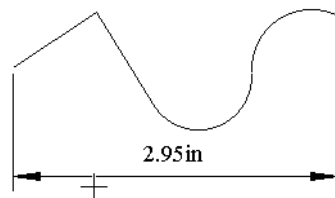
1. Select the first point.



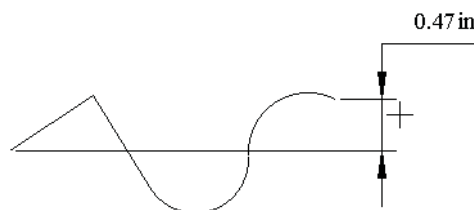
2. Select the second point, or enter the length and angle of the dimensioned line in the Inspector Bar.



3. For a horizontal dimension, move the mouse above or below the two points and click to define the location. You can also enter the length of the witness lines in the Inspector Bar.



4. For a vertical dimension, define the dimension at either side of the two points.



Local menu options:

**Segment Dimensioning, Entity Dimensioning:** See "Segment and Entity Dimensioning" on page 338.

**Horizontal / Vertical Mode only:** creates either type of dimension, no matter where the dimension is located.

## Parallel Dimension



Creates a dimension showing the absolute length of an object.

---

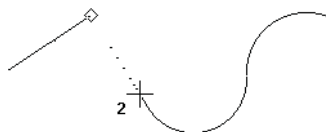
NOTE: To create parallel dimensions automatically, see "Auto Dimension" on page 268.

---

1. Select the first point.

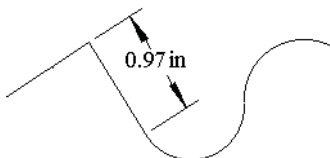


2. Select the second point, or enter the length and angle of the dimensioned line in the Inspector Bar.

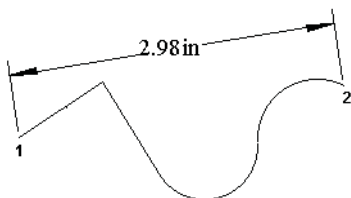




3. Move the mouse and click to define the location. You can also enter the length of the witness lines in the Inspector Bar.



For any two endpoints, the absolute length is parallel to the line between the points.



Local menu options:

**Segment Dimensioning, Entity Dimensioning:** See "Segment and Entity Dimensioning" on page 338.

## Distance Dimension

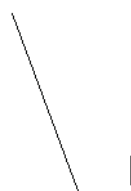


Sets two lines parallel to one another and places a constrained dimension between them. For this tool to be available, **Auto Add Constraints** must be on.

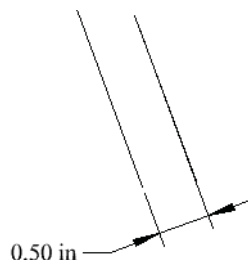


NOTE: To create distance dimensions automatically, see "Auto Dimension" on page 268.

1. Select the two lines, or select them by snapping to either of their endpoints.



2. Place the dimension, and the lines are made parallel.



NOTE: Because **Auto Add Constraints** is on, the Distance dimension appears as a variable in the Calculator Palette and can be edited. See "Constraining Dimensions" on page 269.

## Rotated Dimension

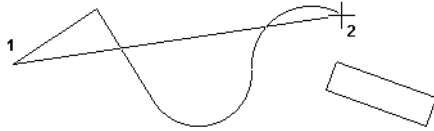


Creates a dimension projected in a specified direction.

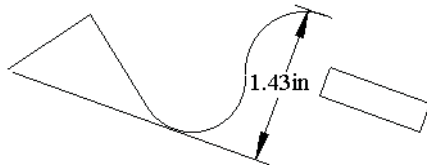
1. Select two points to define the dimension's direction. The dimension will be measured normal to this line. You can also define the first point, then specify the angle of the vector in the Inspector Bar.



1. Select the first point of the object to be dimensioned. Select the second point, or enter the length and angle of the dimensioned line in the Inspector Bar.



2. Move the mouse and click to define the location. You can also enter the length of the witness lines in the Inspector Bar.



#### Local menu options:

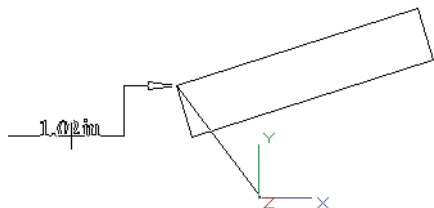
**Segment Dimensioning, Entity Dimensioning:** Available only after the dimension direction has been defined. See "Segment and Entity Dimensioning" on page 338.

### Datum Dimension

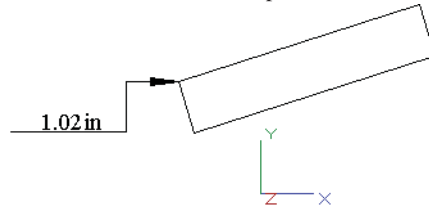


Creates a dimension showing the horizontal or vertical distance from a point. By default, the point is the absolute origin, but you can change this.

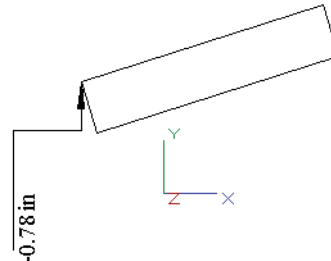
1. Select the point whose distance from the origin is to be displayed. A rubberband line appears, indicating the origin of the dimension.



2. To display the Y coordinate (horizontal dimension text), move the mouse to either side of both the origin and the selected point, and click to locate the dimension. You can also enter the angle and length of the extension lines in the Inspector Bar.



3. To display the X coordinate (vertical dimension text), move the mouse above or below both the origin and the selected point.



#### Local menu options:

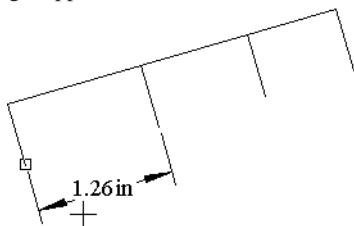
**Set Origin:** By default, datum dimensions are created relative to the absolute origin. Use Set Origin to select a new origin. This origin will remain in effect until changed.

### Baseline Dimension

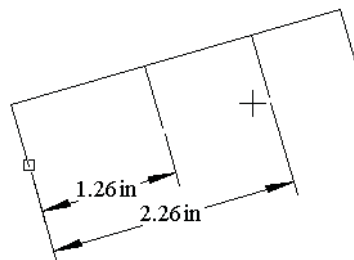


Creates a series of parallel linear dimensions that follow the axis of an existing linear dimension.

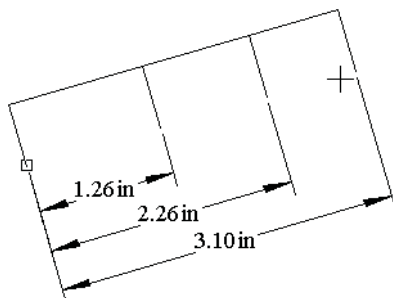
1. Select an existing linear base dimension. Click near the side you want to serve as the baseline. A temporary rectangle appears at the baseline end.



2. Select the first point where you want a new baseline dimension.



3. Continue selecting points. Each new baseline dimension will be created at an offset from the previous one.



**NOTE:** The offset distance is controlled by the **Baseline Increment** value on the **Advanced Format** page of the **Properties** window.

4. When finished, select **Cancel** from the local menu or press Esc.

Local menu option:

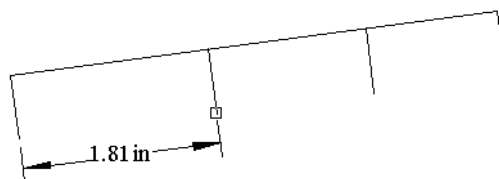
**Select Dimension:** Select a new base dimension.

## Continuous Dimension

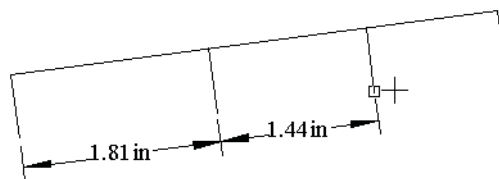


Creates a series of parallel linear dimensions measured from the previous dimension. The dimensions follow the axis of the base dimension.

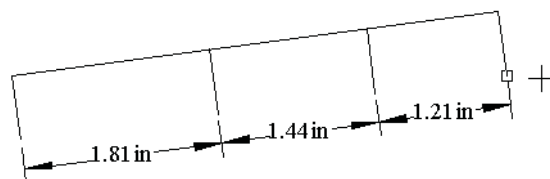
1. Select an existing linear base dimension. Click near the side you want the next dimension to be created. A temporary rectangle appears at the baseline end.



2. Select the first point where you want a new continuous dimension.



3. Continue selecting points. Each new continuous dimension will be measured from the previous dimension.



4. When finished, select **Cancel** from the local menu or press Esc.

Local menu option:

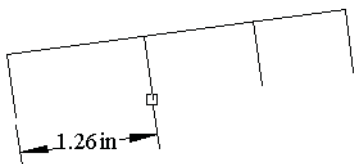
**Select Dimension:** Select a new base dimension.

## Incremental Dimension

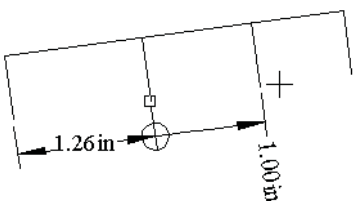


Creates a series of parallel linear dimensions measured from the previous dimension. The dimensions follow the axis of the base dimension, and are displayed normal to the axis.

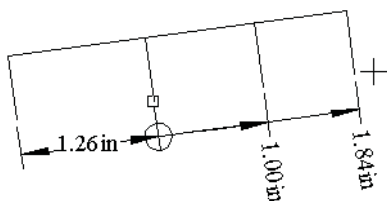
1. Select an existing linear base dimension. Click near the side you want the next dimension to be created. A temporary rectangle appears at the baseline end.



2. Select the first point where you want a new incremental dimension.



3. Continue selecting points. Each new incremental dimension will be measured from the previous dimension.



4. When finished, select **Cancel** from the local menu or press Esc.

Local menu option:

**Select Dimension:** Select a new base dimension.

## Angular Dimension



Creates dimensions measuring angles. You can dimension the angle between two lines, the angle of an arc, between two points of a circle, and between a node and two points.

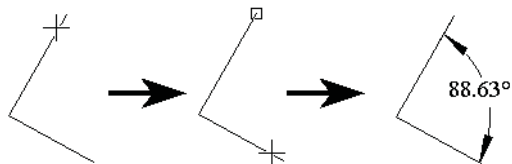
---

*NOTE: To create angular dimensions automatically, see "Auto Dimension" on page 268.*

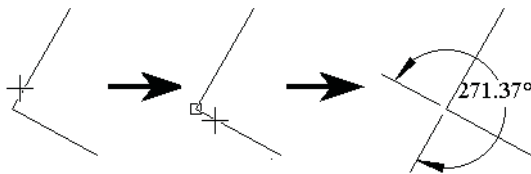
---

### Angle Between Two Lines

1. Select the two lines.
2. Move the mouse to dimension the acute or obtuse angle. Click to locate the dimension, or enter the length and angle in the Inspector Bar.



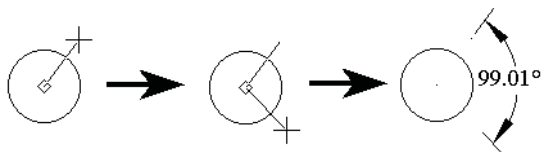
Be careful where you select the lines. If you select close to the angle vertex, you could dimension the complementary angle.



### Angle within a Circle

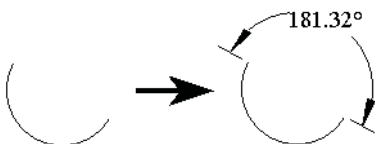
1. Select the circle.
2. Select the start angle, or enter the angle in the Inspector Bar.
3. Select the end angle.

4. Move the mouse to dimension the acute or obtuse angle. Click to locate the dimension, or enter the length and angle in the Inspector Bar.



### Angles of an Arc

1. Select the arc.
2. Move the mouse to dimension the acute or obtuse angle. Click to locate the dimension, or enter the length and angle in the Inspector Bar.

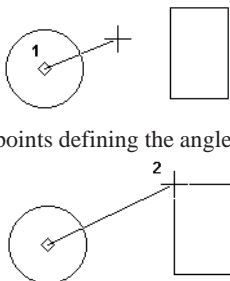


**NOTE:** If **Arc Length** is checked in the **Format** page of the **Properties** window, the arc length will be dimensioned instead of the angle.

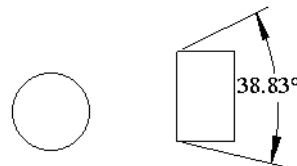
### Local menu option:

**Angle Node** (freeform angle): Dimensions an angle by selecting the angle vertex then two points.

1. Select the angle vertex.
2. Select two points defining the angle.



3. Move the mouse to dimension the acute or obtuse angle. Click to locate the dimension, or enter the length and angle in the Inspector Bar.



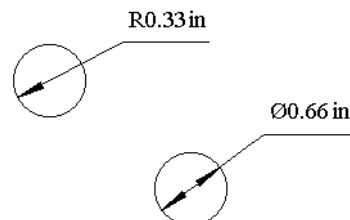
### Radius / Diameter Dimension



Dimensions the radius or diameter of an arc or circle.

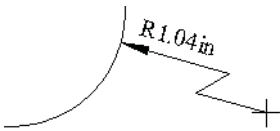
**NOTE:** To create radius or diameter dimensions automatically, see "Auto Dimension" on page 268.

1. Select the arc or circle.
2. Move the mouse and click to locate the dimension, or enter the length and angle in the Inspector Bar.



### Local menu option:

**Large Radius:** If the arc center is out of your drawing space, use this option to display the dimension from outside.



Leader



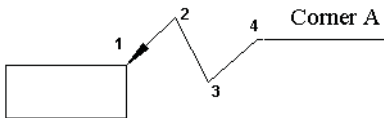
Creates dimension text attached to a simple line leader pointing to a location in your drawing. The leader is similar to a polyline.

NOTE: Check **Draw as Spline** on the **Advanced Format** page of the **Properties** window to create a curves leader rather than line segments.

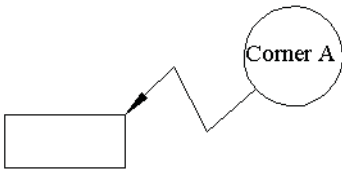
1. Enter the text in the Inspector Bar.
2. Select the leader start point (the end with the arrow). Select or more additional segment endpoints, or enter the length and angle of each segment in the Inspector Bar.

NOTE: The arrowhead is defined by **Arrowheads / 1st** on the **Format** page of the **Properties** window.

3. Double-click to finish, or select **Finish** from the local menu, or press Alt+F.



A text box can be created, by selecting one from the **Advanced Format** page of the **Properties** window.



Wall Dimensions

There is a special tool for dimensioning walls created via the **Architecture** tools. See "Wall Dimension" on page 541.

Quick Dimensions



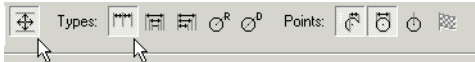
Creates a series of dimensions on one more objects. You can choose the type of dimension and which points will be included.

NOTE: To create a group of various types of dimensions automatically, see "Auto Dimension" on page 268

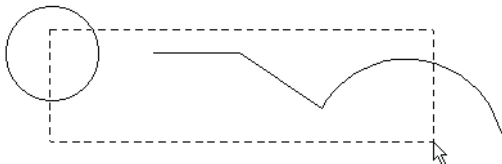
Activate the **Quick** function, then choose three criteria:

- Whether the dimensions will be orthogonal (horizontal / vertical)
- The type of dimension (continuous, baseline, etc.),
- The points between which the dimensions will be created.

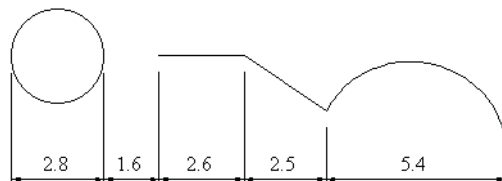
In this example, the dimensions will be **Orthogonal** and **Continuous** (indicated by the arrows in the picture below). Points will be described later in the example.



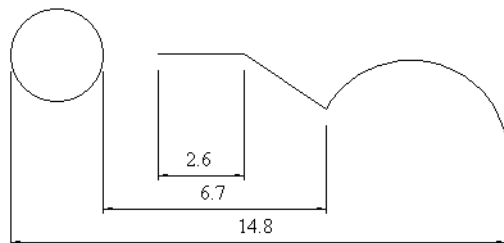
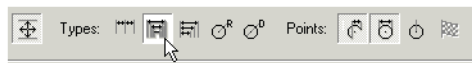
Select the object or objects to dimension. You can use the Shift key to select multiple objects, or drag a selection window. This example consists of one circle and one polyline.



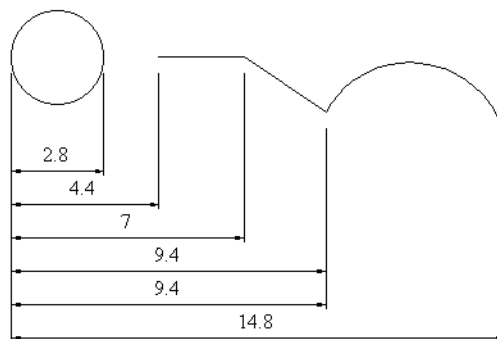
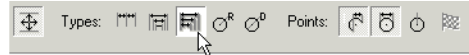
Double-click when all objects are selected. Then move the cursor and click to place the dimensions. Moving the mouse to the right or left of the objects creates vertical dimensions; moving above or below the objects, as shown below, creates horizontal dimensions.



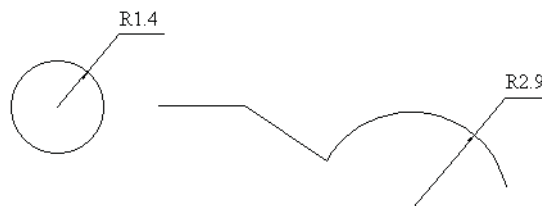
This would be the result with **Staggered** dimensions:



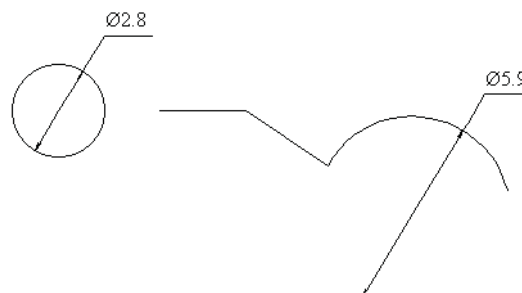
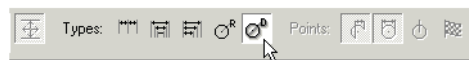
This would be the results with **Baseline** dimensions.



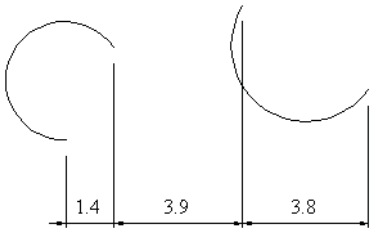
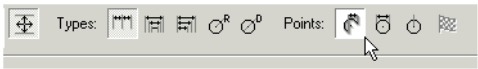
If **Radius** is selected, a radius dimension is assigned to all arc or circle segments found in the selected objects.



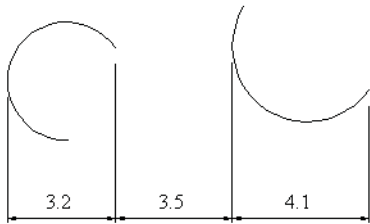
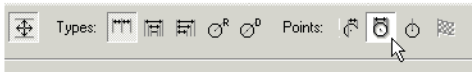
If **Diameter** is selected, a diameter dimension is assigned to all arc or circle segments found in the selected objects.



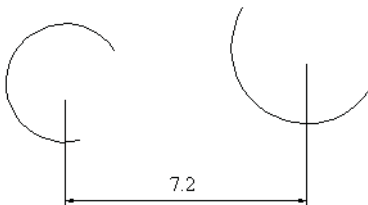
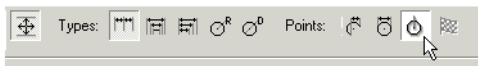
**Points** affect dimensioning of arcs, curves, and segments of polylines. **Start / End** creates dimensions between all start and end points of arcs.



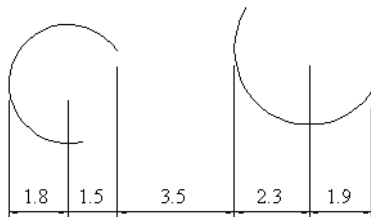
**Dimension** measures the overall distance of the objects.



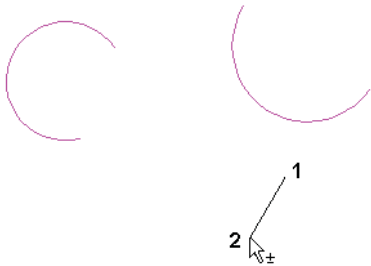
**Arc Center** measures between center points.



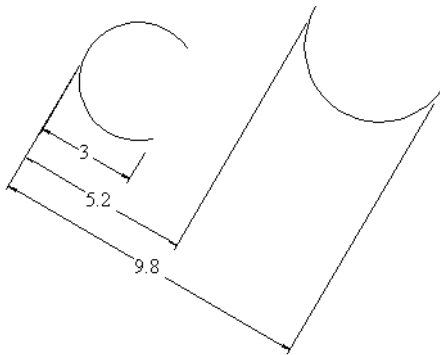
You can combine **Points** options. This example uses both **Dimension** and **Arc Center**.



If **Orthogonal** is not selected, you can define the dimension direction. Select two points to define the direction **normal** to the desired dimension line.



This is the result of non-orthogonal, baseline dimensions.





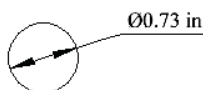
## Smart Dimensions



Creates a dimension based on the object selected.

NOTE: To create a group of various types of dimensions automatically, see "Auto Dimension" on page 268.

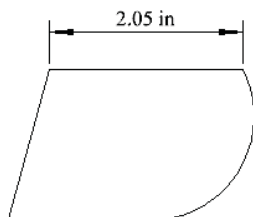
If you select a circle, a diameter dimension will be created.



In the Inspector Bar, you can change the text, set the angle of the dimension, and set the length of the leader line.

Text	Angle	Length
<%%c0.73>	22.16	0.45 in

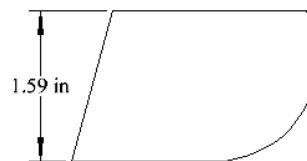
If you select a line or line segment, its length dimension will be created.



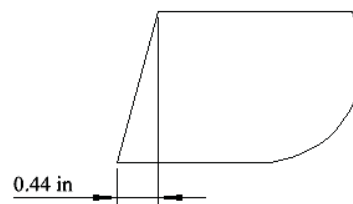
If the line segment is not orthogonal, there are three possible dimensions you can create: **Parallel**



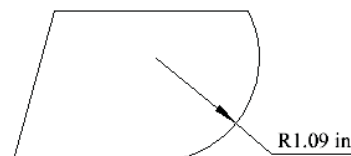
**Orthogonal** (vertical)



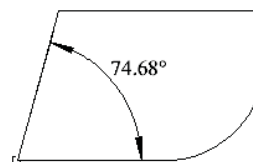
**Orthogonal** (horizontal)



If you select an arc or arc segment, a radial dimension will be created.

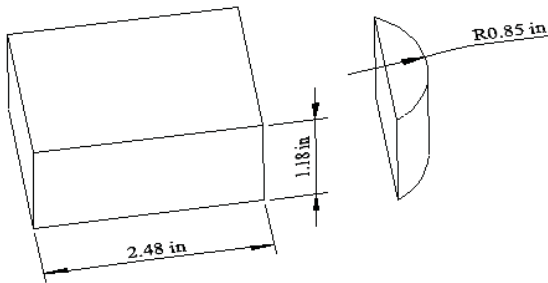


To create an angular dimension, press Shift and select the two lines.



**Smart** dimensions also work for linear and circular edges of solid ACIS objects (Available in TurboCAD Pro and Platinum only). You can use this feature to dimension

cross-section generated through the Drafting Palette, as long and the Surface Engine option in the Drafting Palette was turned off when the section was created.



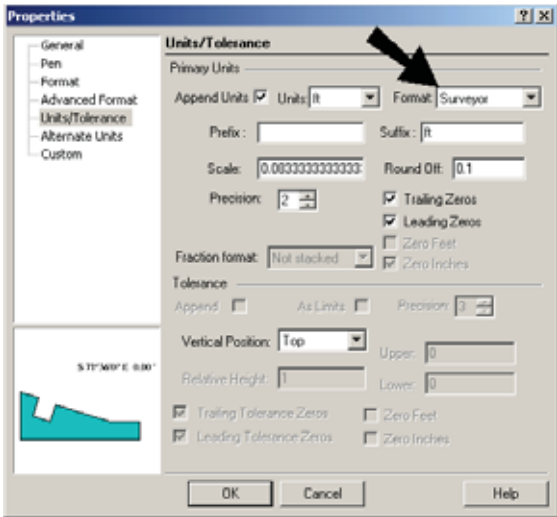
Linear and angular dimensions can also be created for **3D Polylines**.

**Surveyor Dimensions**

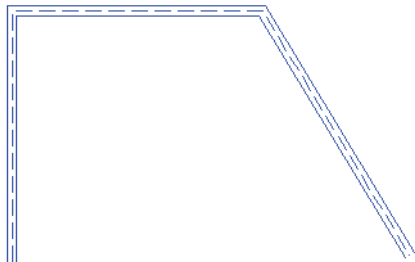
Linear and Angular dimensions can be displayed in surveyor format.

For linear dimensions, surveyor format is angle (degree-minutes-seconds) and length. For angles, surveyor format is degree-minutes-seconds.

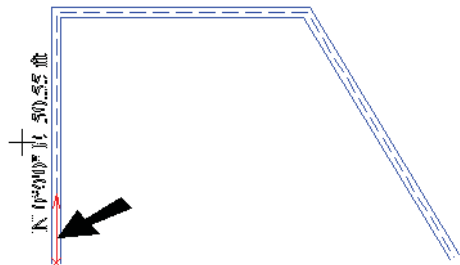
Before creating surveyor dimensions, open the **Properties** for any dimension tool. On the **Units / Tolerance** page, set the **Format** to **Surveyor**.



In this example, surveyor dimensions will be placed using **Parallel Dimensions**, on three segments of this **Multi Line Polyline**.



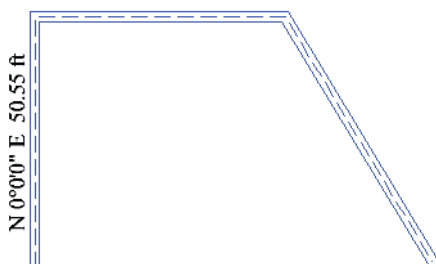
1. After setting **Surveyor** in the dimension tools' **Properties**, activate the dimension tool you want to use (in this case, **Parallel**, but you can use any linear dimension). Click the segment you want to dimension. (Only segment dimensioning is available when creating surveyor dimensions; you cannot click two points.) The red arrow indicates the direction in which the segment is measured.



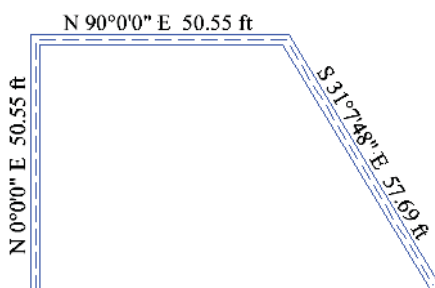
2. If you want the segment measured from the other side, select **Invert** from the Inspector Bar or local menu.



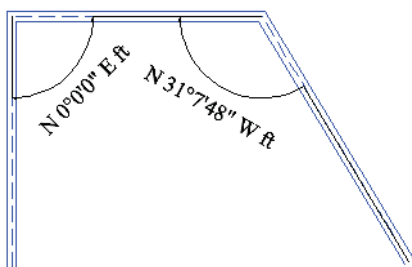
- Click a second time to place the dimension.



- Dimension additional segments the same way.



You can dimension an angle using surveyor dimensions as well.



## Dimensions in Viewports

Dimensioning within a viewport in PaperSpace will provide dimensions of the object's 2D projection onto the monitor screen, not of the object itself. The dimensions will be scaled correctly according to the scale of the view, but misleading dimensions can result if the monitor screen is not exactly parallel to the object.

To dimension an object in non-parallel views within a viewport, you must enter the correct measurements manually in the Inspector Bar or in the dimension's **Properties**. When non parallel views are used, dimensioning is best done in Model Space.

See "Viewports" on page 657.

## MultiLeader Dimensions

*Available in TurboCAD Pro and Platinum only*

**Menu:** Insert / Dimension / MultiLeader

MultiLeader dimension are a special category of dimension tool designed to support entities imported from external DWG files. These leaders attached a single piece of content to multiple leader arrows. the content may be text, a block, or nothing. The content and structure of multileaders is defined by the MultiLeader style.

Local menu option:

**Arrowhead First:** The first click places an arrowhead. Turning this option on turns off Content First.

**Content First:** The first click places the content. Turning this option on turns off Arrowhead first.

**Fixed Leader Points:** Limit the number of points within each leader to the number specified in the style. The default is 2, one at the arrowhead and one at the content. Once both points have been placed, a new leader will begin for the current multileader. If this option is off leaders may have as many points as you wish.

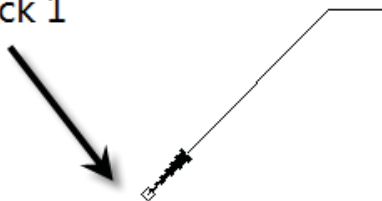
**Insert Next Leader:** This option is only available if Fixed Leader Points is turned off. When available the option finishes the current leader at its last selected point and begins the next leader.

**Finish:** Finishes the entry for all of the leaders.

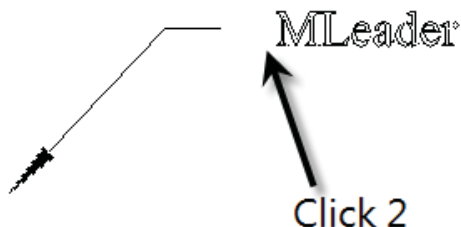
- Select the MultiLeader tool.

- Click to select the first point for the leader.  
If the Content First option is selected this will be the location of the content, otherwise it will be the location of the first arrowhead.

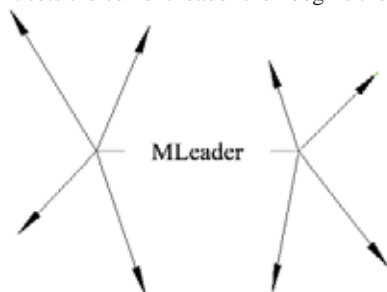
Click 1



- Click to select the second point for the leader.  
If the Content First option is selected this will be the location of the first arrow head, otherwise it will be the location of the first content.



- If the default settings and style is in use a new leader will begin after the second click.
- Click to set the arrow head of the next leader.  
The content position is already set. So with the standard limit of two points per leader selecting each point sets the current leader then begins the next.



- If you have Fixed Leader Points turned off, continue entering points as needed then use Insert Next Leader to move on to the next leader in the sequence.
- Click Finish when you are done creating leaders.

Using the Edit tool you can add and subtract leaders on existing multileaders. you can also add, subtract, and move nodes of multileaders.

## MultiLeader Properties

MultiLeader have different properties than standard dimensions.

### MultiLeader Format

**MultiLeader Format**

Overall Scale:  ☐ Annotative

Arrowhead  
 Size:

Leader  
 Leadertype:  Color:   
 Line type:  Line width:

☒ Landing Landing distance:

Content type:

**Overall Scale:** Sets the overall scale for the multileader.

**Annotative:** Set whether the multileader is annotative or not. This has no effect within the application is strictly a setting to support external DWG applications.

**Arrowhead:** Sets the type of arrowhead used in the multileader.

**Size:** Sets the size of the leader arrowheads.

**Leader Type:** Sets whether the type is a line, spline, or none.

**Color:** Sets the color for the leaders.

**Line Type:** Sets the line type for the leaders.

**Line Width:** Sets the line width for the leaders.

**Landing:** Sets whether there will be a landing, a horizontal line attaching the end of leaders to the content.

**Landing Distance:** Sets the landing width.

**Content Type:** This is controlled by the Style

## MultiLeader Content

The feature of the MultiLeader Properties Content page vary depending upon the content type specified in the selected MultiLeader Style. If the Content Type in the style is set to none the page will be blank. If the Content Type is set to text the page will be as follows:

The screenshot shows the 'MultiLeader Content' dialog box with the 'Text' tab selected. The 'Text Style' is set to 'Standard' and 'Height' is '0.25 in'. There are checkboxes for 'Frame around text' and 'Background Mask', both currently unchecked. 'Width' is '0 in' and 'Color' is 'By Block'. Under 'Text Position', 'Justify' is 'Left Justificat', 'Text Rotation' is '0', 'Direction' is 'Horizontal', 'Landing Gap' is '0.25 in', 'Left attachment' is 'Middle of top', 'Right attachment' is 'Middle of top', and 'Angle Type' is 'As inserted'. Under 'Text LineSpacing', 'Style' is 'At least', 'Factor' is '1', and 'Distance' is '6.666667'.

**Text Style:** Set which Text style from the Style manager is used in the multileader.

**Height:** Sets the height of the text. This will override the settings in the styles.

**Frame around text:** specifies whether a box will be drawn around the text of the multileader.

**Width:** Sets the width of the paragraph.

**Background Mask:** Creates a mask for objects behind the text.

**Color:** Sets the text color.

**Justify:** Specifies the text justification.

**Left Attachment:** Specifies the vertical position where the leaders on the left side of the content.

**Right Attachment:** Specifies the vertical position where the leaders on the right side of the content.

**Text Rotation:** Specifies the angle of the content text.

**Landing Gap:** Specifies the distance between the end of the landing and the content.

**Style:** Select **At Least** to automatically set the line size relative to the largest character of a line. Select **Exactly** to keep all lines the same size.

**Factor:** Sets the distance between lines.

If the Content Type in the style is set to block the Content page will look as follows:

The screenshot shows the 'MultiLeader Content' dialog box with the 'Block' tab selected. 'Block Source' is 'Block\_0', 'Block attachment' is 'Insertion point', 'Block Color' is 'By Block', 'Block Rotation' is '0', 'Scale X' is '1', 'Scale Y' is '1', and 'Scale Z' is '1'.

**Block Source:** Specifies the block to be used in the content.

**Block Attachment:** Specifies whether the block will be attached to the multileader using its geometric center of the block, or attached using the block reference point (insertion point).

**Block Color:** Sets the block color.

**Block Rotation:** specifies the angle for the rotation of the block insertion.

**Scale X:** Sets a scaling factor for the block along its X axis.

**Scale Y:** Sets a scaling factor for the block along its Y axis.

**Scale Z:** Sets a scaling factor for the block along its Z axis.

## MultiLeader Styles

You can manage and create MultiLeader styles using the Style manager. For more on the Style Manager see "Style Manager" on page 564.

In the Style Manager, there is one style, “Standard,” listed under “MultiLeader styles.”

NOTE: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window.

8. You can change the “Standard” style, but if you want to preserve this style, make sure “Standard” is highlighted, then click **Create New**.
9. Assign a name or accept the default.  
This creates a new which is a copy of “Standard.”
10. To change the current multileader to the new style, open its **Properties** to the **General** page.

Drawing Symbols

The **AddOns** menu (*in TurboCAD Pro only*) contains several commonly-used symbols you can insert into your drawing. Most of these tools can also be found on the **Special Tools** toolbar, displayed by right-clicking in any toolbar and selecting **Special Tools**.

Drawing Markers

Available in TurboCAD Pro and Deluxe only

Menu: Tools / Architecture / Markers

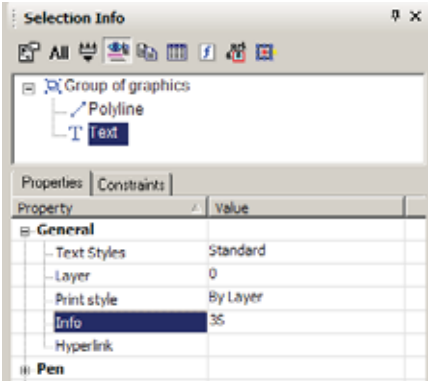
Use drawing markers to add a shape with text inside. You can choose between a circle, square, diamond, and hexagon. These tools can also be found on the **Drawing Markers** toolbar.



The marker is inserted with default text (1S, 2S, etc.).



You can edit the text on the **General** page of the marker’s **Properties**. In addition, you can modify properties of either the shape or text by using the Selection Info palette.

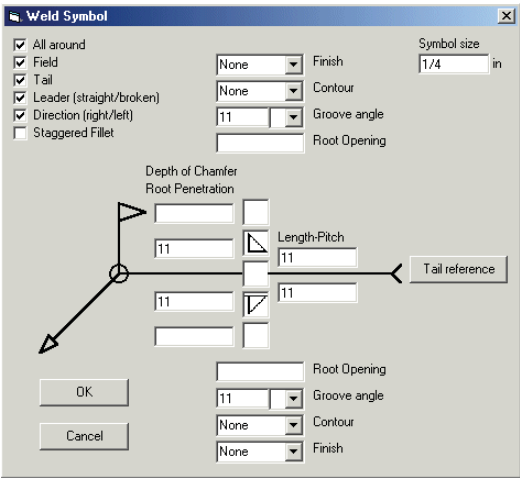


Weld Symbols

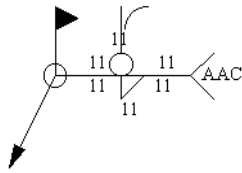
Available in TurboCAD Pro and Platinum only



Use the **Weld Symbol** window to enter the symbol parameters.



Click OK when finished, and locate the symbol in your drawing.



To edit a weld symbol, open its **Properties** window (see "Object Properties" on page 79). Open the **Other** page, click **Weld Symbol**, and click **Go To Page**.

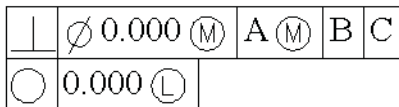
This opens the original design window, in which you can change any symbol parameters.

## Geometric Tolerance

*Available in TurboCAD Pro and Platinum only*



Tolerance is used to specify by how much a dimension can vary. Geometric tolerancing is a way to show maximum allowable deviations of form, profile, orientation, location, and runout of a feature.



The first of two mandatory symbol frames contains a symbol that represents the geometric characteristic to which a tolerance is being applied, such as form, orientation, or runout. Form tolerances control straightness, flatness, circularity, cylindricity, and profiles of lines and surfaces.

The second mandatory frame contains the tolerance value. Where applicable, the tolerance value is preceded by a diameter symbol and followed by a material condition symbol.

For a tolerance frame with two tolerance values, the second compartments contains the value of Tolerance 1 and is followed by a third, identical compartment that holds the

value of Tolerance 2. Other (optional) compartments commonly contain a pair of symbols each. These are a datum reference letter and a material condition symbol.

1. Select the Tolerance tool.
2. Click to place the insertion point of the tolerance.
3. Click to set the angle of the tolerance insertion and to finish.

The **Tolerance** window provides controls for customizing the symbol frames.

**Sym:** Select one of the available geometric characteristic symbols.

**Tolerance (#):** Specify three parameters - the tolerance value and two symbols, diameter and material condition, preceding the value and succeeding it, respectively. The diameter symbol can be inserted by toggling the box preceding the tolerance value field.

**MC (Material Conditions):** Relevant for features that can vary in size.

- **Maximum material condition (M or MMC):** A feature contains the maximum amount of material stated in the limits. At MMC, a hole has minimum diameter, whereas a shaft has maximum diameter.
- **Least material condition (L or LMC):** A feature contains the minimum amount of material stated in the limits. At LMC, a hole has maximum diameter, whereas a shaft has minimum diameter.
- **Regardless of Feature Size (S or RFS):** A feature can be any size within the stated limits.

**Datum (#):** A theoretically point, axis, or plane from which you make measurements and verify dimensions. Usually, two or three mutually perpendicular planes perform this task best. These are jointly called the datum reference frame. Specify one of three datum references - primary, secondary, and tertiary (A, B, and C). The datum reference can consist of a value and a modifying symbol.

**Height Protected Tolerance Zone:** Controls the variation in height of the extended portion of a fixed perpendicular part and refines the tolerance to that specified by positional tolerances. Enter a value in the data entry field. A projected tolerance zone symbol (P) can be inserted or discarded by toggling the box succeeding the height entry field.

Use the **Format** page to set the parameters for the lines and text of the Tolerance.

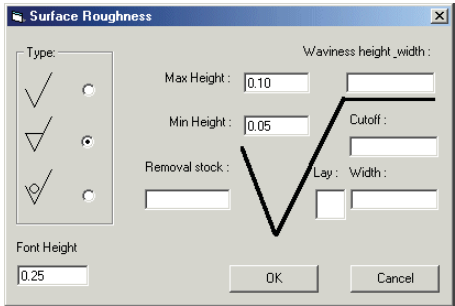
To edit a tolerance symbol, open its **Properties** window (see "Object Properties" on page 79) and click **Tolerance**.

**Surface Roughness**

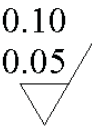
*Available in TurboCAD Pro and Platinum only*



Use the **Surface Roughness** window to enter the symbol parameters.



Click OK when finished, and locate the symbol in your drawing.



To edit a surface roughness symbol, open its **Properties** window (see "Object Properties" on page 79). Open the **Other** page, click **Surface Roughness**, and click **Go To Page**.

This opens the original design window, in which you can change any symbol parameters.

**Hatching**

Hatching is a way to fill a closed 2D object with a pattern. The pattern that will be used, or whether the fill will be solid, is set in the **Brush** page the **Hatch tools' Properties** window (see "Brush Properties" on page 82).

You can display the **Hatch** toolbar by right-clicking in any toolbar area and selecting **Hatch**.

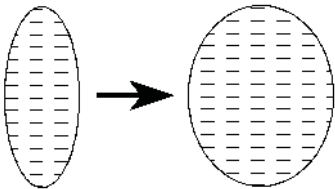


You can also fill a closed object by adjusting the **Brush** settings in the object's **Properties** window. Another way is to use the Colors and Brushes palette; see "Colors and Brushes" on page 49.

*NOTE: If no pattern is specified when an object is selected for hatching, the fill will be solid.*

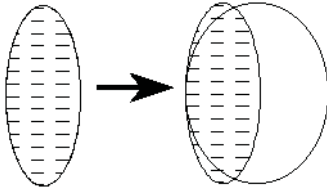
**Associative Hatching**

Associative hatches are linked to their boundaries, and update when their boundaries are modified.





Non-associative hatches are independent of their boundaries.



To set hatching to be associative, check **Associative Hatch** on the **Display** page of the **Drawing Setup (Options / Display)**.

To break the link between an associative hatch and its boundary, select both the boundary and the hatch, and select **Drop Link** from the Inspector Bar or local menu.

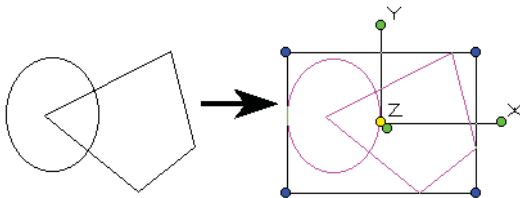


## Hatch

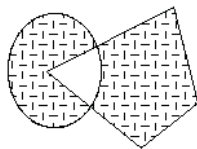


Applies a hatch pattern to a selected closed area. The closed objects must be selected *before* the **Hatch** tool can be used.

1. Set the hatch pattern by opening the **Properties** window of the **Hatch** tool. On the **Brush** page, select the pattern and other hatch parameters.
2. Use the **Select** tool to select the objects you want to hatch.



3. Click **Hatch** or select **Format / Create Hatch**.



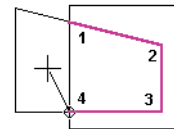
The hatch is not applied to overlapping areas.

## Path Hatching

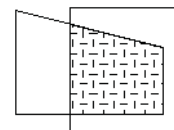


Applies a hatch pattern to a closed boundary of segments, defined manually.

1. Set the hatch pattern by opening the **Properties** window of the **Hatch** tool. On the **Brush** page, select the pattern and other hatch parameters.
2. Activate **Path Hatching**.
3. Select points to define segments of the boundary. The points do not have to lie on existing objects; they can be anywhere. A magenta line indicates the progress of the boundary.



4. Select the first point again to close the boundary, or select **Close** from the local menu or Inspector Bar.



### Local menu option:

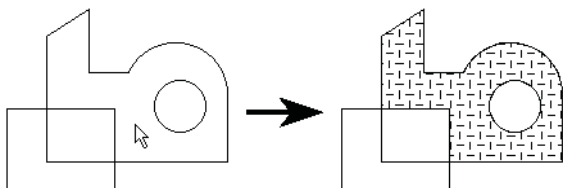
**By Entity:** Use this option to select the entire perimeter of an object. If you have already selected boundary points, **By Entity** will select the perimeter up to the next point. This is especially useful when you are trying to hatch curved objects.

## Pick Point and Hatch

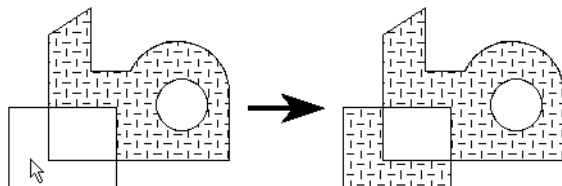


Applies a hatch pattern to a closed boundary, by clicking one point within the desired boundary. This tool recognizes overlaps and islands.

1. Set the hatch pattern by opening the **Properties** window of the **Hatch** tool. On the **Brush** page, select the pattern and other hatch parameters.
2. Activate **Pick Point and Hatch**.
3. Click any point within the desired boundary to create the hatch.



4. To hatch another area, you must activate the tool again and pick another point.



## Pick Point Hatching

**Pick Point Hatching** is an enhanced version of Pick Point Hatch.



Applies a hatch pattern to a closed boundary, by clicking one point within the desired boundary. This tool recognizes overlaps and islands.

1. With Pick Point Hatching you can pre-set the hatch's pattern Line Color, Line Pattern, Line Width, Brush Color and Brush Pattern from the Properties toolbar directly above the drawing area.
2. Activate **Pick Point and Hatch**.
3. Click any point within the desired boundary to create the hatch.
4. You can hatch additional areas by clicking until you cancel out of the tool.

## Editing a Hatch Pattern

You can change a hatch pattern by selecting an object and changing the pattern listed on the **Brush** page of its **Properties** window (see "Brush Properties" on page 82).

If you want to edit a hatch pattern, you will first have to explode it (see "Exploding Objects" on page 247). For associative hatches, you must first explode the object into its boundary and hatch components.

# 10 Working in 3D

This section focuses on setting up your drawing to work in 3D, creating 3D objects, and manipulating the view so that you can see your objects from any angle.

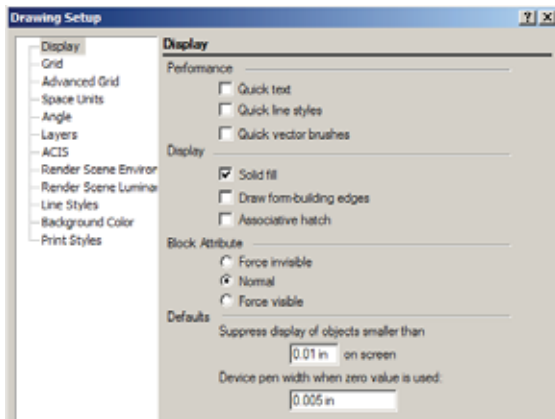
You can use 3D tools (as well as most 3D editing tools) in both wireframe and render modes (*in TurboCAD Pro only*). See "Creating and Editing Objects in Render Mode" on page 496.

## 3D Drawing Setup

Each drawing has a Drawing Setup (**Options / Drawing Setup**) in which you can control how 3D objects are displayed or rendered.

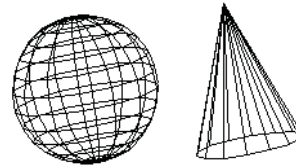
Three of the **Drawing Setup** controls are relevant for 3D models - **Display**, **ACIS**, and **Render Scene Environment**.

### Display

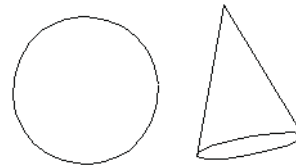


The options on this page that is relevant to 3D is in the **Display** section. For the remaining options, see "Display Options" on page 67.

**Draw form-building edges:** When checked, the entire structure of 3D objects is shown. When not checked, the minimum number of edges needed to show the object will be displayed.



With form-building edges

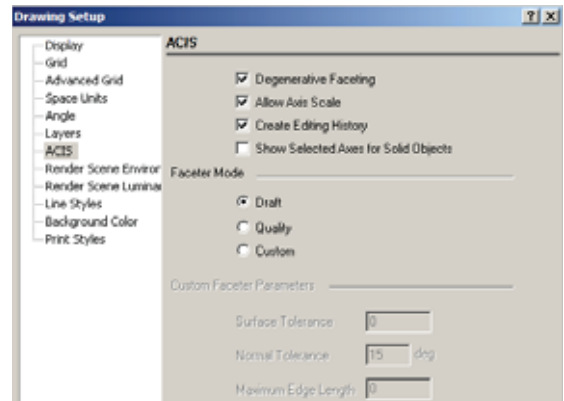


Without form-building edges

NOTE: You can make selected edges non form-building with the **Edit Tool**. See "Editing Nodes of Edges of Exploded Surfaces" on page 429.

## ACIS

Available in TurboCAD Pro and Platinum only



Controls faceting of ACIS 3D solid (not surface) objects, thereby the accuracy of the model representation. Faceting generates polygonal representations of object faces, while maintaining the edge consistency between adjacent faces.

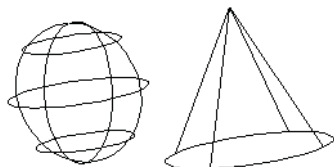
---

**NOTE:** *The faceted representation of a face is also called a mesh.*

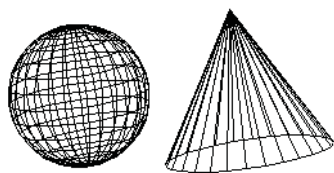
---

Faceting is used during rendering, and faceting refinements may have a significant impact on the rendering process. Tighter facets (higher in number) produce a smoother rendered surface, but slow down the rendering process.

**Degenerative Faceting:** Reduces the number of edges without generating facets. Should be used if your model contains a significant number of 3D objects, because the drawing loading time will be reduced. (For files of tens of megabytes, the loading time reduction may be more than ten times.) However, rendering such a model may take much longer.



**With degenerative faceting**



**Without degenerative faceting**

---

**NOTE:** *To see the effects of degenerative faceting, check **Draw form-building edges** on the **Display** page of the **Drawing Setup (Options / Display)**.*

---

**Allow Axis Scale:** ACIS solid objects can be scaled by entering different scale values in the Inspector Bar for each axis. If not checked, scaling will be uniform, and this uniform values must be entered in the scale fields for at least two axes.

---

**NOTE:** *This parameter is only relevant for ACIS solids. Use the **Selection Info** palette if you are unsure about an object's type. For example, if you create a solid box, you will need to explode it once to make it an ACIS solid.*

---

**Creating Editing History:** Enables you to edit 3D objects, after they are created, in the Selection Info palette. See "Editing 3D Objects using Selection Info" on page 487.

**Show Selected Axes for Solid Objects:** Applies to objects that will be assembled by axis. See "Assemble by Axis" on page 480.

**Faceter Mode:** Three types of faceting are available:

- **Draft:** The default mode. Default settings cannot be changed.
- **Quality:** Provides better quality than **Draft**, but the default settings cannot be changed.
- **Custom:** Enables you to control default tolerance parameters.

**Custom Faceter Parameters:** If **Custom** is the faceter mode, you can set the following parameters:

- **Surface Tolerance:** The maximum distance between the facet and the part of the solid it is representing, thereby setting how closely the facets represent the surface.
- **Normal Tolerance:** The maximum angle between two adjacent nodes of a facet (actually the normal deviation), thereby setting how accurately the facets represent the solid, and the rendering quality. This control is usually independent of model size.
- **Maximum Edge Length:** The maximum facet edge length. As the length decreases, the number of facets increases. This is the way to subdivide facets into more planar faces.

## Render Scene Environment



Options for rendering that control the background and foreground. These effects can be seen only when using **Quality** rendering. See "Rendering" on page 495 and "Environments" on page 530.

## 3D Views

While working in 3D, there are many ways to change how you view your drawing. Views are seen by the "camera" - equivalent to your eye. The camera is located in a specific place and faces a specific direction. See "The Camera" on page 419.

There are several standard orthographic and isometric views, and if you need additional perspective you can change how the camera is oriented. You can move the camera using the **Move Camera** commands (see "Camera Movements" on page 421), and dynamically examine the model using the **Walk Through** tools (see "Walk Through Tools" on page 421). To save and display additional views, you can create camera objects (see "Camera Objects" on page 422).

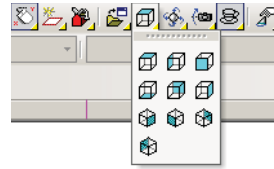
---

*TIP: If you use a wheel mouse, keep the middle mouse button pressed, and drag the cursor around the screen to dynamically rotate the model.*

---

## Standard Views

The ten standard views can be accessed from the **View / Camera** menu.



You can also display this toolbar separately by right-clicking in the toolbar area and selecting **Standard Views**.

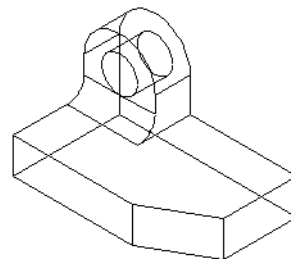


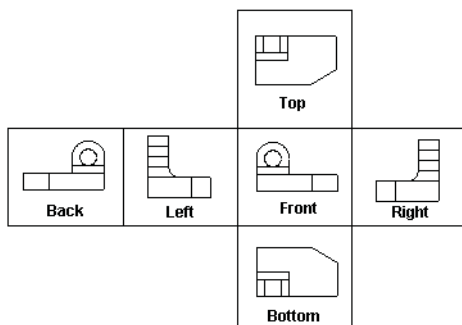
And, by default, these tools are available on the local (right-click) menu.

## Orthographic Views

The first six views on the toolbar are orthographic views - Top (World Plan), Left, Front, Bottom, Back, Right.

If your model was enclosed in a box, these views reflect how the model would appear by looking directly at each of the box faces. If you "unfold" the box, you can see the relationship between the views.





**Front View:** Looks at the XZ plane of the World Coordinate System, while facing +Y.

**Top View:** Looks at the XY plane while facing -Z (down).

---

*TIP: The **Top** view can be also obtained from the **Plan** view by using the **Roll** walk through tool. See "Roll" on page 421*

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**Left View:** Looks at the YZ plane while facing +X.

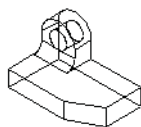
**Bottom View:** Looks at the XY plane while facing +Z (up).

**Back View:** Looks at the XZ plane while facing -Y.

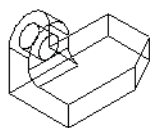
**Right View:** Looks at the YZ plane while facing -X.

## Isometric Views

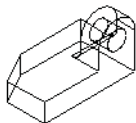
The last four views on the toolbar are isometric - angled views. The views are named for compass directions (Isometric\_SE is the view looking from the southeast).



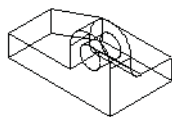
Isometric\_SE



Isometric\_SW



Isometric\_NE



Isometric\_NW

## Saving 3D Views

You can save 3D views using the methods described in "Saving Views" on page 102. In addition, you can use camera objects to save views with specific parameters and open them in their own windows. See "Camera Objects" on page 422.

## 3D Coordinate Systems

Each model has an internal, fixed coordinate system - the WCS, or World Coordinate System. The WCS cannot be changed, but you can define your own coordinate system - the UCS, or User Coordinate System. When you start a new drawing, the UCS by default coincides with WCS. All objects are defined and manipulated with respect to the current UCS.

- Selecting an object using **2D Selector** mode (see "2D / 3D Selector" on page 181) changes the UCS (and therefore the workplane) to the one attached to the selected object.
- When you move an object in **3D Selector** mode, or an object that is part of a group or block, the UCS and workplane do not change. The Entity Coordinate System and its workplane travel with the object.

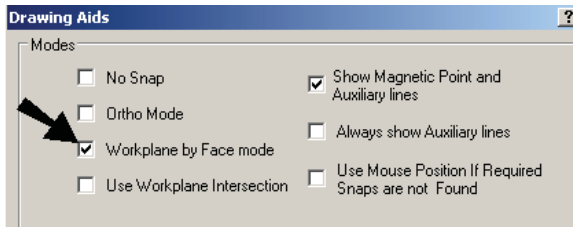
The UCS indicator is helpful if you want to see how the UCS moves. You can display markers for the UCS and WCS via the Local menu toolbars menu, and customize the markers' appearance via the **Preference** page of the **Program Setup (Options / Preference)**. The WCS indicator is always shown at the lower left corner of the screen; the UCS indicator is shown at the UCS origin.

## Workplanes

A workplane is the plane on which 2D objects are created, and on which most 3D objects are based. In 2D, you always work on the same workplane - the XY plane of the current UCS. In 3D, however, the workplane may change frequently.

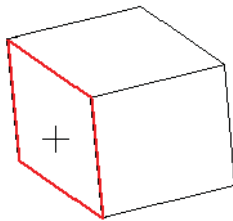
In 3D, the workplane is still in the XY plane by default. But if you have 3D objects in your model, you can temporarily move the workplane to the facet where your cursor is. This

is set in the **Drawing Aids** window, which can be opened by right-clicking on the SNAP or GEO field at the bottom of the screen.

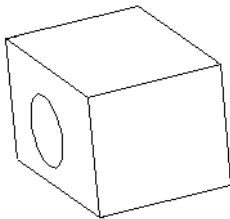


NOTE: For details on **Drawing Aids**, see "Snap Settings" on page 108.

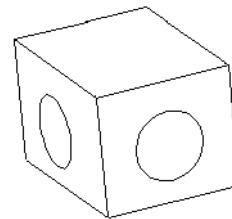
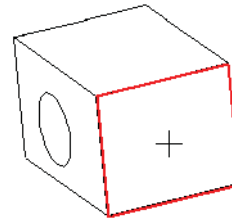
When **Workplane by Face mode** is checked, the facet will highlight in red when the cursor passes over it.



The object you draw will be placed on that facet.

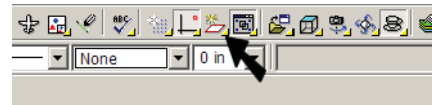


The facet workplane is temporary; after the object is created, the workplane returns to the XY plane. If you highlight another facet, that workplane will be temporarily active for the next object you create.



NOTE: If you create an object with no facet highlighted, the default workplane will be used.

If you uncheck **Workplane by Face mode** in **Drawing Aids**, the workplane will always be the default workplane. If you don't want to use the XY plane of the current UCS, you can use one of the **Workplane** commands. These commands can be found on the flyout toolbar on the main toolbar.



You can also display the **Workplane** toolbar by right-clicking in any toolbar area and selecting **Workplane**.

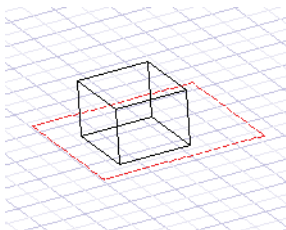


---

NOTE: *The Design Director also contains tools for manipulating workplanes, creating workplanes, and displaying the view according to a selected workplane. See "Design Director: Workplanes" on page 131.*

---

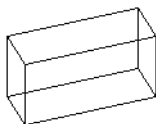
If you are working with the grid turned on (see "Grid" on page 107), the grid will lie on the current workplane.



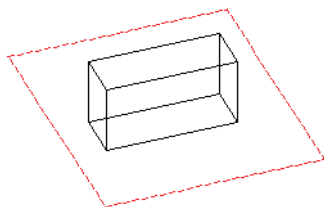
## Displaying the Workplane



Displays or hides the red indicator representing the current workplane.



**WorkPlane hidden**



**WorkPlane displayed**

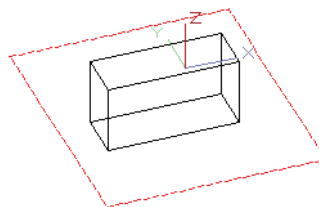
---

NOTE: *If **Workplane by Face mode** is checked in the **Drawing Aids** window, the location of the workplane will change temporarily when you highlight a facet and create an object on it. As soon as the object on that facet workplane is finished, the workplane returns to the default location.*

---

Initially the indicator is sized to fit the entire window, but you can change this, as well as the angle and location of the workplane (see "Editing the Workplane" on page 369).

You can also display the origin of the workplane coordinate system by opening the **Preferences** (Options / Preference) and checking **Show User CS**.



## Fit to Window



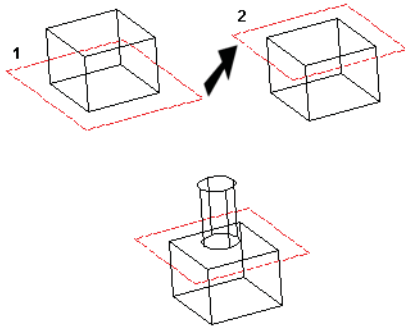
Fits the workplane rectangle to the current window. This is useful after a view change such as zoom or pan.

## Changing the Workplane

While working in 3D, you will probably have to change the workplane frequently to create objects that have the correct location and orientation.



For example, create a cube based on the default workplane. If you want to create a cylinder on the top face of the cube, you must move the workplane up to this face. Then you can place the cylinder in the correct location.



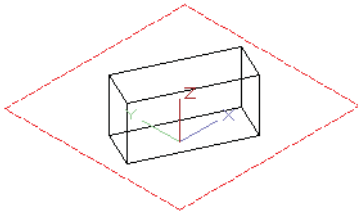
The workplane can also change when you select objects, depending on whether the 2D or 3D Selector is used. See "2D / 3D Selector" on page 181.

### Workplane by View

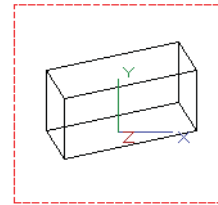


Sets the workplane according to the current view.

1. Position the model to the desired view. In this example, the current workplane is **By World**.



2. Select **By View**. The workplane is turned so that it is parallel to the view plane. The Z axis becomes perpendicular to the drawing screen.



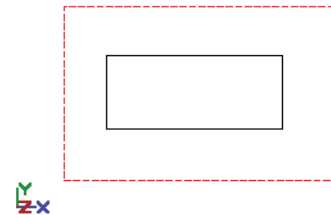
### Workplane by World



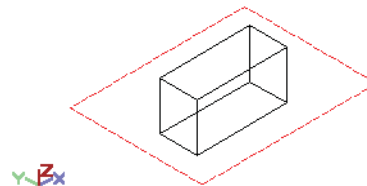
Sets the workplane based on the WCS (World Coordinate System).

**NOTE:** ***By World** is the default workplane when selecting objects that reside on different workplanes.*

You can display the WCS at the lower left corner of the screen by opening the **Preferences** and checking **Show World CS**.



By World, top view



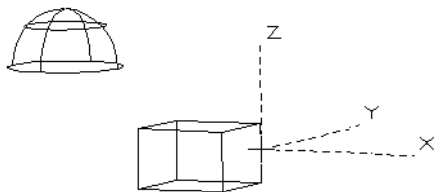
By World, Isometric\_SW view

## Workplane by Entity



Sets the workplane according to the coordinate system of a selected object, block, or group.

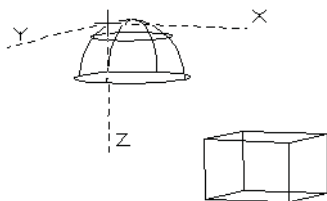
1. Click **By Entity**, and the cursor becomes a dashed representation of the current UCS. Click the desired object.



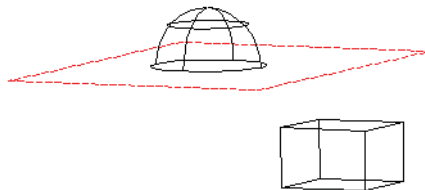
The workplane is placed along the plane on which the object was created.



2. To change the workplane to another object, use the tool again. Click the desired object.



The workplane is positioned according to the selected object.




---

**TIP:** If an object is listed in the Graphics section of the Design Director, you can set the view to an object's **Workplane by Entity**, and set this workplane as the current workplane. See "Design Director: Graphics" on page 133.

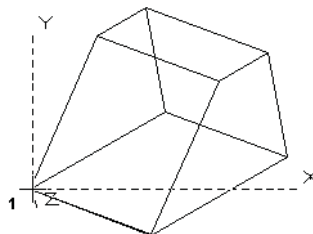
---

## Workplane by 3 Points

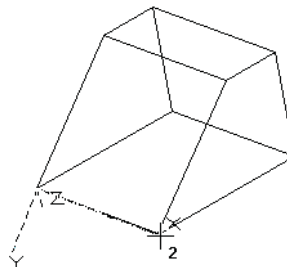


Sets the workplane to fit three points. You can select these points, or enter their coordinates in the Coordinate Fields.

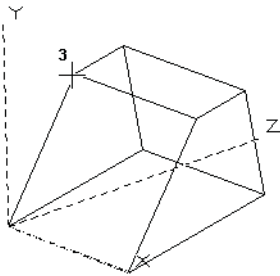
1. Select the UCS origin.



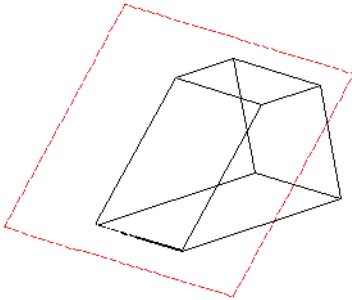
2. Select the second point to define the +X direction.



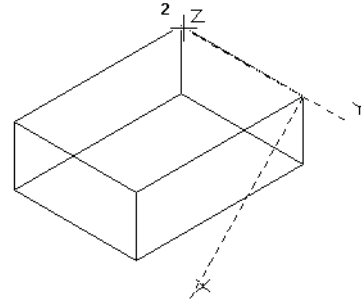
3. Select any point that lies on the desired X-Y plane.



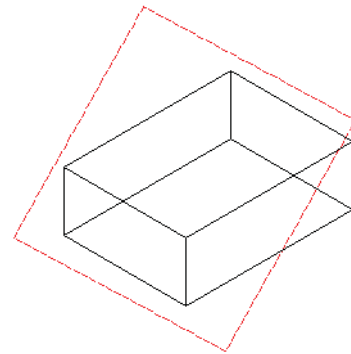
The workplane is created.



2. Select the second point to define the +Z direction.



The workplane is created.

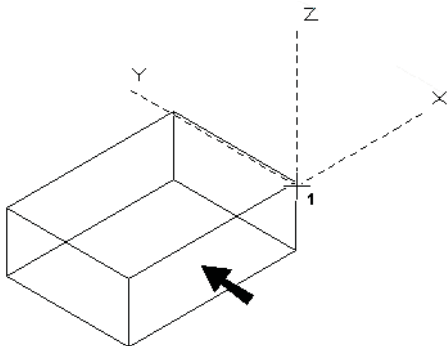


## Workplane by Z axis

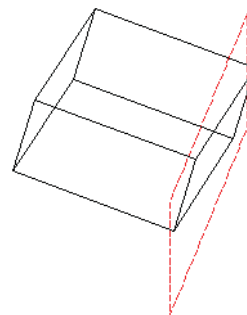


Sets the workplane by defining the Z axis of the UCS. The X-Y plane is perpendicular to this axis.

1. Select the UCS origin, or enter the coordinates in the Coordinate Fields. In this example, the workplane is to be aligned with face shown.



In this example, because of the view, it may not appear that the workplane was aligned correctly. You can rotate the model to verify that the workplane is indeed aligned with the desired plane.

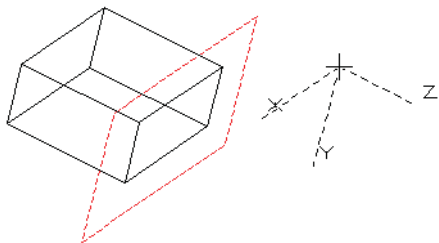


## Workplane Origin

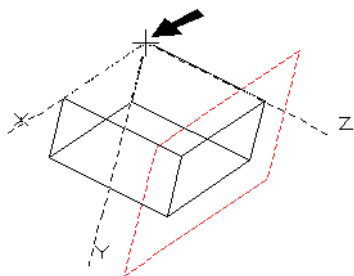


Sets the workplane parallel to (offset from) the current workplane, by relocating the UCS origin.

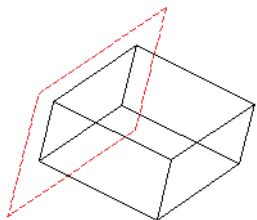
1. When you select UCS Origin, the cursor becomes a set of dashed axes, with the X-Y plane parallel to the current X-Y workplane.



2. Select a new UCS origin, or enter the coordinates in the Coordinate Fields.



The parallel workplane is created.



## Workplane by Facet



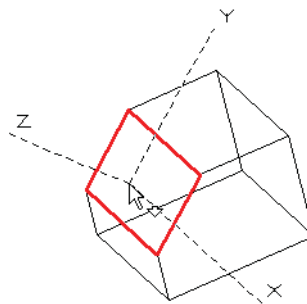
Sets the workplane to be aligned along a facet of a solid.

---

NOTE: The workplane is set to a facet automatically by default, as long as **Workplane by Face mode** is checked in the **Drawing Aids** window. You can open this window by right-clicking on the SNAP or GEO field at the bottom of the screen. See "Snap Settings" on page 108.

---

1. Move the cursor to the desired facet, which is highlighted in red, and the workplane axes are displayed.

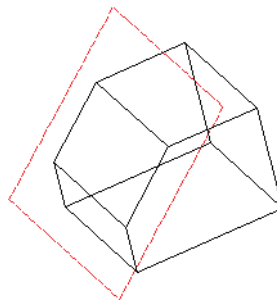



---

NOTE: To select a facet behind or in front of the indicated facet, use the Page Up and Page Down keys.

---

2. Click to create the workplane.



## Saving and Recalling WorkPlanes



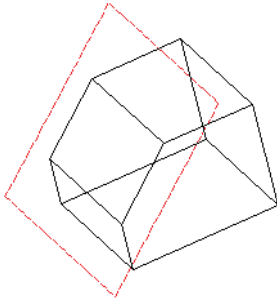
Saves workplanes so that you can recall them later.

---

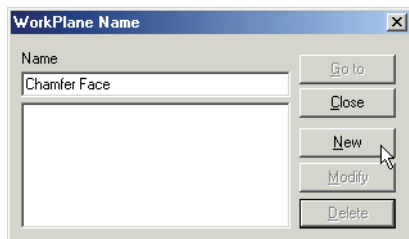
*NOTE: You can also create saved workplanes in the Design Director. See "Design Director: Workplanes" on page 131.*

---

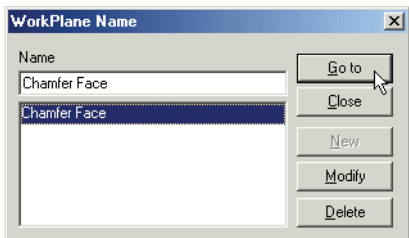
1. Create the desired workplane.



2. Click **Named**, and assign a name for the workplane. Click **New** to add it to the list.



3. Now, when a different workplane is active, you can recall the one you saved. Click **Named** again, find the desired workplane on the list, and click **Go to**. (You can also double-click on the name.)



## Previous



Recalls the previous workplane. Using this tool repeatedly will toggle between the current and previous workplanes; it will not scroll back through several workplanes.

## Editing the Workplane

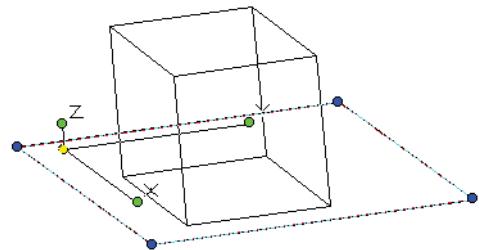


Enables you to move or rotate the workplane, and to scale the workplane indicator.

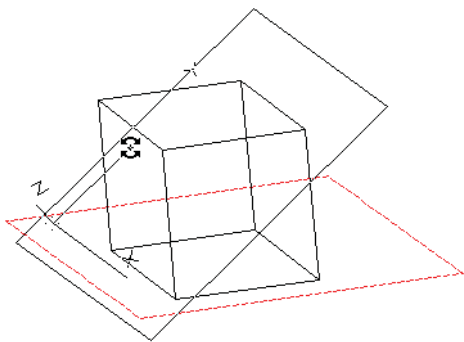
- Drag the blue handles to change the scale (only affects the indicator).
- Use the green handles to rotate the workplane about a specific axis.
- Use the yellow reference point to move the workplane.
- Press D to select the workplane's reference point if you want to move it.

You can also use the scale, position, and rotation fields in the Inspector Bar.

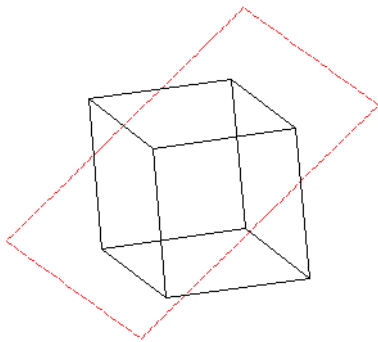
1. Click **Edit**, and the workplane is displayed with editing handles.



2. Click on a green rotation handle (Y in this case), and move the mouse to rotate the workplane.



3. Click anywhere outside the workplane to finish and accept the new position.



## Display Intersections with 3D Objects

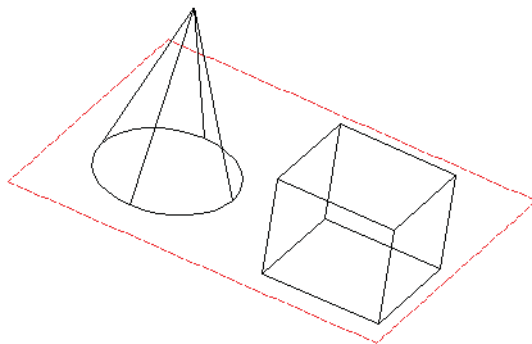


Enables you to visualize where the current workplane intersects with 3D objects. This is a toggle command; if selected, the interactions will always be displayed whenever the workplane is displayed.

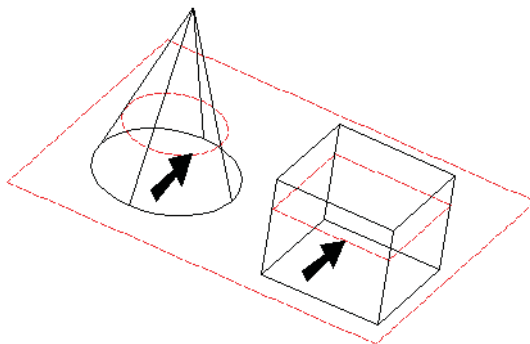
---

NOTE: You can snap to points along intersection lines and curves. See "Workplane Intersections Snap" on page 114.

---



**Workplane without interactions**

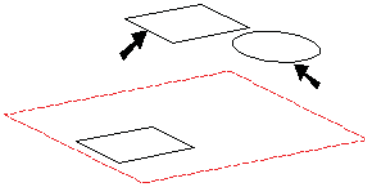


**Workplane with interactions**

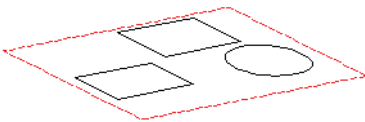
## Place on WorkPlane

Enables you to place 2D objects on the current workplane.

1. Select the objects you want to move.



2. Select **Format / Place on Workplane**. The objects are moved onto the current workplane.



This tool can be useful, for example, for objects that have been selected with the **3D Selector** (see "2D / 3D Selector" on page 181) and moved from their original workplane. If you try using a 2D tool like **Fillet** on two lines that sit on different workplanes, the tool will not work; one of the lines must be moved to the workplane of the other.

## Creating 3D Objects

You can display the **3D Object** toolbar by right-clicking in any toolbar area and selecting **3D Object**.

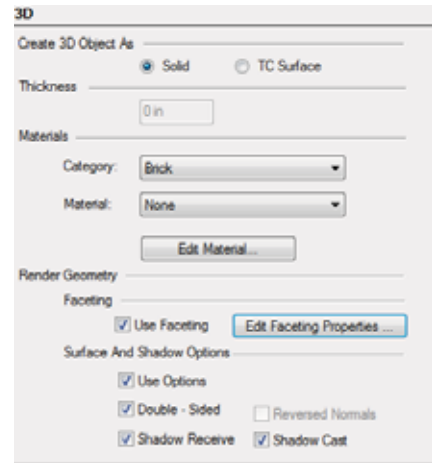


There are three types of 3D objects you can create:

- **Standard 3D objects:** Objects created entirely within as 3D tool. See "Standard 3D Objects" on page 373.
- **3D Profile objects:** Objects created by performing functions on 2D profile objects. See "Profile Objects" on page 388.
- **Modified 2D objects.** 2D objects that are assigned a thickness. See "Creating 3D Objects by Editing 2D Objects" on page 409.

## 3D Properties

Controls how objects are created, and their materials.



**NOTE:** For options that control how 3D objects are displayed, see "ACIS" on page 359.

The ACIS engine is Available in TurboCAD Pro and Platinum only. Therefore, you can only work with solids and surfaces in the Pro version, though TurboCAD surfaces are available in all versions.

**Create 3D Object As:** Choose between **Solid** (ACIS representation) and **TC Surface** (TurboCAD representation). 3D objects are created as solids by default, but you can change this **Properties** setting for all 3D tools, or for individual objects.

- **Solid** objects are created using the ACIS solid modeling engine. Solids are more realistic than surfaces, because objects have volume as well as shape. The solid model assumes that a 3D object comprises a framework, a "skin" (set of surfaces) encasing the framework, and inner "body."
- **TC Surface** objects are created using the TurboCAD internal graphics engine. The "inner body" concept does not apply to surfaces, but when an object is trimmed or cut, the resultant object is entirely covered by surface elements.

You can create surfaces that are ACIS objects by converting existing objects. See "Surface and Solid Conversion Operations" on page 466.

---

*TIP: You can explode a solid object twice to turn it into an exploded (node-editable) surface object.*

---

**Thickness:** Relevant for 2D objects. Assigning a 2D object a thickness makes it a 3D object. Closed 2D objects become solid or surface volumes; open objects (lines, arcs) become surfaces. See "Creating 3D Objects by Editing 2D Objects" on page 409.

---

*NOTE: The thickness is assigned perpendicular to the workplane in which the object was drawn, regardless of the current workplane.*

---

**Materials:** Apply photorealistic colors and textures to the selected 3D object (your model) by using preset materials or by creating new materials. Materials can also be assigned By Block. When set By Block the object will use the material assigned to the block instance in which it resides.

Materials are only viewable in certain render modes. See "Materials" on page 522.

## Render Geometry

*Available in TurboCAD Pro and Platinum only*

These properties are used to speed up rendering and have more efficient memory use. These properties work only in render modes and do not affect wireframe or hidden line modes.

### Faceting

A set of options that define the faceter mode for the object. Faceting options can be enabled only for ACIS (solid) objects.

**Use Faceting:** This option turns the local faceter mode on/off. When it is off (by default), this section does not affect the render. If it is on, then render will use the local faceter mode instead of the one from the ACIS dialog to control rendering of this object.

**Edit Faceting Properties:** This button opens the dialog with local faceter mode settings. Dialog parameters fully match the parameters available in the ACIS Custom parameters.

## Section Surface And Shadows

This set of additional options define the behavior of the graphic during rendering.

**Use Options:** This option turns the use of this set of options on/off. If this option is off (by default), then these options do not affect the render. If it is on, then rendering will use the parameters from this section for this object.

**Double-Sided:** (Off by default) When On this option indicates to the rendering engine that the primitive should be treated as double-sided. By default a primitive is single-sided.

**Reversed Normals:** (Off by default) When On this option indicates to the rendering engine that orientation of normals will be reversed.

**Shadow Receive:** (On by default) When Off this option indicates to the rendering engine that the primitive will no longer receive any shadows.

**Shadow Cast:** (On by default) When Off this option indicates to the rendering engine that the primitive will no longer cast any shadows.

## TC Surface Properties

If an object is created as a surface, the options on this page of the **Properties** will be enabled. To create a surface object, **TC Surface** must be checked in the object's **3D** page of its **Properties**.

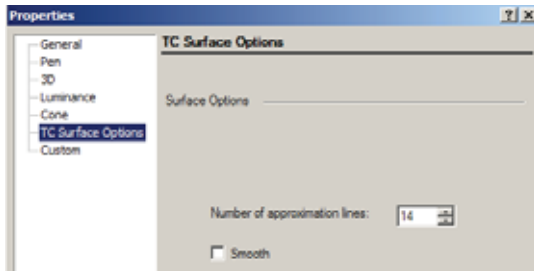
---

*NOTE: Solid (ACIS) objects are Available in TurboCAD Pro and Platinum only.*

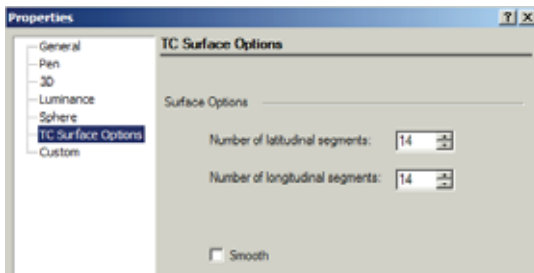
---



Parameters on this page depend on the type of 3D object. For example, these are a cone's parameters:

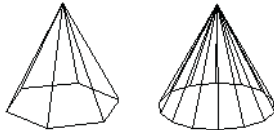


These are the parameters for a sphere:



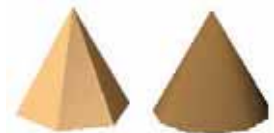
Most 3D objects have a **TC Surface Options** page resembling one of the above examples.

- **Number of approximation lines, latitudinal, longitudinal segments:** The number of segments that represent the object. To see the segments, **Draw form-building edges** must be checked in the **Display** page of the **Drawing Setup (Options / Display)**.



Low vs. high number of approximation lines

- **Smooth:** Smooths an object when rendering.



## Standard 3D Objects

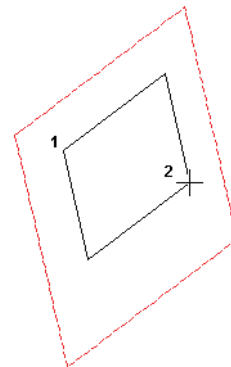
Standard 3D objects, also known as *Primitives*, are created entirely within a 3D tool, with no reference to any other objects.

### Box

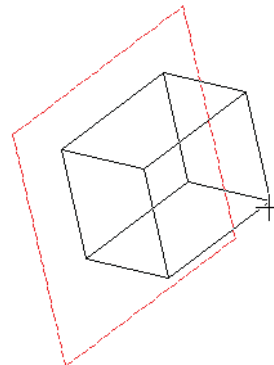


Creates an orthogonal, rectangular prism.

1. Select two opposite corners for the rectangular base. The rectangle is created on the current workplane.



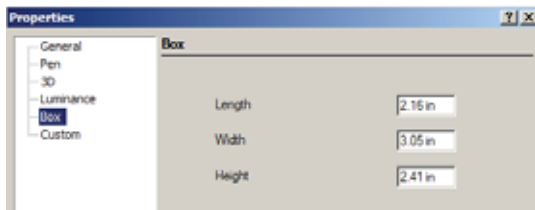
2. Select the third point to define the box height. The height is normal to the workplane.



You can also enter the length, width, and height in the Inspector Bar.

## Box Properties

The **Properties** window of a box contains a **Box** page, in which you can set the length, width, and height.

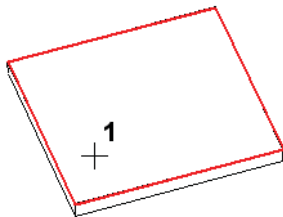


## Rotated Box

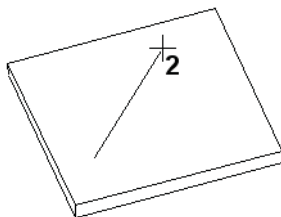


Creates a rectangular prism, based on a rectangle defined by three points.

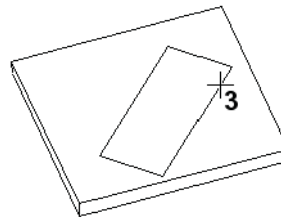
1. The rotated box will be placed on top of this shallow box. The first click defines one corner of the base rectangle.



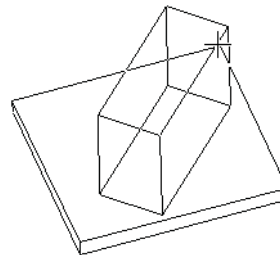
2. The second click defines one edge of the base rectangle.



3. The third click defines the width of the base rectangle.

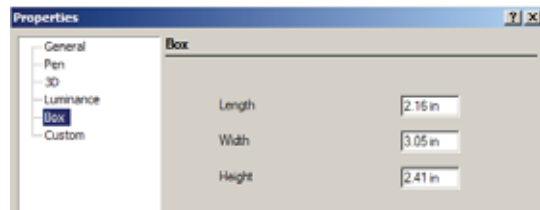


4. The fourth and final click defines the height of the box.



You can also enter the length, width, and angle of the edge of the base rectangle, and the height, in the Inspector Bar.

The **Properties** window of a rotated box contains a **Box** page, in which you can set the length, width, and height.

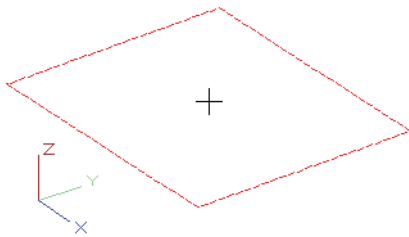


## Sphere

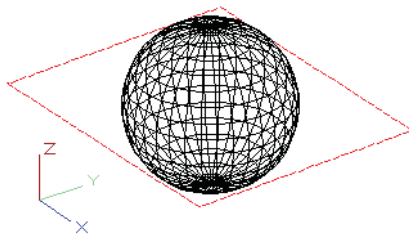


Creates a sphere centered on the current workplane.

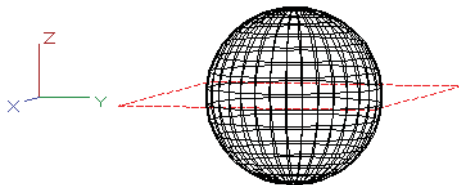
1. Select the center point of the sphere. The point will be projected onto the current workplane.



2. Click to size the sphere, or enter the radius in the Inspector Bar.

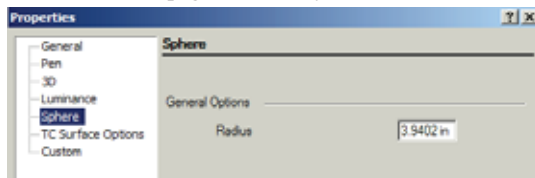


The sphere is created, centered on the workplane.



### Sphere Properties

The **Properties** window of a sphere (and hemisphere) contains a **Sphere** page, in which you can set the radius.



For **TC Surface Options**, see "TC Surface Properties" on page 372.

### Hemisphere



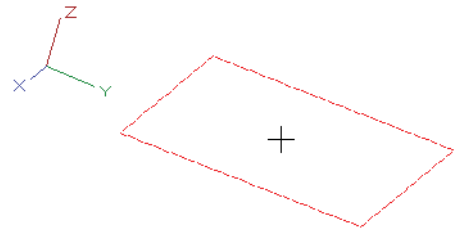
Creates a hemisphere (half of a sphere) with its circular base on the current workplane.

---

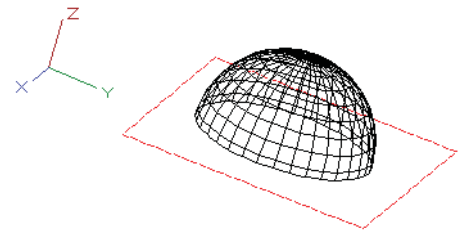
*NOTE: A hemisphere is considered to be a **Sphere** object, and its **Properties** contain a **Sphere** page. For details, see "Sphere Properties" on page 375.*

---

1. Select the center point of the hemisphere. The point will be created on the current workplane.



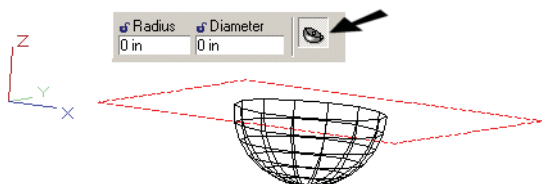
2. Click to size the hemisphere, or enter the radius in the Inspector Bar.



The hemisphere is created on the side of the positive Z axis.

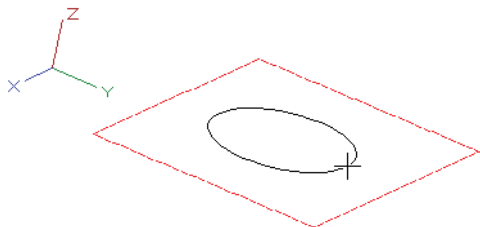
Local menu option:

**Downward:** Creates a hemisphere in the negative Z direction, “below” the workplane.

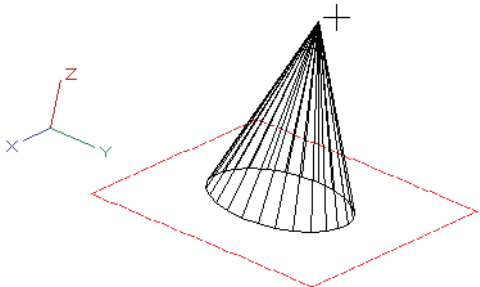
**Cone**

Creates a cone - by default, an object with a circular base that tapers to a point at the top. There are local menu options for creating non-standard cones.

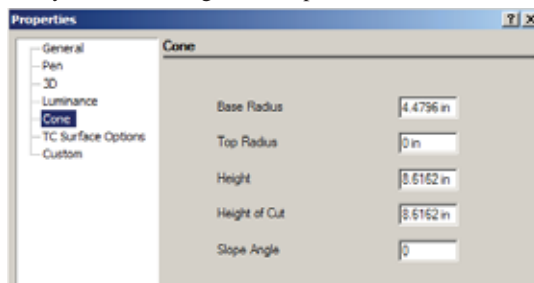
1. Create the circular base by selecting the centerpoint and a point on the circumference. Or enter the radius, diameter, or circumference in the Inspector Bar. The circle is created on the current workplane.



2. Select a third point to specify the cone height, or enter the height in the Inspector Bar. You can create the cone on either side of the workplane. The top point is directly above the base centerpoint.

**Cone Properties**

The **Properties** window of a cone contains a **Cone** page, in which you can set the geometric parameters.



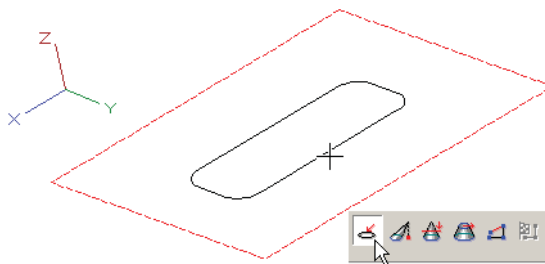
For **TC Surface Options**, see "TC Surface Properties" on page 372.

Local menu options:

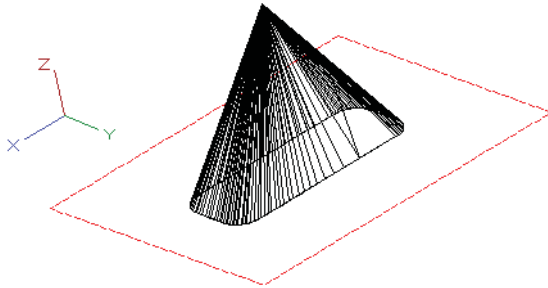
NOTE: These options can be used separately or in combination. **Cut Cone by Plane** and **Cut Cone by Lofting** cannot be used together.

**Specify 2D Base:** Uses an existing 2D object as the base.

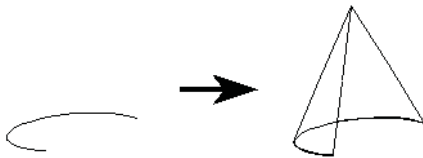
1. Select the 2D object. (You cannot use 2D objects that are contained within a group or block.)



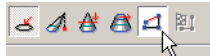
2. Select a point to define the height, or enter the height in the Inspector Bar. The height is perpendicular to the workplane of the 2D object.



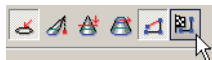
You can also select a 2D open object. In this case, a partial-cone surface object will be created.



You can also use a compound profile, open or closed, for the 2D base. (A compound profile is a series of connected lines and/or arcs.) Make sure **Specify 2D Base** and **Use Compound Profile** are both active, and select the compound profile.

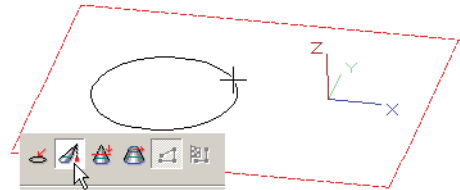


When the profile is selected, select **Finish Selection of Profile**.

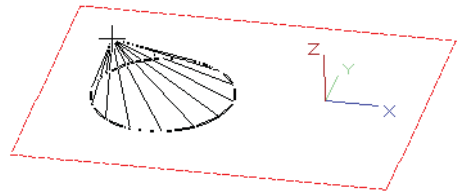


**Specify Object Height Base:** Creates an offset cone by using a specified point for the cone tip.

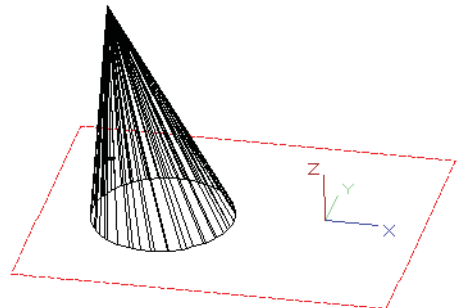
1. Create the circular base.



2. Select the point which is directly under the desired tip.

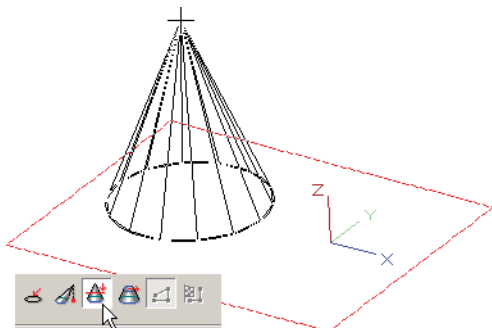


3. Select a third point to specify the cone height, or enter the height in the Inspector Bar. The top point is directly above the selected point.

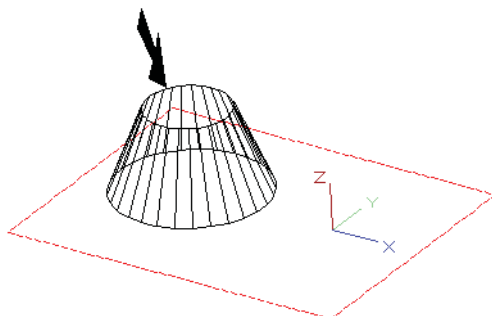


**Cut Cone by Plane:** Creates a truncated cone by cutting off the tip.

1. Create a standard cone.

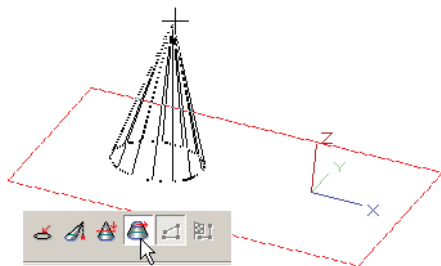


2. Move the mouse outward to create the top of the cone. You can also enter the height of cut in the Inspector Bar.

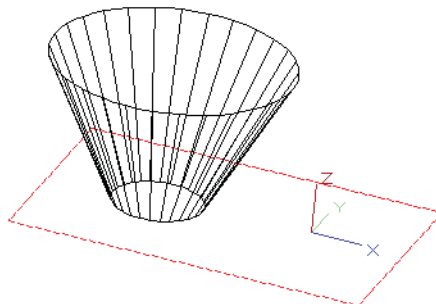


**Cut Cone as Lofting:** Creates an inverted, truncated cone by using a scaling factor for the cone base.

1. Create a standard cone.



2. Move the mouse outward to create the top of the cone. You can also enter a scaling factor in the Inspector Bar.

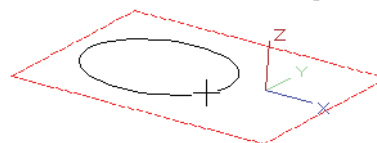


## Cylinder

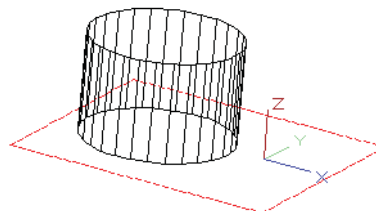


Creates a cylinder - by default, a circular prism.

1. Create the circular base by selecting the centerpoint and a point on the circumference. Or enter the radius, diameter, or circumference in the Inspector Bar. The circle is created on the current workplane.

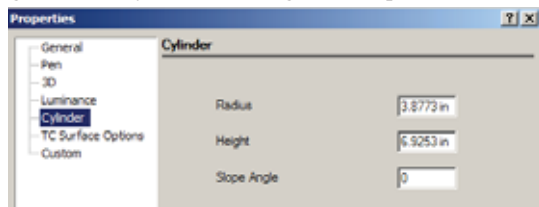


2. Select a third point to specify the cylinder height, or enter the height in the Inspector Bar. You can create the cylinder on either side of the workplane.



## Cylinder Properties

The **Properties** window of a cylinder contains a **Cylinder** page, in which you can set the geometric parameters.

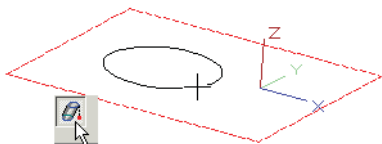


For **TC Surface Options**, see "TC Surface Properties" on page 372.

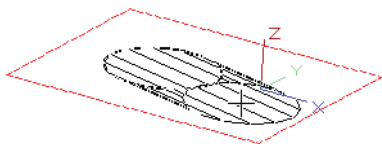
Local menu option:

**Specify Object Height Base:** Creates an offset cylinder by using a specified point for the top face.

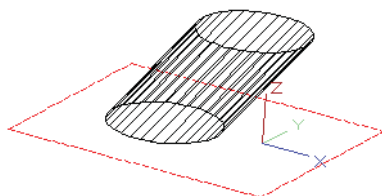
1. Create the circular base.



2. Select the point which is directly under the desired center of the top face.



3. Select a third point to specify the cylinder height, or enter the height in the Inspector Bar.

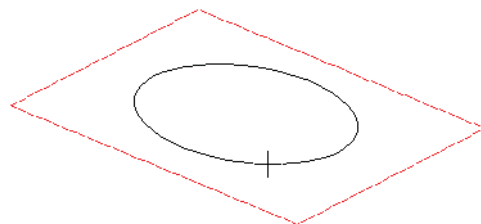


## Torus

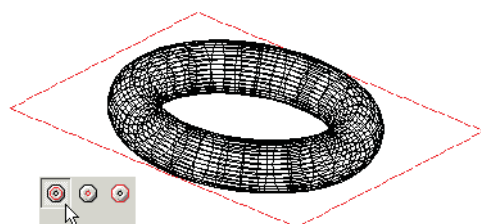


Creates a donut-shaped object by extruding a circle along a circular path.

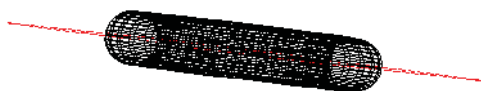
1. Create the circular base by selecting the centerpoint and a point on the circumference. Or enter the base radius in the Inspector Bar. The circle is created on the current workplane.



2. The default option is **Center**, which means the cross-section of the torus uses the base circle as its center. Click to define the tube size, or enter the tube radius in the Inspector Bar.

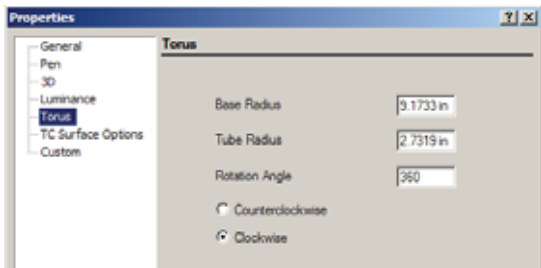


The torus is created, centered on the workplane.



Torus Properties

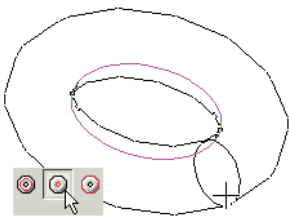
The **Properties** window of a torus contains a **Torus** page, in which you can set the geometric parameters.



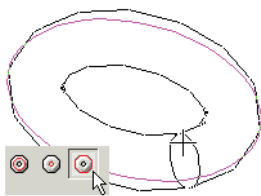
For **TC Surface Options**, see "TC Surface Properties" on page 372.

Local menu options:

**Inner Radius:** The base circle is the inner radius; the torus proceeds outward.



**Outer Radius:** The base circle is the outer radius; the torus proceeds inward.

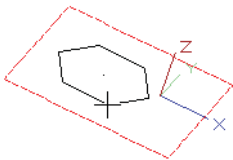


Polygonal Prism

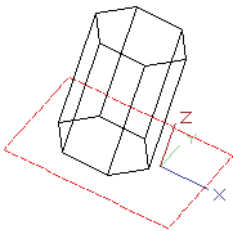


Creates a prism by defining a polygon (multi-sided, equal lengths, closed) as its base shape.

1. Create the base polygon by selecting the centerpoint and one of the vertices. You can enter the number of sides, angle, and radius or side length in the Inspector Bar. The polygon is created on the current workplane.

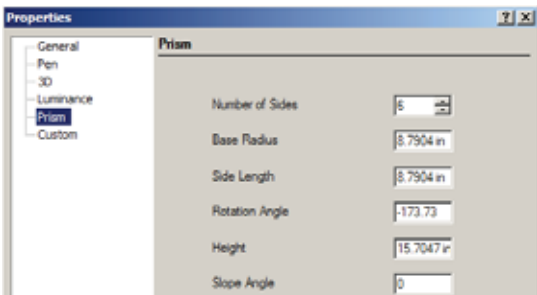


2. Select a third point to specify the prism height, or enter the height in the Inspector Bar. You can create the prism on either side of the workplane.



Prism Properties

The **Properties** window of a prism contains a **Prism** page, in which you can set the geometric parameters.



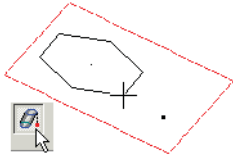
For **TC Surface Options**, see "TC Surface Properties" on page 372.



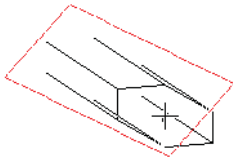
Local menu options:

**Specify Object Height Base:** Creates an offset prism by using a specified point for the top face.

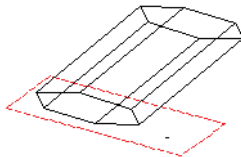
1. Create the base polygon.



2. Select the point which is directly under the desired center of the top face.



3. Select a third point to specify the prism height, or enter the height in the Inspector Bar.



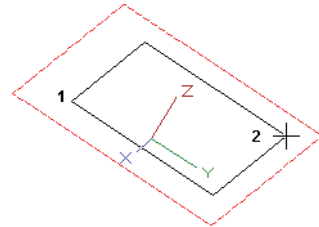
## Wedge



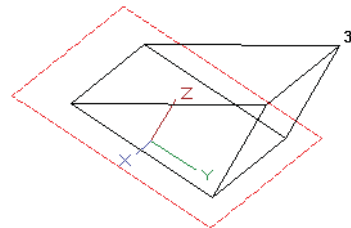
Creates a triangular wedge - a box cut diagonally in half.

1. Create the base rectangle by selecting its opposite corners. You can also enter the length and width in the Inspector Bar. The rectangle is created on the current workplane.

The order of selecting points is important - the wedge will be extruded from the *second* point.

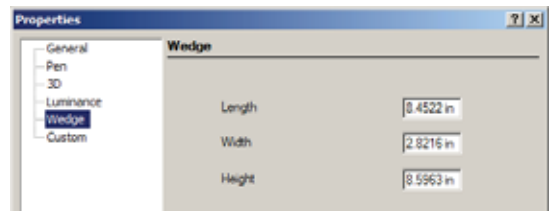


2. Select a third point to specify the wedge height, or enter the height in the Inspector Bar. You can create the wedge on either side of the workplane.



## Wedge Properties

The **Properties** window of a wedge contains a **Wedge** page, in which you can set the geometric parameters.



For **TC Surface Options**, see "TC Surface Properties" on page 372.

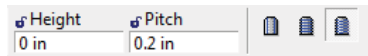
## Thread

Available in TurboCAD Pro and Platinum only



Thread adds a threaded section to any cylindrical surface, including the outside of a cylinder, or the inside of a cylindrical hole. The hole can be the result of using a Boolean to subtract a cylinder from a solid, or created with the hole tool if the hole has no chamfer.

Local menu options:



**Height:** Sets the total height of the thread.

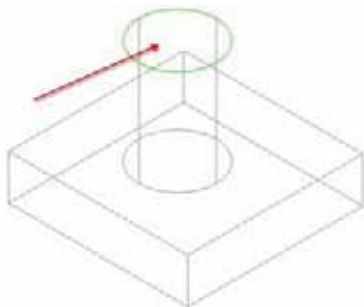
**Pitch:** Set the distance between ridges on the thread.

**Schematic:** Sets the thread display and generation of the thread to a simpler form appropriate for schematics. This form takes the least computation.

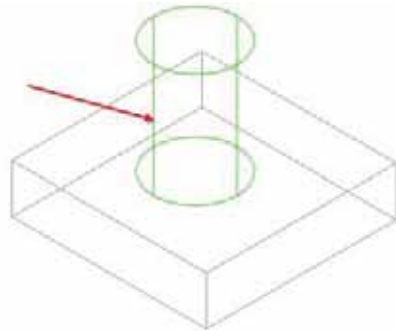
**Simple:** Sets the thread display and generation to a middle form appropriate in which the threads are shown as parallel ridges. This form takes the moderate computation.

**Genuine:** Sets the thread display and generation to an exact form with true helical threads. This form takes the most computation.

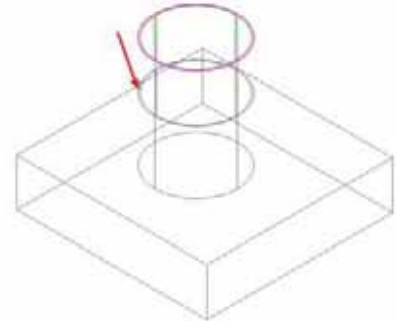
1. Specify the **Pitch** and **Thread** type in the Inspector bar
2. Click on a planar face at one end of the cylinder (or hole). The thread height will be measured from this face.



3. Select the side of the cylinder (or hole).



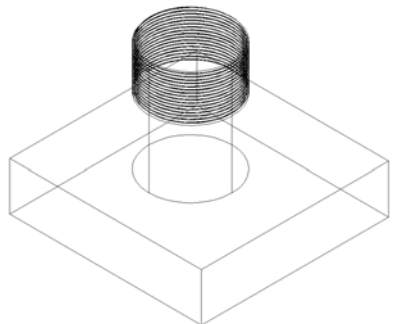
4. Move the mouse to specify the **Height**, or enter a value in the **Height** field of the Inspector Bar.



5. If you are specifying the height by using the mouse, click to finish the thread.

Or

If you are specifying the height by using the Inspector bar, press Enter to set the height.



## 3D Mesh



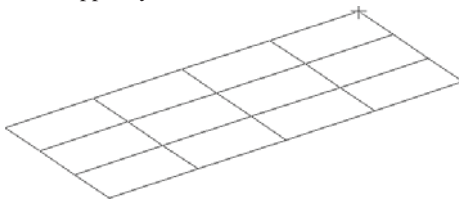
Creates a mesh that represents a set of facets. The base of a mesh is a rectangle with the sides parallel to the X and Y axes.

**NOTE:** The **Terrain** tools provide another way to create a meshed surface. See "Terrain" on page 560.

To create this mesh, TurboCAD defines a matrix consisting of X and Y nodes, with each node separated by a **Step** value. The default number of nodes is 14 for each axis, but this can be changed in the Inspector Bar.

In addition, you can specify **Interpolation nodes**, which are nodes created between the main X and Y nodes. If you use interpolation nodes, Z values are interpolated between main nodes, thus creating a smoother mesh.

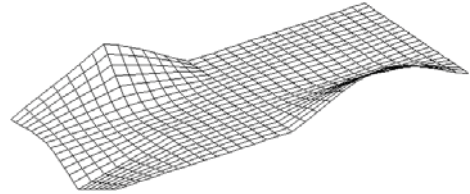
1. In the Inspector Bar, enter the number of nodes and interpolation nodes. If you know the **Step** value, you can enter that as well.
2. Place the rectangular mesh by selecting a corner point. If the **Step** values are not defined, sizing the rectangle will establish them. Notice that the interpolation nodes do not appear yet.



3. A coordinates table appears, in which you can enter Z values for each X-Y node. In this table, you can also modify the position of X and Y mesh lines, but you cannot edit X and Y coordinates of individual nodes at this stage of creating the mesh.

	Y	X1	X2	X3	X4
X		-1.75	0.33	2.41	4.5
Y1	-3.89	0	0	-1	0
Y2	0.4	2	0	0	0
Y3	4.7	0	0	0	0
Y4	9	0	0	0	1.5
Y5	13.3	0	0	0	0

4. Click OK. The mesh now contains the interpolation nodes, and the Z values are linear interpolations between the main X-Y nodes.

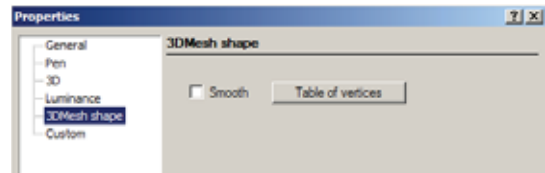


**TIP:** Use the **Edit Tool** to move a single node or a group of nodes (see "Edit Tool in 3D" on page 427).

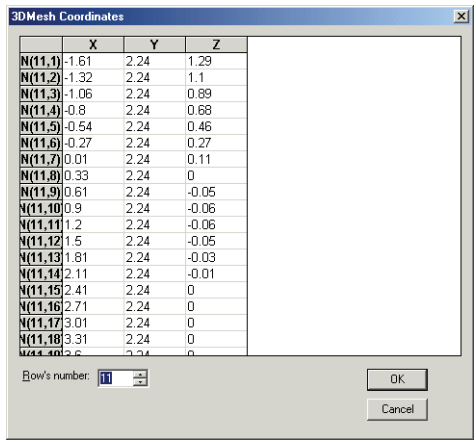


## 3D Mesh Properties

Enables you to change properties of the 3D mesh.



**Table of vertices:** Displays the 3D Mesh Coordinates dialog, in which you can adjust the coordinates of each node. Use the **Row Number** spin control to choose the row of the mesh base.



This is different than the coordinates table you used to create the mesh. In this table you can edit coordinates of all points, including interpolation nodes.

**Smooth:** Gives the mesh a smooth appearance in render mode.

**Helix**

*Available in TurboCAD Pro and Platinum only*

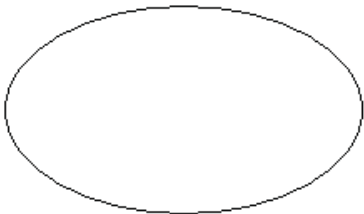


This tool creates a curvilinear helix that can be used for creating springs and other helical elements.

- 1. Click to place the Base Point of the helix.
- 2. Move the mouse to specify the Base Radius, or enter a value in the Radius or Diameter fields of the Inspector Bar.
- 3. If you are specifying the radius by using the mouse, click to finish the extrusion.

Or

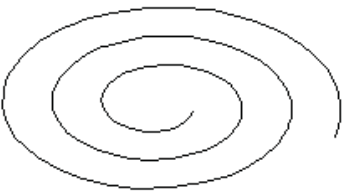
If you are specifying the radius/diameter by using the Inspector bar, press Enter to set the radius/diameter.



- 4. Move the mouse to specify the Top Radius, or enter a value in the Radius or Diameter fields of the Inspector Bar.
- 5. If you are specifying the radius by using the mouse, click to finish the set the Top Radius.

Or

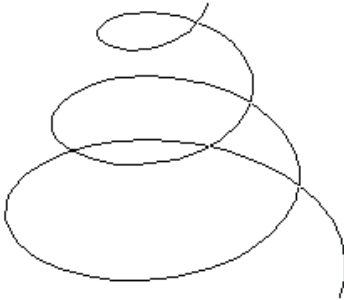
If you are specifying the radius/diameter by using the Inspector bar, press Enter to set the radius/diameter.



- 6. Move the mouse to specify the height of the helix, or enter values in the Turns and Turn Height fields of the Inspector Bar.
- 7. If you are specifying the height by using the mouse, click to finish the helix.

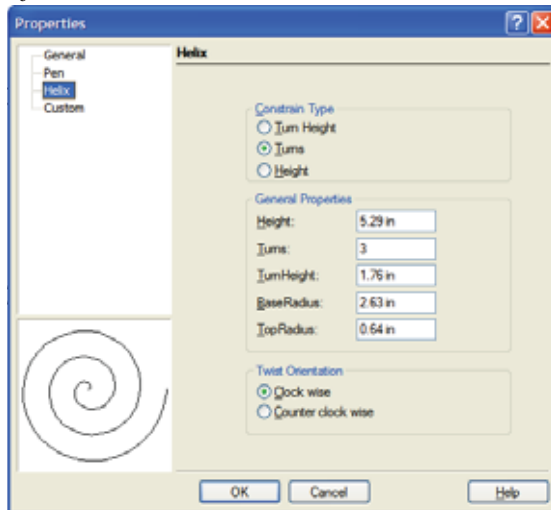
Or

If you are specifying the Turns and Turn Height by using the Inspector bar, press Enter to set the finish the helix.



### Helix Properties

The Properties window of a Helix object contains a Helix page, in which you can set parameters defining how the object is created.



**Constrain Type:** This controls the dynamic for changing the properties by semi-locking the selected property. When one of the other properties is changed, the designated constrained property remains constant, and the third property is altered so that the constrained property can remain constant. The constrained properties work in the following pattern:

- If the Turn Height property is constrained, changing the Height property will also change the Turns property, and changing the Turns property will change the Height property. If the Turn Height value is changed the Height property will also change.
- If the Height property is constrained, changing the Turn Height property will also change the Turns property, and changing the Turns property will change the Turn Height property. If the Height value is changed the Turn Height property will also change.
- If the Turns property is constrained, changing the Height property will also change the Turn Height property, and changing the Turn Height property will change the Height property. If the Turns value is changed the Height property will also change.

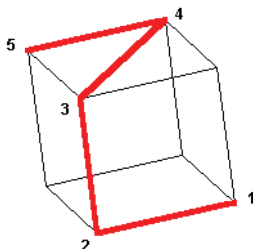
General Properties:

- **Height:** Sets the total height of the helix.
- **Turns:** Sets the number of turns in the helix.
- **Turn Height:** Sets the height of each complete turn in the helix.
- **Base Radius:** Sets the radius of the first turn of the helix.
- **Top Radius:** sets the radius of the final turn of the helix.
- **Twist Orientation:** these options specify the direction that the helix will turn, clockwise, or counter-clockwise.

## 3D Polyline



Similar to 2D Polyline (see "Polyline" on page 138), but you can place line segments anywhere in 3D space; you are not restricted to the current workplane.



You can enter coordinates in the Coordinate Fields, but there are no input fields in the Inspector Bar for this tool.

Unlike the 2D Polyline, you cannot add arc segments, nor can you manipulate line thicknesses.

---

**NOTE:** If you want to create a 3D polyline by joining existing linear and/or non-linear segments, use the **3D Polyline** option of **Join Polyline**. See "Join Polyline" on page 243.

---

**3D Polylines** can be used in two other tools, **3D Fillet** and **Pipe**.

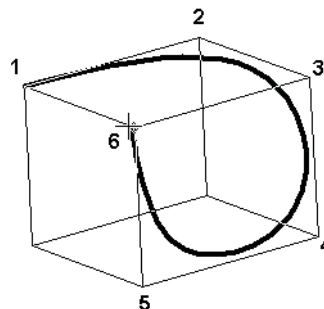
### 3D Spline by Control Points

**Menu:** Insert / 3D Object / 3D Spline / By Control Points

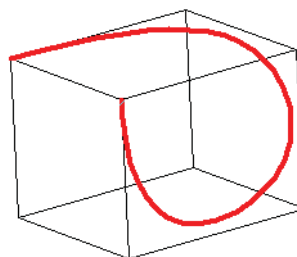


Creates a curve by selecting its control points, similar to the related 2D function (see "Spline by Control Points" on page 170), but you can place line segments anywhere in 3D space. The curve does not pass through the points (unlike a Bezier curve), rather it uses them as a guide.

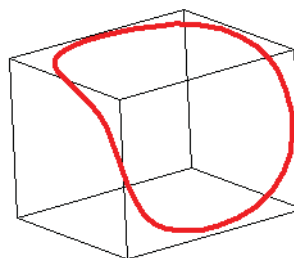
1. Select the control points in the desired order.



2. After selecting the last point, select **Finish** from the local menu or press Alt+F. You can also double-click the last point.



3. If you want to close the spline, choose **Close** from the local menu instead of **Finish**. In this case, the spline will no longer touch the first and last control points.



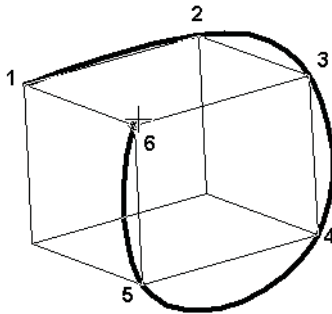
Once created, you can use the **Edit Tool** to change the shape of a spline and add knots. See "Editing Splines and Bezier Curves" on page 227.

### 3D Spline by Fit Points

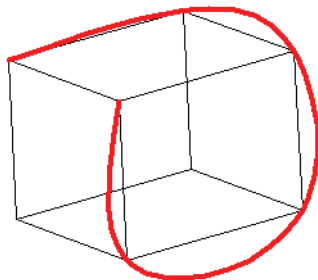


Creates a curve by selecting its control points, similar to the related 2D function (see "Spline by Fit Points" on page 170), but you can place line segments anywhere in 3D space. The curve passes through the points (unlike a spline, which uses control points as a guide).

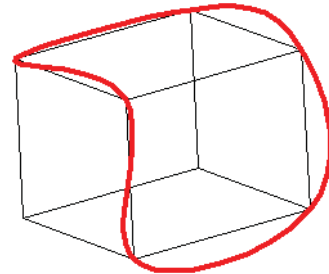
1. Select the control points in the desired order.



2. After selecting the last point, select **Finish** from the local menu or press Alt+F. You can also double-click the last point.



3. If you want to close the spline, choose **Close** from the local menu instead of **Finish**. In this case, the spline will no longer touch the first and last control points.



Once created, you can use the **Edit Tool** to change the shape of a spline and add knots. See "Editing Splines and Bezier Curves" on page 227.

### Bolt

*Available in TurboCAD Pro and Platinum only.*



**NOTE:** You can display the **Special Tools** toolbar by right-clicking in any toolbar area and selecting **Special Tools**.

Creates a threaded bolt with a cylindrical, hexagonal, or square head. Hexagonal and square heads are automatically filleted.

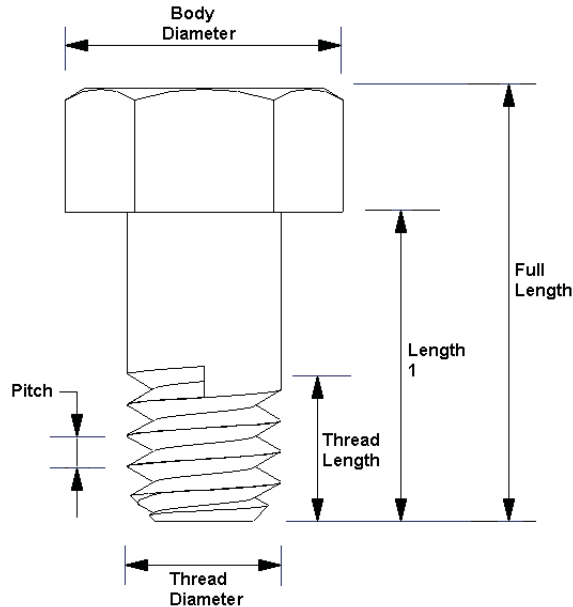
1. Use the Inspector Bar or local menu to select the type of bolt to be created: circular, hexagonal, or square head.



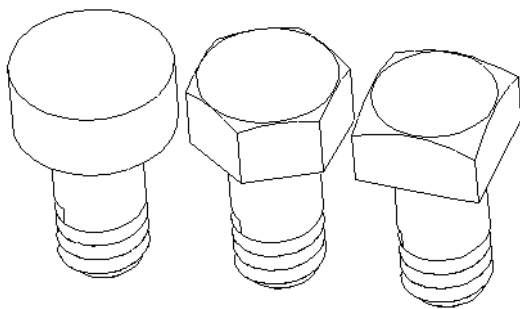
2. Select the center point of the bolt's base (bottom of threads). You can then specify dimensions using the mouse or by entering them in the Inspector Bar fields.

Length1	ThrDiam	ThrLen	Pitch	BodyDiam	FullLen
0	0	1	0.5	0	0

The bolt parameters are shown below:



- After the last click, or after pressing Enter if you use the Inspector Bar, the bolt is created. The three types are shown below.



## Profile Objects

Profile objects are based on existing 2D objects. The original 2D profile remains unmodified by the 3D tool that references it.

One advantage to profile objects is that you can change them by simply editing the profile (or profiles) on which they are based. See "Profile Editing" on page 429.

## Compound Profiles

Available in TurboCAD Pro and Platinum only

A compound profile is an open or closed chain of connected curves or lines, or any set of closed curves or closed polylines. You can use compound profiles when you do not want to create a polyline, or convert a chain into a polyline. Compound profiles can be used as paths for sweeps etc., and as profiles that are swept along paths.

You can use blocks as compound profiles, however there are restrictions. You can only use one element in a block as a path for any given operation. Also, only closed sequences of curves, lines etc. (chains) or closed objects can be used to sweep, revolve etc. All such closed objects in the block selected to sweep, revolve, etc will be used in the resulting 3D object.

## Prism



Creates a 3D object between two 2D objects that lie on different planes. The planes do not have to be parallel.

Both profile objects must be the same type and must have the same number of vertices. For example, you can create a prism between two circles or two rectangles, but not between a circle and a rectangle. Splines and Bezier curves must have the same number of control points.

(If you want to use more than two profiles, or profiles of different type, see "Lofting" on page 402.)

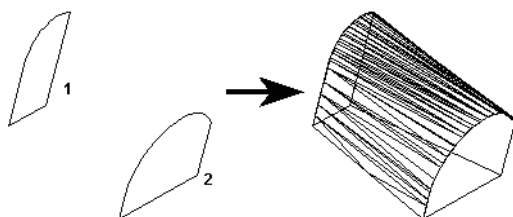


NOTE: A prism is considered to be a **Lofting** object, and its **Properties** contain a **Lofting Shape** page. For details, see "Lofting Shape Properties" on page 404.

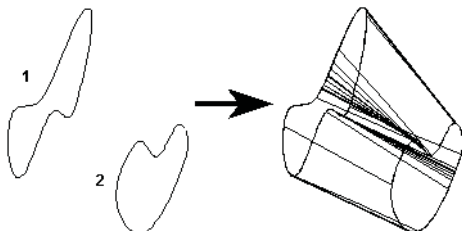
1. If you want to select only simple (single-object) curves, make sure **Use Compound Profile** is **not** selected. If you want to use compound profiles, see "Prism with Compound Profiles" on page 389.



2. Select the two 2D objects.

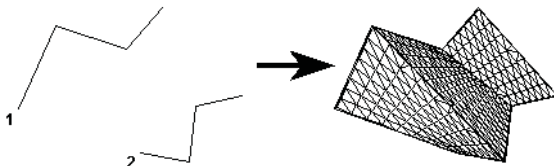


Polyline to polyline



Spline to spline

You do not have to select closed 2D objects. If you use open objects, a 3D surface will result.



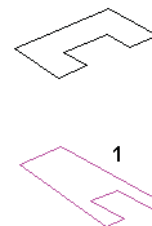
### Prism with Compound Profiles

A compound profile is an open or closed chain of connected curves or lines. You can use compound profiles when you do not want to create a polyline, or convert a chain into a polyline.

1. Make sure **Use Compound Profile** is selected.



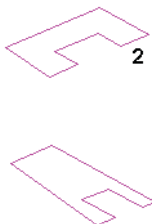
2. Select the first compound profile, which is automatically identified as a chain. To deselect any curve in the chain, select it again (it will turn green). In this example, Profile 1 is a series of connected lines, not a polyline.



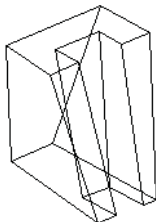
3. When the profile is selected, click **Finish Selection of Profile**, or select it from the local menu.



4. Select the second profile using the same steps.



- When you finish the second profile selection, the prism is created.




---

NOTE: Compound profiles can be modified, which updates the 3D objects upon which they are based. See "Updating Compound Profiles" on page 431.

---

## Simple Extrude



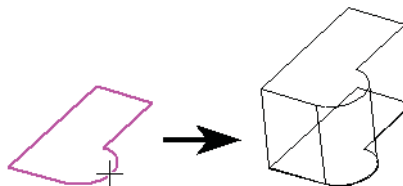
Creates a 3D object by extruding (sweeping) a 2D open or closed profile along a path normal to the workplane of the 2D object. If you extrude an open profile, a surface will be created.

- If you want to select only a simple (single-object) curve, make sure **Use Compound Profile** is **not** selected.
- If you want to use a compound profile, which is a series of connected lines and/or arcs, select **Use Compound Profile**.

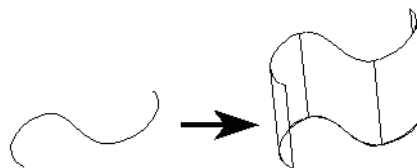


- If necessary, select **Finish Selection of Profile** to continue.

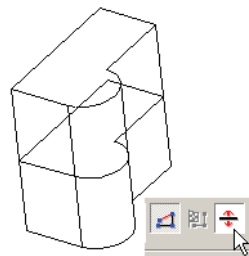
- Select the 2D open or closed profile to extrude. Move the mouse to extrude the profile, or enter a value in the Height field of the Inspector Bar.



If you select an open profile, the resulting 3D object will be a surface.



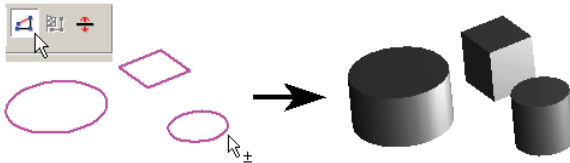
If you select **Two Sided Extrude**, the solid extrusion will be created on either side of the profile.



### Extruding Multiple Profiles

You can also select multiple closed regions, or profiles, to extrude all at one time.

Make sure **Use Compound Profile** is selected. Select the first profile, then press Shift to select additional profiles. Each profile is extruded the same distance.



If you select nested regions, you can create islands, and regions within islands.



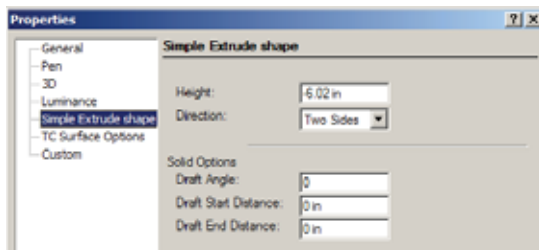

---

NOTE: Compound profiles can be modified, which updates the 3D objects upon which they are based. See "Updating Compound Profiles" on page 431.

---

### Simple Extrude Shape Properties

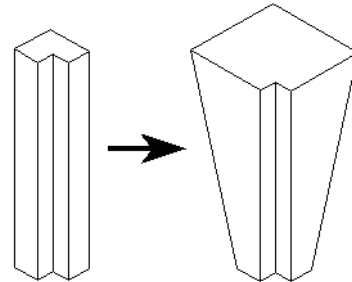
The **Properties** window of an extrude object contains a **Simple Extrude Shape** page, in which you can set parameters defining how the object is created.



- **Height:** The distance of the extrusion.
- **Direction:** Switch between one-sided and two-sided.

**Solid Options:** For these parameters to be accessible, the extrude must be created as a solid. *Available in TurboCAD Pro and Platinum only.*

- **Draft Angle:** Creates an extrusion of increasing or decreasing cross-section. Enter the angle of deviation from the extrusion path.



- **Draft Start / End Distance:** If **Draft Angle** = 0, you can specify a draft angle by entering the offset distances.
- **Offset:** Creates a hole in the extrusion at a distance from the outside of the extrusion equal to the Offset value.

For **TC Surface Options**, see "TC Surface Properties" on page 372.

---

NOTE: Text can be used as part of a compound profile while extruding.

---

### Twisted Extrude



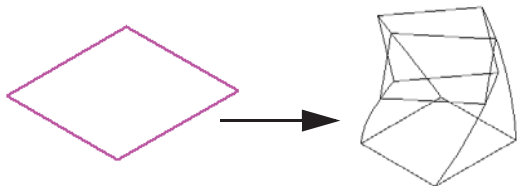
Creates a 3D object by extruding (sweeping) a 2D open or closed profile along a path normal to the workplane of the 2D object and imparting a twist. If you extrude an open profile, a surface will be created.

1. If you want to select only a simple (single-object) curve, make sure **Use Compound Profile** is **not** selected.

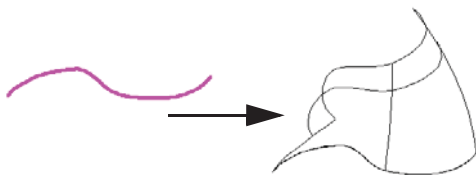
2. If you want to use a compound profile, which is a series of connected lines and/or arcs, select **Use Compound Profile**.



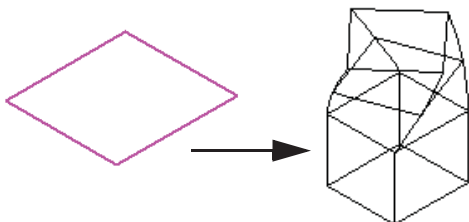
3. If necessary, select **Finish Selection of Profile** to continue.
4. Select the 2D open or closed profile to extrude. Move the mouse to extrude the profile, or enter a value in the Height field of the Inspector Bar.



If you select an open profile, the resultant object will be a surface.



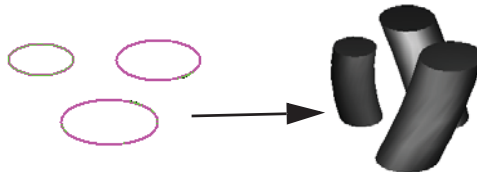
If you select **Two Sided Extrude**, the solid extrusion will be created on either side of the profile.



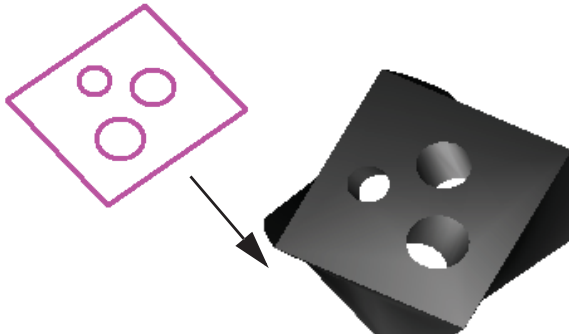
## Extruding Multiple Profiles

You can also select multiple closed regions, or profiles, to extrude all at one time.

Make sure **Use Compound Profile** is selected. Select the first profile, then press Shift to select additional profiles. Each profile is extruded the same distance.



If you select nested regions, you can create islands, and regions within islands.




---

*NOTE: Compound profiles can be modified, which updates the 3D objects upon which they are based. See "Updating Compound Profiles" on page 431.*

---

### Twisted Extrude Shape Properties

The **Properties** window of an extrude object contains a **Twisted Extrude Shape** page, in which you can set parameters defining how the object is created.

**Twisted Extrude shape**

Height:	20.4 in
Direction:	One Side
Draft Angle:	0
Draft Start Distance:	0 in
Draft End Distance:	0 in
Offset:	0 in

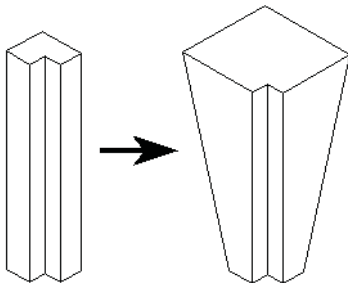
**Twist Options**

Twist Angle:	90
Distance Type:	Normal
Twist Start Distance:	0 in
Twist End Distance:	10 in
Continuity:	G0

- **Height:** The distance of the extrusion.
- **Direction:** Switch between one-sided and two-sided.

For the following parameters to be accessible, the extrude must be created as a solid. *Available in TurboCAD Pro and Platinum only.*

- **Draft Angle:** Creates an extrusion of increasing or decreasing cross-section. Enter the angle of deviation from the extrusion path.



- **Draft Start / End Distance:** If **Draft Angle** = 0, you can specify a draft angle by entering the offset distances.

- **Offset:** Creates a hole in the extrusion at a distance from the outside of the extrusion equal to the Offset value.

#### Twist Options:

- **Twist Angle:** Specifies the total rotation of the twist.
- **Distance Type:** Specifies where the twist will be applied to the extrusion. Normal applies the extrude between the Start and End distance. Full height applies the twist to the entire extrusion. Twist to Top applies the twist from the Start distance to the top of the extrusion.
- **Twist Start Distance:** Where the twist starts along the extrusion.
- **Twist End Distance:** Where the twist ends along the extrusion.
- **Continuity:** Specifies the smoothness of the transition between the straight sections and the twisted sections. G0 is the most abrupt transition, G1 is a more moderate transition, and G2 is the smoothest transition.

For **TC Surface Options**, see "TC Surface Properties" on page 372.

### Quick Pull

*Available in TurboCAD Pro and Platinum only*



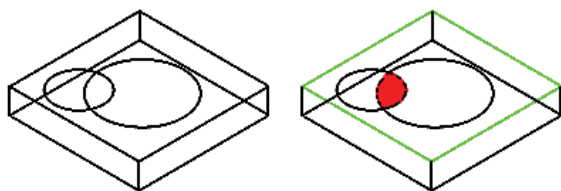
Pulls or pushes a 3D extension of a face by extruding 2D a closed area on the face of a 3D object. The extrusion extends along a path normal to the face of the 3D object.

The closed area must be coplanar with the Flat face of a 3D object. It may consist of:

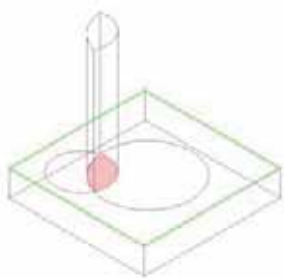
- Any closed 2D entity. E.g. a rectangle
- Any area that can be hatched by picking a point (with zero gap tolerance)
- Areas enclosed by crossing coplanar, linear geometry, including edges and geometry in blocks E.g. intersecting lines that create a closure
- 3D faces. E.g. The face of a cube

- Any combination of the above. Areas created by geometry (including edges on faces) drawn coplanar to any face of a 3D solid E.g. A circle is coplanar with the face of a cube. It overlaps the edge of the cube. The edge divides the circle into two parts, one is a valid area, or intersecting lines that create a closure on the face of a 3D object, with or without defining geometry of the 3D entity.

1. Select the 2D closed area to extrude.



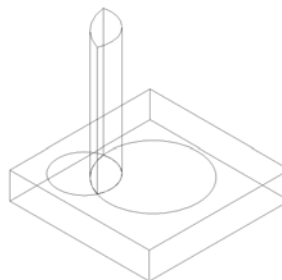
2. Move the mouse to extrude the face, or enter a value in the Height field of the Inspector Bar.



3. If you are specifying the height by using the mouse, click to finish the extrusion.

Or

If you are specifying the height by using the height field in the Inspector bar, press Enter to finish the extrusion.

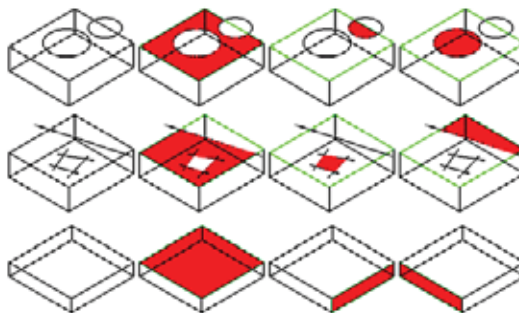



---

*Note: If the Two Sided property is set, the solid extrusion will be created on either side of the profile.*

---

Examples of areas that can be extruded:



### Quick Pull Properties

The Properties window of an Quick Pull object contains a Simple Extrude Shape page, in which you can set parameters defining how the object is created.

**Height:** The distance of the extrusion.

**Direction:** Switch between one-sided and two-sided.

**Solid Options:** For these parameters to be accessible, the extrusion must be created as a solid. Available in TurboCAD Pro and Platinum only.

- Draft Angle:** Creates an extrusion of increasing or decreasing cross-section. Enter the angle of deviation from the extrusion path.

- **Draft Start / End Distance:** If Draft Angle = 0, you can specify a draft angle by entering the draft distances.
- **Offset:** Creates a hole in the extrusion at a distance from the outside of the extrusion equal to the Offset value.

---

*Note: The Quick Pull and the Simple Extrude tool use the same Property Value Preset. Therefore, changing the property settings in one tool will change the settings in the other.*

---

## Sweep

Available in TurboCAD Pro, and Platinum Only



Creates a 3D object by driving a 2D profile along a path.

1. Start with one or more 2D profiles. The profiles can be open or closed. Add a 2D or 3D path. Typically the path intersects the profile and is approximately perpendicular to it, but these conditions are not required.



2. Activate **Sweep**. If the profiles consist of compound curves, make sure **Use Compound Profile** is active.



3. Select the first 2D profile. If the profile is open, select **Finish Selection of Path** to end the selection. You can then select more profiles if necessary, by pressing the Shift key.

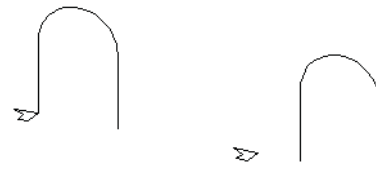


4. Then select the 2D or 3D path over which you want to sweep the 2D profiles. If the sweep path consists of more than one curve, make sure **Use Compound Path** is active. When finished, click **Finish Selection of Path**.

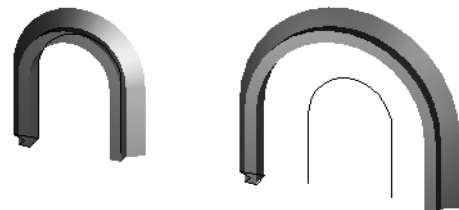


As stated before, the 2D profile and the sweep path do not have to intersect. However, the results will vary depending on how far apart the profiles are.

Consider this comparison, in which the profiles on the left intersect and the profiles on the right do not.



Here are the **Sweep** results: the solid on the right is swept over the offset of the sweep path, and is therefore larger.



### Local menu options

**Rigid Sweep:** Use this option if you want the cross-sections of the solid to remain parallel to the original swept profile.

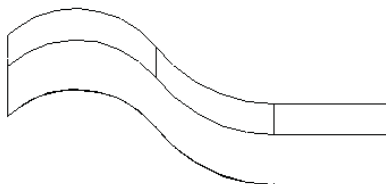
This solid does not use **Rigid Sweep**; the cross-sections of the solid are always normal to the sweep path.



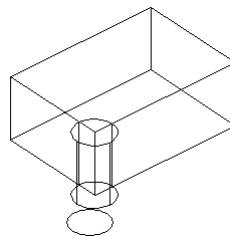
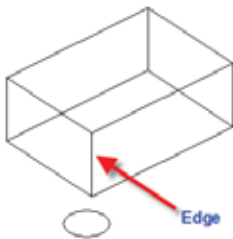
This is the same solid using **Rigid Sweep**; the sections are always parallel to one another, and to the original profile.



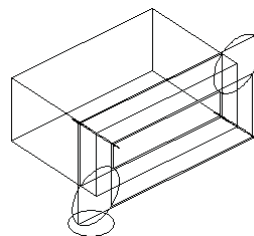
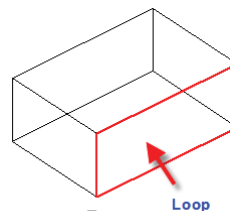
If you view the **Rigid Sweep** from this angle, you can see how the sections are parallel.



**Use Edge as Path:** When this option is on you can use the edge of a 3D object as a path. You cannot use this option with a compound profile. (Available in TurboCAD Platinum Only)



**Use Loop as Path:** When this option is on you can use the Loop of a 3D object as a path. A loop is all of the edges which enclose a single face of a 3D object. You have to select the 3D object first, then its face. You cannot use this option with a compound profile. (Available in TurboCAD Platinum Only)




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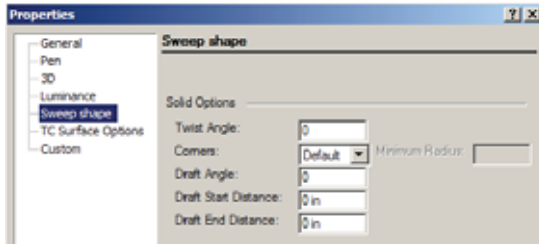
**NOTE:** If you are trying to sweep by using a block as the path, you must have the Compound Path option turned on.

---



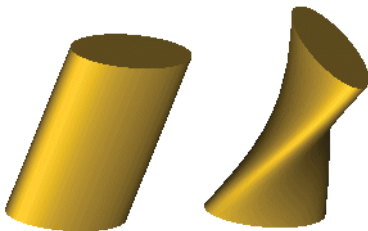
### Sweep Shape Properties

The **Properties** window of an sweep object contains a **Sweep Shape** page, in which you can set geometric parameters.



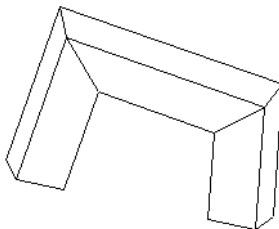
For these parameters to be accessible, the sweep must be created as a solid. *Available in TurboCAD Pro and Platinum only.*

**Twist Angle:** The angle by which the sweep is twisted along the path.

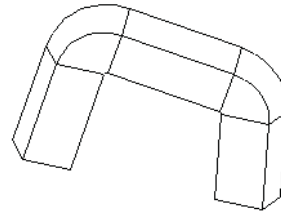


**Corners:** Defines how the joints of multi-segmented extrusions will be formed.

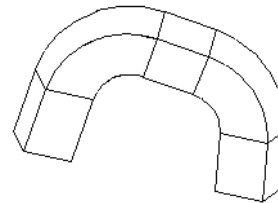
- **Default:** Sharp corners.



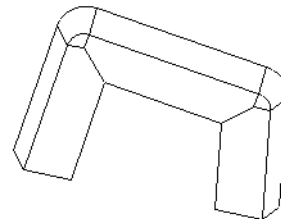
- **Bend:** Rounds the corners. If you don't specify a **Minimum Radius**, the rounding will be minimal.



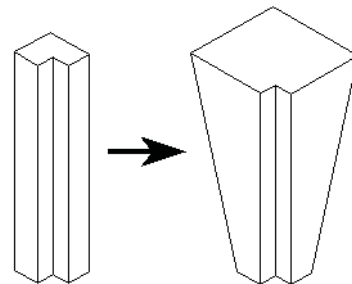
If you want a larger rounding radius, specify a **Minimum Radius**.



- **Crimp:** Keeps internal corners sharp, performs a minimal rounding on external corners.



**Draft Angle:** Creates an extrusion of increasing or decreasing cross-section. Enter the angle of deviation from the extrusion path.



**Draft Start / End Distance:** If **Draft Angle** = 0, you can specify a draft angle by entering the offset distances.

Sweep does not allow you to create a **TC Surface**.

## Rail Sweep

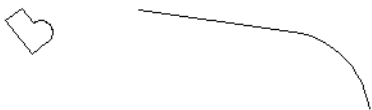
*Available in TurboCAD Pro, Platinum and Deluxe Only*



Creates a 3D object by driving a 2D profile along a path. The profile and path do not have to touch or be in different workplanes; the profile section will be “brought” to the start of the path, made normal to the path, and swept along it.

If you are going to use Compound profiles for both the path and the profile you will need a base point. The base point specifies the point through which the profile will follow the path. To create a base point, use the Point tool and place a point at the location adjacent to the profile that you wish to be used. For example, at the end of a line, or at the center of an arc or circle.

1. Start with one 2D profile. The profile can be open or closed. Add a 2D or 3D path. The path can lie anywhere, and in any workplane. In this example, profile and path lie in the same workplane.



2. Activate **Rail Sweep**.
3. If the profiles consist of compound curves, make sure **Use Compound Profile** is active.



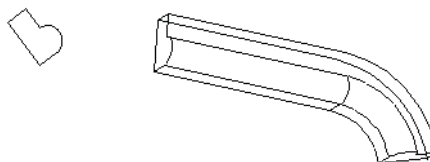
4. Select the first 2D profile. If the profile is open, select **Finish Selection of Path** to end the selection. You can then select more profiles if necessary, by pressing the Shift key.



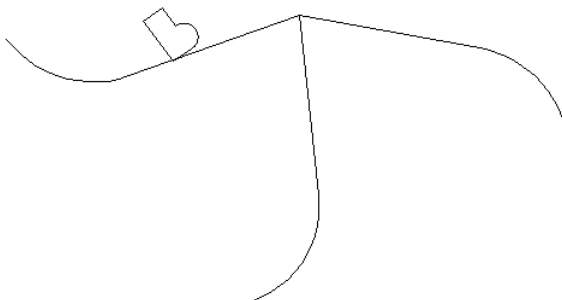
5. If you are using a Compound profile for the profile you must specify the base point to be used. at this step. Select the point you created for this purpose.
6. Then select the sweep path. If the sweep path consists of more than one curve, make sure **Use Compound Profile** is active.



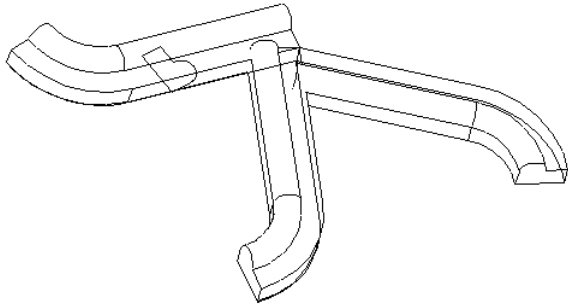
7. The profile is brought to the path (at the base point if one is being used) and swept along it, normal to the path. The profile intersects the path at its reference point (see "Changing the Reference Point" on page 199).



This tool is handy if you have a single profile that you want to sweep over multiple paths, or rails. In this example, there are three paths for the same profile:



Here are the results:

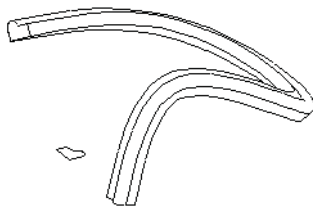


If you used a Compound profile for the profile, you will be able to modify the resulting 3D object by modifying the original profile object, or by moving the base point object. If you used a Compound profile for the path, you will be able to modify the resulting 3D object by modifying the original path object/s.

The sweep path does not have to be 2D. The path in this example was created using **3D Spline by Fit Points**.



This is the result:



#### Local menu option

**Rigid Sweep:** Keeps cross-sections of the solid parallel to one another along the entire path. See explanation under "Sweep" on page 395.

#### **Compound Profiles as Paths**

To use a compound profile as a path

Also see "Sweep Shape Properties" on page 397.

## **Revolve**



Creates a 3D object by revolving a 2D object about a revolution axis.

By default, the profile will be revolved 360 degrees, but you can change this angle or create a spiral. See "Revolution Shape Properties" on page 401.

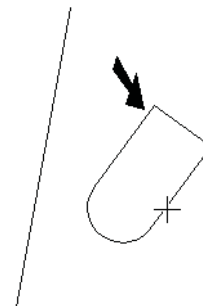
1. If you want to select only a simple (single-object) curve, make sure **Use Compound Profile** is **not** selected. If you want to use a compound profile, see "Revolve with Compound Profiles" on page 400.



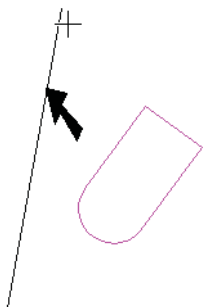
2. If you want to select a line as the axis of revolution, make sure **Select Revolve Axis** is active.



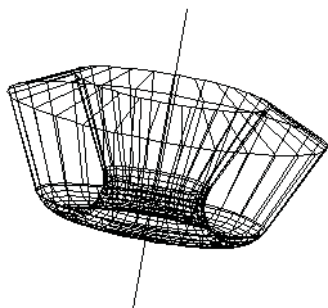
3. Select a 2D object to revolve.



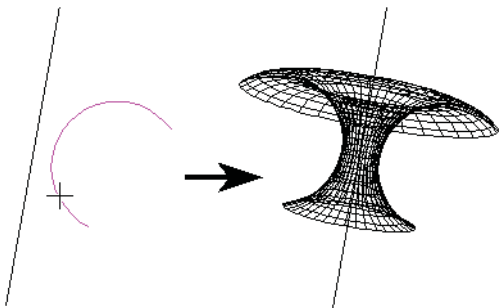
4. Select two points of a revolution axis, or, if **Select Revolve Axis** is active, select the axis line..



The revolved shape is created.



You do not have to select a closed 2D object to revolve. If you use an open object, a 3D surface will result.



### Revolve with Compound Profiles

A compound profile is an open or closed chain of connected curves. You can use compound profiles when you do not want to create a polyline, or convert a chain into a polyline.

1. Make sure **Use Compound Profile** is selected.



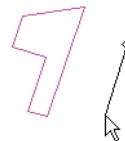
2. Select the compound profile, which is automatically identified as a chain. To deselect any curve in the chain, select it again (it will turn green). In this example, the profile is a series of connected lines, not a polyline.



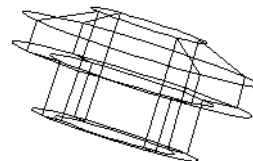
3. You can select more profiles if necessary. When the profiles are selected, click **Finish Selection of Profile**, or select it from the local menu.



4. Select two points of a revolution axis, or, if **Select Revolve Axis** is active, select the axis line.



The revolved shape is created.



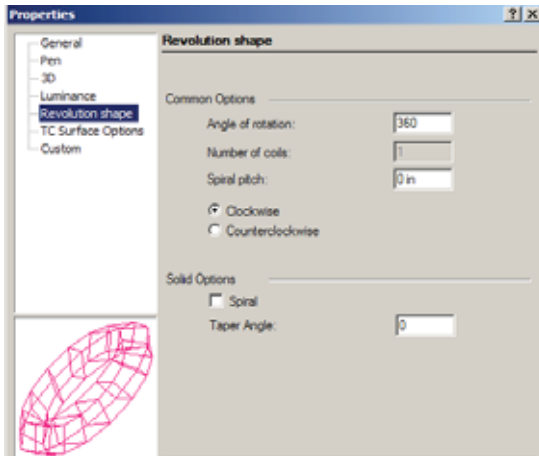

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*NOTE: Compound profiles can be modified, which updates the 3D objects upon which they are based. See "Updating Compound Profiles" on page 431.*

---

## Revolution Shape Properties

The **Properties** window of a revolve contains a **Revolution Shape** page, in which you can set geometric parameters.



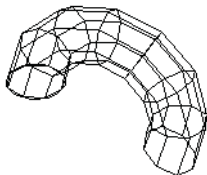
For the Solid parameters to be accessible, the revolve must be created as a solid. In the **3D** page of the **Properties** window, make sure **Solid** is selected under **Create Object As**.

**Spiral:** The revolution will proceed outward in a spiral pattern.

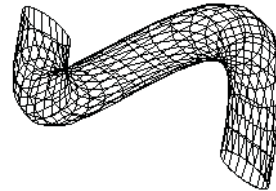
**Taper Angle:** The revolution will have a draft angle, with the section increasing or decreasing as it progresses.

The remaining parameters (**Common Options**) appear for both surface and solid objects.

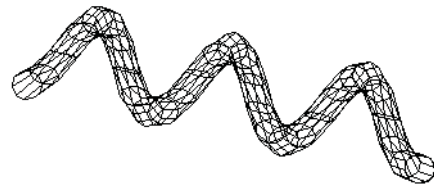
**Angle of rotation:** Enter a value less than 360 for a partial revolve.



**Spiral pitch:** The lateral distance between the start and end of the revolve, enabling you to create a spiral. The angle of rotation must be 360.



**Number of coils:** If **Spiral pitch** is nonzero, you can specify the total number of revolutions.



**Clockwise / Counterclockwise:** Sets the direction of revolution. The direction depends on how the revolution axis was selected.

For **TC Surface Options**, see "TC Surface Properties" on page 372.

## Lofting

Available in TurboCAD Pro and Platinum only



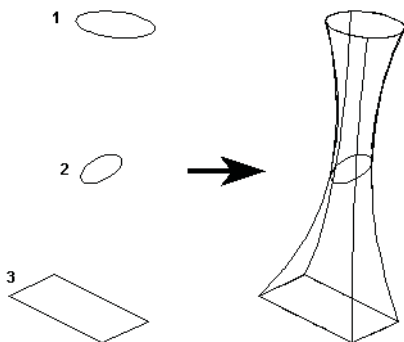
Creates a 3D object by connecting 2D profiles. The profiles lie on different planes, and the planes do not have to be parallel. The profiles are connected using NURBS (Non-Uniform Rational b-Spline) calculations.

The profiles can be closed or open, but they must be consistent - all open or all closed.

1. If you want to select only simple (single-object) curves, make sure **Use Compound Profile** is **not** selected. If you want to use compound profiles, see "Lofting with Compound Profiles" on page 402.

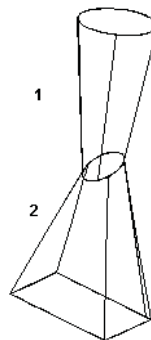


2. Select the profiles, in the desired order. Selection order is important
3. Select **Finish** from the local menu, or double-click on the last profile, to create the loft.

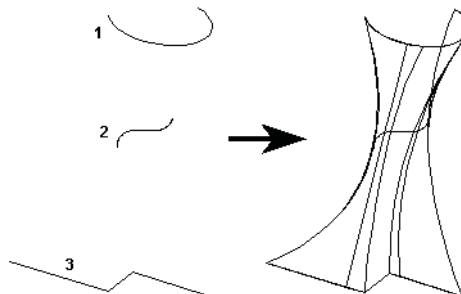


The loft has no sharp corners at the profiles.

To create corners, you can create two separate lofts.



If you use open profiles, a 3D surface will result.



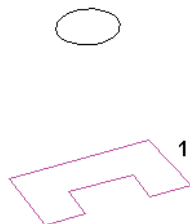
### Lofting with Compound Profiles

A compound profile is an open or closed chain of connected curves. You can use compound profiles when you do not want to create a polyline, or convert a chain into a polyline.

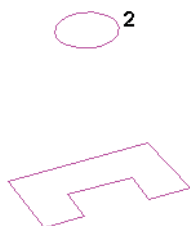
1. Make sure **Use Compound Profile** is selected.



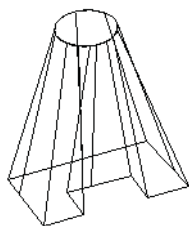
2. Select the compound profile, which is automatically identified as a chain. To deselect any curve in the chain, select it again (it will turn green). In this example, Profile 1 is a series of connected lines, not a polyline.



3. When the profile is selected, click **Finish Selection of Profile**, or select it from the local menu.
4. Select the next profiles using the same steps. If subsequent profiles are simple (not compound), you can turn off **Use Compound Profile**.



5. When the last profile is selected, click **Finish** or select it from the local menu.



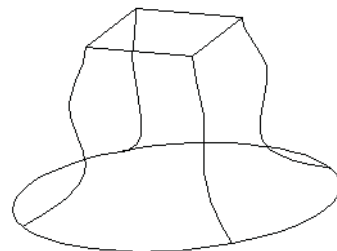

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NOTE: Compound profiles can be modified, which updates the 3D objects upon which they are based. See "Updating Compound Profiles" on page 431.

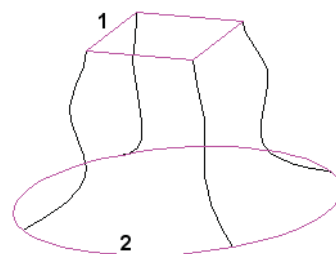
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### Lofting with Guide Lines

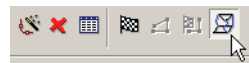
You can specify guides lines when creating the loft object. In this example, the lower circle and upper square are the loft profiles. The four curves between the profiles are guide lines.



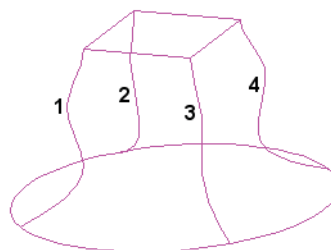
1. Activate **Loft**, and select the loft profiles in the desired order.



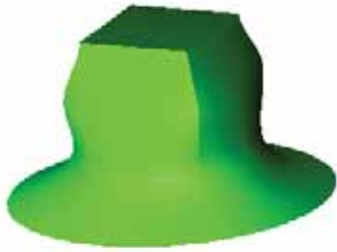
2. Click **Select Guide Lines**, or select it from the local menu.



3. Select each of the guide lines.



- 4. Click **Finish** or select it from the local menu. The 3D object transitions between the loft profiles, following each of the guide lines.

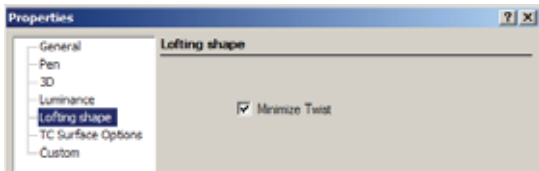


**Guide Lines**

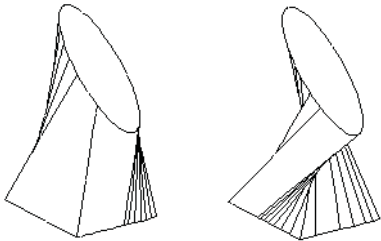
Guide lines can be arcs, 2D or 3D Splines or lines. Beziers, and polyline cannot be used as guide lines. Guide line must touch every profile used in the loft. One end of each guide line must terminate on the first profile used in the loft. The other end of that guide line must terminate on the last profile used in the loft. If guide lines do not meet these requirements they will be ignored.

**Lofting Shape Properties**

The **Properties** window of a loft contains a **Lofting Shape** page, in which you can minimize twist.



**Minimize twist:** The lengths of segments between profiles will be minimized. (Available only if the object is created as a solid. In the **3D** page of the **Properties** window, make sure **Solid** is selected under **Create Object As.**)



**Effect of Minimize Twist**

For **TC Surface Options**, see "TC Surface Properties" on page 372.

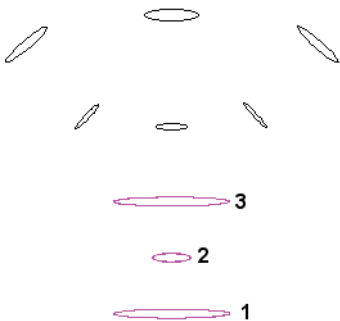
**Branched Lofting**

*Available in TurboCAD Pro and Platinum only*



Creates a lofting object by defining profiles along a trunk, and profiles along two or more branches. The profiles lie on different planes, and the planes do not have to be parallel. The result is one 3D object.

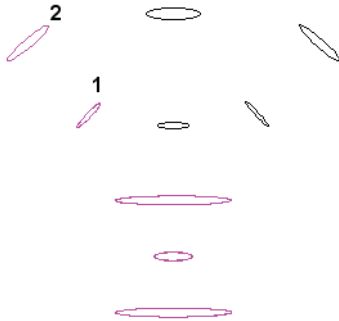
- 1. Select the profiles that define the trunk. The branching will start at the last profile of the trunk.



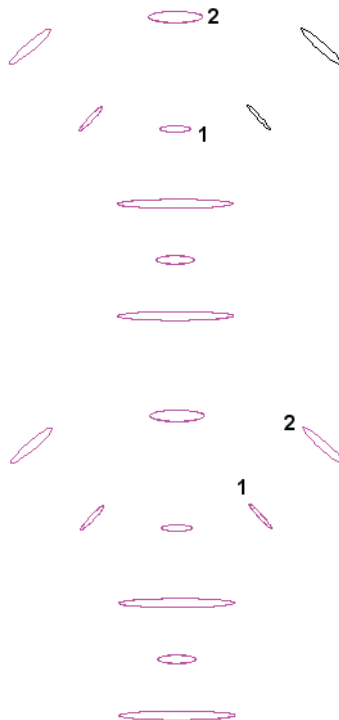
- 2. Select **Finish selection of trunk** from the local menu or Inspector Bar.



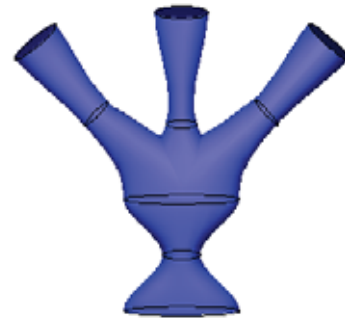
3. Select the profiles that define the first branch, starting at the profile closest to the end of the trunk.



4. Select **Finish** selection of branch.
5. Select profiles for additional branches the same way.



6. When all branches are defined, select **Finish**. The lofting object is created.



### Face2Face Lofting

*Available in TurboCAD Pro and Platinum only*

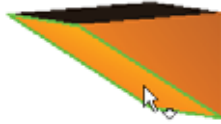


Creates a 3D lofting object between two faces of existing 3D objects. The result is one 3D object.

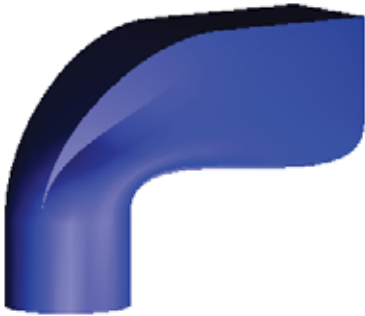
1. Select the first face, in this example, the top of the cylinder. Selection order is important.



2. Select the second face, in this case, the sloped face of the wedge.



3. The result is a loft object that proceeds smoothly between the two faces. The two original objects and the new loft object are now merged into one object.



The resulting object takes the properties (such as Pen) of the object where you selected the first face.

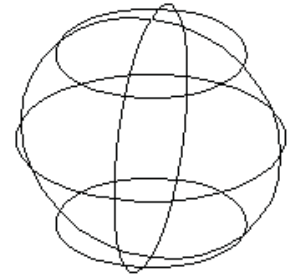
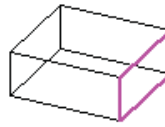
## Extrude to Face

*Available in TurboCAD Pro and Platinum only*

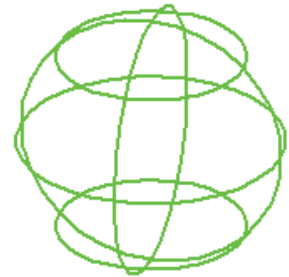
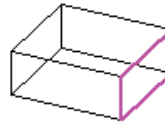


Extends a 3D object by extruding selected faces to a selected 3D object and performing a boolean.

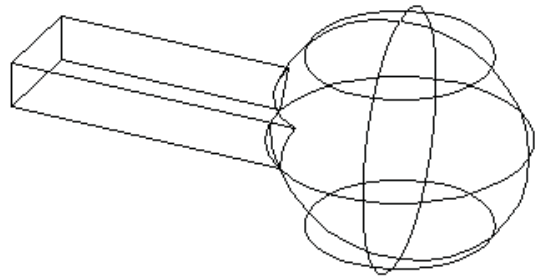
1. Select the face of a 3D object.



2. Select the 3D object to be the target of the extrusion.



3. The result is a new 3D object.

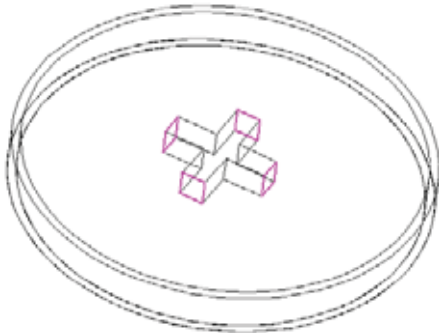


If the boolean option is set to Subtract, the extrusion will be subtracted from the target body, and the original 3D object from which the extrusion was made will be removed.

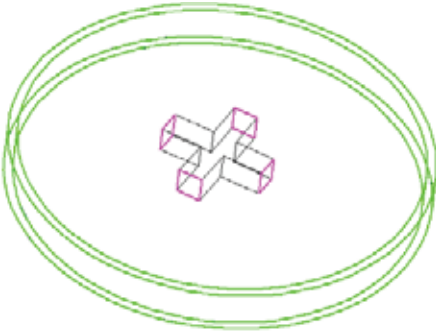


You can extrude multiple faces at one time, but you can extrude to only a single 3D object.

1. Hold down the SHIFT key to select multiple faces.



2. Release the SHIFT key and select the target object.

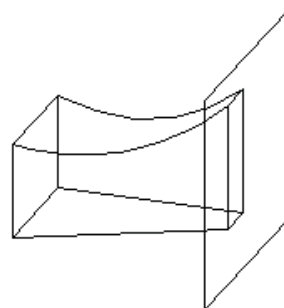
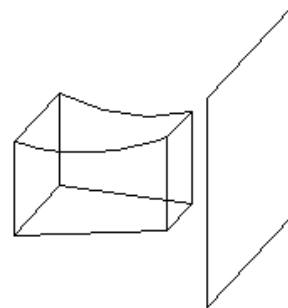


3. You will have a multiple extension.

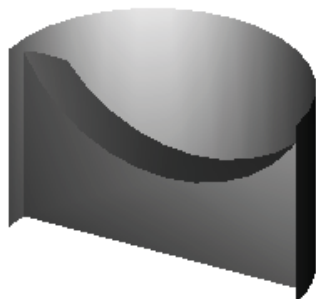
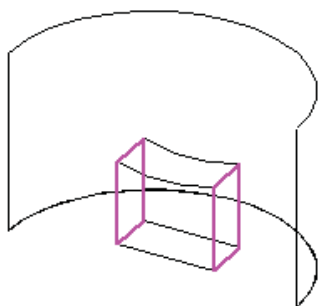


There are some limitations that must be respected to create valid results:

- Resulting extrusions must intersect the target body when extended, or the operation will be invalid.
- You cannot extrude adjacent faces. Adjacent faces are extended along with the extrusion face. Adjacent curved faces and adjacent slanted faces will continue their curve or slant when the selected face it extruded



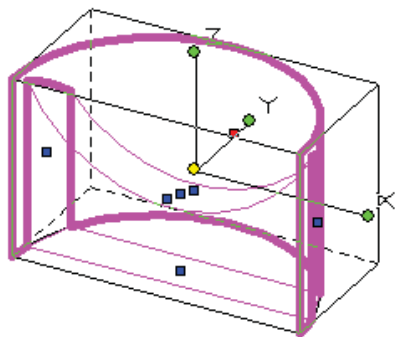
- You can extend to ACIS surfaces, but only if the target body completely intersects the resulting extrusion.



- You cannot extrude a face to the same 3D object that the extrusion face belongs to.

When extruding to a ACIS surface you can remove the remnant “excess” of the target surface using the Facet Edit tool.

1. Select the facet.



2. Press the DELETE key.



## Profile Along Path

*Available in TurboCAD Pro, and Platinum Only*



Places a 2D profile along a 2D or 3D path.

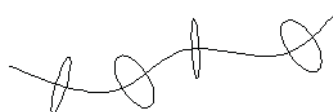
1. Select the profile you want to project, then select a point on the path where you want the projection.



2. Click more points where you want profile projections.



If you rotate the model, you can see the profiles in 3D.



3. Select **Finish** from the local menu or Inspector Bar to exit the function.

Local menu options:

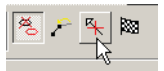
**Make Copy Profile:** Use this profile to make multiple copies along the path. When unselected, you can **Finish** or select another profile.



**One Step Back:** Removes profiles in the reverse order in which they were created on the path.



**Unselect Profile:** Unselects the current profile, enabling you to select another one.



## Graphic Along Path

*Available in TurboCAD Pro, and Platinum Only*



Places a 2D profile or 3D object along a 2D or 3D path.

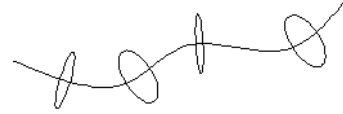
1. Select the profile or object you want to project, then select a point on the path where you want the projection.



2. Click more points where you want profile projections.



If you rotate the model, you can see the profiles in 3D.



3. Select **Finish** from the local menu or Inspector Bar to exit the function.

Local menu options:

**Sets:** Specifies a number of copies of the object that will be placed from the insertion point onward. The space between the insertions will be equal to the value in the Distance field.

**Distance:** Set the distance between the multiple insertions specified by the Sets field.

**Delete Original:** Deleted the original object.

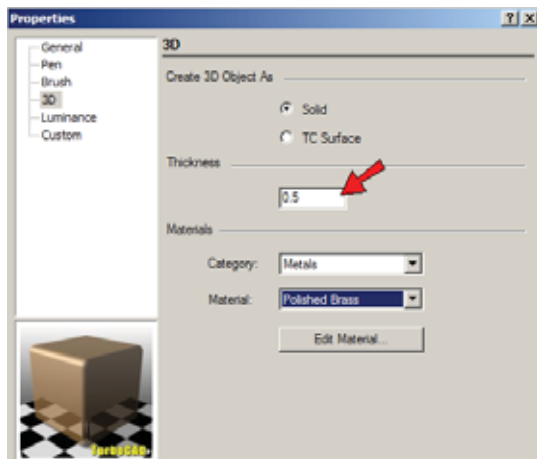
**Flip Original:** Flips the inserted objects, creating a mirror of the original.

**Set Graphic on Path Start Point:** Sets the insertion at the start point of the path.

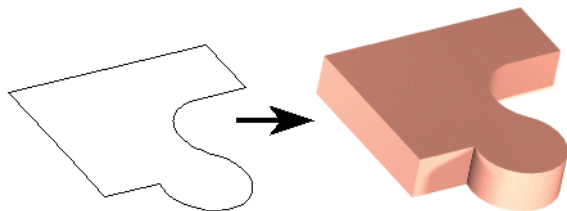
**Set Graphic on Path End Point:** Sets the insertion at the end point of the path. Sets will be ignored if this setting is on.

## Creating 3D Objects by Editing 2D Objects

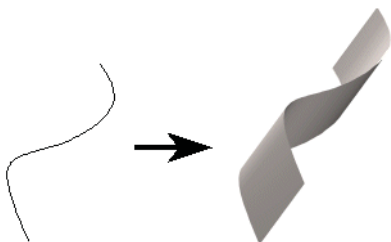
An easy way to create a 3D object is to give depth to a 2D object. This is done by opening the **Properties** window of a 2D object (see "Object Properties" on page 79) to the **3D** page, and entering a thickness.



For closed 2D objects, a solid will result. You can use the **Properties** window to create a surface object instead. See "3D Properties" on page 371.



For an open 2D object, a surface will result.




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**NOTE:** *The original profile can be edited (**Select Edit** or **Edit Tool**) just like any 2D object. Whatever changes you make, the thickness will remain unchanged (unless you change it).*

---

## Intersection and Projection

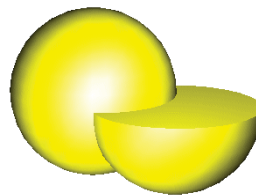
These two tools create curves, one by finding the intersection of two solid objects, and the other by projecting a 2D object onto a 3D solid.

### Intersection

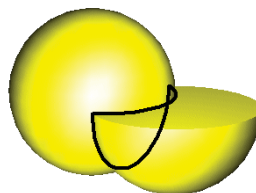


Creates curves along the intersection of two solids (not surfaces). This tool does not work with surface objects.

Select the first and second solid objects. This example shows a sphere and a hemisphere.



The intersection curves are created.

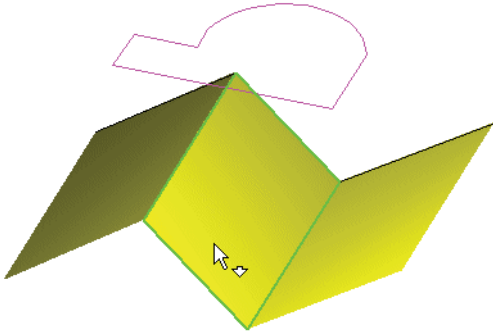


### Projection



Projects a 2D curve onto a solid (not surface) face. The 2D curve can be open or closed.

This example shows a 2D polyline and a **Simple Extrude** generated from a line-segmented polyline. First select the 2D curve. Then select the face. Even though the **Simple Extrude** is one object, its faces are still considered separate for this tool.

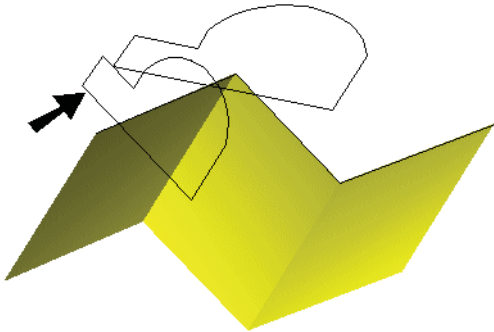



---

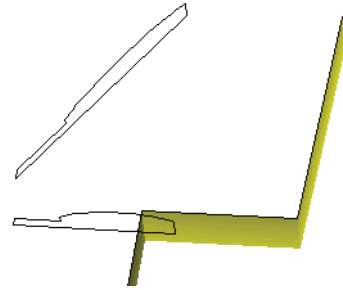
*NOTE: Even when you are projecting onto a surface such as this one, the object type must be a solid, as set in the **3D** page of the object's **Properties**.*

---

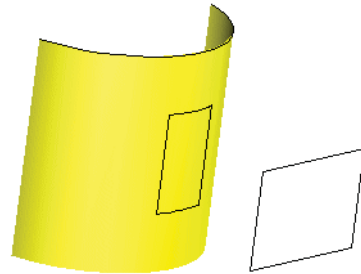
The projection curves are created. For a planar face, the curves can be placed on the face's theoretical extension.



In this view, the projection can be seen more clearly.



If you are projecting onto a curved face, the projection will be scaled toward the center of the curved face.



## Patterns

*Available in TurboCAD Platinum only*

Patterns are arrays of Solid ACIS 3D objects copied in specific arrangements, and controlled parametrically. Patterns are solid object and they and/or their elements can be manipulated by 3D tools, like Bend, and patterns can be Booleaned with other 3D objects. There are five pattern tools: Array, On curve, Radial, Spherical, Cylindrical.

### Array Pattern

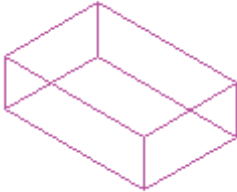
*Available in TurboCAD Platinum only*



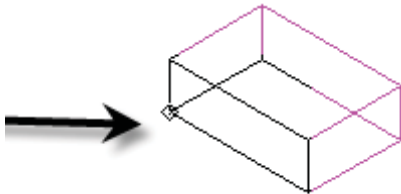
Creates a rectilinear array composed of columns, rows and levels.

1. Select the Array Pattern tool.

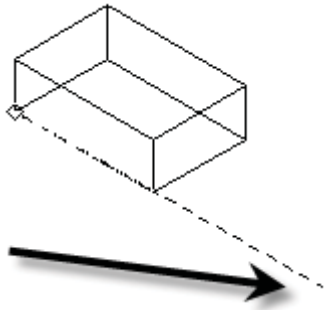
2. In the Inspector bar set the Number of columns, rows and levels.
3. Select a 3D solid.



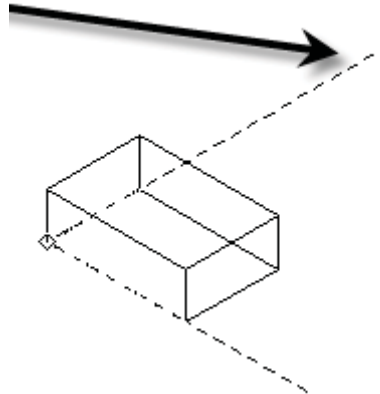
4. Click to specify the origin of the array.



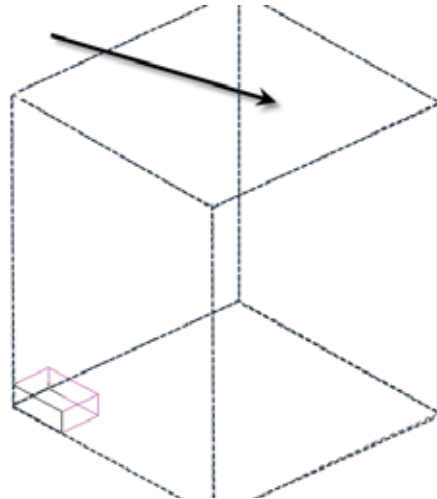
5. Move the cursor and click to set the X axis of the array.



6. Move the cursor and click to set the Y axis of the array.



7. Move the cursor and click to set the Z axis of the array.





The array is created.



The axis values set the distance from center to center of the source object.

---

**WARNING:** You must have at least two rows to have more than one levels in this array. Reducing rows to one will eliminate the level option until more rows are added.

---

#### Local menu options:

**Create Association:** Creates an association between the pattern and the destination object. If subsequently the destination object is edited the pattern will update to reflect the changes made. The Leave Destination and By Entity options must be active for associations to be available.

**Leave Source:** If this option is on the original 3D object (source) will remain. If it is off the object will be deleted.

**Leave Destination:** If this option is on the 3D object used as for By Entity will remain. If it is off the object will be deleted.

**By Entity:** If this option is on you can select an existing box to set the axis size of the array. With this option on you select the box instead of specifying the origin and axis values. The axis values will be the axis values of the box.

**Fit Pattern:** If this option is on the axis lengths will define the total length between the centers of the outermost elements in the array along each axis. In other words “fit” within the lengths. If the option is off the axis lengths specify the distance between centers of adjacent objects.

**Rectangular Base:** Forces the the Y-axis to be 90 degrees to the X-axis

**Hexagonal:** Creates a hexagonal grid array. If this option is on you will only be allowed to enter an X-axis, unless you use the By Entity setting.

---

*NOTE: Array Properties are available only through the Selection Info Palette..*

---

#### Array Properties:

The properties available will vary depending upon how you created your pattern.

**Fit:** If on the array will be fit within the specified axes lengths. Otherwise the axis lengths specify the distance between the centers of objects.

**Cols:** Sets the number of columns

**Rows:** Sets the number of rows

**Levels:** Sets the number of levels.

**X-axis Length:** Sets the X axis length.

**Y-Axis Length:** Sets the Y axis length.

**Height:** Sets the Z axis length.

### **Pattern by Curve**

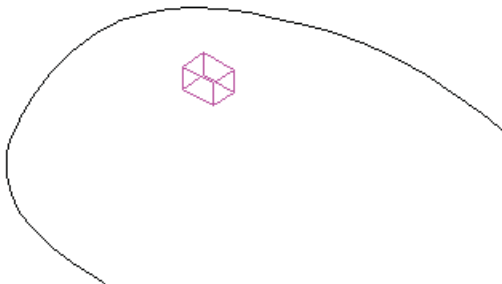
*Available in TurboCAD Platinum only*



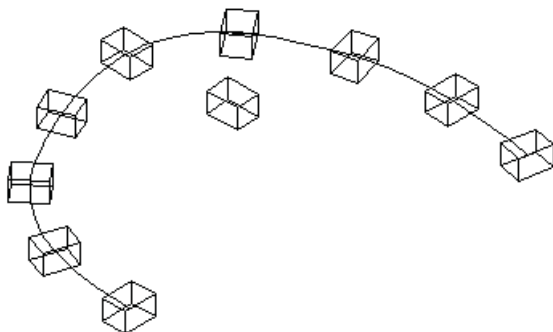
Creates an array of objects along an existing curve. Bezier's can be used as the curve, so can 2D or 3D splines. Polylines, circles, arcs and lines cannot be used.

1. Select the Pattern by Curve tool.
2. In the Inspector bar specify the number of Sets, then press Enter.

3. Select a 3D solid.



4. Click on a curve to finish the pattern.



#### Local menu options:

**Create Association:** Creates an association between the pattern and the destination object. If subsequently the destination object is edited the pattern will update to reflect the changes made. The Leave Destination option must be active for association to be available.

**Leave Source:** If this option is on the original 3D object (source) will remain. If it is off the object will be deleted.

**Leave Destination:** If this option is on the curve will remain. If it is off the curve will be deleted.

**Non Rotate:** If this option is selected the tool creates the designated pattern but the arrayed objects retain the orientation of the source object and are not rotated at each position.

#### Array Properties:

The properties available will vary depending upon how you created your pattern.

**Fit:** If on the array will be fit within the specified axes lengths. Otherwise the axis lengths specify the distance between the centers of objects.

**Count of Elements:** Sets the number of items in the pattern.

**Non Rotate:** If this option is selected the tool creates the designated pattern but the arrayed objects retain the orientation of the source object and are not rotated at each position.

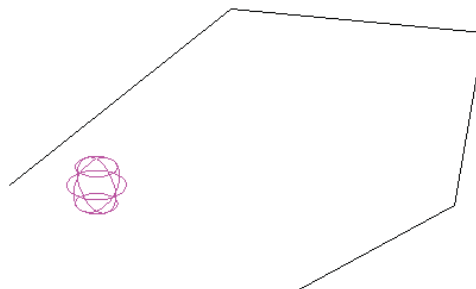
### **Pattern by Polyline**

*Available in TurboCAD Platinum only*

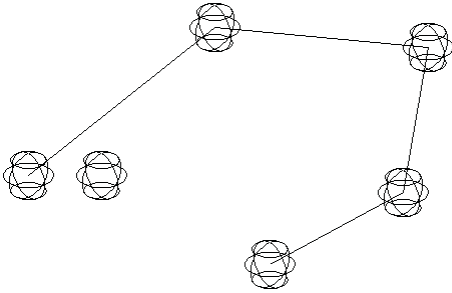


Creates an array of objects along a 2D or 3D polyline.

1. Select the Pattern by Polyline tool.
2. In the Inspector bar specify the number of Sets, then press Enter.
3. Select a 3D solid.



- Click on a polyline to finish the pattern.



#### Local menu options:

**Create Association:** Creates an association between the pattern and the destination object. If subsequently the destination object is edited the pattern will update to reflect the changes made. The Leave Destination option must be active for association to be available.

**Leave Source:** If this option is on the original 3D object (source) will remain. If it is off the object will be deleted.

**Leave Destination:** If this option is on the curve will remain. If it is off the curve will be deleted.

**Non Rotate:** If this option is selected the tool creates the designated pattern but the arrayed objects retain the orientation of the source object and are not rotated at each position.

#### Array Properties:

The properties available will vary depending upon how you created your pattern.

**Fit:** If on the array will be fit within the specified axes lengths. Otherwise the axis lengths specify the distance between the centers of objects.

**Count of Elements:** Sets the number of items in the pattern.

**Non Rotate:** If this option is selected the tool creates the designated pattern but the arrayed objects retain the orientation of the source object and are not rotated at each position.

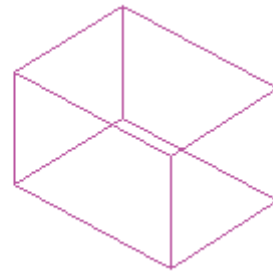
## **Radial Pattern**

*Available in TurboCAD Platinum only*

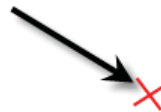
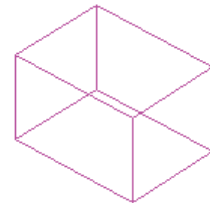


Creates a variety of radial array patterns.

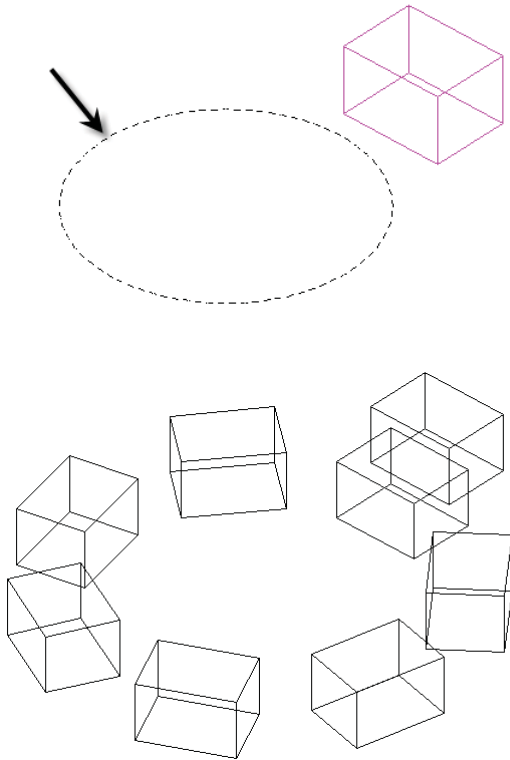
- Select the Radial Pattern tool.
- Select a 3D solid.



- Click to set the center of radial pattern.



4. Click to set the radius of the pattern.



#### Local menu options:

**Create Association:** Creates an association between the pattern and the destination object. If subsequently the destination object is edited the pattern will update to reflect the changes made. The Leave Destination and By Entity options must be active for associations to be available.

**Leave Source:** If this option is on the original 3D object (source) will remain. If it is off the object will be deleted.

**Leave Destination:** If this option is on the 3D object used as for By Entity will remain. If it is off the object will be deleted.

**By Entity:** If this option is on you can select an existing circle or an arc or a circular curve on a 3D object to set the size of the array. With this option on you select the circle instead of specifying the center and radius values.

**Elliptical Pattern:** If this option is selected the tool creates a circular pattern. The number of rings is ignored. This is the default option.

**Polar Grid Pattern:** If this option is selected the tool creates a polar grid pattern, similar to the polar longitude lines on a globe. The angular number is ignored.

**Radial Pattern:** If this option is selected the tool creates a radial pattern, similar to the spokes of a wheel.

**Non-Rotate:** If this option is selected the tool creates the designated pattern but the arrayed objects retain the orientation of the source object and are not rotated at each position. This option is not available for if Polar Grid is selected.

**Fit Pattern:** If this option is on the radius will define the total distance between the center of the outermost elements in the array and the center of the array. In other words “fit” within the radius. If the option is off the radius specifies the distance between centers of adjacent rings. This option is not available for elliptical patterns.

**Hex Symmetry:** If this option is on the resulting Polar grid pattern will use a hexagonal arrangement for the spokes instead of a quadrant arrangement. This option is only available if Polar grid is selected..

**On Arc:** If this option is selected the tool will prompt you to specify the desired arc after you specify the radius. The arc is specified by selecting two points in sequence. This option will be ignored if By Entity is selected.

#### Array Properties:

The properties available will vary depending upon how you created your pattern.

**Fit:** If this option is on the array will be fit within the specified axes lengths. Otherwise the axis lengths specify the distance between the centers of objects.

**Count of Elements:** Sets the number of items in the pattern.

**Start Angle:** Sets the angle at which the pattern starts.

**End Angle:** Sets the angle at which the pattern ends.

**Non-Rotate:** If this option is selected the tool creates the designated pattern but the arrayed objects retain the orientation of the source object and are not rotated at each position. This option is not available if Polar Grid is selected.

**Fit Pattern:** If this option is on the radius will define the total distance between the center of the outermost elements in the array and the center of the array. In other words “fit” within the radius. If the option is off the radius specifies the distance between centers of adjacent rings. This option is not available for elliptical patterns.

**Hex Symmetry:** If this option is on the resulting Polar grid pattern will use a hexagonal arrangement for the spokes instead of a quadrant arrangement. This option is only available if Polar grid is selected..

**Radius:** Sets the radius for the pattern.

#### Array Properties:

The properties available will vary depending upon how you created your pattern.

**Count of Latitudes:** Sets the number of latitudes in the pattern.

**Radius:** Sets the radius for the pattern.

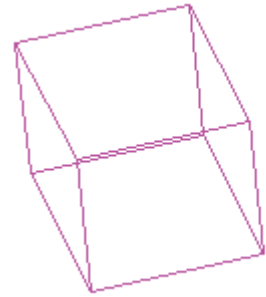
### **Spherical Pattern**

*Available in TurboCAD Platinum only*



1. Select the Spherical Pattern tool.
2. In the Inspector bar set the Number of latitudes.

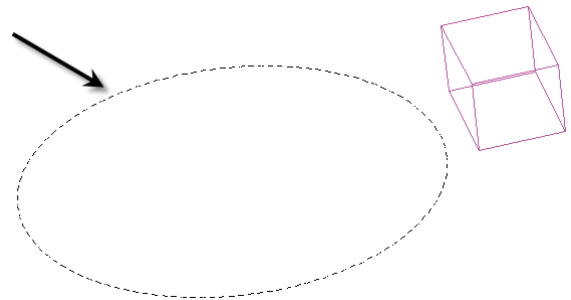
3. Select a 3D solid.

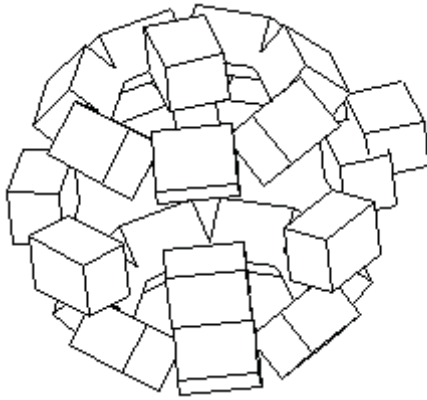


4. Click to set the center of radial pattern.



5. Click to set the radius of the pattern.





**Create Association:** Creates an association between the pattern and the destination object. If subsequently the destination object is edited the pattern will update to reflect the changes made. The Leave Destination and By Entity options must be active for associations to be available.

**Leave Source:** If this option is on the original 3D object (source) will remain. If it is off the object will be deleted.

**Leave Destination:** If this option is on the 3D object used as for By Entity will remain. If it is off the object will be deleted.

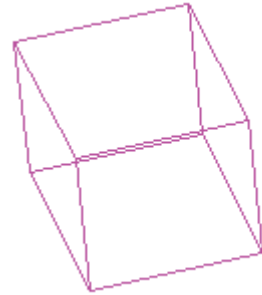
## Cylindrical Pattern

*Available in TurboCAD Platinum only*

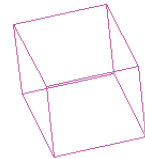


1. Select the Cylindrical Pattern tool.

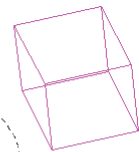
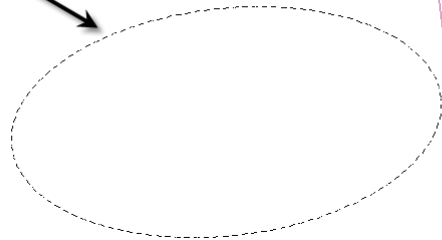
2. Select a 3D solid.



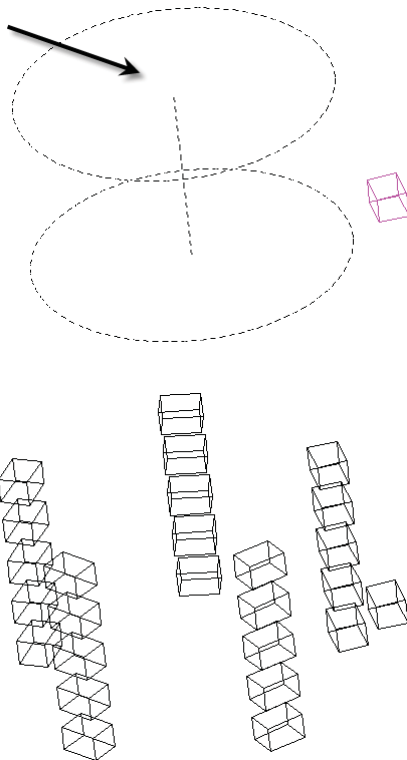
3. Click to set the center of radial pattern.



4. Click to set the radius of the pattern.



5. Click to set the height of the pattern.



#### Local menu options:

**Leave Source:** If this option is on the original 3D object (source) will remain. If it is off the object will be deleted.

**Create Association:** Creates an association between the pattern and the destination object. If subsequently the destination object is edited the pattern will update to reflect the changes made. The Leave Destination and By Entity options must be active for associations to be available.

**Leave Destination:** If this option is on the 3D object used as for By Entity will remain. If it is off the object will be deleted.

**By Entity:** If this option is on you can select an existing circle or an arc or a circular curve on a 3D object to set the size of the array. With this option on you select the circle instead of specifying the center and radius values.

**Fit Pattern:** If this option is on the height will define the total distance between the top and bottom of the array. In other words “fit” within the height. If the option is off the height specifies the distance between each of the axial sets.

**Hexagonal:** Creates a hexagonal alternating cylindrical array.

**Radius:** Sets the radius for the pattern.

## ***Snaps and Dimensions in 3D***

The following snaps work with 3D objects: **Nearest on Facet** and **Center of Extents**. Other snaps work in 3D, but are projected onto the current workplane.

Therefore, to apply a dimension to a 3D object, you must set the workplane to the plane where you want the dimension to appear. In other words, you can display *projected* measurements in 3D.

For ACIS solid objects (*TurboCAD Pro only*), you can apply **Radius** and **Diameter** dimensions to arc-based objects. You must turn on **Degenerative Faceting** in the **ACIS** page (**Options/ACIS**). These dimensions are non-associative.

## **Examining the 3D Model**

Once the model contains one or more 3D objects, there are several ways to change the way you see the model.

### ***The Camera***

In 3D space, you can view a model from any point. TurboCAD uses the concept of a “camera” - the camera represents your eye as you view the current scene. You can easily use the standard orthogonal and isometric views (see “Standard Views” on page 361), but camera tools enable you to view from any angle.

The current model view contains a default camera, placed at the center of the screen, facing into the model.

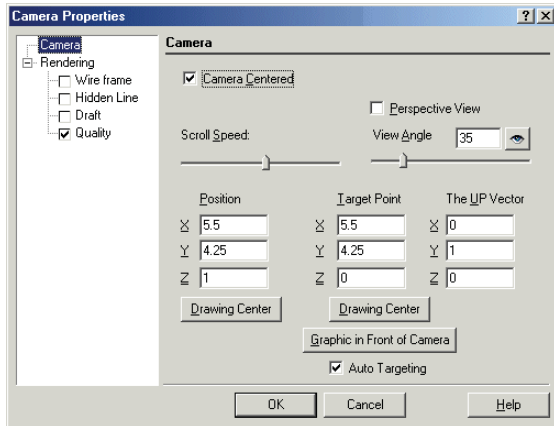
---

*NOTE: While “camera” means the perspective of the current view, “camera object” is something different. Camera objects are created to save and display multiple views, or views with specific parameters. See “Camera Objects” on page 422.*

---

## Camera Properties

Controls the position and target of the camera, as well as rendering controls. For the rendering properties, see "Render (Camera) Properties" on page 497.



**Camera Centered:** Enables the camera to turn around its own center when you use the **Camera Turn** tools (see "Camera Movements" on page 421). When not checked, the camera will turn around the **Target Point**.

**Scroll Speed:** The speed of the camera movement.

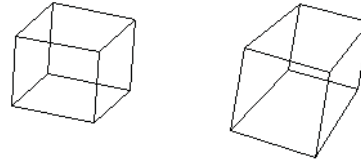
**Perspective View:** The visual effect of a perspective projection is similar to that of photographic systems and of the human visual system. The size of the perspective projection of an object varies inversely with the distance of that object from the center of projection.

---

NOTE: When using **Perspective View**, objects cannot be located or moved beyond the horizon. Select **Display Horizon** to display the horizon, in order to better orient yourself to the space.

---

- **View Angle:** The degree of perspective foreshortening. The view gets wider as the setting gets larger.



Standard view and perspective view

**Position:** The physical location of the camera.

- **Drawing Center:** Assigns the target point coordinates as the drawing center. The drawing center is always the geometric center of all the objects in the drawing.

---

NOTE: You can also access this view by selecting **View / Camera / Look to Drawing Center**.

---

**Target Point:** The point that the camera faces.

- **Drawing Center:** Moves the camera to the current drawing center.

**The UP Vector:** Indicates which way is up for the camera. A positive Z value rotates the camera up; a negative Z value rotates the camera down.

**Graphic in Front of Camera:** Select the object closest to the desired view center, and the target point is attached to that object.

---

NOTE: You can also access this view by selecting **View / Camera / Look to in Front Graphic**.

---

**Auto Targeting:** Sets the center of rotation on the object closest to the center of the screen (the default camera position). This is relevant when moving objects using the keyboard and mouse (such as Shift + right mouse button).



## Camera Movements

Incremental controls for adjusting the camera, and therefore your view of the model. As the camera moves through space, the visible objects will seem to move in the opposite direction of the camera motion.

These commands can be accessed via the **Camera** toolbar.

You can also display the **Camera Tools** toolbar by right-clicking in any toolbar area and selecting **Camera Tools**.



**Camera Turn Up:** Turns the camera up.

**Camera Turn Down:** Turns the camera down.

**Camera Turn Right:** Turns the camera to the right.

**Camera Turn Left:** Turns the camera to the left.

**Camera Rotate Right:** Rolls the camera clockwise.

**Camera Rotate Left:** Rolls the camera counter-clockwise.

**Camera Up:** Moves the camera up.

**Camera Down:** Moves the camera down.

**Camera Right:** Moves the camera to the right.

**Camera Left:** Moves the camera to the left.

## Walk Through Tools

Dynamic (moving) controls for moving the camera.

These commands can be accessed through the **Walk Through** toolbar.

You can display the **Walk Through** toolbar by right-clicking in any toolbar area and selecting **Walk Through**.



## Examine

**Hotkey:** Ctrl + Right mouse button



Views the model from different angles. Drag the mouse to the right to see the left side of the object. Drag downward to see the top of the object.

## Slide

**Hotkey:** Ctrl + Shift + Right mouse button



Moves the camera along a plane parallel to the screen. You can move the camera up, down, left or right, but not forward or backward. Drag the mouse straight up to move the viewpoint upward, drag to the left to move it left. If you drag the mouse along an angle, the viewpoint will move along the corresponding angle. It's important to remember that the camera will move in the same direction as the cursor, which means that the Model Space will appear to move in the opposite direction.

## Roll

**Hotkey:** Shift + Right mouse button or Ctrl with the arrow keys.



Rotates the camera around the axis that passes through the camera. This tool will rotate the viewpoint either clockwise or counterclockwise. Drag the mouse to the left to rotate the viewpoint clockwise; drag to the right to rotate counterclockwise.

Vertical movement of the mouse will behave identically to the **Walk** control.

## Walk



Simulates walking around on the current plane. This is a handy way to walk through a 3D house model, for example.

Positions the camera on the current horizontal plane, also known as the viewplane. Moving the mouse forward or backward moves the viewpoint in the corresponding direction.

Left or right movement turns the camera in the corresponding direction. To move the viewpoint forward, drag the cursor toward the top of the screen. To move backward, drag toward the bottom of the screen. If you drag diagonally upward toward the left or right, the viewpoint will move forward at an angle; if you drag diagonally downward to the left or right, the viewpoint will move backward at an angle.

---

**NOTE:** *This tool will move forward or backward only when the **Perspective View** option is checked on the **Camera** page of the **Camera Properties**.*

---

## Turn



Rotates the camera around its center. This changes the angle of the viewpoint without relocating the camera. The viewpoint angle can be changed upward or downward. The viewpoint angle can be also be changed to the left or right. To turn the viewpoint right, drag the mouse to the right of the screen. To turn downward, drag toward the bottom of the screen. As with **Slide**, the viewpoint will move in the same direction as the cursor. This will have the effect of moving the view of the model on the screen in the opposite direction of the cursor motion.

## Local Menu Options

While using the **Walk Through** controls, there are several options available on the local menu and Inspector Bar.

**Undo Move:** Returns the view to the position in which the current mode was invoked.



**Straighten:** Returns the view to a level orientation.



**Discrete Movement:** Limits camera motion. When on, the camera will only move when the mouse is moving.



**Continuous Movement:** Allows the camera to move as long as the left mouse button is held down. The camera movement will be in the direction the mouse was dragged. This is the default option.



**Camera properties:** Opens the **Camera Properties** window. See "Camera Properties" on page 420



## Camera Objects

*Available in TurboCAD Pro and Platinum only*

Camera objects are used to create and save views with specific parameters. The view for each camera object can be displayed in a separate window. Each new camera is assigned a name by default (Camera 1, Camera 2, etc.) but this name can be changed.

You can display the **Camera** toolbar by right-clicking in any toolbar area and selecting **Camera**.



The difference between camera objects and saved views (see "Saving Views" on page 102) is that saved views do not open in separate windows, and do not include any additional parameters, such as render mode.

Camera indicators show you the location and orientation of the camera. Indicator visibility can be controlled in the **Camera Options** window (**View / Cameras**).

---

**NOTE:** You can also create and manipulate camera objects, and group cameras into camera sets, via the *Design Director*. See "Design Director: Cameras" on page 131.

---

### Camera Object by View

Inserts the camera at the center of the view, so that the camera view is the same as the current model view. The properties of this camera are those of the default camera representing the current view, described in the **Camera** page of the **Camera Properties** (see "Camera Properties" on page 420).

The camera object itself is not visible in the current window, and a separate view window will not be created, since you are already looking at the camera view.

### Camera Object Normal to View



Inserts the camera perpendicular to the current model view. The camera will face the same way as **Camera by View**, but can be placed anywhere.

### Camera Object Parallel to View



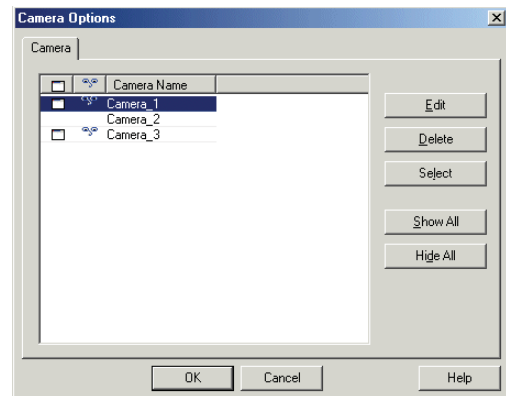
Inserts the camera on a plane parallel to the current model view. The first point places the camera, and the second point defines its direction.



### Camera By 2 Points



Inserts the camera by picking two points. The first point places the camera and the second point defines its direction.

### Camera Object Views



To display the view seen by the camera object, you can create a separate view window. To create a view window, click the  icon for the relevant camera. The  icon is used to show or hide the camera indicator.

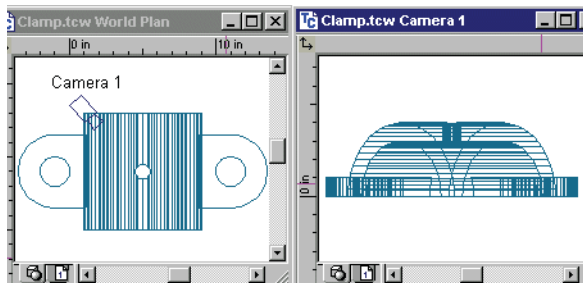
You can also use this dialog to edit and delete camera objects.

**Select:** Selects the highlighted camera in the drawing. This option is useful when the drawing is large, and you are unable to find the camera in the screen.

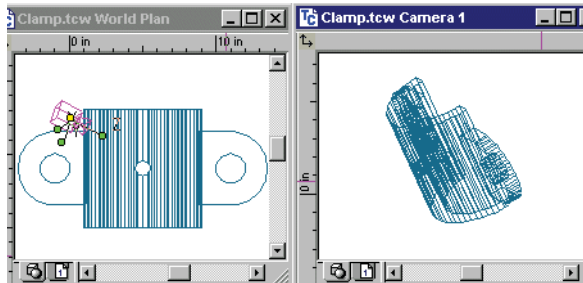
**Show All:** Makes all camera objects visible.

**Hide All:** Hide all camera objects.

To display all view windows, select **Window / Cascade** or **Window / Tile**.

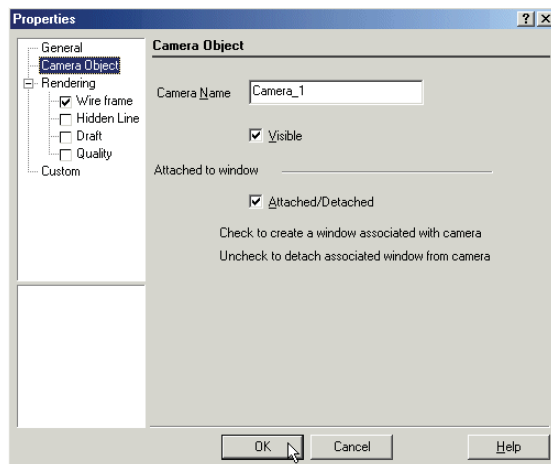


If you change the direction of a camera in one window, the views in any attached windows will update accordingly. If you use other view controls in an attached window, the camera position will be changed.



## Camera Object Properties

To access the camera object **Properties**, double-click a camera object.



**Visible** - Check this box to display the camera object indicator in the view window.

**Attached / Detached** - Check this box to create a new drawing window containing the camera view. Updates to the camera will update the view. Unchecking the box will disconnect the window from the view, so that updates to the camera will not affect the view.

## Camera Sets

A camera set is a group of cameras, which can be handy if you want to set rendering, perspective, and visibility parameters for multiple cameras. Camera sets are created and manipulated in the Design Director. See "Design Director: Cameras" on page 131.

## QuickTime Movies

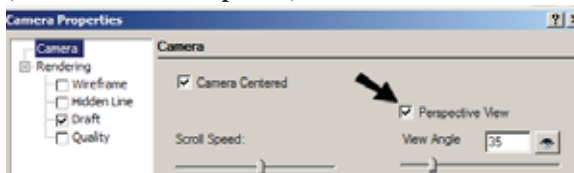
If the model is rendered by LightWorks, you can export the scene to a QuickTime movie format. This format does not create an actual animated movie, rather it exports to a format in which you can rotate and view objects from different angles.

The QuickTime player can be downloaded for free from [www.apple.com](http://www.apple.com).

There are two types of QuickTime movies:

- **Panoramic:** Constructed from a series of images taken from a fixed viewpoint. You can rotate around the viewpoint and look in all directions.
- **Object:** Constructed from a series of images taken from different angles. You can rotate the objects in any direction, to view all sides.

Before you can create a movie, the scene must be in **Perspective** mode, which is set in the **Camera Properties** (**View / Camera / Properties**).



Also, the LightWorks rendering must be one of the following modes:

- **Flat** (Draft rendering)
- **Gouraud** (Draft rendering)
- **Phong** (Draft rendering)
- **Preview** (Quality rendering)
- **Full** (Quality rendering)

See "Render (Camera) Properties" on page 497.

## Creating a Movie

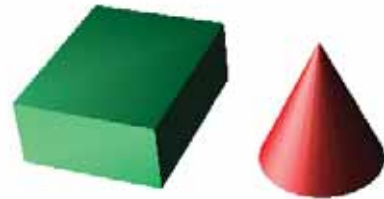
Panoramic QT Movie



Create Object QT Movie



1. Create the 3D model and view it in **Perspective** mode.



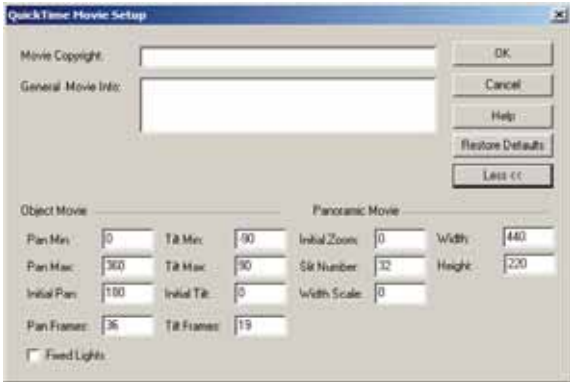
2. Create either the Panoramic or Object movie. You are asked for a file name, and you are notified when the movie is created.
3. Play the movie using the QuickTime Viewer. With either movie type, you can pan or rotate the scene by holding the left mouse button and moving the mouse.



Movie Setup



Opens the **QuickTime Movie Setup** window.



To display the lower section of the window, click the **More** button.

**Movie Copyright** and **General Movie Info**: Identifying information about the movie.

**Object Movie properties**

**Pan Min, Pan Max**: Limits of horizontal pan position, in degrees.

**Initial Pan**: The starting horizontal angle, in degrees, for when the movie is first opened.

**Pan Frames**: The number of frames to generate between the **Pan Min** and **Pan Max** positions. With the default value of 36, frames are rendered every 10 degrees.

**Tilt Min, Tilt Max**: Limits of vertical pan position, in degrees.

**Initial Tilt**: The starting vertical tilt angle, in degrees, for when the movie is first opened.

**Tilt Frames**: The number of frames to generate between the **Tilt Min** and **Tilt Max** positions. With the default value of 19, frames are rendered every 10 degrees.

**Fixed Lights**: As the camera moves around an object, the lights will stay fixed relative to the camera. This gives the effect that the camera is static and the object is rotating, which may look more natural in some cases.

---

*NOTE: Object movie files contain a large number of frames, and can therefore take a long time to create. With the default values of 36 pan frames and 19 tilt frames, 684 frames will be created.*

---

**Panoramic Movie properties**

**Initial Zoom**: Defined by a value between 0 and 100.

**Slit Number**: The number of individual images that should be rendered. The default value should be used for all but advanced users; setting this value too low could result in distortion, and setting it too high can result in minimal quality gain.

**Width Scale**: Movie width will be set as follows:  
$$\text{Movie width} = (\text{Width Scale}) * (\text{Width} / 2) - \text{Width}$$
When you increase the movie width, be aware that rendering time and movie file size also increases.

**Width, Height**: The dimensions of the movie viewing window.

# 11 Editing in 3D

This section covers ways to change 3D object geometry, including node editing and Boolean and blending operations.

You can use most 3D editing tools (and all 3D object tools) in both wireframe and render modes (*in TurboCAD Pro only*). See "Creating and Editing Objects in Render Mode" on page 496.

## Modifying Object Geometry

As with 2D geometry, there are several ways to modify the geometry of an object. Node editing (**Edit Tool**) can be used on an object itself, or on the profile used to create an object. (A profile can also be moved, rotated, or scaled in **Select Edit** mode.)

Objects in 3D can be moved, scaled, copied, and rotated in **Select Edit** mode (see "Select Edit" on page 198), in the same way as for 2D objects. The 3D Selector must be active for work in 3D (see "2D / 3D Selector" on page 181). In 3D **Select Edit**, you can move in 3 dimensions and rotate about 3 axes.

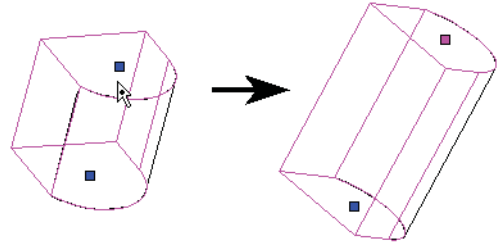
*Solids are only available in TurboCAD Pro.*

### Edit Tool in 3D

Most 3D objects do not have nodes to edit. The exceptions are **Box**, **Cone**, **Sphere**, **Hemisphere**, **Revolve**, and **Extrude**. These objects have limited nodes that can be used only for scaling and orientation. These nodes are lost once Boolean operations are performed.

Editing nodes of 3D objects works the same way as for 2D objects (See "Edit Tool" on page 219). The main difference is that nodes can be moved anywhere in 3D space. Also, 3D

nodes cannot be added or deleted from objects themselves (though they can be added or deleted while node editing profiles).



While node editing is limited, if possible at all, for most 3D solids, you *can* node edit an exploded surface object.

You can change a solid object to a TurboCAD (non-ACIS) surface object via the **3D** page of its **Properties** window. Explode the surface so that it can be node edited.

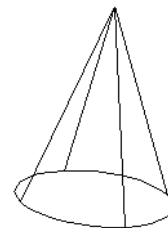
---

**TIP:** You can explode a solid object twice to turn it into an exploded (node editable) surface object.

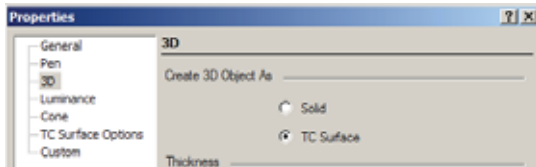
---

### Editing Nodes of Exploded Surfaces

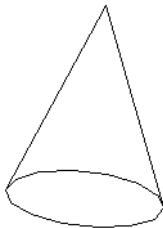
This example uses a basic cone. If it was created as a solid, and you try to node edit it, you will see that it cannot be selected.



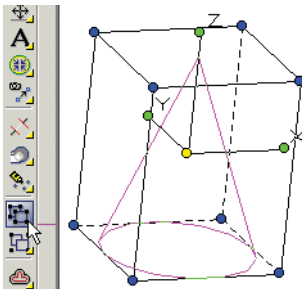
1. Select the cone and open its **Properties** (see "Object Properties" on page 79). On the **3D** page, select **TC Surface**.



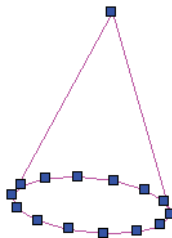
The cone is now comprised of surfaces, but it still cannot be node edited.



2. Select the cone again, and click **Explode** (or select **Format / Explode**).

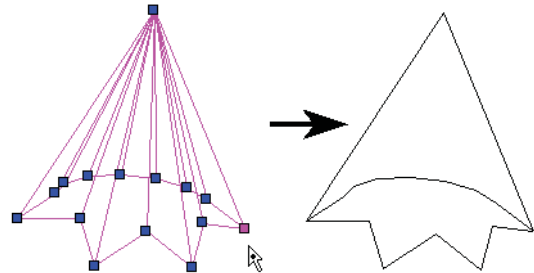


3. You can now node edit the cone nodes.

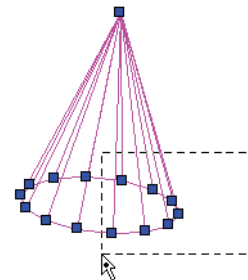


*NOTE: You also could have exploded the original solid object twice to obtain an exploded (node editable) surface.*

4. Drag individual nodes to change the shape of the cone.

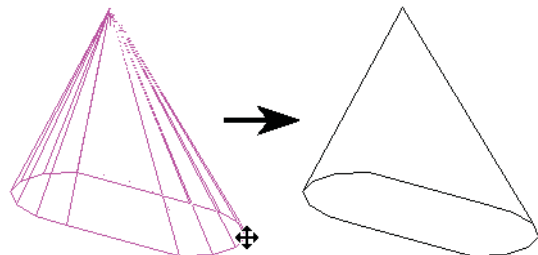


You can also drag a group of nodes simultaneously. Use a selection window to select the desired group of nodes.



After they are selected, the nodes appear in magenta.

Now when you drag any one of the selected nodes, the entire group moves.

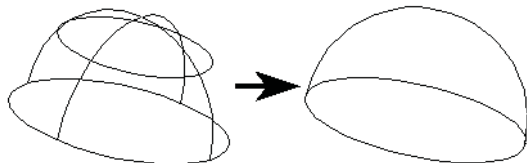




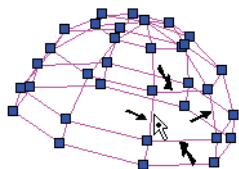
## Editing Nodes of Edges of Exploded Surfaces

For exploded 3D surface objects, you can use the **Edit Tool** to control the visibility and usage of edges.

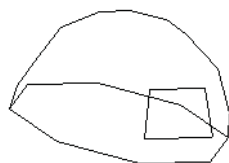
1. Create a solid object and transform it into an exploded surface object, using the methods described in "Editing Nodes of Exploded Surfaces" on page 427.



2. Node edit the object. To make an edge visible, right-click between its nodes and select **Make edge visible**.



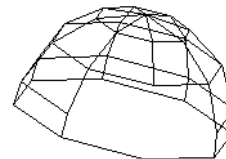
3. Do the same for all edges you want to show. You can hide visible edges by selecting **Make edge invisible** from the local menu. Press Esc twice to leave the **Edit Tool**, and the selected edges are shown.



4. To modify form-building edges, turn them on by checking **Draw form-building edges** on the **Display** page of the **Drawing Setup (Options / Display)**.



5. Right-click on nodes you want to hide and select **Make edge non form-building**.



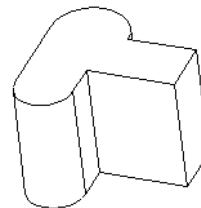
You can display edges by selecting **Make edge form-building**.

## Profile Editing

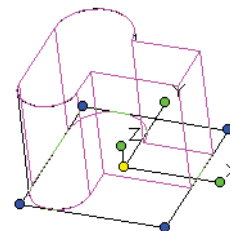
Some types of 3D objects are created by performing a function on a profile (see "Profile Objects" on page 388). Such objects include **Simple Extrude**, **Revolve**, and **Lofting**. Some standard objects are also based on profiles, such as **Cone**, **Wedge**, **Cylinder**, **Polygonal Prism**, and **Torus**. For standard objects, the profiles are created within the 3D tool; profile objects reference a profile that has already been created.

For objects that are based on a 2D profile, you can edit the profile in **Select Edit** mode (see "Select Edit" on page 198).

This example will use a **Simple Extrude** object, created from a closed polyline.



1. In **Select** mode, select the object. Right-click and choose **Select Profile** from the local menu.

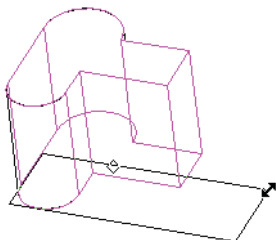


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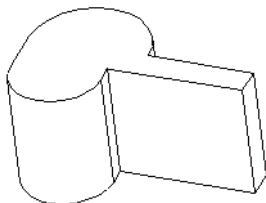
**WARNING:** *The workplane moves to coincide with the selected profile. This may change the workplane you want to be active. If needed, you can use **Previous** to return to the desired workplane.*

---

2. You can move, rotate, or scale the profile as you would any 2D object in **Select Edit**. See "Select Edit" on page 198.



3. Once the profile is edited, click outside to return to the 3D object, and click outside again to clear the selection. (You can also press Esc twice to exit **Select** mode.) The object is now based on the edited profile.

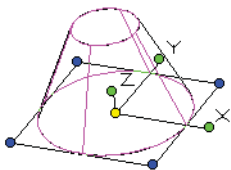


## Editing Multiple Profiles

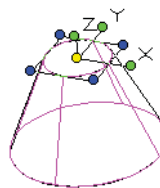
Some objects like truncated cones and lofts use multiple profiles. You can access and edit each profile separately.

This example will use a truncated cone.

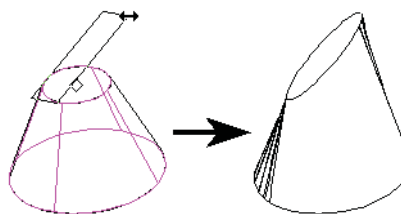
1. In **Select** mode, select the object. Right-click and choose **Select Profile** from the local menu.



2. If the selected profile is not the profile you want to edit, right-click again and choose **Select Next Profile**. You can use this method to scroll forward and backward (**Select Previous Profile**) until you reach the desired profile.



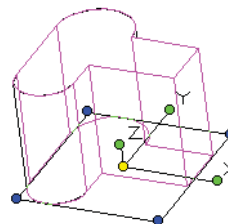
3. Edit the desired profile as needed to change the object.



## Editing Nodes of a Profile

If you want to edit profiles beyond the moving, rotating and scaling possible in **Select Edit**, you can also edit profile nodes. For details, see "Edit Tool" on page 219.

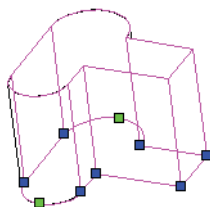
1. In **Select** mode, select the object. Right-click and choose **Select Profile** from the local menu.



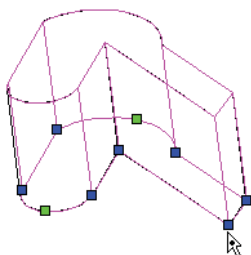
2. Right-click again and select **Edit Node**, or click **Edit Node** in the Inspector Bar.



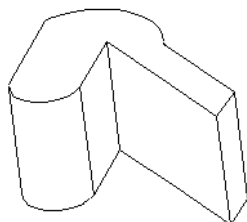
The profile nodes appear.



3. Drag the nodes as needed to change the profile shape.



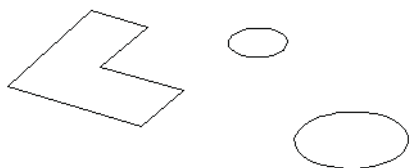
4. Return to **Select** mode, and then clear the selection, by pressing Esc twice.



### Updating Compound Profiles

Profile objects can be based on simple (single-curve) profiles or compound profiles. Once the 3D object is created, the curves that comprise the compound profile can be changed, and the resulting solid will update.

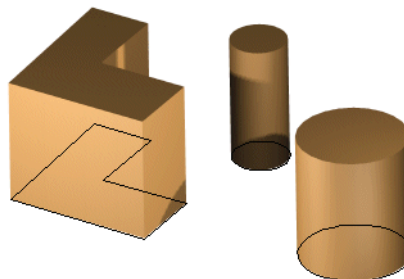
This example uses these three closed 2D compound profiles, but simple profiles will work as well.



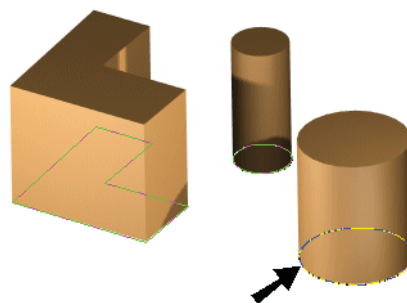
1. In this example, a **Simple Extrude** will be created (see "Simple Extrude" on page 390). Make sure **Use Compound Profile** is selected.



2. Select one profile, then press Shift and select the other two.



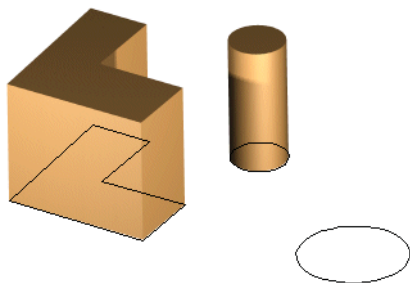
3. There are two ways to update the profile:
  - Select **Update Compound Profile** and select the profile used to create the extrusion.
  - Or **Select** the 3D object. With it selected, right-click and select **Select Profile**. With the profile then selected, right-click again and select **Update Compound Profile**.
4. Click one profile to deselect it.



5. When the profile is modified, select **Finish Selection of Profile**.

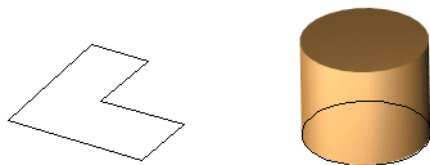


The extrusion updates to reflect the selected profile curves.

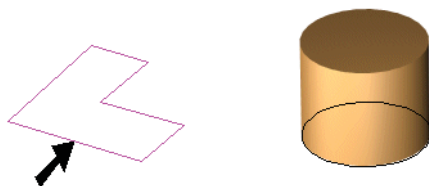


You can also use this tool to add or delete regions from multiple-compound region extrusions. You can also select profiles that were previously unused.

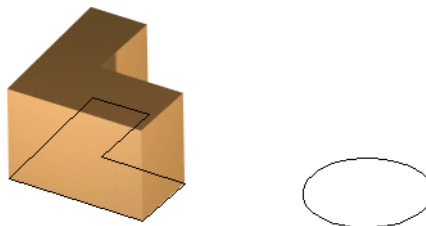
Finally, you can update a 3D object that is based on a simple profile to be based on a compound profile (but not vice-versa). In this example, the extrusion is based on a circle, and was not created using **Use Compound Profile**.



1. Select the extrusion, and select **Select Profile**. Right-click again and select **Update Compound Profile**. Select the compound profile.



2. Click **Finish Selection of Profile**, and the extrusion is now based on the compound profile.



While creating solids based on compound profiles, it is recommended to work with Editing History active (see "Editing 3D Objects using Selection Info" on page 487). This ensures that you will be able to modify the profiles in **Select Edit** mode.

## 3D Boolean Operations

Boolean operations use two existing 3D objects to create a new object. Objects can be combined, subtracted, intersected, or sliced. The objects must both be of the same type - surfaces or solids.

You can display the **Boolean & Facet** toolbar by right-clicking in any toolbar area and selecting **Boolean & Facet**.



These tools are also available on the fly-out toolbar from the **Drawing Tools**.



When Boolean operations are completed, you can edit the shape and location of the objects used to create them. See "Editing 3D Objects using Selection Info" on page 487.

If you plan to assign materials to your 3D objects, it is recommended to do this after the Boolean operations are performed.

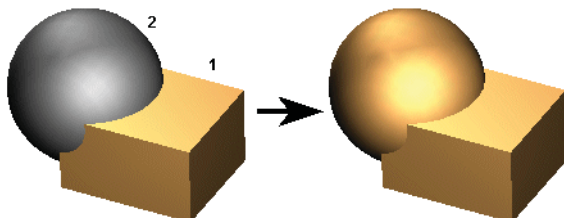
NOTE: The **Imprint** tool also uses Boolean operations, combining a solid with the extrusion of a 2D closed profile. See "Imprint" on page 471.

## 3D Add

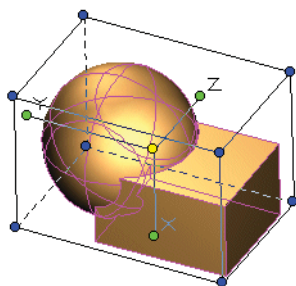


Combines two or more 3D objects into one object. Any overlapping volume is removed.

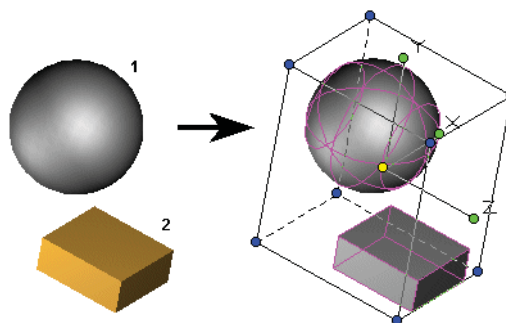
Select two objects to combine, and the second object is joined to the first object. Selection order is important, because the new object will have all the properties of the **first** object selected, such as layer and material.



The resulting object is one object. You can verify this by selecting the object.



The selected objects do not have to overlap. If you combine non-overlapping objects, they will still be combined into one object.



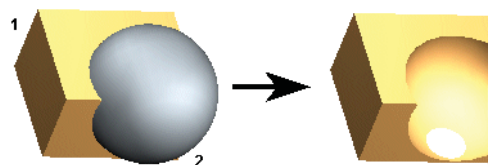
To select multiple objects hold down the shift key while selecting, then select **Finish Selection** from the local menu.

## 3D Subtract



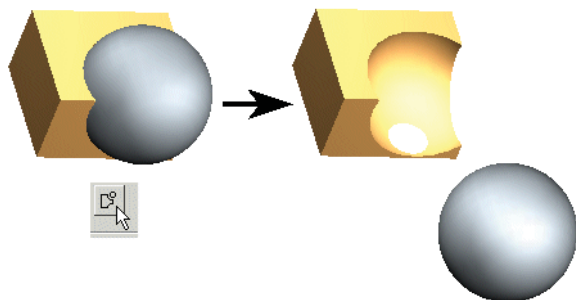
Subtracts one or many 3D objects from one 3D object.

First select the object to subtract from, then select the object to subtract. By default, the second object is deleted.



Local menu option:

**Don't remove the subtrahend:** Keeps the second (subtracted) object. The resultant objects appear the same, but if you move the second object you can see that it was subtracted from the first.



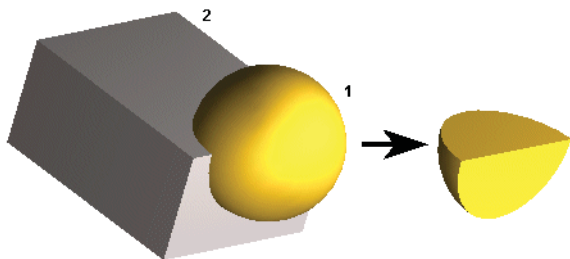
To select multiple objects hold down the shift key while selecting, then select **Finish Selection** from the local menu. The first object will act as the minuend, and all of the subsequent objects will be subtrahends.

### 3D Intersect



Creates one 3D object that is the overlap (intersection) between two objects.

Select two objects that overlap. Selection order is important, because the new object will have all the properties of the **first** object selected, including layer, and material.



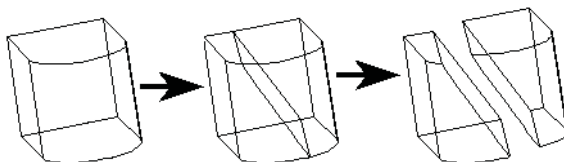
#### Local menu option:

**Keep Original:** When this option is selected the original objects used in the intersection will be retained.

### 3D Slice



Slices, or divides, one or more objects into new objects, by specifying the slicing plane.



NOTE: The **Section** tool is similar, but it creates a section of the object on the specified plane, while leaving the original object intact. See "Sectioning Solids" on page 456.

### Slice by Line

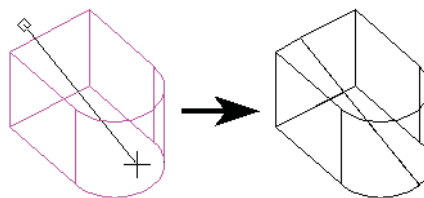
Slices objects by a plane perpendicular to the current view, defined by two points. This is the default option.

1. Select **Slice by Line (2 Points)** from the local menu or Inspector Bar.

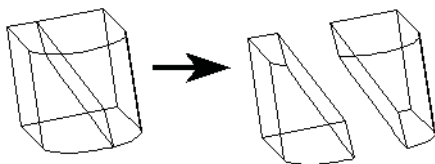


2. Select the objects to slice. You can use Shift to select more than one object.
3. Select two points The slicing plane will pass through the line defined by these points, in the direction normal to the current view (into the screen).

The slice is created, though in the current view the dividing plane appears as a line.



You can rotate the view and move one of the new objects to see how the original object was sliced.



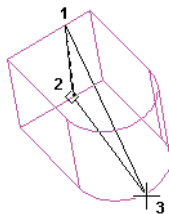
### Slice by Plane

Slices objects by defining the slicing plane. The plane is defined by three points.

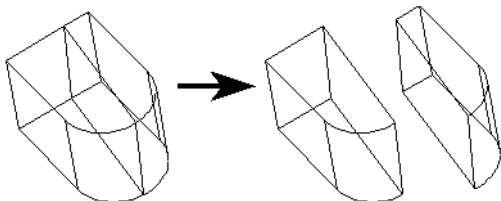
1. Select **Slice by Plane (3 Points)** from the local menu or Inspector Bar.



2. Select the object to section. You can use Shift to select more than one object.
3. Select three points to define the slicing plane.



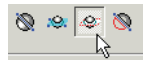
The slice is created. You can move one of the new objects to see how the original object was sliced.



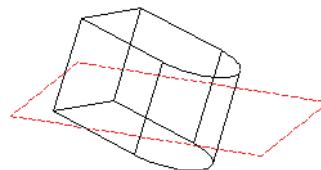
### Slice by Workplane

Slices objects by using the current workplane as the slicing plane. For details on workplanes, see "Workplanes" on page 362.

1. Select **Slice by Workplane** from the local menu or Inspector Bar.



2. Select the object to slice.

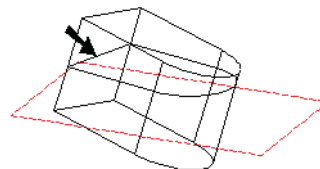



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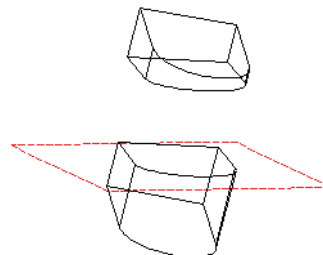
*NOTE: To slice multiple objects, start out using a different option - **Slice by Line** or **Slice by Plane**. Use Shift to select the objects to slice, then click the **Slice by Workplane** icon.*

---

The object is sliced.



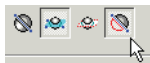
You can move one of the new objects to see how the original object was sliced.



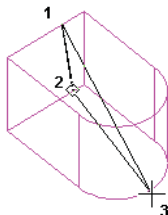
### Delete a Sliced Part

Removes one of the new objects created by slicing objects. This option is used in tandem with one of the other options.

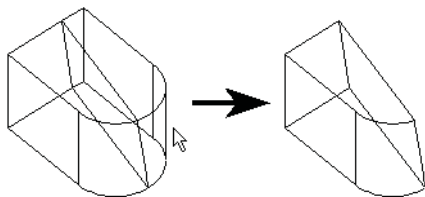
1. Select the desired slicing option (in this case, **Slice by Plane**) and **Delete a Sliced Part** from the local menu or Inspector Bar.



2. Select the object to section. You can use Shift to select more than one object. Slice the object by the selected method, in this case, defining a plane by three points.



3. The slice is created. Select the object you want to delete.




---

*TIP: Objects created as a result of slicing can also be deleted by selecting them and pressing the Delete key.*

---

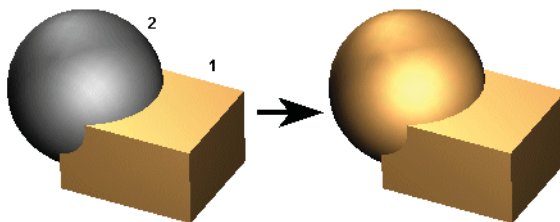
## MultiAdd\_VB6

*Available in TurboCAD Platinum Only*



Combines two or more 3D objects into one object. Any overlapping volume is removed.

Select two or more objects to combine, then click the MultiAdd\_VB6 tool. All of the selected objects will be combined into one object



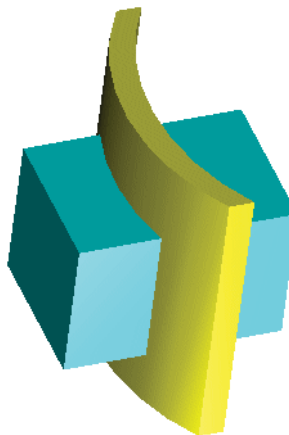
The resulting object is one object. You can verify this by selecting the object.

## Exploding Solids

*Available in TurboCAD Pro and Platinum only*

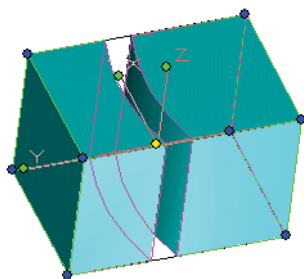
If you use Boolean functions such as **3D Add** or **3D Subtract**, it is possible to get a solid comprised of two or more separate objects that are still considered to be one object. It can be difficult to split this solid using 3D editing tools, but the **Explode** tool can be easily used for this.

1. Start with two solids that overlap.





2. Use **Boolean Subtract** to remove one solid from the other. In **Select** mode the two parts are considered to be one object.



3. With the object still selected, activate **Explode** (**Format / Explode**).

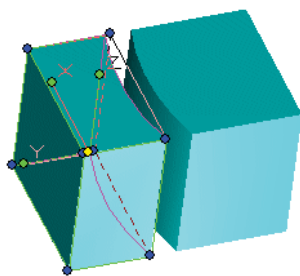



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NOTE: For more information on exploding, see "Exploding Objects" on page 247.

---

The two parts are now considered to be separate objects.

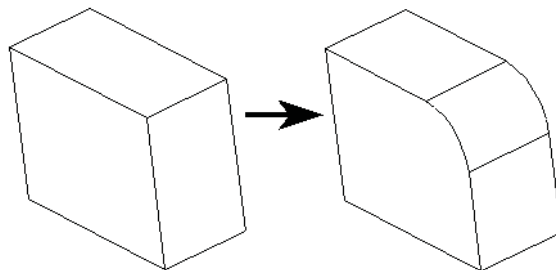


## Fillet Edges

*Available in TurboCAD Pro and Platinum only*



Rounds edges of solid (ACIS) 3D objects.

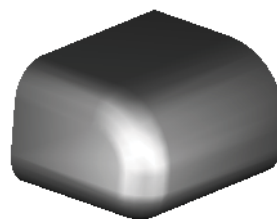


Only solid objects (not surfaces) can be selected for filleting. Filleting can either add or remove material; material is added to a concave edge, and it is removed from a convex edge. You can choose whether to create rounded or mitered vertices.

---

NOTE: You can edit a **Fillet Edges** operation in the Selection Info palette. See "Editing 3D Objects using Selection Info" on page 487.

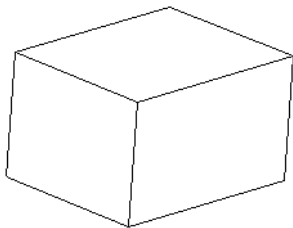
---



## Fillet with Round Vertices

Rounding the vertices produces a smooth rounding along all edges and corners.

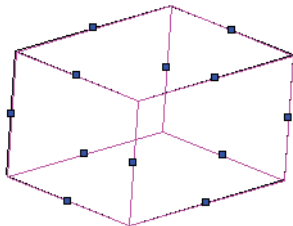
This example will start with a box.



1. Set the **Start Radius** in the Inspector Bar. Select **Round Vertex** in the Inspector Bar or on the local menu.



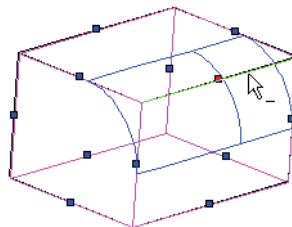
2. Select the object whose edges are to be rounded. Each edge is marked with a blue square.



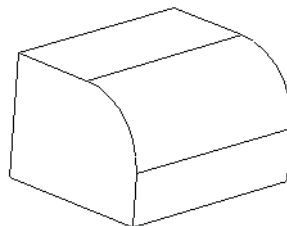
If you select **Hide Marks**, these edge markers will be hidden.



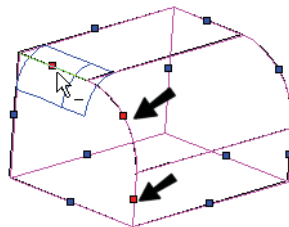
3. Select the first edge to round. A preview appears showing the fillet that will result.



4. Continue selecting edges, if needed. (You can also drag a window to select multiple edges). Selected edges are indicated by a red square.
5. To complete the fillet, select **Finish** from the Inspector Bar or the local menu.



6. In this example, if you select an edge that is smoothly connected to other edges, the connected edges will be selected automatically.

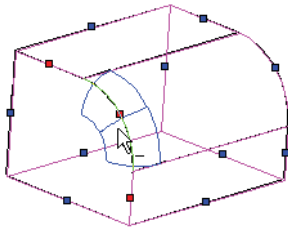



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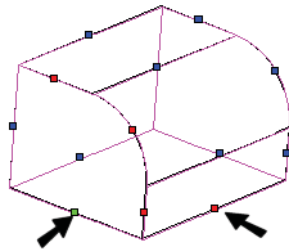
**NOTE:** *Adjacent edges are smoothly connected if they are connected by an arc or rounded vertex.*

---

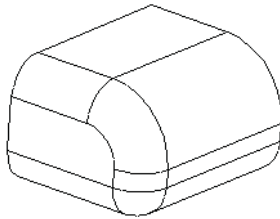
7. Clicking other edges will show the fillet preview along that edge.



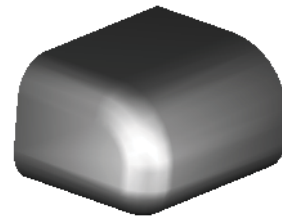
8. In some cases, selecting an edge (such as the bottom front edge) will cause another edge to be rounded as well (the bottom side edge), because they meet at a common vertex. These edges are indicated by a green square.



9. When all edges are selected, set the rounding radius in the Inspector Bar, and select **Finish** from the Inspector Bar or the local menu.



Viewing the results in render mode shows the effect of rounding. The common vertex is smoothly rounded.

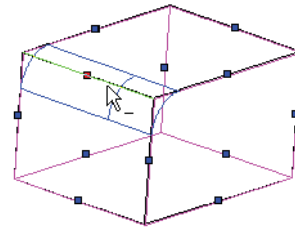


### Fillet with Setbacks

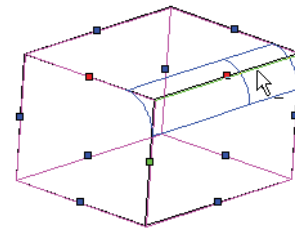
When rounding edges with the **Round Vertex** option, you can enter setbacks in the Inspector Bar. The setback is the distance from the vertex to the point on an edge where the rounding begins.

Setbacks are visible at vertices where all edges are rounded. The setback value must be greater than the edge rounding radius.

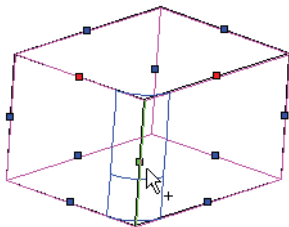
1. Select the object, set the radius, and select the first edge to round. When an edge is selected, the blue square turns red. If you select an edge that is smoothly connected to other edges, the connected edges will be selected automatically.



2. Continue selecting edges.

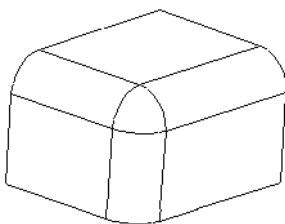


When you select two adjacent edges, any other edge that meets that vertex will be automatically selected as well, indicated by the green square.

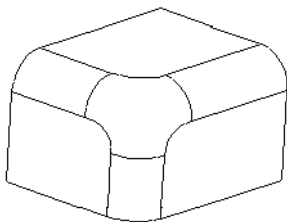


3. When all edges are selected, set the rounding radius in the Inspector Bar, and select **Finish** from the Inspector Bar or the local menu.

If no setbacks are used, the rounding extends along the entire edge.



If setbacks are used (entered in the Inspector Bar), the rounding starts at the specified distance from the vertex.




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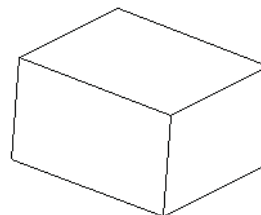
*NOTE: If the setback values are different, where they are applied depends on the structure of the object itself. You may need to experiment to get the desired result.*

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## Fillet with Miter Vertices

Mitering the vertices produces rounded edges that meet at an edge at the corners.

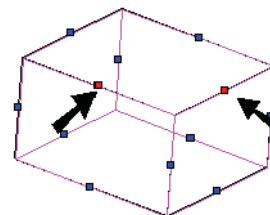
This example will use the following model.



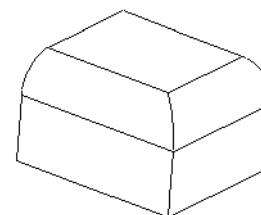
1. Make sure **Round vertex** is disabled.



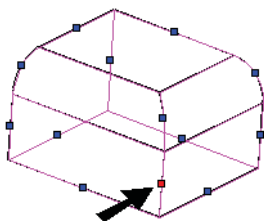
2. Select the object whose edges are to be rounded. Each edge is marked with a blue square.
3. Set the **Start Radius** radius in the Inspector Bar, and select the edges.



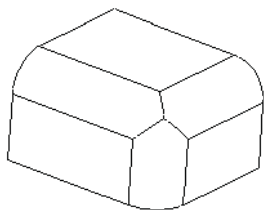
4. The edges is rounded, and meet at a sharp corner.



5. To round another edge, select the object again and select the next edge. Make sure **Pick smooth sequence** is not selected.



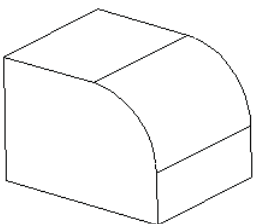
This is the result - the intersection of all three corners has sharp edges.



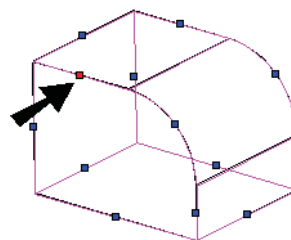
### Pick Smooth Sequence

Adjacent edges are smoothly connected if they are connected by an arc or rounded vertex. When working with **Round vertex**, smoothly connected edges are automatically selected. But this is not the case when **Round vertex** is turned off.

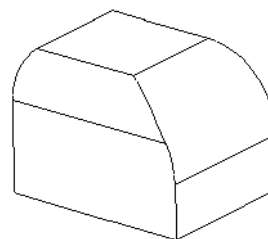
1. Start with a box with one filleted edge, using a large radius.



2. Activate **Fillet** without **Round vertex**, and fillet one edge, using a smaller radius.



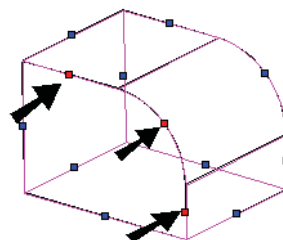
This is the result - only the selected edge was rounded.



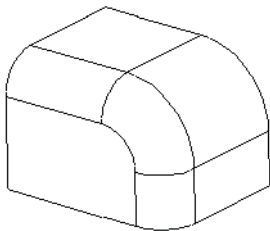
3. Undo, and this time select **Pick smooth sequence**.



4. This time when you select one of these edges for filleting, all edges in the smooth chain are selected.



This is the result - all edges in the chain are rounded.



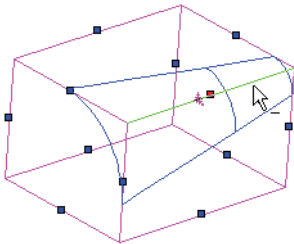
**Fillet with Variable Radius**

For all rounding types, you can use the **Unequal Radius** option to specify the radius at the start and end of the edge.

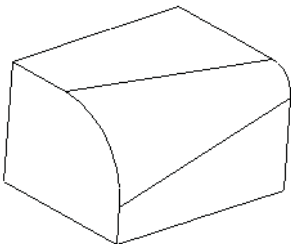


- 1. Specify the **Start Radius** and **End Radius** in the Inspector Bar.
- 2. Select the object, and select the edge or edges to round. For each edge, a small magenta arrow indicates the direction of the edge, so that you will know where the **Start Radius** and **End Radius** will be applied. You can reverse the direction by pressing Ctrl and selecting again.

Start Radius	End Radius	Setback 1	Setback 2
1.2 in	0.4 in	0 in	0 in



The edge is rounded with an varying radius.

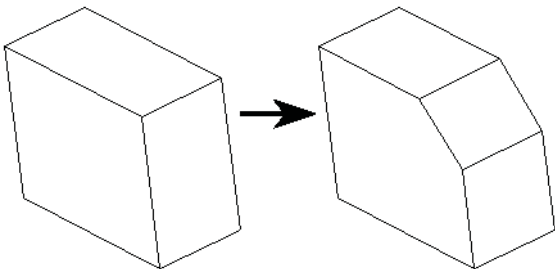


**Chamfer Edges**

*Available in TurboCAD Pro and Platinum only*



Chamfers (bevels) edges of solid 3D objects.



Only solid objects (not surfaces) can be selected for chamfering. Chamfering can either add or remove material; material is added to a concave edge, and it is removed from a convex edge. You can choose whether to create rounded or mitered vertices.

NOTE: You can edit a **Chamfer Edges** operation in the *Selection Info* palette. See "Editing 3D Objects using Selection Info" on page 487.

The chamfer size is set by the **Offset** and **Chamfer Angle** values. Offsets are the distances from the edge. If the **Offset** values are equal, the **Chamfer Angle** is 45 degrees.

Offset 1	Offset 2	Chamfer Angle
3 in	3	45

If you change one **Offset** value, the **Chamfer Angle** updates accordingly.

Offset 1	Offset 2	Chamfer Angle
3 in	2	56.31

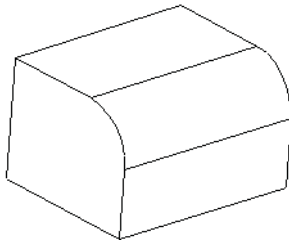
Similarly, if you change the **Chamfer Angle**, the **Offset 2** value will update accordingly.

Offset 1	Offset 2	Chamfer Angle
3 in	1.73 in	60

## Chamfer with Round Vertices

Rounding the vertices produces a smooth rounding along corners.

This example will use the following model, with one filleted edge.



1. Select **Round Vertex** in the Inspector Bar or on the local menu.

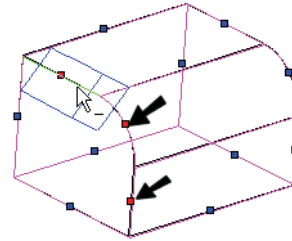


2. Select the object whose edges are to be chamfered. Each edge is marked with a blue square.

If you select **Hide Marks**, these edge markers will be hidden.



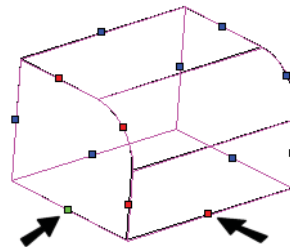
3. Select the first edge to chamfer. When an edge is selected, the blue square turns red. A preview appears indicating the chamfer size. If you select an edge that is smoothly connected to other edges, the connected edges will be selected automatically.



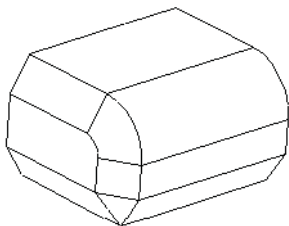
**NOTE:** *Adjacent edges are smoothly connected if they are connected by an arc or rounded vertex.*

4. Continue selecting edges, if needed. (You can also drag a window to select multiple edges).

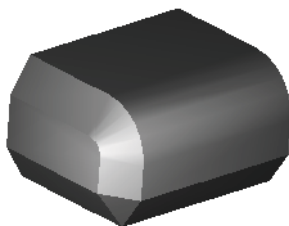
In some cases, selecting an edge (such as the bottom front edge) will cause another edge to be chamfered as well (the bottom side edge), because they meet at a common vertex. These edges are indicated by a green square.



- When all edges are selected, set the chamfering offsets in the Inspector Bar, and select **Finish** from the Inspector Bar or the local menu.



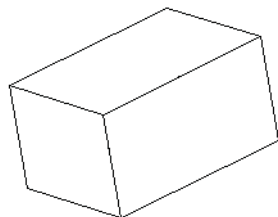
Viewing the results in render mode shows the affect of chamfering. The common vertex is smoothly rounded.



### ***Chamfer with Miter Vertices***

Mitering the vertices produces chamfered edges that meet at an edge at the corners.

This example will use the following model.

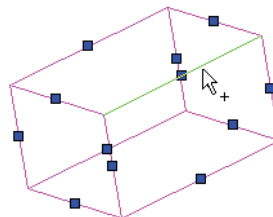


- Make sure **Round vertex** is disabled.

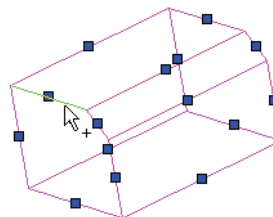


- Select the object whose edges are to be chamfered. Each edge is marked with a blue square.

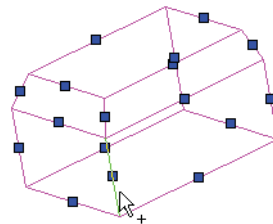
- Set the chamfer offset values in the Inspector Bar, and select the edge to chamfer.



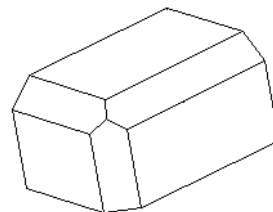
- The edge is chamfered. To chamfer another edge, select the object again and select the next edge.



- Continue to select edges to chamfer.



Any common corners will meet at a sharp edge.

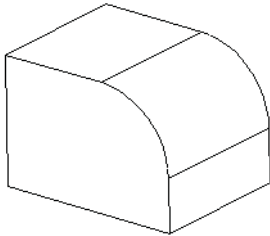




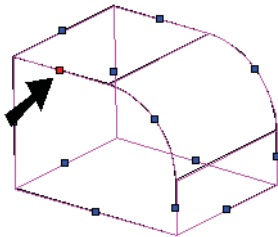
## Pick Smooth Sequence

Adjacent edges are smoothly connected if they are connected by an arc or rounded vertex. When working with **Round vertex**, smoothly connected edges are automatically selected. But this is not the case when **Round vertex** is turned off.

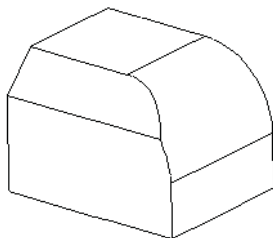
1. Start with a box with one filleted edge, using a large radius.



2. Activate **Chamfer** without **Round vertex**, and chamfer one edge, using a smaller offset.



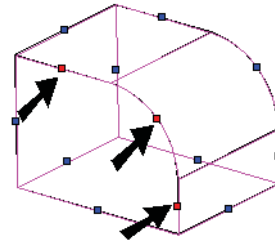
This is the result - only the selected edge was chamfered.



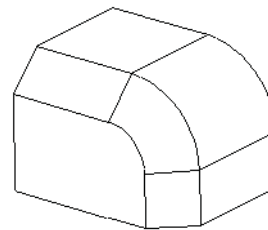
3. Undo, and this time select **Pick smooth sequence**.



4. This time when you select one of these edges for chamfering, all edges in the smooth chain are selected.



This is the result - all edges in the chain are chamfered.



## Bending and Unbending

These tools are used for bending existing sheets and tubes (pipes), or for adding flanges on sheets and tubes. There are also tools for unbending.

These tools are available on the **Modify** toolbar, which you can display by right-clicking in any toolbar area and selecting **3D Modify**.



## Bend Sheet

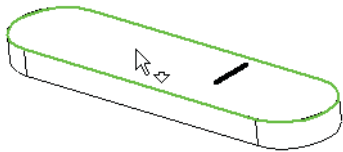
Available in TurboCAD Platinum only



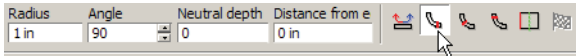
Bends a solid ACIS object, most commonly a sheet with thickness (Box).

NOTE: You can unbend the object using **Unbend Sheet**. See "Unbend Sheet" on page 451.

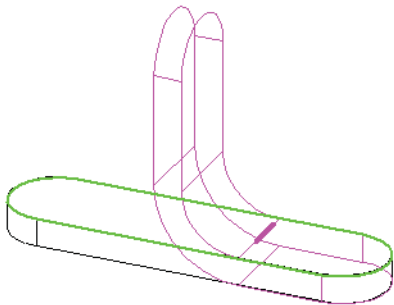
1. Start with a shallow **Box**, or a **Simple Extrude** based on a polyline or rectangle. Add a line to the top.
2. Activate **Bend**. Select the face you want to bend.



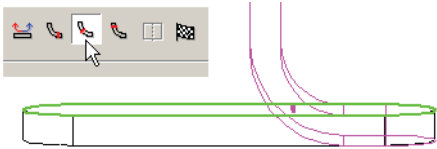
3. Then select the line about which you want to bend the solid. The line must lie on the solid face you selected.
4. Set the bending **Radius** and **Angle**. The default method is **Start**, in which the bend starts where the top face meets the selected line.



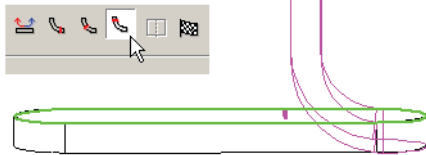
The preview shows how the final result will look.



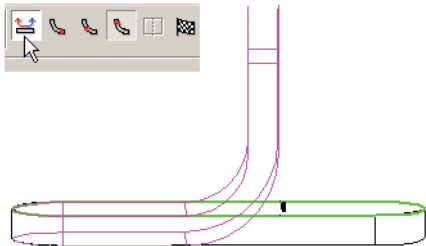
5. Switch to **Center**. In this case, the center of the bent solid meets the selected line (or its projection into the solid).



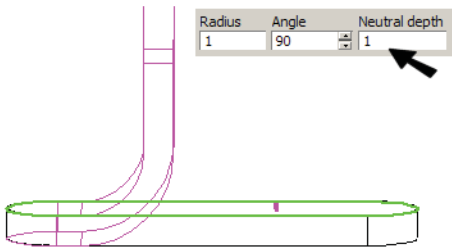
6. Switch to **End**. In this case, the bottom face of the bent solid meets the selected line (or its projection).



7. Select **Left Side**. The bend now starts from the other end of the solid.

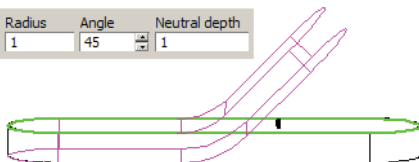


8. The **Neutral Depth** is the distance into the depth of material, along which there will be no tension or compression.

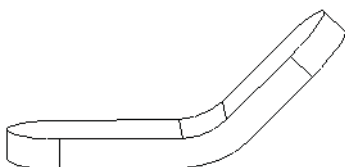


9. The **Angle** is measured from the plane of the bending face.

Radius	Angle	Neutral depth
1	45	1



10. When the parameters are set, select **Finish** from the local menu or Inspector Bar to complete the bend.

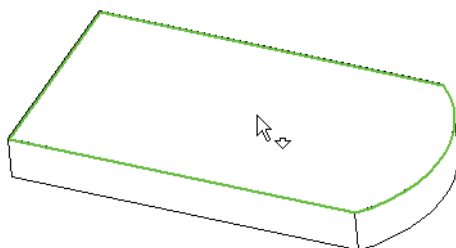


### By Distance from Edge

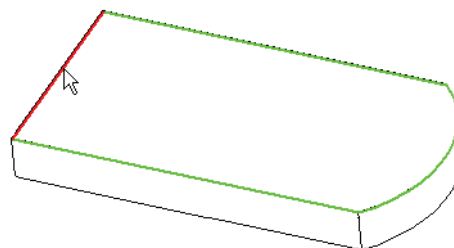
This option of the **Bend** tool enables you to define where the bend starts from a linear edge of the planar face.



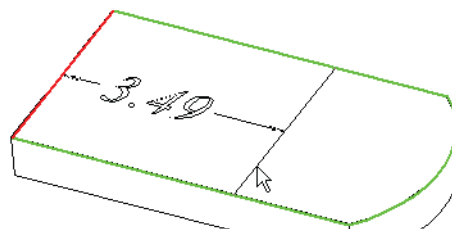
1. Start with a solid that has at least one linear edge. The first step is to select the face that will bend.



2. Then select the linear edge from which the bend distance will be defined.

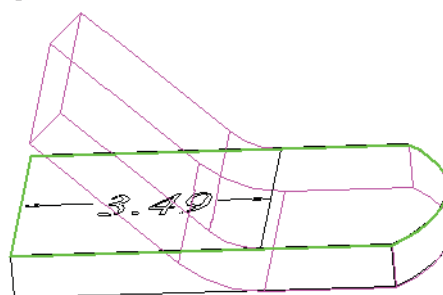


3. Pull the cursor from this edge to set the bend distance. Once you click to set the distance, you can change it by clicking again. You can also set the distance in the Inspector Bar.



Radius	Angle	Neutral depth	Distance from e
1 in	45	0 in	3.49 in

4. The preview shows how the final result will look.



Radius	Angle	Neutral depth	Distance from e	Preview
1 in	45	0 in	3.49 in	

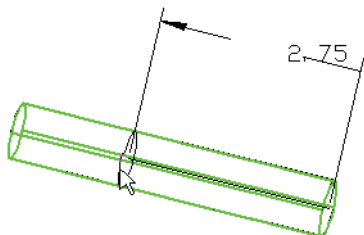
## Tube Bend

Available in TurboCAD Platinum only



Bends a solid (ACIS) cylinder.

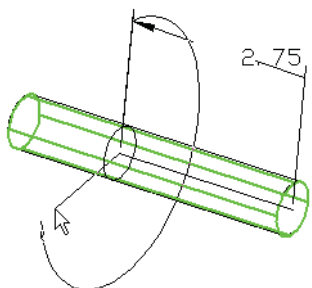
1. Start with a cylinder, and activate **Tube Bend**. Select the cylinder you want to bend. The next click defines the distance where the bend will start.



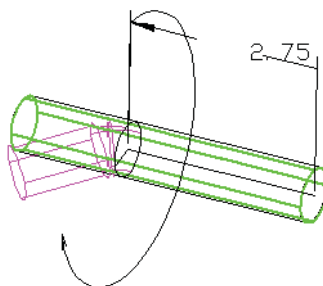
2. This distance is the **Axial distance**, which can be set in the Inspector Bar as well. You can also define the **Radius** and **Angle** here.

Radius	Angle	Neutral depth	Axial distance
0.1 in	45	0.37 in	2.75 in

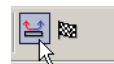
3. The next click defines the **Azimuth angle**, which defines the direction of the bend, relative to the axis of the cylinder.



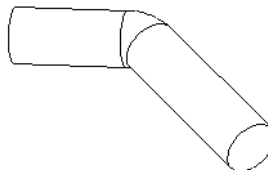
The preview shows how the final result will look. You can still make changes to all values in the Inspector Bar.



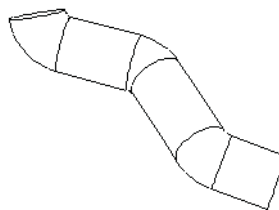
4. Click **Left Side** if you want to bend the cylinder from the other end.



5. Select **Finish** from the local menu or Inspector Bar to complete the bend.



You can use this tool multiple times on the same tube.



## Flange

Available in TurboCAD Platinum only



Adds a bend onto the edge of a solid (ACIS) object, most commonly a sheet with thickness (Box).

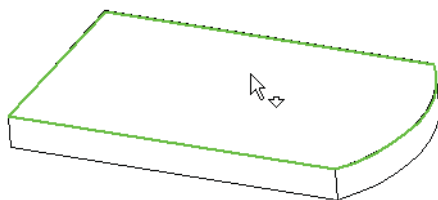
---

NOTE: You can unbend a flange using **Unbend Sheet**. See "Unbend Sheet" on page 451.

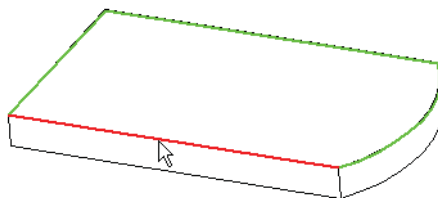
---

1. Start with a shallow **Box**, or a **Simple Extrude** based on a polyline or rectangle.

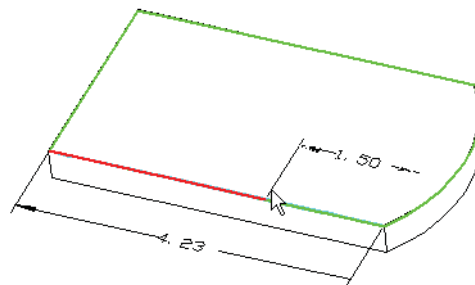
1. Activate **Flange**. Select the face to which you want to add the flange.



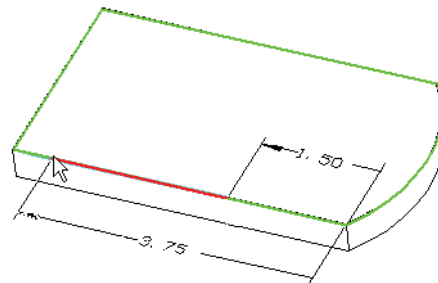
2. The next click defines the linear edge about which the flange will bend.



3. The next click defines how far from one end the flange will start. This distance can be zero. The total length of the edge is also indicated.



4. The next click defines how far from the other end the flange will end.

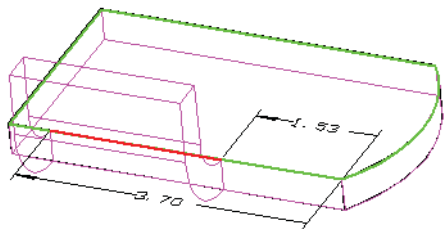


5. These distances are **Start flange position** and **End flange position**, which can be set in the Inspector Bar as well. You can also define the **Radius** and **Angle** here.

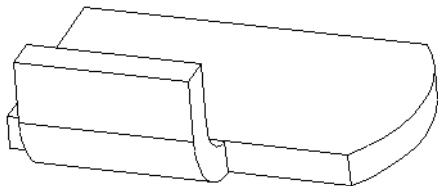
Radius	Angle	Neutral depth	Flange height	Start flange pos	End flange pos
0.5 in	90	0.25 in	2 in	1.53 in	3.7 in

The **Neutral Depth** is the distance into the depth of material, along which there will be no tension or compression. **Flange Length** is the length of the new material beyond the edge.

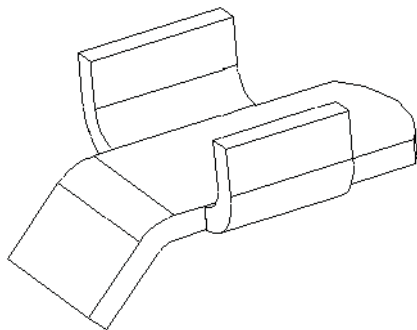
The preview shows how the final result will look. You can still make changes to all values in the Inspector Bar.



6. Select **Finish** from the local menu or Inspector Bar to complete the flange.



You can use this tool multiple times on the same sheet.



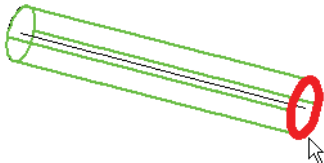
### Tube Flange

*Available in TurboCAD Platinum only*

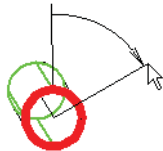


Adds a bend onto a solid (ACIS) cylinder.

1. Start with a cylinder, and activate **Tube Flange**. Select the cylinder you want to bend. The next click defines the end where the flange will be added.



2. The next click defines the **Azimuth angle**, which defines the direction of the bend, relative to the axis of the cylinder.

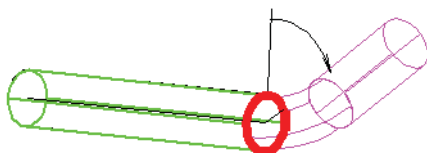


3. This angle can be set in the Inspector Bar as well. You can also define the **Radius** and **Angle** here.

Radius	Angle	Neutral depth	Flange height	Azimuth angle
0.25 in	90	0.37 in	2 in	60.97

The **Neutral Depth** is the distance into the depth of material, along which there will be no tension or compression. **Flange Length** is the length of the new material beyond the current end of the cylinder.

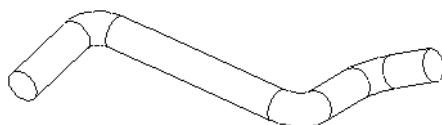
The preview shows how the final result will look. You can still make changes to any value in the Inspector Bar.



4. Select **Finish** from the local menu or Inspector Bar to complete the bend.



You can use this tool multiple times on the same cylinder.



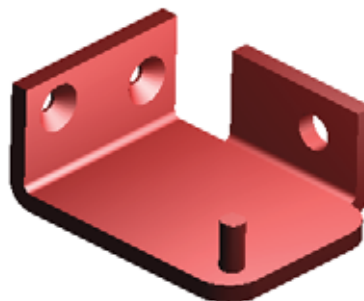
## Unbend Sheet

*Available in TurboCAD Platinum only*



Unbends a sheet that has a bend or flange. This tool is most commonly used on objects that have a **Bend** or **Flange** added (see "Bend Sheet" on page 446 and "Flange" on page 449) or on objects imported into TurboCAD.

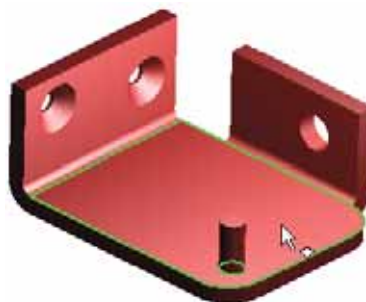
1. Start with a sheet that has a bend and/or flange. This example has a sheet with a cylindrical imprint, a bend with two holes, and a flange with one hole.



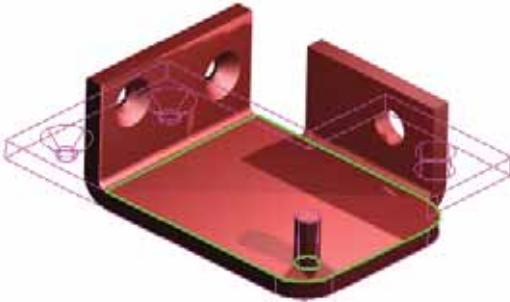
2. Activate **Unbend Sheet**. The **Neutral Depth** is the distance into the depth of material, along which there will be no tension or compression. Changing this parameter will very slightly affect the resulting sheet size.



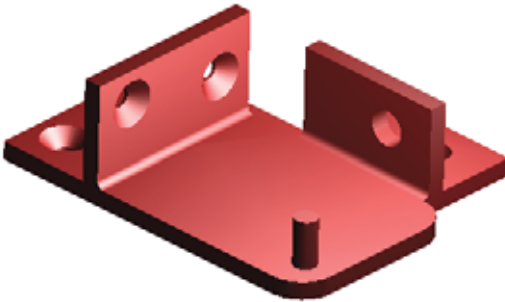
3. Select the face that will be used as the base for unbending. The resulting flat sheet will be flush with this face.



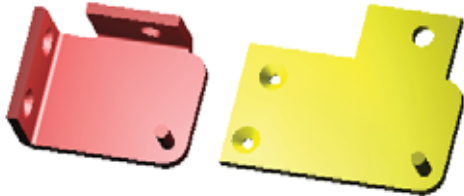
A preview of the resulting sheet will appear.



4. Select **Finish** from the local menu or Inspector Bar. The new, unbent sheet is created, overlapping with the base face.



5. To see the results, you can move the unbent sheet, or delete the original sheet.



## Unfold Face

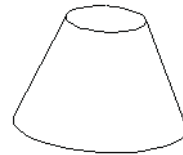
*Available in TurboCAD Pro and Platinum only*



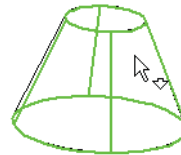
Unfolds faces of an ACIS object. This is also known as obtaining the involute of a surface.

### Example 1

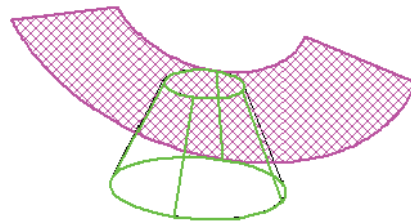
1. For a simple example, start with a truncated cone, using the **Cut Cone by Plane** option.



2. Activate **Unfold Face**. Click when the curved face is highlighted. (Only faces that can be unfolded will be highlighted.)

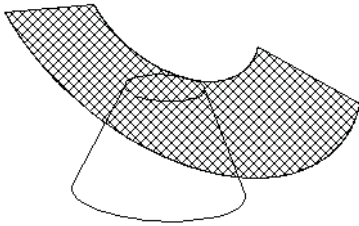


3. A preview of the unfolded face will appear.





- To create the face, select **Finish** from the Inspector Bar or local menu.



### Example 2

- For a more complex example, start with a **Rectangle**, filleted using the **Fillet** tool.



- Create a cone from this polyline, using the **Specify 2D Base** option.



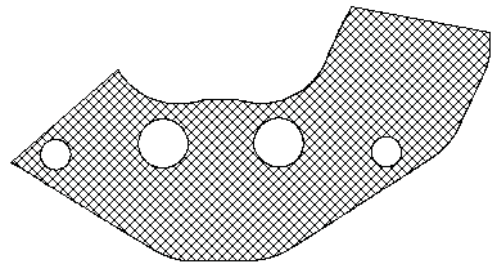
- Create some cylinders and use **3D Subtract** to remove them from the cone.



- Activate **Unfold Face** and click the cone.



- Select **Finish**. If you move or delete the cone, you can see the unfolded face clearly.



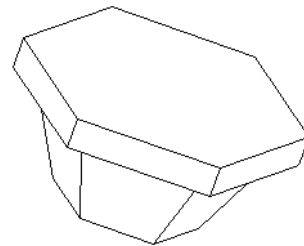
## Shelling Solids

*Available in TurboCAD Pro and Platinum only*

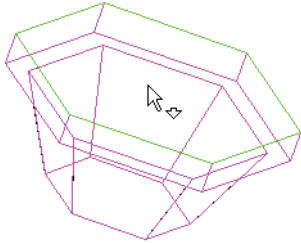


Creates a shell of a specified thickness from a single solid (not surface) object. The new faces are created by offsetting existing faces inside or outside.

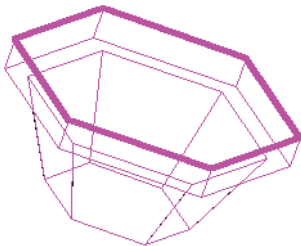
This example will use the solid shown below. (It is actually two solids, one **Lofting** and one **Simple Extrude**, combined into one solid using **3D Add**.)



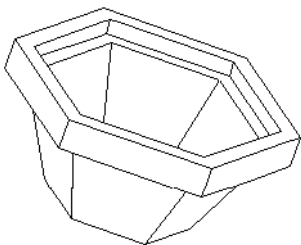
1. Activate **Shell Solid**. Select the object, which turns magenta.
2. Select any faces you want to remain open. Faces are highlighted in green when the cursor passes over them.



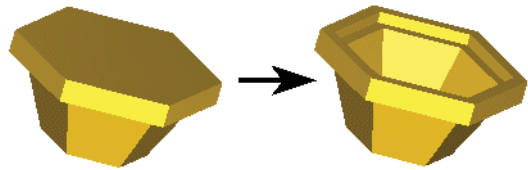
Open faces, when selected, are indicated by a thick magenta line. You can use the Page Down or Page Up key to select the next face.



3. Enter the shell thickness in the Inspector Bar. A positive value created an outward shell, and a negative value creates an inward shell.
4. Select **Finish** from the local menu or Inspector Bar.



Viewing the results in render mode shows the shelling results.



After the shell operation is complete, you can edit it in the Selection Info palette. See "Editing 3D Objects using Selection Info" on page 487

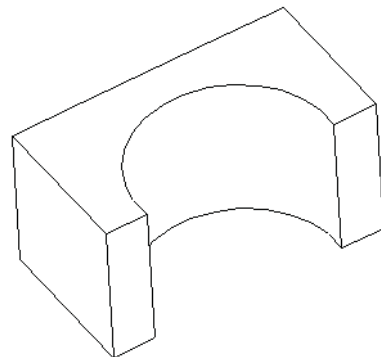
## Facet Offset

*Available in TurboCAD Platinum only*

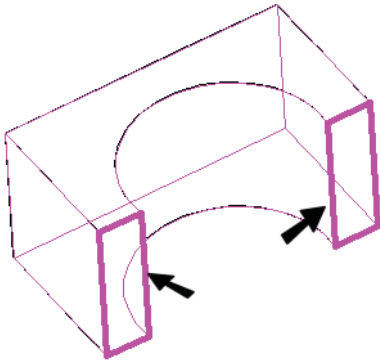


Extends a solid face inward or outward.

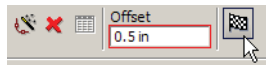
1. Start with a solid object. This example is a **Simple Extrude** based on a polyline.



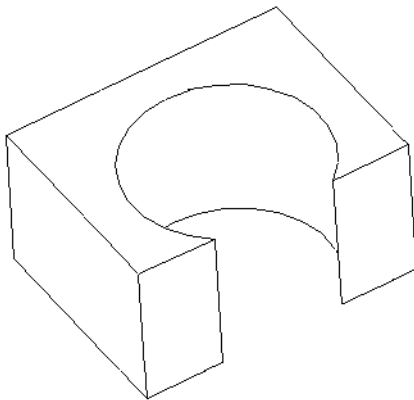
2. Activate **Facet Offset** and select the solid. Then select the face or faces you want to offset.



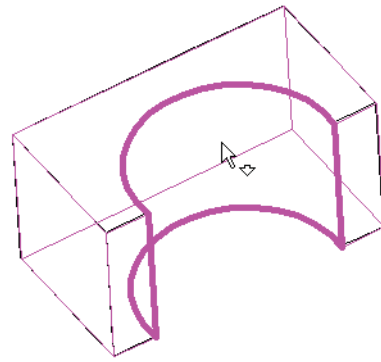
3. Enter the **Offset** distance - a positive value will offset the face outward, and a negative value will offset inward. Select **Finish** to offset the faces.



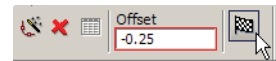
4. The faces are moved outward.



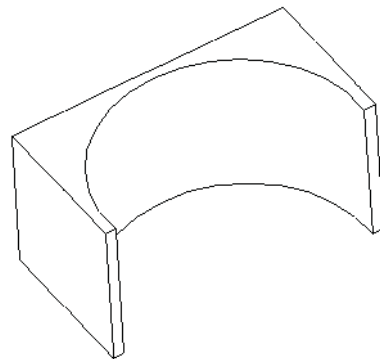
5. Offset faces do not have to be planar. In this example, select the curved face.



6. Use a negative **Offset** and select **Finish**.



The face moves inward.



If the Part History is activate, you can edit the offsets after they are created. See "Editing 3D Objects using Selection Info" on page 487.

---

**NOTE:** If the **Offset** distance will produce a geometrically incorrect solid, or will make the solid disappear, the action will be ignored.

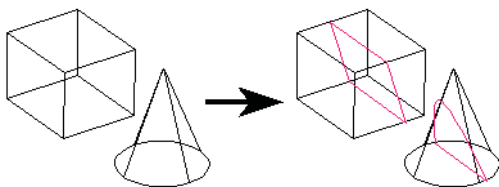
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## Sectioning Solids

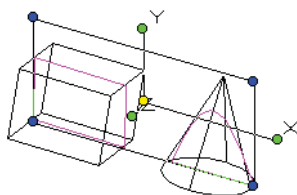
Available in TurboCAD Pro and Platinum only



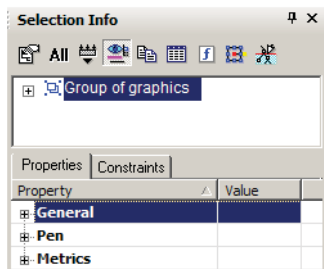
Creates a 2D or 3D section of one or more 3D solid or surface objects.



The resulting sections are grouped into one object.



The sections are considered a "Group of graphics," which can be viewed and manipulated in the Selection Info Palette (see "Selection Info Palette" on page 191).



For a 2D section, the "Group of graphics" is composed of lines, circles, ellipses, circular arcs, elliptical arcs and/or splines as needed to create the section. For a 3D section (**Section by Closed Polyline**), the "Group of graphics" is composed of solids and/or surfaces generated by the intersection of the selected objects and the normal extrusion of the closed polyline.

NOTE: The **3D Slice** tool is similar, but uses the specified plane to divide the object into new objects. See "3D Slice" on page 434.

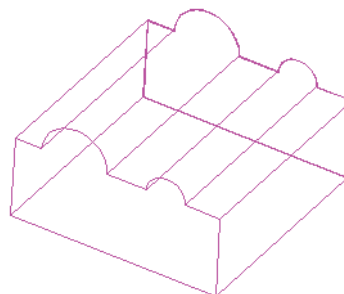
### Section by Line

Creates a section perpendicular to the current view, by defining two points. This is the default option.

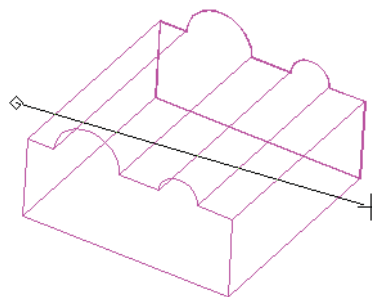
1. Select **Section by Line (2 Points)** from the local menu or Inspector Bar.



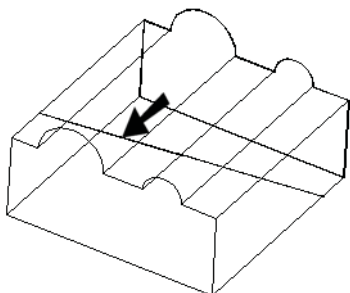
2. Select the object to section. You can use Shift to select more than one object, or use a selection window (first click outside the objects, then drag the window).



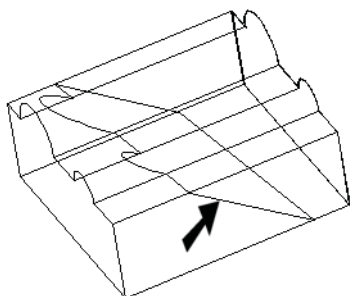
3. Select two points. The section plane will pass through the line defined by these points, in the direction normal to the current view (into the screen).



4. The section is created, but in the current view it appears as a line.



You can rotate the view to see the section.



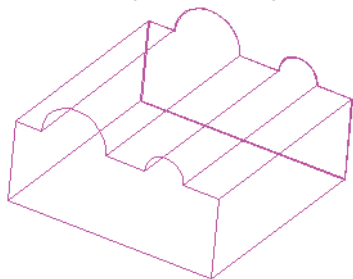
### Section by Plane

Creates a section by defining the sectioning plane. The plane is defined by three points.

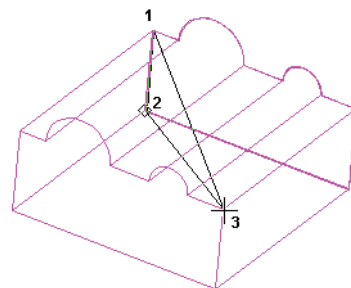
1. Select **Section by Plane (3 Points)** from the local menu or Inspector Bar.



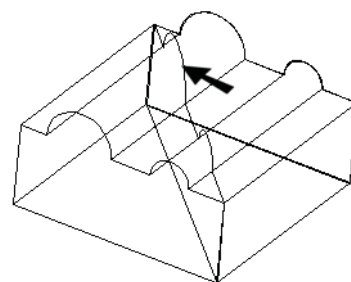
2. Select the object to section. You can use Shift to select more than one object, or use a selection window (first click outside the objects, then drag the window).



3. Select three points to define the sectioning plane.



4. The section is created.



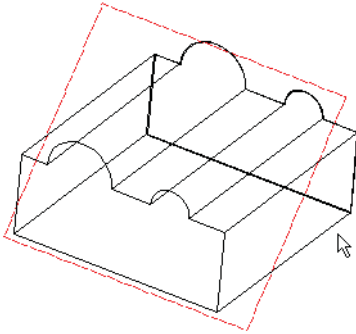
### Section by Workplane

Creates a section by using the current workplane as the sectioning plane. For details on workplanes, see "Workplanes" on page 362.

1. Select **Section by Workplane** from the local menu or Inspector Bar.



2. Select the object to section. You can use Shift to select more than one object, or use a selection window (first click outside the objects, then drag the window).

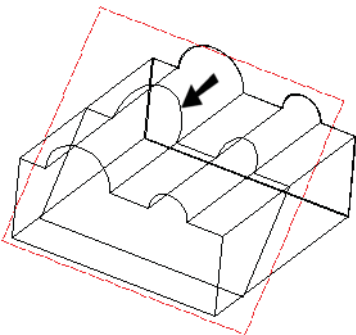



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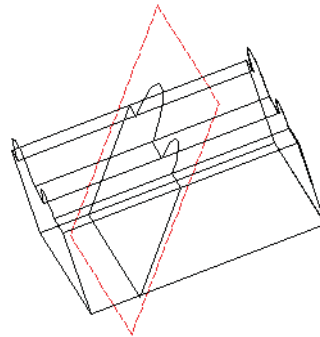
NOTE: To section multiple objects, start out using a different option - **Section by Line** or **Section by Plane**. Use Shift to select the objects to slice, then click the **Section by Workplane** icon.

---

3. The section is created.



You can rotate the view to see that the section lies on the workplane.



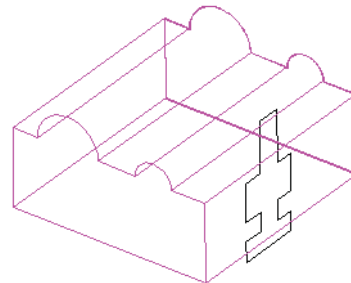
## Section by Closed Polyline

Creates a 3D section by using a closed polyline, rectangle, or polygon to cut through the solid. The section is normal to the plane of the polyline.

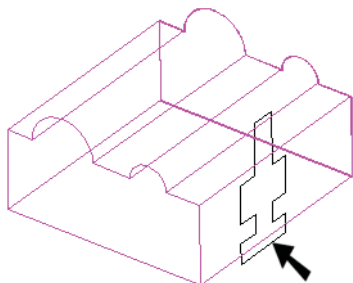
1. Select **Section by Closed Polyline** from the local menu or Inspector Bar.



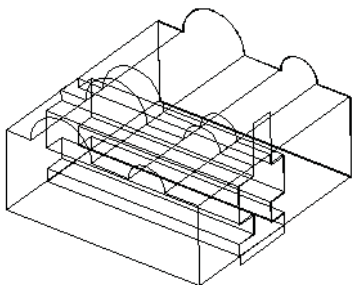
2. Select the object to section. You can use Shift to select more than one object, or use a selection window (first click outside the objects, then drag the window).



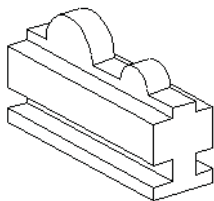
3. Select the closed polyline.



4. The 3D section is created.



You can view the 3D section more clearly by removing the original solid. (The render mode here is **Hidden Line**.)



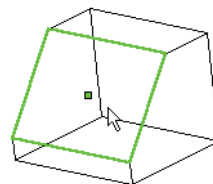
## Facet Edit

*Available in TurboCAD Pro and Platinum only*



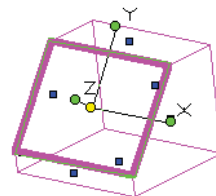
Enables you to move or rotate individual facets of a solid (not surface) object.

1. Select the facet you want to edit. Before it is selected, the facet is highlighted in green and indicated by a green square at its center.

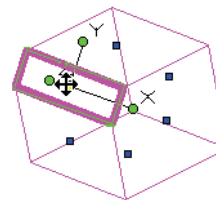


When the facet is selected, all its other facets are indicated by a blue square. You can select another facet for editing at any time while the tool is active.

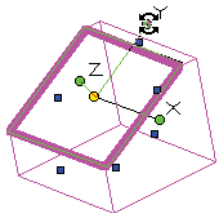
The selected facet appears similar to **Select Edit** mode, with rotation handles for the three axis (relative to the facet plane), and a yellow reference point for moving.



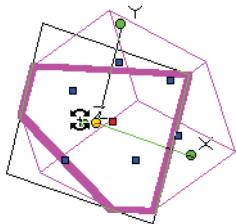
2. Drag the facet by its reference point to move it normal to its current plane.



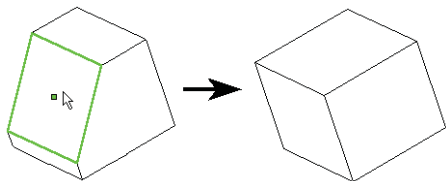
3. You can also rotate about X or Y to change its angle relative to adjacent facets.



4. You can rotate around Z to "tilt" the facet sideways.



5. Lastly, you can delete a node by selecting it and pressing the Delete key. This is possible only when the ACIS object can be reconstructed without the facet.




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NOTE: If you want to delete a facet without reconstructing the object, you can convert the solid to a surface. See "Create Surface from Solid" on page 468.

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#### Local menu options:

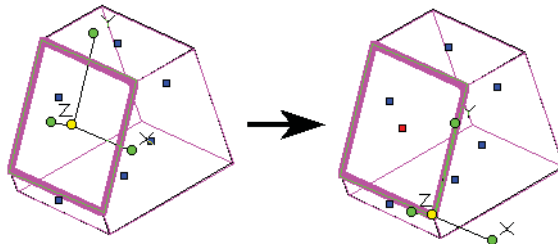
**Properties:** Opens the **Facet Editor Properties**. The options here are the same as for the 3D Selector. See "Selector 3D" on page 183.



**Edit Reference Point:** Moves the reference point of the facet, located by default at the facet center.



Moving the reference point enables you to rotate the facet about different axes.

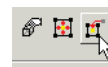



---

NOTE: You can also press the D key to select and move the reference point.

---

**Default Reference Point:** If the reference point was moved, this option moves it back to its default location at the center of the facet.





## Facet Deform

*Available in TurboCAD Pro and Platinum only*

These functions enable you to change the shape of a surface or solid face by applying a uniform pressure, or by specifying deformation points and constraints.

Facet Deform works only with ACIS solids and surface. It does not work on TurboCAD surface objects, objects that have **TC Surface** selected in the **3D** page of the **Properties**. See "3D Properties" on page 371.

---

**WARNING:** **Facet Deform** causes the edited object to be deleted from the Par Tree. See "Editing 3D Objects using Selection Info" on page 487.

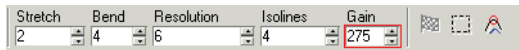
---

## Pressure Load



Enables you to deform a face or surface by applying a uniform pressure over the entire face. The pressure is always normal to the face. The result is a NURBS surface or solid.

1. Activate the **Pressure Load** function, and set the pressure parameters (defined below).



**Stretch:** A factor controlling elasticity - the resistance of the face to stretching. The higher the value the higher the resistance, resulting in less deformation.

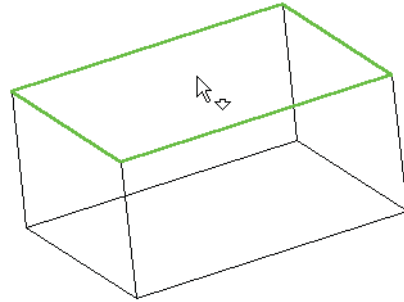
**Bend:** A factor controlling the resistance of the face to bending (flexure) and torsion. Higher values prevent sharp changes to the face.

**Resolution:** The number of internal grids (knots) used to control the deformation. The higher the value, the greater the influence of **Bend** and **Stretch** factors. Lower values, however, yield faster results.

**Isolines:** Number of display lines. This does not affect the results, but is used to better visualize the deformed face.

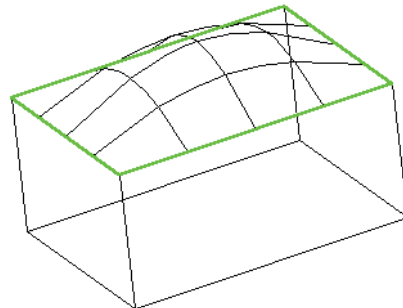
**Gain:** The pressure value. A gain of 1,000 on a 10 x 10 face will have more of an effect than the same gain on a 100 x 100 face. Negative values can be used.

2. Select the face or surface you want to deform. In this example, the pressure will be applied to the top face of a box.

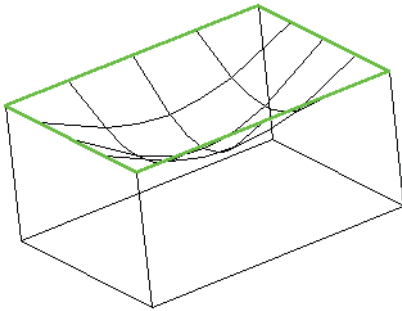


3. A preview of the deformed face will appear, according to the parameters you set. You can adjust the values and press Enter to update the preview. If you need a better representation of the deformation, increase the number of isolines.

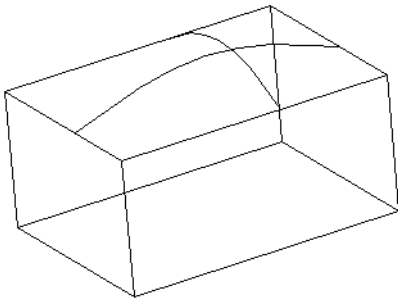
A positive **Gain** value pushes the face up.



If you switch the **Gain** to a negative value, the face will be pushed inward.

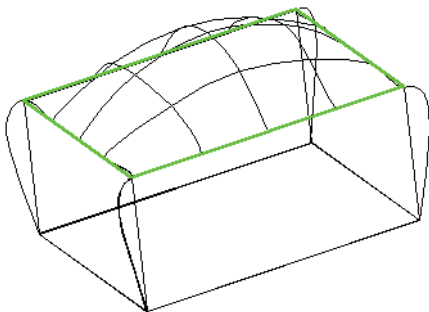


4. When you are satisfied with the deformation, select **Finish** from the local menu or Inspector Bar.

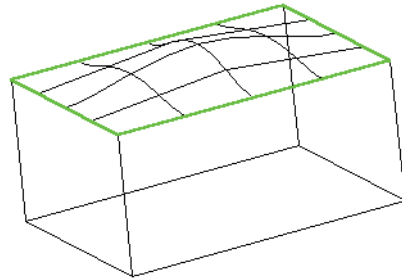
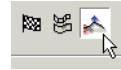


Local menu options:

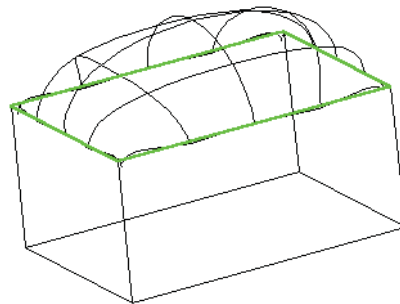
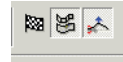
**Linked neighboring faces:** Adjacent faces will also be deformed.



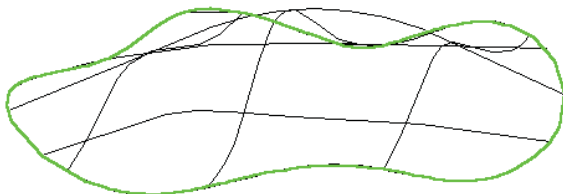
**Fix tangent:** Tangency at the face boundaries will be maintained. In this example, the face will remain horizontal along its boundaries.



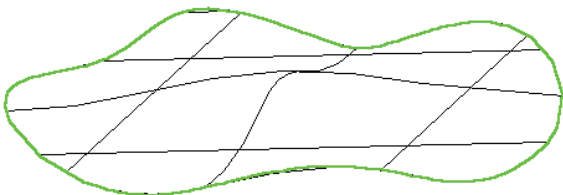
You can combine **both** local menu options to produce a deformation that is tangent to adjacent faces, with minimal deformation of the neighboring faces..



You can also deform a stand-alone surface such as this one, created by using **Create Surface from Profile** on a closed spline. (See "Create Surface from Profile" on page 466.)



In this case, the local menu option **Linked neighboring faces** is not available. If you select **Fix Tangent**, the face remains horizontal along its boundary.

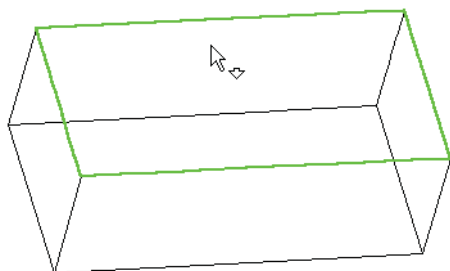


## Deform to Point

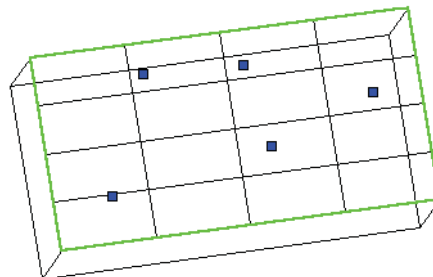


Enables you to deform a face or surface by moving one or more points on the face. The result is a NURBS surface or solid.

1. Activate the **Deform to Point** function and select the face you want to deform - in this case, the top face of the box.

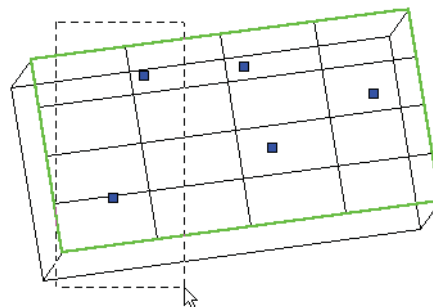


2. The next step is to define deformation points and, if necessary, constraint points. You can click anywhere on the face to create points. If you created points in advance, use the **Vertex** snap to select them.



Any of these points that are not moved will be used as constraints; they will hold the face in place.

3. Now select one or more points that will move. You can select points individually, or use a selection box.

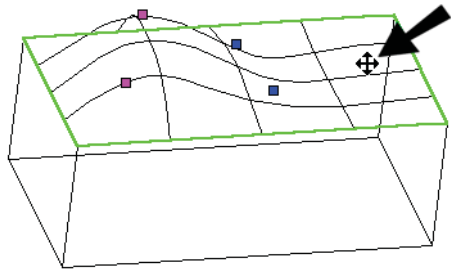


4. Once points are selected, you can move them manually by dragging their reference point, or enter delta values in the Inspector Bar. In this example, the two selected points will move up in the Z direction.

Stretch	Bend	Resolution	Isolines	DeltaX	DeltaY	DeltaZ
2	2	6	4	0 in	0 in	0.35

NOTE: For explanations of the other parameters (**Stretch**, **Bend**, etc.), see "Pressure Load" on page 461.

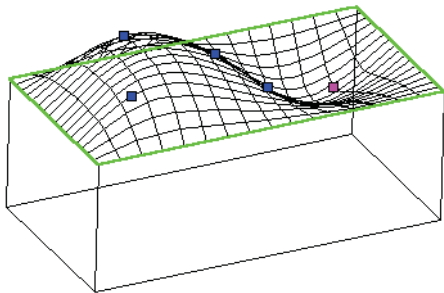
5. Press Enter to move the points, and the face preview is updated. If necessary, you can now move other points. In this case, a single point is selected (rather than using a selection window).



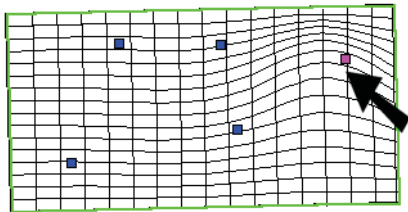
6. This point will move in both Y and Z.

Stretch	Bend	Resolution	Isolines	DeltaX	DeltaY	DeltaZ
2	2	6	4	0	0.15	-0.5

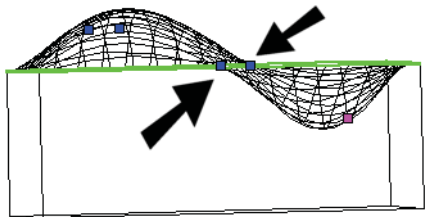
The preview now shows all three moved points. The two points that were not moved are used to hold the face in place. You can increase the number of isolines to better visualize the deformed face.



In this view, you can see the movement in Y of the single point shown.



And in this view, you can see the effect of the constraint points, those not selected to move.



Local menu options:

NOTE: For **Linked neighboring faces** and **Fix Tangent**, see "Pressure Load" on page 461.

**Delete constraint:** Deletes the selected point or points. The deformation or constraint effect on the face will also be removed.



**One Step Back:** Use this if you want to change delta values for the points just moved.



**Move Along Normal:** Moves the points in a direction normal to the surface. In this case, you only need to specify a single **Direction** value.



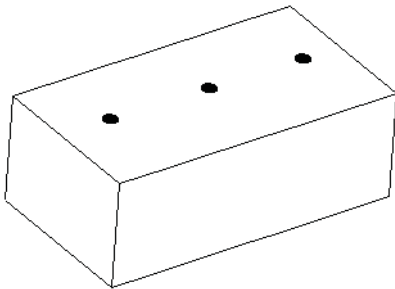
# Hole

Available in TurboCAD Platinum only



Uses a Boolean subtract operation to remove hole volumes from a solid object (or add material, in the case of a **Boss**).

1. Start with a solid object, like a box.
2. Create points for each hole you want to create (see "Point" on page 137). The points must lie on the face from which the hole will be removed.

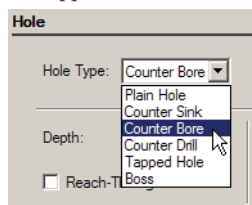


NOTE: As long as **Workplane by Face mode** is checked in **Drawing Aids**, the points will be placed on the facet where your cursor is. (See "Snap Settings" on page 108.) Otherwise, you can use **Workplane by Facet** to place points on the correct facet.

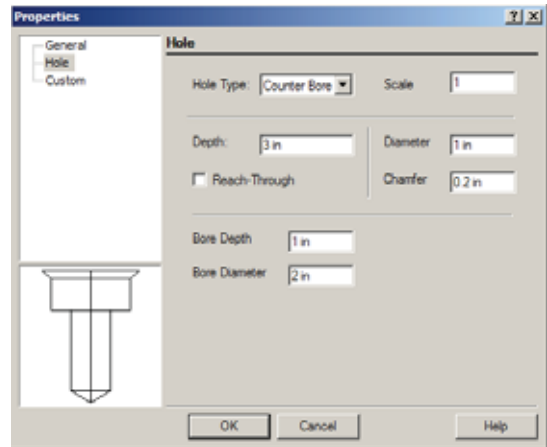
3. Activate **Hole**, and open the **Properties**.



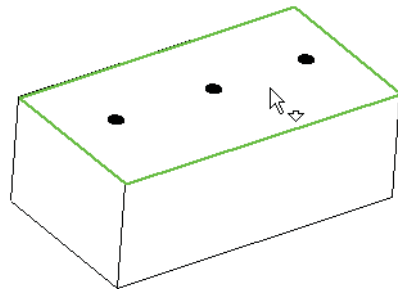
4. On the **Hole** page, select the type of hole. The types of holes are Plain hole, Counter sink, Counter bore, Counter drill, Tapped, and Boss.



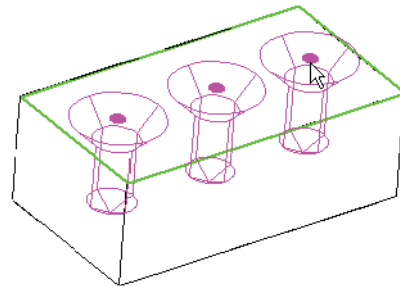
5. The other parameters on this page depend on the hole type.



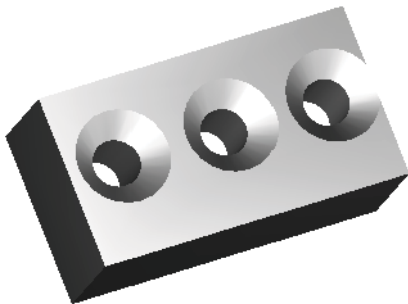
6. Close the **Properties**, and select the face on which the holes will be created.



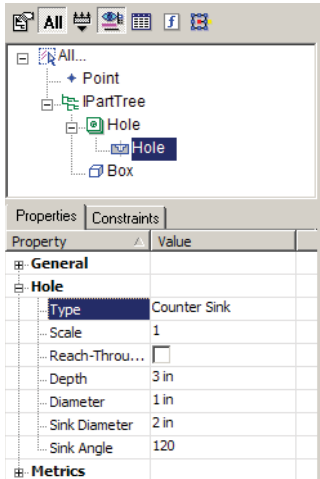
7. Select the points. Use the Shift key to select multiple points. The preview shows what the final result will look like.



- 8. Select **Finish** from the local menu or Inspector Bar, and the holes are created.



You can edit hole parameters after they are created. See "Editing 3D Objects using Selection Info" on page 487.



## Surface and Solid Conversion Operations

*Available in TurboCAD Pro and Platinum only*

In TurboCAD, “surface” refers to a 3D ACIS object, whereas a TurboCAD surface (when **TC Surface** is checked in the **3D** page of an object’s **Properties**) refers to an object created with TurboCAD’s internal engine, which is not an ACIS object. (See "3D Properties" on page 371.)

These are conversion operations that create surfaces or solids from existing objects, including surfaces, faces, and profiles.

You can display the **Solid/Surface** toolbar by right-clicking in any toolbar area and selecting **Solid/Surface**.

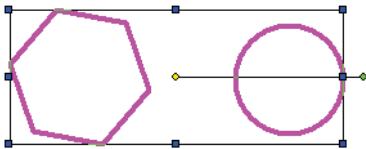


### Create Surface from Profile

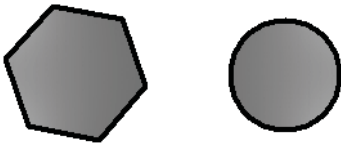


Enables you to create an ACIS surface within a closed profile. The profile can be 2D or 3D.

- 1. Select one or more closed profiles.



- 2. Activate the **Surface from Profile** function, and the boundaries are filled by one or more surfaces. You can see the surfaces in render mode.

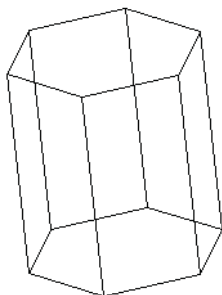


## Create Surface from Face(s)

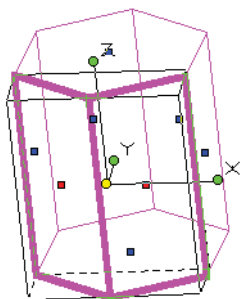


Enables you to create a surface from selected planar facets of solid objects.

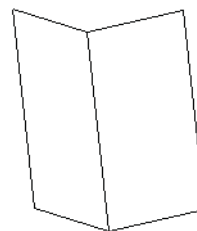
1. Start with a solid that has planar facets. This example uses a 6-sided polygon extruded (using **Simple Extrude**) into a solid.



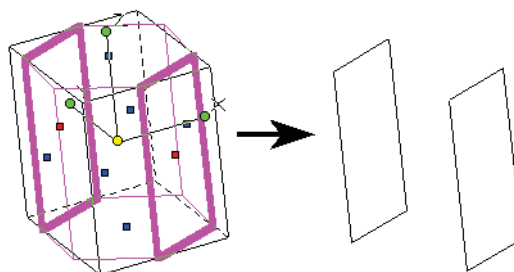
2. Activate **Facet Edit** and select one or more faces. (See "Facet Edit" on page 459.) Use Shift to select multiple faces.



3. Activate the **Surface from Face(s)** function, and the selected faces are now surfaces. The rest of the solid is deleted.



The selected faces do not have to be contiguous; surfaces can be created from any face(s).

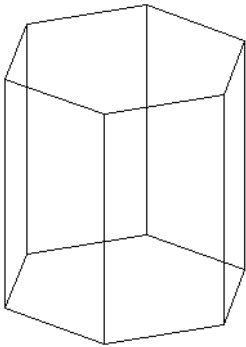


## Create Surface from Solid

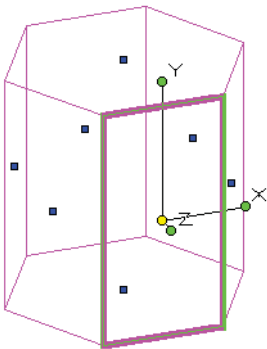


Enables you to convert a solid volume into a hollow form; each solid face is converted into a surface.

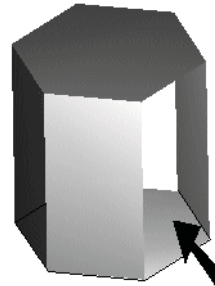
1. Start by selecting one or more solids. This example uses a 6-sided polygon extruded (using **Simple Extrude**) into a solid.



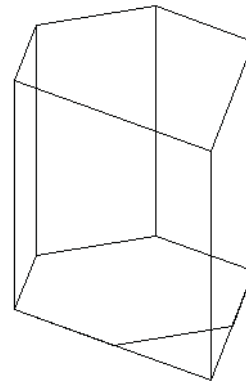
2. Activate the **Surface from Solid** function, and the solid is hollowed out. Each of its facets is now a surface. To more easily see the results, activate **Facet Edit** and select one of the vertical faces. (See "Facet Edit" on page 459.)



3. Delete the face (using the Del key), and view the model in render mode to see the missing facet.

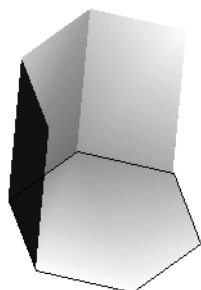


**NOTE:** If you used **Facet Edit** on a solid, without converting it into surfaces, this would be the result - the other faces would update to accommodate the deleted face.

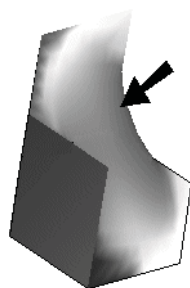




One usage of surfaces is to recreate a solid from the changed volume. If you delete a few more faces . . .



... you can then use **Create Solid from Surface** to add one missing face and produce a new solid. (See "Create Solid from Surface" on page 470.)

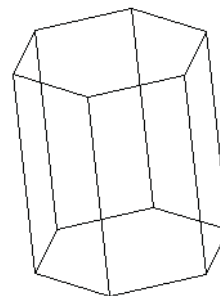


### **Create Solid from Face(s)**

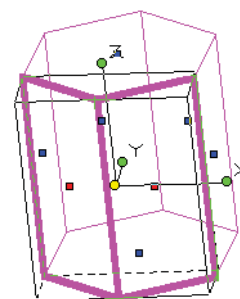


Enables you to convert one or more surfaces / faces into a solid. The faces must be contiguous, but they do not need to share boundaries over the entire boundary length.

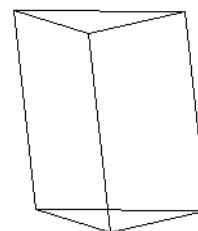
1. Start with one or more solids. This example uses a 6-sided polygon extruded (using **Simple Extrude**) into a solid.



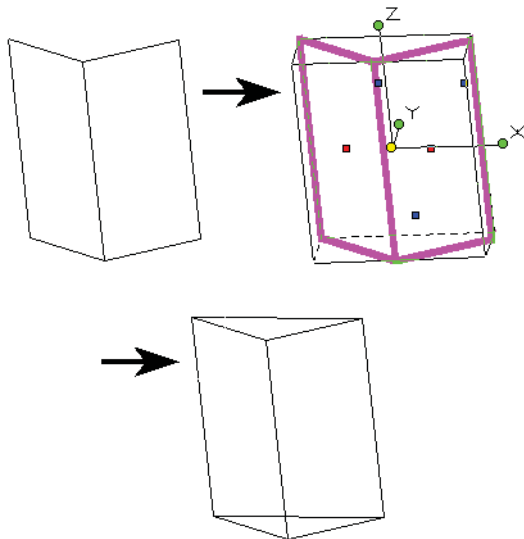
2. Activate **Facet Edit** and select one or more faces. (See "Facet Edit" on page 459.) Use Shift to select multiple faces.



3. Activate the **Solid from Face(s)** function, and the selected faces are now facets of a solid. Faces are added as needed to complete the solid.



You do not have to start with a solid; this function also works with objects that are already surfaces.

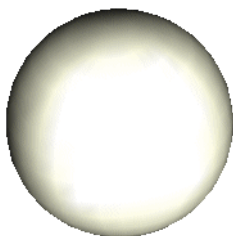


## Create Solid from Surface

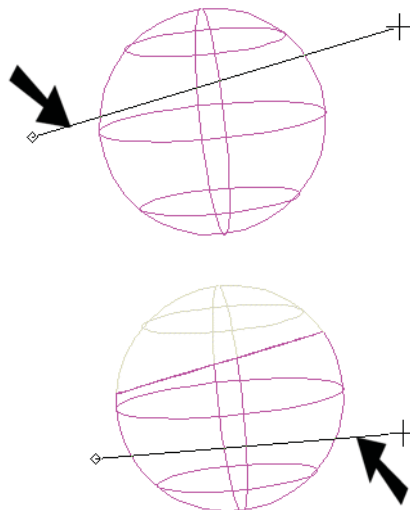


Enables you to convert a surface into a solid volume, within the surfaces' boundaries.

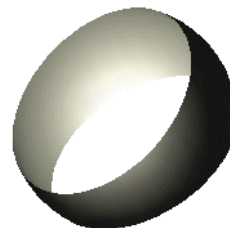
1. Start with a solid or surface. If you have a solid, select it and use **Create Surface From Solid** to convert it into a surface. This example uses a sphere.



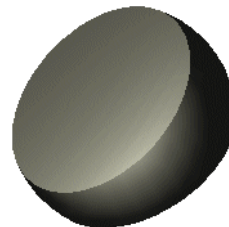
2. Once you have a surface, activate **3D Slice** in order to cut the surface (see "3D Slice" on page 434). Make one or two slice lines.



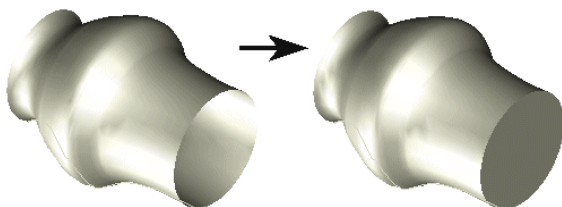
3. Select and delete the sliced portions.



4. Now select what's left of the surface and activate **Create Solid From Surface**. The surface is converted into a solid volume, within the boundaries of the original surface (holes are filled).



As a similar example, create a revolved surface (see "Revolve" on page 399). This is an ACIS object. Simply select it and make it a solid.



## Imprint

Available in TurboCAD Pro and Platinum only



Modifies a facet of a solid (not surface) object by imprinting a closed 2D profile object (polyline, polygon, circle, arc, or curve). Multiple profiles can be imprinted at once. You can also add a fillet to the bottom and/or chamfer to the top.

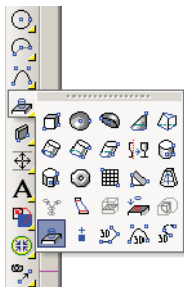
**Imprint** uses Boolean operations of **Add** and **Subtract** on the original solid and the extrusion of the 2D closed profile. See "3D Boolean Operations" on page 432.

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*NOTE: The closed 2D profile object must be located on a facet of the 3D object.*

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Imprint is also available on the fly-out toolbar from the **Drawing Tools**.




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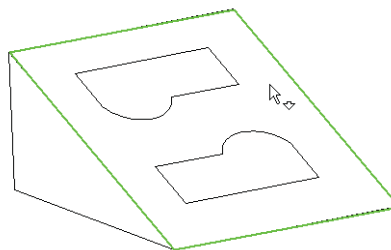
*NOTE: You can use text as part of the compound profile for imprinting.*

---

## Imprint Auto Detect

Adds or subtracts the extrusion of the closed profile to/from the solid, depending on which direction the profile is extruded.

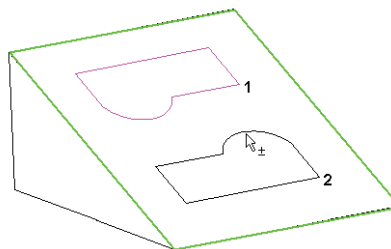
1. Start with a solid object with one or more profiles on one facet. Activate **Imprint** and select the facet that will be imprinted.



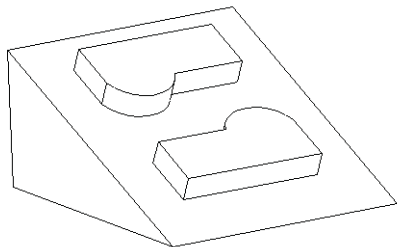
1. Make sure **Auto Detect** is selected. In this example, the last two options in the Inspector Bar have no blend (fillet or chamfer).



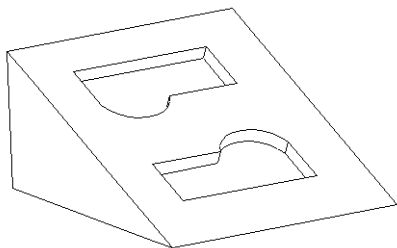
2. Select the closed profile. If you want to select more than one profile, press the Shift key.



3. Pull the profiles away from the solid, and click to create the imprint, or enter a **Height** in the Inspector Bar. Because the extrusion direction was outward, the extrusions were added to the solid.



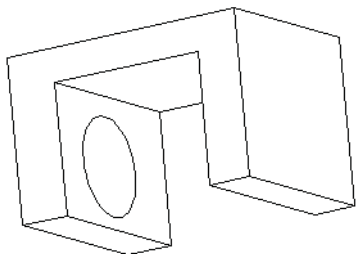
4. If you **Undo** and recreate the imprint, pushing into the solid this time, the extrusions will be subtracted from the solid.



## ***Imprint Add***

Adds the extrusion of the closed profile to the solid, regardless of the extrusion direction.

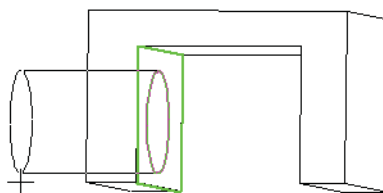
1. Start with a solid like this, with a closed profile within one of the inside facets.



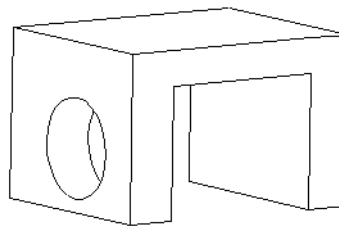
2. Activate **Imprint**. For the sake of comparison, first select **Auto Detect**. In this example, the last two options in the Inspector Bar have no blend (fillet or chamfer).



3. Select the inner face that contains the profile, then select the profile itself. Extrude the profile this way:



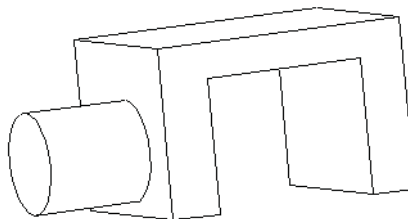
This is the result: because the profile was extruded inward with respect to its face, its extrusion was subtracted from the solid.



4. Undo, and now select **Add**.



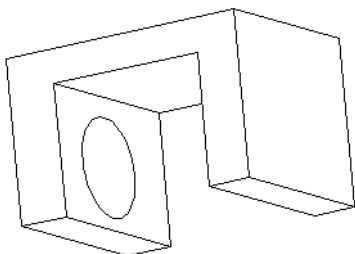
5. Select and extrude the profile the same way, and now its extrusion is added to the solid.



## Imprint Subtract

Subtracts the extrusion of the closed profile from the solid, regardless of the extrusion direction.

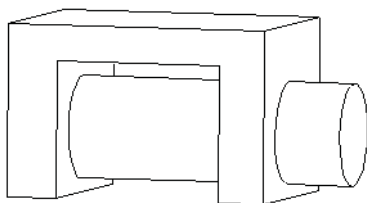
1. Start with a solid like this, with a closed profile along one of the inside facets.



2. Activate **Imprint**. For the sake of comparison, first select **Auto Detect**. In this example, the last two options in the Inspector Bar have no blend (fillet or chamfer).



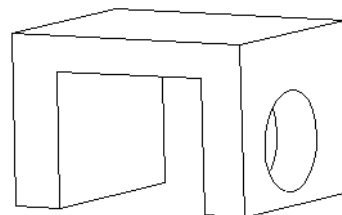
3. Select the inner face that contains the profile, then select the profile itself. Extrude the profile this way. Because the profile was extruded outward with respect to its face, its extrusion was added to the solid.



4. Undo, and now select **Subtract**.



5. Select and extrude the profile the same way, and now its extrusion is subtracted from the solid.



## Imprint with Chamfer or Fillet

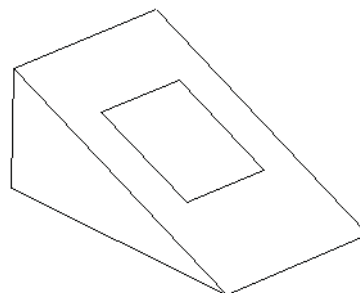
You can add a chamfer or fillet to the top or bottom of the imprint.

---

NOTE: For details on these edge tools, see "Fillet Edges" on page 437 and "Chamfer Edges" on page 442.

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1. Start with a solid object with one or more profiles on one facet. Activate **Imprint** and select the facet that will be imprinted.



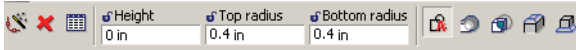
2. In this example, **Auto Detect** is selected. When the last two options are set to **Normal Top** and **Normal bottom**, the imprint edges will be sharp.



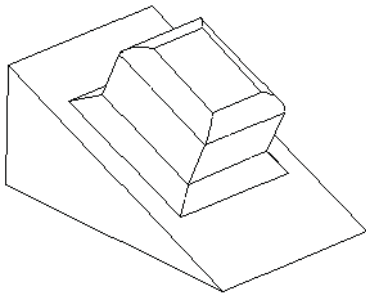
3. To fillet or chamfer, click the option icons to set them to **Fillet Top / Bottom** or **Chamfer Top / Bottom**. Clicking repeatedly will scroll through the options. In this example, the bottom has a chamfer, and the top has a fillet.



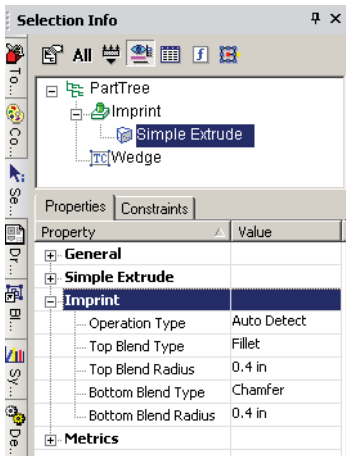
4. Set the **Top radius** and **Bottom radius**.



5. Select and extrude the profile. The top is filleted and the bottom is chamfered.



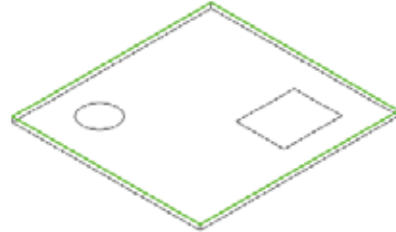
6. You can edit all parameters of the imprint, including changing or removing fillets and chamfers, in Selection Info palette.



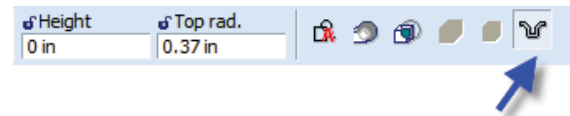
## Imprint with Dimple Sheet Metal

You can use the imprint tool to create a dimple effect as in sheet metal.

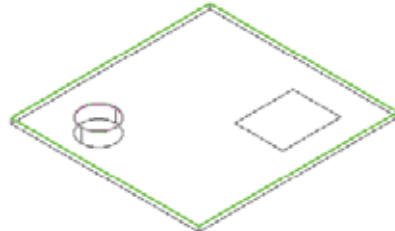
1. Start with a solid object with one or more profiles on one facet. Activate **Imprint** and select the facet that will be imprinted.



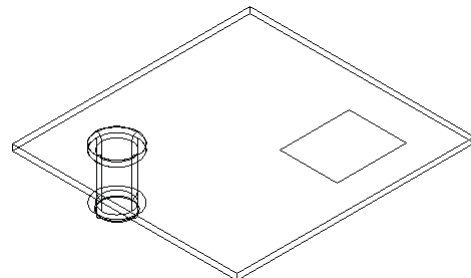
2. Select the **Dimple sheet metal** option.



3. Select a closed profile.



The height must be a negative value greater than the sheet thickness. The top radius must be equal to or greater than the thickness of the sheet.

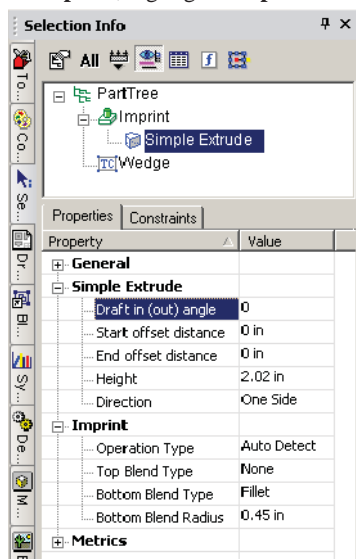


## Editing Imprint Parameters

The Selection Info palette can be used to edit parameters of the imprint itself, as well as the fillets and/or chamfers.

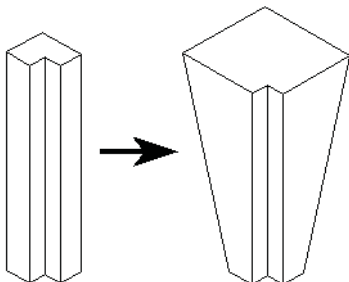
NOTE: For general information on this palette, see "Selection Info Palette" on page 191.

Select the imprint, and its **Part Tree** appears at the top of the palette. Under **Imprint**, highlight **Simple Extrude**.



At the bottom of the palette, the **Simple Extrude** category contains parameters for the imprint body:

- **Draft Angle:** Creates an extrusion of increasing or decreasing cross-section. Enter the angle of deviation from the extrusion path.

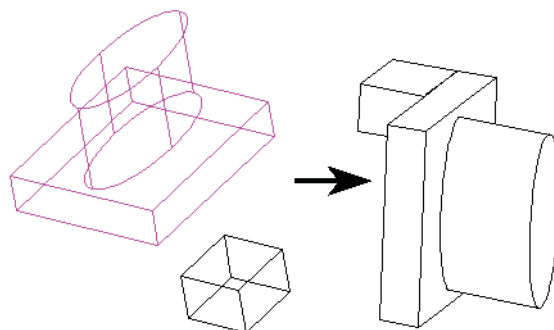


- **Draft Start / End Distance:** If **Draft Angle** = 0, you can specify a draft angle by entering the offset distances.
- **Height:** The distance of the extrusion.
- **Direction:** Switch between one-sided and two-sided.

The **Imprint** category is where you can change the imprint type (**Auto Detect**, **Add**, or **Subtract**) and change or remove fillets and chamfers and change their radii.

## Assembling

The **Assemble** tools are used to position a 2D or 3D object relative to another object.



The **Assemble** tools can be used on 2D and 3D objects. For the objects you want to use, be sure the Selector is set correctly (2D, 3D, or both). See "2D / 3D Selector" on page 181.

These tools are available on the **3D Modify** toolbar, which you can display by right-clicking in any toolbar area and selecting **3D Modify**.



NOTE: The **Transform** tools also enable you to move objects, as well as scale and copy, and do not require information as accurate as that required by the **Assemble** tools. In addition, you can record **Transform** operations for use on other objects. See "Transforming" on page 212.

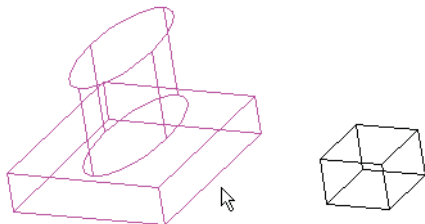
## Assemble by 3 Points



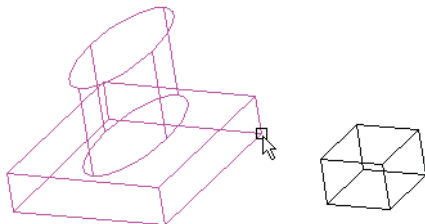
Changes the position of an object by changing the location of one point, two points (line), or three points (plane). The source points typically lie on the object to be moved, although this is not required. Destination points can lie on destination objects, or they can be specified in the Coordinate Fields.

### To assemble by one point:

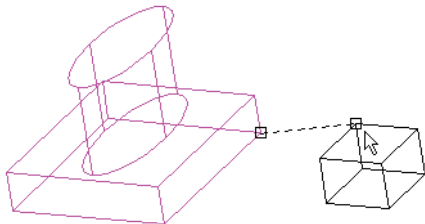
1. Select the object to be repositioned.



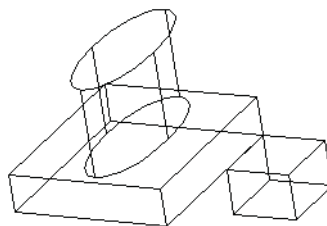
2. Select the source point.



3. Select the destination point.

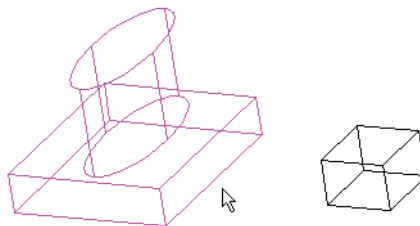


4. Select **Finish** from the local menu or Inspector Bar. The object is moved so that its source point meets the destination point, keeping its orientation.

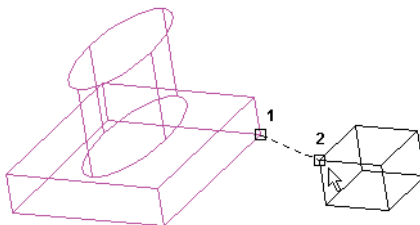


### To assemble by two points (line):

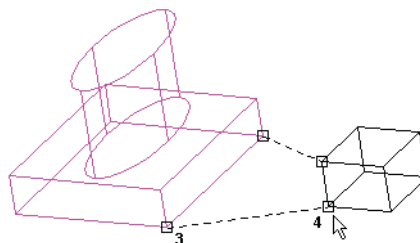
1. Select the object to be repositioned.



2. Select the first source point and its destination. This first set of points determines the actual object position; the remaining points set the orientation.

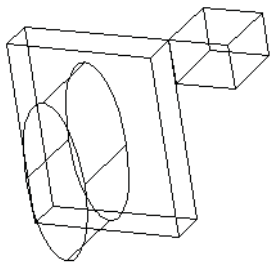


3. Select the second source point and its destination.





4. Select **Finish** from the local menu or Inspector Bar. The object is moved so that its first source point meets the first destination point, and the line between source points becomes aligned with the line between destination points.



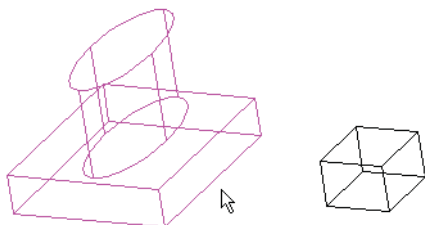

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NOTE: For another way to assemble by edges, see "Assemble by Edge and Point" on page 477.

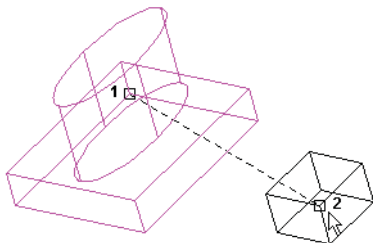
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#### To assemble by three points (plane):

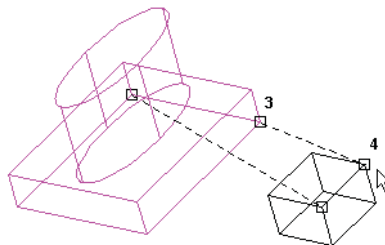
1. Select the object to be repositioned.



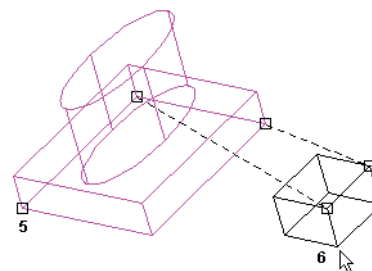
2. Select the first source point and its destination. This first set of points determines the actual object position; the remaining points set the orientation.



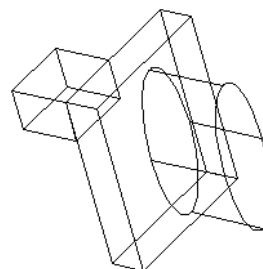
3. Select the second source point and its destination.



4. Select the third source point and its destination.



The object is moved so that its first source point meets the first destination point, and the plane defined by the source points becomes aligned with the plane defined by the destination points.



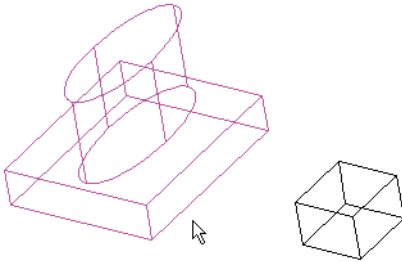
### Assemble by Edge and Point



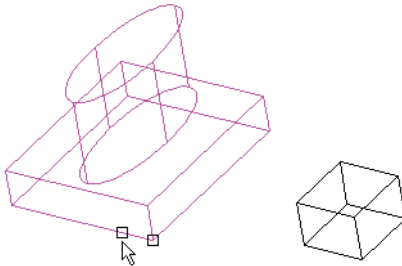
Changes the position of an object by changing the location and alignment of an edge. You can also add a source point and destination point to change the rotation as well.

**To assemble by a pair of edges:**

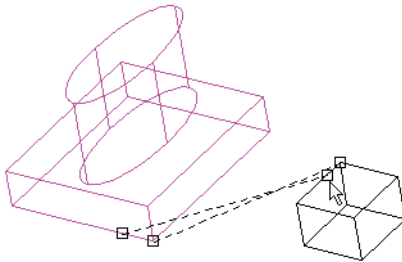
1. Select the object to be repositioned.



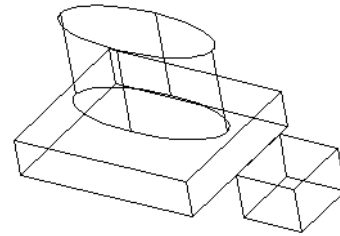
2. Select the source edge. The new alignment of the edge depends on where you select the edge; the point you select will be moved to the point you select on the destination edge.



3. Select the destination edge. The dotted lines indicate how the object will be aligned.

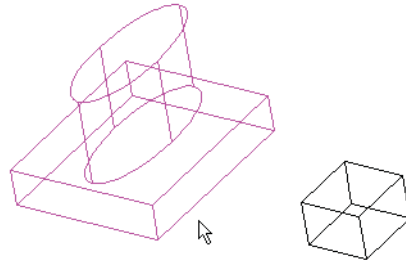


4. Select **Finish** from the local menu or Inspector Bar. The object is moved so that its source edge meets the destination edge, connected at the two selected points.

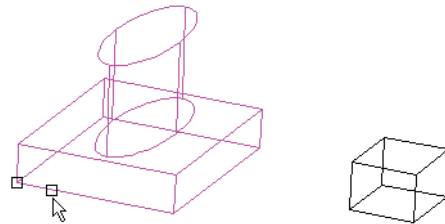


#### To assemble by edges and a plane:

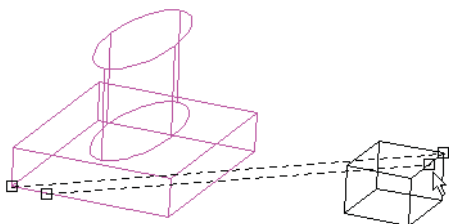
1. Select the object to be repositioned.



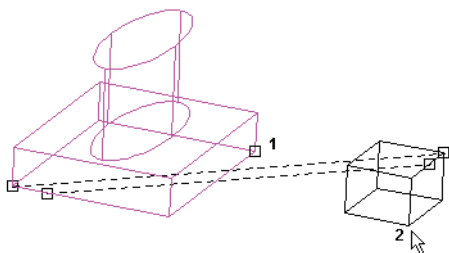
2. Select the source edge. The new alignment of the edge depends on where you select the edge; the point you select will be moved to the point you select on the destination edge.



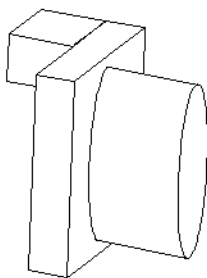
3. Select the destination edge. The dotted lines indicate how the object will be aligned.



4. Select a point on the source plane (not on the selected edge) and a point on the destination plane.



The object is moved so that its source edge meets the destination edge, connected at the two selected points. The rotation is set by the points on the source and destination planes. The results are shown here in Hidden Line render mode.



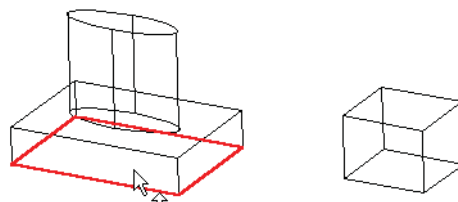
## Assemble by Facet

*Available in TurboCAD Pro and Platinum only*

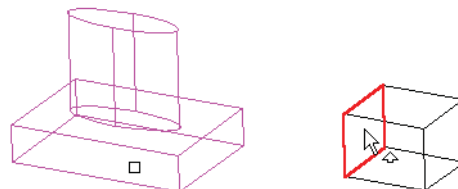


Changes the position of an object by aligning facets.

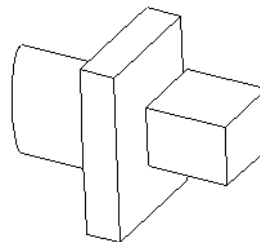
1. Select the source facet of the object to be repositioned. To select a facet behind or in front of the indicated facet, you can use the Page Up and Page Down keys.



2. Select the destination facet.



The object is moved so that the source facet meets the destination facet. The results are shown here in **Hidden Line** render mode.



## Assemble by Axis

Available in TurboCAD Pro and Platinum only

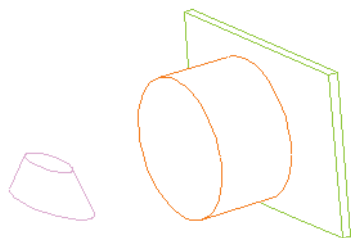


Changes the position of an object by aligning axes.

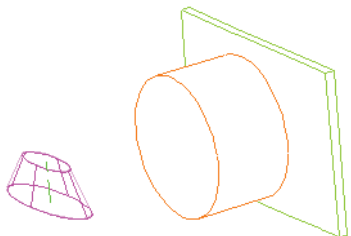
### Assembling Circular or Rotated Objects

For round objects that have a rotational axis (such as **Cylinder**, **Cone**, **Polygonal Prism**, **Revolve**, or **Simple Extrude** based on a circular 2D object), you need to set the assembly axis before the objects can be assembled.

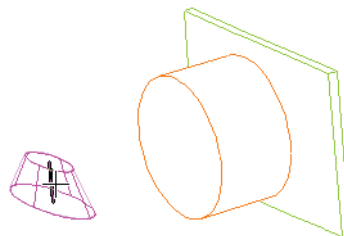
This example will assemble a truncated cone based on an ellipse, aligning it with the axis of a cylinder.



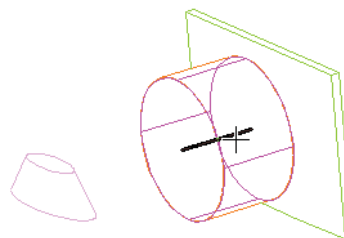
1. Activate **Set Assembly Axis**.
2. Click one of the round objects (the truncated cone, in this case). Its axis is displayed as a dashed line.



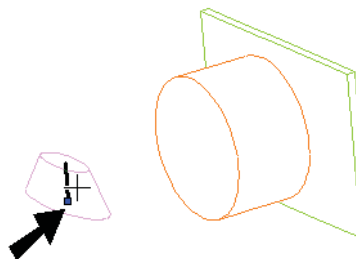
3. Click the axis to define it for the assembly.



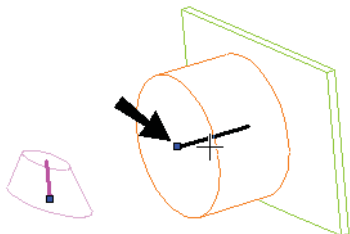
4. Select **Finish** from the Inspector Bar or local menu.
5. Do the same to define the axis for the other round object (the cylinder).



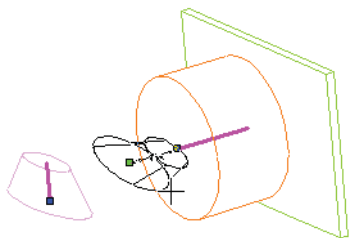
6. Select **Finish** from the Inspector Bar or local menu.
7. Activate **Assemble by Axis**. Click the axis of the object you want to move. Be sure to select the axis near the endpoint that will meet the endpoint of the other axis.



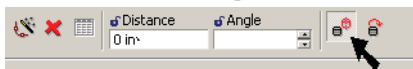
8. Click the destination axis, near the endpoint where the other axis will meet it.



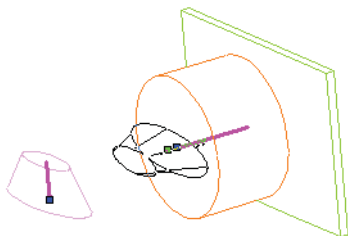
A preview of the moved object appears. You can move it in either direction along the axis.



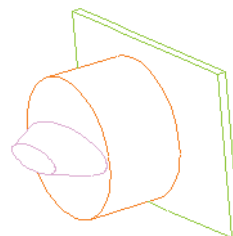
9. Use the mouse or enter the move distance. If you want to rotate the object, you can enter an Angle. If the moved object is oriented backward, select **Turn Over** from the local menu or Inspector Bar.



10. To rotate it with the mouse, select **Rotate**.  
If you turned or rotated the object, the preview will update.



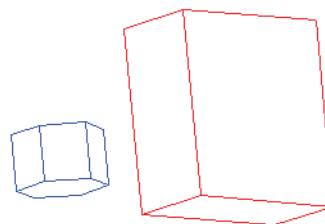
11. When the distance and angle are defined, the object moves so that the two axes are aligned.



### Assembling Planar or Linear Objects

For objects that have no assembly axis defined, you can define the axis during the assembly process.

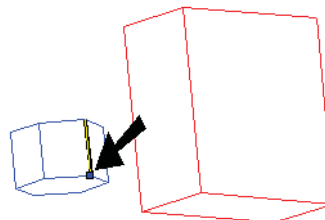
This example uses a polygonal prism and a box.



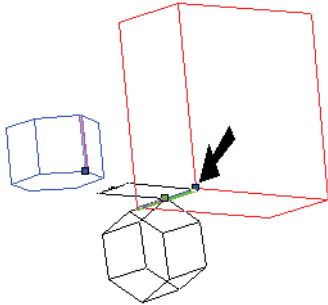
1. Activate **Assemble By Axis**, and select **Use Any Line** from the local menu or Inspector Bar.



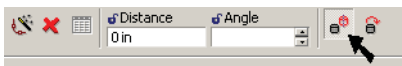
2. Select the object you want to move. Then click a line or edge you want to use as the assembly axis. Be sure to click near the endpoint of the axis which will meet the other axis.



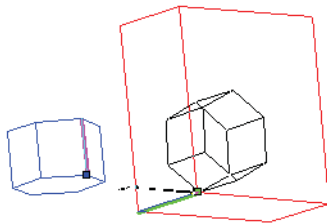
- Click an edge or line as the destination axis. Select it near the endpoint that will meet the other axis. A preview of the moved object appears. You can move it in either direction along the axis.



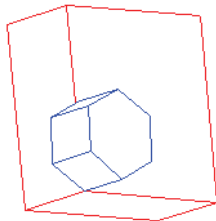
- Use the mouse or enter the move distance. If you want to rotate the object, you can enter an Angle. If the moved object is oriented backward, select **Turn Over** from the local menu or Inspector Bar.



- To rotate it with the mouse, select **Rotate**. If you turned or rotated the object, the preview will update.



- When the distance and angle are defined, the object moves so that the two axes are aligned.



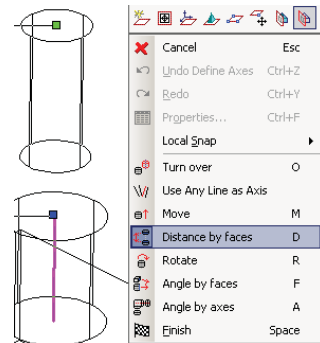
## Distance by faces, Angle by faces and Angle by axes

Make sure to create sufficient axes to setup for these modes.

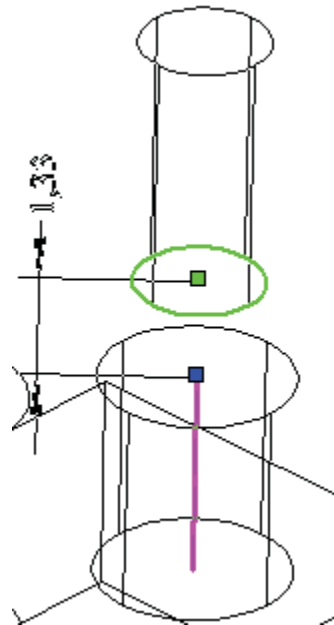
Make two axes of two bodies coincident, then select one of modes by using hot keys or local menu

### Distance by Faces

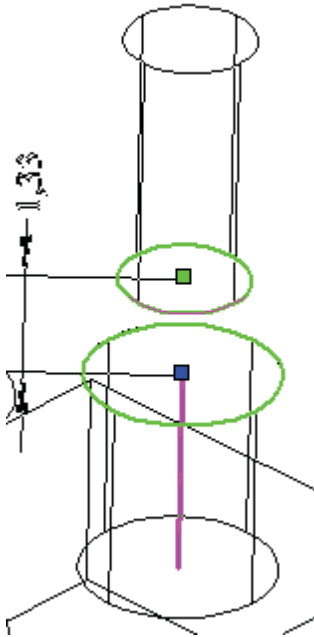
Select the Distance by faces mode.



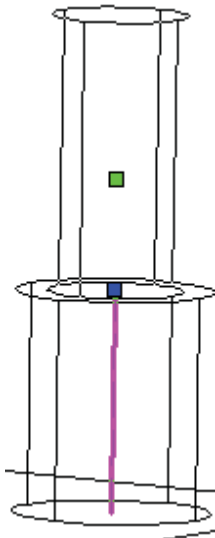
Define the first face on the body which will be moved.



Define destination face

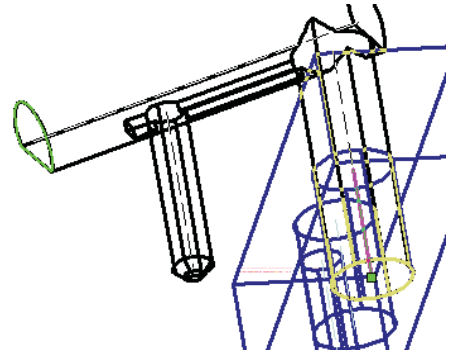


The body will be moved by the distance between selected faces along the common axis and two faces will become coincident.

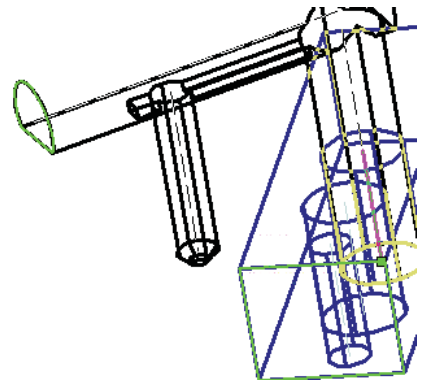


### Angle by Faces

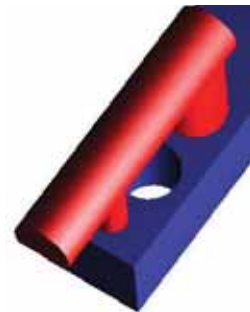
Select face on body you want to rotate.



Select the second face (on any body) you wish the first face to be parallel to.



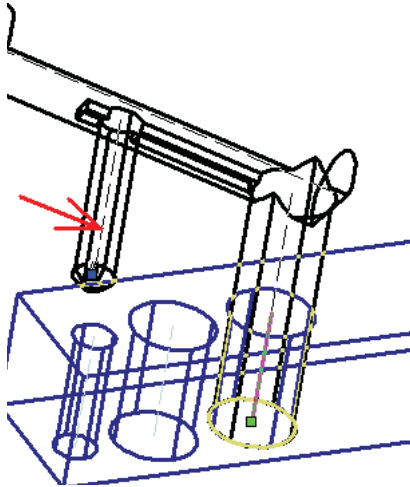
If the rotation around the common axis is available, then the normals of the two defined faces will become coincident/parallel.



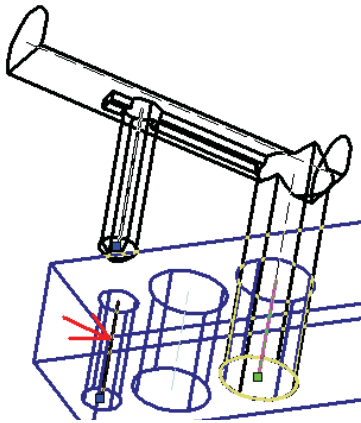
## Angle by Axes

Mode 1:

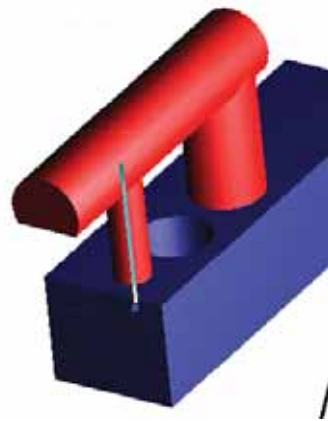
Select the second axis on first body; the one you are moving.



Select the second axis on the body you are aligning to.

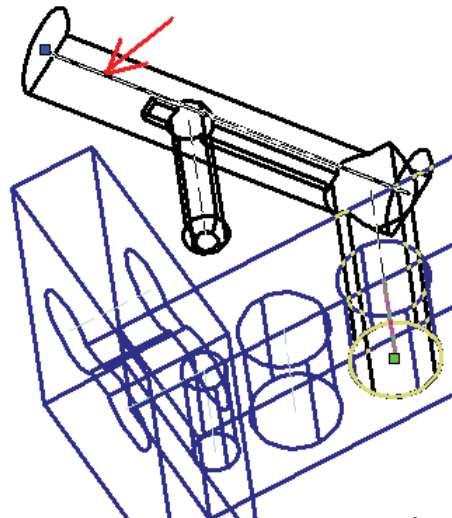


If the rotation around the common axis is available, the directions of selected axes will become coincident.



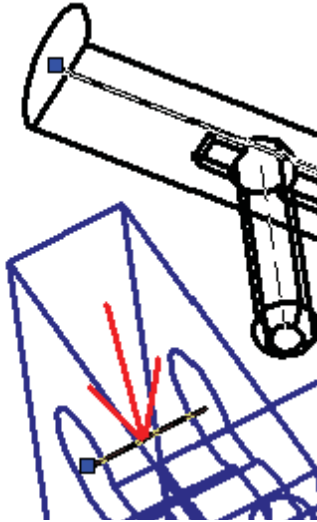
Mode 2:

Select the second axis on first body; the one you are moving.

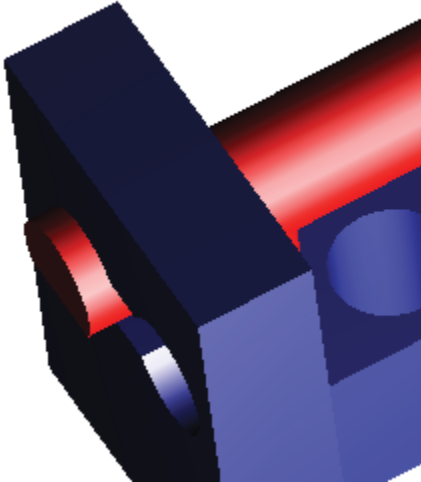




Select the second axis on the body you are aligning to.



If the rotation around the common axis is available, the directions of selected axes will become coincident.



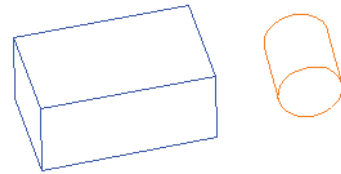
## Assemble by Tangents

*Available in TurboCAD Pro and Platinum only*



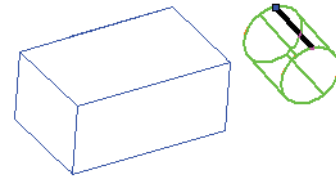
Changes the position of an object by aligning a cylindrical face tangent to another cylindrical face, or to a flat face.

This example will assemble a cylinder and a box.

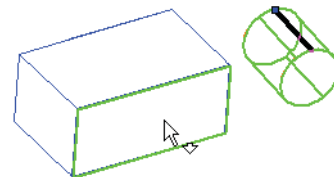


### Aligning to a Face

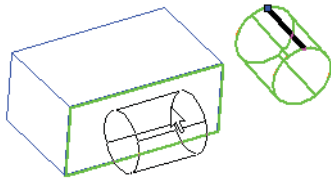
1. Activate **Assemble by Tangents**. Select the object that will move (the cylinder). Be sure to select the axis where you want the object to be placed on the second object.



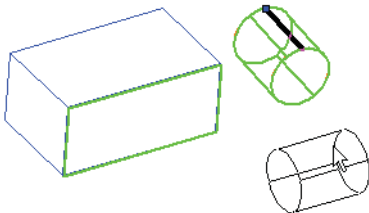
2. To align the object to a face, and not to a specific axis or point, click the face of the second object. This face can be planar or cylindrical. Where you click defines the initial location of the first object.



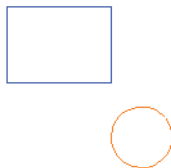
A preview of the moved object appears.



3. You can move the preview anywhere, and it remains tangent to the face.

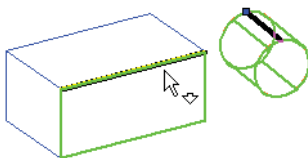


4. When you place the object, and view it in **Side** view, you can see that the cylinder is tangent to the face of the box.

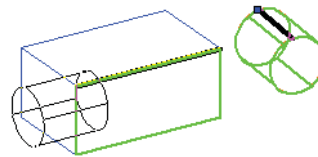


## Aligning to a Face and Edge

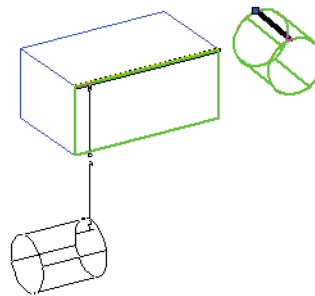
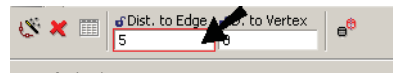
1. You can align a cylindrical face to both a face and edge. When selecting the second face, click on one of its edges.



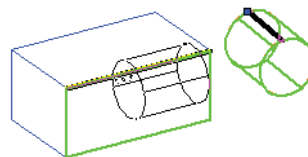
The preview shows the cylinder aligned to both the face and edge.



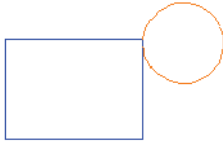
2. A **Distance to Edge** value will move the cylinder away from the edge, in an offset direction while staying tangent to the face.



3. A **Distance to Vertex** value will move the cylinder along the edge.



- When you place the object, and view it in **Side** view, you can see that the cylinder is tangent to the face and edge of the box.



## Editing 3D Objects using Selection Info

Available in TurboCAD Pro and Platinum only

NOTE: For general information on this palette, see "Selection Info Palette" on page 191.

You can use the Selection Info palette to edit properties and parameters of 3D objects, taking into account the order in which the objects were created. Objects must be created as solids; surface objects cannot be edited this way (see "3D Properties" on page 371).

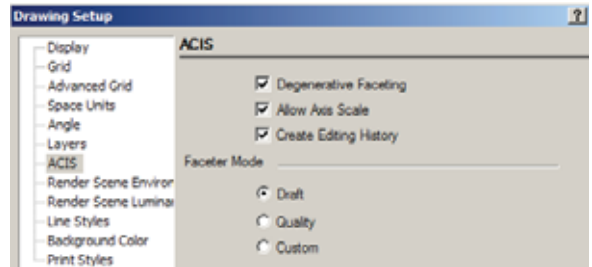
In addition to seeing what objects were used in which operations, you can also edit certain properties of these operations:

- **Fillet Edges** and **Chamfer Edges**: add or subtract affected edges, modify the chamfer length or fillet radius.
- **Boolean operations**: Change the size or location of the original objects combined in the Boolean operation.
- **Shell**: Change the open faces, set inward or outward shelling, modify the shell thickness.
- **Imprint**: Change the type of imprint, add a draft angle, and change the fillets and chamfers at the edges. See "Editing Imprint Parameters" on page 475.

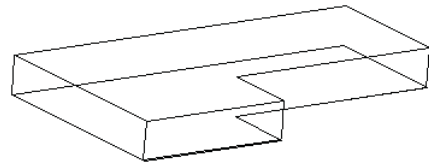
After you make a change to any operation, all subsequent operations will update to reflect the change.

WARNING: **Facet Deform** operations cause the edited objects to be deleted from the editing history.

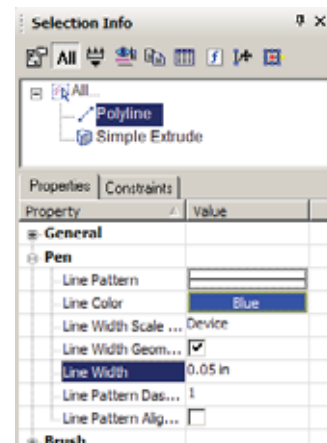
In order for object history to be recorded, you must check **Create Editing History** in the ACIS page of **Drawing Setup (Options / ACIS)**.



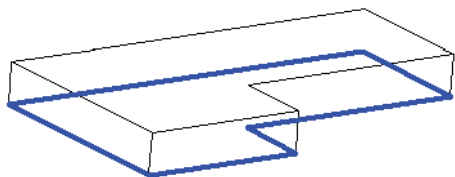
- This example started with a Simple Extrude, based on a linear polyline.



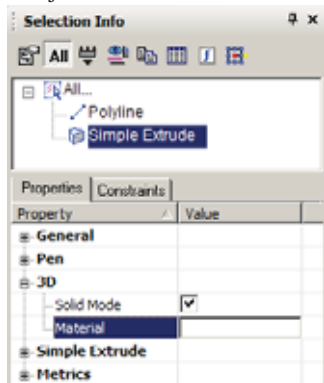
- Highlight Polyline and change its color and line weight.



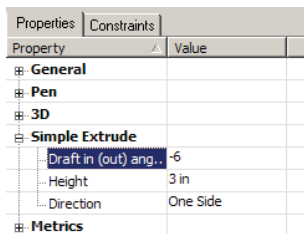
Now the polyline is easily differentiated from the solid.



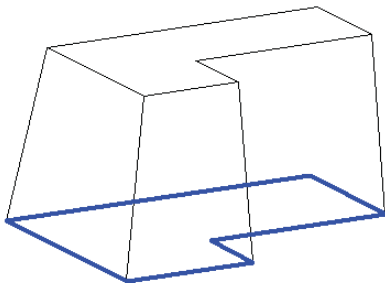
For the Simple Extrude, you can edit all parameters that are available in its **Properties** window. For example, the **3D** category has an option to change the object into a surface object (uncheck **Solid Mode**), and to set the object's **Material**.



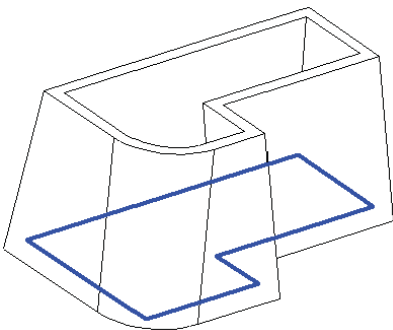
3. Under **Simple Extrude**, increase the **Height** and add a slight **Draft Angle**.



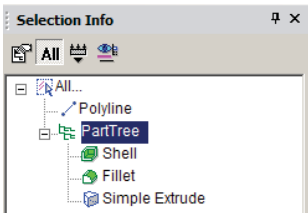
The solid is now higher with drafted walls.



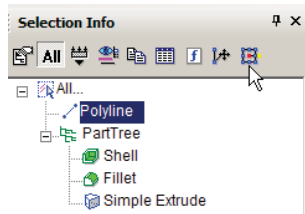
4. Use **Fillet Edges** to round one edge (see "Fillet Edges" on page 437.) Then use **Shell Solid** to shell this shape, leaving the top face open (see "Shelling Solids" on page 453.)



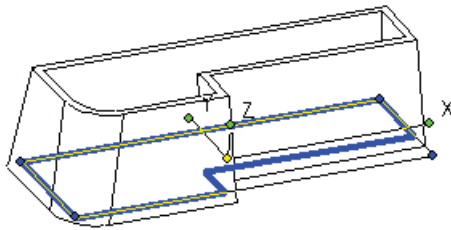
In Selection Info, the Extrude, Fillet and Shell operations are listed under "Part Tree."



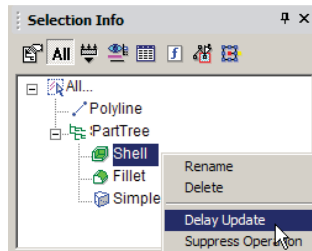
5. The Simple Extrude is based on the polyline, and changing the polyline will update the solid. Highlight Polyline and click **Select**.



6. Scale the polyline in Select Edit mode to make it longer. The solid shape, as well as the fillet and shelling update automatically.

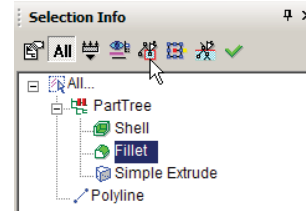


7. Subsequent operations do not have to update automatically. To disable this, right-click on “Shell” and select **Delay Update**.

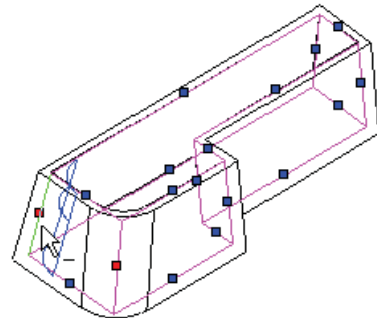


**NOTE:** If you click **Suppress Operation**, the operation will be removed from part history, though its item will still remain in the list. You can bring the operation back into the model by selecting **Unsuppress Operation**.

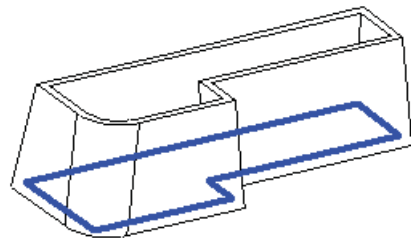
8. To edit the fillet, highlight it and select **Edit in Place**.



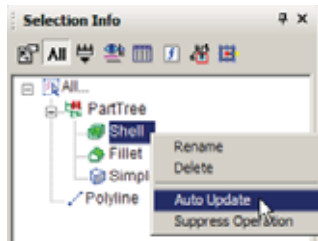
9. **Edit in Place** enables you to return to the original operation and make changes. Click another edge for filleting. You could also change parameters such as the fillet radius.



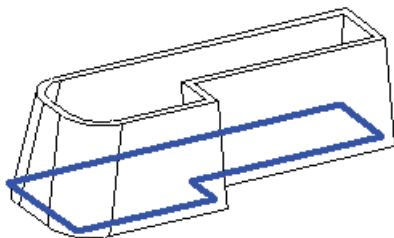
10. Select **Finish** to update the operation. The shell has not updated.



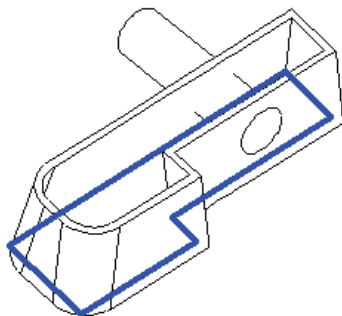
11. To update the shell, right-click t and select **Auto Update**.



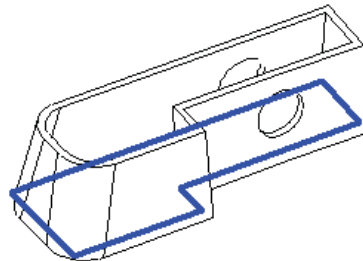
Now the shell reflects the new filleted edge.



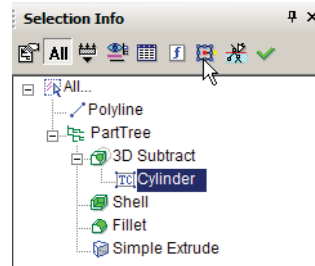
12. For the next operation, create a cylinder, with its workplane located on one of the vertical faces.



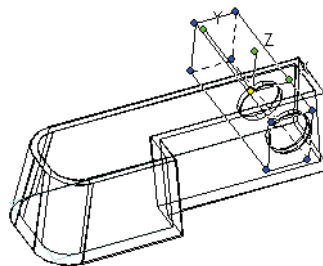
13. Then use the Boolean operation **3D Subtract** to remove the cylinder from the shelled solid. (See "3D Subtract" on page 433.)



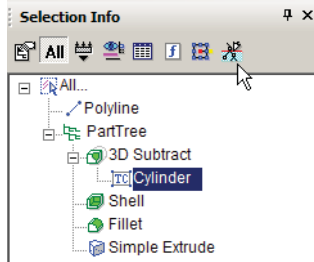
14. The **3D Subtract** operation is now listed. If you expand it, you can see the **Cylinder** on which it is based. Highlight the cylinder, and click **Select**.



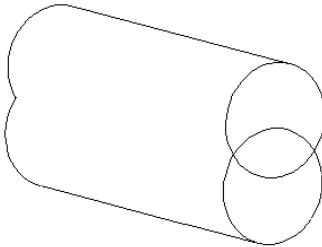
15. Move the cylinder, which in turn updates the 3D Subtract operation.



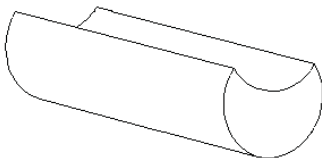
16. For another way to edit the cylinder, select **Edit Content**.



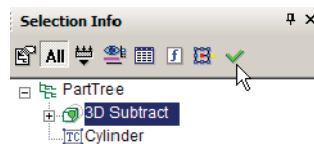
17. The cylinder is now the only object in the drawing area, and any changes will modify the operations currently based on the cylinder. Make a copy of the cylinder so that they overlap.



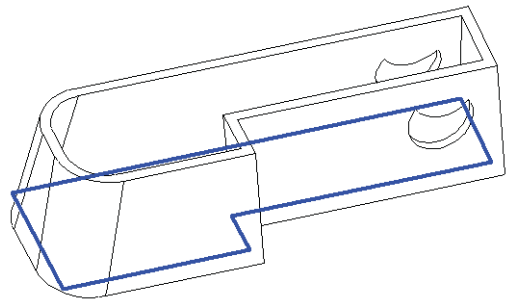
18. Then use **3D Subtract** to get this shape:



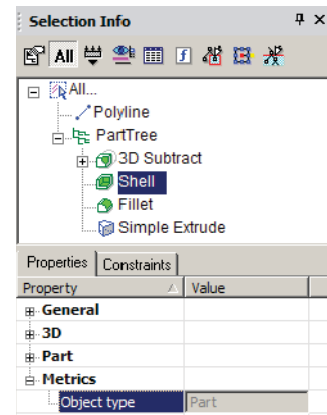
19. When finished, click **Finish Editing Content**.



The entire solid is updated, based on the modified cylinder.



20. For one of the solid operations, open **Metrics**. It only informs you that the object is a solid, as opposed to a surface.



21. Click **Show Physical Metrics**.



This displays properties such as volume, area, center of gravity, and moments of inertia.

Properties		Constraints
Property	Value	
General		
3D		
Part		
Metrics		
Object type	Part	
Solid-state Physi...		
Volume	30.142	
Surface Area	248.040	
Center of Gra...	x=7.41 in y=5.65 i	
x	7.41 in	
y	5.65 in	
z	0.99 in	
Moments of I...		

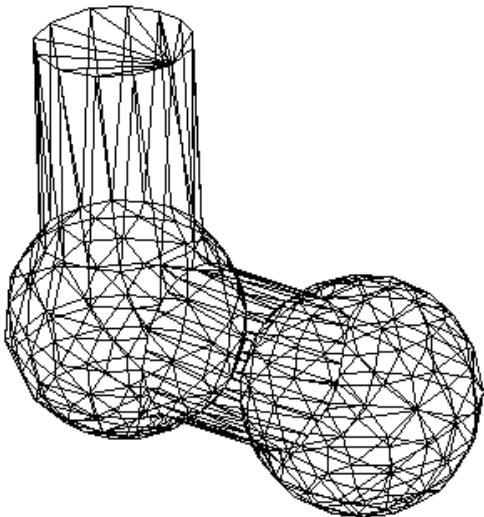
TC Mesh Simplification

Available in TurboCAD Pro and Platinum only



Simplifies meshes by reducing the total polygon count. For example you can use it for reducing the number of the polygons in the laser scanned model (e.g. from 400000 to 4000 triangles).

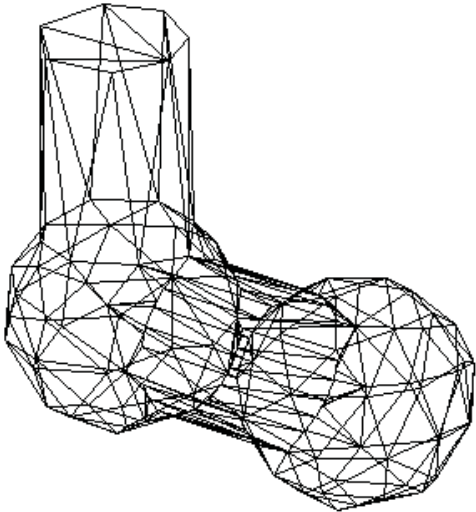
- 1. Select a mesh or TC Surface.



- 2. Select the **Percent to Keep**.

% to keep	Polygons in selection
50	564

- 3. Click the **Finish** button or select **Finish** from the local menu.



Local Menu Options

**Ignore Boundaries, Contract Boundaries at end, Fix Boundaries:** this switch tells the simplifier how should it process the models boundaries (boundary = set of the edges where each edge belongs to the one triangle only)

**Do Full Update before simplification:** you should use this setting if the simplifier fails or its result is incorrect. Usually it means that the simplifier's input model was incorrect. You may try to heal the model by using “Do Full Update” in this case.

XClip

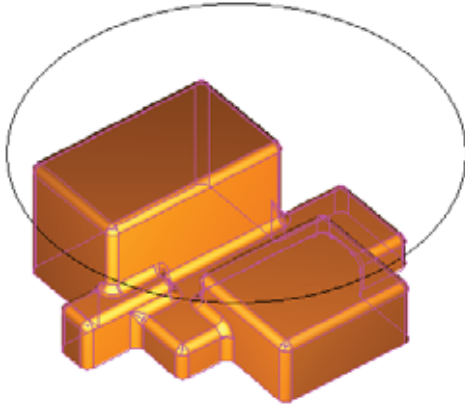


The XClip tool creates a cropped/clipped display of a selected external reference or block reference based upon a selected boundary.

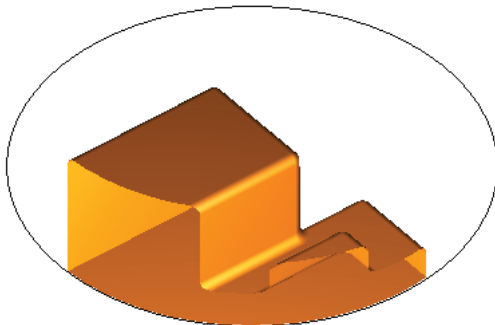


You can use any circle, or closed polyline consisting of only straight segment as a boundary.

1. Select an xref or a block or a group of xrefs or blocks.



2. In the local menu select the Select Polyline option.
3. Click on the desired cutting line.



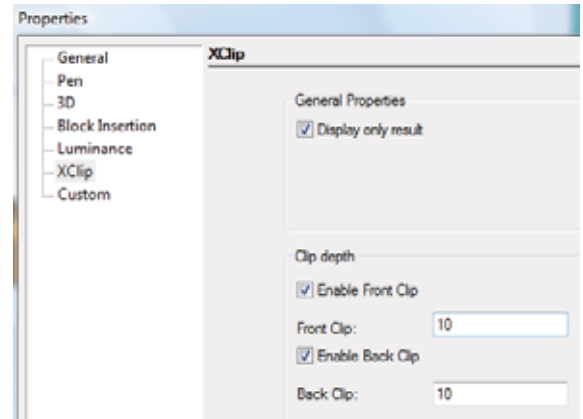
A cropped version of the xref or block is created. The original xref or block insertion is destroyed. The xclip and the selected boundary are not associative, so updating the boundary does not update the xclip.

If the xref/block contained 3D object these objects are shown as “hollowed” whether they are surfaces or ACIS solids. Xclipping does not create new geometry for the clipped entities so the missing faces are simply not displayed. In this regard xclips are not like booleans.

Regardless of the current UCS, the clip depth is applied parallel to the clipping boundary.

## XClip Properties

XClip properties provide added control for how xclips are displayed.



**Display only result:** if this option is unchecked the clipping boundary will be ignored, and all of the geometry of the clipped blocks or xrefs will be displayed.

**Enable front clip:** If this option is selected the xclip will clip everything in the clipped entities above a specified height. Front clipping always occurs parallel to the original clip boundary.

**Front Clip:** Sets the height for front clipping.

**Enable back clip:** If this option is selected the xclip will clip everything in the clipped entities below a specified height. Front clipping always occurs parallel to the original clip boundary.

**Back Clip:** Sets the depth for back clipping.

In the following picture the the xclip has a Front Clip of 12in. and a Back Clip of 1in.





# 12 Rendering

One of the most important tools for controlling the view of your model is **Render** mode. In render mode, all 3D objects are displayed as shaded, or without hidden lines. Higher level rendering also enables you to view materials and textures, showing a realistic view of what your model will actually look like. And in order to create a realistic rendered view, light effects must be added.

To further enhance your render, you can assign materials and luminance qualities to objects. You can also create a render scene environment, which affects the background and foreground of your model.

## Rendering

Displays all objects, or only selected objects, as shaded, or without hidden lines.

### Creating a Rendered View

You can display the **Render** toolbar by right-clicking in any toolbar area and selecting **Render**.



By default, these tools are also available on the local (right-click) menu.

When creating the first render of a model, the easiest way is to select one of the render modes and accept the default lights.

---

*TIP: You can modify the lights by selecting **View / Lights**. For more details, see "Lights" on page 513.*

---

There are several ways to modify and enhance your rendering:

- Adjust render properties (see "Render (Camera) Properties" on page 497)
- Modify or add lights (see "Lights" on page 513)
- Define object materials (see "Materials" on page 522)

- Create a render scene environment (see "Environments" on page 530)

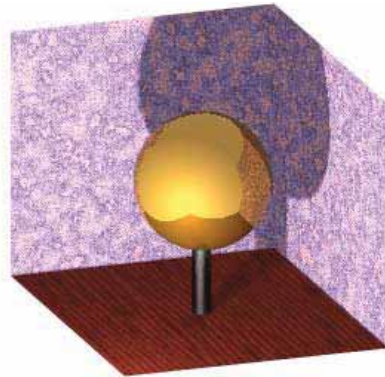
You can use all 3D tools (as well as most 3D editing tools) in both wireframe and render modes. See "Creating and Editing Objects in Render Mode" on page 496.

To render only specific objects, select those objects before selecting the render mode. To return to Wireframe or another render mode, click the relevant icon or use the **View / Camera** menu.

## Quality Rendering



Creates the most photorealistic image with respect to shading, color, reflection, texture, and other qualities. Only the LightWorks rendering type is available for this render mode. For complex models, this type of rendering can be slow.

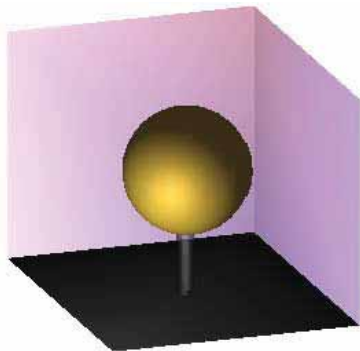


See "Quality Render Properties" on page 500.

## Draft Rendering



Quickly creates a rendered view. The parameters are more limited than with **Quality** rendering, and the view will not be as detailed. The OpenGL and LightWorks rendering types are available for this mode.

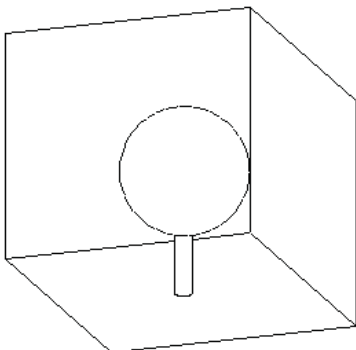


See "Draft Render Properties" on page 499.

## Hidden Line



Eliminates the hidden lines from the view.

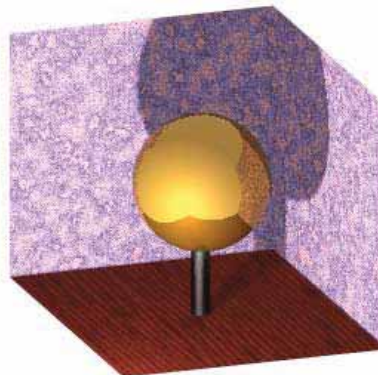


See "Hidden Line Properties" on page 498.

## Advanced Render-Render Styles



Render Styles are used to give you maximum control over the rendering process. A fair variety of preset values are provided including both raytracing and sketch types of rendering



See "Render Styles Properties" on page 500.

---

*NOTE: There are detailed help files available from the Editors for Luminances, Environments, Materials, and Render Styles. These detailed help files describe the function of all settings, and the effect of value changes to parameters in these dialogs*

---

## Creating and Editing Objects in Render Mode

All 3D geometry and most 3D editing tools can be used while in render mode.

You can edit objects using **Select Edit** and **Edit Tool**, perform 3D Boolean commands (with the exception of the Boolean operation **3D Slice**) and manipulate the workplane.

Tools for creating and modifying 2D geometry *cannot* be used in render mode. This includes text and tools for inserting and manipulating pictures.

*TIP: If you need access to 2D profiles for creating profile objects, you can see them in render mode if you check **Foreground Wireframe / Nonrenderable objects** in the **Camera Properties (View / Camera / Properties)** for the various render modes.*

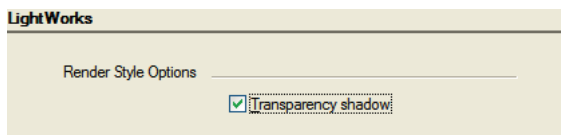
Drag and drop maneuvers are also not available in render mode. Therefore, you cannot drag objects in or out of the Blocks or Symbols Palettes (though the **Format** menu commands are available). You also cannot drag objects into other drawings.

The **Copy** (Ctrl+C) command also works differently in render mode. Rather than copy only the selected objects, the entire scene is captured as a bitmap. It can be inserted into another file or viewport by using **Edit / Paste Special**.

### LightWork Settings

The LightWork settings under **Options/LightWork** will effect any render or saved raster file created with render styles. They are on by default.

**Transparency Shadow:** Specifies whether a render created with a render style will support shadows cast by transparent objects. This toggles the Shadow Transparency setting for every Luminance which has Shadow Transparency turned on.

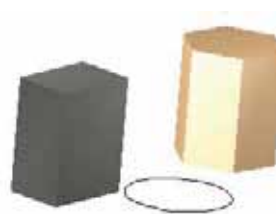


## Render (Camera) Properties

The properties for all render modes contain the following parameters:

**Foreground Wireframe:** Controls wireframe display in render mode. These parameters can be helpful when objects are hidden by other objects, or when not all objects are rendered.

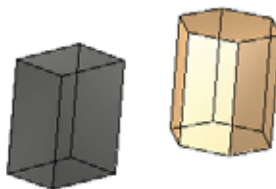
- **Nonrenderable Objects:** If checked, objects that cannot be rendered are displayed as wireframes during rendering. Nonrenderable objects include most 2D objects, though some 2D objects can be rendered, such as images.



If not checked, these objects will not be seen.

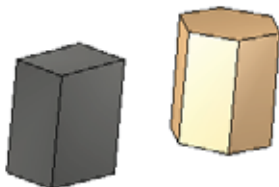


- **Renderable Objects:** Displays the contour lines of renderable objects. Renderable objects include 3D objects and certain 2D objects such as images.



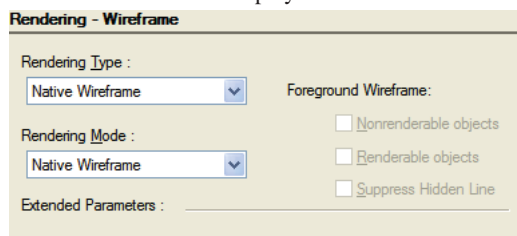
- **Suppress Hidden Line:** Available for Draft and Quality rendering only. If **Renderable Objects** is checked, the hidden contour lines are not displayed.

Properties of the hidden line display are taken from the **Extended Parameters** found on the **Hidden Line Properties**.



## Wireframe Properties

Parameters for wireframe display.



**Rendering type:** The graphics engine that will be used to create the render. The differences between the options relate to the different libraries used for object representation.

- **OpenGL:** All contour lines are displayed.
- **LightWorks:** All contour lines are displayed, but color is not displayed.
- **Native Wireframe:** The standard wireframe mode. For fast drawing, only main contour lines are displayed for ACIS objects.

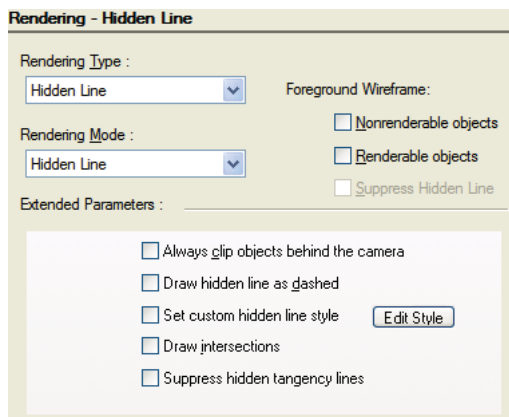
**Rendering mode:** The available modes depend on the rendering type:

- **Wireframe:** All contour lines are displayed.
- **Wire OpenGL:** One-color wireframe rendering.

- **Wire Material Color OpenGL:** Adds color highlighting in accordance with LightWorks algorithms.
- **Native Wireframe:** The standard wireframe mode. For fast drawing, only the main contour lines are displayed for ACIS objects.

## Hidden Line Properties

Parameters for **Hidden Line** rendering (see "Hidden Line" on page 496).



**Rendering type:** The graphics engine that will be used to create the render.

- **Hidden Line:** The standard display mode - transparent rendering.
- **LightWorks:** A for of Non-transparent rendering Hidden lines are completely hidden.
- **Redsdk:** Uses the Redsdk engine for hidden line mode. See the Redsdk section for details on controlling display with Redsdk. This option is only available if Native Draw is set to Redsdk.

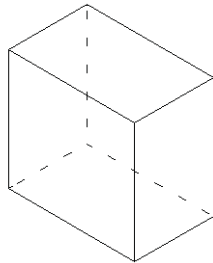
**Rendering mode:** The following modes are available for both rendering types:

- **Hidden Line:** Displays only the contour. Display of intersections and hidden lines depends on the options on the lower part of window. Colors are displayed.
- **Hidden OpenGL:** Non-transparent wireframe render, suppresses hidden lines. Colors are not displayed.

- **Hidden Material Color OpenGL:** Non-transparent wireframe rendering, with suppressed hidden lines. Materials and lighting are also rendered.
- **Redsdk Hidden Line:** Uses the Redsdk engine for hidden line mode. See the Redsdk section for details on controlling display with Redsdk. This option is only available if Native Draw is set to Redsdk.

**Always clip objects behind the camera:** Creates cross-sections by placing a camera inside the object. This means that the object will be "cut" by a plane perpendicular to the camera view direction.

**Draw hidden line as dashed:** Displays all hidden lines as dashed.

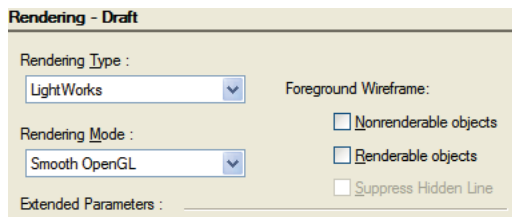


**Set custom hidden line:** Enables you to specify a custom line for hidden lines. Click **Edit** to open the Pen Properties window (see "Pen Properties" on page 81.)

**Draw Intersections:** Displays the lines of object intersections.

## Draft Render Properties

Parameters for **Draft** rendering (see "Draft Rendering" on page 496).



**Rendering Type:** The graphics engine that will be used to create the render.

- **OpenGL:** Allows quick rendering in order to gauge the shape and placement of objects. Because of its speed, OpenGL is best suited for walk-throughs to quickly review the structure of the drawing, but not to show all details. For example, an object's pen color is used rather than its defined material.
- **LightWorks:** Used to create higher quality renders, but can take time.
- **Redsdk:** Uses the Redsdk engine for draft rendering. See the Redsdk section for details on controlling display with Redsdk. This option is only available if Native Draw is set to Redsdk.

**Rendering Mode:** The available modes depend on the rendering type:

- **Flat:** LightWorks' fastest method, produces images quickly but with low image quality. Each facet is painted uniformly with an average color.
- **Gouraud:** Supports smooth shading, does not support textures, shadows, refraction or reflections of light. This is a higher quality render than **Flat**.
- **Phong:** Supports textures and shadows, but not shadows or refraction or reflections of light. It is a higher quality render than **Gouraud**.
- **Flat OpenGL:** LightWorks OpenGL's fastest but lowest-quality method, does not support smoothing.
- **Smooth OpenGL:** The most commonly-used LightWorks OpenGL method, and the default for **Draft** rendering. Speed and quality are optimized.
- **Redsdk Draft:** Uses the Redsdk engine for draft rendering. See the Redsdk section for details on controlling display with Redsdk. This option is only available if Native Draw is set to Redsdk.

## Quality Render Properties

Parameters for **Quality** rendering (see "Quality Rendering" on page 495).

**Rendering - Quality**

Rendering Type : LightWorks

Rendering Mode : Radiosity

Extended Parameters :

Render Mode : Flat Iterations : 2

Quality : 0.97 Max Area : 0.97 Min Area : 0.002

Foreground Wireframe:

☐ Nonrenderable objects

☐ Renderable objects

☐ Suppress Hidden Line

**Rendering Type:** Only the LightWorks option is available.

**Rendering Mode:**

- **Preview:** Supports texture, shadows and lighting effects. Creates rendered images faster than **Full**, but of lower quality.
- **Full:** Supports texture, shadows and lighting effects. This is a higher quality render than **Preview**. **Full** creates rendered images faster than **Raytrace** options, but of lower quality.
- **Raytrace Preview:** Uses raytracing to create medium-quality rendering. Raytracing is a method of calculating light paths, shadows, and reflections. This is a higher quality render than **Full**. **Raytrace Preview** and creates rendered images faster than **Raytrace Full**, but of lower quality.
- **Raytrace Full:** Uses raytracing to create high-quality rendering. It supports texture, shadows and lighting effects. Precision calculations and finer resolution create a higher quality render than **Raytrace Preview**. **Raytrace Full** creates very high quality renderings at the cost of speed.
- **Radiosity:** Generates light distribution throughout the scene; the resolution is independent of any particular viewpoint. The radiosity algorithm is intended for

solving the lighting problem, not the visibility problem. Once the light distribution has been generated, images can be rendered using standard visibility and shading algorithms. Radiosity is light-dependant and requires careful preparation to create the highest quality imaging.

**Extended Parameters:** Available if **Radiosity** is used.

- **Render Mode:** Sets the render method **Radiosity** will use to draw the image after making calculations.
- **Iterations:** The level of refinement in the **Radiosity** calculations. The higher the number, the finer the results. A typical value is between 20 and 30.
- **Quality:** The integrated parameter for improving the quality of an object's shadow displaying. Each quality value corresponds to the default set of radiosity parameters. Two of them, **Max Area** and **Min Area** can be adjusted manually to smooth the object shadow.

## Render Styles Properties

Parameters for **Render Styles** rendering (see "Advanced Render-Render Styles" on page 496).

**Render Style**

Category: Depth of field

Render Style: Focus plane 1 m

Edit Render Style...

Foreground Wireframe:

☐ Nonrenderable objects

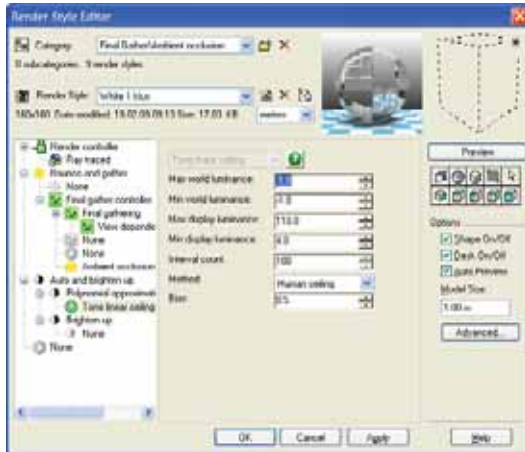
☐ Renderable objects

☐ Suppress Hidden Line

- **Category:** Allows you to select the category or the render style. The following Categories are provided by default: Depth of field, Final Gather (with Ambient Occlusion and Weather subcategories), Ray Traced, and Sketch.
- **Render Style:** Allows you to select the render style from the selected category.



- **Edit Render Style:** Opens the Render Style editor for modifying existing render styles and creating new render styles. The editor will open to the currently selected style.



## Depth of Field

### Focus plane 1 m



**Description:** Render style intended to reproduce the focusing effect.

This rendering style uses a ray casting algorithm and Depth of field effect, reproducing the effect when a real camera is focused on objects a certain distance away and objects nearer or further away appear blurred.

Focus plane = 1.0 m

Focus plane is set using the 'Depth of field \Focus plane' parameter.

### Focus plane 2 m



**Description:** Render style intended to reproduce the focusing effect.

This rendering style uses a ray casting algorithm and Depth of field effect, reproducing the effect when a real camera is focused on objects a certain distance away and objects nearer or further away appear blurred.

Focus plane = 2.0 m

Focus plane is set using the 'Depth of field \Focus plane' parameter.

### Focus plane 5 m



**Description:** Render style intended to reproduce the focusing effect.

This rendering style uses ray casting algorithm and Depth of field effect, reproducing the effect when a real camera is focused on objects a certain distance away and objects nearer or further away appear blurred.

Focus plane = 5.0 m

Focus plane is set using the 'Depth of field \Focus plane' parameter.

### Focus plane 10 m



**Description:** Render style intended to reproduce the focusing effect.

This rendering style uses a ray casting algorithm and Depth of field effect, reproducing the effect when a real camera is focused on objects a certain distance away and objects nearer or further away appear blurred.

Focus plane = 10.0 m

Focus plane is set using the 'Depth of field \Focus plane' parameter.

### **Final Gather**

#### **Final Gather\Draft**



**Description:** Render style intended for preliminary rendering.

This rendering style uses the Final Gather algorithm with minimum render quality settings, without regard for reflectance or transparency. It allows for the quickest rendering while creating or editing a scene.

### **Preview 10**



**Description:** Render style intended for preliminary rendering of models that are about 10 meters in size.

This rendering style uses the Final Gather algorithm with minimum render quality settings, but takes into account size and overall illumination of the model. It allows for a quick rendering while creating or editing a scene.

Accurate lighting = 10%, Model Size = 10 m.

Accurate lighting is set by 'bounce and gather\ lighting, accuracy' parameter; model size is set by 'bounce and gather\ Model Size' parameter.

### **Standard 10m**



**Description:** Render style intended for rendering of models that are about 10 meters in size.

This rendering style uses the Final Gather algorithm with medium render quality settings.

Accurate lighting = 40%, Model Size =10 m.

Accurate lighting is set by 'Bounce and gather\ Lighting, Accuracy' parameter; model size is set by 'Bounce and gather\ Model Size' parameter.

### Standard 30m



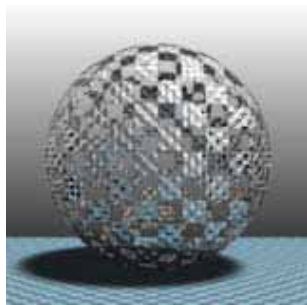
**Description:** Render style intended for rendering of models that are about 30 meters in size.

This rendering style uses the Final Gather algorithm with medium render quality settings.

Accurate lighting = 40%, Model Size =30 m.

Accurate lighting is set by 'Bounce and gather\ Lighting, Accuracy' parameter; model size is set by 'Bounce and gather\ Model Size' parameter.

### Standard 50m



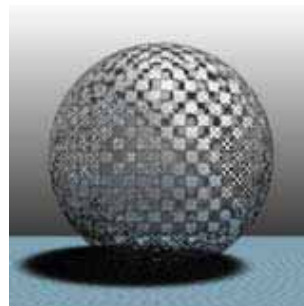
**Description:** Render style intended for rendering of models that are about 50 meters in size.

This rendering style uses the Final Gather algorithm with medium render quality settings.

Accurate lighting = 40%, Model Size =50 m.

Accurate lighting is set by 'Bounce and gather\ Lighting, Accuracy' parameter; model size is set by 'Bounce and gather\ Model Size' parameter.

### Standard 100m



**Description:** Render style intended for rendering of models that are about 100 meters in size.

This rendering style uses the Final Gather algorithm with medium render quality settings.

Accurate lighting = 40%, Model Size =100 m.

Accurate lighting is set by 'Bounce and gather\ Lighting, Accuracy' parameter; model size is set by 'Bounce and gather\ Model Size' parameter.

### Presentation 10m



**Description:** Render style intended for rendering of models that are about 10 meters in size.

This rendering style uses the Final Gather algorithm with high render quality.

Accurate lighting = 80%, Model Size = 10 m.

Accurate lighting is set by 'Bounce and gather\ Lighting, Accuracy' parameter; model size is set by 'Bounce and gather\ Model Size' parameter.

## ***Ambient Occlusion***

### **White 0.1 klux**



**Description:** Render style intended for rendering of models that are about 10 meters in size.

This rendering style uses the Final Gather algorithm with medium render quality settings and an ambient lighting effect.

Ambient lighting intensity = 0.1 Kilolux, Ambient lighting color = White.

Lighting intensity is set by 'Ambient occlusion\Intensity' parameter, color is set by 'Ambient occlusion\Colour' parameter.

### **White 0.5 klux**



**Description:** Render style intended for rendering of models that are about 10 meters in size.

This rendering style uses the Final Gather algorithm with medium render quality settings and ambient lighting effect.

Ambient lighting intensity = 0.5 Kilolux, Ambient lighting color = White.

Lighting intensity is set by 'Ambient occlusion\Intensity' parameter, color is set by 'Ambient occlusion\Colour' parameter.

### **White 1.0 klux**



**Description:** Render style intended for rendering of models that are about 10 meters in size.

This rendering style uses the Final Gather algorithm with medium render quality settings and an ambient lighting effect.

Ambient lighting intensity = 1.0 Kilolux, Ambient lighting color = White.

Lighting intensity is set by 'Ambient occlusion\Intensity' parameter, color is set by 'Ambient occlusion\Colour' parameter.

### Warm 0.1 klux



**Description:** Render style intended for rendering of models that are about 10 meters in size.

This rendering style uses the Final Gather algorithm with medium render quality settings and an ambient lighting effect.

Ambient lighting intensity = 0.1 Kilolux, Ambient lighting color temperature = 3000.0.

Lighting intensity is set by 'Ambient occlusion\Intensity' parameter; color is set by 'Ambient occlusion\Colour temperature' parameter.

### Warm 0.5 klux



**Description:** Render style intended for rendering of models that are about 10 meters in size.

This rendering style uses the Final Gather algorithm with medium render quality settings and an ambient lighting effect.

Ambient lighting intensity = 0.5 Kilolux, Ambient lighting color temperature = 3000.0.

Lighting intensity is set by 'Ambient occlusion\Intensity' parameter; color is set by 'Ambient occlusion\Colour temperature' parameter.

### Warm 1.0 klux



**Description:** Render style intended for rendering of models that are about 10 meters in size.

This rendering style uses the Final Gather algorithm with medium render quality settings and an ambient lighting effect.

Ambient lighting intensity = 1.0 Kilolux, Ambient lighting color temperature = 3000.0.

Lighting intensity is set by 'Ambient occlusion\Intensity' parameter; color is set by 'Ambient occlusion\Colour temperature' parameter.

### Blue 0.1 klux



Description: Render style intended for rendering of models that are about 10 meters in size.

This rendering style uses the Final Gather algorithm with medium render quality settings and an ambient lighting effect.

Ambient lighting intensity = 0.1 Kilolux, Ambient lighting color temperature = 7500.0.

Lighting intensity is set by 'Ambient occlusion\Intensity' parameter; color is set by 'Ambient occlusion\Colour temperature' parameter.

### Blue 0.5 klux



**Description:** Render style intended for rendering of models that are about 10 meters in size.

This rendering style uses the Final Gather algorithm with medium render quality settings and an ambient lighting effect.

Ambient lighting intensity = 0.5 Kilolux, Ambient lighting color temperature = 7500.0.

Lighting intensity is set by 'Ambient occlusion\Intensity' parameter; color is set by 'Ambient occlusion\Colour temperature' parameter.

### Blue 1.0 klux



**Description:** Render style intended for rendering of models that are about 10 meters in size.

This rendering style uses the Final Gather algorithm with medium render quality settings and an ambient lighting effect.

Ambient lighting intensity = 1.0 Kilolux, Ambient lighting color temperature = 7500.0.

Lighting intensity is set by 'Ambient occlusion\Intensity' parameter; color is set by 'Ambient occlusion\Colour temperature' parameter.

## Weather

### Clear Sky



**Description:** Render style intended to produce rendering with a specified illumination.



This rendering style uses the Final Gather algorithm with minimum render quality settings with illumination level set up under Tone Linear Ceiling to emulate a clear sky.

### Cloudy



**Description:** Render style intended to produce rendering with a specified illumination.

This rendering style uses the Final Gather algorithm with minimum render quality settings with illumination level set up under Tone Linear Ceiling to emulate a cloudy sky.

### Hazy



**Description:** Render style intended to produce rendering with a specified illumination.

This rendering style uses the Final Gather algorithm with minimum render quality settings with illumination level set up under Tone Linear Ceiling to emulate a hazy sky.

### Night - Overcast



**Description:** Render style intended to produce rendering with a specified illumination.

This rendering style uses the Final Gather algorithm with minimum render quality settings with illumination level set up under Tone Linear Ceiling to emulate night an overcast sky.

### Overcast



**Description:** Render style intended to produce rendering with a specified illumination.

This rendering style uses the Final Gather algorithm with minimum render quality settings with illumination level set up under Tone Linear Ceiling to emulate an overcast sky.

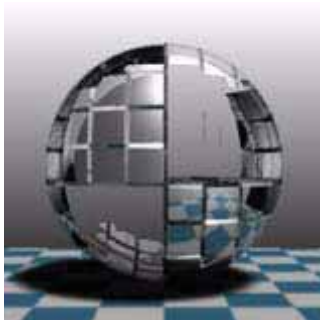
### Twilight - Clear



**Description:** Render style intended to produce rendering with a specified illumination.

This rendering style uses the Final Gather algorithm with minimum render quality settings with illumination level set up under Tone Linear Ceiling to emulate twilight with a clear sky.

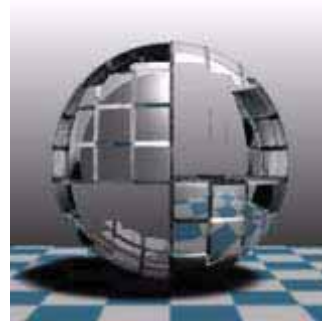
### Twilight - Cloudy



**Description:** Render style intended to produce rendering with a specified illumination.

This rendering style uses the Final Gather algorithm with minimum render quality settings with illumination level set up under Tone Linear Ceiling to emulate twilight with a cloudy sky.

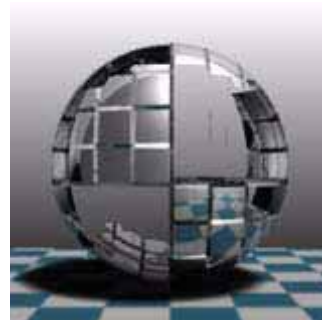
### Twilight - Hazy



**Description:** Render style intended to produce rendering with a specified illumination.

This rendering style uses the Final Gather algorithm with minimum render quality settings with illumination level set up under Tone Linear Ceiling to emulate twilight with a hazy sky.

### Twilight - Overcast



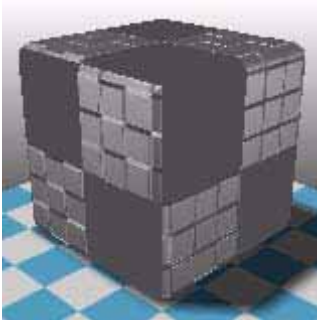
**Description:** Render style intended to produce rendering with a specified illumination.

This rendering style uses the Final Gather algorithm with minimum render quality settings with illumination level set up under Tone Linear Ceiling to emulate twilight with an overcast sky.



## Ray Traced

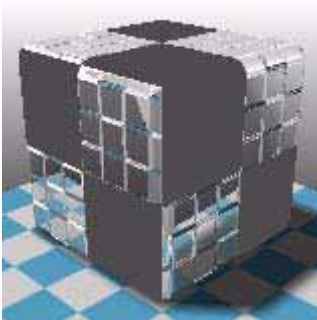
### Preview



**Description:** Render style intended for preliminary rendering.

This rendering style uses ray casting algorithm with minimum render quality settings. It allows for a quick rendering while creating or editing a scene.

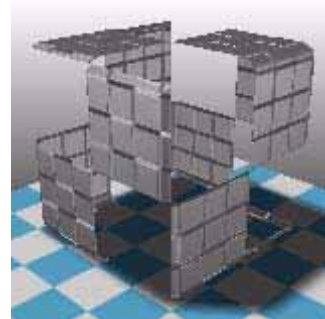
### No Transparency



**Description:** Render style intended for rendering without regard for transparency.

This rendering style uses ray casting algorithm with Transparency = Off. It allows for a quick preliminary rendering for scenes that have a large number of transparent materials of 'Glass' type.

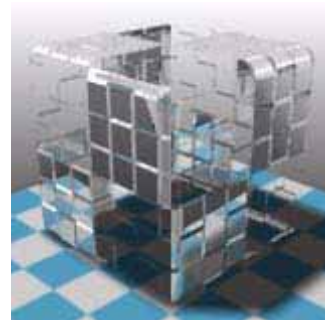
### No Reflectance



**Description:** Render style intended for rendering without regard for reflectance.

This rendering style uses ray casting algorithm with Reflectance = Off. It allows for a quick preliminary rendering for scenes that have a large number of reflective materials of 'Mirror' type.

### \Standard

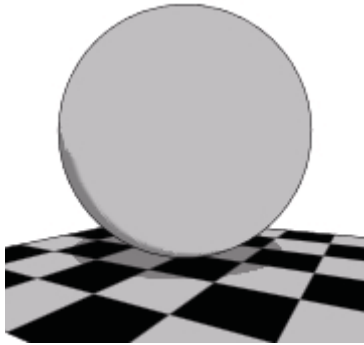


**Description:** Render style intended for medium render quality renderings.

This rendering style uses ray casting algorithm with optimum settings for time/render quality. It allows for a quick rendering for the majority of scenes.

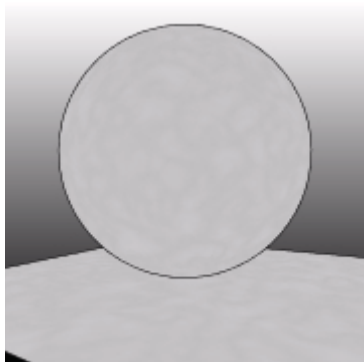
## ***Sketch***

### **Cartoon**



**Description:** Render style intended to create a cartoon effect.

### **Colour Wash**



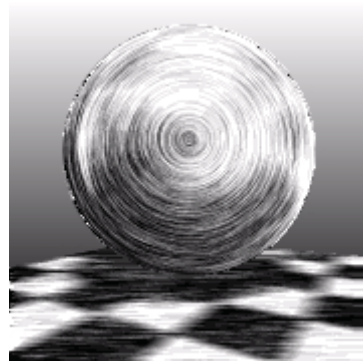
**Description:** Render style intended to create a colour wash effect.

## **Color Contour**



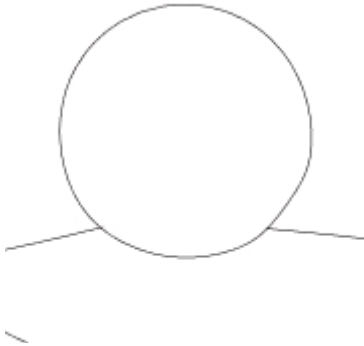
**Description:** Render style intended to create a colour contour effect.

## **Gray Contour**



**Description:** Render style intended to create a gray contour effect.

### Hand Drawn



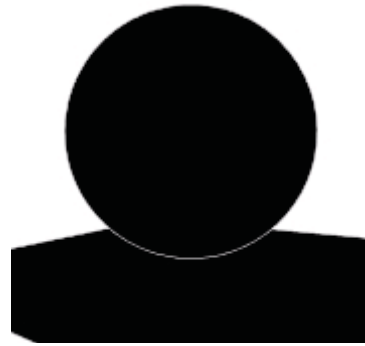
**Description:** Render style intended to create a hand drawn effect.

### Hatch



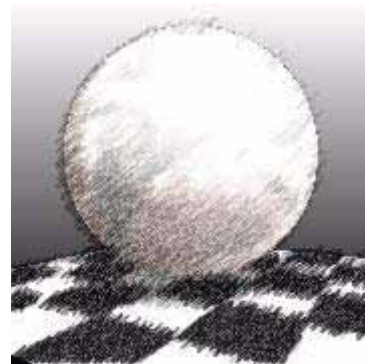
**Description:** Render style intended to create a hatch effect.

### Ink Print



**Description:** Render style intended to create an ink print effect.

### Lines and Colour Fill



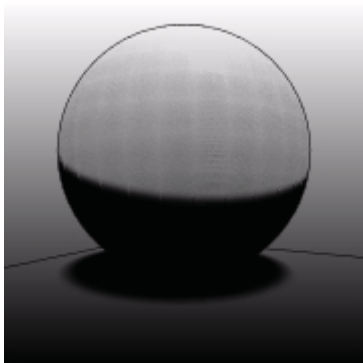
**Description:** Render style intended to create a line and colour fill effect.

## Mosaic



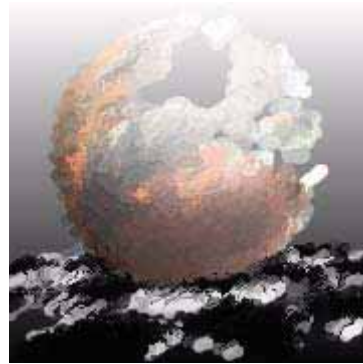
**Description:** Render style intended to create a mosaic effect.

## Lines and Shadow



**Description:** Render style intended to create a lines and shadow effect.

## Oil Painting



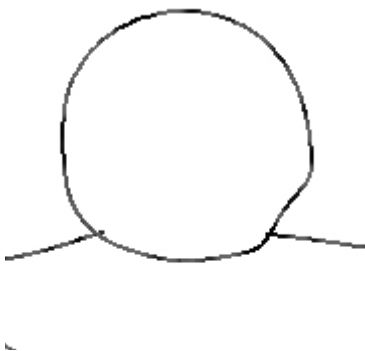
**Description:** Render style intended to create an oil painting effect.

## Soft Pencil



**Description:** Render style intended to create a soft pencil effect.

## Rough Pencil



**Description:** Render style intended to create a rough pencil effect.

## Stipple



**Description:** Render style intended to create a stipple effect.

## Lights

Creating realistic rendered views requires realistic light effects. There are several types of lights you can create and position, whose effects can be viewed while rendering.

The first time you render a model, you can choose to have a set of default lights created. You can use these, modify them, and/or add your own lights.

*NOTE: You can also create and manipulate lights, and group lights into sets, via the Design Director. See "Design Director: Lights" on page 132.*

## Creating Lights - Light Types

You can add lights to your model by using the **Insert / Light** options, or by selecting **View / Lights** and clicking **New**. You can place the light and define its direction (where applicable) manually on the screen, or enter the coordinates.

You can display the **Lights** toolbar by right-clicking in any toolbar area and selecting **Lights**.

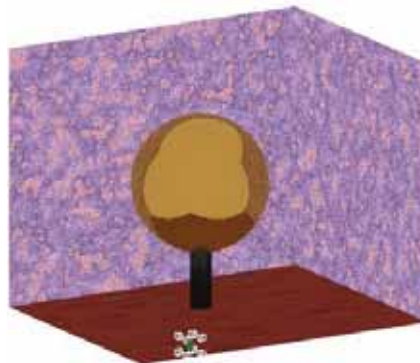


*NOTE: It is important to test changes made in lighting as you proceed. Changing all of the light settings in one step can create unexpected results.*

## Ambient Light



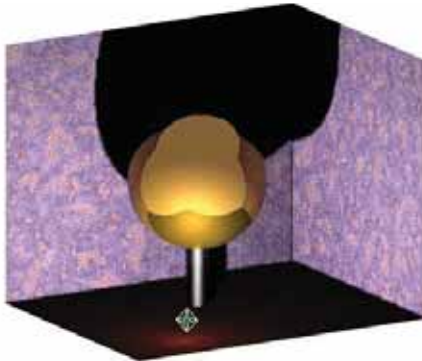
Provides light equally from all directions. Light from an ambient source shines equally on all sides of an object. The **Position** and **Direction** options are turned off because they are irrelevant.



## Point Light



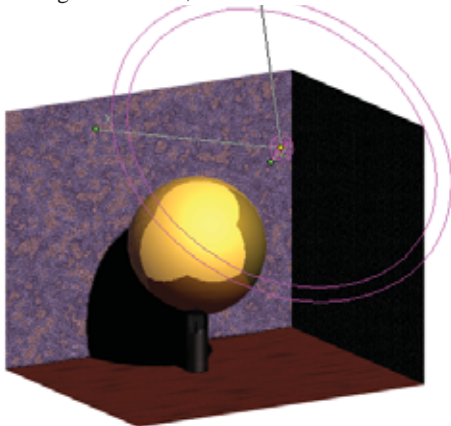
Provides light from a selected position. Light from a point source will illuminate each surface of an object depending upon the position of the surface to the light. A surface perpendicular to the light will receive more illumination than a surface that is at an angle. A surface that faces away from the point source will be dark.



## Directional Light



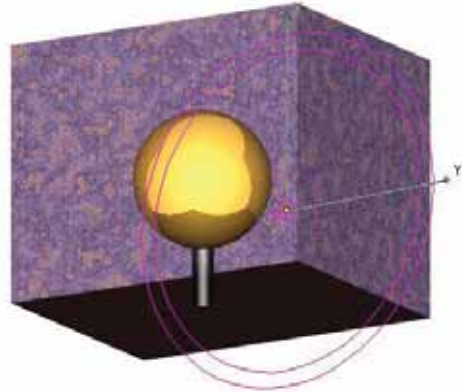
Provides light that has a specific direction. The direction is specified by an origin and a selected point. The length of the directional light is infinite, and does not fade with distance.



## Headlight



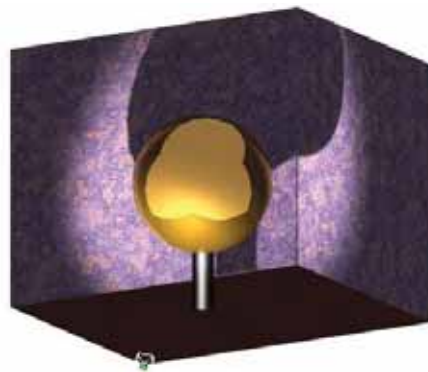
Provides illumination from the camera position. In effect, a headlight is a directional light that uses the camera position as the directional point. As the camera moves, the direction of the light source is updated.



## Spot Light



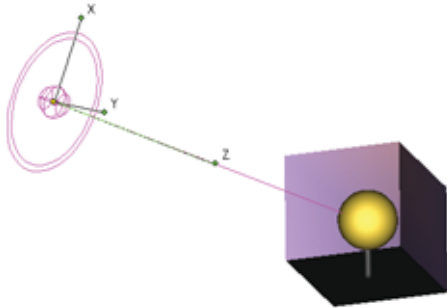
Provides directional light originating from a point and constrained to a cone.



Sky Light

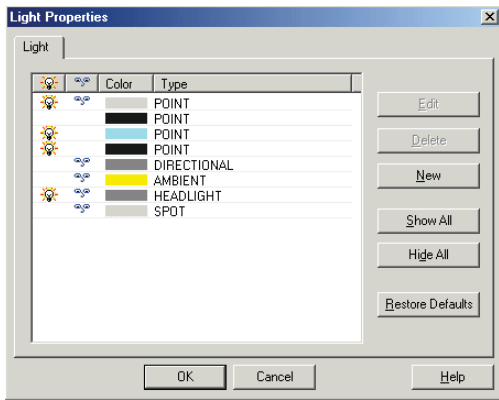




A directional light source that simulates outdoor sunlight.



Controlling Lights

Controlling lights is done via the **Light Properties** window, opened by selecting **View / Lights**.

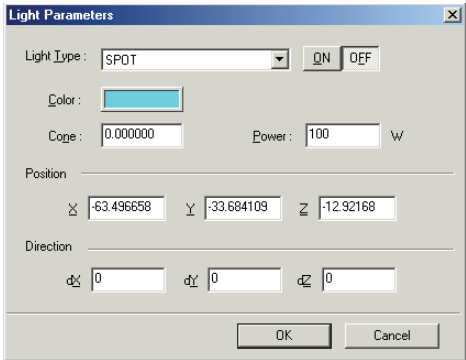


**Light tab:** Shows all of the currently defined lights. An initial list of default lights can be created when you first enter draft or quality render mode. The  symbol indicates whether the light is turned on. The  symbol indicates whether the light indicator will be displayed. See "Light Indicators" on page 515.

**Edit:** Click to open the **Properties** window for the selected light. See "Editing Lights" on page 516.

**Delete:** Deletes the selected light.

**New:** Click to add a new light. The **Light Parameters** window appears, in which you can specify the light type and parameters. For details on the parameters, see "Light Properties" on page 516.



**Show All:** Displays all light indicators.

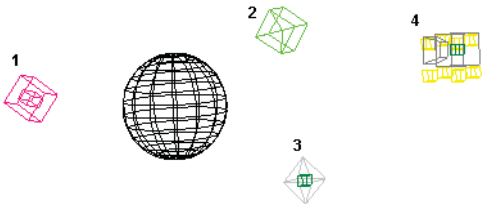
**Hide All:** Hides all light indicators.

**Restore Defaults:** Returns to the initial default light set. New lights will be deleted, and modified lights will be reset to their original settings.

Light Indicators

Each light has an indicator which can be displayed. Indicators appear as 3D icons that indicate the defined light color and brightness. The visibility of indicators can be set in the **Light Properties** window.

In the example below, Light 1 is a **Directional**, Light 2 is a **Spot**, Light 3 is **Headlight**, and Light 4 is **Point**.



When a light is on, the indicator will be dark. When the light is off, the indicator will be light.



Light indicators can be selected like any other 3D object. Repositioning and rotating a light indicator will reset the light properties. Light properties can be viewed and edited by selecting the light indicator and opening the **Properties** window.

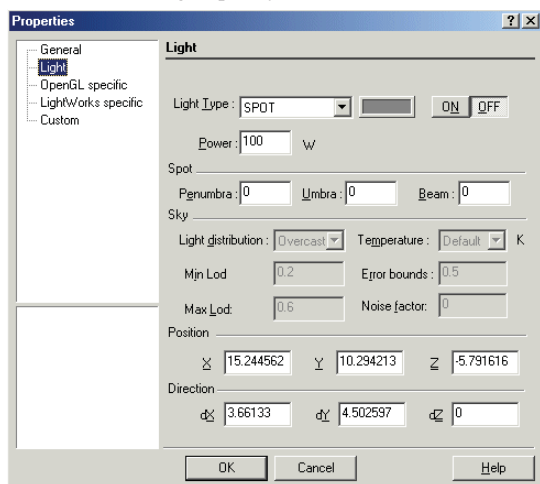
## Editing Lights

For every light in your model, you can edit its properties. You can access the **Properties** window by double-clicking the light indicator (see "Object Properties" on page 79) or by selecting **Lights** and clicking **Edit** for the desired light.

Three pages of a light's properties are specific to lights: **Light**, **OpenGL Specific**, and **LightWorks Specific**.

## Light Properties

Parameters of the light quality and location.



**Light Type:** Type and color of the light, and whether it is on or off.

**Power:** The intensity of the light, in watts. The higher the setting, the brighter the light. This parameter is relevant only for **Radiosity** rendering. The absolute values (0 through 255) of the color components handled through the **Color** controls determine the absolute value of the light power. For example, if only the Red color is enabled and its color value is N

(N=R+G+B), then the actual value of the light power is determined by the following expression ( $P \cdot N / 255$ ) where P is the power value set in the **Power** field.

---

*TIP: Setting wattage values too high can cause objects to have a washed-out appearance.*

---

**Spot:** Parameters that apply only to **Spot** lights - sources that emit light constrained to a cone.

- **Penumbra:** The angle of the beam.
- **Umbra:** The angle of the dark center of the spot light.
- **Beam:** The sharpness of the spotlight. A value of 0 means the light will be evenly dispersed. A higher value means more light will be concentrated toward the center of the beam.

**Sky:** Parameters that apply only to **Sky** lights.

- **Light distribution:** The level of sky clarity, ranging from clear to overcast. For a clear sky, the brightest part of the sky surrounds the sun. For an overcast sky, the brightest part of the sky is straight up.
- **Min Lod:** Minimum level of detail for area source decomposition. Usually this value is between 0.0 and 1.0, closer to 0.0, but values can be greater than 1.0 as well. This parameter determines the initial sampling for lighting calculation. If **Min Lod** is too low, shadow boundaries may not be reproduced correctly. If it is too high, rendering may take a long time. Practical values are between 0.0 and 0.5.
- **Max Lod:** Maximum level of detail for area source decomposition. This delimits the maximum amount of work for any point being illuminated. If set to the same value as **Min Lod**, calculations in regions of rapidly-varying irradiance may fail. If **Max Lod** is smaller than **Min Lod**, their values will be reversed.
- **Temperature:** The ambient temperature, in degrees K. Used to set the sky color.
- **Error bounds:** When **Max Lod** is sufficiently larger than **Min Lod**, the shader will perform adaptive sampling, once its initial sampling is complete. The **Error bounds** parameter then dictates the accuracy of



the solution. An value of 0.1 means that the maximum error associated with the illumination of a particular point of interest is less than 10% of the energy associated with the illumination arriving at the point. Values much smaller than 0.1 are not unusual; the default value (0.5) discourages adaptive sampling.

- **Noise factor:** You can add noise to provide patterns which look naturally chaotic and noisy. A value of 0.0 means no noise; a value of 1.0 means maximum noise.

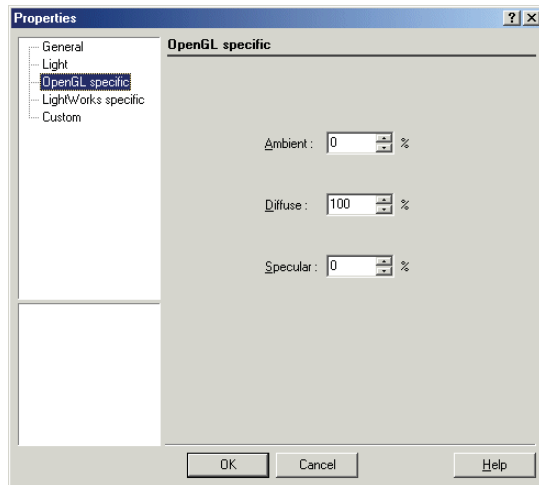
**Position:** The location or origin of the light.

**Direction:** For lights with a specific orientation, these are delta values in X, Y, and Z representing the directional vector.

### OpenGL Specific Properties

Parameters relevant for OpenGL rendering.

The Open GL rendering engine creates views of lower quality than the LightWorks engine, but at a higher speed.



**Ambient:** The amount of ambient light - light provided equally from all directions.

**Diffuse:** The quantity of diffuse light reflected by each surface that is lit by the light. When diffuse light strikes a surface, it is reflected in every direction equally. This makes the surface appear as if it has a rough or matte finish.

**Specular:** The quantity of specular light reflected by each surface that is lit by the light. When specular light strikes a surface, it reflects in only one direction. This makes the surface appear as if it has a smooth, mirror-like finish.

---

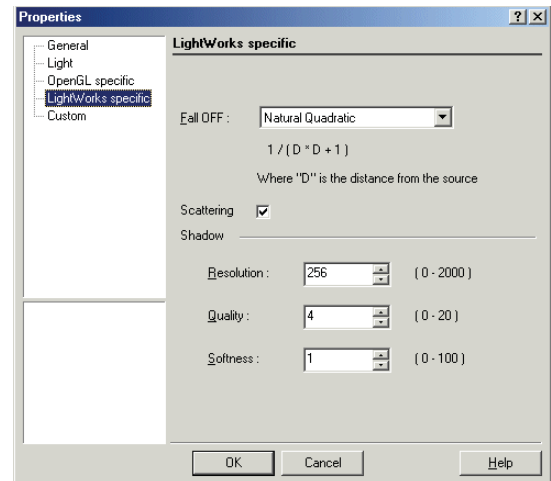
*WARNING: The Open GL settings are cumulative. When all three light component values are set to the maximum, the light source will generate three times the amount of light of a single maximum setting.*

---

### LightWorks Specific Properties

Parameters relevant for LightWorks rendering.

The LightWorks rendering engine creates views of high quality, but view creation can take time.



**Fall Off:** Relevant for **Point** and **Spot** light sources, determines the way in which the light's intensity is attenuated as a function of the distance from the source. By default this parameter is set to "No attenuation", i.e. light intensity is constant. Other available fall-off types include:

- **Natural Linear:**  $1/(d+1)$
- **Natural Quadratic:**  $1/(d*d+1)$
- **Linear:**  $1/d$
- **Quadratic:**  $1/d*d$

Where  $d$  = the distance from the light source.

**Scattering:** Creates the effect of light being scattered by a foggy atmosphere.

**Shadow:** Controls shadow display. Increasing **Resolution** enables you to smooth the shadow edges. You may get some light and shadow effects by specifying **Quality** and **Softness** parameters.

## Light Sets

A light set is a group of lights, which can be handy if you want to set identical parameters for multiple lights. Light sets are created and manipulated in the Design Director. See "Design Director: Lights" on page 132.

## Luminance

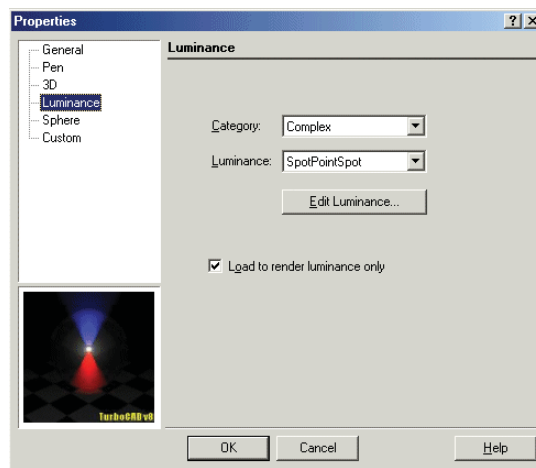
Another way to add lights to your drawing is to assign luminance properties to the entire mode, or to individual 3D objects. Luminance is one or more lights attached directly to objects, or to the overall WCS or UCS. Luminance differs from lights (see "Lights" on page 513) in that the range of luminance effects is much larger than those available for drawing lights.

Luminance properties are parameters of the LightWorks rendering engine. An object's luminance can be seen either in **Draft** or **Quality** rendering, as long as an appropriate LightWorks rendering mode is used.

Several luminances and categories are provided by default. You can create a new luminance via the **Luminances Editor** (see "Editing Luminances" on page 520).

## Luminance Properties

You can access luminance properties via the **Properties** window of an object (see "Object Properties" on page 79), or by using the Luminances Palette (see "Luminances Palette" on page 521).



**NOTE:** If you are specifying luminance for the overall drawing, use the **Render Scene Luminance**. Drawing luminance can be applied either to the WCS (World Coordinate System) or UCS (User Coordinate System). See "WCS and UCS" on page 105.

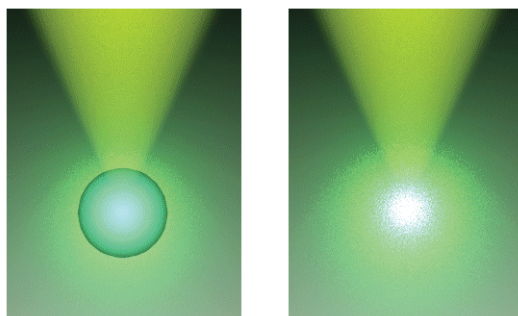
**Category:** Three categories are provided for you, but you can create new ones.

- **Single:** Defines only one light type. See "Single Luminance" on page 519.
- **Complex:** Defines two or more lights. See "Complex Luminance" on page 520.
- **Floresc Architec AV:** A set of light types representing realistic lights, based on \*.ies files from Lithonia Lighting. These files are stored in the Luminance\Photometric\ies folder.

**Luminance:** This field depends on the category. See the relevant section for details.

**Edit Luminance:** Opens the **Luminance Editor**. See "Editing Luminances" on page 520.

**Load to render Luminance only:** Check if you want to view only the object lights, and not the object itself.



**Normal Render (Showing Object), and Load to Render Luminance Only**

## Light Types

There are several types of lights you can use to define an object's luminance. Each type has relevant parameters such as intensity, color, direction, and location.

**Ambient:** Illuminates all surfaces equally, regardless of orientation.

**Area:** Simulates a luminous surface.

**Area Goniometric:** Combines area and goniometric shaders.

**Area Sky:** Similar to **Sky**, but models only sky light that passes through portals, such as a window or door.

**Distant:** Light that is constant and parallel (non-conical).

**Eye:** A light source located at the view point.

**Goniometric:** Directional light, whose direction is specified by an \*.ies file (a text-based file). This luminance is useful for creating lighting based on manufacturers' lighting data, such as room lights, building lights, and street lights.

**Point:** Light from a selected position.

**Projector:** Projects a graphic image onto a solid object. An image file with full path must be specified.

**Simple Environment:** A light shader that uses an environment map to light a scene.

**Simple Sky:** A simplified sky shader which models the illumination from a uniformly bright, single-color, hemispherical dome representing the sky.

**Sky:** An area light that simulates sky light, providing soft and natural shadows. This light is modeled as a hemisphere of infinite radius, positioned above the center of the model.

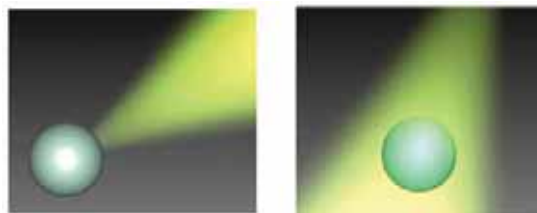
**Spot:** Provides directional light that is constrained to a cone.

**Sun:** Projects a spot light from a very distant point, to simulate outdoor sunlight. You can combine **Sun** and **Sky** lights for a realistic rendering effect.

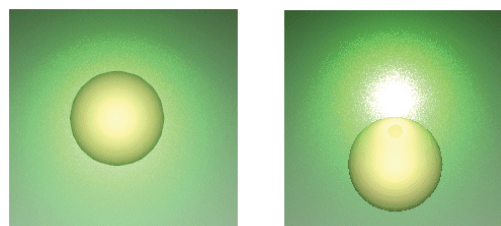
## Single Luminance

Defines only one light. You can select one of the provided luminances, or create a new luminance via the **Luminance Editor** (see "Editing Luminances" on page 520).

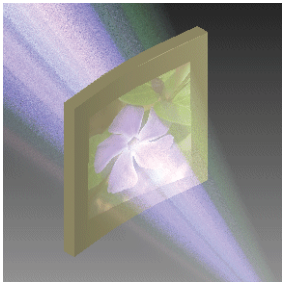
Examples of **Single Luminance** light types are shown below.



**Spot Light**



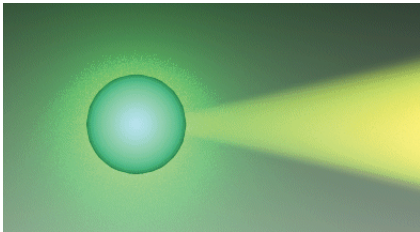
**Point Light**



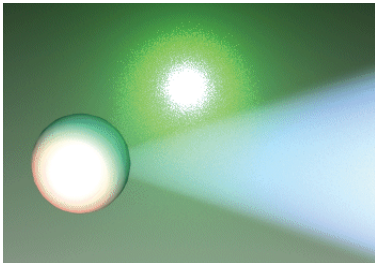
Projector (Projects an Image)

Complex Luminance

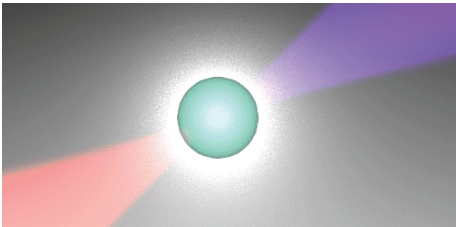
Assign two or more lights. You can use one of the provided defaults, or you can create a new luminance via the **Luminance Editor** (see "Editing Luminances" on page 520). All lights types available for the **Single** category are also available for **Complex**.



Spot and Point Lights



Spot, Point, and Distant Lights

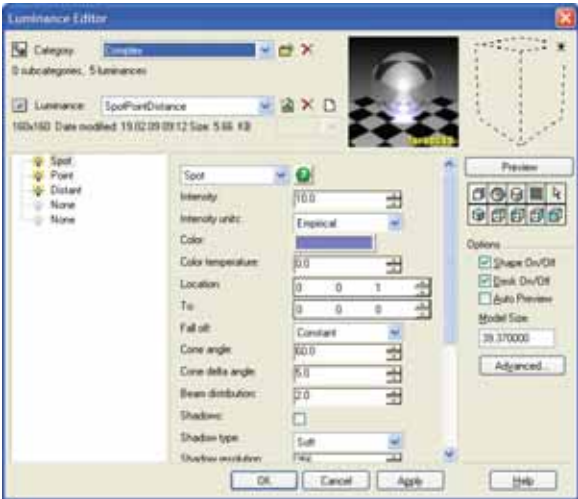


Point and Two Spot Lights

Editing Luminances

The **Luminance Editor** can also be accessed from the **Luminance** page of an object's **Properties**. You can also access it from the Luminances Palette; select the item in the palette, then right-click and select **Edit Luminance**. Or double-right-click on the item, or click **Edit Luminance**.

Or you can select a luminance on the Luminances Palette, right-click and select **Edit Luminance**.



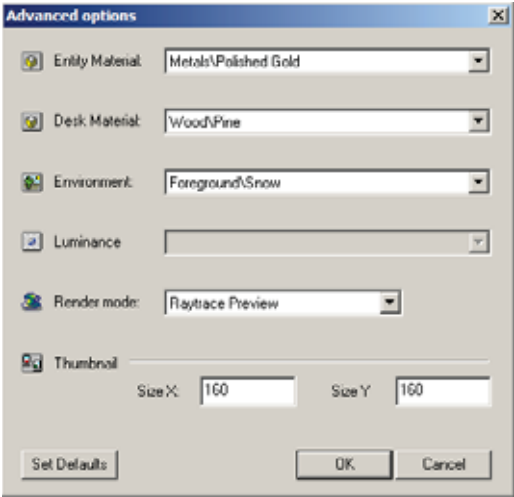
To create a new luminance, click **New Luminance**. You can also define a new category, or add a new luminance to an existing category. Click **Delete** to delete a luminance or category.

*TIP: You can also create a subcategory within an existing category. Click **New Category**, and use the format "Category\Subcategory."*

**Complex** luminances can contain up to five light types; each type is represented by a page on the **Luminance Editor**. For each page, select the desired light type (see "Light Types" on page 519), and edit its parameters as needed. The available parameters depend on the selected light type.

In the **Preview** section, you can change the shape and orientation of the preview object. Under **Options**, you can toggle the display of the preview object or desk (the surface below the object). If **Auto Preview** is checked, the preview will update each time a change is made (otherwise you must click the **Preview** button).

Click **Advanced** to change the appearance of the preview object or desk, or the background environment. You can also change the rendering type and thumbnail size.

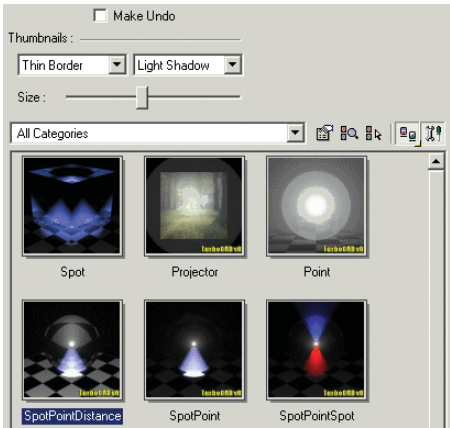


You can see the new desk and environment in the **Preview** window.



### Luminances Palette

Provides easy and fast access to object luminances.



All luminances that have been defined are displayed here, and can be assigned to objects and edited. Use the **Thumbnail** controls to adjust how the items are displayed, and use the **Size** slider to adjust the item size. Check **Make Undo** to add luminance to the **Undo / Redo** history of the model.

To apply a luminance to an object, first select the object. Then double-click the palette item to apply it. You can also right-click on a luminance and select **Set Luminance**.

To remove a luminance from an object, select the object. Then in the palette, right-click and select **Set None**.

To find objects that have a specific luminance, select the luminance in the palette, then click **Select by Luminance**.



To find the luminance of an object, select the object and then click **Find Luminance of Selection**.



To edit a luminance, select the item in the palette, then right-click and select **Edit Luminance**. Or double-right-click on the item, or click **Edit Luminance**.



To switch the luminance display from thumbnails to list format, click **View Thumbnails** and select a different view.



You can hide or show the palette fields and toolbars by clicking **Show / Hide Options**.



**Loading and Saving Luminances**

If you want to save luminance properties for future use, select **Options / Luminance / Save Luminances**, or right-click in the Luminances Palette and select **Save Luminances**. Use the categories on the right side of the **Save** window to specify which luminances you want to save. Luminance data will be written to a \*.dat file, located in the **Luminance** folder.

Any new luminance you define will be automatically saved in the TurboCAD file as well.

To load a luminance \*.dat file, select **Load Luminances** from the **Options / Luminance** menu, or from the local menu of the palette.

**Materials**

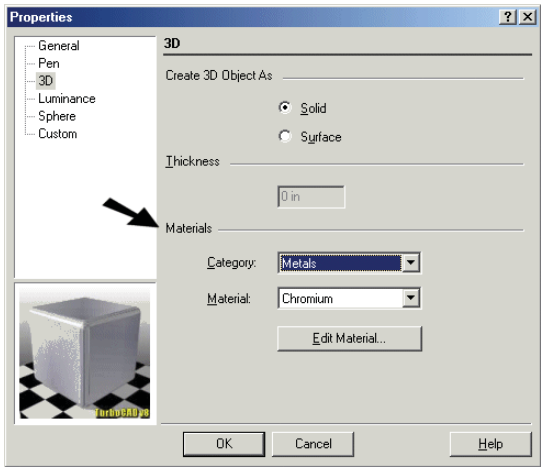
By applying materials to a 3D object, you can obtain a more realistic view of the model during a render.

An object's materials can be seen in **Quality** render mode, and in certain types of **Draft** rendering.

Several materials and categories are provided by default. You can create a new material via the **Materials Editor** (see "Editing Materials" on page 523).

**Material Properties**

You can access material properties via the **3D** page of an object's **Properties** window (see "Object Properties" on page 79), or by using the Materials Palette.



**Category:** Groups of materials such as glass, metals, stone, and wood.

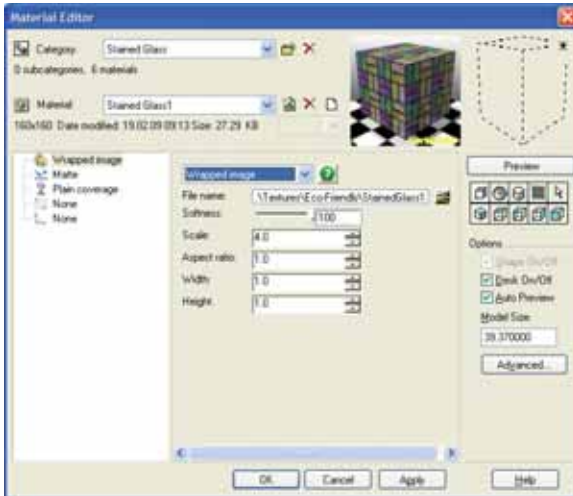
**Material:** This field depends on the category. For example, if the **Category** is Metals, the materials include brass, iron, aluminum, etc.

**Edit Material:** Opens the **Materials Editor**. See "Editing Materials" on page 523.



## Editing Materials

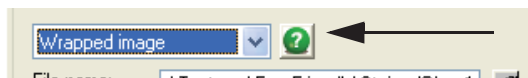
The **Materials Editor** can also be accessed from the **3D** page of an object's **Properties**. You can also access it from the Materials Palette; select the item in the palette, then right-click and select **Edit Materials**. Or double-right-click on the item, or click **Edit Materials**.



Each material has five categories of parameters you can set: Pattern, Reflectance, Transparency, Texture and Wrapping. Not all categories are used for each material, only those that are relevant.

For more information on Materials HDR, HDRI, Luminances and all other LightWorks.

1. Select a material in the Materials Palette
2. Right click and select Edit Materials.
3. When the Material Editor appears click the small green help button. This will open the LightWorks help, which is replete with details on functions and settings.



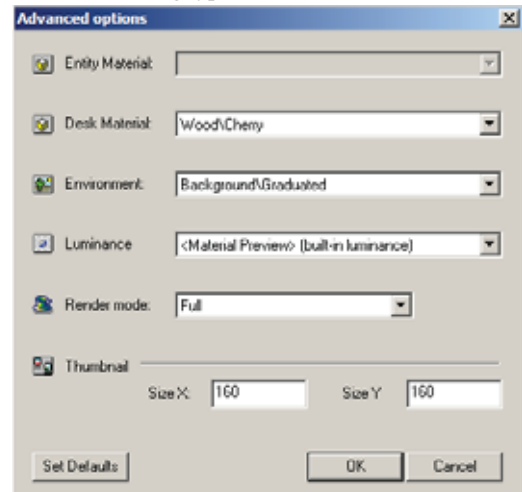
**NOTE:** *Materials in TurboCAD is a vast topic unto itself, and cannot be covered fully here. The TurboCAD Community has a web page where you can get much more information:*

<http://www3.turbocadcommunity.com/tiki-index.php?page=Lightworks>

Another reference is the book *Photorealism in TurboCAD Pro*, which is available at [www.cadcouse.com](http://www.cadcouse.com).

In the **Preview** section, you can change the shape and orientation of the preview object. Under **Options**, you can toggle the display of the desk (the surface below the preview object). If **Auto Preview** is checked, the preview will update each time a change is made (otherwise you must click the **Preview** button).

Click **Advanced** to change the material of the preview desk or background environment. You can also change the luminance, rendering type, and thumbnail size.

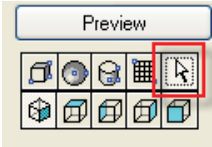


You can see the new desk and environment in the **Preview** window.



You can also see approximately how the material will appear on an object by entering a value into the **Model Size** field. Simply specify a size that is close to the size of the object to which you will be applying the material.

Alternately, you can use the selection arrow under the preview.



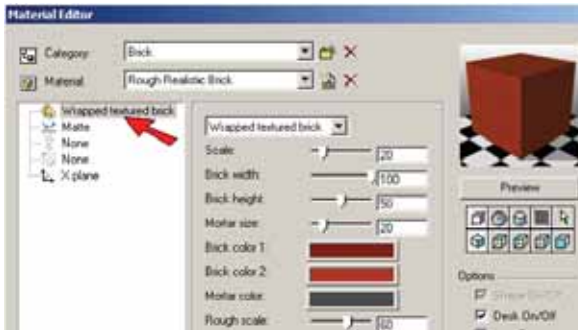
This will show you exactly how the material will appear on the selected object.

*Warning: If the selection arrow is the specified option for the preview object, and no object is selected, you will get a warning that a preview cannot be generated. This will also happen if the selection arrow is the default and you open the Material editor with no object selected.*

Pattern

*NOTE: TurboCAD has several defined patterns, but if you want to apply your own image as a pattern, see "Wrapping an Image" on page 527.*

The first set of parameters in the **Material Editor** is for the material's pattern. For example, look at a Brick material. In this example, the pattern is "Wrapped texture brick."



This pattern is defined by dimensions and colors specific to bricks and mortar.

For granite, the pattern parameters are related to mineral colors.



And for wood floors, the pattern parameters are for planks and grooves.



Materials such as glass have no pattern; the pattern is defined as "Plain."



To see the entire list of available patterns, open the drop-down menu. Each type of pattern has its own set of unique parameters.



## Reflectance

This category defines a material's luster: its brightness, polish, dullness, etc. For example, a brick's reflectance is "Matte" - it is flat and does not reflect light.



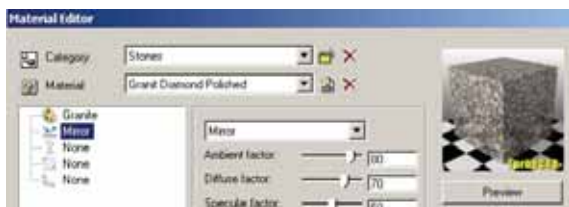
Reflectance of most metals is "Conductor" - a set of parameters designed to simulate how metals interact with light.



Glass has its own Reflectance settings.



Some stone materials have "Mirror" reflectance to simulate shininess.



## Transparency

This category is relevant for transparent glass and plastic materials. For example, "Eroded" transparency simulates a bumpy or uneven glass surface.



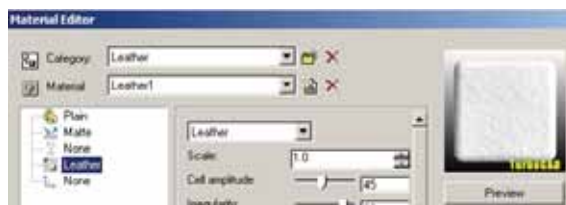
Not all glass or plastic materials have Transparency defined. For example, "Tinted Glass" is smooth, and its transparency is actually defined as part of the "Glass" reflectance.



This is a good example of how complex, and powerful, the TurboCAD materials engine is. You can get similar results by defining parameters in different categories.

## Texture

This category enables you to make a surface appear rough or bumpy. The rendering engine simulates shadows along the surface to create the textured effect. For example, you can define a Leather texture.



“Ice” is defined with a rough texture.



Wrapping

This category defines how a pattern or image wraps around surfaces of a 3D object.

For materials, such as Granite, the wrapping is defined as “None,” which pastes the pattern without distortion onto faces along the three major planes.



The “UV” wrapping is similar to “None” except that you can specify how the pattern is scaled.



With “Auto Axis” the pattern is oriented according to the WCS, regardless of where the faces of the 3D object are located and oriented.



You can also specify the axis along which the pattern will be projected.

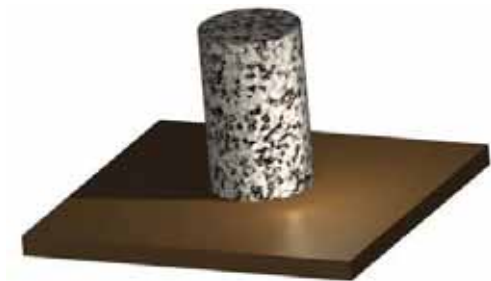


Creating a New Material

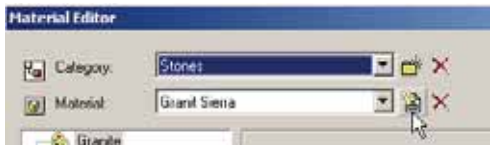
You can create a new material in two ways: modifying an existing material, or creating a new one from scratch.

Modifying an Existing Material

In this example, the cylinder has the material “Stones - Granite Sierra” and the plate below is “Metals - Antique Brass.”



1. To create a new material from an existing material, find the existing material in the **Material Editor**, Then click **New Material**.



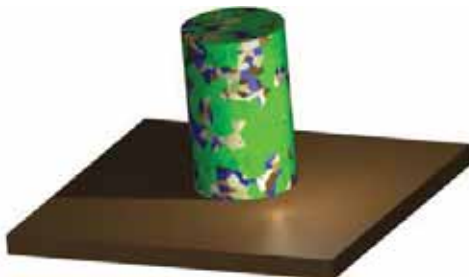
2. Assign a name for the new material. It will be placed in the same category as the original.



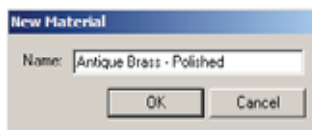
3. In the Pattern category, increase the **Scale** (which is the size of the repeated image, in drawing units), and change the various mineral colors.



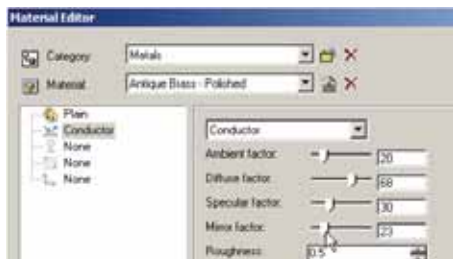
This is the result - a larger scale granite with different colors.



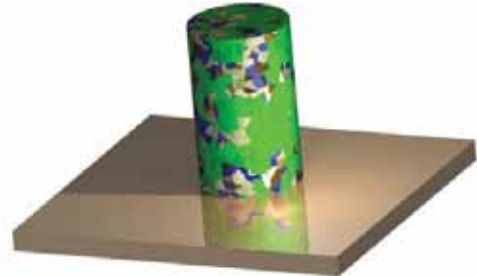
4. For the plate, make a new material from the antique brass.



5. In the Reflectance category, increase the **Mirror Factor**.



This changes the plate surface to reflect the cylinder.

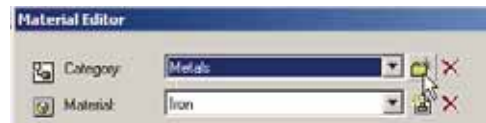


## Wrapping an Image

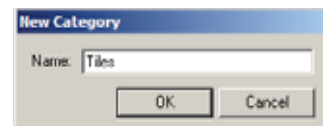
If you have your own image you want to use as a material, this is defined in the Pattern category. This example uses a ceramic tile:



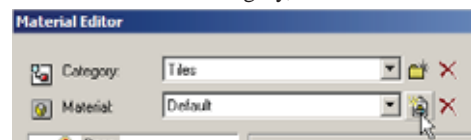
1. In the **Material Editor**, click **New Category**.



2. Assign the name "Tiles."



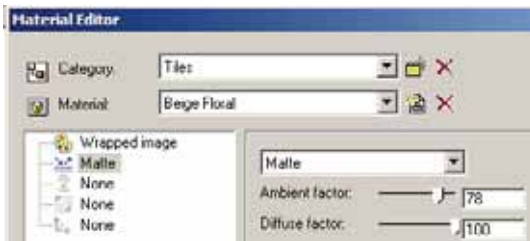
3. With "Tiles" as the category, click **New Material**.



- Assign a material name. For the Pattern category, select “Wrapped Image” and click the **Browse** icon next to **File name**.



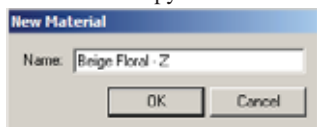
- For Reflectance, use “Matte” with a high Ambience Factor.



- Use “Auto Axis” for Wrapping.



- This material is good for the cylinder, but the material used for the plate should be slightly different. Use **New Material** to create a copy of the tile material.



- The Wrapping here should be “Z plane” so that only the top face of the plate will be covered.



Here is the result with “Auto Axis” applied to the cylinder and “Z-plane” applied to the plate.



- If you want to change the aspect ratio of the tiles on the cylinder, switch to “UV” and increase the “U scale.”

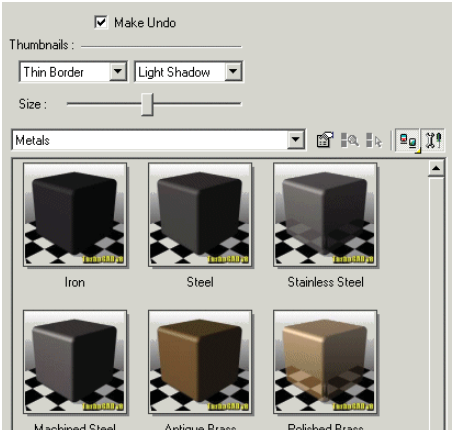


This makes the cylinder’s tiles twice as wide, with the same height.



Materials Palette

Provides easy and fast access to object materials.



All materials that have been defined are displayed here, and can be assigned to objects and edited. Use the **Thumbnail** controls to adjust how the items are displayed, and use the **Size** slider to adjust the item size.

To apply a material to an object, first select the object. Then double-click the palette item to apply it. You can also right-click on a material and select **Set Material**.

To remove a material from an object, select the object. Then in the palette, right-click and select **Set None**.

To find objects that have a specific material, select the material in the palette, then click **Select by Material**.



To find the material of an object, select the object and then click **Find Material of Selection**.



To edit a material, select the item in the palette, then right-click and select **Edit Material**. Or double-right-click on the item, or click **Edit Material**.



To switch the luminance display from thumbnails to list format, click **View Thumbnails** and select a different view.



You can hide or show the palette fields and toolbars by clicking **Show / Hide Options**.



Right-click on the Material palette will open a local menu that provides a wide array of options.

**Set Material**  
Set None

Drag on Facet

Select By Material  
Find Material of Selection  
Change Materials Category...

☒ Show Options  
☒ Show Toolbar

☒ View Thumbnails  
View List  
View Details  
Arrange Icons ▶

Refresh  
Update Thumbnails  
Rebuild Thumbnails ▶

Load Materials...  
Save Materials...  
Restore Default Materials

Edit Materials...

Dragging and Dropping Materials

Another way to apply materials to object is to select a material in the Material palette, then drag-and-drop it onto the desired object.

Drag on Facet

Selecting the **Drag on facet** setting in the local menu of the **Material** palette will allow you to drag a material to a single facet of an object.



## Loading and Saving Materials

If you want to save luminance properties for future use, select **Save Materials**, or right-click in the Materials Palette and select **Save Materials**. Use the categories on the right side of the **Save** window to specify which materials you want to save. Material data will be written to a \*.mat file, located in the **Materials** folder.

Any new material you define will be automatically saved in the TurboCAD file as well.

To load a materials \*.mat file, select **Load Materials** from the **Options / Materials** menu, or from the local menu of the palette.

## Environments

As part of the **Drawing Setup**, you can specify render options that affect the appearance of the background and foreground, as well as overall tone. These effects can be seen only when using **Quality** render.



Quality rendered view with no Environment

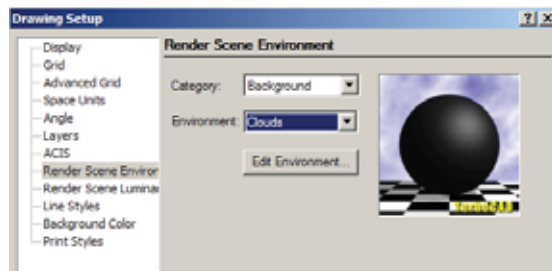


Render Scene Environment:  
Background = Clouds, Foreground = Snow

Several environments and categories are provided by default. You can create a new environment via the **Environments Editor** (see "Editing Environments" on page 531).

## Environments Properties

You can access environment properties via the **Render Scene Environment** page of the **Drawing Setup (Options / Render Scene Environment)**, or by using the Environments Palette (see "Environments Palette" on page 532).



### Category:

- **Background:** How the area behind the model appears, such as graduated colors or clouds.
- **Cubical Maps:** Maps a photo image.
- **Foreground:** How the area in front of the model appears, such as snow or fog.

- **Images:** Applies an image as the model background. The graphic can be in \*.bmp, \*.jpg, \*.tif, or \*.lwi format.
- **Plain:** Applies a plain color as the model background.
- **Tone:** Brightens the model or moves the perspective up or down.

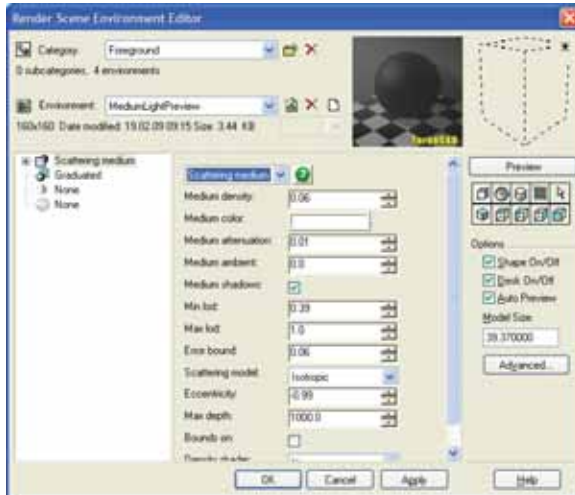
**Environment:** This field depends on the category. For example, if the category is **Plain**, several colors will appear.

**Edit Environment:** Opens the **Environments Editor**. See "Editing Environments" on page 531.

## Editing Environments

**Menu:** Options / Environments / Edit Environment

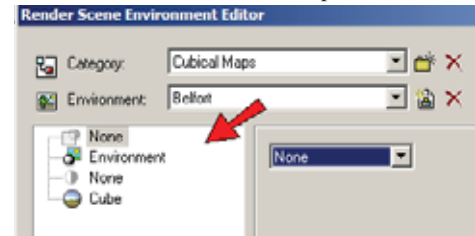
The **Render Scene Environment Editor** can also be accessed from the **Render Scene Environment** page of the **Drawing Setup**. You can also access it from the Environments Palette; select the item in the palette, then right-click and select **Edit Environments**. Or double-right-click on the item, or click **Edit Environments**.



To create a new environment, click **New Environment**. You can also define a new category, or add a new environment to an existing category. You can also click **Delete** to delete an environment or category.

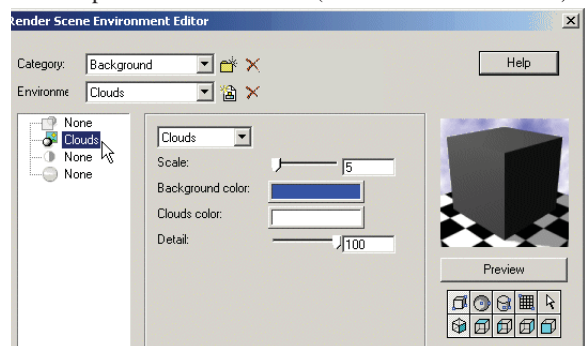
*TIP: You can also create a subcategory within an existing category. Click **New Category**, and use the format "Category\Subcategory."*

Each environment consists of four categories: Foreground (fog, light scattering, etc.), Background (clouds, planes, images, etc.), Tone (Brighten Up or Scale), and Global Environment (Cube, Panorama, etc.). These categories are found on the left side of the **Render Scene Environment Editor**, and each has its own relevant parameters.

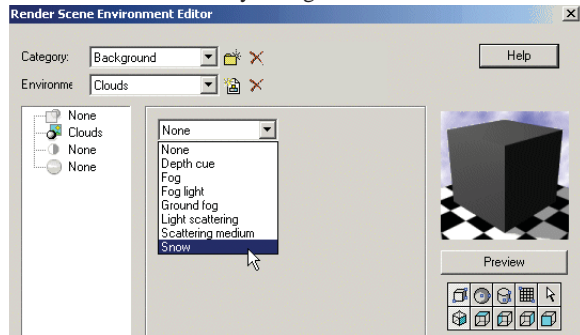


In the example above, the environment **Ground Fog** has two sets of parameters: **Ground Fog** (fog parameters) and **Graduated** (top and bottom colors). Other environments have different parameters.

If you switch to the background environment **Clouds**, you can edit parameters for **Clouds** (color and scale of clouds).

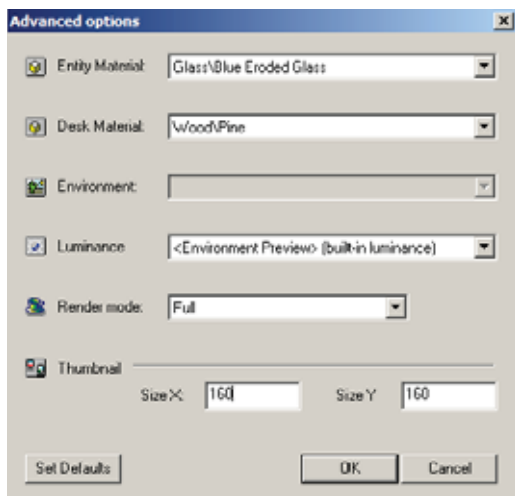


For any environment you can add set of parameters to the existing defaults. Highlight any **None** page and select a set of parameters from the dropdown list. In this example, you can add snow to a cloudy background.



In the **Preview** section, you can change the shape and orientation of the preview object and desk (the surface beneath the object). Under **Options**, you can toggle the display of the preview object or desk. If **Auto Preview** is checked, the preview will update each time a change is made (otherwise you must click the **Preview** button).

Click **Advanced** to change the material of the preview object or desk. You can also change the luminance, rendering type, and thumbnail size.

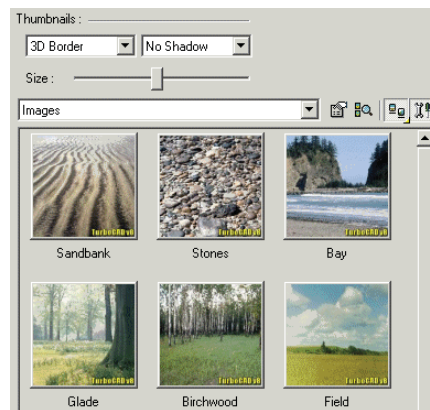


You can see the new materials in the **Preview** window.



## Environments Palette

Provides easy and fast access to environments



All environments that have been defined are displayed here, and can be assigned to your drawing and/or edited. Use the **Thumbnail** controls to adjust how the items are displayed, and use the **Size** slider to adjust the item size. Check **Make Undo** to add luminance to the **Undo / Redo** history of the model.

To apply an environment to the drawing, double-click the palette item to apply it. You can also right-click on a luminance and select **Set Environment**.

To remove an environment from a drawing, right-click in the palette and select **Set None**.

To find the environment of the drawing, click **Find Environment**.





To edit an environment, select the item in the palette, then right-click and select **Edit Environment**. Or double-right-click on the item, or click **Edit Environment**.



To switch the environment display from thumbnails to list format, click **View Thumbnails** and select a different view.



You can hide or show the palette fields and toolbars by clicking **Show / Hide Options**.



## Loading and Saving Environments

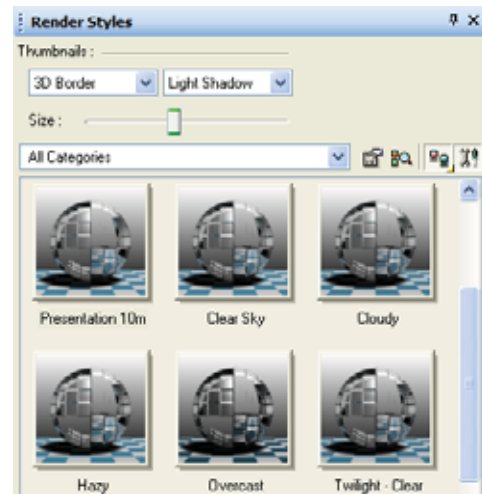
If you want to save environments for future use, select **Save Environments**, or right-click in the Environments Palette and select **Save Environments**. Use the categories on the right side of the **Save** window to specify which environments you want to save. Environment data will be written to a \*.dat file, located in the **RenderSceneEnv** folder.

Any new environment you define will be automatically saved in the TurboCAD file as well.

To load an environment \*.dat file, select **Load Environments** from the **Options / Environments** menu, or from the local menu of the palette.

## Render Styles Palette

Provides easy and fast access to render styles.



All render styles that have been defined are displayed here, and can be assigned to objects and edited. Use the **Thumbnail** controls to adjust how the items are displayed, and use the **Size** slider to adjust the item size. Check **Make Undo** to add luminance to the **Undo / Redo** history of the model.

To apply a render style double-click the palette item. You can also right-click on a material and select **Set Render Style**.

To deselect all the render styles, in the palette, right-click and select **Set None**.

To find the current render style click **Find Render Style**.



To edit a render style, select the item in the palette, then right-click and select **Edit Render Style**, or click **Edit Render Style**.



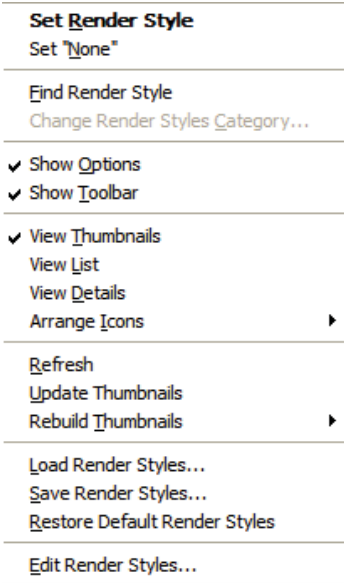
To switch the render style display from thumbnails to list format, click **View Thumbnails** and select a different view.



You can hide or show the palette fields and toolbars by clicking **Show / Hide Options**.



Right-click on the Render Style palette will open a local menu that provides a wide array of options.



### Loading and Saving Render Styles

If you want to save Render Style properties for future use, select **Save Render Styles**, or right-click in the Render Styles Palette and select **Save Render Styles**. Use the categories on the right side of the **Save** window to specify which render styles you want to save. Render style data will be written to a \*.dat file, located in the **Render Styles** folder.

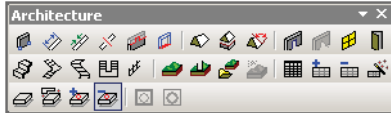
Any new render style you define will be automatically saved in the TurboCAD file as well.

To load a render style \*.dat file, select **Load Render Styles** from the **Render Styles** menu, or from the local menu of the palette.

# 13 Architecture Tools

TurboCAD provides specialized tools for architectural drawings, allowing you to quickly create and edit walls, window and door openings, and roofs.

You can display the **Architecture** toolbar by right-clicking in any toolbar area and selecting **Architecture**.



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NOTE: The **Tools / Architecture** menu also contains **Makers**. See "Drawing Markers" on page 354.

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NOTE: To get started quickly with architectural tools try using the *Architectural* template. It has many of the features and settings for Architectural objects and styles preset.

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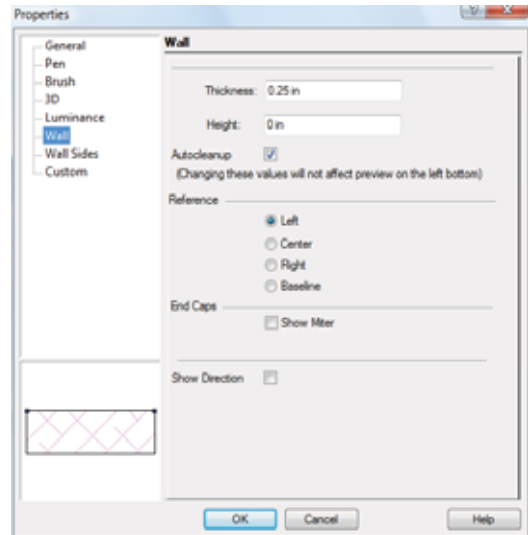
## Wall Tools

The **Wall** tools help you quickly work out a wall design, by creating smart objects. Walls heal at intersections, and they can take block insertions that automatically align correctly.

The typical sequence to create a floor plan starts with a wall layout. When the walls have been drawn, insert doors, windows and/or openings. After that, you can adjust the wall layout by moving or rotating any wall. When you adjust a wall, the walls that are attached to it will adjust to it. Any inserted doors, windows, and openings also adjust.

## Wall Properties

Options for wall alignment, thickness, end caps, and direction.



**Thickness:** The thickness of the wall (distance between the double lines).

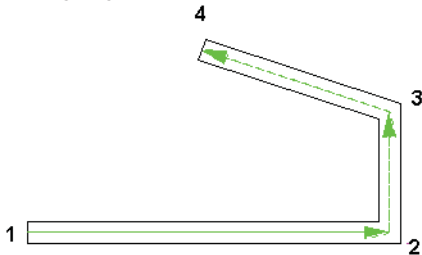
**Height:** The vertical height of the wall.

**Autocleanup:** This option sets whether the wall will automatically cleanup (heal) at intersection with other walls. This can be useful if you want to show the overlap between walls.

**Reference:** Choose whether to draw the wall by its center, left side, right side, or Base line. Right and left sides are determined by "facing" toward the start point. This setting affects how the object is snapped, and where the nodes are located (for the **Edit Tool**).

**End Caps:** Check **Show Miter** to close the ends of a wall.

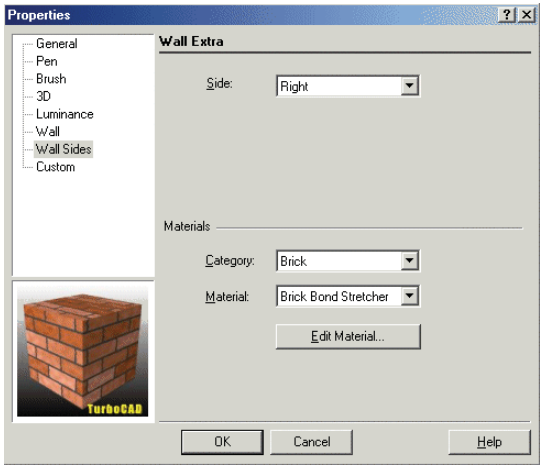
**Show Direction:** Displays a wall's direction, from its start point to its end point. This is useful when defining end caps, and when assigning materials to different sides of the wall.



NOTE: The location of the directional arrow depends on which **Reference** option was used (Left, Right, or Center.)

Wall Side

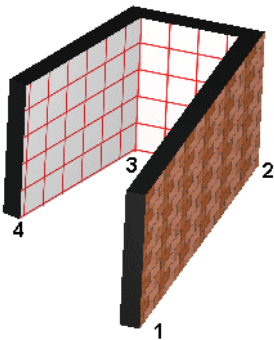
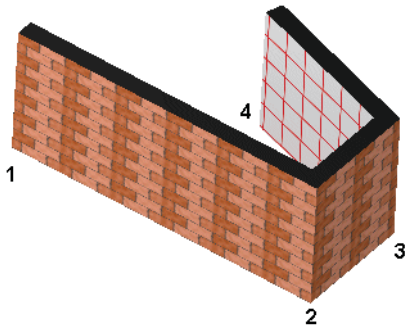
Enables you to assign different materials to either side of a wall. The wall must have height, which is defined by entering a **Thickness** value in the **3D** page of the wall's **Properties**.



**Side:** **Left** and **Right** depend on the wall's direction, assuming you are standing at the start point and facing the end point. To show the directional arrow, check **Show Direction** on the **Wall** page of the **Properties**.

For details on materials, see "Materials" on page 522.

In this example, the walls were created in the order shown. The **Right** sides of the walls have a brick material, and the **Left** sides have a grid pattern.

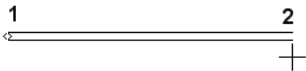


Inserting Walls



Creates segments of double lines that represent wall layouts. The difference between this tool and the **Double Line** tools is that walls automatically heal at corners and intersections. In addition, you can attach walls to existing walls which will also heal.

- 1. Select two points to draw the first wall segment.



**TIP:** Use **Shift** to activate **Ortho** mode if you want to draw horizontal and vertical walls.

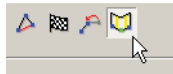
2. Select the endpoint for the next wall segment. The corner is automatically healed; no intersection lines are shown.



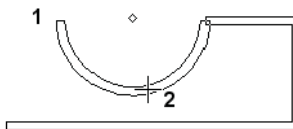
3. Select more wall endpoints as needed.



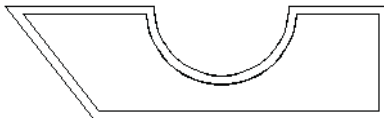
4. To add a circular wall segment, select **Arc Wall** from the local menu or Inspector Bar.



5. To create the arc segment, first select the endpoint, then select a point along the circumference.



6. To return to linear walls, turn off **Arc Wall**. Continue selecting wall endpoints. Select **Finish** from the local menu or Inspector Bar. If you want to close the wall, select **Close** from the local menu or Inspector Bar.



Any wall segment can be deleted or moved like any other object.

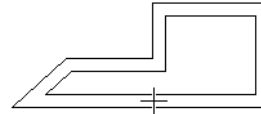
## Linking and Healing

Walls have two aspects which define how they interact with one another: Linking and Healing. Linking means that walls are associative (attached). If you move or modify a wall, walls that are linked to it will modify themselves to remain attached. Healing means that the intersection of walls is cleaned to show a joint without overlapping elements.

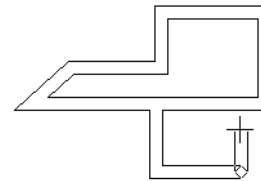
## Attaching Walls

Any wall you add to an existing wall will be automatically healed at their intersections.

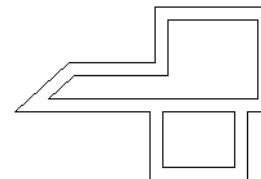
1. Click inside an existing wall at the point where you want to add another wall.



2. Select segment endpoints.



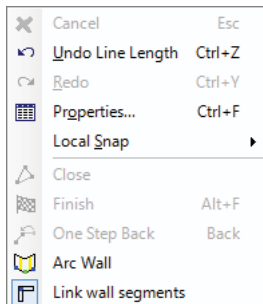
3. If you extend a segment to an existing wall, this intersection will be healed as well.
4. If you extend a segment using the T-Meet tool or Meet 2 Lines tool into an existing wall, this intersection will be healed and linked.



## Link Wall Segments

The Link Wall Segments (local menu) option controls whether walls become linked as they are drawn. It is on by default, but If it is turned off walls will not become linked. You can tell if a wall is linked by selecting it. Any other

segments linked to that wall will be highlighted. Since walls automatically linked when you draw then in proximity this option can be useful for drawing a wall close to another wall without having them link. For example, this can be useful for drawing cubicle walls in an office setting.



## Drop Link between Wall Segments

You can break the link between any two walls segments.

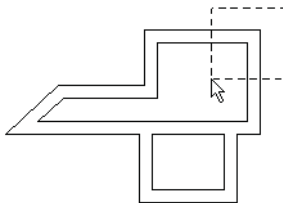
1. Select the two segments
2. Right click to open the local menu and select Drop Link.

## Moving Walls

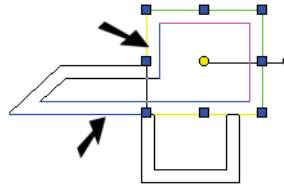
To move a single wall, select it as you would any other object and use the **Select Edit** tools to move, scale, or rotate it. (See "Select Edit" on page 198.)

Moving a wall will affect its adjacent walls.

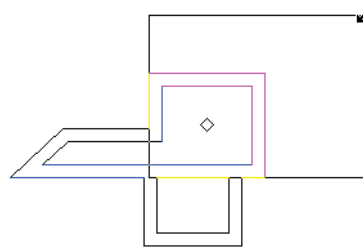
1. In **Select** mode, select the wall you want to move. To select multiple walls, use Shift or drag a selection window.



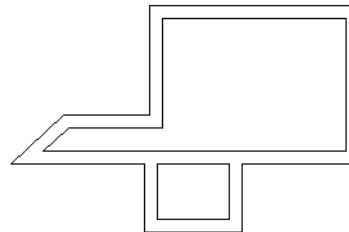
In addition to the walls you selected, adjacent walls are marked in blue, so that you will see all affected walls before performing the edit.



2. Use the **Select Edit** tools to move, rotate, or scale the selected wall or walls.



The affected walls extend or shrink to meet the new walls, but their orientation does not change. If the walls contain inserted windows, door, or other blocks or openings, they will adjust as well.



## Splitting and Joining Walls

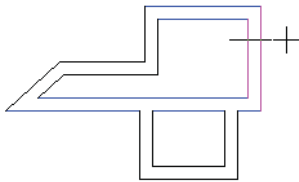
Available in TurboCAD Pro and Platinum only



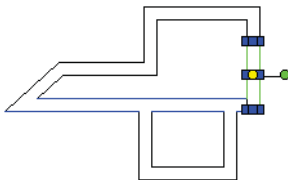
Splits a wall into two separate walls, and joins split walls into one wall. This tool works on straight walls only, not curved walls.

To use the **Split** tool:

1. Select the wall you want to split.
2. Position the cursor where you want to split the wall, and click to split.

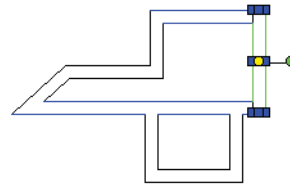


3. Because walls automatically heal, you will not be able to see the split (unless you use end caps). If you select one of the walls, you can see that the original wall was split.



To use the **Join** tool:

1. Select the two segments you want to join. The walls are combined into one wall.



## Editing Walls

The **Edit Tool** can be used to move or resize walls.

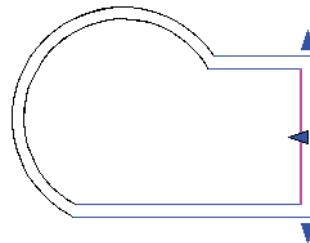



---

NOTE: For details on this tool, see "Edit Tool" on page 219.

---

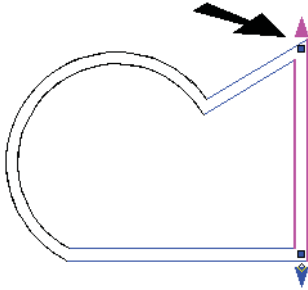
1. This example shows a set of walls with one arc segment. Activate the **Edit Tool** and click the segment you want to change.



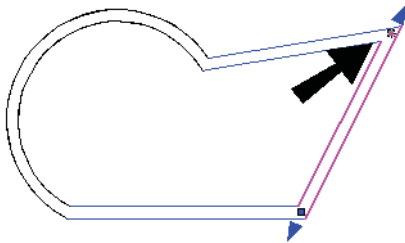
2. Select **Edit Nodes**.



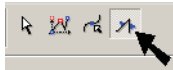
3. With this options, you can move the node at either end. If you click and drag the arrow, you are constrained to the line of the wall.



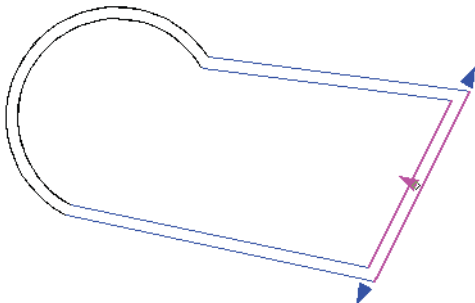
4. If you click and drag a node, you can move it anywhere.



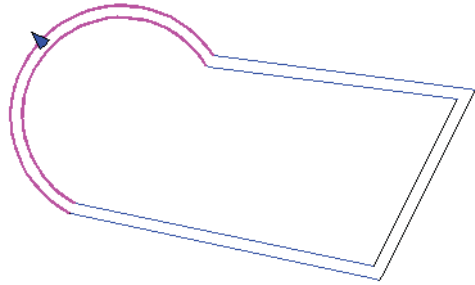
5. Switch to **Edit Segments**.



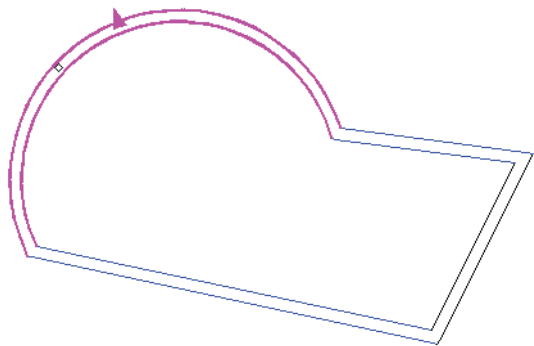
6. Now you can move the entire wall, constrained to its current orientation



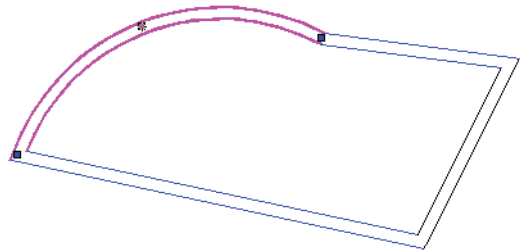
7. You can also change arc segments with the **Edit Tool**.



8. With **Edit Segments**, the arc center stays the same and the arc radius changes.



9. With **Edit Nodes**, the arc endpoints stay the same.





## Wall Dimension

Available in TurboCAD Pro and Platinum only



Specialized architectural dimensioning tool for walls. This tool is used in conjunction with the Style Manager. See "AEC Dimension Styles" on page 599.

## Add Top Wall Modifiers

Available in TurboCAD Pro and Platinum only



Enables you to change the top height of a wall by trimming or extending it to a single-line object.

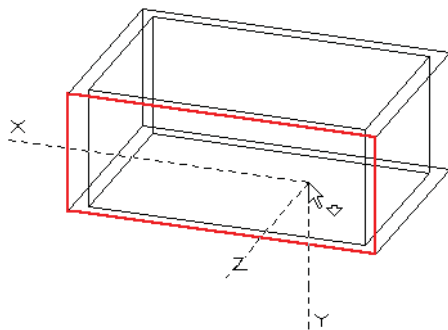
The object can be a line, arc, or polyline. It must be at least as long as the wall, and cannot extend below the bottom of the wall. In addition, it must lie in the same workplane as the wall.

---

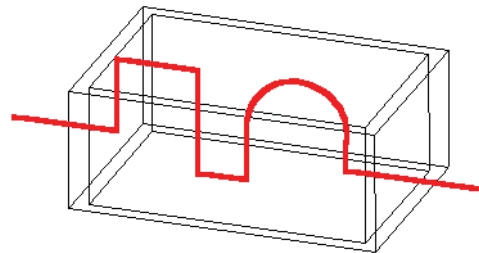
**NOTE:** The way to assign height to a wall is to open its **Properties** to the **3D** page and enter a **Height**.

---

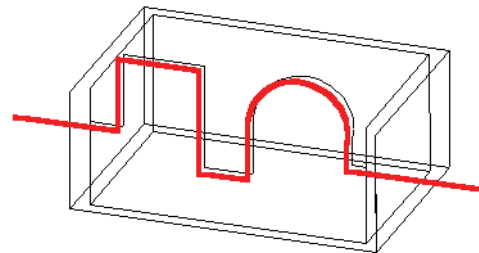
1. Set the workplane to the wall you want to modify. The easiest tool is **By Facet**.



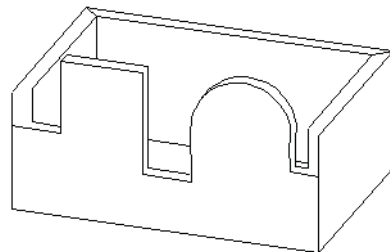
2. Create the object you want to use as a modifier. Be sure that the object meets or extends past the wall ends.



3. Activate **Add Top Wall Modifier**, select the wall to modify, and select the 2D modifier. The wall is trimmed or extended to meet the modifier.



The results may be easier to see in **Hidden Line** mode.




---

**NOTE:** If the modifier length is insufficient, you can edit it. The modifier will be applied once it reaches the required length.

---

If you add another Top modifier, it will cancel the effect of the previous one.

## Add Bottom Wall Modifiers

Available in TurboCAD Platinum only



Enables you to change the bottom height of a wall by trimming or extending it to a single-line object.

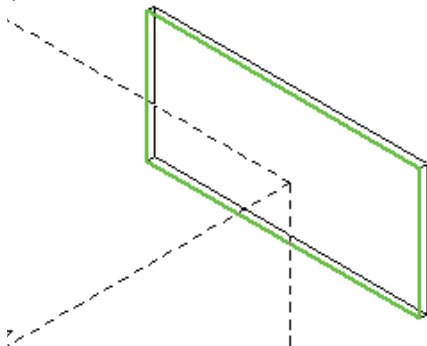
The object can be a line, arc, or polyline. It must be at least as long as the wall, and cannot extend below the bottom of the wall. In addition, it must lie in the same workplane as the wall.

---

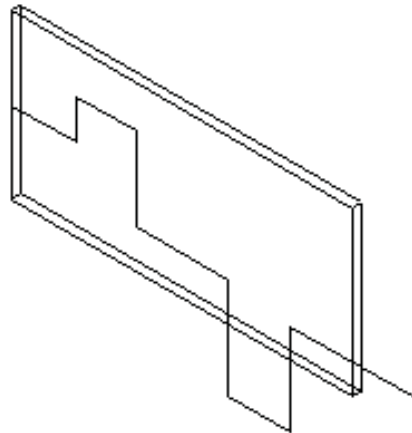
NOTE: The way to assign height to a wall is to open its **Properties** to the **3D** page and enter a **Height**.

---

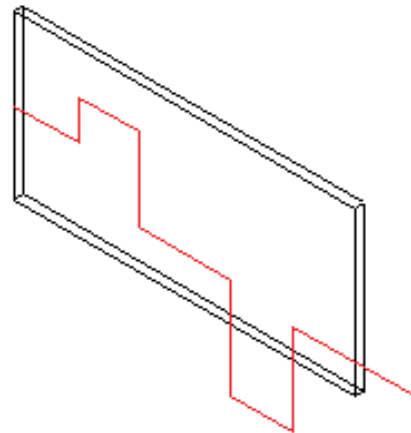
1. Set the workplane to the wall you want to modify. The easiest tool is **By Facet**.



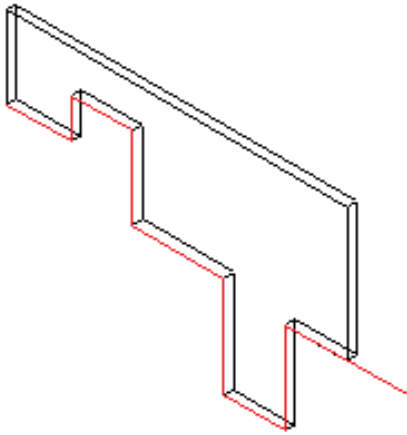
2. Create the object you want to use as a modifier. Be sure that the object meets or extends past the wall ends.



3. Activate **Add Bottom Wall Modifier**, select the wall to modify, and select the 2D modifier. The wall is trimmed or extended to meet the modifier.



The results may be easier to see in **Hidden Line** mode.




---

*NOTE: If the modifier length is insufficient, you can edit it. The modifier will be applied once it reaches the required length.*

---

If you add another Bottom modifier, it will cancel the effect of the previous one.

You can use Top and Bottom modifiers on the same wall.

## Roof Wall Modifier

*Available in TurboCAD Pro and Platinum only*



Enables you to extend the Top height of a wall to meet the roof. This is especially important if you have edited the Slope angle of a roof so that you have an open gable end.

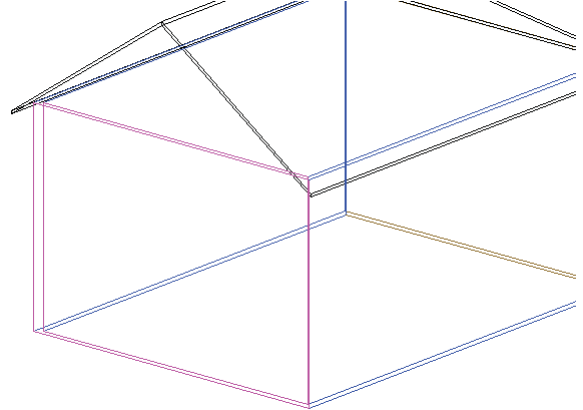
---

*NOTE: The way to assign height to a wall is to open its **Properties** to the **3D** page and enter a **Height**.*

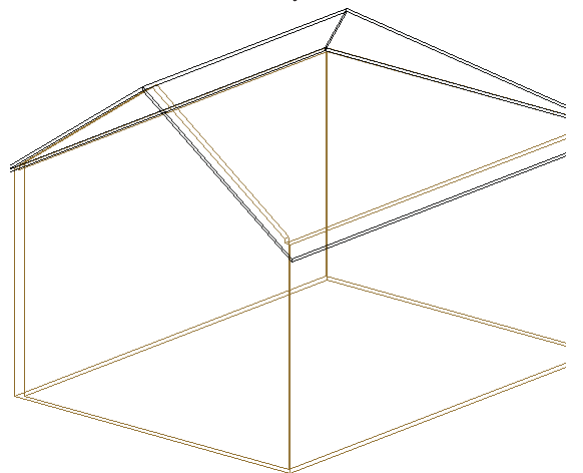
---

1. Select the Roof Wall Modifier tool.

2. Select the wall you want to modify..



3. Select the roof to which you want the wall to extend.



The wall will now extend to the roof.

Top Wall Modifiers will override Roof Wall Modifiers, and Roof Wall Modifiers will override Top Wall Modifiers, but you can use Bottom Wall Modifiers, in conjunction with Roof Wall Modifiers.

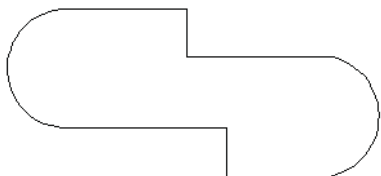
## Convert to Wall

Available in TurboCAD Pro and Platinum only

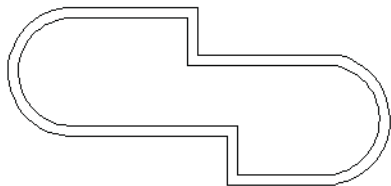


Converts single-line objects into walls. This is useful for creating curved walls.

1. Select the object to convert. You can select lines, arcs, circles, polylines, etc. Multiple objects can be selected.



2. Select **Finish** from the local menu or Inspector Bar. The wall is created with the default thickness, but you can modify its **Properties**,




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NOTE: The way the wall is created in relation to the original object depends on the **Reference** parameter in the **Wall** page of the **Properties** window.

---

## Inserting 2D Blocks in Walls

When a 2D block is inserted into a wall, it rotates to align itself with the wall. The direction of the rotation depends upon the direction in which the wall was drawn; the top of the block will align to the right side of the selected wall. (The right side is determined by facing the wall's start point from the end point.)

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NOTE: This method of inserting blocks applies to 2D blocks, and 2D (plan) representation of walls. To create walls and doors that you can see in 2D as well as 3D, see "Openings" on page 548.

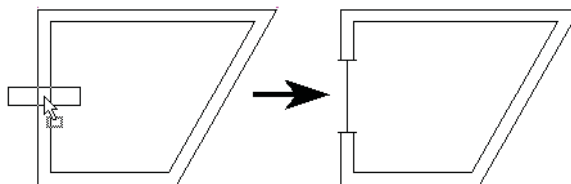
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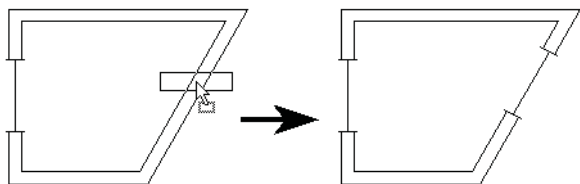
1. Create the 2D representation of the desired insertion (door, window, etc.). Create a block from these objects. You can do this by selecting the objects and dragging them into the Blocks Palette, or by selecting **Format / Create Block**.



For more information on blocks, see "Blocks" on page 291.

2. Drag the block from the Blocks Palette onto the wall at the point where you want to insert it. The block will automatically align itself with the wall, and a cutout will be created.





## Roofs

*Available in TurboCAD Pro and Platinum only*

You can easily add and modify roofs to a building. The roof slope can be constant or you can change the slopes of individual roof faces.

### Add Roof



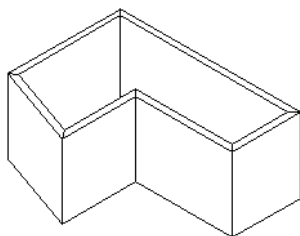
Creates a roof with a constant slope angle.

1. Create the walls of the building. If you do not use the **Wall** tools, you can use any 3D shape, or even a 2D outline.

---

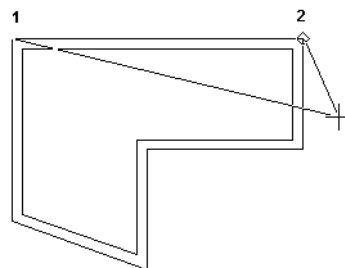
**NOTE:** If you use the **Wall** tool, you can give the walls height by entering a **Thickness** in the **3D** page of the walls' **Properties**.

---

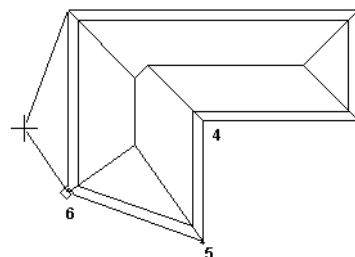


2. If necessary, bring the workplane to the correct level. You can use **By Facet** to set the workplane to the top of the walls.

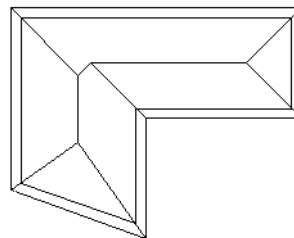
3. Activate **Add Roof**, and trace the outline of the roof.



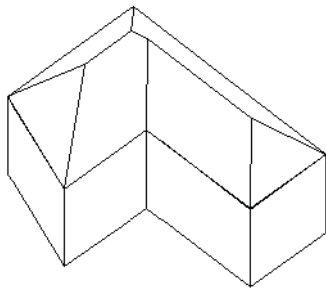
With each corner point you select, the roof adjusts to create a closed form. .



4. After you select the last point, select **Finish** from the local menu or Inspector Bar. The ridge and valley lines are calculated to maintain a constant slope angle.

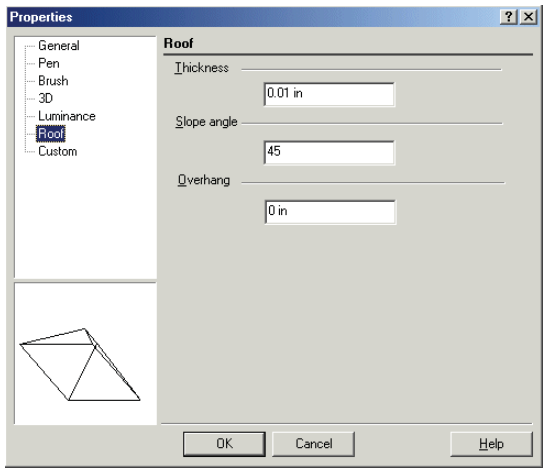


Rotate the view to see the roof.



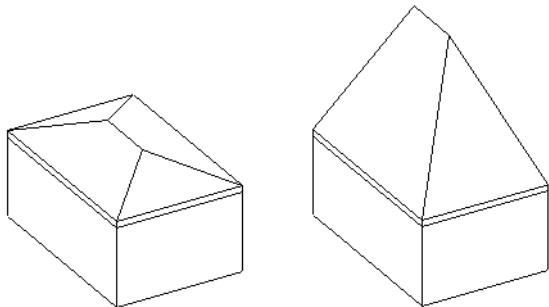
Roof Properties

Thickness, slope angle, and overhang length of the roof.



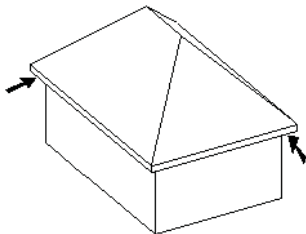
**Thickness:** The thickness of the roof.

**Slope Angle:** The higher the angle, the steeper the roof slope.

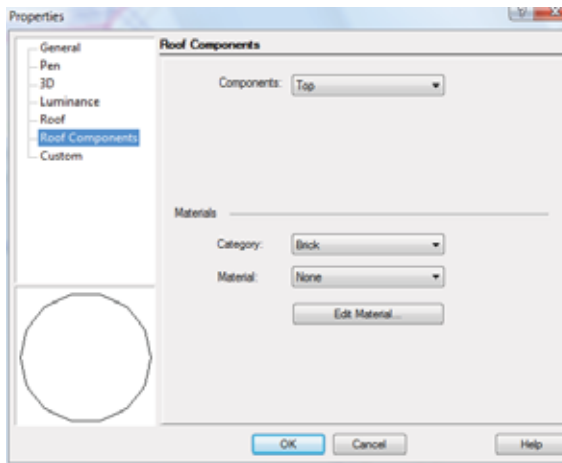


NOTE: This parameter changes the slope of the entire roof. To change the slope of one facet, use the **Edit Slope Angle** tool.

**Overhang:** The distance the roof extends past the defined roof outline.



The Roof Components properties allow for the setting of different Materials to each component of the roof. The components available are as Top, Bottom, Gable and Fascia

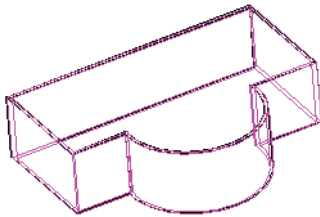


## Add Roof by Walls

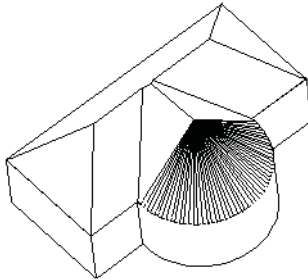


Creates a wall defined by a series of walls.

1. Start with a closed chain of walls, and activate **Add Roof by Walls**.
2. In the Properties, set the **Slope Angle** and **Overhang**.
3. Press Shift and select all walls in the chain.



4. Select Finish to create the roof.

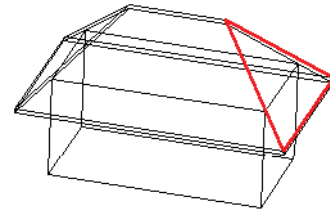


## Edit Slope Angle

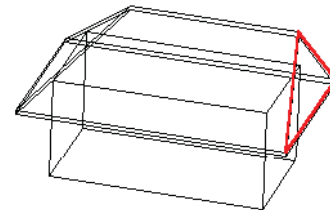


Changes the slope of individual roof faces.

1. Select the roof face whose slope you want to change.



2. Enter the new slope angle in the Inspector Bar.



## Windows and Doors

These two architectural tools enable you to insert “smart” windows and doors into walls. Large libraries for doors and windows are available, and all features and dimensions can be specified.

*The expanded library of windows and doors is available in TurboCAD Platinum only.*

### Window



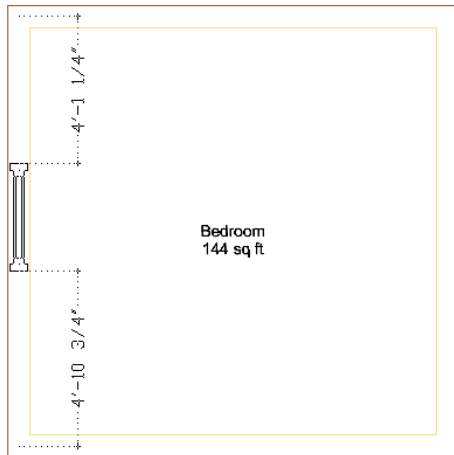
Inserts “smart” windows into walls.

Before inserting windows, you first need to use the Style Manager to define window styles. For the entire process of creating windows, see “Window Styles” on page 571.

For the various types of windows you can insert, see “Window Types” on page 575.

1. Select the tool and hover over a wall.
2. Click on the wall to insert the window.

You can use snaps to exactly place the window. You will also be shown dynamic dimensions when you are inserting the window that indicate the window's distance from the two nearest walls.



Dynamic dimensions will also appear when you use the Edit tool on windows.

## Door



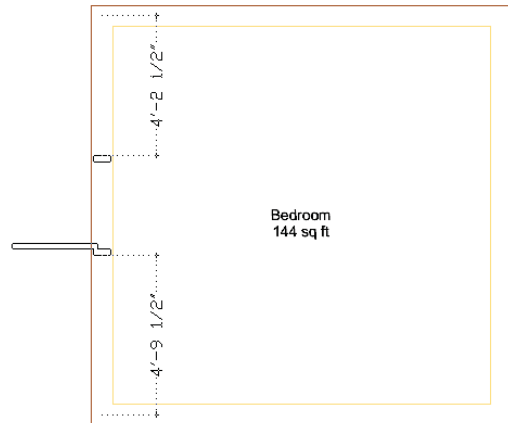
Inserts "smart" doors into walls.

Before inserting doors, you first need to use the Style Manager to define door styles. For the entire process of creating doors, see "Door Styles" on page 579.

For the various types of windows you can insert, see "Door Types" on page 582.

1. Select the tool and hover over a wall.
2. Click on the wall to insert the door.

You can use snaps to exactly place the door. You will also be shown dynamic dimensions when you are inserting the window that indicate the door's distance from the two nearest walls.



Dynamic dimensions will also appear when you use the Edit tool on doors.

## Openings

*Available in TurboCAD Pro and Platinum only*

Enables you to insert a 2D / 3D opening into a wall, representing a window, door, etc.

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**NOTE:** *To insert actual "smart" windows and doors, see "Windows and Doors" on page 547*

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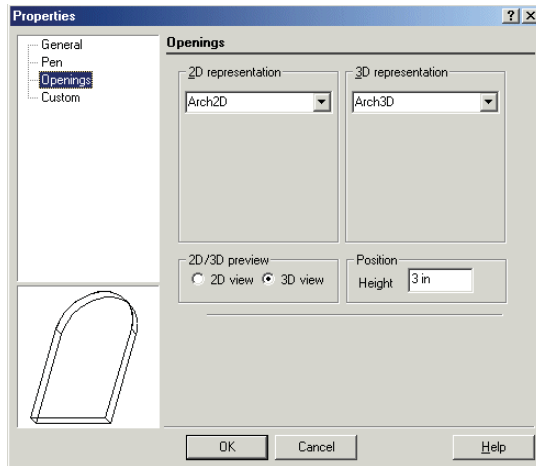
In this tool, each opening is represented by two blocks - a 2D and 3D view, so that the block can be seen in both orthographic and isometric views. The blocks must be created in advance before they can be inserted into a wall. See "Creating a Block" on page 291.

The blocks do not have to be the same thickness as the walls they will be inserted into.



## Opening Properties

Before an opening can be inserted, you must select the 2D and 3D blocks (previously created) that will be used to represent it. This is done in the **Properties** window - opened by right-clicking on the **Insert Opening** icon.



**2D/3D representation:** The blocks that will be used to represent the opening. The 2D block will be used in plan views; the 3D block will be used in all other views.

**2D/3D preview:** The preview is displayed on the left side of the window, so that you can verify the selected block is correct.

**Position / Height:** The distance from the workplane where the opening will be inserted. This is useful for windows that need to be located a certain height from the floor.

---

*TIP: If you want to create and save more than one type of opening, create separate styles (accessed from the **General** page of the **Properties** window). This way you will not have to recreate properties of openings you have already inserted. See "Property Value Presets" on page 84.*

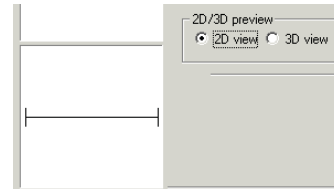
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## Insert Opening

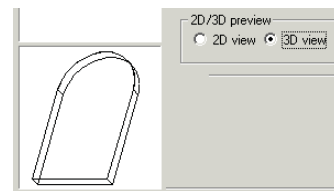


In this example, the opening has the following properties:

- For 2D representation: a block consisting of three lines



- For 3D representation: a block consisting of an arched doorway, created by making a polyline (see "Polyline" on page 138) and extruding it (see "Simple Extrude" on page 390).

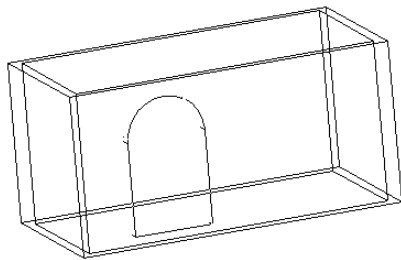


- A value was assigned for **Height**, so that the block will be inserted above the floor.

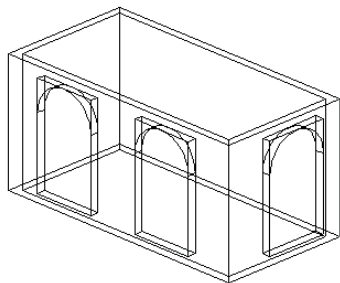
Remember, the blocks have to be created in advance, and are assigned to the opening in the **Opening Properties**.

1. If necessary, change the workplane back to **By World**. You may have changed the workplane while creating one of the blocks, but the workplane must be along the bottom of the walls.
2. Activate **Insert Opening**, and make sure **Anchor to Wall** is selected. (**Anchor to Roof** is similar, except that openings will tilt themselves to fit along a sloped roof.)
3. Drag the opening to one of the walls. The block is dragged by its reference point, which can be changed if needed. See "Relocating a Block Reference Point" on page 295.

The block aligns itself to the wall, and cuts the opening.



4. Drag more blocks onto the walls. The block will change its alignment to fit any wall.

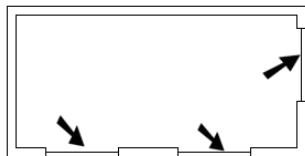



---

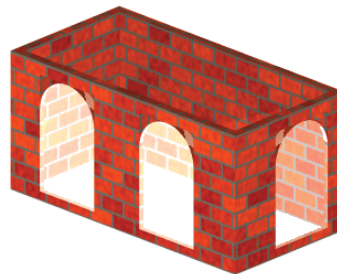
NOTE: This tool only creates **rectangular** cutouts, even if the block is non-rectangular. You can change this by modifying the block. See "Edit Opening Modifier" on page 550.

---

In **Plan** view, the openings are represented by the 2D block.



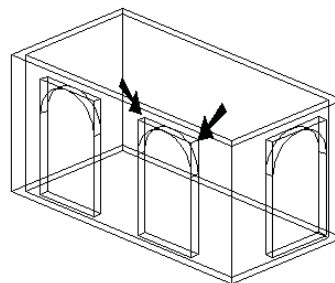
If you assign the 3D block a material such as glass (in the **3D** page of its **Properties**), and assign a brick material for the walls, you can see the results in render mode.



## Edit Opening Modifier



By default, when using the **Insert Opening** tool, an opening is inserted into a wall by creating a rectangular cutout, even when the block is not rectangular, as in the case of an arched doorway or a circular window.

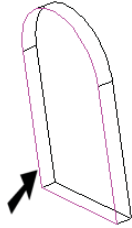


For non-rectangular openings, you can apply a modifier that will define the shape of the cutout.

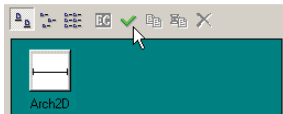
1. In the Blocks Palette, select the 3D block whose cutout you want to modify, and click **Edit Contents**.



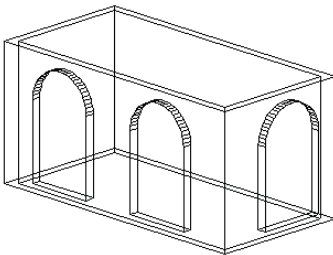
2. Create the closed, 2D object you want to use as the cutout. Change the workplane if necessary. In this example, a polyline was added to the door face - an exact trace of the doorway shape.



3. Activate **Edit Opening Modifier**. If a modifier was previously defined, it will be highlighted.
4. Select the object to be used as the modifier.
5. Select **Finish** from the local menu or Inspector Bar.
6. In the Blocks Palette, click **Finish Edit Content**



The openings now show the revised cutout shape.




---

*NOTE: If you want to continue adding more openings, you will have to change the workplane back to **Workplane by World**.*

---

## Slabs

*Available in TurboCAD Platinum only*

You can add a slab by clicking inside a room of closed walls, or by selecting a closed polyline. You can also create holes in slabs.

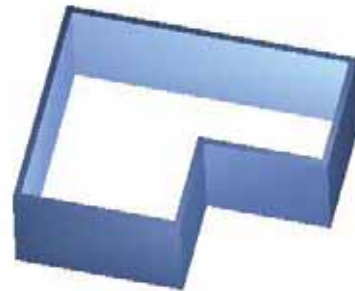
Before creating slabs, you should define slab styles. See "Slab Styles" on page 597.

### Add Slab by Click

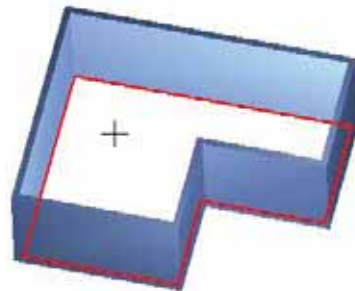


Creates a slab in a room, defined by a closed set of walls.

1. Start with a set of closed walls.



2. Open the tool's **Properties** and define the slab style on the **General** page. See "Slab Styles" on page 597. You can also change a slab's style using the Selection Info palette.
3. Activate **Add Slab by Click** and click anywhere inside the room.



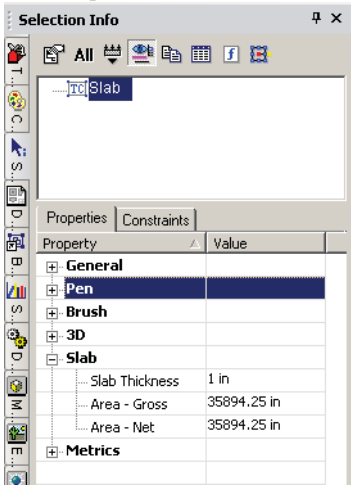
The slab is created inside the room, between the walls.



4. You can also specify to offset the slab from the walls by using the **Offset** field in the Inspector bar.



5. The thickness and area of the slab are listed in the Selection Info palette.



Local Menu Option

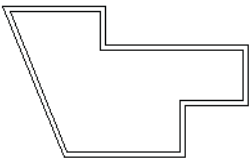
**Use Wall Thickness:** This option can be useful using walls with different widths. If this option is selected, the offset value for each side of the slab will be calculated separately as  $offset = offset * wall\_width$ . In this case offset value is dimensionless quantity and acts as a multiplier of the wall width. I

**Convert to Slab**



Creates a slab inside a closed curve (rectangle, circle, etc.) or polyline. The curves or polylines can be created using double line tools.

1. Start with a closed curve or polyline. This example uses a polyline created with **Double Line Polyline**.



2. Open the tool's **Properties** and define the slab style on the **General** page. See "Slab Styles" on page 597. You can also change a slab's style using the Selection Info palette.
3. Activate **Convert to Slab** and click the polyline. The slab is created inside the polyline.



4. To create walls from the double lines, you can use **Convert to Wall** (see "Convert to Wall" on page 544).

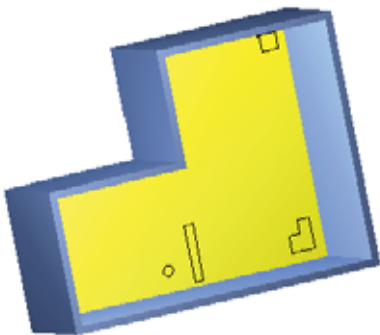


Add Hole

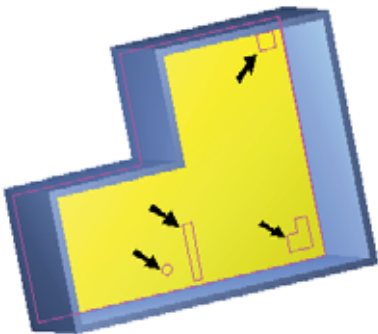


Creates holes in a slab, defined by closed curves or polylines.

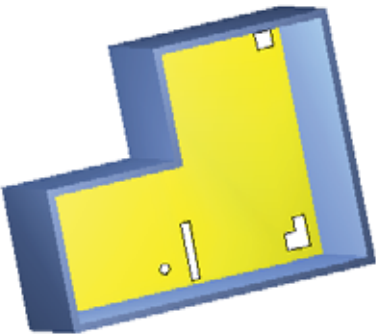
- 1. Start with a slab, and use one or more polylines or curves (rectangles, circles, etc.) to define the holes.



- 2. Activate **Add Hole** and click the slab on which you want to create holes. Then click the object representing the hole. If you want to create multiple holes, press Shift and click all of the hole objects.



- 3. Select **Finish** from the Inspector Bar or local menu. The material inside the hole objects is removed.



In the Selection Info palette, **Area - Gross** represents the area of the slab not including hole removal. **Area - Net** is the total area, with holes removed.

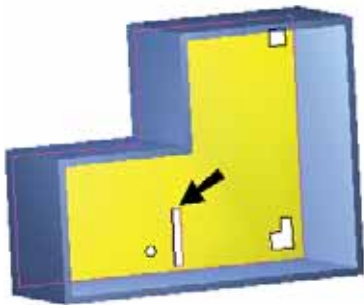
Properties		Constraints	
Property		Value	
General			
Pen			
Brush			
3D			
Slab			
Slab Thickness		1 in	
Area - Gross		35894.25 in	
Area - Net		35000.01 in	
Metrics			

Delete Hole

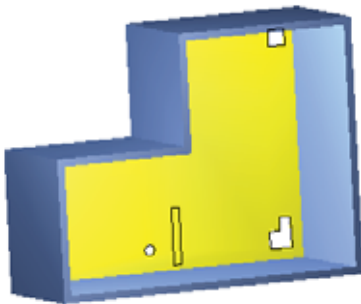


Removes holes already defined in a slab.

1. Start with a slab that has one or more holes defined. Activate **Delete Hole** and click the slab. Then click the object or objects whose hole you want to remove. If you want to remove more than one hole, press Shift and click multiple objects.



2. Select **Finish** from the Inspector Bar or local menu. The hole is removed.



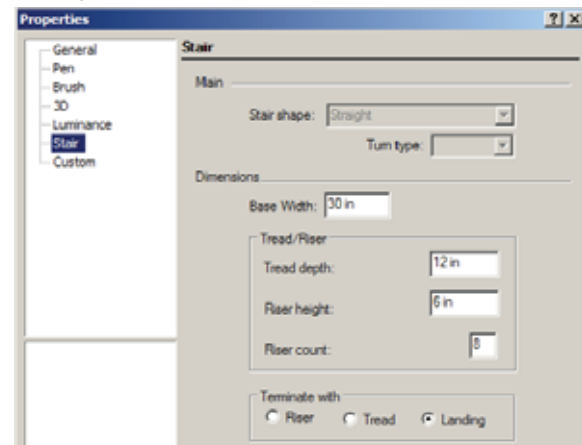
## Stairs

These tools enable you to create four types of staircases: straight, spiral, multi-landing, and U-shaped.

*Available in TurboCAD Pro only. Spiral, Multi-Landing, and U-Shaped stairs are Available in TurboCAD Platinum only.*

## Stair Properties

The **Stair** page of the **Properties** contains parameters for defining the size of treads and risers.



**Stair shape** and **Turn type**: These vary depending on the type of stair you are creating. These options are explained for each stair type.

**Base Width**: The width of the stairs. You can define this here, or when defining the stair geometry.

**Tread depth**: The horizontal depth of each step.

**Riser height**: The vertical height of each step.

**Riser count**: The number of stairs. You can define this here, or when defining the stair geometry.

**Terminate with**: Defines what will appear at the top of the stairs.

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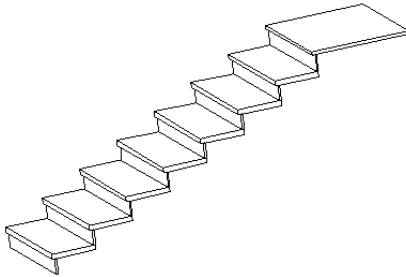
*NOTE: Other stair properties, such as tread and riser thickness, nosing length, and materials can be set using the Style Manager. See "Stairs Styles" on page 602. Once you have styles defined, you can set a stair's style in the **General** page of the **Properties**.*

---

## Straight Stair



Inserts a straight line of stairs.



The first two clicks define the stair width, and the third defines the length. After the stairs are created, you can use the **Properties** to change the width, number of risers, etc.

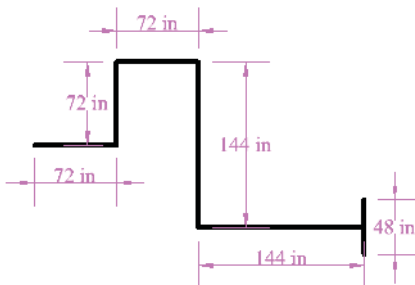
## Multi Landing Stair

Available in TurboCAD Platinum only.

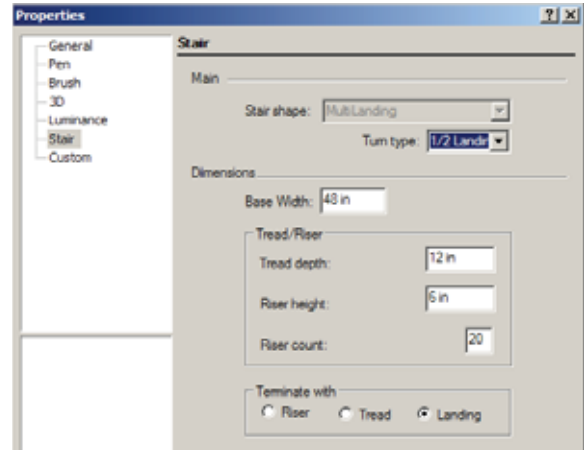


Creates two or more lines of stairs.

- For reference, it may be helpful to lay out the stairs using lines in **World Plan**. In this example, the right-most vertical line represents the width of the stairs, and the other lines represent the path of the stair case.

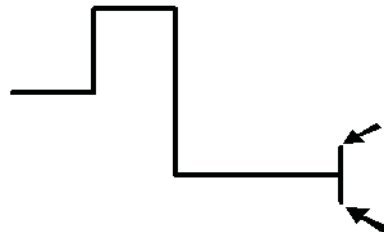


- Activate **Multi Landing Stair** and open the **Properties** to the **Stair** page. This example shows **1/2 Landing**. The remaining parameters control the vertical and horizontal dimensions of each stair.

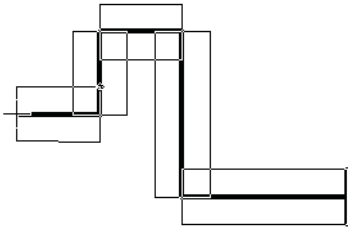


**NOTE:** Other stair properties, such as tread and riser thickness, nosing length, and materials can be set using the **Style Manager**. See "Stairs Styles" on page 602. Once you have styles defined, you can set a stair's style in the **General** page of the **Properties**.

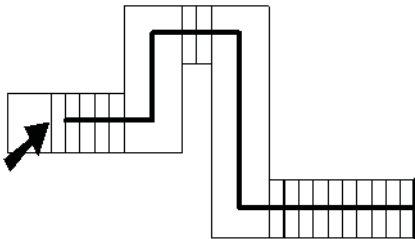
- The first two clicks define the width of the stairs, or you can use the Inspector Bar. (If this value is different than the **Base Width** set in the **Properties**, then **Base Width** will be overridden. However, you can always change this value later in the **Properties**.)



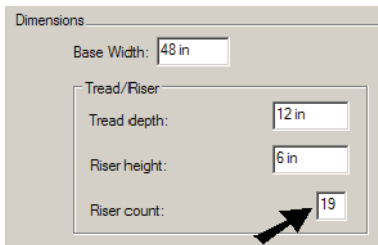
4. The next clicks, or values in the Inspector Bar, define the path of the stairs. This example uses five segments.



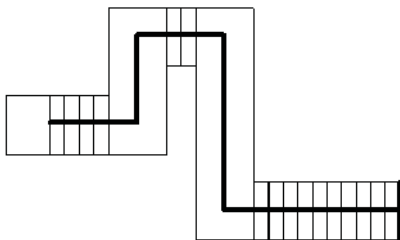
5. Select **Finish** to create the stairs. In **World Plan**, you can see whether the number of stairs is correct. In this example, there is one stair too many.



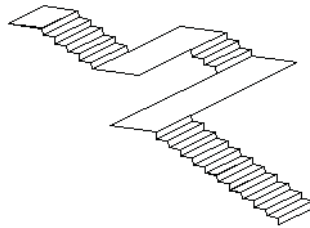
6. To correct this, open the stair's **Properties** and set the correct **Riser count**.



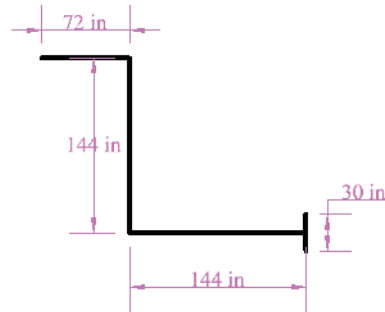
Now the number is correct.



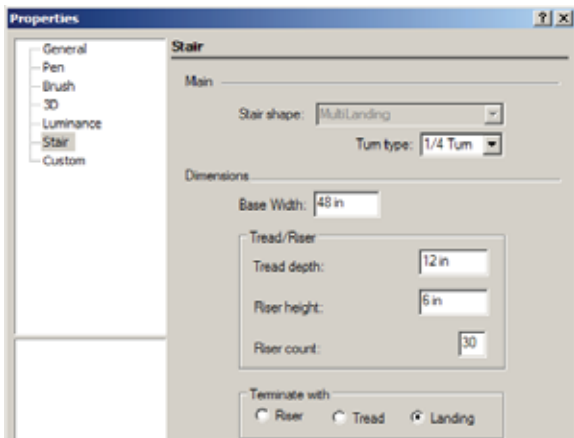
7. In Isometric view, this is the **1/2 Landing** staircase. The second, fourth, sixth, etc. segments of the stair path are created as landings.



8. For the other turn types, each segment is created as a staircase. So start with a stair line like this, with fewer segments.

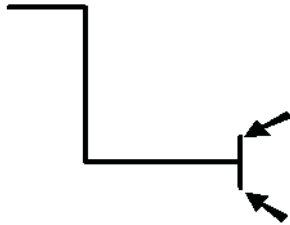


9. Select **1/4 Turn**, and set the **Riser Count** to an approximately accurate value (it can always be corrected later).

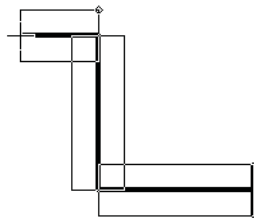




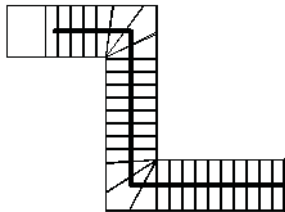
10. As before, set the width of the stairs.



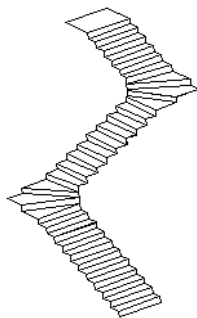
11. Then select the stair path.



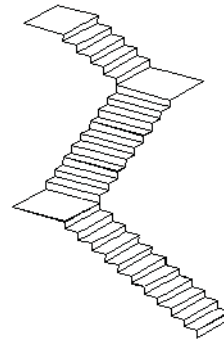
12. Select **Finish**, and adjust the **Riser Count** if necessary.



This is the **1/4 Turn** staircase in Isometric view. There are no landings; the stairs proceed along the turns.



This would be the result if you selected **1/4 Landing**. Each segment is a staircase, and square landings are created between each segment.



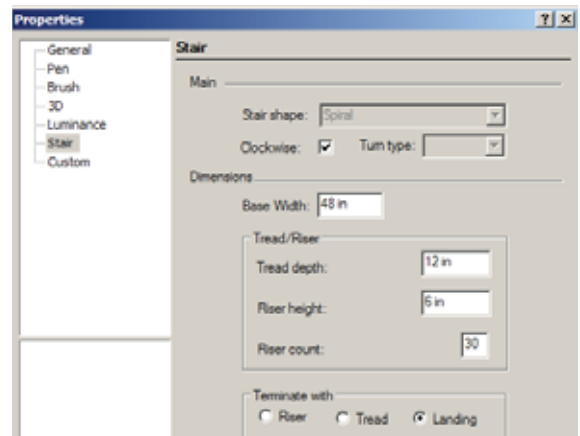
## Spiral Stair

Available in TurboCAD Platinum only.



Creates a spiral staircase.

1. Activate **Spiral Stair** and open the **Properties** to the **Stair** page. This example will be **Clockwise**. The remaining parameters control the vertical and horizontal dimensions of each stair.

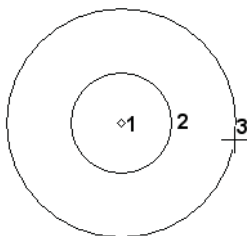


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NOTE: Other stair properties, such as tread and riser thickness, nosing length, and materials can be set using the *Style Manager*. See "Stairs Styles" on page 602. Once you have styles defined, you can set a stair's style in the **General** page of the **Properties**.

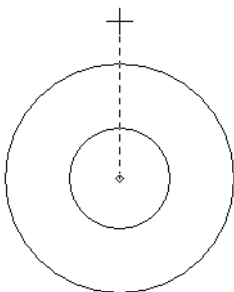
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- The next three clicks define the plan dimensions of the stairs. The first sets the center, the second sets the interior radius, and the third sets the exterior radius. These values can also be set in the Inspector Bar.

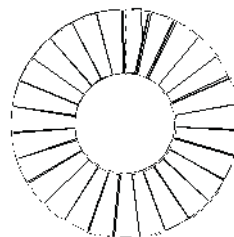


The space between the interior and exterior radius is the width of the stairs. If this value is different than the **Base Width** set in the **Properties**, then **Base Width** will be overridden. However, you can always change this value later in the **Properties**.

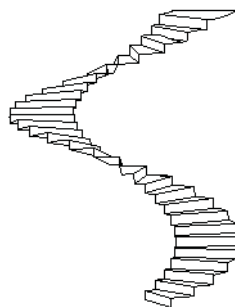
- Next, set the angle from the circle center, where the first stair will be.



The spiral staircase is created.



This is the clockwise spiral staircase in Isometric view. The height of the staircase is the **Riser Count** times **Riser Height**, both of which can be adjusted in the **Properties**.



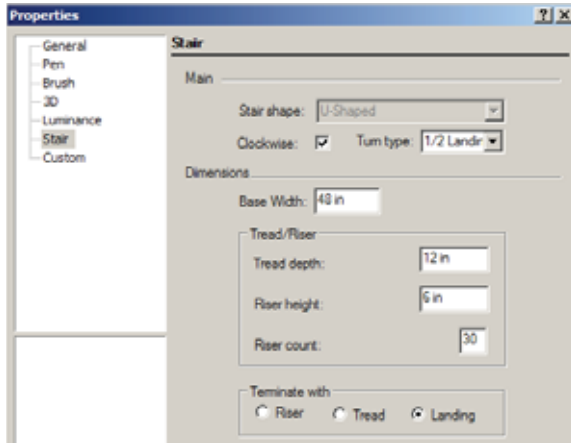
### U-Shaped Stair

Available in TurboCAD Platinum only.



Creates a line of stairs that doubles back on itself.

1. Activate **U-Shaped Stair** and open the **Properties** to the **Stair** page. This example will be **Clockwise** with a **1/2 Landing**. The remaining parameters control the vertical and horizontal dimensions of each stair.



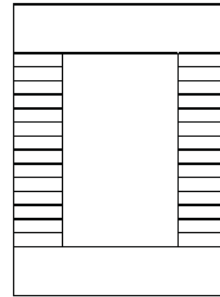
NOTE: Other stair properties, such as tread and riser thickness, nosing length, and materials can be set using the Style Manager. See "Stairs Styles" on page 602. Once you have styles defined, you can set a stair's style in the **General** page of the **Properties**.

2. The next three clicks define the plan dimensions of the stairs. The first two clicks sets the stair width and angle. The third click sets the offset - this is the distance between the exterior edge of one stair run and the interior edge of the other stair run. These values can also be set in the Inspector Bar.

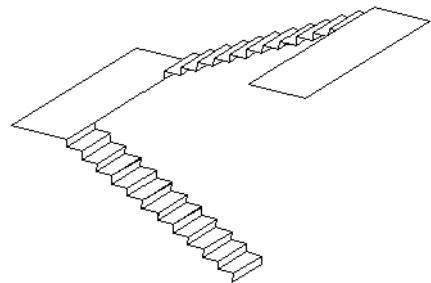


The space between the first two clicks is the width of the stairs. If this value is different than the **Base Width** set in the **Properties**, then **Base Width** will be overridden. However, you can always change this value later in the **Properties**.

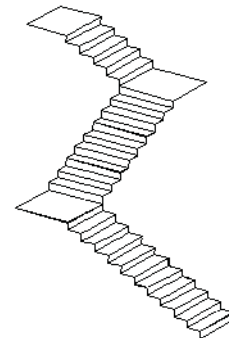
The U-shaped staircase is created.



This is the clockwise staircase in Isometric view. The height of the staircase is the **Riser Count** times **Riser Height**, both of which can be adjusted in the **Properties**. Each stair run has half the **Riser Count**, and a landing runs between them.



This would be the result if you selected **1/2 Turn**. Each segment is a staircase, and there are no landings. The total number of risers is divided among the three sets of stairs.

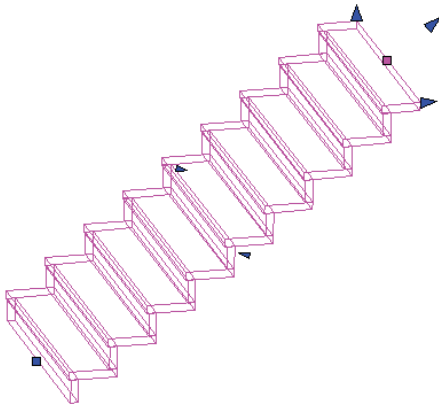


## Editing Stairs

The **Edit Tool** can be used to move or resize walls.



1. This example shows a stair. Activate the **Edit Tool** and click the stair you want to change.



2. If you click and drag a node, you can move it anywhere.
3. The Inspector bar gives you the option to apply a specific delta x and y for any selected node. It also gives the ability to edit the Height of the stairs.

Delta X	Delta Y	Height
0 in	0 in	1.8 in

## Railings

Available in TurboCAD Pro and Platinum only



Inserts a linear railing or a railing along a staircase. This tool works in conjunction with the Style Manager. See "Rail Styles" on page 604.

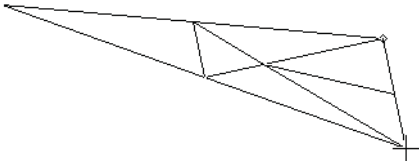
## Terrain

Available in TurboCAD Pro and Platinum only

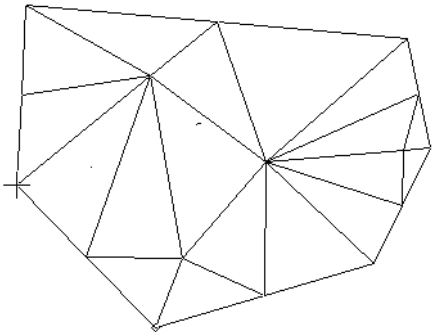


Creates a topographical terrain, represented by triangulated network. You can create a terrain from scratch, or import coordinates from a file.

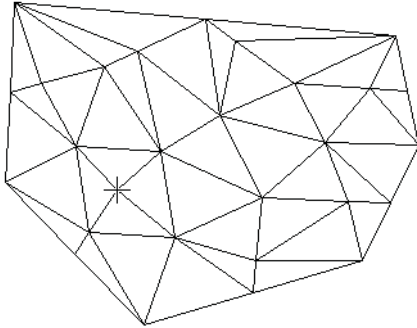
1. Activate **Terrain**, and click points to define the outer boundary (or enter coordinates in the Coordinate Fields). As you click points, the surface triangulates.



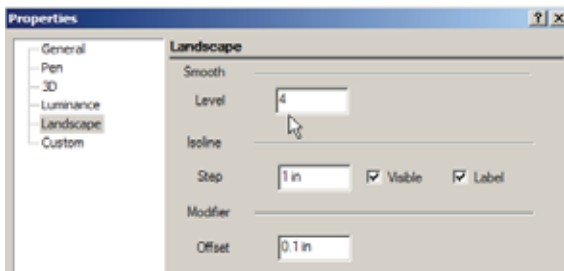
2. Continue defining the outer boundary.



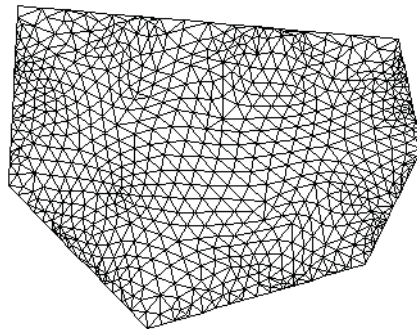
- Define interior points as well. Each point you define will become a node you can edit. You do not need to define all nodes now; you can add nodes later.



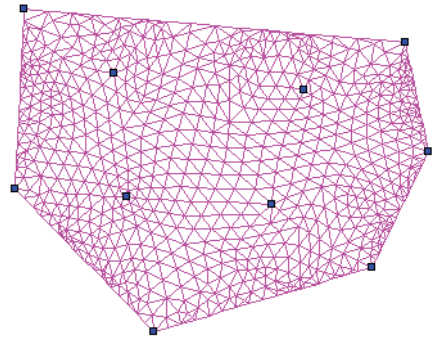
- Open the terrain's **Properties** to the **Landscape** page. Under **Smooth**, increase the **Level**.



This increases the level of triangulation, resulting in a more accurate surface.



- Activate the **Edit Tool** on the terrain. Each point you defined when creating the terrain is represented as a node.

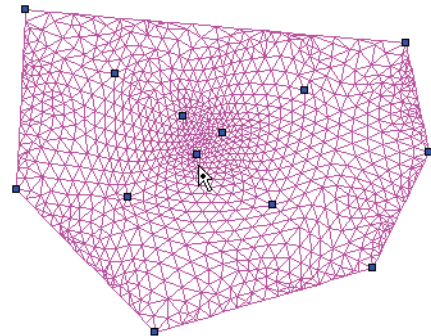



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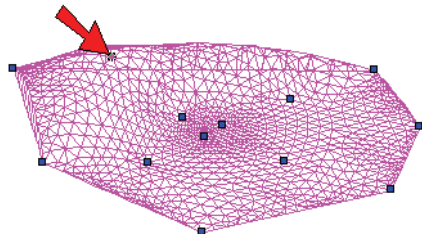
NOTE: For details on this tool, see "Edit Tool" on page 219.

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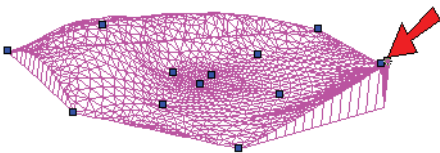
- If you want to add more nodes, right-click where you want the new node and select **Add Node**.



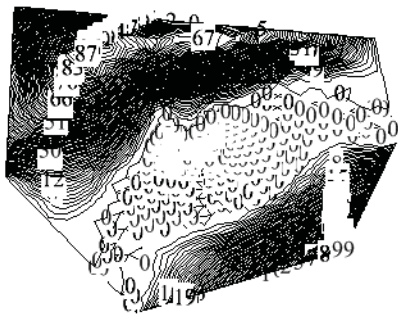
7. To modify the terrain, click a node and use the Coordinate Fields to change its Z position. You can also drag a node while in a Front or Side view. Adjacent nodes will remain in place.



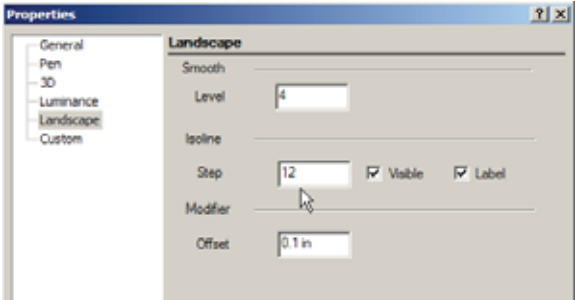
8. You can also change the elevation of nodes along the boundary.



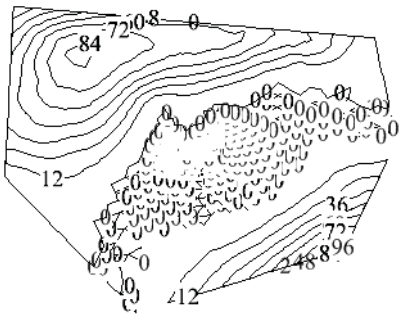
9. Switch to **World Plan**. This view shows the terrain as a series of isolines, each labeled with its elevation.



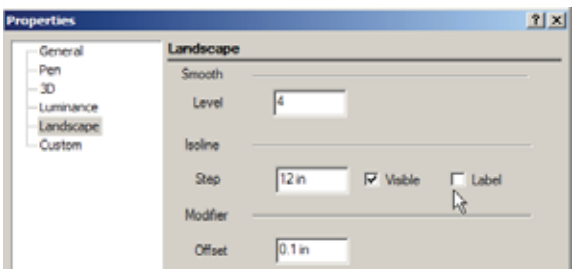
10. In the **Landscape** page, change the **Step**. In this example, an isoline will be displayed every 12 inches.



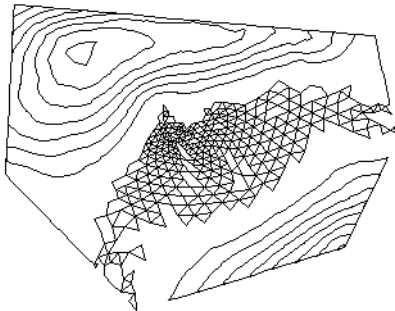
This makes the isoline display less dense.



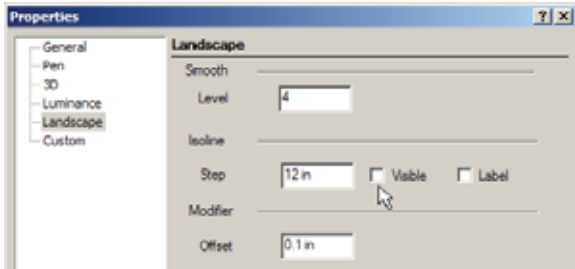
11. Uncheck **Label**.



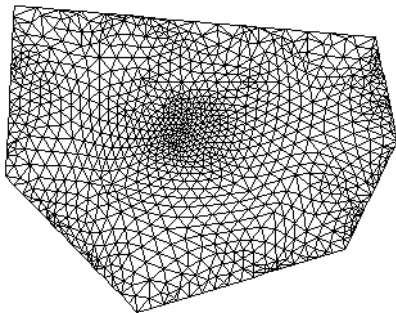
This removes the elevation value from each isoline.



12. Uncheck **Visible**.



This removes the isoline display.



If isolines are displayed, when you move out of World Plan view, they will not be visible.

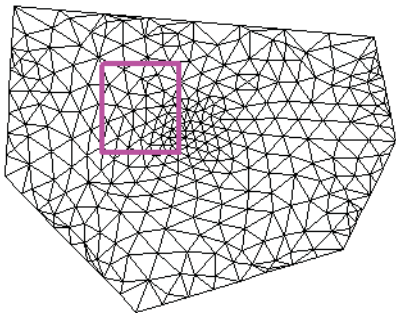
## Terrain Modifier

Available in TurboCAD Pro and Platinum only



You can add or remove material from a terrain around a closed, 2D shape.

1. Start with a terrain, and create a closed 2D shape, such as a rectangle.



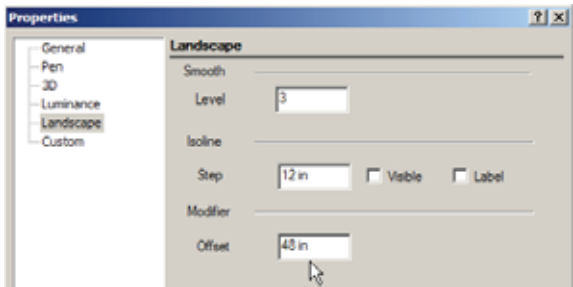
2. In this example, the shape is placed vertically so that part of it is below the terrain, and part of it is above.



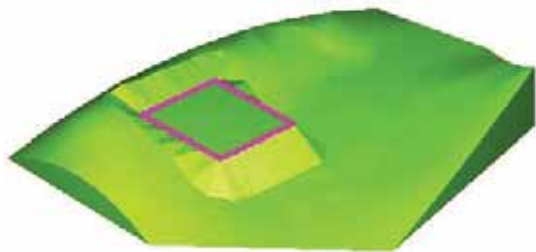
3. Activate **Terrain Modifier**, and select the terrain to modify. Then select the closed shape. Material is added or removed as needed to meet the shape.



4. To slope the terrain around the modified shape, add an **Offset** value.



This is the result:



### Import Terrain



Terrain data you can import must be listed in a .txt or .xyz file. The format for each coordinate should be X, Y, and Z values, separated by commas or spaces.

Activate **Import Terrain**, and define the original point. This is where the center of the terrain will be placed. Then select the data file.

### Terrain from Selected Points



Creates a terrain from a set of selected points. The points must all have the same Z coordinate.

For details on points, "Point" on page 137). To create the terrain, first select the points, then activate **Terrain from selected Points**.

## Schedule



Inserts a schedule - a table detailing all selected windows and doors. This tool works in conjunction with the Style Manager. See "Schedule Styles" on page 592.

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NOTE: The **Fill Schedule Wizard** is another way to add a schedule, that scans for objects on specific layers. See "Fill Schedule Wizard" on page 595.

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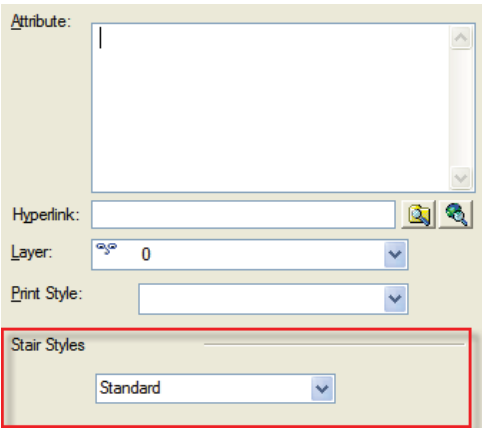
## Style Manager

The Style Manager enables you to define styles for the following commonly-used objects:

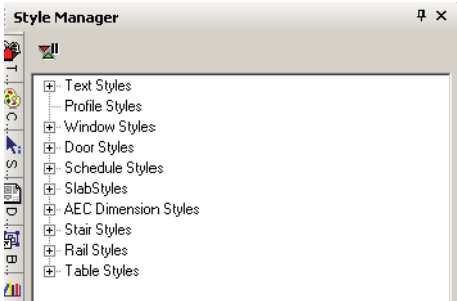
- Text
- Profiles (for modifying windows and doors)
- Windows
- Doors
- Walls
- Schedules
- Slabs
- AEC Dimensions
- Dimensions
- Stairs
- Rails
- Tables



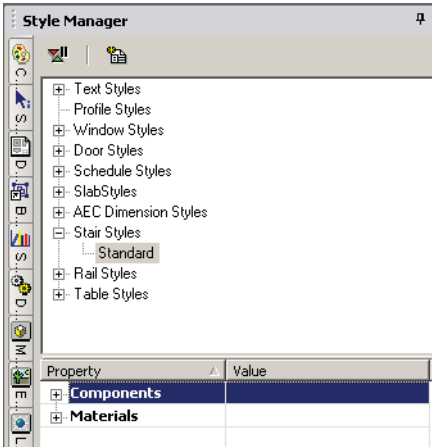
You select assign a style to an object through the object properties on the General page. Simply select the style in the Style drop-down box at the bottom of the page.



You can open the Style Manager by selecting **View / Style Manager Palette**.



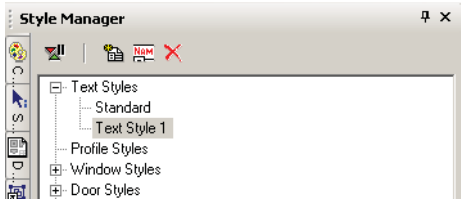
Each category in the Style Manager contains one pre-defined style, called “Standard.” For each of these s, you can define relevant properties.



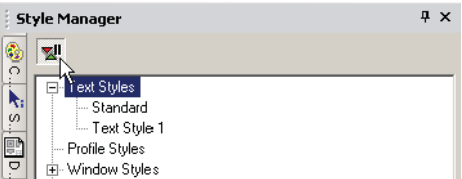
To create a new style, highlight a style you want to base the new style on, and click **Create New Style**.



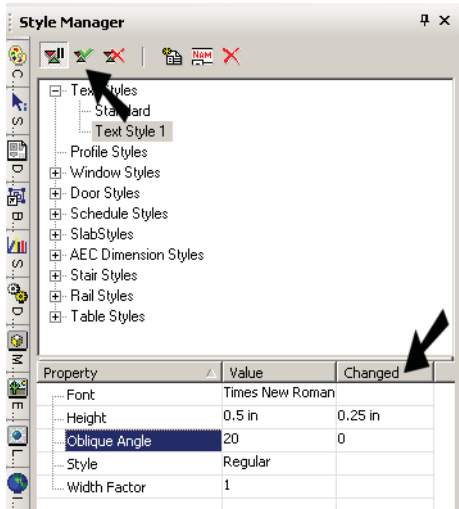
Once you have new style, there are icons at the top you can use to **Rename** or **Delete** it.



The **Delay Style Modification** icon can be enabled if you don't want to update your style with each property change.



If **Delay Style Modification** is enabled, you can view both old and new properties in the lower pane. At the top is an icon for updating the entire style at once, or for clearing changes not yet implemented.



Savings as Templates

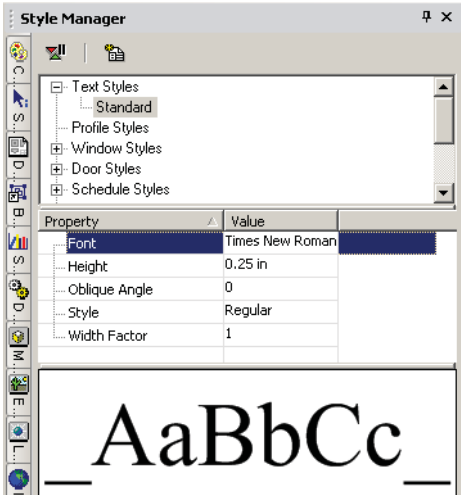
You can save styles in template files, so that you don't have to create styles from scratch each time. To do this, set up the styles you want for doors, schedules, slabs, etc. Then use **File / Save As** to save the file as a \*.tct file (TurboCAD Template). Place the template file in the "Template" folder of the TurboCAD root directory. Then when you want to open the template, use **File / New**, and select **New from Template**.

NOTE: This also applies to tool **Properties**; set the properties you want, such as door dimensions or wall width and height, and save them as part of the template file as well.

Text Styles

The can be used to set various styles for text. For details on creating text, see "Text" on page 325.

In the Style Manager, there is one style, "Standard," listed under "Text styles." This style defines properties such as font and height. A preview showing a layout the style is on the lowest pane.



NOTE: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window.

- 1. You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted and click **Create New Style**.

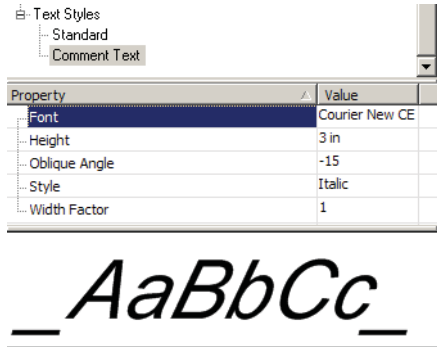


- 2. Assign a name or accept the default.

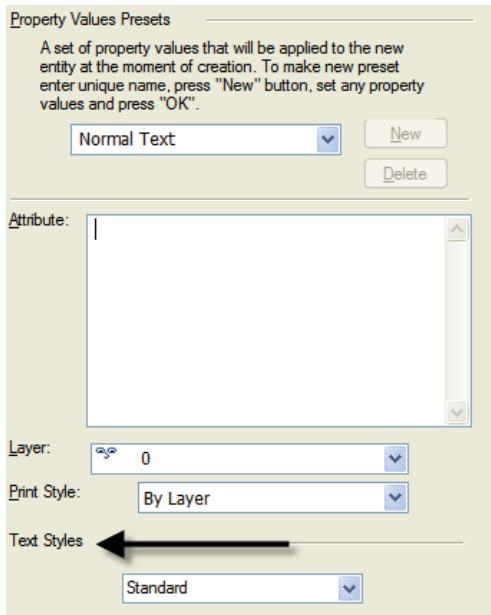


This creates a new style which is a copy of "Standard."

3. Highlight the new style you created, and change some of the parameters.



4. To apply this style, open **Properties** of a text string to the **General** page.



NOTE: If a text string already has other parameters defined, such as color or a text box, these properties will remain. Only the changed style parameters, such as font or height, will update.

### Profile Styles

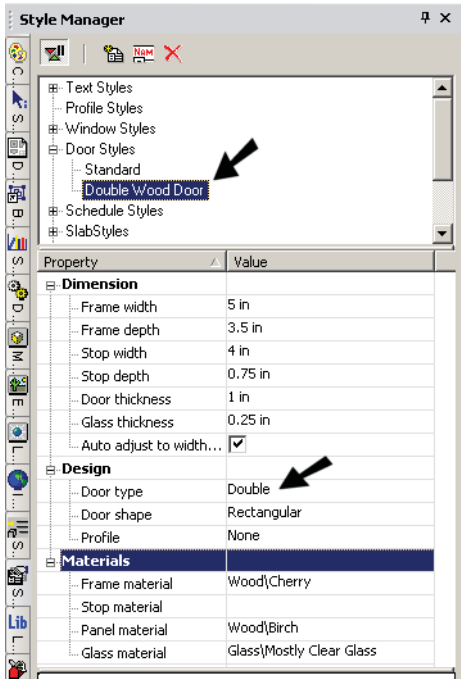
Profiles are used to define custom shapes for doors, and windows. Once you have created the profile, the profile is selected as part of the window or door's profile.

NOTE: If you want to save styles to a template, see "Savings as Templates" on page 566.

The first example will show how to use a profile for a door shape.

#### Example 1: Door Profile with One Boundary

1. Before creating the door, create a new door style. This one is a double door with a rectangular shape.



NOTE: For details on door s, see "Door Styles" on page 579.

2. Create walls and insert the door. Use the door's **Properties** window to set the overall size and elevation of the door.



This is the door that will be modified with a profile.

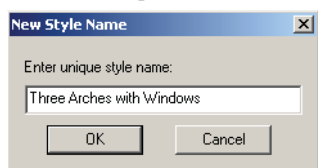
3. First, look at the **Delay Style Modification** icon; if it is enabled, you cannot create a new profile. Make sure **Delay Style Modification** is off.



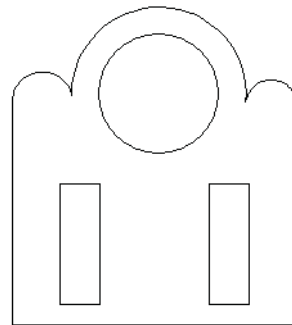
4. Highlight **Profile styles** and click **Create New Style**.



5. Assign a name to the profile.



6. When you create a profile, you switch to **Edit Geometry** mode, and anything you have in your file is removed from view. Because you are creating a 2D profile, switch to **World Plan** view.
7. Use **Polyline**, **Arc** or **Circle** tools to create closed curves for boundaries and holes. In this example, the outer shape and the two rectangles were created using Polyline. The circle was created using **Circle Center and Point**.

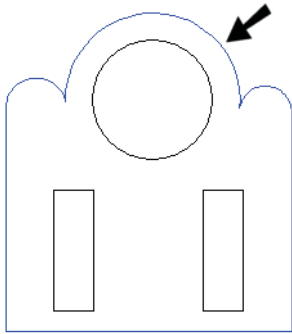


You can have one or more boundaries, but boundaries cannot overlap. Holes must be entirely within a boundary.

8. To define which curves will be used for boundaries, select **Edit Boundary Profile**.



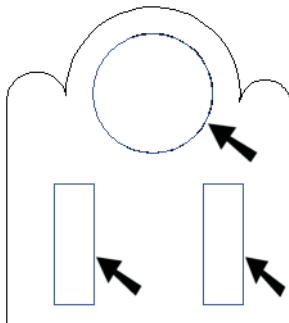
9. Click the boundary curve or curves, which turn blue when selected.



10. Select **Finish** from the local menu or Inspector Bar.
11. To define which curves (if any) will be used for holes, select **Edit Hole Profiles**.

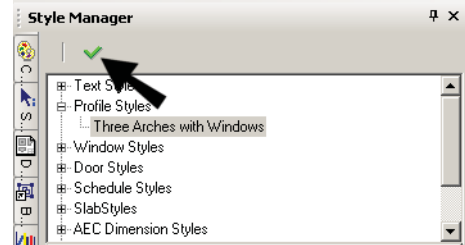


12. Click the hole curve or curves, which turn blue when selected.

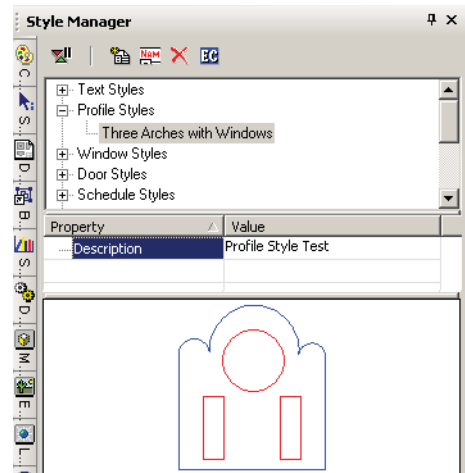


13. Select **Finish** from the local menu or Inspector Bar.

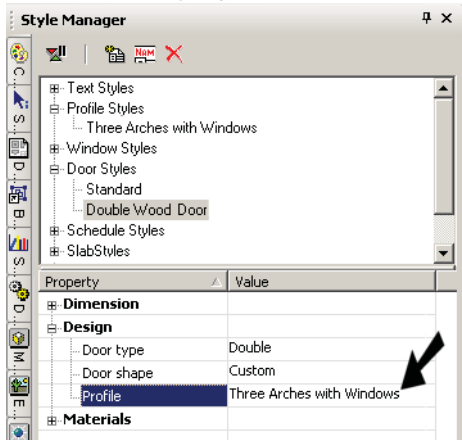
14. When the boundaries and holes are defined, click **Finish to Edit Geometry** in the Style Manager.



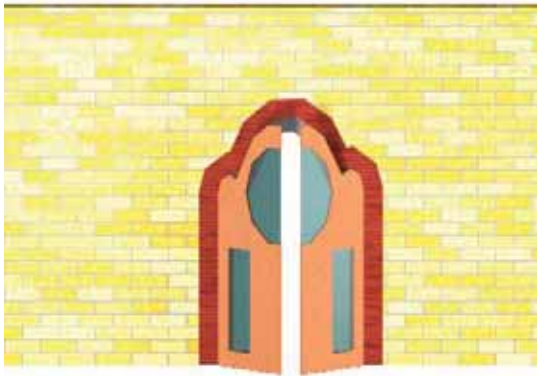
The profile now appears in the Preview area of the Style Manager. Boundary profiles are shown in blue, holes are shown in red.



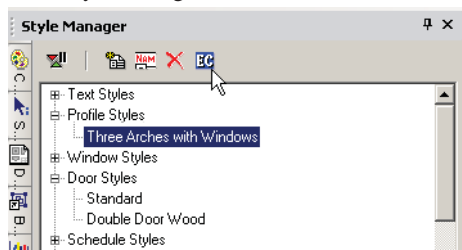
15. Now return to the door profile. Under **Design**, set the **Profile** to the one you just created.



The door outline changes to match the profile. The hole profiles define the door glass, and these areas have the material specified for **Glass** in the door style.



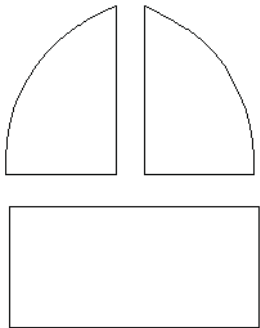
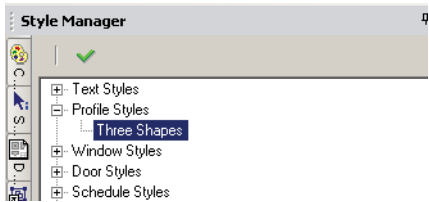
16. If you want to make changes to the profile, highlight it in the Style Manager, and then click **Edit Content**.



17. You will return to **Edit Geometry** mode, where you can make your changes. When you're finished editing, click **Finish to Edit Geometry**. The door or window that uses the profile will update automatically.

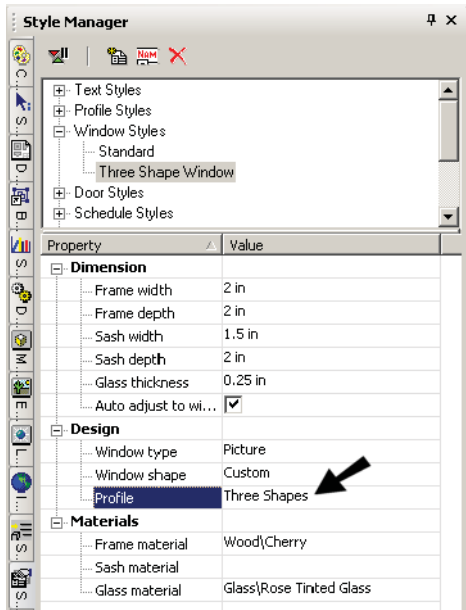
### Example 2: Window Profile with Multiple Boundaries

1. In this example, create a new profile that will be used in a window.
2. This profile has three closed profiles, each created using **Polyline**.



Hole profiles are not generally needed for windows, since the boundary profiles define where the window glass is. If you define holes in window profiles, holes will be placed in the glass.

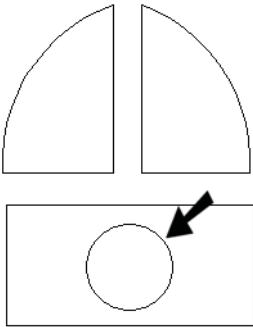
3. Now create a window style that uses the profile. This example uses a **Picture** window.



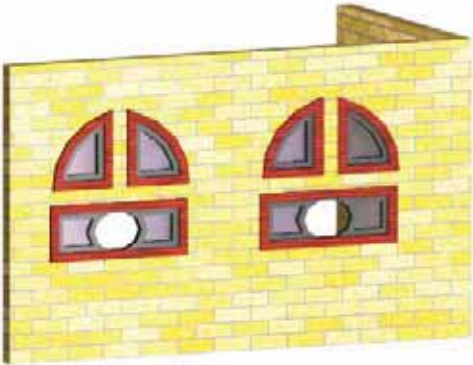
4. Use the window **Properties** to define overall dimensions, and insert a window or two in a wall. Each window consists of three parts, defined by the boundary profiles.



5. To see what hole profiles would do, use **Edit Content** edit the profile. Add another profile within one of the boundaries. Use **Tools / Architecture / Profile / Edit Hole Profile** to define the new curve as a hole. Then click **Finish to Edit Geometry**.



The glass now has a hole where the new profile curve was created.

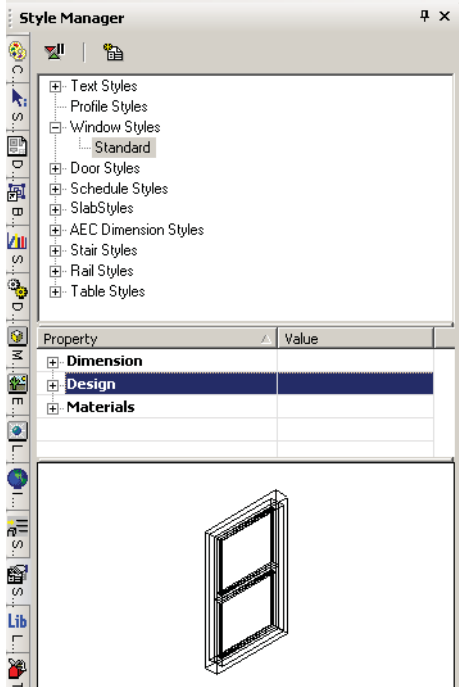


### Window Styles

The **Window** tool can be used to insert windows into walls, but you need to first define window styles.

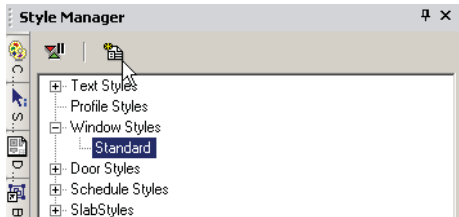
*NOTE: If you want to save styles to a template, see "Savings as Templates" on page 566.*

In the Style Manager, there is one style, “Standard,” listed under “Windows.” A preview showing a previous of the window style is on the lowest pane.



NOTE: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window.

1. You can change the “Standard” style, but if you want to preserve this style, make sure “Standard” is highlighted and click **Create New Style**.

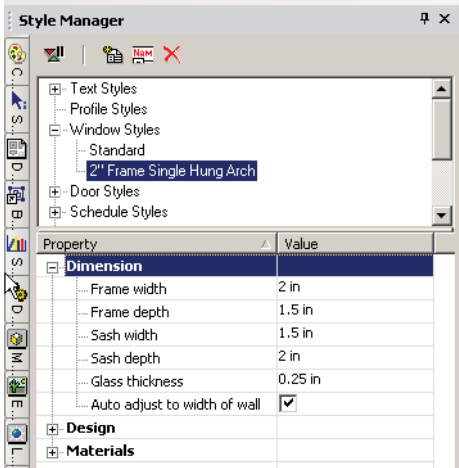


2. Assign a name.



This creates a new style which is a copy of “Standard.”

3. There are three categories of properties. Open **Dimension**, where you can specify measurements of the frame, sash, and glass.



- **Auto Adjust to Width of Wall:** Sets the depth of the window so that it will cut all the way through the wall.

4. Open **Design**, where you can specify the window type (double hung, transom, etc.) and shape (rectangular, arch, octagon, etc.). For a list of available window types, see “Window Types” on page 575.

Property	Value
<b>Dimension</b>	
<b>Design</b>	
Window type	Awning Transom
Window shape	Rectangular
Profile	None
Muntins	Muntins
<b>Materials</b>	

5. You can also select, edit and create muntins. See “Window Muntins” on page 577

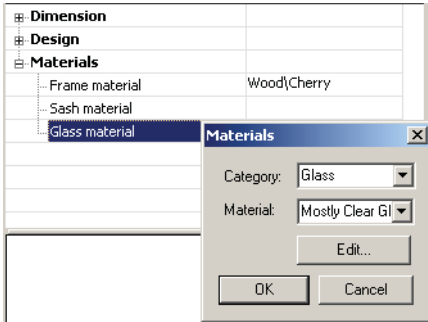


NOTE: **Profile** is used when you've created a profile for changing the window shape, or for adding holes. See "Profile Styles" on page 567.

6. The last category is **Materials**, in which you can set the materials of the frame, sash, and glass. If you leave any material blank (such as the **Sash Material** below), the object will be colored according to the **Pen** color set in the window's **Properties**.

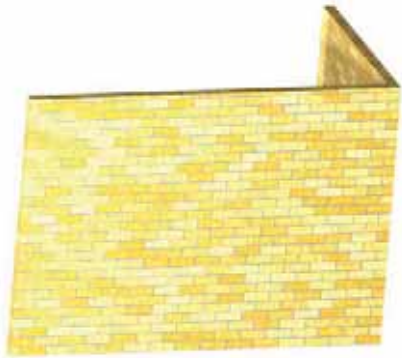
Property	Value
[-] Dimension	
[-] Design	
[-] Materials	
Frame material	
Sash material	
Glass material	
Muntins material	

To set or change a material, just click in the field. In the **Materials** window that opens, select the **Category** and **Material**.

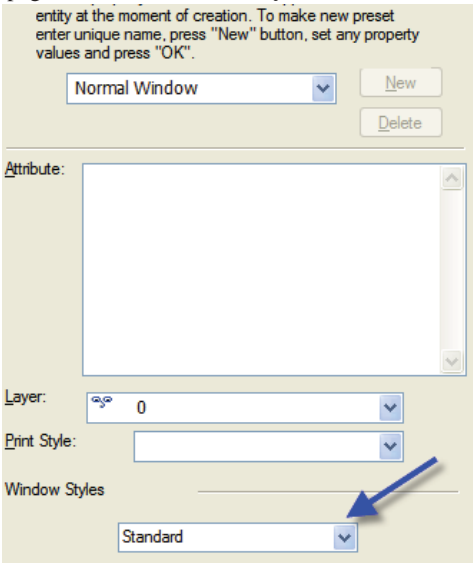


NOTE: For details on materials, see "Materials" on page 522.

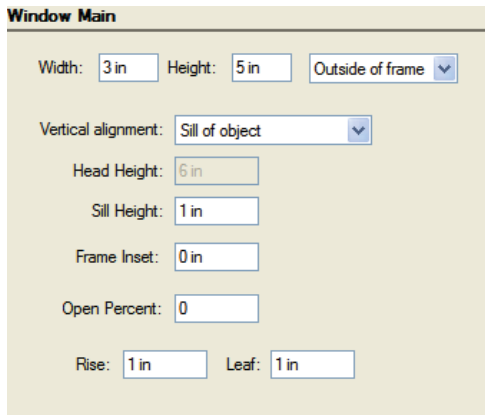
7. Before creating any windows, you must have at least one wall defined. See "Inserting Walls" on page 536. To assign height to a wall, define a **Thickness** in the **3D** page of the wall's **Properties**.



8. This is the icon for **Window**. Right-click on the icon to set the tool's **Properties**, and click the icon when you're ready to insert a window.
9. Open the **Window** tool's **Properties**. On the **General** page, select the **Window Style**.

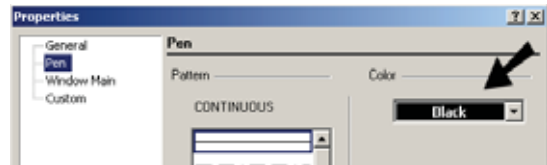


10. On the **Window Main** page, set the overall dimension of the window. The preview at the lower left updates when you change the values.



- **Width** and **Height**: Overall outer dimensions of the window.
- **Inside/Outside of Frame**: Specifies that the dimensions of the window are applied by measuring to the inside or outside of the window frame.
- **Vertical alignment**: Specifies whether the window's vertical position is specified by the Sill Height, or the Head of the window.
- **Head Height**: Specifies the vertical location of the top of the window. This field is only accessible if the vertical alignment option is set to **Head of object**.
- **Sill Height**: Specifies the vertical location of the sill of the window. This field is only accessible if the vertical alignment option is set to **Sill of object**.
- **Rise**: The distance from the top corner to the top center, in the case of arched or peaked windows.
- **Frame Inset**: The distance from the front of the wall to the window frame.
- **Open Percent**: Defines how wide the window is open.
- **Leaf**: in the case of uneven windows, the size of one of the panes.

11. On the **Pen** page, specify the color of the window. This color will be applied to any part of the window that does not have a material assignment.



12. When all parameters are set, activate **Window** and click the wall to place the windows. The wall material within the window area is removed. No matter where you click on the wall, the windows are placed according to the **Elevation** value. In this example, the sash is colored black, which is the default **Pen** color. The frame and glass have the assigned materials.

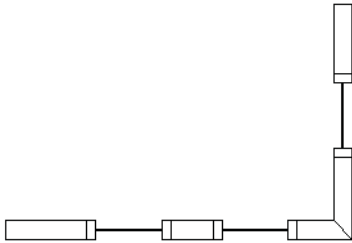


13. If you click another wall, the window properly aligns to it.



14. You can orient of the window to the wall by using the **Flip Left-Right**, **Flip Inside-Outside** controls in the Local menu, or Inspector bar.

In **World Plan** view, you can see how the windows cut the walls.



All parameters available on the **Properties** window, including **Window style**, are also available on the Selection Info palette. If you move a window, it will remain within the plane of the wall.

### Window Types

These are the standard window types available in the **Design** category of the Style Manager.

---

NOTE: *The expanded library of windows and doors is Available in TurboCAD Platinum only.*

---

#### Single Hung



#### Double Hung



#### Pass Through (no glass)



#### Picture (does not open)



**Single Casement**



**Double Casement**



**Glider**



**Awning Transom**



**Hopper Transom**



**Single Hopper**



**Single Awning****Vertical Pivot****Horizontal Pivot**

**Uneven Single Hung, Uneven Double Hung:** the two panes have different heights.

**Window Muntins**

*Available in TurboCAD Platinum only*

Window muntins are available in several patterns that you can customize into many different styles:

- Rectangular
- Diamond
- Prairie 9 Lights
- Prairie 12 Lights
- Starburst
- Sunburst

The Starburst and Sunburst patterns are combined rounded/rectangular patterns with a half-round or a quarter-round spoked top and can be used for half- and quarter-round as well as rectangular muntins. The other patterns are primarily rectangular but will fit into a half-round.

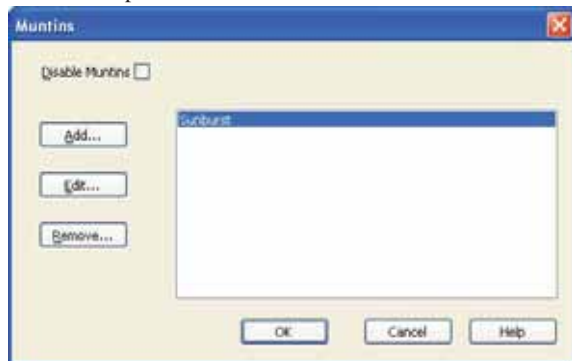
Multiple muntin blocks can be assigned to each style. This allows you to have different patterns of muntins for each panel of glass, or to combine muntin patterns.

To setup muntins component please do the following:

1. Open the Style Manager Palette.
2. Navigate to the Window styles.
3. Select the window style you wish to add muntins to.
4. Expand the Design properties category and click Muntins.
5. The Muntins dialog will appear.

This dialog contains list of all muntins assigned to the window style.

For example you can add muntins with Rectangular pattern to the one door glass component and muntins with diamond to other component.



**Add:** displays the Muntins Block dialog where you can define a name for the muntins block and its properties.

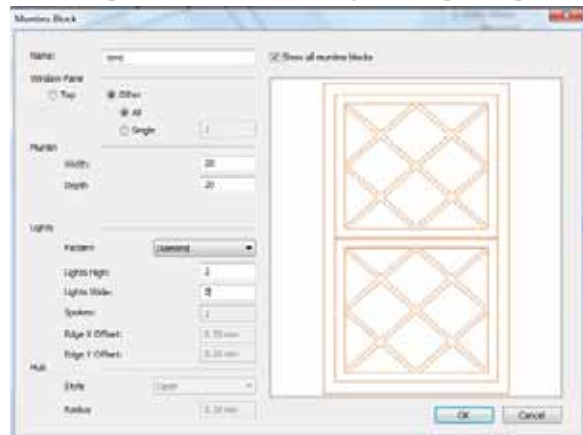
**Edit:** displays the Muntins Block dialog with properties for the selected muntins.

**Delete:** Deletes the selected muntins block.

**Disable Muntins:** when ON – muntins are disabled.

1. Open the Style Manager and select a door style you wish to add muntins to.
2. Expand the Design properties category and click Muntins.
3. The Muntins dialog will open.
4. Click the Add button to open Muntins block dialog.

5. Muntins can be applied to All window glass components, or to individual glass component pane.



If you specify a single panel you must specify the number of the panel to be used. The number of the panel corresponds to the order in which you selected the hole profiles.

*NOTE: If you cannot see the window because the preview is rotated, click on the preview with the center mouse button (wheel) hold and drag. This will rotate the preview.*

6. Define the muntins width and depth.
7. Select the Rectangular in Pattern combo-box.
8. Enter number of horizontal muntins lights in “Lights High” edit box.
9. Enter number of vertical muntins lights in “Lights High” edit box.
10. Click OK to close the Muntins Block dialog.
11. Click OK to close Muntins dialog.

The Diamond Muntins pattern uses the same options as the Rectangular pattern.

Prairie-9 Lights pattern, and the Prairie-12 Lights pattern use two alternate options:

**Edge X Offset and Edge Y Offset:** these specify the distance of the outer muntins from the panel edge.

The Starburst Muntins pattern has two additional options:

**Spokes:** which sets the number of spokes that radiate out from the center to form the burst pattern.

**Center:** which sets whether the hub will be at the Top or Bottom of the radial pattern.

The Sunburst Muntins pattern has two additional options.

**Style:** this sets the whether the center of the sunburst will be an open circle or a closed disk.

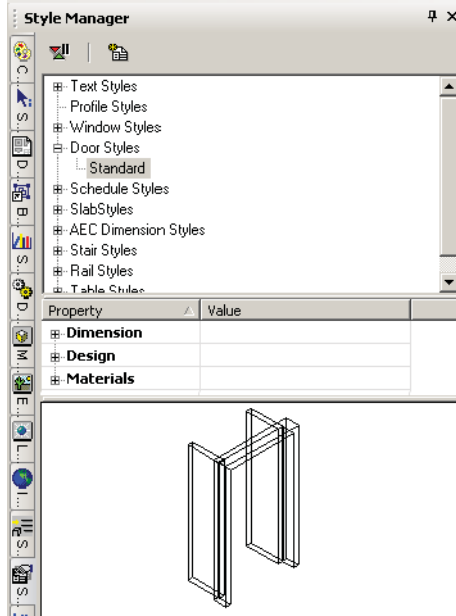
**Radius:** this sets the radius for the center of the sunburst.

## Door Styles

The **Door** tool can be used to insert doors into walls, but you need to first define door styles.

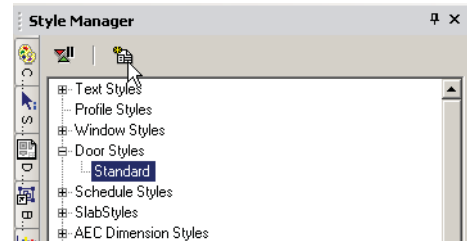
*NOTE: If you want to save styles to a template, see "Savings as Templates" on page 566.*

In the Style Manager, there is one style, "Standard," listed under "Door styles." A preview showing a previous of the door style is on the lowest pane.

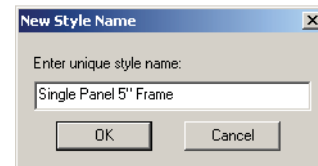


*NOTE: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window.*

1. You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted and click **Create New Style**.

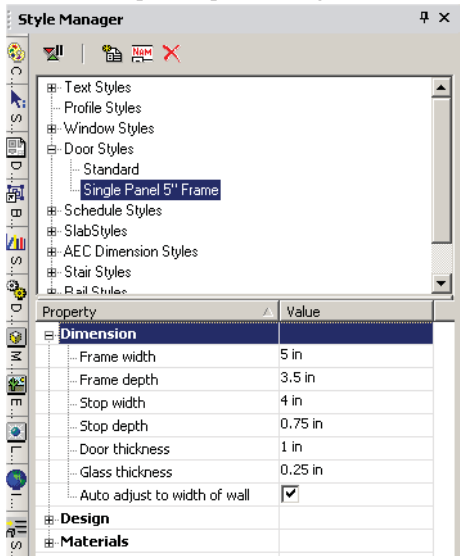


2. Assign a name.



This creates a new style which is a copy of "Standard."

3. There are three categories of properties. Open **Dimension**, where you can specify measurements of the frame, stop, door panel, and glass.



- **Auto Adjust to Width of Wall:** Sets the depth of the door so that it will cut all the way through the wall.
4. Open **Design**, where you can specify the door type (single, double, pocket, etc.) and shape (rectangular, arch, half round, etc.). For a list of available door types, see "Door Types" on page 582.

Property	Value
Dimension	
Design	
Door type	Single
Door shape	Custom
Profile	Profile Style 1
Muntins	Muntins
Materials	

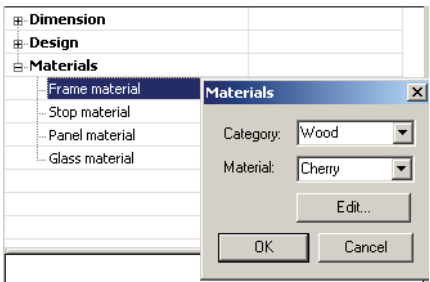
5. You can also select, edit and create muntins. See "Door Muntins" on page 584

NOTE: **Profile** is used when you've created a profile for changing the door shape, or for adding holes. See "Profile Styles" on page 567.

6. The last category is **Materials**, in which you can set the materials of the frame, stop, panel, and glass. If you leave any material blank (such as the **Stop Material** below), the object will be colored according to the **Pen** color set in the window's **Properties**.

Property	Value
Dimension	
Design	
Materials	
Frame material	
Stop material	
Panel material	
Glass material	
Muntins material	

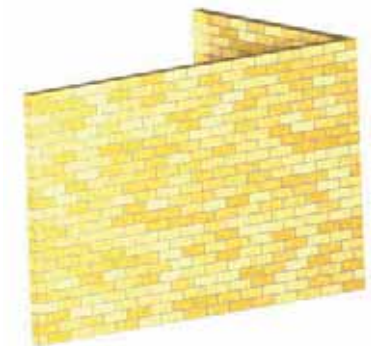
To set or change a material, just click in the field. In the **Materials** window that opens, select the **Category** and **Material**.



NOTE: For details on materials, see "Materials" on page 522. Glass material will only be used if the door uses a profile that includes holes. See "Profile Styles" on page 567.



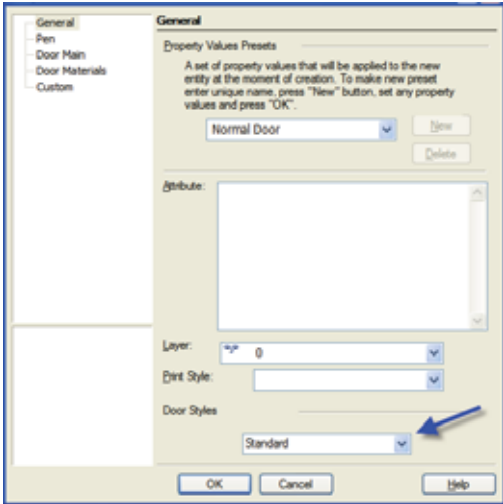
7. Before creating any doors, you must have at least one wall defined. See "Inserting Walls" on page 536 To assign height to a wall, define a **Thickness** in the **3D** page of the wall's **Properties**.



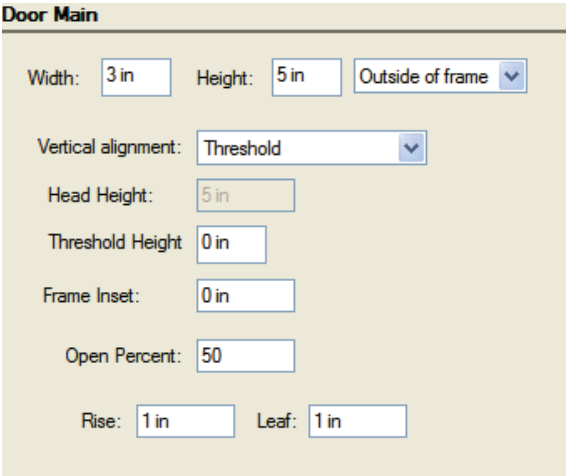
8. This is the icon for **Door**. Right-click on the icon to set the tool's **Properties**, and click the icon when you're ready to insert a door.



9. Open the **Door** tool's **Properties**. On the **General** page, select the **Door Style**.

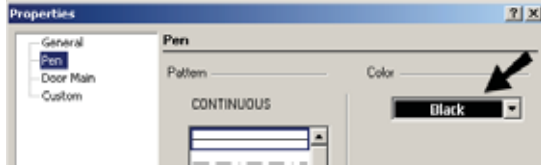


10. On the **Door Main** page, set the overall dimension of the door. The preview at the lower left updates when you change the values.



- **Width** and **Height**: Overall outer dimensions of the window.
- **Inside/Outside of Frame**: Specifies that the dimensions of the window are applied by measuring to the inside or outside of the window frame.
- **Vertical alignment**: Specifies whether the widows vertical position is specified by the Threshold Height, or the Head of the window.
- **Head Height**: Specifies the vertical location of the top of the window. This field is only accessible if the vertical alignment option is set to **Head of object**.
- **Threshold Height**: Specifies the vertical location of the sill of the window. This field is only accessible if the vertical alignment option is set to **Threshold**.
- **Rise**: The distance from the top corner to the top center, in the case of arched or peaked windows.
- **Frame Inset**: The distance from the front of the wall to the window frame.
- **Elevation**: The distance from the bottom of the wall to the bottom of the window. This value is also available on the Inspector Bar.

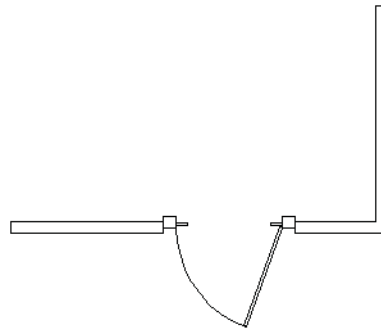
- **Open Percent:** Defines how wide the window is open.
  - **Leaf:** in the case of uneven doors, the size of one of the panes.
11. On the **Pen** page, specify the color of the door. This color will be applied to any part of the door that does not have a material assignment.



12. When all parameters are set, click the wall to place the door. The wall material within the door area is removed. No matter where you click on the wall, the door is placed according to the **Elevation** value. In this example, the door stop is colored black, which is the default **Pen** color. The frame and panel have the assigned materials.



In **World Plan** view, you can see how the door cuts the wall.



13. You can orient of the window to the wall by using the **Flip Left-Right**, **Flip Inside-Outside** controls in the Local menu, or Inspector bar.

All parameters that available on the **Properties** window, including **Door style**, are also available on the Selection Info palette. If you move a door, it will remain within the plane of the wall.

## Door Types

These are the standard door types available in the **Design** category of the Style Manager.

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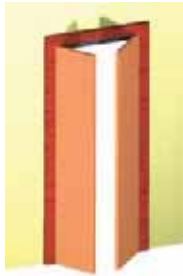
*NOTE: The expanded library of windows and doors is Available in TurboCAD Platinum only.*

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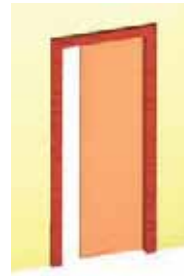
### Single



**Double**



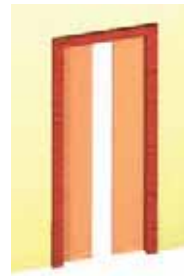
**Pocket**



**Double Opposing**



**Double Pocket**



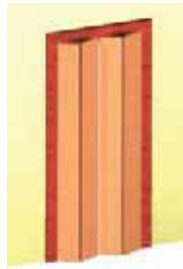
**Bifold**



**Sliding Double**



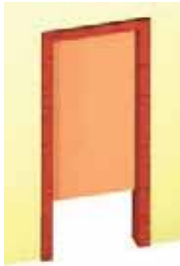
**Bifold Double, Panel**



**Sliding Triple**



## Overhead



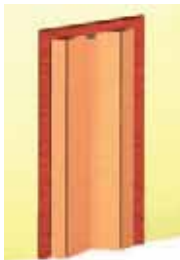
## Revolving



## Pass Through (no panel)



## Accordion



## Communicating



## Door Muntins

*Available in TurboCAD Platinum only*

Door muntins can be applied only to doors with glass components.

To make a door with glass components you must create profile first (Style Manager | Profile Styles).

As soon as you have profile style created, navigate to the door styles, select the door style you wish add muntins to and select the profile as door shape (Door Shape = Custom, Profile = "Profile Style 1")

Door muntins are available in several patterns that you can customize into many different styles:

- Rectangular
- Diamond
- Prairie 9 Lights
- Prairie 12 Lights
- Starburst
- Sunburst

The Starburst and Sunburst patterns are combined rounded/rectangular patterns with a half-round or a quarter-round spoked top and can be used for half- and quarter-round as well as rectangular muntins. The other patterns are primarily rectangular but will fit into a half-round.

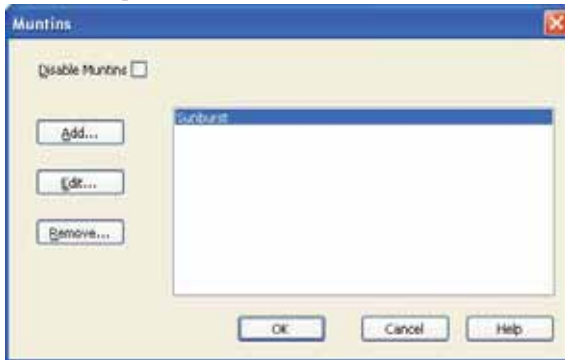
Multiple muntin blocks can be assigned to each style. This allows you to have different patterns of muntins for each panel of glass, or to combine muntin patterns.

To setup muntins component please do the following:

1. Open the Style Manager Palette.
2. Navigate to the Door styles.
3. Select the door style you wish to add muntins to.
4. Expand the Design properties category and click Muntins.
5. The Muntins dialog will appear.

This dialog contains list of all muntins assigned to the door style.

For example you can add muntins with Rectangular pattern to the one door glass component and muntins with diamond to other component.



**Add:** displays the Muntins Block dialog where you can define a name for the muntins block and its properties.

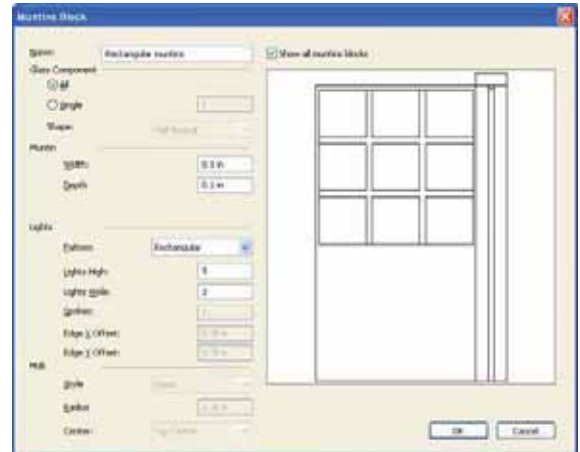
**Edit:** displays the Muntins Block dialog with properties for the selected muntins.

**Delete:** Deletes the selected muntins block.

**Disable Muntins:** when ON – muntins are disabled.

1. Open the Style Manager and select a door style you wish to add muntins to.
2. Expand the Design properties category and click Muntins.
3. The Muntins dialog will open.
4. Click the Add button to open Muntins block dialog.

5. Muntins can be applied to All door glass components, or to individual glass component pane.



If you specify a single panel you must specify the number of the panel to be used. The number of the panel corresponds to the order in which you selected the hole profiles.

---

*NOTE: If you cannot see the door because the preview is rotated, click on the preview with the center mouse button (wheel) hold and drag. This will rotate the preview.*

---

6. Define the muntins width and depth.
7. Select the Rectangular in Pattern combo-box.
8. Enter number of horizontal muntins lights in "Lights High" edit box.
9. Enter number of vertical muntins lights in "Lights High" edit box.
10. Click OK to close the Muntins Block dialog.
11. Click OK to close Muntins dialog.

The Diamond Muntins pattern uses the same options as the Rectangular pattern.

Prairie-9 Lights pattern, and the Prairie-12 Lights pattern use two alternate options:

**Edge X Offset and Edge Y Offset:** these specify the distance of the outer muntins from the panel edge.

The Starburst Muntins pattern has two additional options:

**Spokes:** which sets the number of spokes that radiate out from the center to form the burst pattern.

**Center:** which sets whether the hub will be at the Top or Bottom of the radial pattern.

The Sunburst Muntins pattern has two additional options.

**Style:** this sets the whether the center of the sunburst will be an open circle or a closed disk.

**Radius:** this sets the radius for the center of the sunburst.

## Custom Blocks for Windows and Doors

*Available in TurboCAD Platinum Only*

This functionality provides the means to attach one or several blocks to window and doors to provide additional geometry e.g. a window sill, door knob, shutters.

You can attach custom block to entire window/door or to different parts.

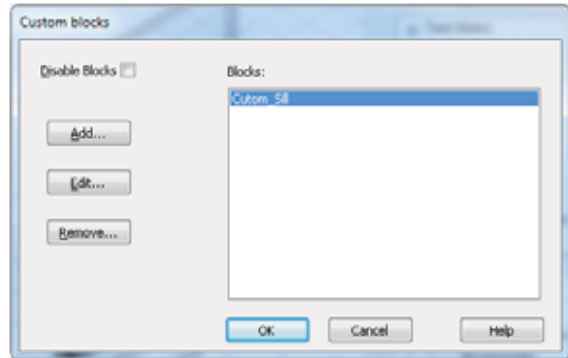
To make a door with components you must create the block first. The block used for this must be composed only of ACIS solids.

Multiple muntin blocks can be assigned to each style. This allows you to have different patterns of muntins for each panel of glass, or to combine muntin patterns.

To setup muntins component please do the following:

1. Open the Style Manager Palette.
2. Navigate to the Door or Window styles.
3. Select the style you wish to add blocks to.
4. Expand the Design properties category and click Blocks.
5. The Custom Blocks dialog will appear.

This dialog contains list of all blocks assigned to the door or window style.



### Custom Block Dialog:

**Disable Blocks:** When ON - custom blocks are ignored.

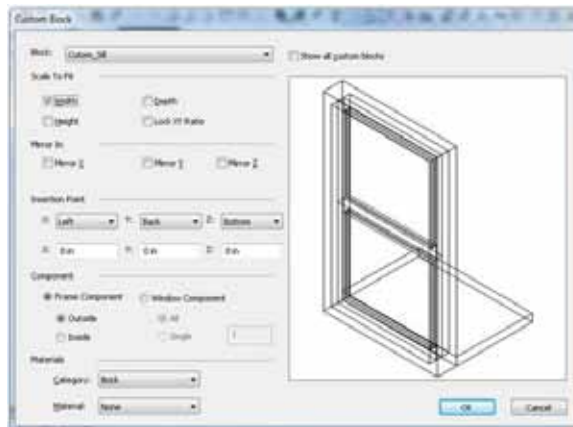
**Add:** displays Custom Block dialog to add new custom block.

**Edit:** displays Custom Block dialog to edit selected custom block

**Remove:** deletes selected custom block

**Blocks list:** contains list of blocks attached to the window style.

### Custom Blocks Editor



**Block combo-box:** Used for selecting the block (from the list of blocks) that is attached to the window/door style.

### Scale To Fix Controls Group

**Width check box:** when ON – block is scaled to fit window/door width.

**Height check box:** when ON – block is scaled to fit window/door height.

**Depth check box:** when ON – block is scaled to fit window/door depth.

**Lock XY Ratio check box:** works in combination with one of the (Width, Height, and Depth) check boxes.

For example, when Width is ON and you click Lock XY Ratio the block is scaled in a way to keep XY ratio.

### Mirror In Controls

**Mirror X check box:** mirrors block relative X axis.

**Mirror Y check box:** mirrors block relative Y axis.

**Mirror Z check box:** mirrors block relative Z axis.

### Insertion Point controls group

**X:** combo-box: Defines how block is attached relative to window/door in X Axis (to Left Side, Center or Right Side)

**Y:** combo-box: Defines how block is attached relative to window/door in Y Axis (to Back Side, Center or Front Side)

**Z:** combo-box: Defines how block is attached relative to window/door in Z Axis (to Bottom Side, Center or Top Side)

**X: Y: Z: edit boxes:** define the offset from block origin point

### Component Controls Group

Frame Component:

- Outside – block is attached to entire window
- Inside - block is attached to inside frame

Window Component

- All – block is attached to all window component
- Single – block is attached to single window component

### Materials Controls Group

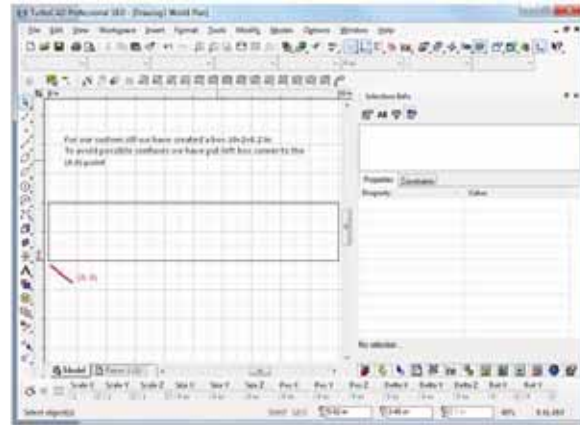
These controls allow define material that will be used for this custom block in render.

### Example of use:

Suppose you would like to create custom sill for our window style.

#### Create a block.

First, we create a geometry for our sill (just a box with 10x2x0.2 in size

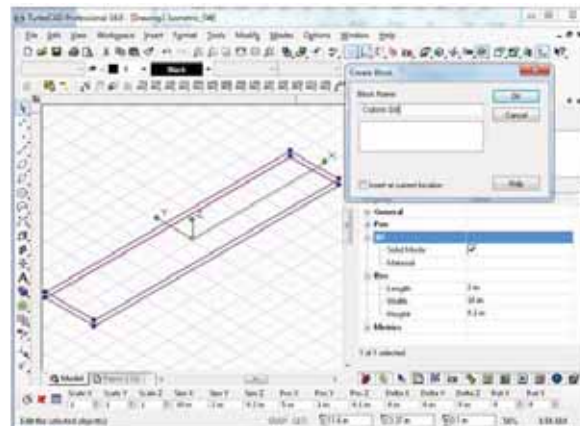


*NOTE: Block used for Custom Blocks must be composed of ACIS solids. For best results align your block with the origin of the drawing.*

### Make Block

Select the box and select **Format | Create Block**.

Enter “Custom Sill” as the Block name and click OK.



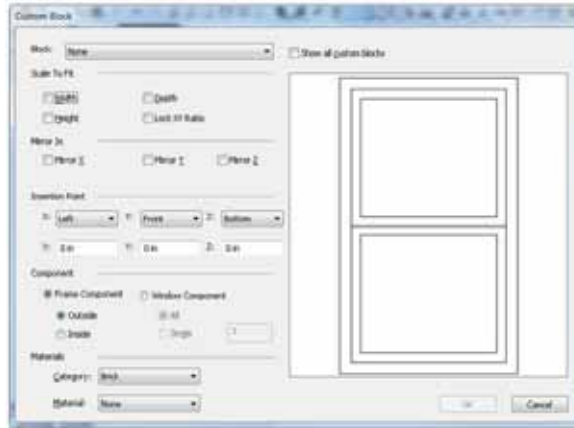


Navigate to the style manager palette, select Standard window style and click the Block property in the properties of the window style.

The Custom Block dialog opens. It contains a list of all the blocks that are attached to the window style.



Click the **Add** button to add new block to the window style. The Custom Block Editor dialog opens.

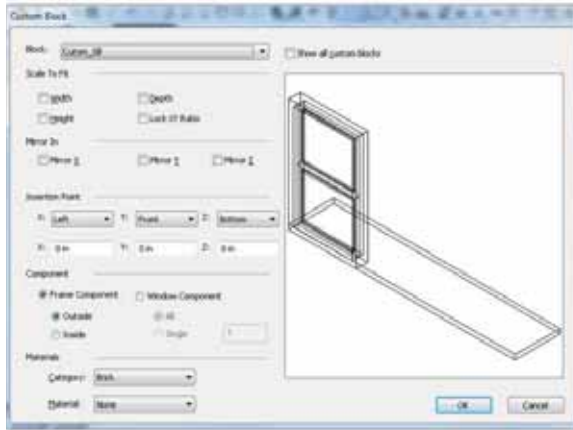


Click the Block combo-box and select the desired block.



just after block is selected you will see additional geometry displayed in preview

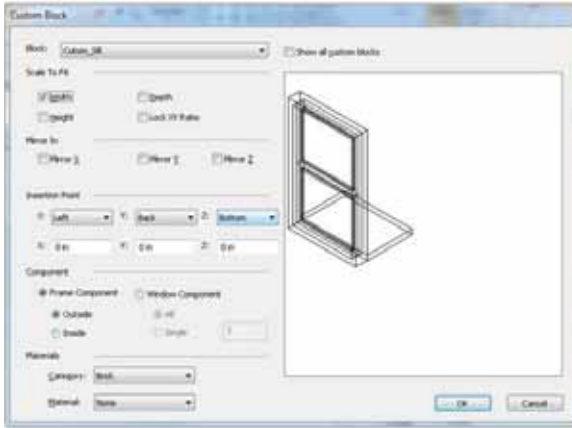
Right click on the preview pane to display pop-up menu and select Isometric\_SE item from it (to have better point of view)



Now let's adjust the block size and position.  
Check Width check box (to scale the block width to the width of window).



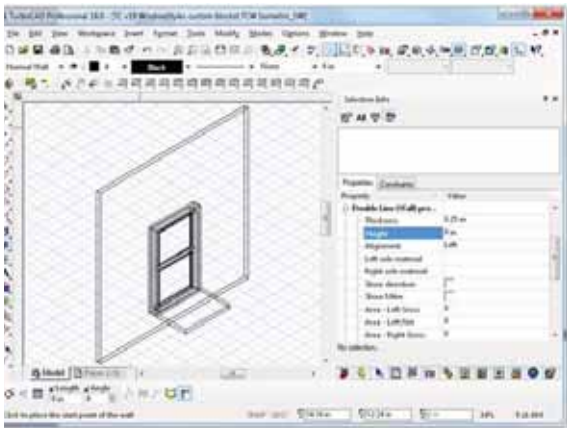
Select Back from Y: combo-box (to place the block to back side of window)



Click OK.

Click OK again.

Now create a wall in drawing and insert a window (of Standard style) into it.



## Wall Styles

The Wall tool is used to insert Walls. Every wall is based upon a defined Wall style created and managed in the Style Manager.

**NOTE:** If you want to save styles to a template, see “Saving Styles as Templates” on page 501.

In the Style Manager, there is one style, “Standard,” listed under “Wall Styles.” A preview showing a previous of the Wall style is shown on the lowest pane of the palette.

1. You can change the “Standard” style, but if you want to preserve this style, make sure “Standard” is highlighted and click **Create New Style**.
2. Assign a name. This creates a new style which is a copy of “Standard.”

## Component Walls

Walls are composed of components with each component representing a part of the walls geometry. By default every Wall Style has at least one component of the “Standard” type. Components are essentially long boxes defined by the components properties.

### To add components to a wall style

1. Select that wall style in the Style Manager.
2. Increase value in the **Number of Components** field to the desired value.

### To delete components from a wall style

1. Select that wall style in the Style Manager.
2. Decrease the value in the Number of Components field to the desired value.

**WARNING:** Components are added sequentially to the Wall style. If you increase the number of components the new components will be added to the bottom of the list of components. If you decrease the number of components the components at the end of the list will be deleted. The best practice is to add, build and complete each component in turn before adding more.

## Component Wall Properties

There are nine categories of wall properties. Several of them have sub-properties:

**Name:** This property is used to specify the name of the component

**Pen Color:** This property is used to specify the Pen Color of the component.

**Width:** These properties are used to specify the width of the component.

**Edge Offset:** These properties are used to specify the horizontal placement of the component.

**Bottom Elevation:** These properties are used to specify the bottom of the component.

**Top Elevation:** These properties are used to specify the top of the component.

**Dimension:** These properties are used to specify the width of the component.

**Brush:** These properties are used to specify the brush style of the component.

**Component material:** This property is used to specify the material of the component.

**Component draw priority:** This property is used to specify how the component will interact and intersect with other components and walls.

## Direction

Walls have direction with a right and left side. Whether a side is Left or Right depends on the wall's direction. To determine side is by assuming you are standing at the start point wall and facing the end point of the wall. Another way to visualize In other words, if a wall is drawn from left to right, the side toward the top of the screen is the left side of the wall, and the side toward the bottom of the screen is the right side of the wall. You can show a walls direction by checking the **Show Direction** on the **Wall** page of the wall's properties. Horizontal values use for defining components may have a negative or positive value. The left side of the wall is the positive direction, and the right side of the wall is the negative direction.

## Width

The width of a wall is measured from the Edge Offset of the wall. There are four Width properties that combine to set the actual width of a component:

**Width:** this value sets base width value for the component.

**Plus Wall Width:** if this value is checked, the wall's modified Width value will be added to the Component's Width value to specify the total width of the component.

**Operator:** specifies how the Value will modify the wall's Width (added, subtracted, multiplied, or divided) before it is added to the components Width.

**Value:** this number is used to modify the wall's Width before it is added to the component Width.

---

NOTE: A wall's width is specified in the wall's properties

---

These for properties combine in the following pattern:

Actual Component Width = Width + (Wall Width <operator> Value) - depending upon the operator selected. In other words:

Actual Component Width = Width + (Wall Width + Value) - if adding

Actual Component Width = Width + (Wall Width - Value) - if subtracting

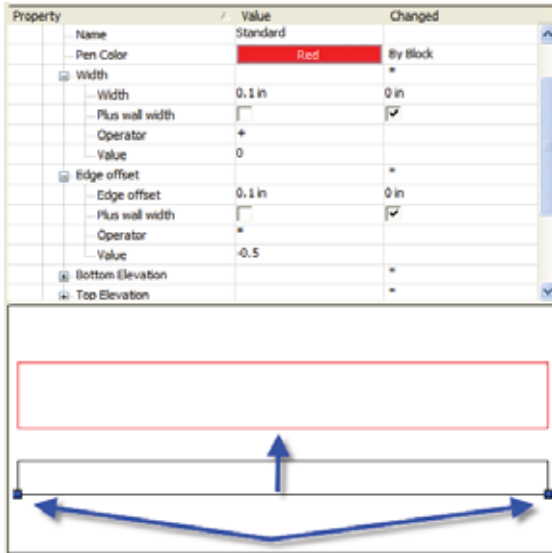
Actual Component Width = Width + (Wall Width \* Value) - if multiplying

Actual Component Width = Width + (Wall Width / Value) - if dividing

So, if the Width of the component is 2 feet, and the Width of the wall is 1 foot, if the operator is Divide, and the Value is set to 4, the components actual width will be 2.25 feet.) Or  $2 + (1/4)$ .

## Edge Offset

There are four Width properties that combine to set the Edge Offset of a component: Offsets are measured from the baseline of the wall. The baseline of a wall is indicated in the Style Manager by the two blue nodes. The direction arrow of always resides on the baseline.



**Edge Offset:** this value sets base offset value for the component.

**Plus Wall Width:** if this value is checked, the wall's modified Width value will be added to the Component's Edge Offset value to specify the total offset of the component.

**Operator:** specifies how the Value will modify the wall's Width (added, subtracted, multiplied, or divided) before it is added to the components Edge Offset.

**Value:** this number is used to modify the wall's Width before it is added to the component Edge Offset.

---

NOTE: A wall's width is specified in the wall's properties

---

These properties combine in the following pattern:

Actual Offset = Edge Offset + (Wall Width <operator> Value) - depending upon the operator selected. In other words:

Actual Offset = Edge Offset + (Wall Width + Value) - if adding

Actual Offset = Edge Offset + (Wall Width - Value) - if subtracting

Actual Offset = Edge Offset + (Wall Width \* Value) - if multiplying

Actual Offset = Edge Offset + (Wall Width / Value) - if dividing

So, if the Edge Offset of the component is 4 inches, and the Width of the wall is 6 inches, if the operator is Add, and the Value is set to 4, the components actual offset will be 14 inches.) Or  $4+(6+4)$ .

## Bottom Elevation

The Bottom Elevation is specified by two properties:

**Offset:** this value specifies the distance between the From Elevation and the bottom of the component.

**From Elevation:** this setting provides four positions along the wall height, from which the Offset can be measured.

- **Wall Bottom** — this is the absolute lowest point of the wall as measured from the position of the wall on the Z axis (Z location). It is normally the same as the Baseline, but it may vary if a Wall Modifier has been applied to the wall.
- **Wall Top** — this is the absolute highest point of the wall as measured from the position of the wall on the Z axis (Z location). It is normally the same as the Base Height, but it may vary if a Wall Modifier has been applied to the wall.
- **Base Height** — this is the Height value specified in the Wall's Height property.
- **Baseline** — this is the walls Z location, its position on the Z axis

## Top Elevation

The Bottom Elevation is specified by two properties:

**Offset:** this value specifies the distance between the From Elevation and the top of the component.

**From Elevation:** this setting provides four positions along the wall height, from which the Offset can be measured.

- **Wall Bottom** — this is the absolute lowest point of the wall as measured from the position of the wall on the Z axis (Z location). It is normally the same as the Baseline, but it may vary if a Wall Modifier has been applied to the wall.
- **Wall Top** — this is the absolute highest point of the wall as measured from the position of the wall on the Z axis (Z location). It is normally the same as the Base Height, but it may vary if a Wall Modifier has been applied to the wall.
- **Base Height** — this is the Height value specified in the Wall's Height property.
- **Baseline** — this is the walls Z location, its position on the Z axis

## Dimension

Specifies how a components width is to be dimensioned with an AEC dimension. This setting will only have an affect if the Wall Width option in the AEC Dimension style is set to **Wall Components From Style**.

The options are:

- From Left Side
- From Center
- From Right Side

## Component Draw Priority

The Component Draw Priority is used to specify how components will draw, heal and clean up with other components and other walls. The following conditions apply with the Component Draw Priority:

- Components with the same Draw Priority will heal to each other.
- Components with a lower value will pass through components with a higher value.
- Draw Priority takes precedence over Draw order

## Schedule Styles

The **Schedule** tool can be used to create a table in your file detailing all selected walls, windows, doors, and/or slabs.

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NOTE: The **Fill Schedule Wizard** is another way to add a schedule, that scans for objects on specific layers. See "Fill Schedule Wizard" on page 595.

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NOTE: If you want to save styles to a template, see "Savings as Templates" on page 566.

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1. This example starts with a model that has walls and slabs. There are two types of windows and three types of doors.

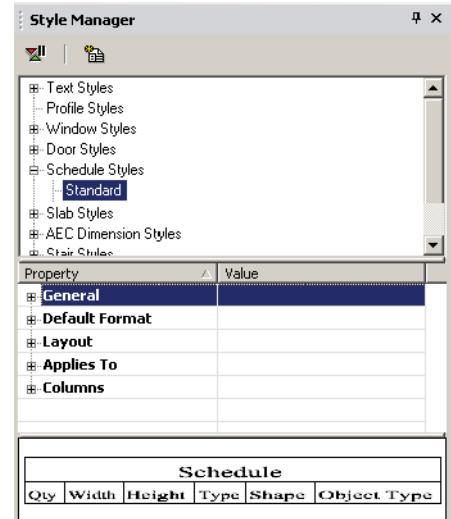



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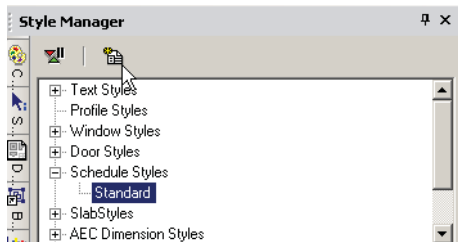
NOTE: For details on these components, see: "Window" on page 547 "Door" on page 548 "Slabs" on page 551

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In the Style Manager, there is one style, “Standard,” listed under “Schedule styles.”



- 1. You can change the “Standard” style, but if you want to preserve this style, make sure “Standard” is highlighted, then click **Create New Style**.

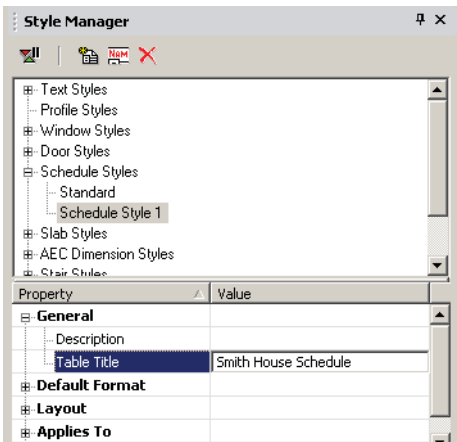


- 2. Assign a name or accept the default.

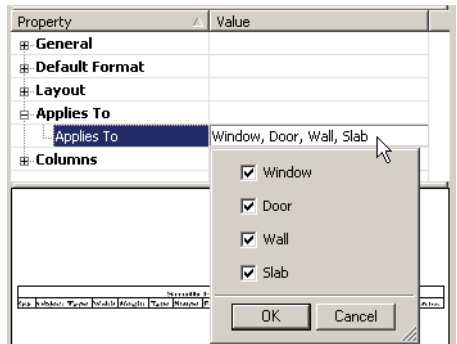


This creates a new style which is a copy of “Standard.”

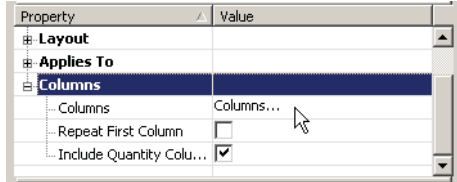
- 3. Highlight the new style, and open the **General** category. This is where you can define the schedule title.



- 4. **Default Format** and **Layout** contain various options for the text and cells in the table. Next to **Applies To**, click the **Value** field to specify which components you want to include in the schedule.



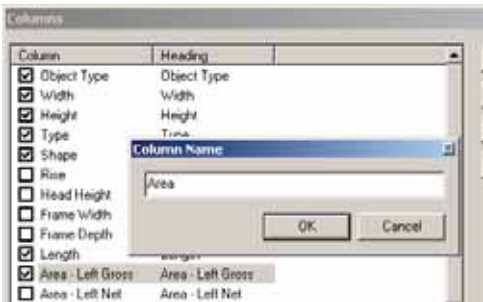
- 5. To specify the columns, click the **Value** field next to **Columns**.



6. This opens a menu from which you can select the schedule's columns. To move a column up or down, highlight it and click **Move Up** or **Move Down**. In this example, **Object Type** was moved to the top of the list.



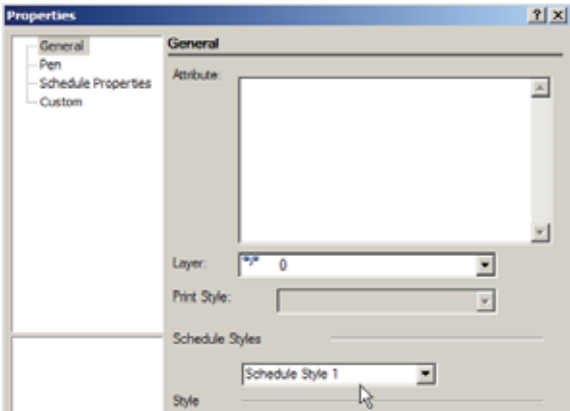
7. If you want to change the name of a column, click its name in the **Heading** field. In this example, “Area - Left Gross” was changed to “Area.”



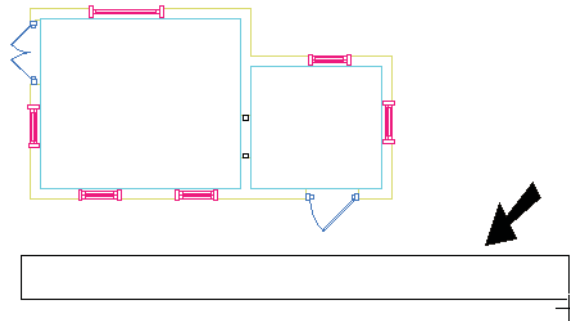
8. Click OK to end the column definitions.
9. This is the icon for **Schedule**. Right-click on the tool to set the tool's **Properties**



10. Open the **Schedule** tool's **Properties** to the **General** page.



11. Switch to **World Plan** view. Activate **Schedule**, and click two corners to define the schedule's width.



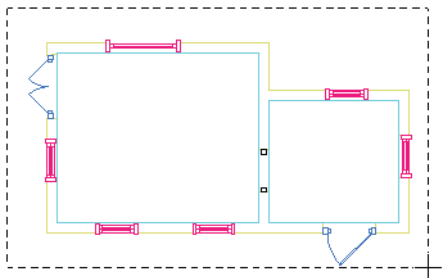
When first created, the schedule contains only the title and column headers.

Smith House Schedule							
Qty	Object Type	Width	Height	Type	Shape	Length	Area

12. To build the schedule, select **Add Object to Schedule**.



13. First click the schedule you want to add to, then select the windows and doors to include. Use Shift to select multiple objects, or use a selection window.



14. When finished, select **Finish** from the local menu or Inspector Bar. The windows and doors appear in the schedule.

Smith House Schedule						
Qty	Object Type	Width	Height	Type	Shape	Area
1	Wall	6.00	100.00		?	120.00 79.17
2	Wall	6.00	100.00		?	102.00 66.67
1	Wall	6.00	100.00		?	120.00 79.25
2	Wall	6.00	100.00		?	80.00 51.48
1	Wall	6.00	100.00		?	75.00 47.92
1	Door	36.00	72.00	Double-Dhung	Half Round	? ?
1	Door	30.00	84.00	Double-Dhung	Rectangular	? ?
5	Window	24.00	40.00	Awning Transom	Rectangular	? ?
1	Door	24.00	70.00	Double-Dhung	Rectangular	? ?
2	Slab	?	?		?	? ?
1	Window	42.00	42.00	Awning Transom	Oval	? ?

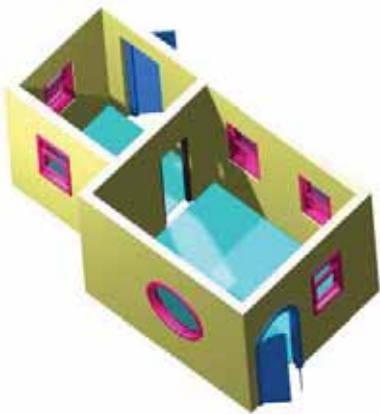
15. If you want to remove objects, use **Remove Object from Schedule**. All schedule objects are highlighted; select the windows and doors you want to remove. Select Finish, and the objects are removed from the schedule.

Fill Schedule Wizard



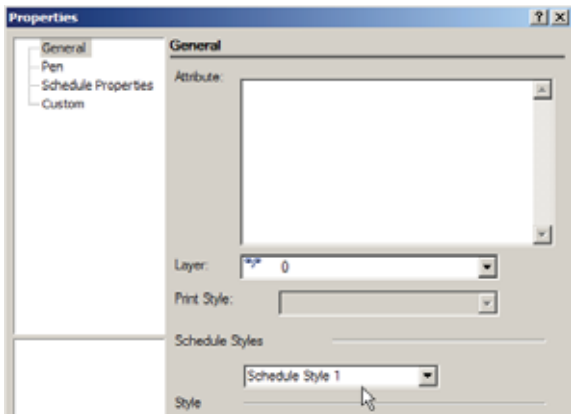
The **Fill Schedule Wizard** enables you to scan your drawing for objects to include in the schedule; you do not have to select objects manually. The wizard scans for specific types of objects on specific layers, which is useful for large drawings which may have objects on invisible layers, or in which slabs are not easy to select because of multiple floors.

1. This examples uses a model that has walls and slabs. There are two types of windows and three types of doors.

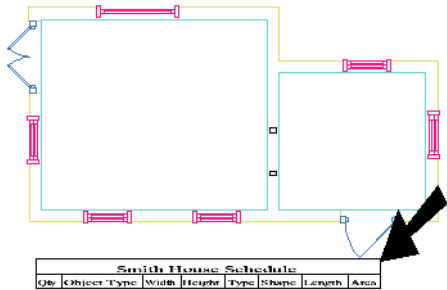


2. Use the Style Manager to create a schedule style (see "Schedule Styles" on page 592).

3. To create the schedule with the new style, open the **Schedule** tool's **Properties** to the **General** page.



4. Switch to **World Plan** view and use the **Schedule** tool to place the schedule, by clicking two corner points.



5. Select **Fill Schedule Wizard**.

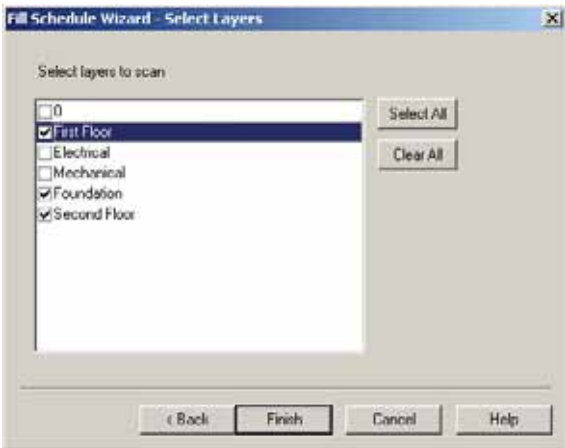


6. Click the schedule you want to fill.

7. In the wizard, check the types of objects to include. (The list that appears in the window depends on what was defined as part of the schedule style.) Then click **Next**.



8. Select the layers you want to include in the scan, and click **Next**.



NOTE: For details on layers, see "Layers" on page 116.



The schedule is filled with all objects found on the selected layers.

Smith House Schedule						
Qty	Object Type	Width	Height	Type	Shape	Length
1	Wall	6.00	100.00		?	120.00
1	Wall	6.00	100.00		?	102.00
1	Wall	6.00	100.00		?	120.12
1	Wall	6.00	100.00		?	102.00
1	Wall	6.00	100.00		?	80.00
1	Wall	6.00	100.00		?	75.00
1	Wall	6.00	100.00		?	80.12
1	Door	36.00	72.00	Double-Dhung	Half Round	?
1	Door	30.00	84.00	Double-Dhung	Rectangular	?
5	Window	24.00	40.00	Awning Transom	Rectangular	?
1	Door	24.00	70.00	Double-Dhung	Rectangular	?
2	Slab	?	?		?	?
1	Window	42.00	42.00	Awning Transom	Oval	?

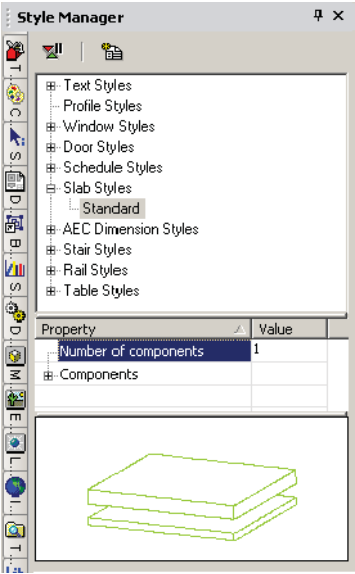
Slab Styles

Available in TurboCAD Platinum only

The Style Manager can be used to set various styles for slabs. For details on creating slabs, see "Slabs" on page 551.

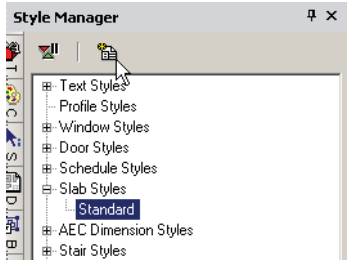
NOTE: If you want to save styles to a template, see "Savings as Templates" on page 566.

In the Style Manager, there is one style, "Standard," listed under "Slab styles." A preview showing a previous of the slab style is on the lowest pane.

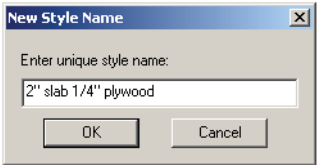


NOTE: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window.

- 1. You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted and click **Create New Style**.

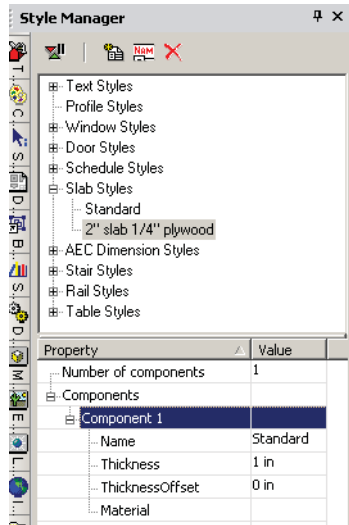


- 2. Assign a name.

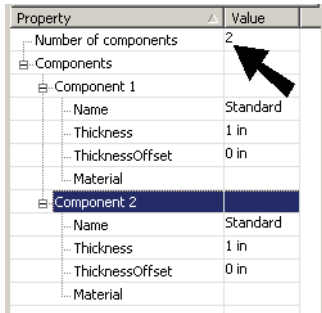


This creates a new style which is a copy of "Standard."

By default, a slab consists of one component, for which you can specify a name, thickness, offset, and material.

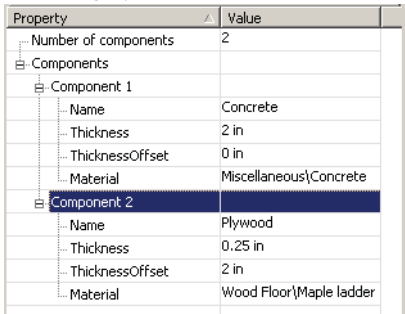


- 3. In this example, there will be two components. Change the number and press Enter, and an additional component is added.



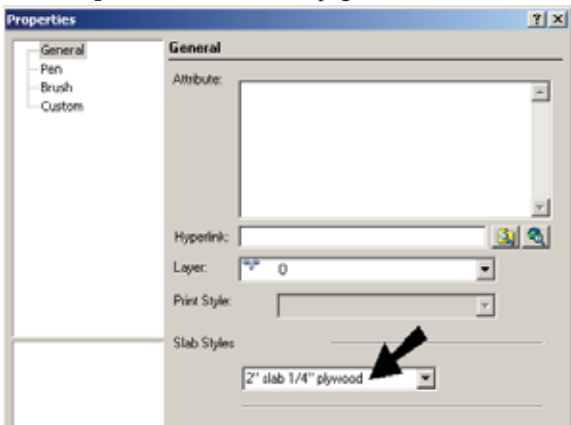
- 4. Component 1 is a 2” concrete slab with zero offset, which means the slab will be created flush with the bottom of the walls. Component 2 is 1/4” wood with an

offset of 2”, so that it will sit directly atop the concrete slab. To define material, click inside the field, then select a category and material.

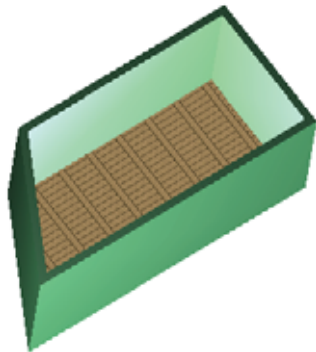


NOTE: For details on materials, see "Materials" on page 522.

- 5. These are the icons for **Add Slab by Click** and **Convert to Slab**. Right-click on either icon to set the tool's **Properties**, and click the relevant icon to create the slab.
- 6. To create the slab with the new style, open the tool's **Properties** to the **General** page.



7. Create the slab using one of the slab tools.



8. In this example, if you remove the walls, both components of the slab can be seen.

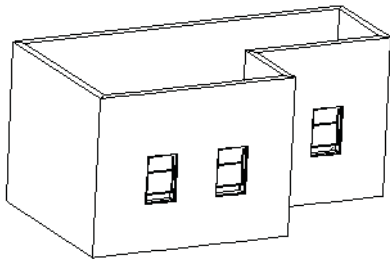


### AEC Dimension Styles

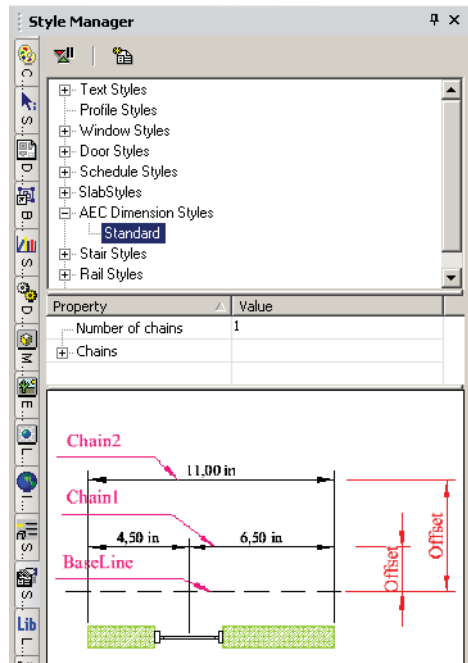
The Style Manager can be used to set various styles for wall dimensions. These dimensions are created with the **Wall Dimension** tool.

*NOTE: If you want to save styles to a template, see "Savings as Templates" on page 566.*

1. Start with some walls, and add openings like windows or doors.

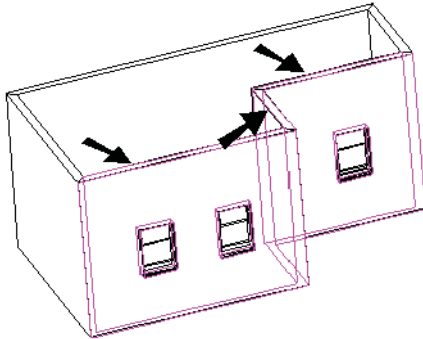


In the Style Manager, there is one style, "Standard," listed under "AEC Dimension styles." This style has one chain, and a preview showing a layout of AEC Dimension styles is below.

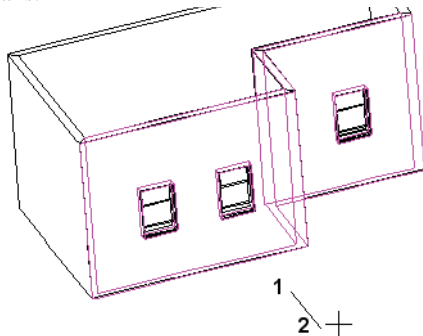


*NOTE: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window.*

2. Activate **Wall Dimension**, and select one chain of walls, using the Shift key for multiple selection. Select **Finish Selection**.

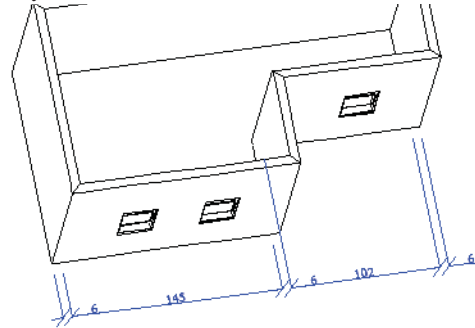


3. The next click defines the location of the dimension baseline. The subsequent click defines the dimension angle - if you want the dimensions to proceed along the wall chain, the angle should be perpendicular to the walls.



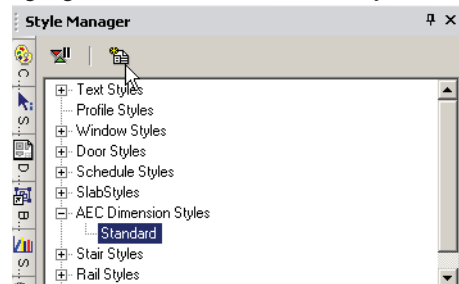
*TIP: To ensure a vertical or horizontal angle line, you can use the X or Y lock in the Coordinate Field.*

4. This is the resulting dimension chain in the current style.

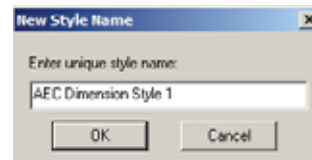


*NOTE: If you want to change the properties of the dimensions themselves, such as font or leader lines, see "Dimension Properties" on page 332.*

5. You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted, then click **Create New Style**.

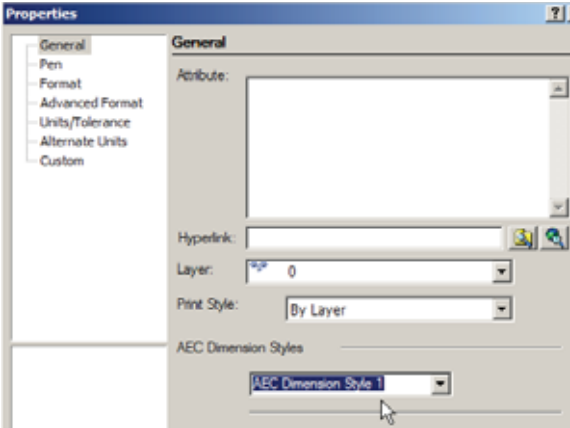


6. Assign a name or accept the default.

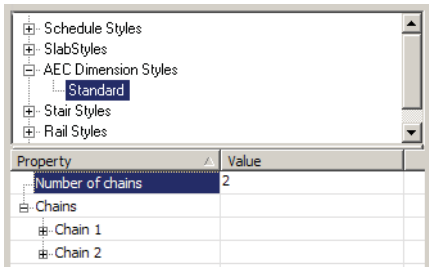


This creates a new style which is a copy of "Standard."

7. To change the current wall dimension to the new style, open its **Properties** to the **General** page.

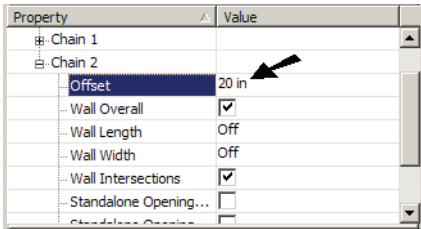


8. With the new style highlighted, change the **Number of chains** to 2.

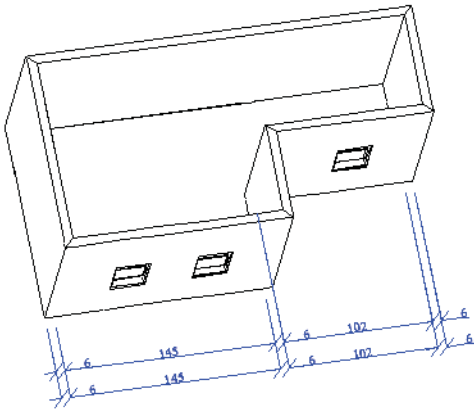


**NOTE:** If **Delay Style Modification** is enabled, you can see the old and new values for each field. Then you can update the style by clicking **Apply Style Changes**. If there is no delay, then all changes are implemented immediately.

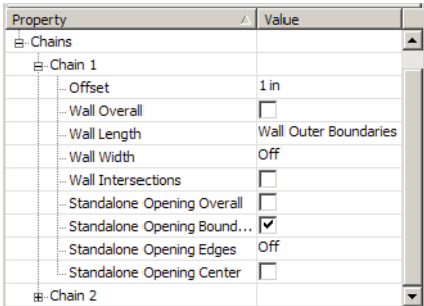
9. To separate Chain 2 from Chain 1, add an **Offset**.



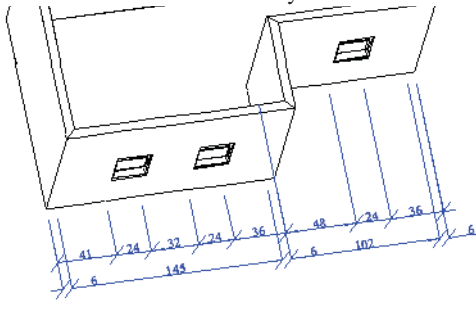
Chain 1 is offset from the baseline, which is where you first clicked to define the dimension line's location. Chain 2's offset is its distance from Chain 1.



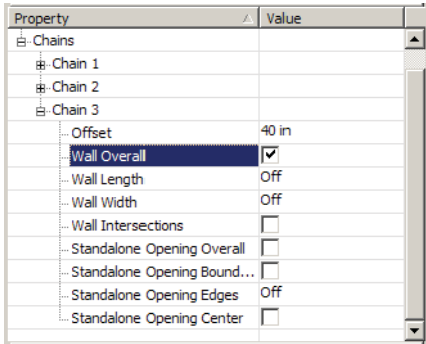
10. Open the properties for Chain 1 and make these changes.



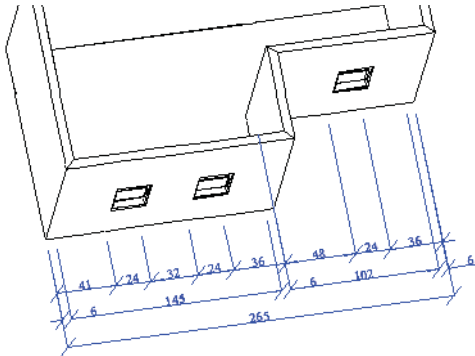
Now Chain 1 dimensions the openings. Chain 2 is still the same as the "Standard" style.



11. To add an overall dimension, add Chain 3 and offset it from Chain 2. Only **Wall Overall** should be checked.



This is the result.



### Dimension Styles

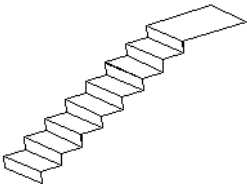
For styles and Style Manager usage for standard dimensions see “Dimension Styles” on page 337.

### Stairs Styles

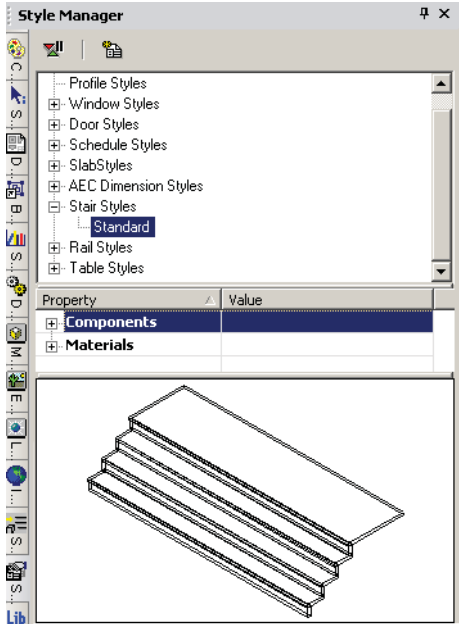
The Style Manager can be used to set various styles for stairs. For details on the types of stairs you can create, see “Stairs” on page 554.

NOTE: If you want to save styles to a template, see “Savings as Templates” on page 566.

1. Start with a staircase. This example uses a **Simple Stair**.



In the Style Manager, there is one style, “Standard,” listed under “Stair styles.” This style has two categories: components and materials. The graphic at the bottom shows a preview of the style.



NOTE: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window.

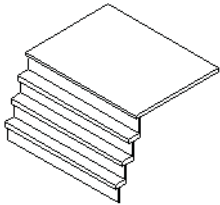
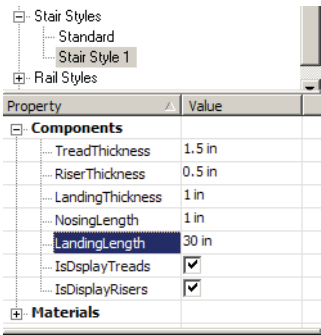
2. You can change the “Standard” style, but if you want to preserve this style, make sure “Standard” is highlighted, then click **Create New Style**.



3. Assign a name or accept the default.

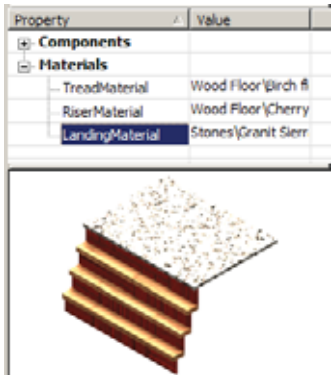


4. The new style is a copy of the “Standard” style. To make some changes, open **Components** and change the parameters, such as tread and riser thickness, etc. **Nosing Length** is how far the tread extends past the riser below. To view your changes, you can click inside the preview window to zoom in. (If you right-click in the preview window, a menu appears that enables you to change the view as well as render style.)



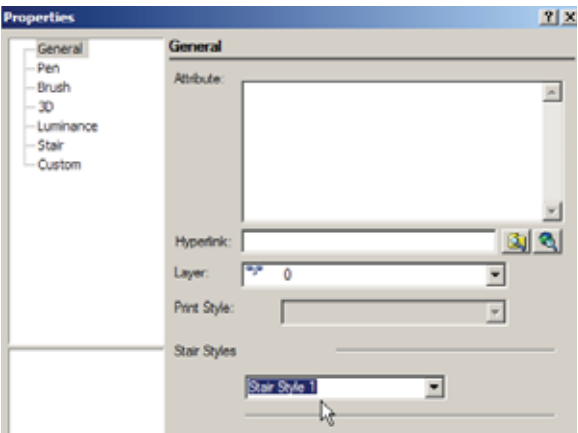
NOTE: Other stair properties, such as stair width, tread depth, and riser height, are set on the **Stair** page of the stair's **Properties**.

5. Open the **Materials** category, in which you can assign different materials to the treads, risers, and landing. Set the preview to **Quality Rendering** to see the materials.



NOTE: For details on materials, see "Materials" on page 522.

6. To change the staircase in the model so that it has the new style, open its **Properties** to the **General** page. All styles defined for stairs will be available under **Stair styles**.



The staircase now has the properties of the new style.



### Rail Styles

Available in TurboCAD Pro and Platinum only

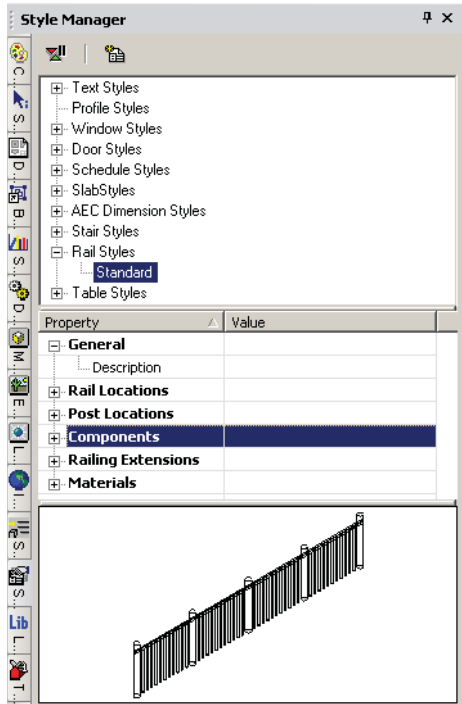
The **Railing** tool can be used to create a standalone, linear railing, or a railing along a staircase.

NOTE: If you want to save styles to a template, see "Savings as Templates" on page 566.

Menu: Tools / Architecture / Railing

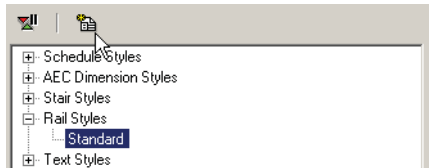


In the Style Manager, there is one style, "Standard," listed under "Rail styles." A preview showing a layout of this rail style is below.



NOTE: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window.

1. You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted, then click **Create New Style**.



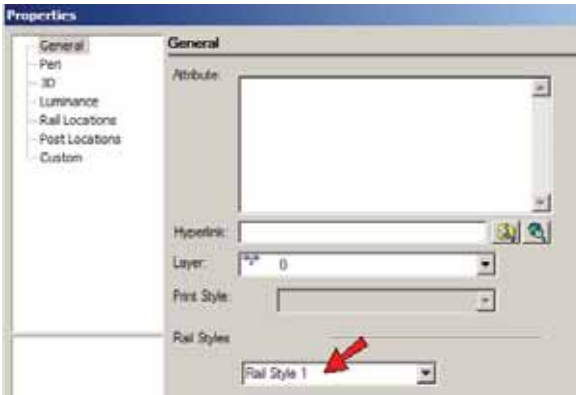


2. Assign a name or accept the default.



This creates a new style which is a copy of “Standard.”

3. To create the railing with the new style, open the tool’s **Properties** to the **General** page.



4. Highlight the new rail style, and open the **Rail Locations** category. This category sets which rails are displayed, their heights along horizontal and sloped segments, and their offsets from the vertical posts. In this example, only a **Handrail** is included.

Property	Value
<b>General</b>	
<b>Rail Locations</b>	
Allow vary	<input type="checkbox"/>
Include Guardrail	<input type="checkbox"/>
Guardrail Horizontal Height	2.1 in
Guardrail Sloping Height	2.1 in
Guardrail Offset from Post	0.15 in
Guardrail Side for Offset	Right
Include Handrail	<input checked="" type="checkbox"/>
Handrail Horizontal Height	30 in
Handrail Sloping Height	30 in
Handrail Offset from Post	0.5 in
Handrail Side for Offset	Auto
Include Bottomrail	<input type="checkbox"/>

5. Open **Post Locations**. This category sets the types of posts included, and their vertical extensions relative to rails. **Fixed posts** are placed at the ends and corners,

**Dynamic posts** are placed at set intervals along the railing. **Balusters** are placed between dynamic posts. In this example, only balusters are included.

Property	Value
<b>Rail Locations</b>	
<b>Post Locations</b>	
Allow vary	<input type="checkbox"/>
Fixed Posts	<input type="checkbox"/>
Extension of ALL Posts from Top...	0.3 in
Extension of ALL Posts from Flo...	0 in
Fixed Posts at Railing Corners	<input type="checkbox"/>
Dynamic Posts	<input type="checkbox"/>
Dynamic Posts Max. Center to ...	36 in
Balusters	<input checked="" type="checkbox"/>
Extension of Balusters from Floo...	2 in
Balusters Max. Center to Center...	4 in
Override Stair Tread Length	<input type="checkbox"/>
Number Balusters per tread	2

NOTE: You can either specify the number of balusters per tread, or baluster spacing. If the number per tread is entered, this will by default override the spacing. If you check **Override**, the spacing value will be used.

6. Open **Components**, in which you specify the shapes and dimensions of rails and posts.

Property	Value
<b>Components</b>	
Guardrail Profile Name	Rectangular
Guardrail Scale	Scale To Fit
Guardrail Width	0.1 in
Guardrail Depth	0.1 in
Guardrail Justification	Middle Center
Handrail Profile Name	Circular
Handrail Scale	Scale To Fit
Handrail Width	2 in
Handrail Depth	2 in
Handrail Justification	Middle Center
Bottomrail Profile Name	Rectangular

7. Open **Railing Extensions**, in which you can specify horizontal extensions of rails, relative to posts and landings.

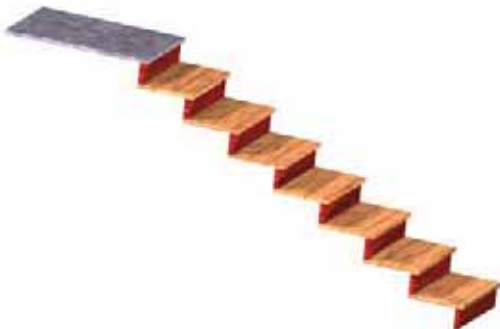
Property	Value
<b>Railing Extensions</b>	
... Allow vary	<input type="checkbox"/>
... Use Stair Landing Extension at Floor..	<input checked="" type="checkbox"/>
... Handrail Ext. at Top of Entire Stair	0.5 in
... Add Tread Length to Handrail Ext. ...	<input type="checkbox"/>
... Handrail Ext. at Bottom of Entire St...	0.5 in
... Add Tread Length to Handrail Ext. ...	<input type="checkbox"/>
... Guardrail Ext. at Top of Entire Stair	0.5 in
... Add Tread Length to Guardrail Ext....	<input type="checkbox"/>
... Guardrail Ext. at Bottom of Entire S...	0.5 in
... Add Tread Length to Guardrail Ext ...	<input type="checkbox"/>
... Use Stair Landing Extensions at La...	<input type="checkbox"/>
... Handrail Ext. at Top of Flight	4 in
... Add Tread Length to Handrail Ext. ...	<input type="checkbox"/>
... Handrail Ext. at Bottom of Flight	4 in
... Add Tread Length to Handrail Ext. ...	<input type="checkbox"/>

8. Finally, open **Materials**, in which you can specify the material for each post and rail type.

Property	Value
<b>General</b>	
<b>Rail Locations</b>	
<b>Post Locations</b>	
<b>Components</b>	
<b>Railing Extensions</b>	
<b>Materials</b>	
... Guardrail Material	
... Handrail Material	Metals\Polished
... Bottomrail Material	
... First Fixed post Material	
... Last Fixed Post Material	
... Other Fixed Post Material	
... Dynamic Post Material	
... Baluster Material	Metals\Iron

NOTE: For details on materials, see "Materials" on page 522.

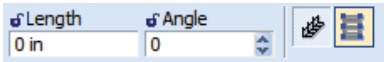
9. This example is for a stair railing, so start by creating a staircase. (For details, see "Stairs" on page 554.)



10. Activate **Railing**. By default, **Stair Railing** is active



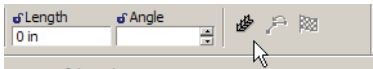
11. To place rails on both sides of the stair use the Attach to Both Sides option.



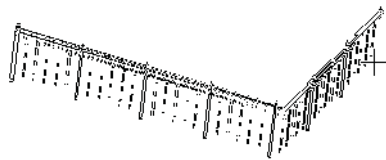
12. Select the staircase, and the railing is created.



The other type of railing is standalone, defined by linear segments. To create this, make sure **Stair Railing** is not selected.



This railing is defined by one or more segments, similar to a polyline or wall. A preview of the railing appears while defining the segments. When finished, select **Finish** from the local menu or Inspector Bar.



This example includes a handrail, guardrail, and bottom rail. The posts include a fixed post at the end, dynamic posts along the length, and balusters between bottom rail and guard rail.



### Table Styles

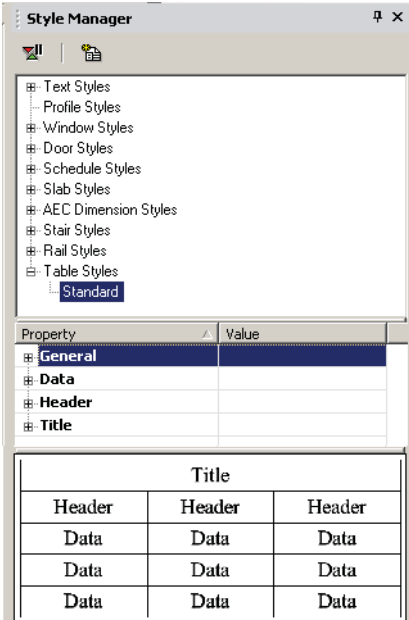
Tables can be used to define table properties. Table can be created using **Insert / Table** (see "Tables" on page 619) or they are created automatically when you create a report (see "Custom Properties, Database, and Reports" on page 624).

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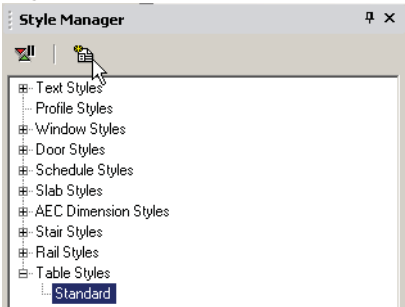
*NOTE: If you want to save styles to a template, see "Savings as Templates" on page 566.*

---

In the Style Manager, there is one style, "Standard," listed under "Tables."



1. You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted, then click **Create New Style**.

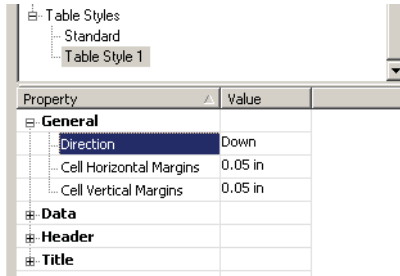


2. Assign a name or accept the default.

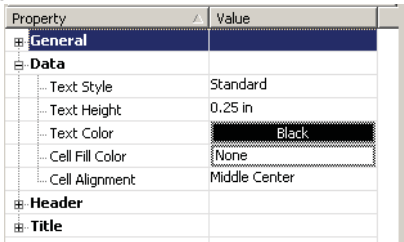


This creates a new style which is a copy of “Standard.”

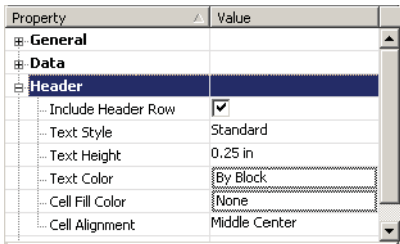
3. Highlight the new style, and open the **General** category. This is where you can define margins and direction. If **Direction** is down, the header and title appear at the top of the table.



4. The **Data** category contains properties of the text and colors in the main cells of the table (not the headers or title). For details on defining text, see "Text Styles" on page 566.



5. The **Header** and **Title** categories are similar to **Data**. They have the additional option whether to display header or title rows.



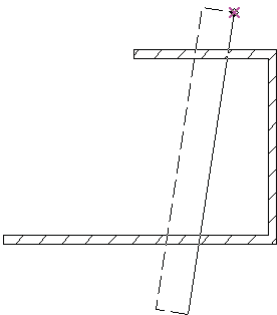
## Section/Elevation

### Section Line



Create a section line.

1. Define the segment start point.
2. Define the end point, or specify the length and angle in the inspector bar.



### Vertical



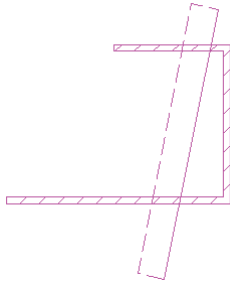
Create a vertical section.

1. Create a 3D element. Now use the section line tool (See "Section Line" on page 608) to cross the pertinent 3D elements.
2. Select vertical section tool. Right click to open the local menu. Select either 2D Section /Elevation or 3D Section /Elevation.

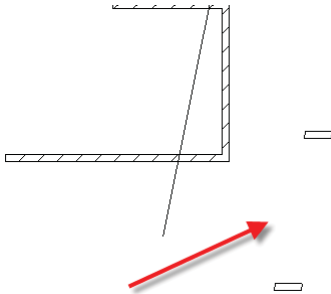
Alternatively, you can also select 2D Section /Elevation or 3D Section /Elevation in the inspector bar.



3. Select the Section line.
4. Select the pertinent 3D elements.



5. Select finish from local menu or in the inspector bar. You will then have the section and will be prompted to select section position. Click in the drawing to place the section.



You can edit the size, orientation and position of the section line using edit tool. (see "Edit Tool" on page 219)

3. Select the Section line.
4. Select the pertinent 3D elements.
5. Select finish from the local menu or in the inspector bar. You will then have the section and will be prompted to select section position. Click in the drawing to place the section.

You can edit the size, orientation and position of the section line using the edit tool. (see "Edit Tool" on page 219)

## House Wizard Toolset

*Available in TurboCAD Pro, Platinum, and Deluxe only*

The House Wizard is a set of tools designed to allow for accelerated layout of a house design. It is structured to provide a set of room boxes that can be snapped together quickly. Once the initial layout of spaces is complete the click of a single button will generate a preliminary 3D model of the house, including walls, slabs, and doors. You can then quickly finalize the model to your needs.

In order for the House Wizard to work correctly you must start with either the "House Template", or the "House Template Metric".

To get the maximum utility out of the House Wizard you should turn on the House Wizard toolbar. Go to **Customize**, navigate to the Toolbars page and select House Wizard.



## Horizontal



Create a horizontal section.

1. Create a 3D element. Now use section line tool (See "Section Line" on page 608) that cross the pertinent 3D elements.
2. Select horizontal section tool. Right click to open the local menu. Select either 2D Section /Elevation or 3D Section /Elevation.

Alternatively, you can also select 2D Section /Elevation or 3D Section /Elevation from the inspector bar.

## House Wizard

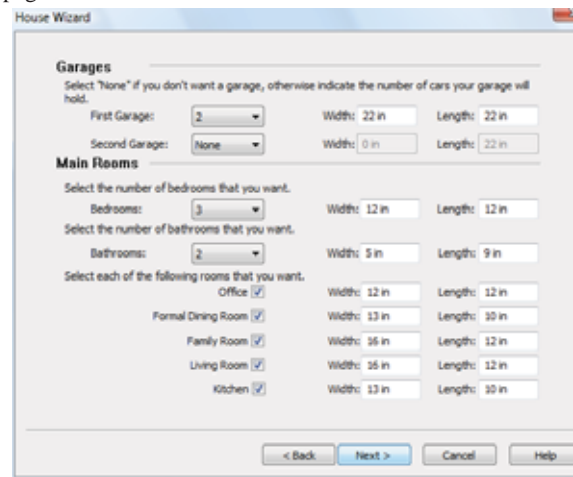
The first step in using the House Wizard toolset is to use the House Wizard itself. The Wizard will generate all of the room boxes that you select.

The Page One of the wizard is simply an introduction, Click Next to move to Page two



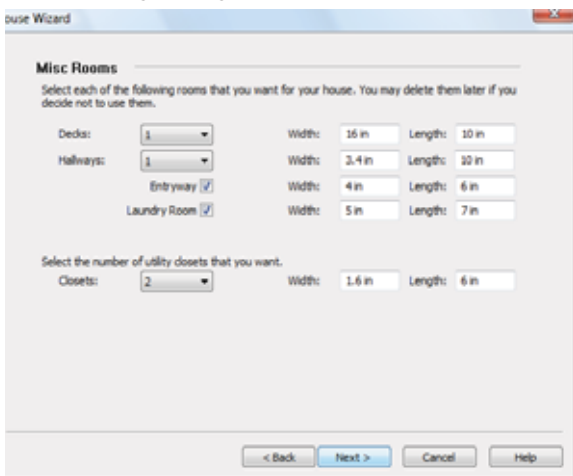
Page Two of the wizard is used to specify the number and size of the Garages and Main Rooms in your house design. The Garage drop-down controls allow you to specify the number of cars the garage is to hold. The Bedroom and Bathroom drop-down controls allow you to specify the

number of those rooms you want. You can specify the additional rooms you want using the check boxes on the page.



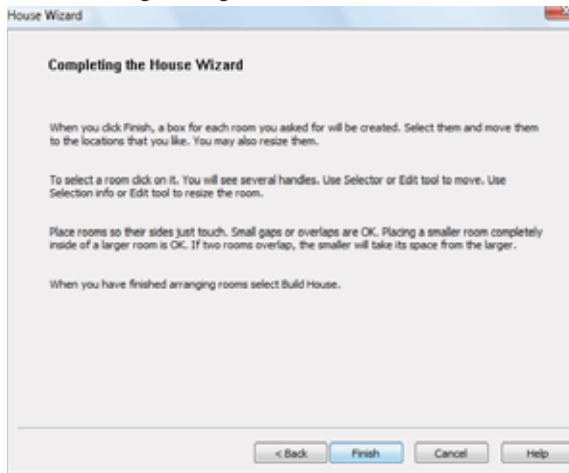
While on this page you can use the Width and Length fields to override the default sizes for the rooms.

Click Next to go to Page Three of the Wizard.



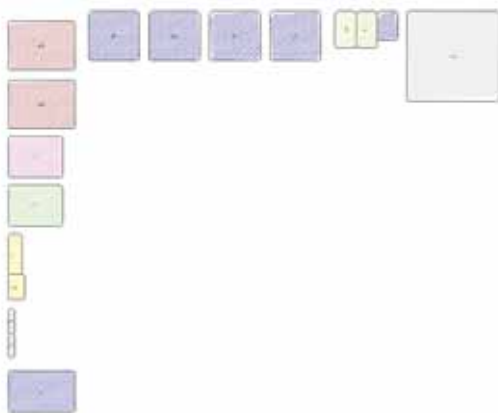
On Page Three of the Wizard you can specify the size and number of decks you want in your design, as well as miscellaneous rooms and passageways. For all of these areas, you have the option of choosing None.

Click Next to go to Page Three of the Wizard.



Page Four of the Wizard gives you some final tips on how to use the boxes that the wizard generates. Read the info carefully then click the Finish button.

The result of the Wizard will be a 2D layout of all the boxes you generated.



## Arranging and Snapping Room Boxes

To arrange the room boxes into your design simply pick up each box and drag it into position, and drop it.

- If you position the boxes so that are aligned closely edge-to-edge the dropped box will "snap" to the adjacent box with edges aligned, and if possible with corners aligned.
- It is a good idea to turn Off Dynamic snaps while you are arranging your boxes, since the two types of snapping may conflict in some instances.
- You can resize the boxes while they are selected by using the blue handles on the edge of the selector. If you hover over the edge of an adjacent box while resizing the edge you are adjusting will snap to align with the edge you are hovering over.
- For best results, it is suggested that you put small ancillary rooms within larger rooms e.g. bedrooms.

Once you are done arranging your rooms, click the Build House button.

Here is an example of some rooms laid out together:



Notice the overlaps between the rooms.

Here is the result after pushing the Build House button:



Notice how the House Wizard has automatically add room dimensions, calculated the individual room areas, and automatically placed doors at logical positions within the house.

## House Setup

This will open the House Wizard Setup dialog. With this dialog you can specify which style elements will be used to generate each of the components of your house. Changes in these settings will only be reflected in your model after the design is rebuilt with Build House.



## Hide-Show Room Boxes

This option will hide room boxes if they are currently showing, or it will show room boxes if they are currently hidden. This function does not override layer visibility.

## Delete Room Boxes

This will delete all of the room boxes in you drawing.

Warning: You will not be prompted to verify that you want to delete the boxes, so if you accidentally delete the boxes, use Undo immediately.

## Insert Room Boxes

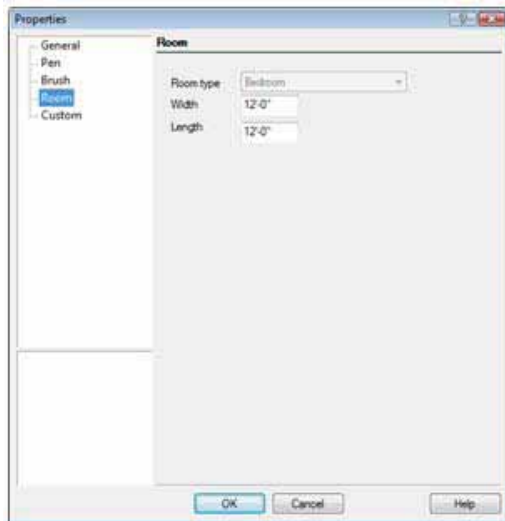
You do not have to use the House Wizard to generate your rooms, nor do you have to reuse the Wizard if you forgot a room. The Insert Room Box tools e.g. Insert Bedroom will allow you to add room boxes to your layout at any time.

Simply select an Insert Room Box tool then click at each location that you want to place that room type. The tool will remain active until you press ESC or choose a different tool.



## Changing Room Size While Inserting

When you are using the Insert Room tools you can dynamically change the size of the room you are inserting by changing the Width and Length parameters in the Inspector bar, or by using the local menu to access the tools properties.





# 14 Woodworking

## Layout

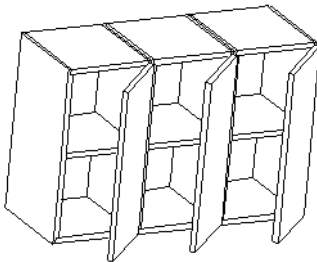
*Available in TurboCAD Pro and Platinum only*

The **Layout** tool can be used for furniture design, and glass or metal cutting. The results show the optimal way to cut rectangular sheets or planks into rectangular parts.

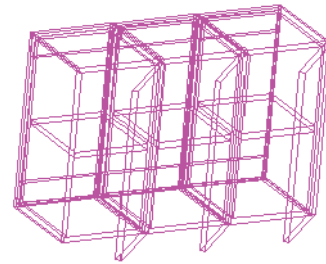


(This icon is found on the **Groups and Blocks** toolbar, which you can display by right-clicking in any toolbar area and selecting **Groups and Blocks**.)

1. Start with the object or objects you want to lay out. This example uses a parametric symbol from the “Cabinets” folder of the Library palette. (The object does not have to be a symbol or part.)



2. Activate **Layout**, and click two points of a selection window around the objects. You can skip selecting objects if you are going to manually specify the planks.



3. Select **Finish** from the local menu or Inspector Bar.
4. In the **Layout** window, check all of the pieces you want to include in the layout.

Plank ...	Width	Height	Quantity	Rotation
<input checked="" type="checkbox"/> 1	24 in	13 in	6	Yes
<input checked="" type="checkbox"/> 2	11.25 in	13 in	6	Yes
<input checked="" type="checkbox"/> 3	11.25 in	23.25 in	6	Yes
<input checked="" type="checkbox"/> 4	11.25 in	3 in	6	Yes
<input checked="" type="checkbox"/> 5	10.38 in	11.75 in	3	Yes

Add Plank...

Swap Width and Height ☒ Allow Planks Rotation

Sheet Parameters

Width: 52 in Height: 48 in Kerf: 0.13 in

Edge Trim

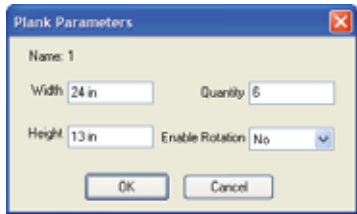
Left: 0.25 in Right: 0.25 in Top: 0.25 in Bottom: 0.25 in

< Back Next > Cancel

In the table you can:

Plank ...	Width	Height	Quantity	Rotation
<input checked="" type="checkbox"/> 1	24 in	13 in	6	No
<input checked="" type="checkbox"/> 2	11.25 in	13 in	6	No
<input checked="" type="checkbox"/> 3	11.25 in	23.25 in	6	No
<input checked="" type="checkbox"/> 4	11.25 in	3 in	6	No
<input checked="" type="checkbox"/> 5	10.38 in	11.75 in	3	No

- Double-click on any plank to open the **Plank Parameters** dialog. Where you can specify the properties of each plank individually, Including the quantity.



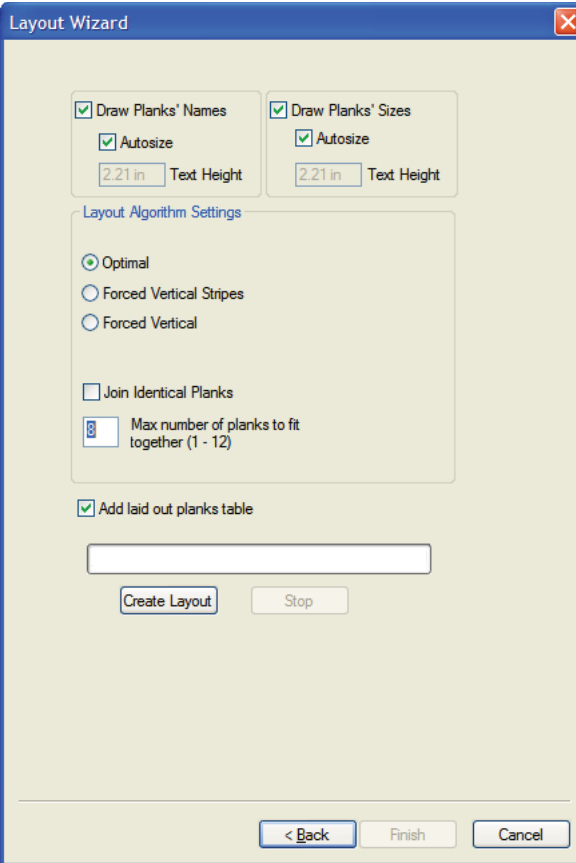
- Click **Add Plank**. Which will open the Plank Parameters dialog and allow you to manually create and specify planks.
- Click **Swap Width and Height**. Which will swap those values for the selected plank.
- Select or de-select **Allow Plank Rotation**. This will set the rotation parameter for all of the planks. If plank rotation is allowed the layout of the planks will be optimized to fit the sheet. If it is disallowed you can control how the planks will layout relative to the sheets grain.

At the bottom of the window you can specify:

- **Size of Board:** defines the board or sheets from which the pieces will be cut.
- **Kerf:** Defines the width of the cutting blade.
- **Edge Trim:** defines trimming cuts in case the boards have ragged or damaged edges.

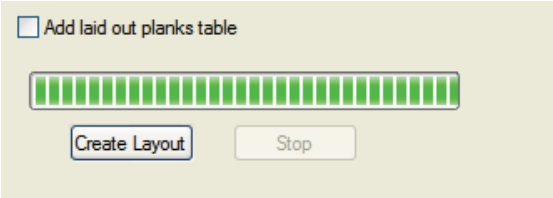
5. Click **Next**.

6. In the **Format** window, set the following options:

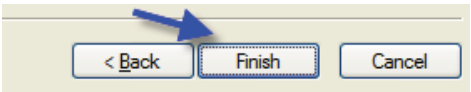


- **Draw Planks' Names:** If checked, block attributes containing the plank name will be assigned to each piece.
- **Draw Planks' Sizes:** If checked, block attributes containing the dimensions of the planks will be assigned to each piece.
- **Autosize:** Automatically chooses the size of the text.
- **Layout:** Check **Forced Vertical Stripes** if you want through cuts, check **Forced Vertical** if you want through cuts along with using **Join Identical Planks**, if you want otherwise, check **Optimal**.

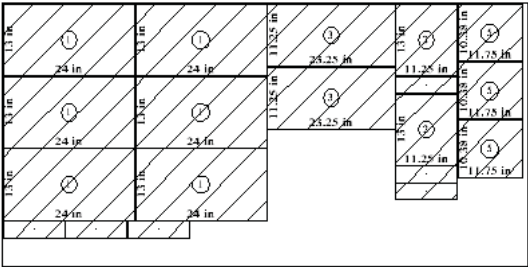
- **Algorithm options:** Check **Join Identical Planks** if you have more than six identical pieces. And set the maximum number of parts from which pieces will be cut.
  - Check **Add laid out planks table** to insert a table with all of the plank parameters after you inset the layout.
7. Click **Create Layout** to begin calculating the layout. (You can interrupt the process by clicking **Stop**.)



8. When the calculation process is complete, click **Finish**.



9. Click in the drawing to bring in the board layout.



If you checked **Add Name or Add Sizes**, you will see the blocks indicated on each piece. Identical pieces have the same block letter.

10. If you checked **Add laid out planks table**, click in the drawing to place the table with the layout data,

Quantity	Height	Width	Plank Name
6	13 in	24 in	1
6	13 in	11.25 in	2
6	23.25 in	11.25 in	3
6	3 in	11.25 in	4
3	11.75 in	10.38 in	5



# 15 Database, Tables, and Reports

## Tables

**Insert Table** enables you to insert an empty table, and **Modify Table** enables you to add or edit text in the table.

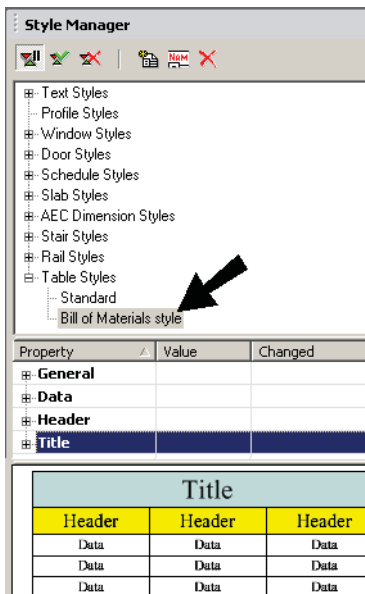
**Insert Table** is available on the **Draw** menu, and **Modify Table** is available on the **Modify** menu. Both tools are also available on the **Text** toolbar. You can display the **Text** toolbar by right-clicking on any toolbar area and selecting **Text**.



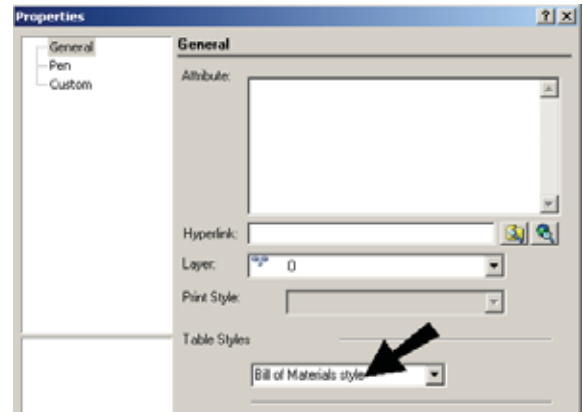
### Insert Table



Before creating a table, you can define a table style (see "Table Styles" on page 607). Styles are useful for creating tables with specific text properties, cell colors, and header / title rows.



To apply the style to the table, open the tool's **Properties** and select the style on the **General** page.



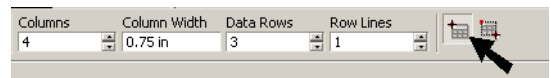
You can also change an existing table's style in **Properties**, or in the Selection Info palette.

There are two methods of inserting a table: **Specify Insert Point** and **Specify Window**.

### Specify Insert Point

With this method, you click the top left point of the table.

1. Make sure **Specify Insert Point** is selected in the local menu or Inspector Bar. The number of columns and rows, as well as column width and row lines (number of lines of text) is set in the Inspector Bar.



2. Click once in the file, and the table is inserted.


3. If you don't want to insert another table, press Esc or start a new tool.

- 4. If you need to change the size of an individual row or column, use the **Edit Tool**. See "Changing Rows and Columns, Merging Cells" on page 623.

**Specify Window**

With this method, you insert the table by clicking two corner points. You can either fix the cell dimensions, or you can fix the number of rows / columns. (You can also mix options for sizing and numbers for rows and columns.)

Make sure **Specify Window** is selected in the local menu or Inspector Bar.

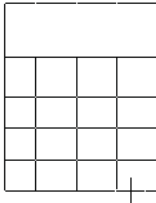


**Fixed Number of Rows / Columns**

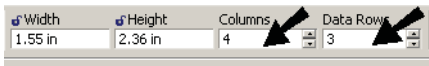
- 1. Activate both **Calculated Row Height** and **Calculated Column width**.



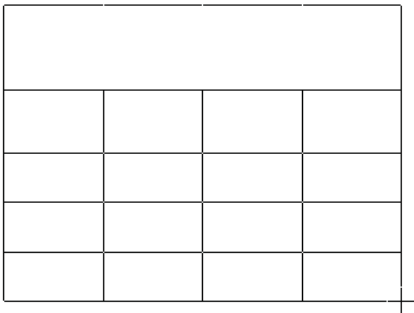
- 2. Click once to set the top left corner of the table.



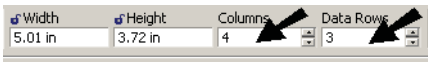
- 3. Set the number of columns and rows.



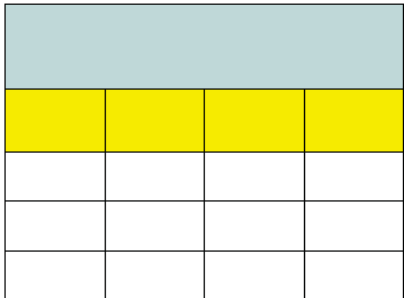
- 4. Move the mouse to size the table.



The number of cells remains the same, no matter how the table is sized.



- 5. Click the second corner to insert the table. If you specified a table style, it will be applied.



- 6. If you need to change the size of an individual row or column, use the **Edit Tool**. See "Changing Rows and Columns, Merging Cells" on page 623.

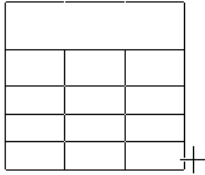


**Fixed Cell Size**

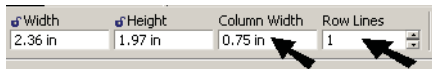
1. Turn off both **Calculated Row Height** and **Calculated Column width**.



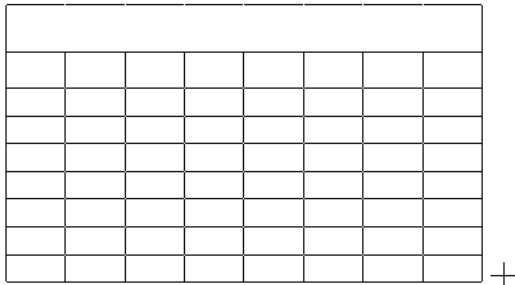
2. Click once to set the top left corner of the table.



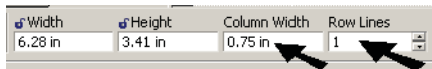
3. Set the column width and number of text lines per row.



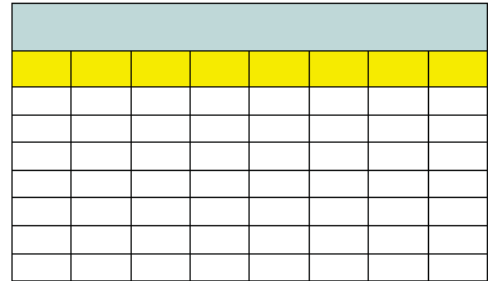
4. Move the mouse to size the table.



The cell size remains the same, no matter how the table is sized. Cells are added or removed as needed.



5. Click the second corner to insert the table. If you specified a table style, it will be applied.



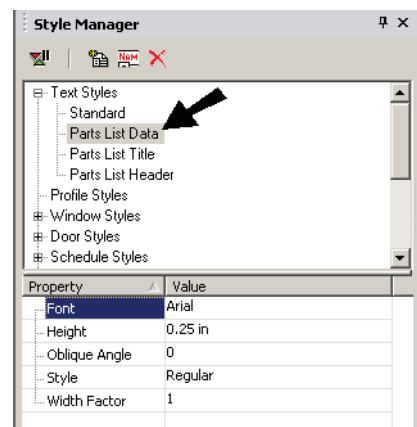
6. If you need to change the size of an individual row or column, use the **Edit Tool**. See "Changing Rows and Columns, Merging Cells" on page 623.

**Modify Table**

**Modify Table** is used to add text to cells, or to edit existing cell text. It can also be used to change properties of individual cells.

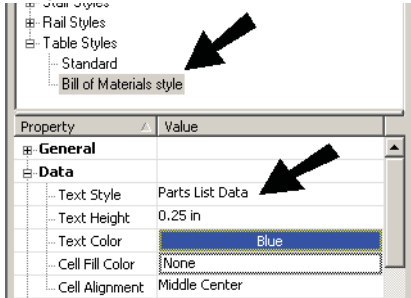
**Adding or Editing Cell Text**

If you are using styles, the table style will refer to a text style for each type of text (data, header, and title). So unless you want to use standard text, define text styles first.

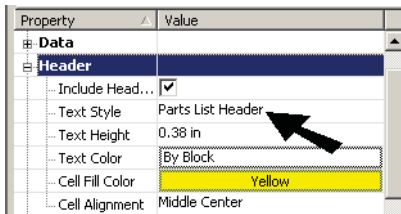


NOTE: For details on text styles, see "Text Styles" on page 566.

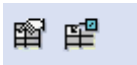
Open the table style you want to use, and specify the text style you want to use for **Data** text. You can also set text color and height here.



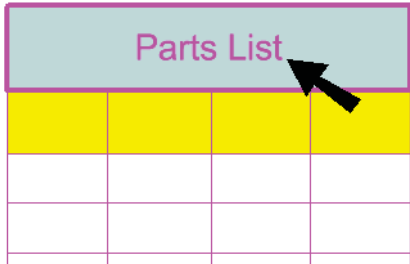
There are also categories for **Header** and **Title**, which can each have their own text style.



- 1. To add text to the table, activate **Modify Table**.
- 2. Make sure **Edit Cell Text** is active in the local menu or Inspector Bar.



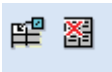
- 3. Select the table you want to modify, then click in a cell where you want to place text.



- 4. Click in each cell and type the text.  
If your drawing has blocks you can also opt to insert a block into the cell.

Parts List			
Number	Material	Cost	Vendor
1	Plastic	\$12	Smith
2	Wood	\$19	Jones
3	Steel	\$3	Brown

- 5. If you want to remove text from a cell, click it and select **Clear Cell Content** from the local menu or Inspector Bar.



- 6. When the table text is complete, end **Modify Text**. The properties of the cell text will match their text styles.

Parts List			
Number	Material	Cost	Vendor
1	Plastic	\$12	Smith
2	Wood	\$19	Jones
3	Steel	\$3	Brown

## Changing Cell Properties

1. If you want to use **Modify Text** to change properties of an individual cell, make sure **Edit Cell Text** is not active.



2. Click the cell you want to change.

Parts List			
Number	Material	Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

3. The properties of the cell can be changed in the Selection Info palette. In this example, the cell's fill color was changed.

Properties	
Constraints	
Property	Value
Current Cell	
Text Style	Standard
Text Height	0.25 in
Text Color	By Block
Cell Fill Color	Aquamarine
Cell Alignment	Middle Center

4. When finished, the cell has the new properties.

Parts List			
Number	Material	Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

## Changing Rows and Columns, Merging Cells

Other than adding text or changing individual cell properties, which are done using the **Modify Text** tool, table changes are made with the **Edit Tool**.

NOTE: For details on the **Edit Tool**, see "Edit Tool" on page 219.

1. To start editing, activate the **Edit Tool** and click the table. You can move any of the nodes to change sizes of single rows or columns.

Parts List			
Number	Material	Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

2. To add a row or column, press Shift and click the cell to the left or above where the new item will go.

Parts List			
Number	Material	Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

NOTE: If you want to Shift-select a new cell, you must first use Shift and click to de-select the current cell.

3. **Insert Row** and **Insert Column** are available on the local menu or Inspector Bar.



The new row is added below the selected cell.

Parts List			
Number	Material	Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

4. To remove a row or column, Shift-select a cell in that row or column.

Parts List			
Number	Material	Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

The row is removed.

Parts List			
Number	Material	Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

5. To merge cells, Shift-select each cell you want to merge.

Parts List			
Number	Material	Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

6. Select **Merge Cells** in the local menu or Inspector Bar.



The selected cells are now one cell.

Parts List			
Number		Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

7. To separate them again, Shift-select the cell and select **Unmerge**.

## Custom Properties, Database, and Reports

*Available in TurboCAD Pro and Platinum only*

All properties of an objects are elements of a database. Standard database property items include line width, color, coordinates of points that comprise the object, etc.

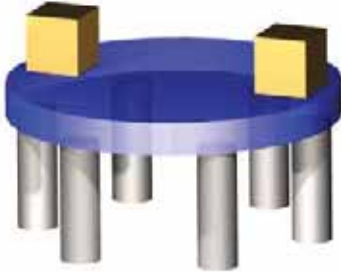
The database management feature enables you to add information to objects and to create reports using this information. For example, you can add custom fields to provide information on suppliers, manufacturers, price, etc. The information can then be presented as a Bill of Materials, Parts List, or other

type of report.

Database management and report creation generally follows these four steps:

- Step 1: Define Custom Fields
- Step 2: Attach Custom Properties to Objects
- Step 3: Define Fields for the Report
- Step 4: Create the Report

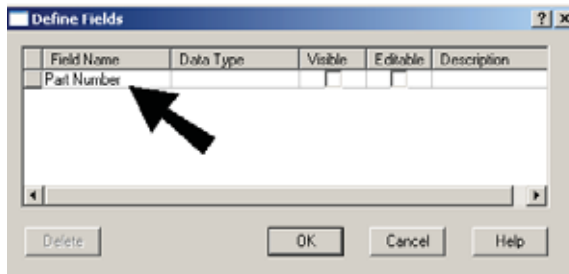
The following example shows how to create some custom properties, then create a report from the custom and standard properties. The objects in the report are shown below: a large plastic disc, two cubes on top, and six steel cylinders below.



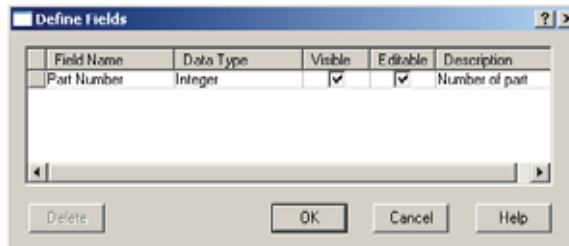
### Step 1: Define Custom Fields

The first step in creating a database is to define the types of information to be recorded.

1. Select **Define Fields**. In the **Define Fields** window, enter the first property: "Part Number."



2. Fill in the remaining fields in this row



#### Data Type:

- **Text:** Alphanumeric information which does not have to be calculated.
- **Integer:** Whole numbers.

- **Double:** Decimal numbers.
- **Currency:** Dollar amounts. This type will not appear in an object's **Custom** page of the **Properties** window.

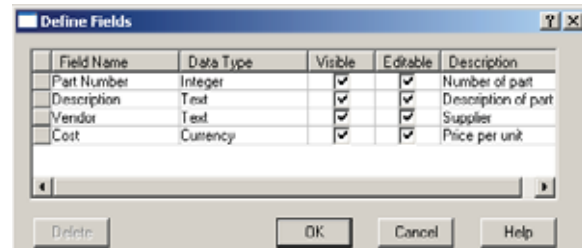
**Visible:** If checked, the field will be visible and available for use in the **Edit Object Data** and **Report** windows.

*TIP: If you know you will not be using certain fields in reports, it is helpful to turn off visibility.*

**Editable:** If checked, the field can be edited in the other database dialogs. Auto fields are always non-editable.

**Description:** An optional field for notes or other comments.

3. To add the next property to the database, click in the first empty field under **Field Name**, and type the new field name. This example uses "Description," "Vendor," and "Cost." Then continue to fill in the other fields.

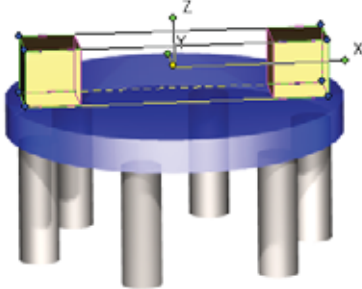


*NOTE: To delete a record, click the square at the beginning of a row, and select **Delete**.*

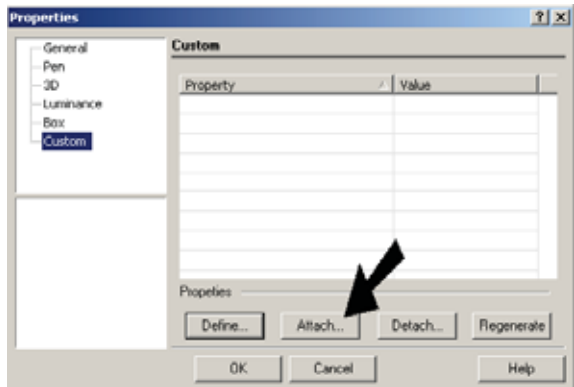
4. Click OK when the custom properties are defined.

**Step 2: Attach Custom Properties to Objects**

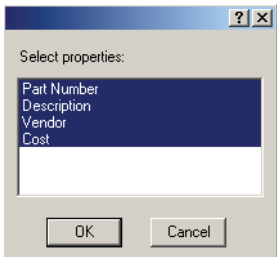
- 1. Select the object whose custom properties you want to define. If multiple objects have the same custom properties, such as the cubes below, select them all.



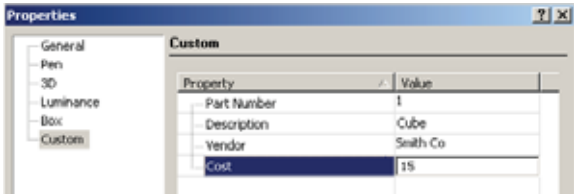
- 2. Open the **Properties** to the **Custom** page. Click **Attach** at the bottom of the window.



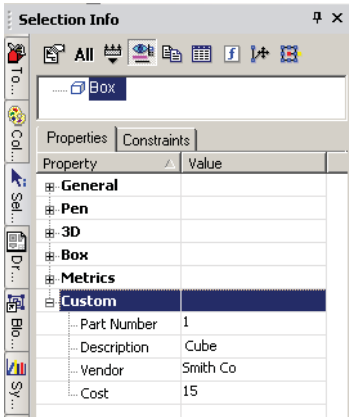
- 3. Select all of the custom properties and click OK.



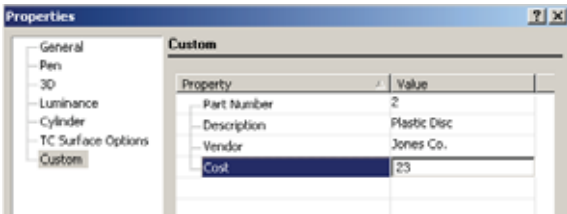
- 4. Now the properties appear on the **Custom** page. Fill in the **Value** for each part.



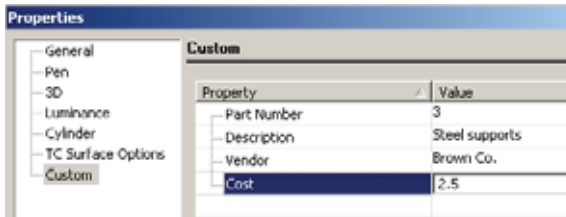
You can also enter or change custom properties in the Selection Info palette.



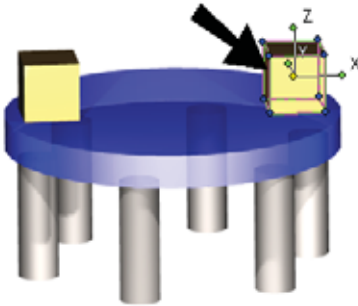
- 5. The previous values were for the two cubes (Part Number 1); the values below are for the large disc (Part Number 2).



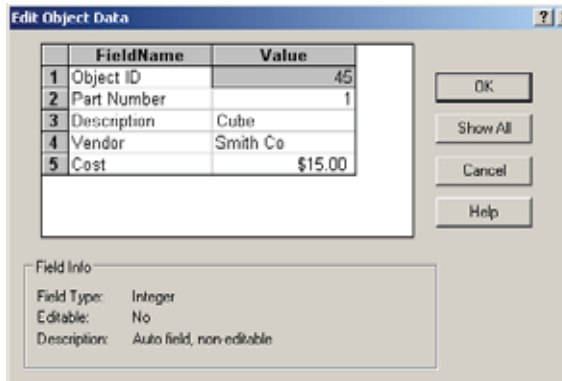
6. These values are for the six support cylinders (Part Number 3).



7. To verify that the custom properties were properly attached, select one of the objects.

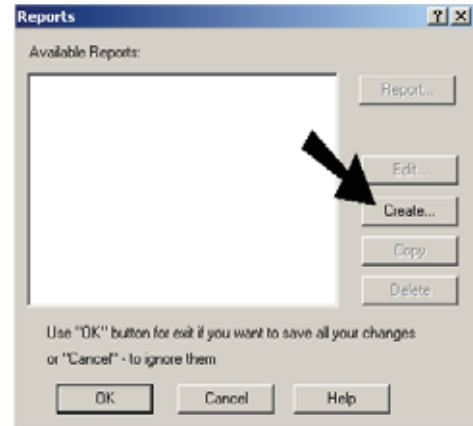


8. Then select **Tools / Database / Edit Object Data**. You can update values in this table, if needed.

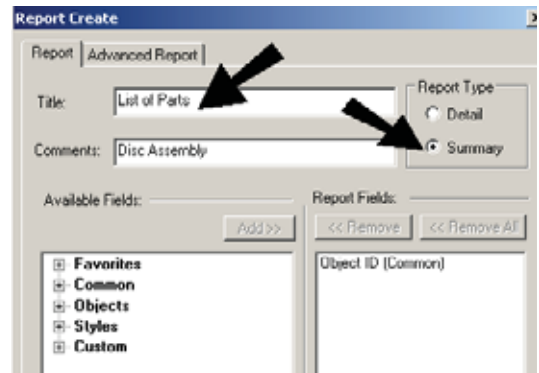


### Step 3: Define Fields for the Report

1. To begin defining the report, select **Tools / Database / Report**. Then click **Create**.

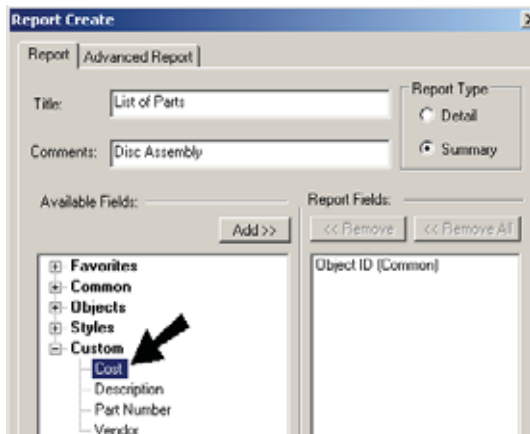


2. Enter a report **Title** and **Comments** (optional). If you choose **Detail**, only values from editable fields may be edited before the report is created. With **Summary**, all values may be edited before printing, even if they come from a non-editable field.

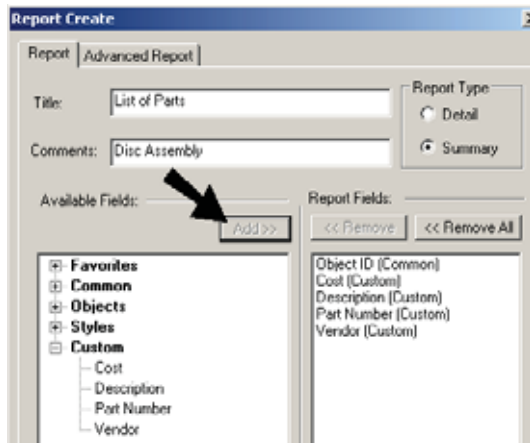


NOTE: "Object ID" is a unique identifier, and is generated automatically when attaching fields to objects.

3. The custom fields appear under the “Custom” category. Click the “plus” sign to expand this category.

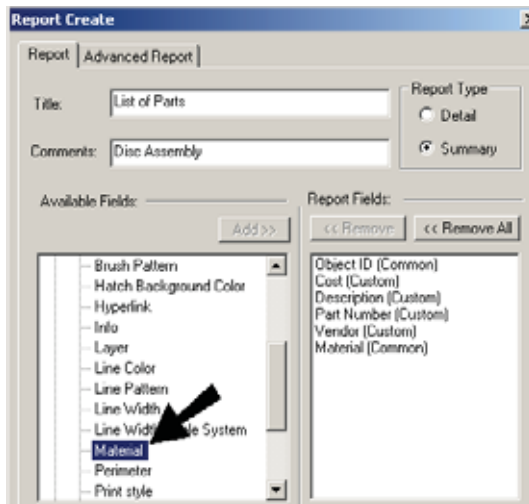


4. For each field you want to appear in the report, click it under **Available Fields**, then click the **Add** button. The field should then appear under **Report Fields**. Repeat these steps for each of the custom fields.



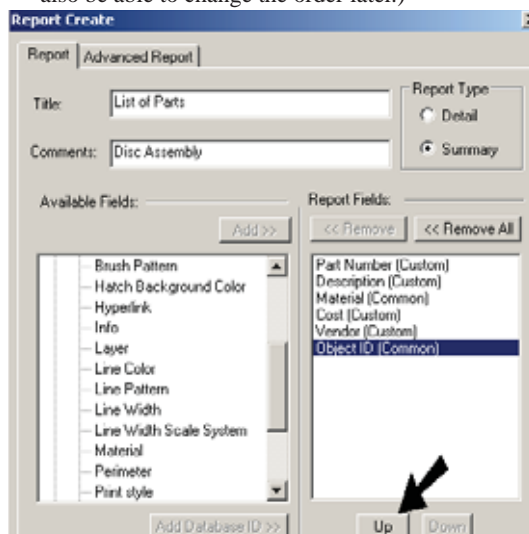
*NOTE: If you have block in your drawing with attributes you can add those attributes to your report.*

5. Non-custom (standard) fields can also be added to a report. In this example **Material** is added, and **Material** is found in the “Common” category.



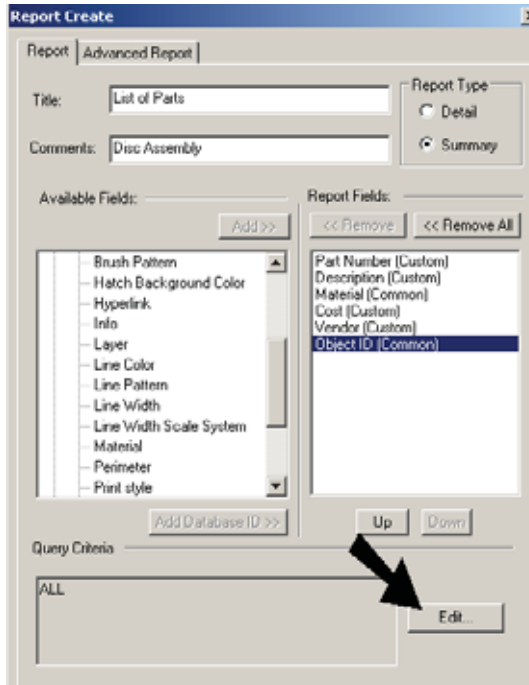
You do not have to create a report using custom fields; reports may contain only standard fields.

6. The order of the Report Fields list is the order that will in the report. To change the order, highlight the field you want to move and click **Up** or **Down**. (You will also be able to change the order later.)

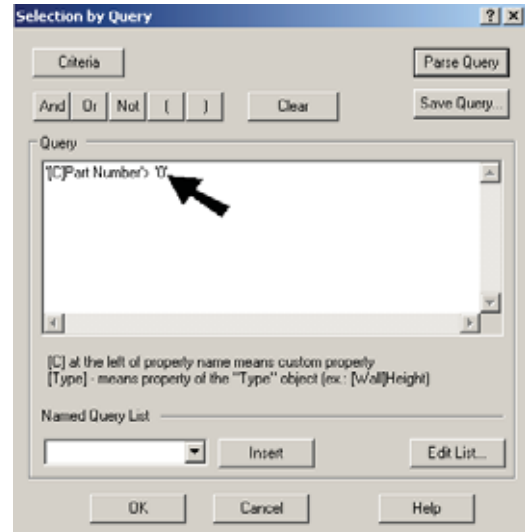




7. You can use a query if you want to filter the objects to be included in the report. If you want to run a query, click **Edit**. Otherwise, all objects in the model will be scanned while creating the report.



An example of a query in this example would be to search for objects that have “Part Number” fields greater than one. This is how the syntax would look:



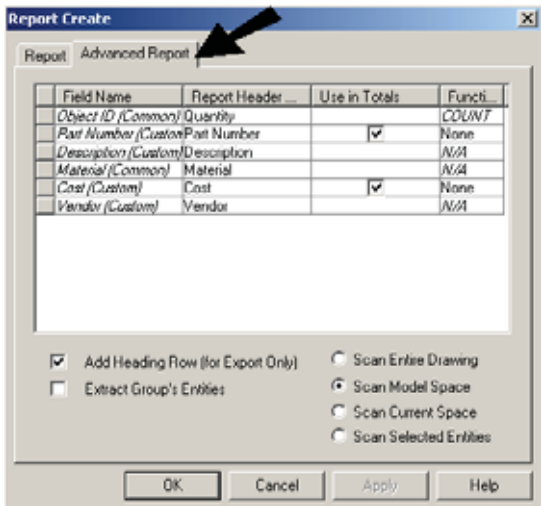
8. If you created a query, click OK to return to the **Report Create** window.

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NOTE: For details on query searches, see “Select by Query” on page 187.

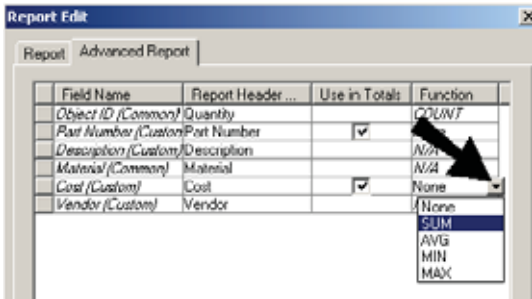
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9. Open the **Advanced Report** table.



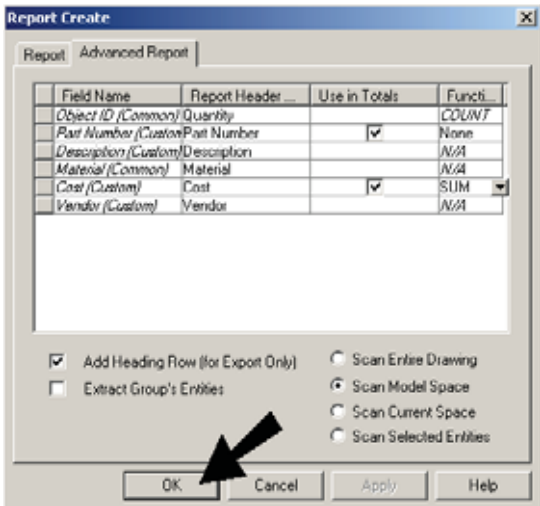
The appearance of this tab depends on whether you are creating a **Detail** or **Summary** report. For a **Detail** report, **Field Names** can be edited. For a **Summary** report, **Report Header Names** may be edited.

10. For the “Copy” row, click the cell in the **Function** column. From the drop-down menu, select **SUM**. This will show the total cost of all objects, not the unit price per object.



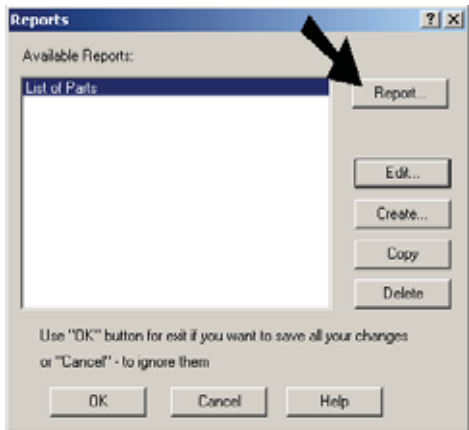
The other function options are **AVG** which calculates the average of all values, and **MIN / MAX** which displays the smallest or largest of the values.

11. When all fields and queries are defined, click **OK** to return to the **Reports** window.

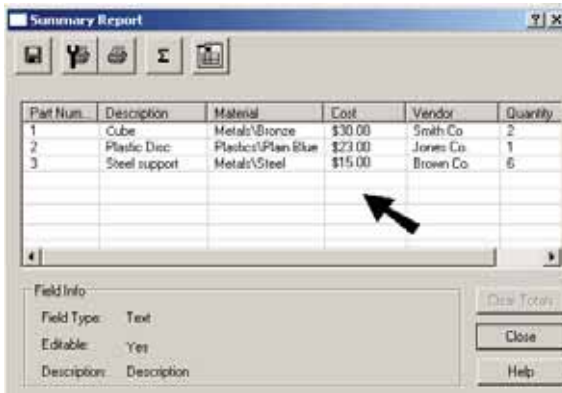


### Step 4: Create the Report

1. The **Reports** window now shows the report you have just created. (You can return to the **Reports** window anytime by selecting **Tools / Database / Report.**) Click **Report**.



2. The **Summary Report** window shows the fields and values that will go in the report. If you want to re-order a column, drag its header to the new location. You can also click on a header to change that column's sorting order. To add a row for cost sums, click in a cell below the last "Cost" items.



3. Then click **Grand Total**.



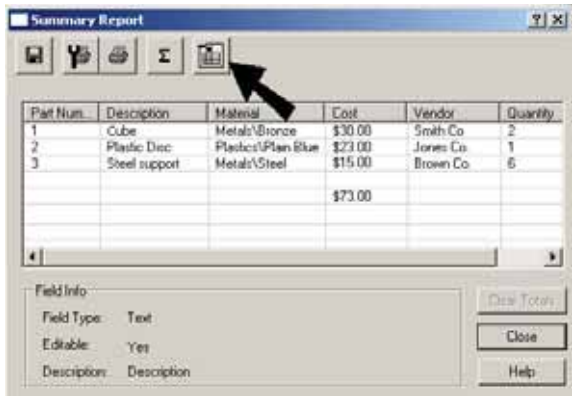
The sum of the cost items is now listed.



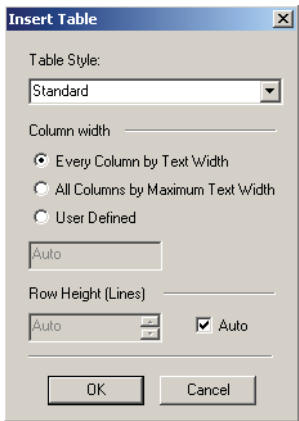
The other available options at the top of the **Summary Report** window are:

- **Export:** Saves the report as a Microsoft Excel Spreadsheet (XLS), Microsoft Access Database (MDB), a Comma Separated Value (CSV), or a Tabbed Text File (TXT).
- **Page Setup:** Sets the format of a printed version of the report. You can add the page number, number of total pages, date, and/or time to the header or footer of each report page. You can also set the margins, alignment, page order, and scale of the report pages, and whether grid lines, row headings, and column headings will be printed.
- **Print:** Prints the report using the **Print Setup** options.

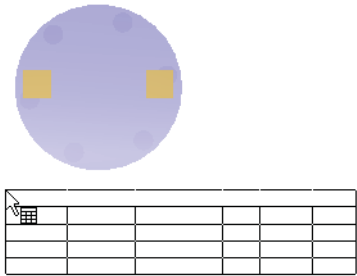
4. To place the report in the file, click **Insert Table into the Drawing**.



5. In the **Insert Table** window, set the sizing options for rows and columns. You can also set the table style here.



6. Click to place the report as a table in the drawing.

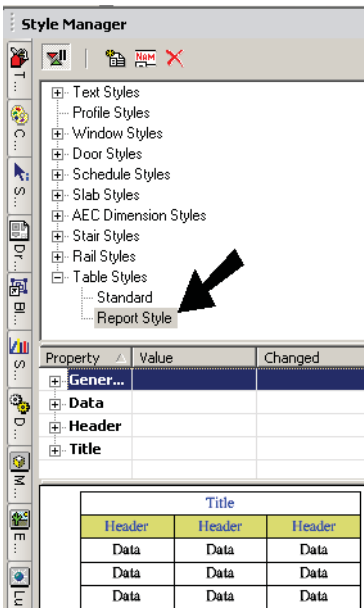


The table contains the custom fields, as well as the **Materials** field.

List of Parts					
Part Number	Description	Material	Cost	Vendor	Quantity
1	Blocks	Metals\Bronze	\$30.00	Smith Co	2
2	Plastic Disc	Plastics\Plain Blue	\$28.00	Jones Co.	1
3	Steel supports	Metals\Steel	\$15.00	Brown Co.	6
Totals:					
			\$73.00		

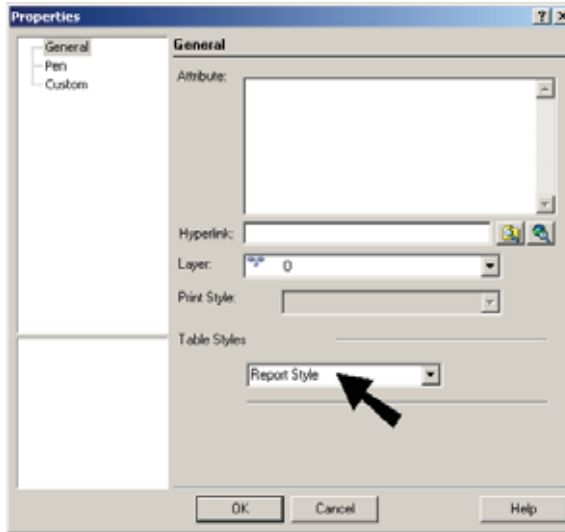
NOTE: If you want to edit or add text to a table, see "Modify Table" on page 621. For other changes such as adding or removing rows / columns, or merging cells, see "Changing Rows and Columns, Merging Cells" on page 623.

7. If you want to change the style of the report table, make sure the style is defined under **Table Styles** in the Style Manager.



NOTE: For details on table styles, see "Table Styles" on page 607.

8. The style is assigned to the table in the **General** page of its **Properties**.



If you change the style, the table will update automatically.

List of Parts					
Part Number	Description	Material	Cost	Vendor	Quantity
1	Blocks	Metals\Bronze	\$30.00	Smith Co	2
2	Plastic Disc	Plastics\Plain Blue	\$28.00	Jones Co.	1
3	Steel supports	Metals\Steel	\$15.00	Brown Co.	6
Totals:					
14			\$73.00		

### Updating the Database and Report

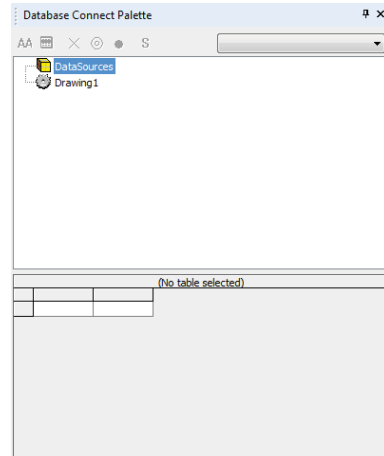
You can update fields values for objects by using **Edit Object Data**, or you can change values in an object's **Properties** or the Selection Info palette.

To update the database, select **Regenerate**.

You will then have to create a new report, using **Report**, and insert the new report into the drawing.

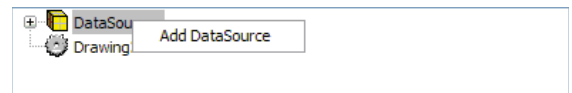
## Database Connection Palette

TurboCAD has the ability to connect to a database and associate the data with objects. Database connectivity and management is handled through the Database Connection Palette.

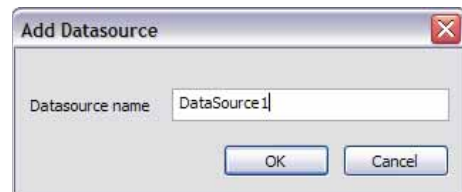


### To Create a Database Connection:

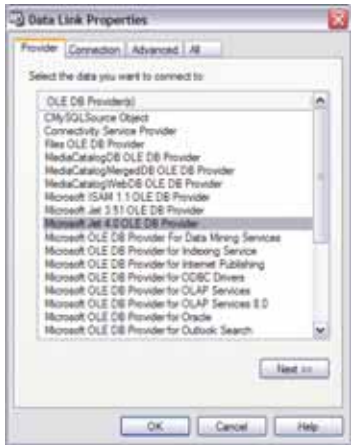
1. Open the **Database Connection Palette**.
2. Right click on the **DataSources** icon and click **Add DataSource**.



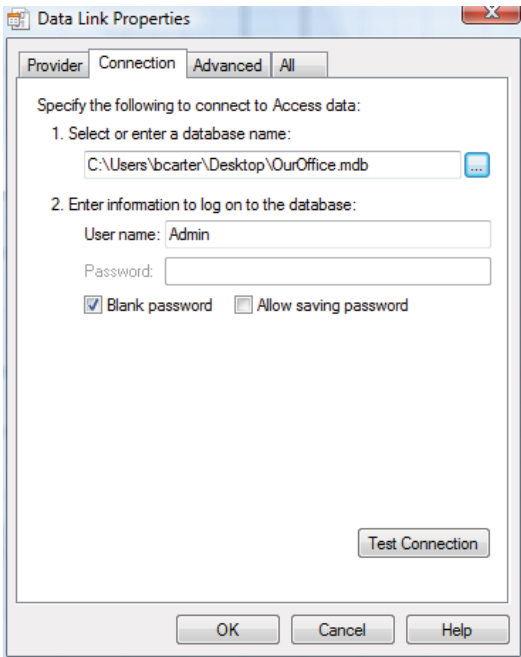
3. In the **Add Datasource** dialog type in the name of the new datasource and click **OK**.



4. The **Data Link Properties** dialog will open.

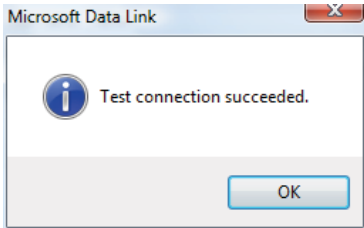


5. On the **Provider** page select the appropriate **OLE DB Provider** type for the datasource to which you wish to connect. Then click **Next**.

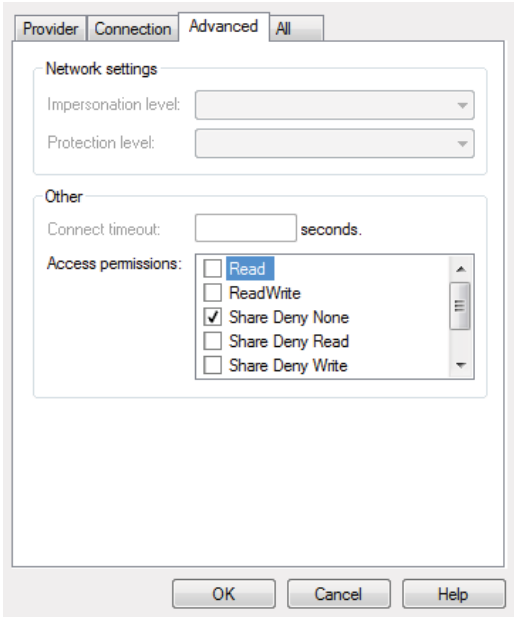


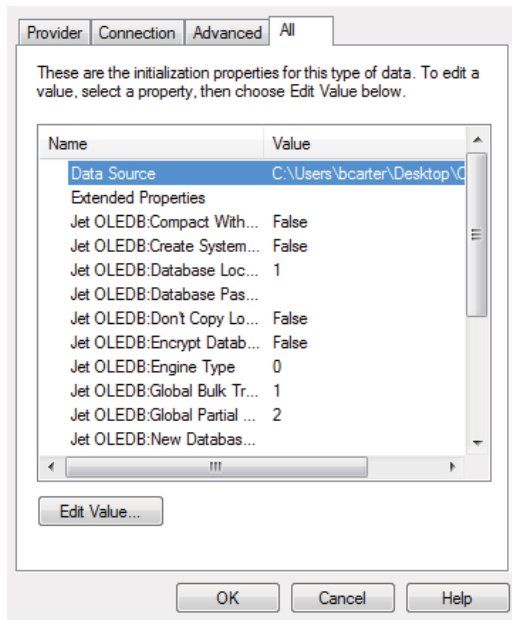
6. The **Connection** page will appear.
7. Specify the database path and name.

8. Setup the log on information.
9. Click the **Test Connection** button. If the connection is valid the **Test Connection Succeeded** message will appear.

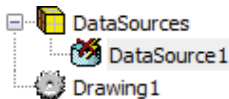


10. At this point you can move on to the **Advanced** and **All** pages, or click **OK**.
11. The features and functions of the Advanced and All pages are network and database specific, and are dependent upon your network and ODBC/OLE DB protocols. Only a person and experienced with your network and database protocols should alter these settings.

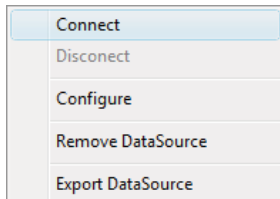




12. After these operations you will see a new datasource object in **DataSources** tree.



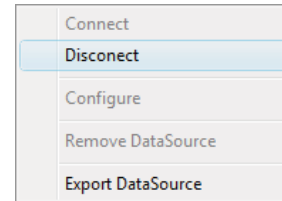
13. The datasource is not yet connected. Right-click on the datasource and select **Connect**.



14. Your database is now connected to TurboCAD as a datasource.
15. When connected a datasource shows all available tables and views in database.

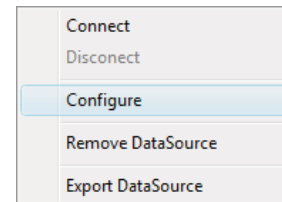
### To Disconnect a Datasource:

1. Select a connected datasource in the **DataSources** tree.
2. Right-click on the datasource and select **Disconnect**.



### To Re-configure a Datasource:

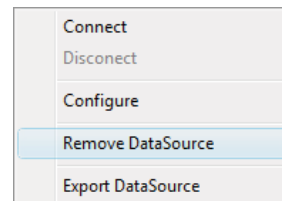
1. Select a datasource in the **DataSources** tree.
2. Right-click on the datasource and select **Configure**. This will open the **Data Link Properties** dialog.



3. Make the required changes and click **OK**.

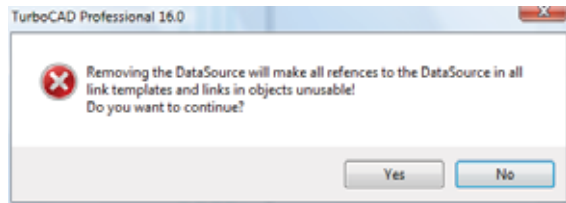
### To Remove a Datasource:

1. Select a datasource in the **DataSources** tree.
2. Right-click on the datasource and select **Remove DataSource**.



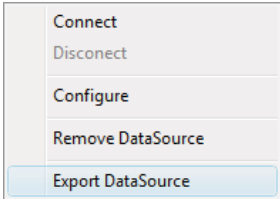
3. You will be prompted with a warning dialog.

4. If you are clear to proceed click **Yes**.

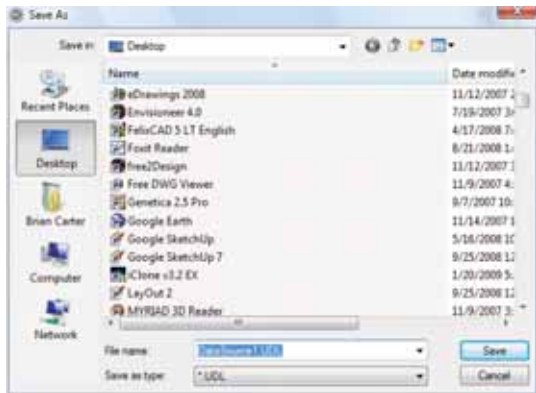


**To Export a Datasource:**

1. Select a datasource in the **DataSources** tree.
2. Right-click on the datasource and select Export DataSource.



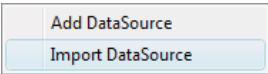
3. This will open Save As dialog.



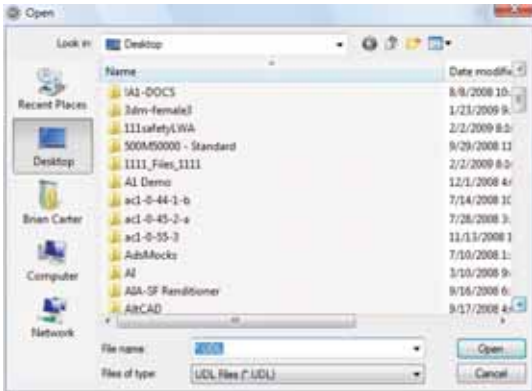
4. Select a location and name for the datasource, and then click **Save**.

**To Import a Datasource:**

1. Right click on the **DataSources** icon and click **Import DataSource**.



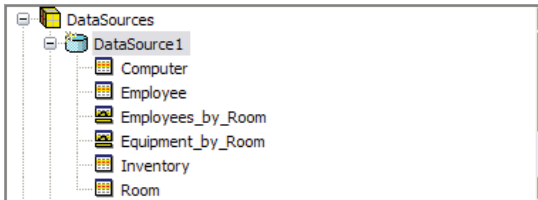
2. This will start the **Open** dialog.



*Note: Connect is only available when a datasource is disconnected. Disconnect is only available when a datasource is connected.*

**Viewing and Editing Data in Tables**

When connected a datasource shows all available tables and views in database. Available tables and views appear as child nodes of the datasource.

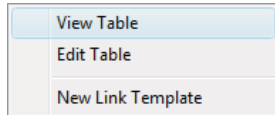


ICON LEGEND: = view = table

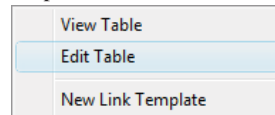


**To Show a Table's Content:**

1. Select the table in the **DataSources** tree.
2. Right-click and select **View Table**, the table will be opened in the lower frame of the **Data base Connection** palette in Read-Only mode.

**To Edit a Table's Content:**

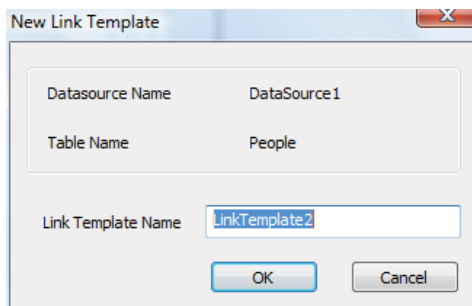
1. Select the table in the **DataSources** tree.
2. Right-click and select **Edit Table**, the table will be opened in the lower frame of the **Data Base Connection** palette in Edit mode.

**Creating Data Link Templates**

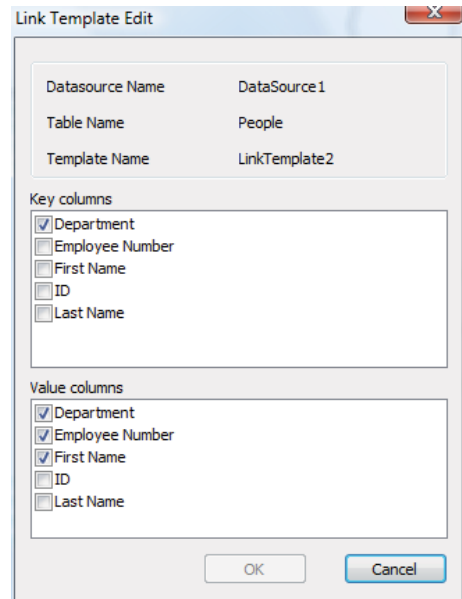
In order to connect data from tables to graphic objects it is necessary to create Link Templates. Link templates set a pattern that defines which the data elements will be linked from the table and associated with designated graphic objects.

**To Create a New Link Template:**

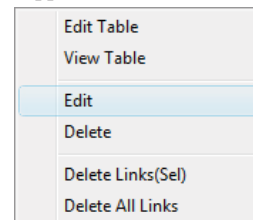
1. Select a table in the **DataSources** tree.
2. Right-click and select **New Link Template**. The **New Link Template** dialog will appear.
3. Type in a name for the link template, and then click **OK**.



4. The **Link Template Edit** dialog will appear.
5. Select the **Key Columns**. These are used to define link information data for linked graphics.
6. Select the **Value Columns**. These values will be set as custom properties of the object.
7. The created link template will be attached to current active drawing.

**To Edit a Link Template:**

1. Select the template in the **DataSources** tree.
2. Right-click and select **Edit**. The **Link Template Edit** dialog will appear.

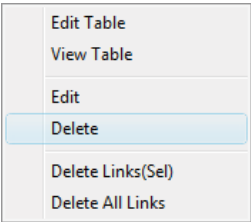


- 1. Make the required changes, and then click **OK**.

**To Remove a Link Template:**

- 1. Select the template in the **DataSources** tree.
- 2. Right-click and select **Delete**.

*Warning: The link template will be immediately deleted. Removing a link template leads to removing all graphic links bound to it.*

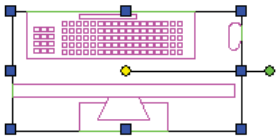


**Linking Data to Graphics**

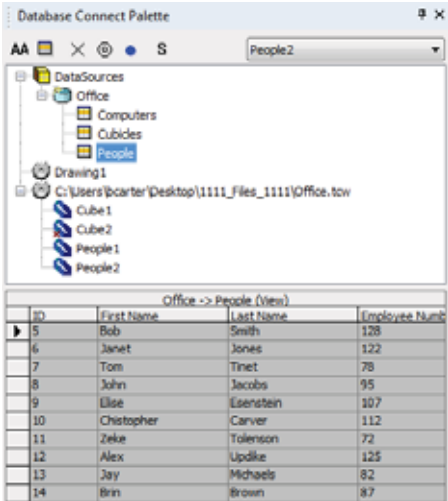
Data is linked to graphics by connecting a table to specific graphics via link templates.

**To Connect Data to Graphics/s:**

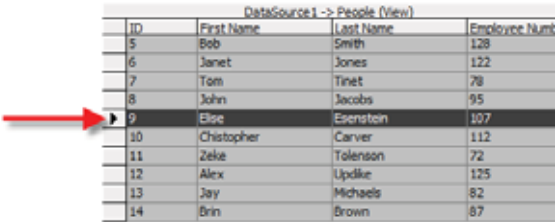
- 1. In a drawing, select one or more graphics.



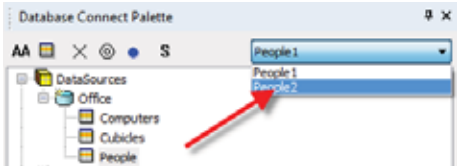
- 2. In the **DataSources** tree, double click on the desired table to select it for linking. This will also open the view of the table in the lower frame of the palette.



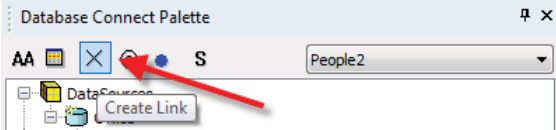
- 3. In the table view, in the lower frame of the palette, click on the record you wish link.



- 4. Use the **Link Template** drop-down box located on the palette toolbar to select the desired link template.



- 5. Click the **Create Link** button on palette toolbar.



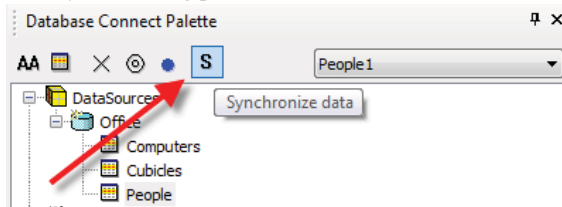
- 6. The link between the selected graphics and the table record is created.

## Synchronizing Data and Graphics

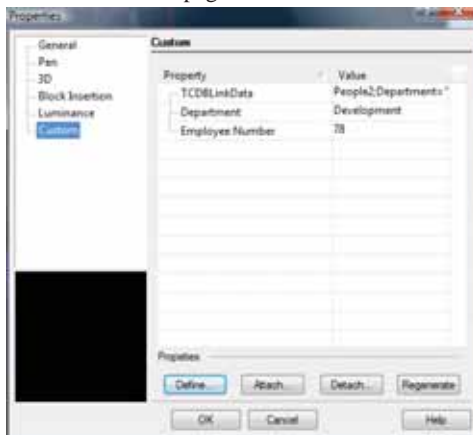
To get data into linked graphics it is necessary to synchronize the data. The synchronizing process copies data from the datasource into all linked graphics and places it in the custom properties of linked graphics.

### To Synchronize Data:

1. Click the **Synchronize Data** button on toolbar to run synchronizing process.

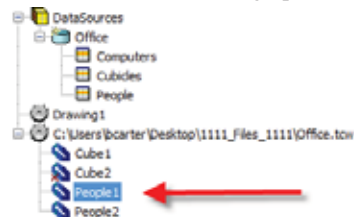


2. Verify that the data has been synchronized by opening the properties of a graphic that is linked and going to the **Custom** page.

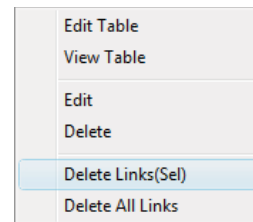


### To Remove Links from Selected Graphics:

1. Select the graphic/s you wish to de-link.
2. In the **DataSources** tree select the link template you wish to de-link from the graphic/s.



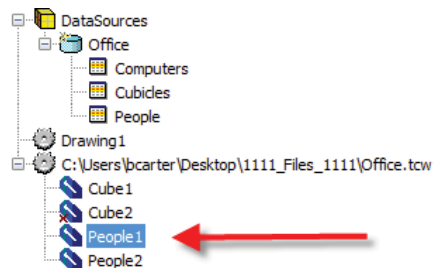
3. Right-click and select **Delete Links(Sel)**.



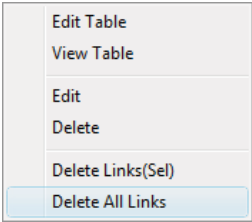
4. Synchronize the data.
5. The links and data have been removed.

### To Remove All Links to a Link Template:

1. In the **DataSources** tree select the link template you wish to de-link.



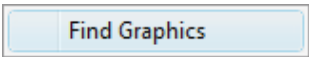
- 2. Right-click and select **Delete All Links**.



- 3. Synchronize the data.
- 4. The links and data have been removed.

**To Find Graphics Linked to a Table Record:**

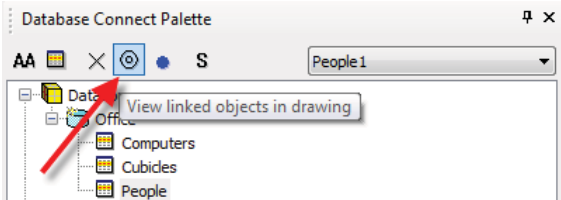
- 1. In the table view, in the lower frame of the palette, click on the record.
- 2. Right-click and select **Find Graphics**.



- 3. All graphics that are linked with current row will be selected.

**To Find Table Record Linked to a Selected Graphics:**

- 1. Select a graphic.
- 2. Click the **View linked** records in dataview button on palette's toolbar.



- 3. All records which are linked to the selected object will be highlighted in the table view.

Office -> People (View)				
ID	First Name	Last Name	Employee Number	Department
5	Bob	Smith	128	Accounting
6	Janet	Jones	122	Design
7	Tom	Tinet	78	Development
8	John	Jacobs	95	Design
9	Elise	Esenstein	107	Accounting
10	Chistopher	Carver	112	Admin
11	Zeke	Tolenson	72	IT
12	Alex	Updike	125	Development
13	Jay	Michaels	82	Design
14	Brin	Brown	87	Development

# 16 Paper Space and Printing

Once your drawing is complete, you may need to view it on paper. This section covers features that are geared toward printing, including Paper Space and viewports. You can also publish your drawing in HTML format for online distribution.

## Paper Space

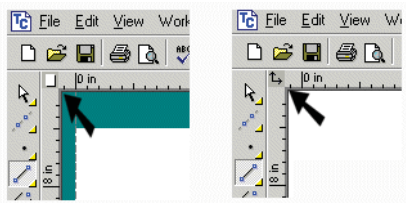
While Model Space is typically used while creating geometry, Paper Space is geared toward creating the final layout of your drawing for printing or plotting on paper. In Paper Space, you typically arrange drawing elements on one or more sheets of paper.

The Paper, the large white rectangle in the middle of the drawing window, shows you how your drawing will be laid out on the page when printed. You do not have to draw on the paper; you can draw anywhere in the drawing space and later fit the drawing to a sheet of paper. You can also turn off the paper sheet shown. The pattern of horizontal and vertical lines, which may be shown on the paper, is the grid that marks exact locations in the drawing.

*TIP: You can use the TC Explorer Palette to see all Paper Spaces defined for any open drawing. See "Print Spaces" on page 94.*

## Switching to Paper Space

You can use the small button at the top left corner, at the intersection of the rulers, to toggle between Model and Paper Spaces.

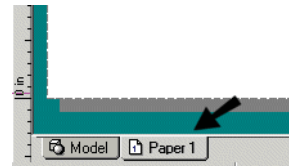


*NOTE: If the rulers are turned off, this button will not appear. Use **Rulers** to toggle the ruler display.*

You can also use the icons on the **Standard** toolbar to switch between spaces.



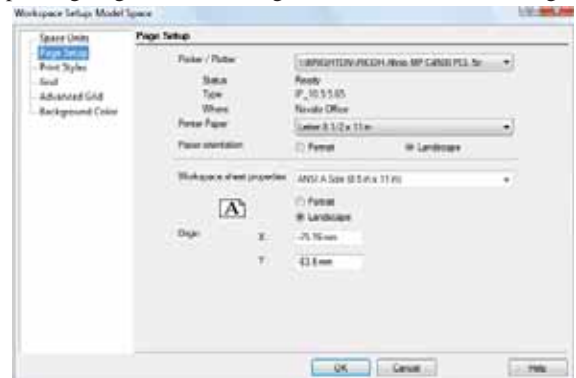
There are also workspace tabs at the lower left corner of the screen. By default, each file contains one tab for Paper Space. Click the tabs to switch spaces.



*NOTE: If scroll bars are turned off, these tabs will not appear. Use **Scroll Bars** to toggle the scroll bar display.*

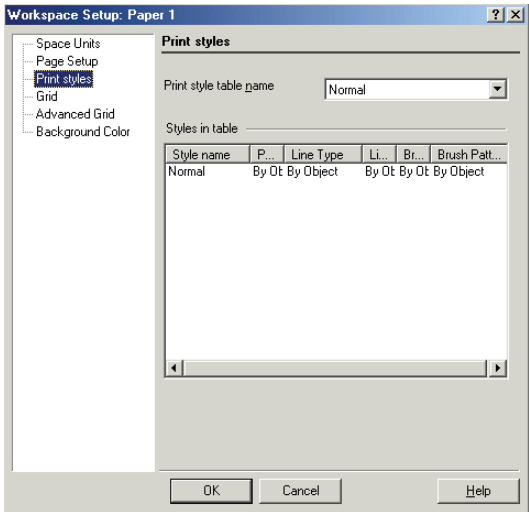
## Paper Space Properties

The **Page Setup** page contains options for the printer, printer paper, worksheet selection, paper size, paper orientation, and printing origin. These settings are saved with the drawing.



The various Paper Spaces can be used for different print setups. For example, you can have one setup for shop drawings on D size paper, and another with renderings on A size paper.

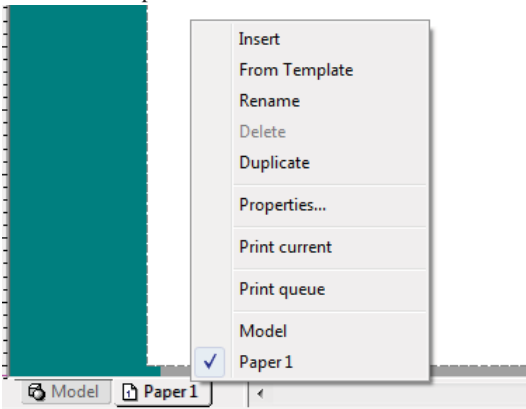
The **Print Styles** page enables you to select a print style.



See "Print Style Options" on page 667.

### Manipulating Paper Spaces

By default, a new file contains one Paper Space, named Paper 1. You can create, delete, or rename spaces by using the **Paper Space** menu, or by right-clicking on any Space tab to access the local menu. These menus can also be used to switch between spaces.



**Insert:** Creates a new, empty Paper Space.

**From Template:** Allows you to import paper space and layouts from existing TCT and TCW files.

**Rename:** Enables you to assign a name to the active Paper Space.

**Delete:** Deletes the Paper Space that was last inserted. You will not be able to delete the last remaining Paper Space of your drawing.

**Duplicate:** Creates a copy of the Paper Space.

**Properties:** Opens the **Properties** window (see "Paper Space Properties" on page 641). Different Paper Spaces can have different properties.

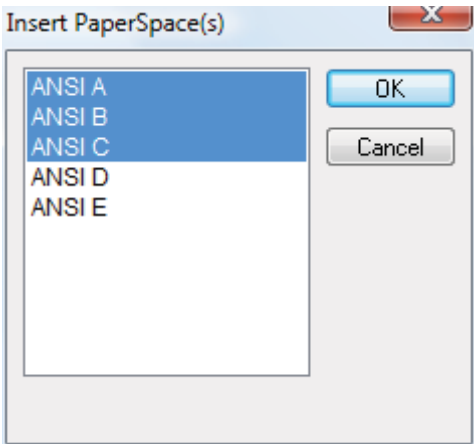
**Print Current:** Opens the Print Dialog for the current space.

**Print Queue:** Opens the Print Queue.

### From Template

To import Paper space layout from existing files:

1. Right click on a space tab then select From Template.
2. The Open dialog will appear.
3. Select the desired file and click Open.
4. The Insert PaperSpace(s) dialog will appear.



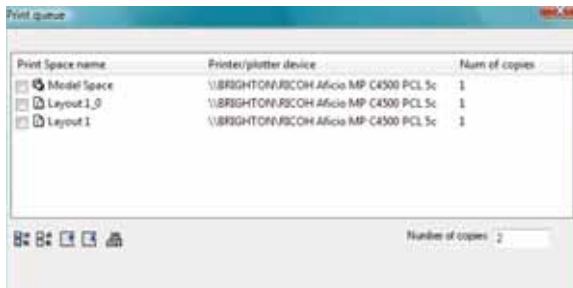
5. Select the Paper Spaces you want.
6. Click OK.
7. The Paper Spaces will be inserted into your drawing.

It is important to import only paper spaces that are using the same unit system (English/Metric). If differing unit systems are used the result will in incorrect scaling of layout elements.

## Print Queue

To import use the Print Queue:

1. Right click on a space tab then select Print Queue.  
Or
2. Select Print Queue from the File menu.
3. The Print Queue dialog will open. If you opened the queue from the space tab, that space will be selected for printing.



In the Print Queue you can select the spaces you want to print. If you double-click on any of the listed spaces you will open the print dialog for that space will open. You can configure the print setup for that space including the printer to be used. The order of the items in the queue sets the order in which the spaces are sent to the printers.

### Options:

**Select All:** Selects all of the spaces for printing.

**Deselect All:** Deselects all of the spaces from printing.

**Move Up:** Moves the currently selected space up in the queue.

**Move Down:** Moves the currently selected space down in the queue.

**Properties:** Opens the Print Properties for the currently selected space.

**Copies:** Sets the number of copies to be printed.

## Drafting Palette - Creating Standard Views

Available in TurboCAD Pro and Platinum only

The Drafting Palette enables you to insert standard views of your model into Paper Space.

*NOTE: If you want to insert predefined views (non-standard views or views you created in advance), see "Viewports" on page 657*

## Drafting Palette Toolbar

**Create Part / Assembly:** Enables you to define a part or a combination of parts (assembly) for creating views. See "Parts and Assemblies" on page 645.



**Delete:** Deletes the selected part, assembly, or view.



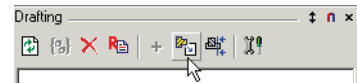
**Rename:** Renames the selected part, assembly, or view.



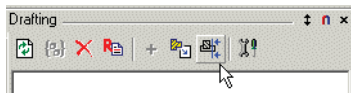
**Add:** Adds the selected standard view that appears in the bottom of the palette to the view list at the top of the palette.



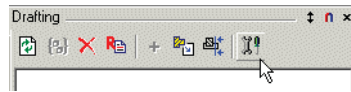
**Insert into Drawing:** Inserts the view selected from the view list into the drawing. (You can also drag the name from the list into the drawing to insert a view.)



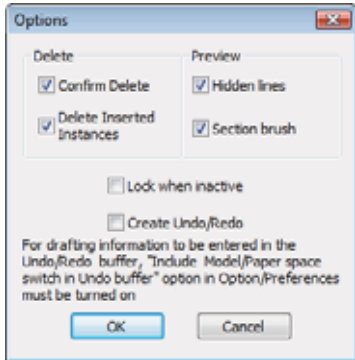
**Create by View Line:** Creates a sectional view from an existing view, by using an existing line or polyline as the section line.



**Drafting Palette Options:** Contains options for deleting objects and what appears in view previews.



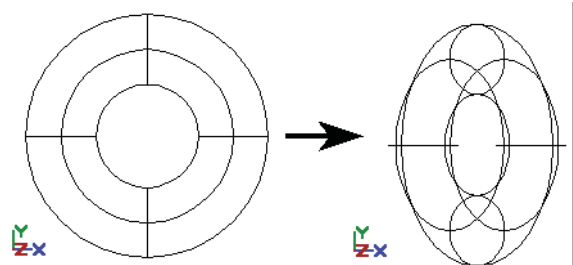
**Lock When Inactive:** When selected Drafting palette objects will not refresh while the Drafting palette is closed.



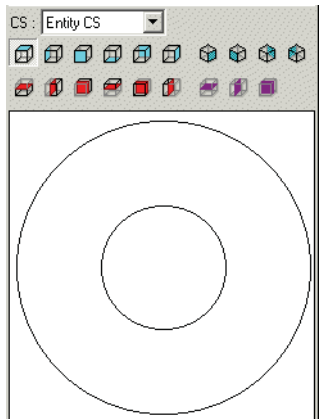
The lower half of the palette contains standard views. The top row contains orthographic and isometric views and the bottom row contains sectional views.



**Entity CS, World CS:** Sets the view based on one of these coordinate systems. As an example, a torus is created and rotated about the Y axis.

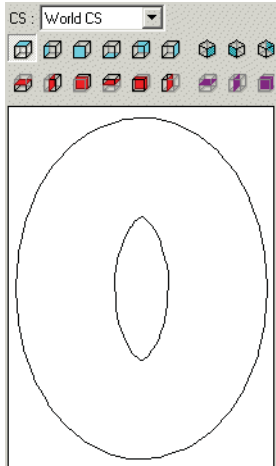


With **Entity CS**, the **Plan** view looks down on the torus in its own CS, even though the torus is rotated.





With **World CS**, the **Plan** view looks down on the rotated torus.

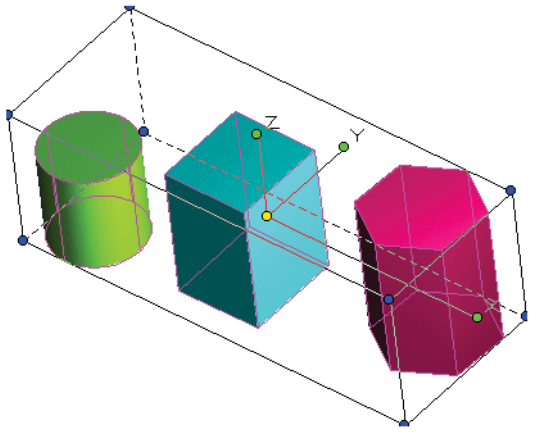


### Parts and Assemblies

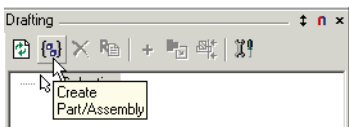
You can create views from individual parts in the model, from selected objects, or from the entire model.

This example uses a cylinder, box, and hexagonal prism.

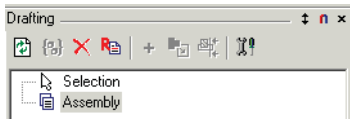
1. For the first assembly, select all three objects.



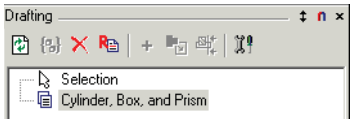
2. Click **Create Part / Assembly**.



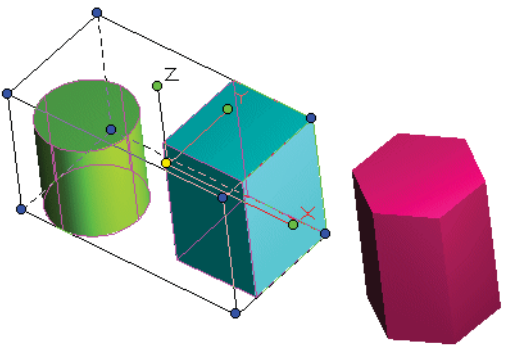
The header **Assembly** appears in the Drafting Palette.



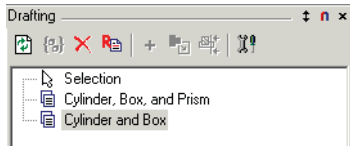
3. This is the default name, but you can change the name using the **Rename** icon, or simply by clicking in the name field and entering the new name.



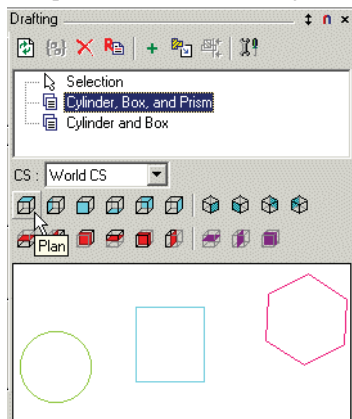
4. For the next assembly, select the cylinder and box.



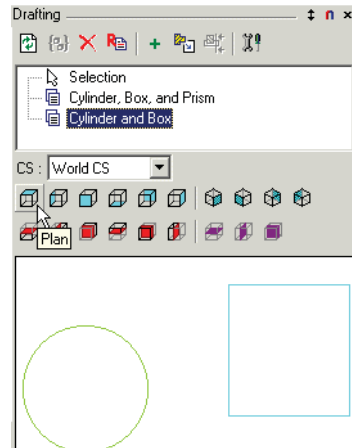
5. Make an assembly from these two objects, and assign a name.



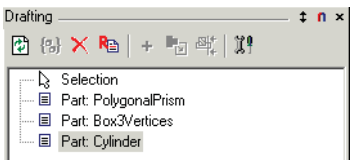
6. To see the difference, first highlight the first assembly, then click the **Plan** view icon. The preview window shows the plan view for all three objects.



7. Then highlight the second assembly and look at the preview of the **Plan** view. Only the cylinder and box appear.



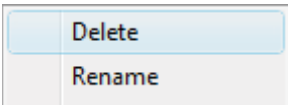
You can also create views of single parts. If you click a single object (cylinder, box, or prism) and click **Create Part / Assembly**, the part name and description appear in the Drafting Palette.



### Deleting Assemblies and Views

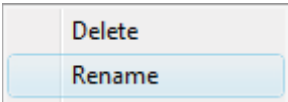
You can delete any assembly or view at any time.

1. Right click on the entity in the Drafting Palette and select delete.
2. When you are prompted if you want to delete the object, select Yes.



### Renaming Assemblies and Views

1. Right click on the entity in the Drafting Palette and select Rename.
2. When the name field highlights, type in the new name.
3. Press Enter to set the new name.

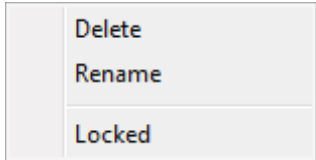


### Locking Assemblies and Views for Caching

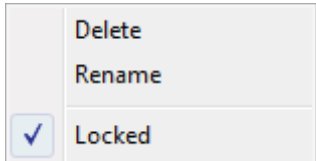
You can lock assemblies and views so that they do not refresh every time you modify the model or the layout. This is especially useful for large or complex models as it reduces refresh time.

### To Lock a View

1. Right click on a view in the Drafting Palette.
2. Select Locked.



When a view is locked a check mark will appear next to the Locked menu Item.



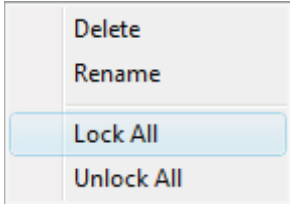
### To Unlock a view

1. Right click on a view in the Drafting Palette.
2. Select Locked, and the check mark will disappear.

### To Lock an Assembly

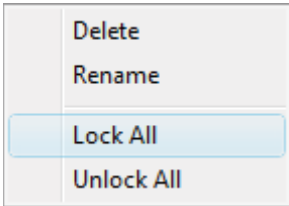
You can lock all of the views in an assembly.

1. Right click on an assembly in the Drafting Palette.
2. Select Lock All

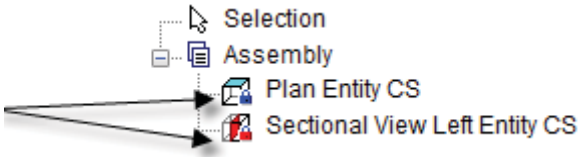


### To Unlock an Assembly

1. Right click on an assembly in the Drafting Palette.
2. Select Unlock All

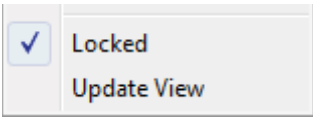


When a view is lock a small lock symbol will appear on the icon of the view. If the view is up to date the lock will be blue. If the model or layout has changed the lock symbol will appear in red.



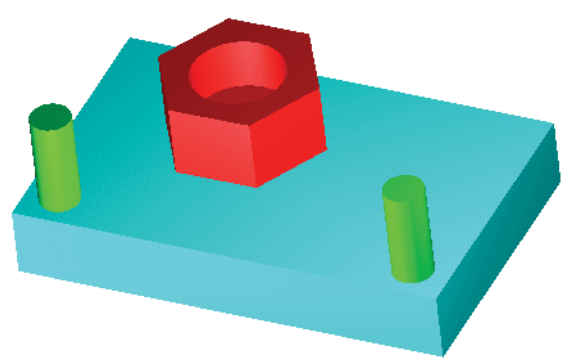
You can update a view at any time to bring it current.

1. Right click on a view in the Drafting Palette.
- Select Update view.



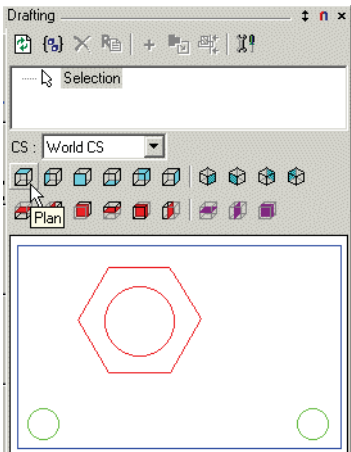
### Inserting Standard Views

This example will use the following solid model, consisting of a box, 2 cylinders, and a polygonal (hexagonal) prism with a subtracted cylinder.



*Note: You can use the Smart Dimension tool*

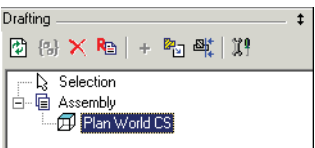
1. First, select the objects you want to include in the view. In this case, select all the objects.
2. In the Drafting Palette, select the standard view from the lower group of icons. Start with a **Plan** view - the preview appears in the palette window.



3. Add this view to the view list by clicking the **Add** icon.

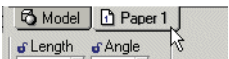


The view name appears on the list. You can change it by clicking the **Rename** icon.

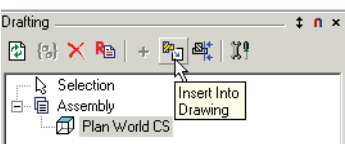


*NOTE: You could have created an assembly of the entire model (see "Parts and Assemblies" on page 645), but selecting objects, then creating and adding a standard view creates an assembly automatically. Note that the **Plan** view appears under the **Assembly** header.*

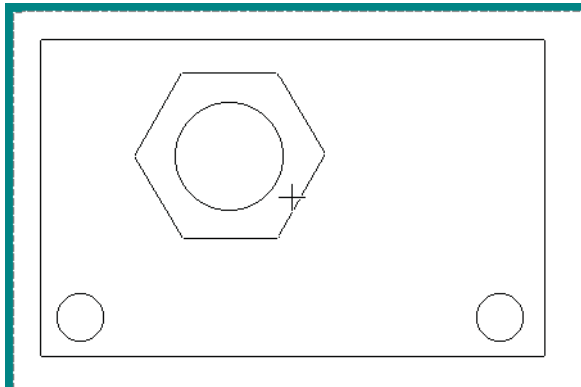
4. Switch to Paper Space. (You can insert views into Model Space as well, but Paper Space is more appropriate.)



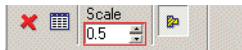
5. Highlight the view name, and click **Insert into Drawing**.



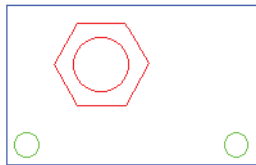
6. Place the view into the Paper Space. In this case, the scale is too large.



7. Press Tab to access the **Scale** field, and enter the new scale value.

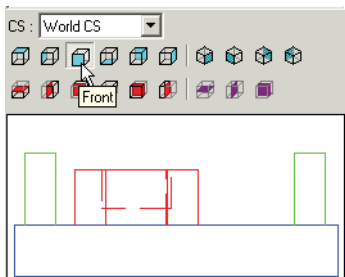


The view is now half as large.

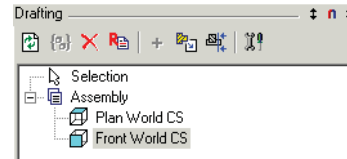


NOTE: You can also change a view's scale in the **Format** page of its **Properties**.

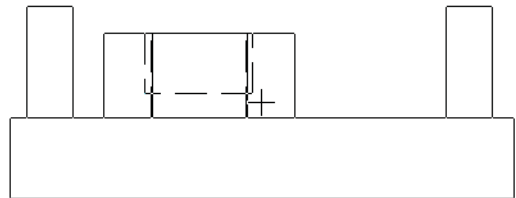
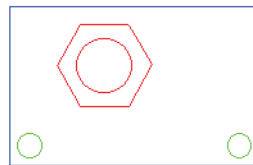
8. For the next view, create a **Front** view.



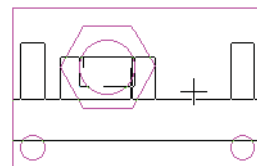
9. **Add** it to the list.



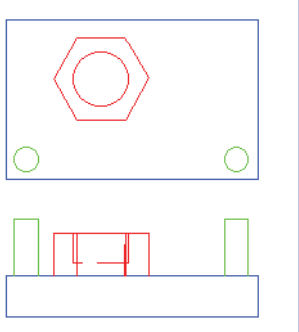
10. Use **Insert into Drawing**, or simply drag it from the palette (drag either the view's name or drag the preview) into the drawing. Again, the scale is too large.



11. To set the scale to that of the **Plan** view, simply drag the **Front** view over the **Plan** view. It assumes the same scale.

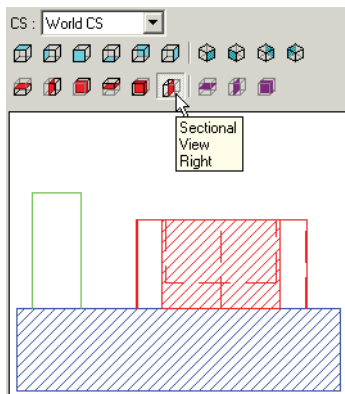


12. This also aligns the **Front** view to the **Plan** view. You can move the Front view up and down, but not left or right. (To break this alignment, but preserve the modified scale, press Shift).

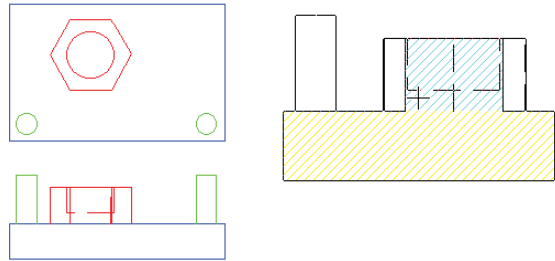


NOTE: You can always move an aligned view (or any view), by selecting it and dragging its reference point. See "Moving Objects in Select Edit" on page 200.

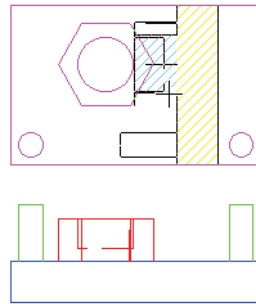
13. The available standard views also include section views. Create a **Sectional View Right**.



14. When this view is inserted, the scale is too large and the orientation is incorrect, relative to the **Plan** view.

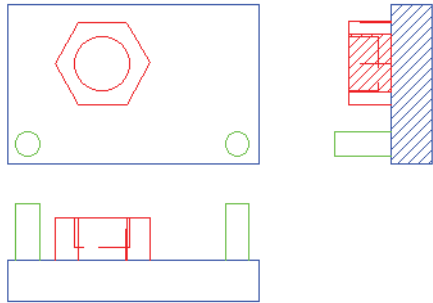


15. Drag the section view over the **Plan** view. This not only sets the scale, but also sets the orientation.

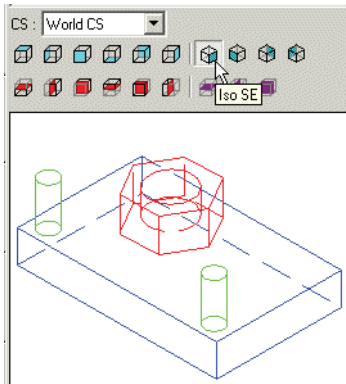


If you press Shift here to break the alignment, the rotation will also change back. Only the modified scale is preserved.

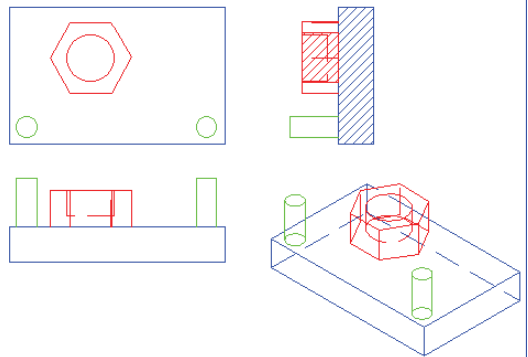
Here are the three views created so far. You can change the hatch pattern of section views in the **Section Brush** page of the view's **Properties**.



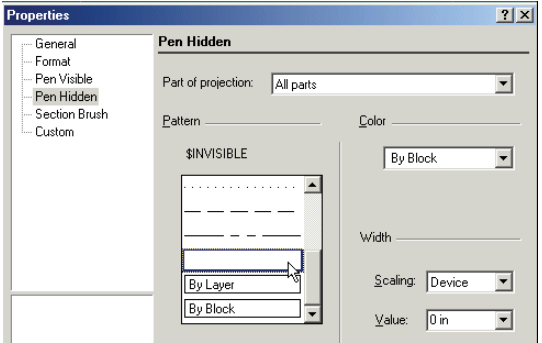
16. For the last standard view, create an **ISO\_SE** view.



17. Insert this view. Because it is isometric, if you try to set the scale by dragging it over one of the other views, the alignment will be diagonal. Therefore, set the scale manually using the **Scale** field.

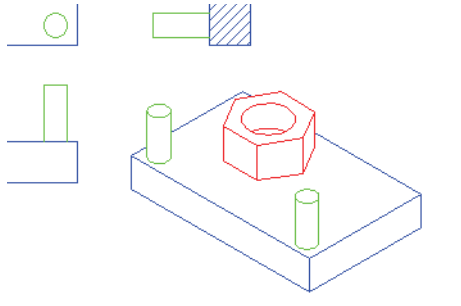


18. To change the appearance of any view, double-click it in **Select** mode to open its **Properties**. For the isometric view, open the **Pen Hidden** page. Set the **Pattern** to **Invisible** so that hidden lines will not be displayed.

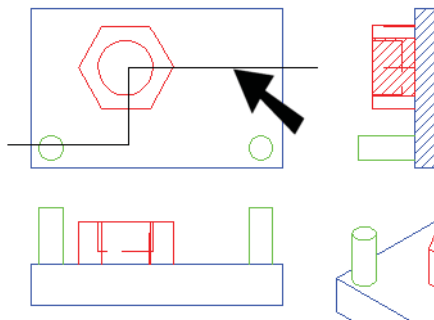


NOTE: For other **Properties** options, see "Properties of Standard Views" on page 655.

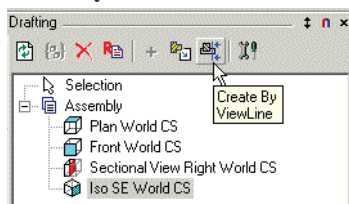
Here is the modified isometric view.



19. Finally, the last view will be a sectional view created from one of the existing views. Use a **Polyline** to draw a multi-segmented section line on the **Plan** view.

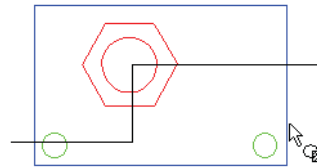


20. Click **Create by View Line**.

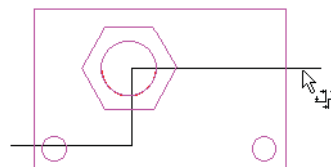


21. If you want to assign text to the view (such as a letter), enter it in the **Text** field of the Inspector Bar.

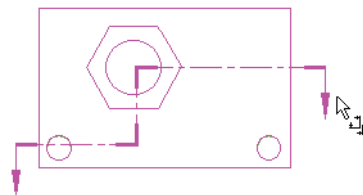
22. First select the view whose sectional view you want to create.



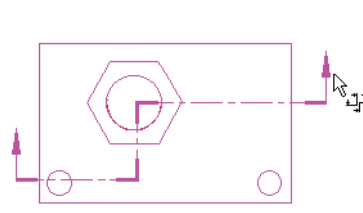
23. Then select the view line, in this case, the polyline.



24. In the next step you define the viewing direction. Move the cursor to switch between the two direction options.



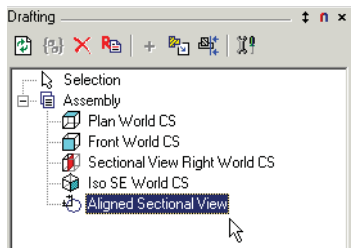
25. Click when the direction is correct.



**NOTE:** You can also change the viewing direction after the section line is created. Open the section line's **Properties** to the **Format** page, and check (or uncheck) **Forward Side**.

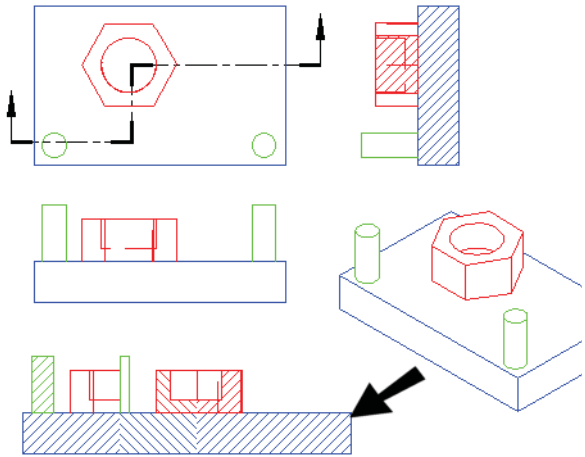


The aligned view is created and is listed in the Drafting Palette, even though it does not yet appear in the drawing.

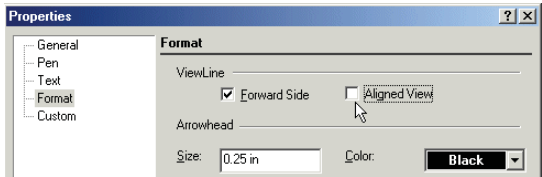


26. Insert the aligned view into the drawing, setting the scale according to the **Plan** view.

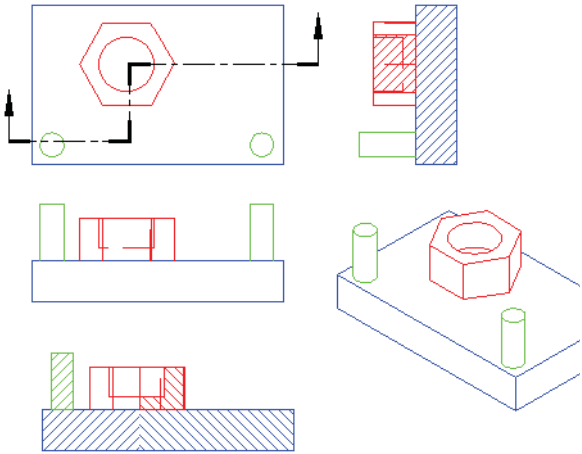
By default, this is an **Aligned** view - what you see is equivalent to “unfolding” the polyline.



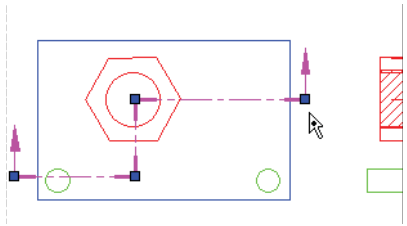
27. To switch to a non-aligned view, open the view's **Properties** to the **Format** page. Deselect **Aligned View**.



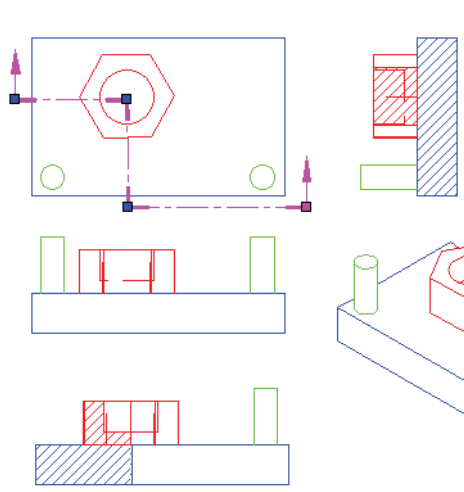
The view is no longer “unfolded.”



28. For another way to change an aligned section view, use the **Edit Tool** on the polyline (see "Edit Tool" on page 219).



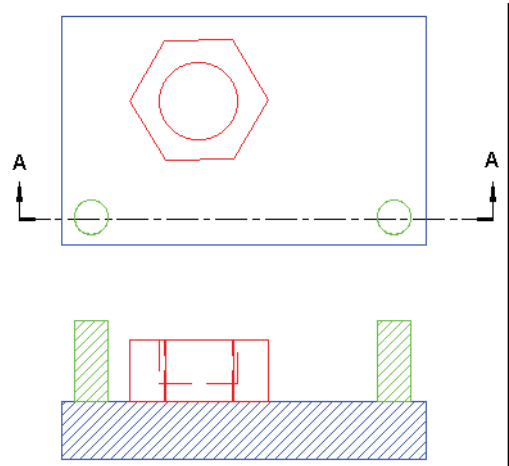
29. Change the polyline by moving, adding, or deleting nodes. The sectional view updates as you change the polyline.



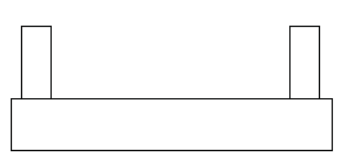
30. If you want to add or change characters on the section line, use the **General** and **Text** pages of the section line's **Properties**. See "Properties of Section Lines" on page 656.

Note that there is a difference between a section and a sectional view. The 3D function **Section** (see "Sectioning Solids" on page 456) creates an actual section of a 3D object, whereas a sectional view is what you see when looking in the direction of the section line.

In this example, the lower view is a sectional view looking in the direction of Line A-A.



And this is the section taken at Line A-A.

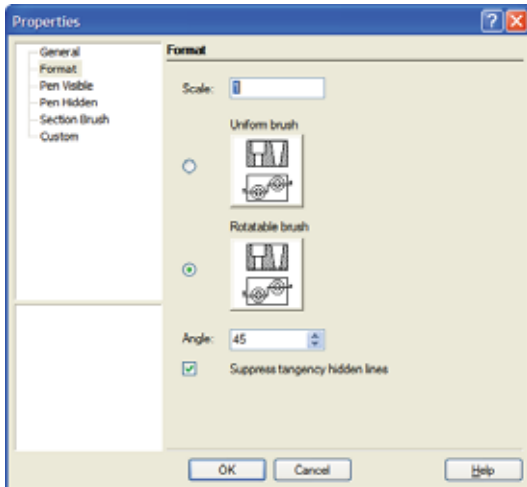


## Properties of Standard Views

For each standard view, the Properties window contains options for changing the scale, pen for visible and hidden lines, and section brushes.

### Format Properties

This window enables you to change the **Scale** of the view.



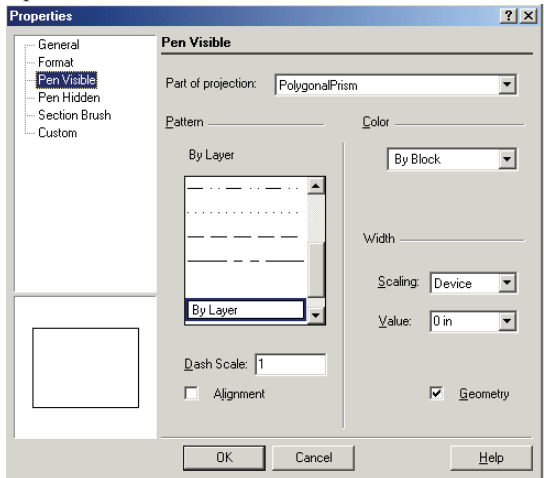
**Brush Rotation:** Sets whether the brush pattern rotates or not when a section line has internal nodes and offsets.

**Angle:** Specifies the angle of brush rotation, if the brush pattern is rotatable

**Suppress tangency hidden lines:** Suppresses overdrawing where two or more lines are overlapping because of tangency.

## Pen Properties

Properties of visible and hidden lines.

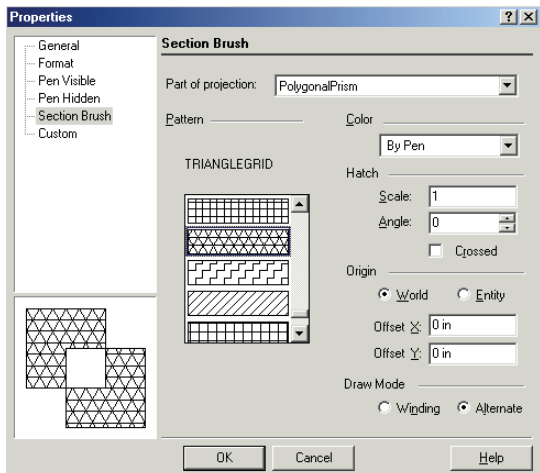


**Part of projection** enables you set pen properties of hidden and visible lines of different objects in the view.

For other options on this page, see "Pen Properties" on page 81.

## Section Brush Properties

Properties of the hatching of section views.



**Part of projection** enables you to set brush properties for different objects in the view.

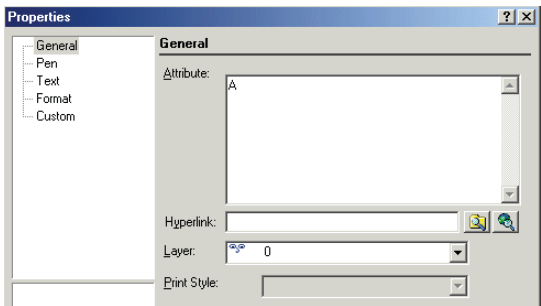
For other options on this page, see "Brush Properties" on page 82.

### Properties of Section Lines

A line or polyline can be used to create a section view.

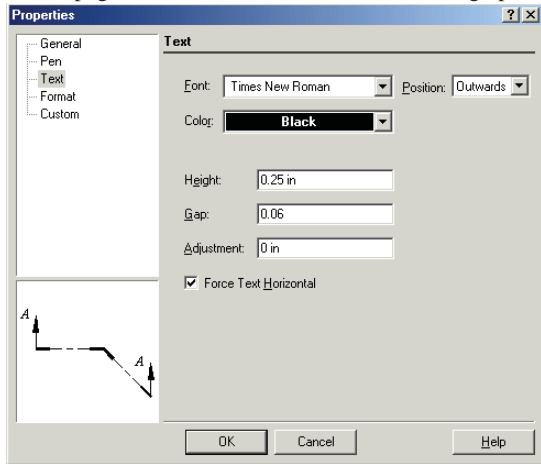
#### General Properties

Use the **Attributes** field to enter or edit characters you want to appear on the section line. The **Text** window contains formatting options for this text.



#### Text Properties

If you entered characters in the **Attributes** field of the **General** page, this window contains text formatting options.



**Position:** Sets the side of the arrow where the text will be located.

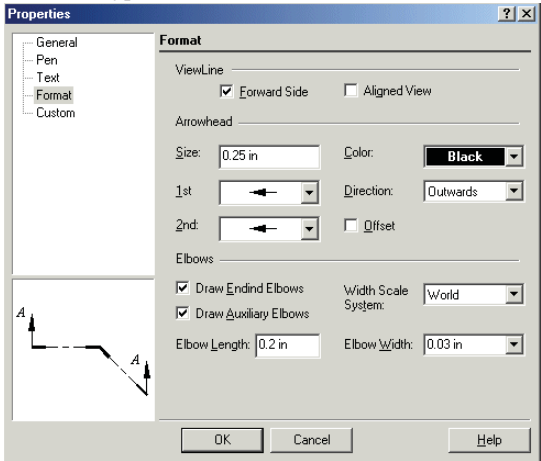
**Gap:** Moves the text away from the tip of the arrowhead.

**Adjustment:** Moves the text in the opposite direction of **Gap**.

**Force Text Horizontal:** Text will be horizontal regardless of the section line orientation.

#### Format Properties

These are properties that define the appearance of the section line, and the type of section created.



#### View Line:

- **Forward Side:** Enables you to switch the viewing direction of the section line.
- **Aligned View:** Select this if you want the section to be created assuming the section line is "unfolded." If unchecked, the section will be created assuming a uniform viewing direction.

**Elbows:** These are the thick additions to the section lines that appear at interior corners and at the ends.

## Viewports



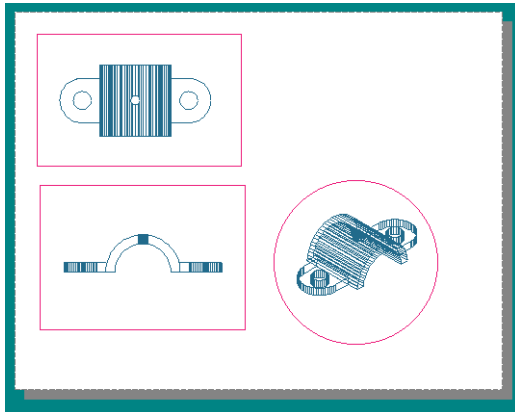
Viewports are used in Paper Space to display one or more views of your model. A viewport has two components: a boundary and the view that it contains. You can only insert views that have already been created - see "Creating a View" on page 103.

---

*NOTE: If you want to insert standard views (views you do not have to pre-define), see "Drafting Palette - Creating Standard Views" on page 643*

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Typically, multiple viewports are created so that you can show several view of the model. If you make changes to your model, any relevant views in the viewports will automatically update.



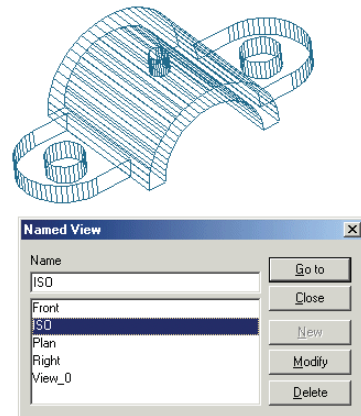
Once a viewport is created, you can use 2D tools and annotation tools (**Text**, **Dimensions**, **Hatching**, etc.) to enhance the Paper Space.

You also have the option of saving a viewport's contents as an image. This is handy for rendering objects in viewports, because it is faster to display an image than to render an object. See "Cache Properties" on page 660.

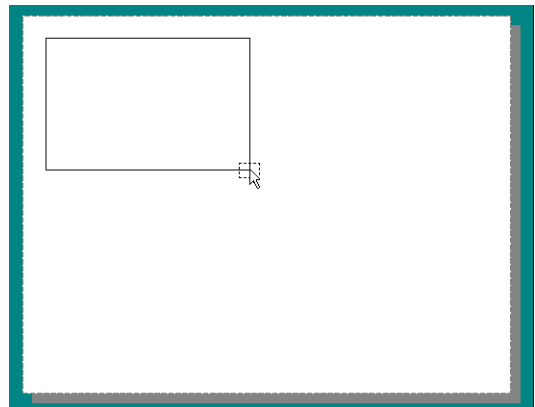
## Inserting Viewports

The following example will be used to demonstrate the use of viewports. To try it yourself, open the file **clamp.tcw** in the **Samples\3D Samples** folder.

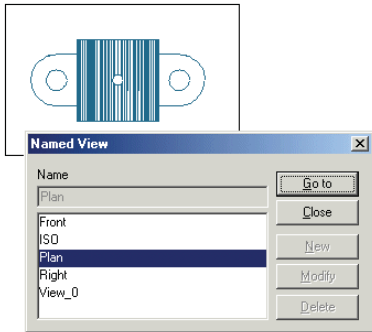
1. Switch to Model Space, and select **View / Named View**. Several views have already been defined. (For details on named views, see "Saving Views" on page 102.)



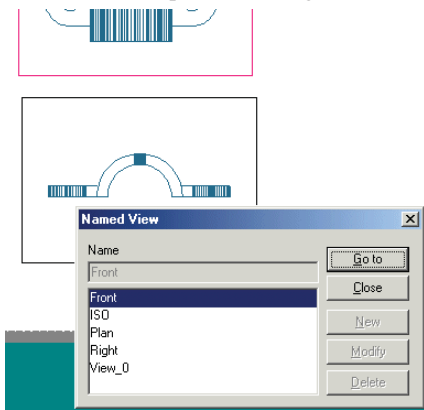
2. There is already one Paper Space tab, containing several viewports. Right-click on the tab and select **Insert** to create a clean Paper Space.
3. Select **Insert / Viewport** (or use the **Insert Viewport** icon) and define a rectangular boundary in one corner.



4. In the **Named View** window, select the desired view (in this case, **Plan**). Clicking **Go To** will display the view without closing the window; this is a good way to check that the view is correct. If you double-click a named view, the view will fill the viewport and the window will close.



5. Insert another viewport containing the **Front** view.



Once a viewport is created, the view inside it can be changed (as well as other parameters) by accessing its **Properties**. See "Viewport Properties" on page 659.

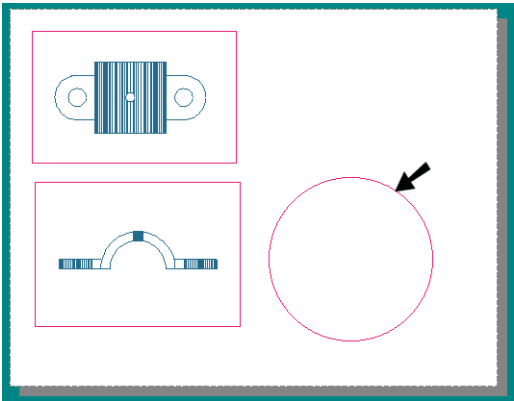
*TIP: If you want to create a viewport of the same size as an existing one, you can copy it (see "Copying Objects" on page 206). You can then open its **Properties** to select another view.*

Local menu option:

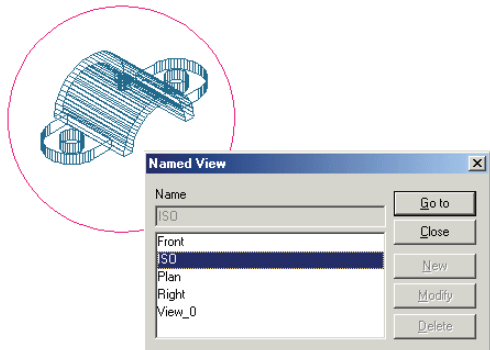
**Shaped Viewport:** Enables you to use any closed 2D object as the viewport boundary.

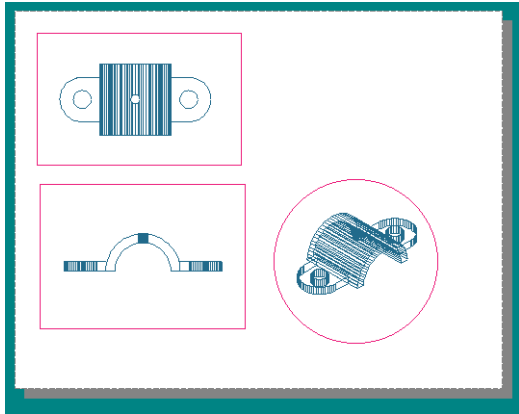


1. In the Paper Space drawing sheet, use the 2D tools to create the closed boundary, in this case, a circle.



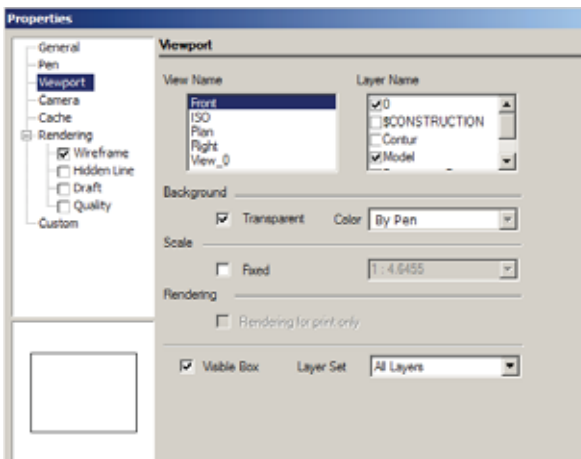
2. Activate **Insert Viewport** with the **Shaped Viewport** option. Select the closed boundary you just created.
3. Select a view as before, in this case, ISO.





## Viewport Properties

Options for the viewport appearance and the objects it contains.

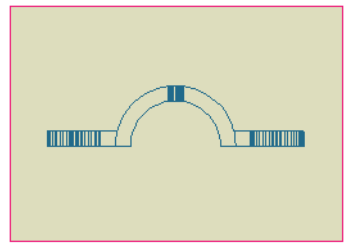


**View Name:** Shows all available views. You can use this list to select another view to fill the viewport.

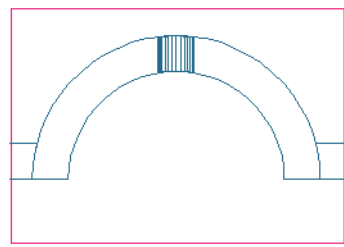
**Layer Name:** Shows all drawing layers. You can check all layers you want displayed.

**Override Layer Visibility:** If this option is on the viewport ignores global layer visibility and uses its own set of visible layers. By default this property is on. But when an DWG/DXF is imported this property is set to “off”.

**Background:** If **Transparent** is not used, you can set a color for the viewport background.



**Scale:** Sets the ratio of the viewing scale for the viewport. To set the scale, check the **Fixed** box and then select or enter the scale. The viewport boundaries do not change, so if the scale increases the view, it may extend past the boundaries.

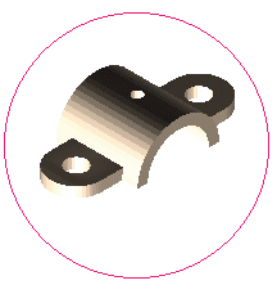
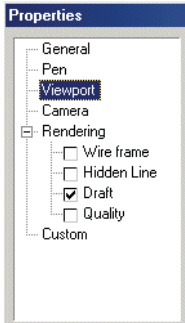



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*NOTE: If dimensions are included in the viewport, the dimension is associative with the viewport. This means that the dimension is scaled simultaneously with the scaling of the viewport. To make the dimension non-associative, select both the viewport and the dimension and choose **Drop link** from the local menu.*

---

**Rendering:** If you want the view inside a viewport to be rendered, check the desired render type on the left side of the **Properties** window.



NOTE: If **Hidden Line**, **Draft**, or **Quality** mode is specified, you can explode the viewport. An exploded viewport becomes a Group of graphics or Picture object, which you can verify in the Selection Info Palette.

**Rendering for print only:** The rendered view appears only when printed, not on the screen.

**Lock Camera:** Using Space Change through a viewport allows you to maneuver the view into model space by panning and turning. The Lock Camera setting causes the view to automatically return to the camera location and direction that were in place before using space change.

**Lock Scale:** Using Space Change through a viewport allows you to maneuver the view into model space through zooming. The Lock Scale setting causes the view to automatically return to the zoom level and view scale that were in place before using space change.

**Visible Box:** Displays or hides the viewport boundary line.

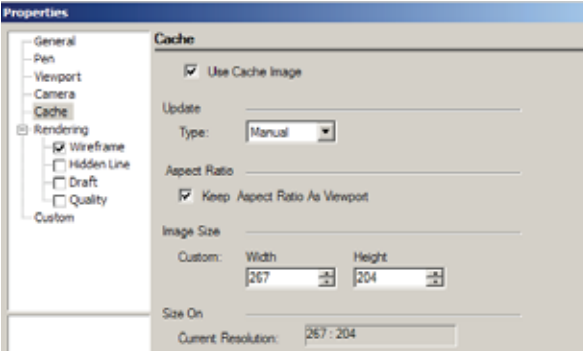
TIP: When a viewport's boundary is invisible, it can be hard to select the viewport. You can also use F6 to scroll through the viewports while selecting.

**Layer Set:** See "Layer Sets" on page 120.

**Viewport is On:** This option allows you to show or hide the content of the viewport. If the option is deselected the content of the viewport will be hidden.

**Cache Properties**

Options for storing images in viewports.



**Use Cache Image:** The viewport image will be stored as a picture, so that the image does not have to be regenerated. This is handy for large, rendered images that can take time to generate.

**Update:** If **Manual** is selected, the generated picture will be updated after you select **Update Viewport Cache**.

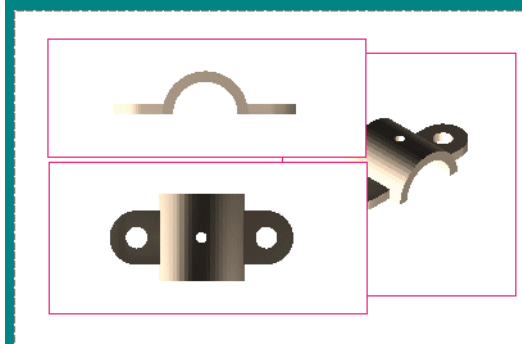
The remaining options on this page control the quality and size of the generated picture.



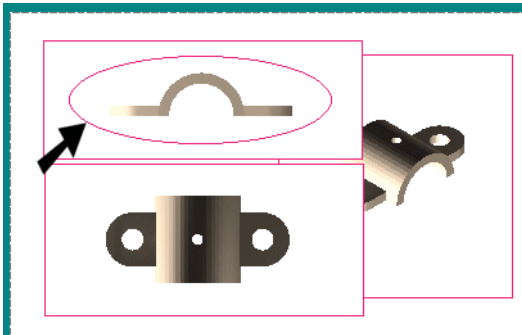
## Update Viewport Boundary



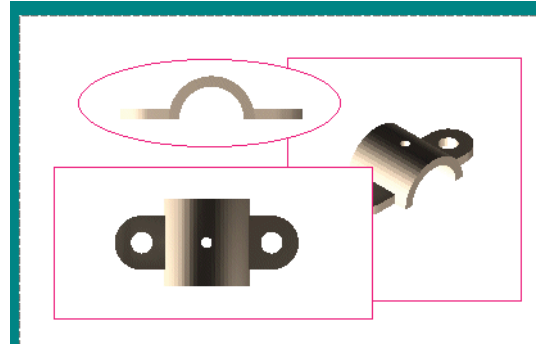
A viewport's boundaries can be replaced with any closed 2D object. This is useful in cases like the one shown below, in which the viewport on the right is partially hidden beneath the other two viewports.



1. Create the closed 2D object you want to use as the new boundary. In this case, the top left viewport boundary will be replaced with an ellipse.



2. Activate **Update Viewport Boundary**, and select the boundary to change. Then select the new boundary.



If the new boundary is too large or too small for the view inside, you can open the viewport **Properties** and change the **Scale**.

*TIP: The conventional way to select the viewport is to click its boundary. It can sometimes be hard to select the viewport if its boundary is invisible. In this case, use F6 or F7 to scroll through the viewports.*

## Overlapping Viewports

Viewports behave like standard 2D objects, in terms of their stacking order and overlapping. You can adjust overlapping viewports using the **Format** menu (**Bring to Front**, **Send to Back**, etc.). See "Stacking Objects" on page 248.

## Floating Model Space

*Available in TurboCAD Pro and Platinum only*



Enables you to use Model Space tools within a viewport in Paper Space. This is useful for making minor adjustments to your model within Paper Space, without having to switch to Model Space. Any substantial changes, however, should be done in Model Space.

1. Select the viewport you want to use to edit the model.
2. Select **Model Space (Floating)**.

3. The viewport is outlined in bold. You can now make minor edits using most of the Model Space tools.
4. Click outside the viewport to finish editing and return to Paper Space.

---

**WARNING:** *If you rotate the view in a floating viewport, the view will **not** return to its original position when you return to Paper Space.*

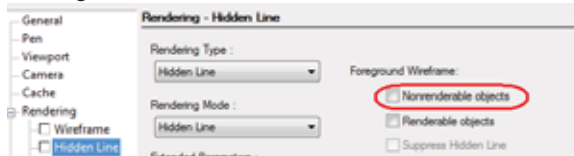
---

## Exploding Viewports

**Wireframe Mode** Available in TurboCAD Pro and Platinum only

**All Other Modes** Available in TurboCAD Platinum only

You can explode a viewport. Wireframe views and Hidden line views are turned into 2D entities. If the viewport is in a rendered (Draft, Quality, advanced) mode the result will be a 2D image.



To work Non-Renderable objects must be turned off in the viewport rendering mode you have selected.

1. Select the viewport.
2. Click Explode.

## Printing

TurboCAD has a wide range of printing features, enabling you to scale and center your drawing on the page or to tile your drawing across multiple pages. TurboCAD also provides a full range of advanced printing options for experienced users.

Paper Space is the work mode used for layout your drawing for printing. See "Paper Space" on page 641.

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**NOTE:** *Under Windows, plotters are treated as printers.*

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## Printer Paper and the Drawing Sheet

One potentially confusing concept is that of the printer paper versus the drawing sheet. Printer paper means the physical sheets of paper that go through the printer. The drawing sheet is the total surface area on which your drawing will be printed. The drawing sheet can correspond to a single sheet of printer paper, or it can span multiple sheets of printer paper.

Both the printer paper and the drawing sheet have adjustable sizes and orientations. This provides a great deal of flexibility when you want to print a large drawing on multiple sheets. You could, for example, print a poster in landscape orientation three feet wide and two feet tall. You could print this virtual sheet out on 32 sheets of 8.5" x 11" paper oriented in portrait, or on 30 sheets of the same size paper oriented in landscape.

## Simple Printing and Tiled Printing

To make a simple print of a drawing that fits on one sheet:

1. Select **File / Print** to open the Print window.
2. For **Print Range**, click **All** to print your entire drawing.
3. Check **Fit on 1 Page**.
4. Select the printer, and click **Properties** to set the paper size and orientation.
5. Return to the **Print** window, and click **OK** to print your drawing. Your drawing will be centered and printed on a single sheet of paper.

If your drawing is larger than the printer sheet, as it is in most cases, you need to use tiled printing. This method prints sections of your drawing on separate pages, which can then be pieced together. The **Page Setup** is used to set parameters for tiled printing (see "Page Setup" on page 664).

---

**TIP:** When tile printing, it is usually best to turn the **Print Crop Marks** option on, unless your printer is capable of printing with zero margins. Crop marks enable you to easily trim the edges of the paper so that the sheets can be pasted onto a backing for display.

---

An example of tiled printing is as follows:

1. Select **File / Page Setup**.
2. On the **Paper** page, use the **Printer Paper** field to specify the paper and orientation.
3. In the **Drawing Sheet Size** fields, specify the size and orientation of the area on which you want to print the drawing. For example, if you want a printout 2 feet tall and 3 feet wide, you would type 2 ft x 3 ft in the list box, and choose the **Landscape** option. Or you can select a standard size.
4. Click **Fit** to place your drawing on the tiled sheets.
5. Click **Print** to print your drawing immediately, or click **OK** and print later using **File / Print**.

Another way to tile print is to specify the number of rows (the number of pages from top to bottom) and columns (the number of pages from left to right). This is done on the **Layout** page of the **Page Setup**. You can then return to the **Paper** page to fit your drawing onto the tiled printer paper. TurboCAD will automatically adjust the dimensions of the drawing sheet to accommodate changes in the number of rows and columns.

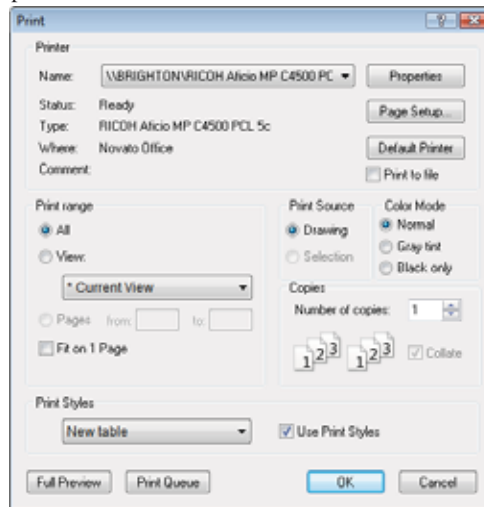
## Print Options

**Hotkey:** Ctrl+P

**Icon on Standard Toolbar:**



Provides a set of tools for printing a single view of a drawing, or printing specific sheets drawing spanning multiple sheets of paper.



**Printer:** Select a printer and manipulate its properties.

- **Name:** Select the desired printer from the drop-down list. All printers set up on your system should appear on this list.
- **Properties:** Opens the **Properties** window for the selected printer.
- **Page Setup:** Controls exactly how drawings will be printed. See "Page Setup" on page 664.
- **Default Printer:** Selects the Windows default printer.
- **Print to File:** Prints the document as a file instead of routing it directly to a printer. You will be prompted to specify the filename and location.

**Print Range:** Options for setting a specific view of the drawing to print, and for selecting a range of pages when printing a drawing that spans multiple printed sheets.

---

*TIP: This is useful for printing tiled drawings if you only want to print specific pages.*

---

- **All:** Prints the entire current drawing. If you choose **Selection** under **Print Source**, only objects that are selected will be printed. If your drawing spans multiple printer sheets, the total number of sheets will be shown in this option title.
- **View:** Prints a single view of the drawing. You can print the current view by default, or click in the list box and choose any named view.
- **Pages:** Prints a range of pages if the **Page Layout** is set to print multiple pages. The printer sheets are counted starting in the lower-left corner from left to right, and from bottom to top.
- **Fit on 1 Page:** Prints everything on a single page. This option can be used either for printing a view or printing the entire drawing.

**Print Source:** Select what you want to print.

- **Drawing:** Prints the current drawing.
- **Selection:** Prints only selected objects.

**Color Mode:**

- **Normal:** Uses the predefined printer settings.
- **Gray tint:** Used for color printers. Disables color mode and prints in black and white, using grey tints for other colors.
- **Black only:** Disables color mode and prints in black only; no grey tints are used.

**Copies:** Sets the number of copies and whether you want multiple copies to be collated.

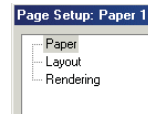
**Print Styles:** Sets the print style to be used while printing. See "Print Style Options" on page 667.

**Full Preview:** Previews your drawing before printing.

## Page Setup

**Menu:** File / Page Setup

Controls exactly how drawings will be printed. The three pages of this window are **Paper**, **Layout**, and **Rendering**.




---

**NOTE:** ***Page Setup** is different than the **Page Setup Wizard** (see "Page Setup Wizard" on page 35), but any relevant parameters set in the Wizard will appear in **Page Setup** as well.*

---

After setting the parameters in this window, you can either print directly from the **Page Setup** (click **Print**), or click OK to accept the changes and return to the drawing. You can then print using **File / Print**, and the **Page Setup** settings will be used.

---

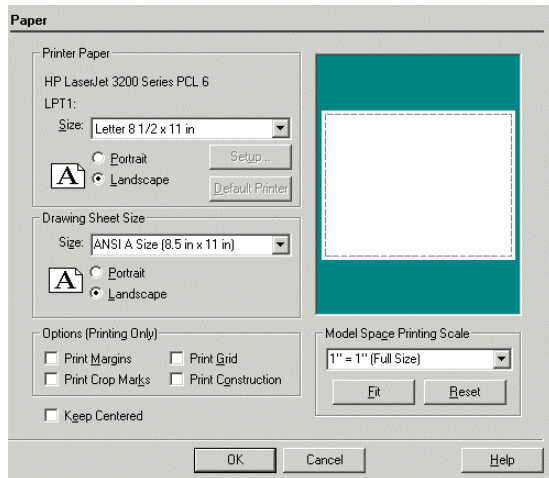
***TIP:** You can also access the **Page Setup** via the **Print** window.*

---

## Page Setup - Paper

Sets the size, orientation, and other parameters of the printer paper and the drawing sheet.

The printer paper is the physical paper that goes through the printer; the drawing sheet is the area on which your drawing will be printed, which can span multiple sheets of printer paper.



**NOTE:** These settings are not retained with the drawing file.

**Printer Paper:** Sets the size and orientation of the printer paper. These options can also be set via the **Print** window.

- **Setup:** Enables you to choose another printer or access functions that directly control your printer. This window is specific to your printer driver, so check your printer or printer driver documentation for details.
- **Default Printer:** Sets the printer paper parameters to those of the default printer.

**Drawing Sheet Size:** Controls the size and orientation of the area on which your drawing will be printed. This area can be imposed on a single sheet of printer paper, or it can span multiple sheets of printer paper. Its orientation can also be set separately from the orientation of the printer paper.

The page display on the right will reflect the relationship between printer paper and drawing sheet. The drawing sheet size will be shown in white (or a custom color if one has been set), and the dashed lines show individual pages of paper. Changing the drawing sheet size may change the number of columns and rows.

**Options:** Enables you to set what elements are printed. These options affect only the printing of the drawing and are not reflected in the appearance of the drawing on screen.

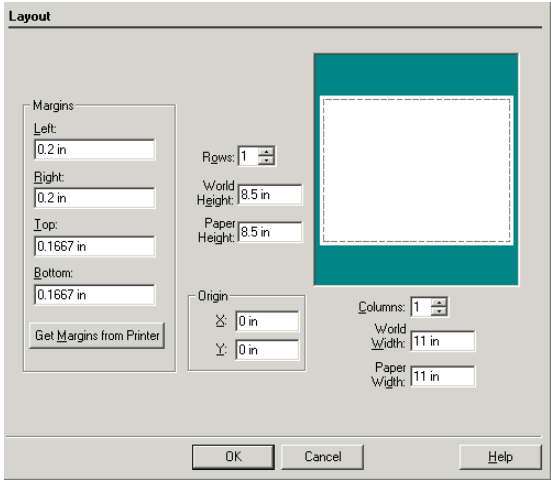
- **Print Margins:** Prints a border along the margin of the printer sheets. This is useful for framing a drawing that is printed on a single sheet.
- **Print Crop Marks:** Prints crop marks on the printer sheets. This is useful if you want to print across multiple printer sheets and crop the sheets so they can be pasted together. You may need to increase your margins to make room for the crop marks.
- **Print Grid:** Prints the grid, as long as it currently displayed. See "Grid" on page 107.
- **Print Construction:** Prints construction geometry. See "Construction Geometry" on page 121.

**Printing Scale:** Scales the drawing so that it fits within the drawing sheet (click **Fit**), or you can choose a custom scale. If the desired scale does not appear in the list, you can type directly using the format 1 in = 1 ft.

**Keep Centered:** Keeps the drawing centered on the drawing sheet. It may be necessary to turn this option off, then on again, for the desired effect.

### Page Setup - Layout

Controls the number of rows and columns (sheets of printer paper arranged vertically and horizontally), as well as other paper parameters.



**Margins:** Margins are the dotted rectangle lines inside the edges of the paper, that let you know whether you are drawing inside or outside the printable area of your drawing. You can specify the top, bottom, left, and right margins of the printer paper in Paper units. You can also enter units other than the default units.

- **Get Margins from Printer:** Sets the minimum margins for your printer and page size.

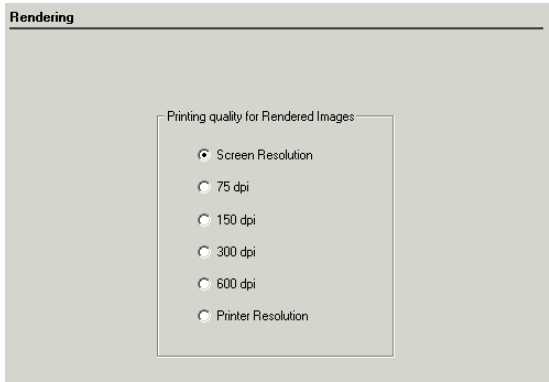
**Rows / Columns:** Sets the number of rows and columns of printer sheet paper for tiled printing. As you increase the number of rows and columns, the **Drawing Sheet Size** (the area on which your drawing will be printed) increases accordingly.

**World Height / Width, Paper Height / Width:** Sets the dimensions of the drawing sheet in world or paper units. As you change these values, the number of rows and columns will automatically update.

**Origin:** Enables you to move the paper to a different location in the drawing. Specify the location of the lower-left corner of the paper in absolute coordinates. This feature is useful if you want to print a particular area of a large drawing by relocating the paper to the area that you want to print.

### Page Setup - Rendering

Sets the resolution at which a rendered drawing will be printed.



**Printing quality for Rendered Images:** Select the resolution for printing rendered drawings.

---

*NOTE: When the resolution is set to a value that is higher than the maximum resolution supported by the printer, the image will be printed at the printer's maximum resolution.*

---

### Printing from Model Space

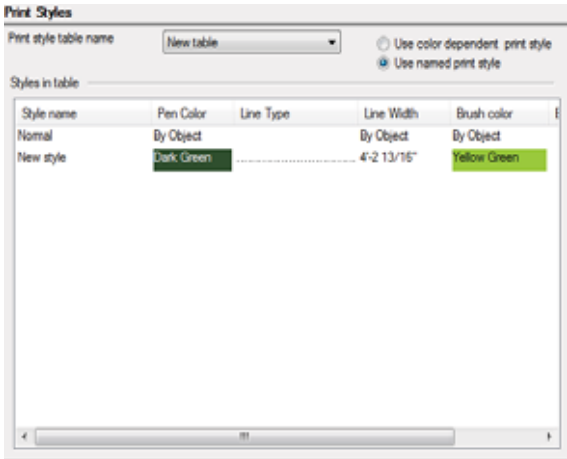
To get a printed copy of the objects appearing in the current window view, you can use **File / Print** and choose to print the current view.

### Print Style Options

The Print Styles Option dialog allows you to View and Specify the current print style.

#### To set the Current Print Style

1. Open the Print Styles Dialog.
2. Select the Style from the Print Style Table Name drop-down.



### Selecting Print Style Type

You can use two different Print Style type: Named Based and Color Based. Only one type of styles can be used in a drawing at the same time.

#### STB / Name Based Print Styles

Name based print styles correspond to AutoCAD STB files. They map a named style to a specific print presentation and group of settings.

#### CTB / Color Based Print Styles

Color based print styles correspond to AutoCAD CTB files. They map each indexed colors to a specific print presentation and group settings.

### To Specify the Type of Styles Used

1. Open the Print Styles Dialog.
2. Use the option button to specify whether you want named or color based print styles.

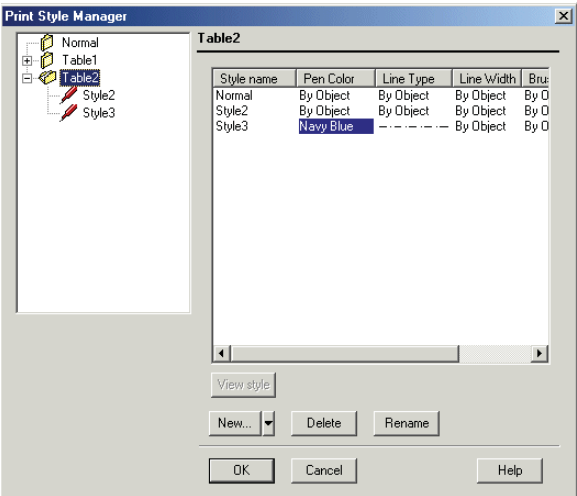
### Print Styles

Enable you to change color, line style, and brush of an object when you send the drawing to print. These settings do not affect the objects on the screen, only how objects are printed.

### Print Style Manager

Menu: Tools / Print Style Manager

Print styles can be created, modified, or imported via the Print Style manager.



**View Style:** Opens the **Properties** window for the selected style.

**New:** Enables you to create a new table, or a new style within the selected table. You can also duplicate items, and import AutoCAD plot styles.

---

NOTE: These features are also available by right-clicking in the table list area.

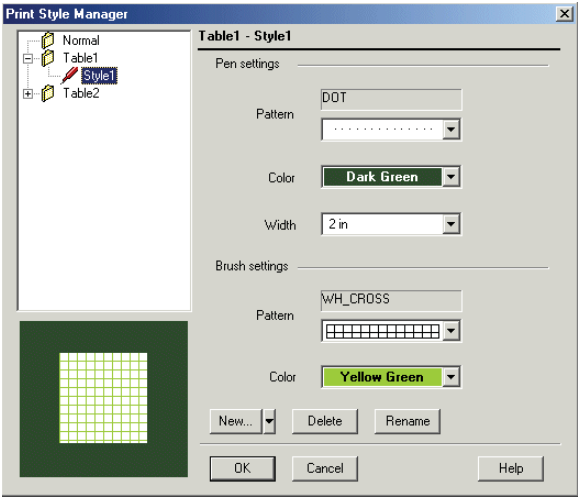
---

**Delete and Rename:** Removes or renames the selected table or style. Deleting a style cannot be undone.



Print Style Properties

To access the properties of any print style, select the style and click **View Style**.



**Pen Settings:** Sets the pattern (line style), color, and width for the pen that will be applied to an object during printing.

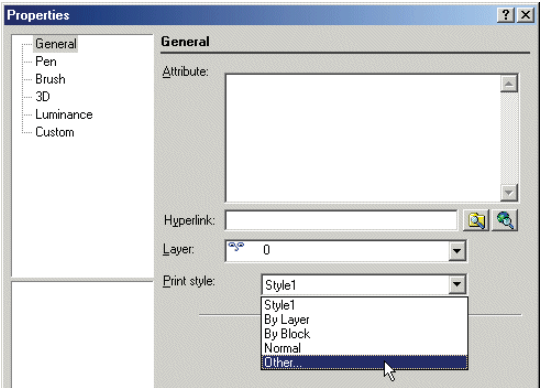
**Brush Settings:** Sets the pattern and color for filling objects during printing. This only applies to objects to which a brush has been assigned.

Applying Print Styles

Once print styles have been created and/or modified, you can add them to your drawing via the **Print Style** page of the **Drawing Setup (Options / Print Styles)**. While in Paper Space, you can also access this page through the **Properties** window.

An object’s print style is set in the **General** page of its **Properties**.

Select “Other” from the drop-down menu to open the **Print Style** manager.

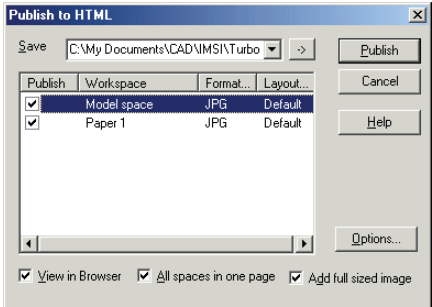


NOTE: *Certain types of objects cannot display print styles.*

Publish as HTML

Available in TurboCAD Pro and Platinum only

Enables you to export your drawing as an HTML file. You can export the whole drawing or any of its workspaces.



**Save:** Select the folder path or click the arrow button to browse. The \*.htm file will be created in this folder, as well as two subfolders to store the graphic images for Paper and Model Spaces.

**Publish:** Click to publish the selected workspaces to HTML.

**Format:** Sets the export format associated with each workspace. Click the column to open the format list. You can choose from \*.gif, \*.jpg, \*.png, \*.wrl, \*.dwf, and \*.mtx.



*NOTE: To display \*.wrl and \*.dwf formats, you need to have an application that will transfer such drawings to your Internet browser. Otherwise you may receive a warning message.*

**Layout:** Click the column to open the format list, and select either **Default** or **Custom**. If **Custom** is selected, you can define the custom layout after clicking **Publish**.

**View in Browser:** Launches the default Internet browser to show the HTML you created.

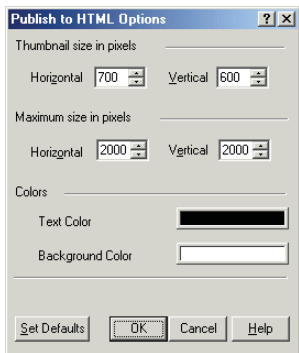
**All spaces in one page:** All selected workspaces will be placed on the same HTML page.

**Add full sized image:** The graphic will be created with the settings defined in the **Maximum size in pixels** control group of the **Options** dialog.

**Options:** Opens the **Publish to HTML Options** window.

## Publish to HTML Options

Parameters for the exported file.



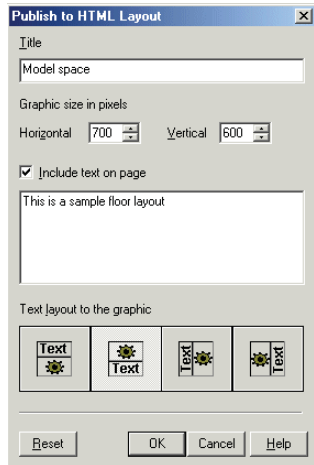
**Thumbnail size in pixels:** Sets the size of the graphic image. You can change these settings if you use a **Custom** layout.

**Maximum size in pixels:** Sets the size of the linked graphic image that will be accessible if **Add full sized image** is checked in the **Publish to HTML** window.

**Colors:** Sets the background and text color. You can define text if you use a **Custom** layout.

## Publish to HTML Layout

Sets the HTML page layout. This window appears when publishing HTML files for one or more workspaces that have a **Custom** layout.



**Title:** Use the default (workspace name) or add a custom title for the graphic.

**Graphic size in pixels:** Sets the size of the graphic image.

**Include text on page:** Adds the text you type in the box to the HTML file.

**Text layout to the graphic:** If text is added, select the text layout position relative to the graphic.



# 17 Customized Programming

You can customize your TurboCAD application by recording and editing macros, and using the SDK.

## Macro Recorder

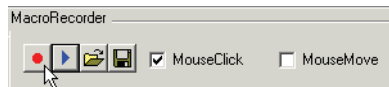
*Available in TurboCAD Pro and Platinum only*

**Menu:** View / MacroRecorder Palette

Creates and plays scripts - scenarios of creating objects and manipulating their properties. You can also record object transformation - move, copy, scale, and rotate. Scripts can be saved for future play.

### Recording and Playing Scripts

To record a script, click **Start Record** in the MacroRecorder palette.



#### Script Options:

- **Mouse Click:** The script will include each mouse click, including work in the Inspector Bar, use of the local menu, etc.).

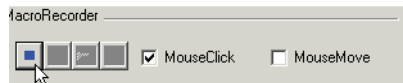
A script created with mouse clicks can be used only in TurboCAD. A script that was created without mouse clicks can be used in any TurboCAD add-on, if the add-on is created based on the TurboCAD Object Model.

*NOTE: A script that uses mouse clicks cannot close windows. For example, if a viewport creation is recorded, when this script is played, the **Named View** window must be closed manually.*

- **Mouse Move:** Available when recording mouse clicks, records all mouse moves as well. For example, if you record the creation of a circle using mouse moves, you are recording exactly how the cursor moves while sizing the circle. Without mouse moves, you will only record the end result of the circle. This option increases the size of the script, and results in slower playback.

Mouse move recording is recommended for scripts in which exact mouse positioning is important, such as work in the **Edit Tool**.

To finish the script, click **Stop Record**.



*WARNING: Keep in mind that the script will only contain the actions performed after starting the script and before ending it. For example, you can record creating a line, stop the recorder and then change the line color. When you play the script, the line will have a different color because the script does not include the color change.*

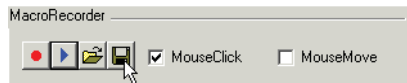
The script text will be displayed in the **MacroRecorder** palette. You can edit the script directly in the palette by placing the cursor on the desired line and changing text.

You can play the script back by clicking **Play Script**.

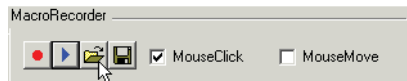


If you clear all items from the screen, or open a new file, playing the script will create new objects on the screen.

To save the script, click **Save Script**. Scripts are saved as \*.tcr (TurboCAD Recorder) files.



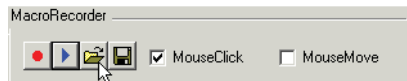
You can play a saved script by clicking **Open Script**.



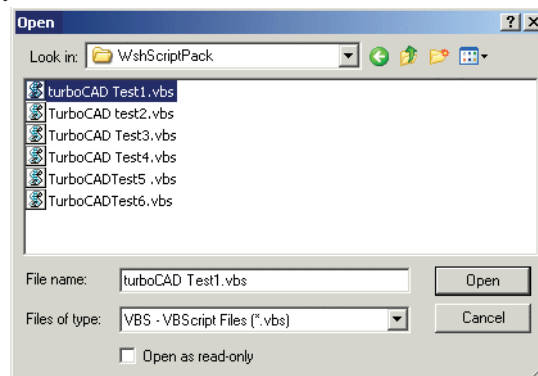
Select the desired script, and it is loaded into the palette. Click **Play Script** to play it.

## Sample Scripts

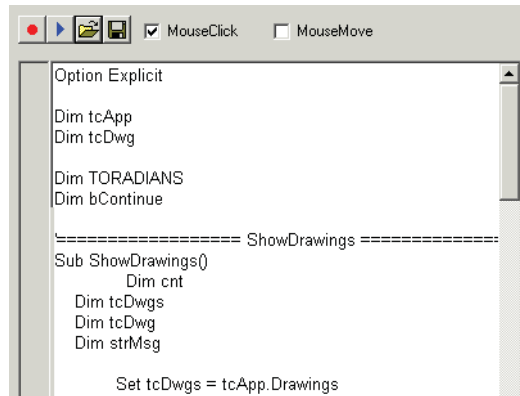
Several sample scripts are provided for you. Click **Open Script**.



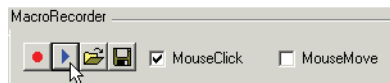
Set **Files of type** to VBS and browse to TurboCAD's installation folder. Browse to SDK \ Samples \ VBS \ WshScriptPack. This folder contains a few scripts you can run.



Select the script you want to run and click **Open**. The script opens the Macro Recorder palette. (You can also open these files in a text editor.)



Click **Play Script**.



Some scripts give you messages related to what is found in your open files. The last script draws several 3D objects.

## Script Limitations

Currently, some TurboCAD objects, such as images, cannot be used in scripts.

Scripts record object creation only, and not all TurboCAD settings can be reproduced. For example, workplane position and views cannot be set in a script. Therefore, set the desired positions of the workplane and views before playing the script.

## SDK

*Available in TurboCAD Pro and Platinum only*

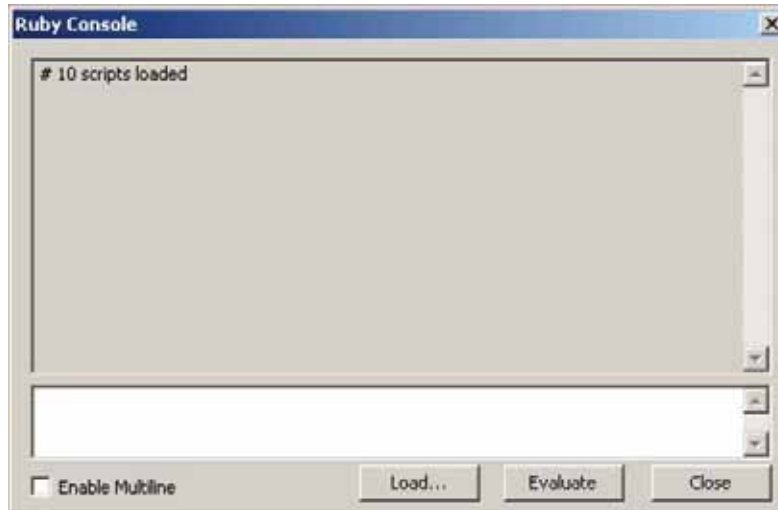
The TurboCAD Software Development Kit enables you to program your own routines within TurboCAD.

If you install TurboCAD using the **Full** installation, an SDK folder is created. This folder contains SDK samples and documentation (online help).

Several SDK tools are provided in TurboCAD, located in the **AddOns** menu. These are located in the sections to which they are relevant. Each tool can be found in the index.

## Using the Ruby Console

When you first start TurboCAD, the Ruby Console opens as well.



You can use the Ruby console to run functions, load ruby scripts, or even define new functions.

The top portion of the Ruby console is the **Output Panel**. This is where Ruby scripts will put text output, and where the Ruby engine will notify you of any errors that it has encountered, or provide other notifications. It is possible to copy text from the Output Panel to the clipboard for reuse in the Input Panel or elsewhere.

The bottom panel of the Ruby console is the **Input Panel**. Here you can type in any functions that you want to call, or define new values or even functions.

The **Load...** button allows you to open a ruby script using a standard "Open" dialog box.

The **Evaluate** button tells Ruby to evaluate the text that you have typed into the Input Panel.

The **Close** button closes the Ruby Console. After you close the Ruby console you can re-open it at any time from the *Scripts/Toggle Ruby Console* menu command.

The **Enable Multiline** checkbox allows you to turn Multiline input off or on.

- When **Enable Multiline** is unchecked (the default), only a single line of input in the input panel is evaluated. This mode is convenient for running already-defined functions or defining and setting variable values on the fly. In this mode, pressing the **Enter** key is the same as pressing the **Evaluate** button.
- When **Enable Multiline** is checked, you are allowed to enter as many different lines of text as you want. In this mode, the **Enter** key works to end the current line and move the cursor to a new line, instead of performing the **Evaluate** function. Multiline mode can be useful if you want to quickly define a simple function right in the Ruby console. After you are done entering lines, be sure to uncheck **Enable Multiline** before pressing **Evaluate**.

## Loading a script with the Load... button

To load a ruby script, click on the Load... button in the Ruby Console. This opens a dialog box that lets you browse to a ruby script and load its functions and other definitions into memory.

Note, however, that loading a script in this way will not automatically execute any of the methods in the script – you'll have to do that from the Ruby console as well. For example, if you want to execute a function **draw\_stuff** which is contained in the ruby script **ConsoleLoadSample.rb** you would do the following:

- Click on the "Load..." button.
- Using the "Open" dialog, browse to the folder containing ConsoleLoadSample.rb.
- Highlight ConsoleLoadSample.rb in the file list, and click "Open". The Output Panel adds a line "true" to indicate the script was opened successfully, but nothing else visibly happens.
- Type "draw\_stuff" in the Input Panel, and click **Evaluate** or press **Enter**. Now the function draw\_stuff will perform whatever tasks it is supposed to.

Once the script has been loaded, it remains in memory for the duration of the TurboCAD session – you won't have to reload the script each time you want to run the draw\_stuff function; simply enter "draw\_stuff" in the input panel and press **Enter** to run the function again.

## Loading a script using the load command

You can also load a script by using the **load** command in the Input Panel. In this case, you must specify the full path to the script, and use double backslashes as path separators. For example:

```
load( "C:\\MyRubyScripts\\draw_plus.rb" )
```

If the script is loaded successfully, the Output Panel will print a "true" response. If not, the Output Panel will print out one or more error messages. As when you use the **Load...** button, loading a script in this fashion does not automatically run any of the functions in the script.

## Defining a function in the Input Panel

To create a new function definition in the Ruby Console is straightforward. Let us walk you through a simple example:

1. Open the Ruby Console, if it's not already open.
2. Place a check mark in the **Enable Multiline** checkbox.
3. Type the following lines into the input panel, using the Enter key to end each line:
 

```
def sayit
  puts "There, I said it!"
end
```
4. Uncheck **Enable Multiline**.
5. Click **Evaluate**. (Note: Don't press Enter. Use the Evaluate button.) The Output Panel should echo all the lines of the function, followed by "nil".
6. Type "sayit" in the Input Panel, and hit **Enter** or press **Evaluate**. The Output Panel should produce "There, I said it," followed by "nil".

## Setting variables in the Input Panel

To set a single variable using the Input Panel, disable Multiline entry, and simply enter the variable's definition, and press Enter or click Evaluate.

```
a = 5
```

press **Enter**

To set a series of variables, enable Multiline entry and enter the variables, then disable Multiline entry and click Evaluate.

```
a = 6
b = 4
c = 5
puts a
puts a*b
puts b+c
```

Disable **Multiline** and click **Evaluate**.

## Clearing the Output Panel

Any time you want to clear the accumulated text in the console's Output Panel, use the cls command in the Input Panel:

```
cls
```

press **Enter**

## More Ruby

Examples of Ruby scripts can be found in the RubyScripts folder within the Programs folder where you installed TurboCAD.

Looking at these examples is the best way to familiarize yourself with Ruby in TurboCAD. For more advanced information there are several online sights dedicated to programming in Ruby, and there are many books available.

You will also want to familiarize yourself with the TurboCAD SDK for a better grasp of TurboCAD's functions. Some of the Ruby functions available emulate the functions of Ruby as used in Google SketchUp. Therefore it is advisable to look at the documentation of Ruby scripting in SketchUp as well.

# Parametric Part Manager

## Introduction

Parametric parts (PPM) are defined using a text description (script). The script defines the structure, editable properties, and output that result in a parametrically editable part.

The script must be saved with a \*.PPM extension. The name of the file determines the name of the part.

## Examining a Script

A simple example of a parametric part is a rectangle where the width, height and rotation angle are defined though parameters. The script of such part might look as follows:

```
// Here is a description of simple rectangle.
H = Parameter("Height", 5, LINEAR, Interval(0, 100));
L = Parameter("Length", 10, LINEAR, Interval(0, 200));
```



```

Angle = Parameter("Angle", 0, ANGULAR, Interval(0, 360));
Rect1 = Rectangle(H, L);
Rect = RotateZ(Rect1, Angle);
Output(Rect);

```

Let's examine each line of this example:

## LINE 1

```
// Here is a description of simple rectangle.
```

The `'//'` indicates a comment. Comments do not affect the behavior of a part. All text following `'//'` to the end of the line are contained by the comment.

## LINE 2

```
H = Parameter("Height", 5, LINEAR, Interval(0, 100));
```

The second line specifies the definition of the 'H' parameter. Let's break out each element of the line to define its function:

H	This is the identifier (name) of the parameter in the part description
=	The equals sign associates the identifier with its definition
Parameter	This is a function. 'Parameter' defines that H is a Parameter
(	Specifies the start of the Parameter function's properties
"Height"	The name of the parameter that will appear in the Properties dialog
,	Indicates the end of one property and the beginning of the next
5	Assigns the default value for H
,	Separates properties
LINEAR	Specifies that H is a linear value
,	Separates properties
Interval(0, 100)	Specifies the allowable values for H as an interval from 0 to 100
)	Specifies the end of the Parameter function's properties
;	End of definition for H

## LINES 3 – 4

```

L = Parameter("Length", 10, LINEAR, Interval(0, 200));
Angle = Parameter("Angle", 0, ANGULAR, Interval(0, 360));

```

The next two lines in the example are similar to the previous one. They define the characteristics of L and Angle parameters in a similar layout. Note that the 'Angle' parameter uses an ANGULAR interval rather than LINEAR.

## LINE 5

```
Rect1 = Rectangle(H, L);
```

This line uses the Rectangle function to define a rectangle called 'Rect1'. It uses the previously defined H and L parameters to specify its properties, height and length. The center of this rectangle will be at the world origin (x=0,y=0,z=0) in the drawing. More on the rectangle tool will be covered later.

## LINE 6

```
Rect = RotateZ(Rect1, Angle);
```

This line defines a new rectangle called 'Rect' which is a rotated version of 'Rect1', using the Angle parameter to define the rotation.

## LINE 7

```
Output (Rect);
```

The last line specifies that the output of the script will be the rotated rectangle called 'Rect'. This is what the be drawn as the part.

## *Script syntax*

The description of a parametric part consists of the entire contents of a text file, except comments, tabs, and other control characters, which are ignored.

Comments are specified either using “/” characters that mean that all subsequent characters up to the end of the line are comments, or using the pair “/\*” and “\*/” that denote beginning and end of the comment, respectively.

A text description is a set of two types of operators:

```
<Identifier>
```

and

```
<Expression>;
```

## *Identifiers*

The <Identifier> defines the symbolic name of an object. It is a set of Roman letters and Arabic numerals, which must start with a letter.

For example valid names would be:

```
PART2a
```

MyPart  
A134

Object identifiers may not be the same as names of functions or such names as PI, or LINEAR. These are reserved words that are used to designate the constants of the scripting language. The list of all reserved names is provided in the reserved word list which appears at the end of this chapter.

## Expressions

Expressions define the associated identifier. Expression syntax matches the expression syntax in the majority of programming languages. They may define numeric value, arithmetic operations, the dependence of the defined object on other objects and function calls.

**The structure of a function call is:**

```
<Function name> (<list of parameters>),
```

**Examples of correct expression syntax:**

```
(D -1/4) * k;  
Polyline(Point(0, 0.25 - 1/8), Point(0, D), Arc1(L-C, - m, m), Point(0,0));  
A = B + 0.5;  
B = 7;
```

## Arithmetic Operations

Arithmetic operations may use the standard arithmetical operators '+' (addition), '-' (subtraction), '\*' (multiplication), '/' (Division) and parenthesis '(' and ')', to determine the sequence of performing arithmetic operations. Object identifiers and numbers serve as operands.

## Script Semantics

A script contains full description of a parametric part. The collection of script operators determines which actions need to be performed to create the resultant object(s). Correct understanding of a script, requires having a clear understanding of how its operators are interpreted.

Identifiers that are used in a *<Expression>* must be defined. In other words it must have been used as:

```
<Identifier> = <Expression>;
```

The list of resultant objects is defined in the *Output( . . )* operator. The *Output( . . )* operator contains a list of which objects are to be displayed in the resulting part. This operator must be present in the script. Each object in the list of arguments for *Output( . . )* must be defined. In other words it must have been used as:

```
<Identifier> = <Expression>;
```

This operator must be present in the script. At least one object must be listed in the Output operator, but you need not output every object used in the script.

The Output operator determines the method that will be used to create an object with this name.

**A correct script describing a parametric part should conform to the following rules:**

1. A script may have more than one *Output( . . )* operator, but any *<Identifier>* should be contained in only one *Output( . . )* operator.
2. For each object used in the *Output( . . )* operator there should be one, and only one, instance of any *<Identifier>*.
3. For each object used in an *<Expression>* there should be one, and only one, instance of any *<Identifier>*.
4. Each identifier should be used only once as *<Identifier>*
5. Each identifier should be at least once in an *<Expression>* operator or in an *Output( . . )* operator.
6. Circular calculation and interdependent referencing are not allowed. The script must not contain interdependent where “Item One” is defined by “Item Two”, and “Item Two” is defined by “Item One”.

The following situations:

```
A = B + 0.5;
B = sin(A);
```

or

```
A = C+5;
B = D+42;
C = (3*(2+A));
D = A/2;
```

are **not** allowed. The first case A and B define each other directly. In the second case A is defined by B through C, and B is defined by A through D. This also means that an identifier is not allowed to depend on itself. For example, you cannot use an operator of this form:

```
H = H*1.05;
```

7. The sequence of script operators is not important (except certain special cases that will be described later); because operators are sorted before the script is run.

## Basic Functions

Probably the most significant advantage of this method of creating parametric parts is the compact size and clarity of the text description of parametric parts in script form. The set of basic functions used in such a description, determines the level of clarity and simplicity of scripts for a particular class of parametric parts.

---

**Note:** *It is intended that the set of basic functions will expand from version to version.*

---

## Description of Parameters

It is important to understand the structure used to within a Parameter function.

### Format:

```
<id> = Parameter(<name>, <default value>, <type>[, <condition1>][,
<condition2>]..);
```

---

Note: The '<>' markers are used to designate elements in the expression, and the '[' ]' markers are used to indicate elements which are optional.

---

**<name>**                      The name displayed in the user interface;

**<default value>**            The default value of the parameter;

**<type>**                      Defines the parameter type. The following example values are possible:

**LINEAR**, means that the parameter is a linear value in the selected linear units of measure.

**ANGULAR**, means that the parameter is an angular value in the selected angular units of measure. (only degrees are available at this time)

**TEXT**, a text string;

**FONT**, a font name;

**COLOR**, an RGB color value;

**MATERIAL**, a material name;

**CHECKBOX**, a logical value, either ON or Off

**<condition>**

These are optional. They define possible restrictions imposed on parameters. Restrictions can be listed in arbitrary order and may take on the following forms:

**Set(<value>,...)** — a list of permissible values of the parameter

**Interval(<minvalue>, <maxvalue>)** — sets the minimum and maximum values of the parameter;

**LessThan(<value>)** — indicates that parameter value should be less than the specified value

**LessOrEqual(<value>)** — indicates that parameter value should not be greater than the specified value

**GreaterThan(<value>)** — indicates that parameter value should be greater than the specified value

**GreaterOrEqual(<value>)** — indicates that parameter value should not be smaller than the specified value

**Set(FolderList)** — particular case of Set operator, when a list of permissible values are defined by operator FolderList.

Restrictions should not contradict each other. For example, you cannot combine GreaterThan(5) with LessThan(2).

When specifying parameter restrictions, it is not allowed to use identifiers or expressions that directly or indirectly depend on other parameters, as arguments of the above-mentioned functions. Only constants or constant expressions can be used, for example: LessOrEqual(PI/2).

**Example of Parameter Description:**

```
Alpha = Parameter("Rotation Angle", 45, ANGULAR, Interval(-90, 90)); // This
creates a parameter used to define a rotation angle. The name is 'Rotation
Angle', the default is 45, the value type is ANGULAR, and the Interval is from
'-90' to '90'
```

## Functions for Creating 2D Entities

The following functions are used to create 2D graphic entities

### **Circle**

The Circle function is used to create circles

**Format:**

```
Circle(<radius>[, <cx>, <cy>]);
```

**<radius>** Defines the circle's radius

**<cx>, <cy>** Defines optional arguments that set the (x, y) coordinates of the circle center. By default, cx = 0, cy = 0

**Example:**

```
Ñ = Circle(D/2, 0, y0);
```

**A more extensive example:**

```
//circle.ppm -- two circles
r1 = Parameter("Radius1", 2.5, LINEAR, Interval(0.0, 10.0));
r2 = Parameter("Radius2", 1.25, LINEAR, Interval(0.0, 10.0));
xc = Parameter("CenterX", 3, LINEAR, Interval(-100, 100));
yc = Parameter("CenterY", 3, LINEAR, Interval(-100, 100));
c1 = Circle(r1); // circle centered on the origin
c2 = Circle(r2, xc, yc); // circle offset from the origin
Output(c1, c2);
```

**Rectangle**

The Rectangle function is used to create rectangles.

**Format:**

```
Rectangle(<width>, <height>[, <cx>, <cy>]);
```

<b>&lt;width&gt;</b>	Defines the rectangle width
<b>&lt;height&gt;</b>	Defines the rectangle height
<b>&lt;cx&gt;, &lt;cy&gt;</b>	Defines optional arguments that set the (x, y) coordinates of the rectangle center. By default, cx = 0, cy = 0

**Example:**

```
rect = Rectangle(W, H, W/2, H/2); // Left bottom corner is in (0,0) point
```

**Polyline**

The Polyline function is used to create polylines consisting of straight line segments and arc segments.

**Format:**

```
Polyline(<list of arguments>);
```

<b>&lt;list of arguments&gt;</b>	Defines the list of arguments, delimited with commas. Arguments define individual segments of a polyline
----------------------------------	--

A line segment is defined by 2 Points.

An arc segment is defined with a Fillet function or with an Arc0 or Arc1 function and two Points on the ends of the arc.

For polylines that contain only straight line segments, the <list of arguments> consists of only 2D points, defined using Point(x,y) function.

**Format:**

```
Point(<cx>, <cy>)
```

<b>&lt;cx&gt;</b>	Defines the x coordinates of the point
-------------------	--

**<cy>** Defines the y coordinates of the point

**For example, a rectangle can be defined in the following way:**

```
rect = Polyline( // no end-of-line is used semicolon here
Point(0,0),      // since this function is on multiple lines
Point(W, 0),
Point(W,H),
Point(0, H),
Point(0,0) );
```

It should be noted that when a polyline's, first and last points are coincident, is called a closed polyline. This type of polyline bounds a certain area, and can be used for creating 3D objects.

Polylines with arc segments are defined by adding auxiliary functions *Arc0* and *Arc1* to the list of arguments. *Arc0* builds the circular arc clockwise, while *Arc1* builds the circular arc counterclockwise.

**Format:**

```
Arc0(<cx>,<cy>),
Arc1(<cx>,<cy>),
```

**<cx>** Defines the x coordinates of the arc center

**<cy>** Defines the y coordinates of the arc center

The start and end point of an arc are defined by the preceding and the following arguments.

*Arc0* and *Arc1* cannot be the first or last argument in the list of arguments. For a polyline that contains only one arc segment, the <list of arguments> consists of 2 Points defined with *Point(x,y)* function and an arc defined with either the *Arc0* or *Arc1* function.

**Example of arc0 and arc1 in a polyline:**

```
//Polyarc.ppm -- polyline with arcs
YSize=5;
XSize=6;
R = 1;
Path = Polyline(Point(0, R), // start at top of rounded lower left corner.
Point(0, YSize-R), // go to bottom of rounded top left corner.
Arc1(0, YSize, R), // make this corner a "cutout"
Point(R, YSize), // left side of top edge
Point(XSize-R, YSize),
Arc0(XSize-R, YSize-R, R), // make this corner a "fillet"
Point(XSize, YSize-R),
Point(XSize, R),
Arc0(XSize-R, R, R), // another fillet
Point(XSize-R, 0),
```



```

        Point(R, 0),
        Arc1(0, 0, R), // another cutout
        Point(0, R));
Output(Path);

```

Another method of creating an arc in a polyline is to use the auxiliary function Fillet, which “smooths” two linear segments that start and end in the preceding point, by adding an arc with the specified radius into the corner. This ensures smoothness at the junction points.

**Format:**

```
Fillet(<radius>);
```

**<radius>** Defines the radius of the fillet

**Example of fillets in a polyline:**

```

// polyfillet.ppm -- polyline with fillets
H = 5;
L = 10;
FR = 1;
p2 = Polyline(// Rectangle with rounded corners
Point(0,0), // lower left corner
Point(L,0), // lower right corner
Fillet(FR), // places fillet at bottom right
Point(L,H), // upper right corner
Fillet(FR), // places fillet at top right
Point(0,H), // upper left corner
Fillet(FR), // places fillet at top left
Point(0,0), // closes rectangle
Fillet(FR) // fillets start/end corner. Since this is a closed shape,
// no following Point function is needed.
);
Output(p2);

```

Fillet and Arcs can be used together in the same Polyline function.

**Example of Arcs and Fillet in a polyline:**

```

Poly1 = Polyline( // Rectangle with rounded corners
Point(0,0),
Point(W - r, 0), Arc1(W - r, r), Point(W, r),
Point(W, H - r), Arc1(W - r, H - r), Point(W - r ,H),
Point(0, H), Fillet(r),
Point(0,0), Fillet(r) );

```

## Functions for Creating 3D Entities from 2D Entities

You can use 2D entities as the basis for creating 3D objects.

### **Thickness**

The Thickness function creates a 3D entity based on the 2D entity by adding thickness. It also allows you to change the thickness property of the 3D object.

#### **Format:**

```
Thickness(<Object>, <value>);
```

<Object>	Defines the initial graphic object
<value>	Defines new value of Thickness

#### **Example of Thickness:**

```
RectA = Rectangle(2, 5);
RectThick = Thickness(RectA, 3);
```

#### **Example of Thickness Used to Create a Box Function:**

```
Input(x0,y0,z0,x1,y1,z1)
R = Rectangle(x1-x0, y1-y0, (x0+x1)/2, (y0+y1)/2);
T = Thickness(R, z1-z0);
Output(Move(T, 0, 0, z0));
```

#### **Another Example of Thickness:**

```
//thickrect.ppm -- draws a 2D rectangle and adds thickness
L = Parameter("Length", 4, LINEAR, Interval(0.1, 20));
W = Parameter("Width", 3, LINEAR, Interval(0.1, 20));
H = Parameter("Height", 1.5, LINEAR, Interval(0.1, 20));
Rect = Rectangle(L, W);
Box = Thickness(Rect, H);
Output(Box);
```

#### **An Example of Thickness with a Circle:**

```
// thickcircle.ppm -- draws a circle and adds thickness
Cylind=Thickness(Circle(1,2,2),2);
Output(Cylind);
```

#### **An Example of Changing Thickness:**

```
// thickcircle2.ppm -- draws a cylinder and changes thickness
Cylind=Thickness(Circle(1,2,2),2);
Cyl2 = Thickness(Cylind, 4); // changes the thickness of the first cylinder
Output(Cyl2);
```

## Sweep

The Sweep function creates a 3D object by extruding a specified profile along a path, defined by a 2D polyline or circle. The profile is defined by a closed 2D polyline or circle.

### Format:

```
Sweep(<profile>, <path>[, <rotation angle>]);
```

<b>&lt;profile&gt;</b>	This defines the profile using a 2D polyline
<b>&lt;path&gt;</b>	This defines the path, along which the profile is “dragged”; the path is defined by a 2D polyline <i><b>Note:</b> The plane of the path and the plane of the profile must not be parallel.</i>
<b>&lt;rotation angle&gt;</b>	This optional argument, defines the rotation angle of the profile relative the Z axis; by default, the argument is equal to zero

### Example of Sweep:

```
Poly1 = Polyline(
  Point(0,0),
  Point(1, 0),
  Point(1,2),
  Point(0, 2),
  Point(0,0) );
PolyProfile = RotateX(Poly1, 90); // the Rotate function will be explained
later
PolyPath = Polyline(
  Point(0,0),
  Point(10, 0),
  Point(10,10),
  Point(0, 10),
  Point(0,0) );
PolySweep = Sweep(PolyProfile, PolyPath);
Output(PolySweep);
```

### Another Example of Sweep

```
//sweep1.ppm
R = 2;
D = 5;
C1 = RotateX(Circle(R, D/2+R, 0),90); // profile
C2 = Circle(D/2, 0, 0); // path
Torus = Sweep(C1,C2);
Output(C1, C2, Torus); //C1 and C2 shown for reference
```

### An Extended Example of Sweep

```
//sweep2.ppm -- another sweep example
```

```

L = Parameter("Length", 5, LINEAR, Interval(0.005, 1000));
W = Parameter("Width", 3, LINEAR, Interval(0.005, 1000));
H = Parameter("Height", 1, LINEAR, Interval(0.1, 3));
FR = Parameter("Fillet Radius", 0.3, LINEAR, Interval(0.001,100));

p = Polyline(Point(0,0), Point(0,H), Point(-FR,H), Point(-FR,0), Point(0,0));
pla = RotateX(p,90,0,0);
p1 = Move(pla, 0, W/2, 0);
p2 = Polyline( Point(0,0), Point(0,W), Fillet(FR), Point(L,W), Fillet(FR),
Point(L,0), Fillet(FR), Point(0,0), Fillet(FR));

s = Sweep(p1, p2); Output(s);

```

## Functions for Creating 3D Entities Directly

3D object may also be created directly without reference to a 2D entity.

### ***Sphere***

The Sphere function is used to create a 3D sphere.

#### **Format:**

```
Sphere(<radius>[,<cx1>,<cy1>,<cz1>]);
```

<b>&lt;radius&gt;</b>	This value specifies the radius of the sphere
<b>&lt;cx1&gt;,&lt;cy1&gt;,&lt;cz1&gt;</b>	These are optional argument used to specify the x, y, z location of the sphere's center point. By default the values for these argument is zero

#### **Sphere Example:**

```
SR1 = Sphere(10,1,3,5.5);
```

#### **Another Sphere example:**

```

//sphere.ppm -- simple sphere example
R = Parameter("Radius", 2.5, LINEAR, Interval(0.01, 20));
cx = Parameter("CenterX", 0, LINEAR, Interval(-100, 100));
cy = Parameter("CenterY", 0, LINEAR, Interval(-100, 100));
cz = Parameter("CenterZ", 0, LINEAR, Interval(-100, 100));
S = Sphere(R, cx, cy, cz);
Output(S);

```

### ***Cone***

The Cone function is used to create a 3D cone.

#### **Format:**

```
Cone(<Height>,<baseradius>[,<topradius>]);
```

<b>&lt;Height&gt;</b>	This value specifies the height of the cone
<b>&lt;baseradius&gt;</b>	This value specifies the radius for the base of the cone
<b>&lt;topradius&gt;</b>	This optional argument specifies a radius for the top of the cone, creating a truncated cone. By default the value for this argument is zero

**Cone Example:**

```
CN1 = Cone(10,5,2);
```

**Another Cone Example**

```
//cone1.ppm -- a simple cone
R = Parameter("BaseRadius", 0.5, LINEAR, Interval(0.01, 10));
H = Parameter("Height", 3, LINEAR, Interval(0.05, 20));
Cone1 = Cone(H, R, 0);
Output(Cone1);
```

**Example of a Truncated Cone:**

```
//cone2.ppm -- a truncated cone
R1 = Parameter("BaseRadius", 0.5, LINEAR, Interval(0.01, 10));
R2 = Parameter("TopRadius", 0.1, LINEAR, Interval(0, 10));
H = Parameter("Height", 3, LINEAR, Interval(0.05, 20));
Cone2 = Cone(H, R1, R2);
Output(Cone2);
```

## Functions for Transforming Geometric Objects

This class of functions is used for moving, and rotating geometric objects. These transformations are related to the transformation of the coordinate system. As always functions create transformed objects, while original objects do not change.

### Move

The Move function is used to move (shift) graphic objects.

**Format:**

```
Move(<Object>, <dx>, <dy> , <dz>[,count]);
```

<b>&lt;Object&gt;</b>	Defines the original graphic object
<b>&lt;dx&gt;, &lt;dy&gt;, &lt;dz&gt;</b>	Defines value of movement along x, y and z axes, respectively
<b>&lt;count&gt;</b>	Defines the number of created objects, where each subsequent object is created by moving the preceding object; this argument is optional, with the default value of 1

**Example of Move:**

```
PolyProfile = Move(Poly1, 1, 3);
```

**Another Example:**

```
//move.ppm -- illustrates the Move function
RB = Parameter("BaseRadius", 2, LINEAR, Interval(0.1, 10));
RT = Parameter("TopRadius", 0.5, LINEAR, Interval(0, 10));
H = Parameter("Height", 4, LINEAR, Interval(0.1, 20));
con1 = Cone(H, RB, RT);
cx = Parameter("CenterX", 5, LINEAR, Interval(-10, 10));
cy = Parameter("CenterY", 0, LINEAR, Interval(-10, 10));
cz = Parameter("CenterZ", 0, LINEAR, Interval(-10, 10));
count = Parameter("Copies", 2, LINEAR, Interval(1, 10));
con2 = Move(con1, cx, cy, cz, count); // create count copies, offsetting each
by cx, cy, cz
Output(con1, con2);
```

**Rotate**

The RotateX, RotateY, RotateZ functions are used to rotate graphic objects around the X, Y and Z axes, respectively.

**Format:**

```
RotateX(<Object>, <rotation angle>[, <cy>, <cz>[, <count>]]);
RotateY(<Object>, <rotation angle>[, <cx>, <cz>[, <count>]]);
RotateZ(<Object>, <rotation angle>[, <cx>, <cy>[, <count>]]);
```

<b>&lt;Object&gt;</b>	Defines the original graphic object
<b>&lt;Rotation angle&gt;</b>	Defines the angle of rotation
<b>&lt;cx&gt;, &lt;cy&gt;, &lt;cz&gt;</b>	Sets an offset for the rotation axis relative to the X, Y and Z axes (in accordance with function names). These arguments are optional; however, only all three arguments can be omitted at once. Default value for each of <cx>, <cy>, <cz> is zero
<b>&lt;count&gt;</b>	Defines the number of created objects, where each subsequent object is created by transforming the preceding object; this argument is optional, with the default value of 1

**Example of Rotate:**

```
PolyProfile = RotateX(Poly1, 90);
```

**Another Example of Rotate:**

```
//rotate.ppm -- demonstrates the rotate functions
c1 = Circle(2, 10, 0); // create a circle
c2 = RotateX(c1, -90, 0, 0); // rotate the circle to lie in the XZ plane
c3 = Move(c2, 0, -0.05, 0); // move it back half the thickness
c4 = Thickness(c3, 0.1);
c5 = RotateZ(c4, 30, 0, 0, 11); //duplicate the circle by rotating about the
Z axis
c6 = Circle(2, 0, 10);
```

```

c7 = Move(c6, 0, 0, -0.05);
c8 = Thickness(c7, 0.1);
c9 = RotateX(c8, -30, 0, 0, 11);
c10 = Circle(2, 0, 0);
c11 = RotateZ(c10, -90, 0, 0);
c12 = Move(c11, 10, 0, -0.05);
c13 = Thickness(c12, 0.1);
c14 = RotateY(c13, 30, 0, 0, 11);
Output(c4, c5, c8, c9, c13, c14);

```

## Functions for Loading External Symbols as Elements

You can load non-parametric external symbols from external files to be a part of a parametric part. The files must be importable (supported) by the CAD system, such as \*.TCW, \*.DWG, \*.SKP

### StaticSymbol

The StaticSymbol function loads non-parametric symbols from external files. When the external symbol's filename is specified with no path information, the symbol is automatically assumed to reside in a sub-folder named Macro that is located in the ppm file's home folder.

#### Format:

```
StaticSymbol(<FileName>[ ,BlockName]);
```

<b>&lt;FileName&gt;</b>	Defines the file name with extension. If the extension is not specified, native file format will be used
<b>&lt;BlockName&gt;</b>	This is an optional argument, which indicates that only the block with the given name should be used as the symbol for loading, and the rest of the contents should be ignored; if the argument is not defined, the active drawing will be loaded as a symbol

#### Example of StaticSymbol:

```

//staticsym1.ppm -- loads an external file from the Macro sub-folder
S = StaticSymbol("ExternalSymbol.tcw");
Output(S); //static symbol from ExternalSymbol.tcw file is inserted on the
drawing

```

### Set(FolderList(...))

To create a list of files in a folder, Set(FolderList(...)) is typically used as the Parameter restriction.

#### Format:

```
<id> = FolderList(<path> <mask> = "*.ppm");
```

<b>&lt;path&gt;</b>	Defines the path to the folder from which the list of files will be created
<b>&lt;mask&gt;</b>	Defines the mask of file names and extensions

**Example of Set(FolderList(..)):**

```
// staticsym2.ppm -- loads an external symbol from a different folder than
Macro
DrawingName = Parameter("Drawing", "Drawing1", Set(FolderList("..\\..\\..\\Draw-
ings", "*.tcw"))); //quantity of "..\\..\\" (before folder Drawings) is equal to
quantity
//of steps over folder tree starting from the Macro sub-folder.
S0 = StaticSymbol("..\\..\\..\\Drawings\\"+DrawingName+".tcw");
//here a static symbol is loaded from a file with a tcw-extension,
// and a filename picked from the FolderList obtained via the DrawingName
parameter.
Output(S0);
```

When specifying a relative path, you must remember that the path is always assumed to start, not at the folder that contains the ppm file, but in a folder below that named "Macro". In the example above, assume for the moment that staticsym2.ppm is located in:

```
C:\Users\Me\Documents\MyCAD\PPM Documentation Samples
```

The path used in the FolderList path and the StaticSymbol path must then implicitly begin at

```
C:\Users\Me\Documents\MyCAD\PPM Documentation Samples\Macro
```

The external symbol is being loaded from:

```
C:\Users\Me\Documents\MyCAD\Drawings
```

That means the script must navigate up three directories to the MyCAD folder, then back down one level to the Drawings folder, so the correct relative path is:

```
..\\..\\..\\Drawings
```

Another example, which loads a specific .tcw file from the Drawings folder:

```
//staticsym3.ppm -- loads a specific file from a different folder
S = StaticSymbol("..\\..\\..\\Drawings\\3DSliceTest.tcw");
//only loads the specific file 3DSliceTest.tcw.
//Remember that the relative path is still rooted in the Macro subfolder.
Output(S);
```

A parametric part (a file with a \*.ppm extension) can be loaded by calling the name of the parametric file as if it were a function, whose arguments are the parameters of the part to be loaded, in the order in which they are described in the file. Refer to “Creating Custom Functions” below for more details on this process.



## Functions for 3D Boolean Operations

Functions of this class are used to perform Boolean operations on 3D geometric objects.

### ***BooleanUnion***

The BooleanUnion function creates an object by adding the specified objects together.

**Format:**

```
BooleanUnion(<Object>, <Object>, ...);
```

<b>&lt;Object&gt;</b>	Defines an object to be used in the Boolean operation. There must be at least two objects
-----------------------	---

#### **Example of BooleanUnion:**

```
S1 = Sphere(5);
S2 = Sphere(5,5,5);
S3 = Sphere(5,5,-5);
S4 = Sphere(5,-5,5);
S5 = Sphere(5,-5,-5);
S6 = BooleanUnion(S1,S2,S3,S4,S5);
Output(S6);
```

#### **Another Example:**

```
R = Parameter("Radius", 8, LINEAR, Interval(0.001, 1000));
s = Sphere(R);
c = Circle(R/3);
c1 = Thickness(c, R*2);
c2 = Move(c1, 0, 0, R); //Cylinder
s1 = BooleanUnion(s, c2); //Sphere with cylinder
Output(s1);
```

### ***BooleanSubtraction***

The BooleanSubtract function creates an object by subtracting the secondary objects from the primary object.

**Format:**

```
BooleanSubtract(<PrimaryObject>, <SecondaryObject>, ...);
```

<b>&lt;PrimaryObject&gt;</b>	Defines an object to be used in the Boolean operation. There is only one primary object
<b>&lt;SecondaryObject&gt;</b>	Defines a secondary object to be subtracted from the primary object There must be at least one or more secondary objects

#### **Example of BooleanSubtract:**

```
S1 = Sphere(5);
S2 = Sphere(5,5,5);
S3 = Sphere(5,5,-5);
S4 = Sphere(5,-5,5);
```

```
S5 = Sphere(5,-5,-5);
S6 = BooleanSubtract(S1,S2,S3,S4,S5);
Output(S6);
```

#### Another Example of BooleanSubtract:

```
R = Parameter("Radius", 8, LINEAR, Interval(0.001, 1000));
s = Sphere(R);
c = Circle(R/3);
c1 = Thickness(c, R*2);
c2 = Move(c1, 0, 0, -R); //Cylinder
s1 = BooleanSubtract(s, c2); //Sphere with hole
Output(s1);
```

### BooleanIntersect

The BooleanIntersect function creates an object derived from the intersection of the primary and secondary objects.

#### Format:

```
BooleanIntersect(<Object>, <Object>)
```

<Object>

Defines an object to be used in the Boolean operation. There must only two objects

#### Example of BooleanIntersect:

```
S1 = Sphere(5);
S2 = Sphere(5,5,5);
S3 = Sphere(5,5,-5);
S4 = Sphere(5,-5,5);
S5 = Sphere(5,-5,-5);
S6 = BooleanIntersect(S1,S2);
Output(S6);
```

## Functions for Modifying 3D Objects

Several functions are available to modify the geometry of 3D objects.

### Fillet Edges

The Fillet Edges function allows rounding one or multiple edges of 3D object.

#### Format:

```
G3Fillet(<Object>,<Edges>, <Radii>);
```

<Object>

Defines the 3D object whose edges are to be rounded

<Edges>	Defines the edge or multiple edges, which are to be filleted. Each edge is defined by Point(xc,yc,zc) or Array of Points.
	Point(xc,yc,zc) is the middle point of an edge to be filleted (for example in the TurboCAD “Fillet Edges” operation, this point is marked with a blue square). Array of Points defines a set of edges to be filleted.
<Radiuses>	Defines the Fillet radiuses. Fillet radiuses are set by Array function. For a single edge the Array contains pair of values, for multiple edges - multiple pairs of values.

**Fillet Edges Example:**

```
Array(Point(x1,y1,z1), Point(x2,y2,z2), Point(x3,y3,z3)); //defines 3 edges
for filleting
//Point(x1,y1,z1), Point(x2,y2,z2), Point(x3,y3,z3); - 3 middle points on 3
edges to be filleted
```

**Another Example:**

```
Array(r1, r2)- //array of radius values for rounding the selected edge. It
defines rounding //radiuses for 2 ends of the selected edge.
//r1 - start radius of fillet
//r2 - end radius of fillet.
```

**Example of Filleting 1 Edge:**

```
G3Fillet(PartA,Point(xc,yc,zc), Array(r1, r2)); //where Point(xc,yc,zc) -
middle of the edge.
```

**Another Example:**

```
Door= G3Fillet(Door0, Point(0, -1, (Height-FHeight-4-3/4)/2), Array(1, 1));
For example (fillet of 1 edge of the box):
x = Parameter("size", 5, LINEAR, GreaterThan(0));
r1 = Parameter("r1", 1, LINEAR, GreaterThan(0));
b0 = Box(0, 0, 0, x, x, x);
b1 = G3Fillet(b0, Point(x/2, 0, 0), Array(r1, r1*2));
Output(b1);
```

**Example of Filleting 4 Edges of a Box:**

```
L = Parameter("Length", 5, LINEAR);
W = Parameter("Width", 3, LINEAR);
H = Parameter("Height", 1, LINEAR);
R = Parameter("Radius",0.5);
g0 = Box(0,0,0,L,W,H);
g1 = G3Fillet(g0, Array(Point(L/2, 0, 0), Point(0, W/2, 0),
                        Point(L/2, W, 0), Point(L, W/2, 0)),
                Array(R, R, R, R, R, R, R, R));
```

```
Output(g1);
```

## Chamfer Edges

The Chamfer Edges function allows chamfering any edge or multiple edges of 3D object.

### Format:

```
G3Chamfer(<Object>, <Edges>, <Offsets>);
```

<b>&lt;Object&gt;</b>	Defines the 3D object whose edges are to be chamfered
<b>&lt;Edges&gt;</b>	Defines the edge or multiple edges, which are to be filleted. Each edge is defined by Point(xc,yc,zc) or Array of Points.  Point(xc,yc,zc) is the middle point of an edge to be filleted (for example in the TurboCAD "ChamferEdges" operation, this point is marked with a blue square). Array of Points defines a set of edges to be chamfered.
<b>&lt;Radiuses&gt;</b>	Defines the Chamfer distances. These are set by Array function. For a single edge the Array contains a pair of distance values, for multiple edges - multiple pairs of distance values.

### A Chamfer Example:

```
Array(d1, d2)- //array of 2 offset values at the ends of an edge.
```

### Another Example:

```
Door= G3Chamfer(Door0, Point(0, -1, (Height-FHeight-4-3/4)/2), Array(1, 1));

//Here Door0 -is the object whose edge is to be chamfered.
//Point(0, -1, (Height-FHeight-4-3/4)/2) - indicates this edge.
//Array(1, 1) sets 2 chamfer distances
```

### Another Example:

```
x = Parameter("size", 5, LINEAR, GreaterThan(0));
r1 = Parameter("r1", 1, LINEAR, GreaterThan(0));
b0 = Box(0, 0, 0, x, x, x);
b2 = G3Chamfer(b0, Point(x/2, x, x), Array(r1, r1+r1));
Output(b2);
```

## G3Offset

The G3Offset function extends a solid face inward or outward.

### Format:

```
G3Offset(<Object>, <Face>, <Offsets>);
```

<b>&lt;Object&gt;</b>	Defines the 3D object whose edges are to be extended
<b>&lt;Face&gt;</b>	Defines the face, which is to be extended. The Face is defined by a Point(x,y,z) belonging to this face

**<Offsets>** Defines the offset distance. A positive value will offset the face outward, and a negative value will offset inward

#### Offset Example:

```
G3Offset(PartA, Point(xf, yf, zf), dist);
```

Where:

PartA — is the 3D object whose faces are to be offset

Point(xf, yf, zf) — is a point for selecting the face to be offset

dist — is the value of face offset

#### Another Example:

```
x = Parameter("size", 5, LINEAR, GreaterThan(0));
r1 = Parameter("r1", 1, LINEAR, GreaterThan(0));
b0 = Box(0, 0, 0, x, x, x);
b3 = G3Offset(b0, Point(x,x/2,x/2), r1/2);
Output(b3);
```

### G3Shell

The G3Shell function allows shelling the shape of solid object, leaving the selected face open. It creates a shell of a specified thickness from a single solid object. The new faces are created by offsetting existing faces inside or outside.

#### Format:

```
G3Shell(<Object>, <Face>, <Thickness>);
```

**<Object>** Defines the 3D object whose edges are to be shelled

**<Face>** Defines the face that should remain open. It is defined by the Point(xc,yc,zc) function which describes a point belonging to this face

**<Thickness>** Defines the shell thickness. A positive value creates an outward shell, and a negative value creates an inward shell

#### Shell Example:

```
G3Shell(PartA, Point(xf, yf, zf), thickn);
```

Where:

Part3 — selects the object which is to be shelled

Point(xf, yf, zf) — is the point on the face, which should remain open

thickn — is the shell thickness

#### Another Example:

```
L = Parameter("Length", 5, LINEAR);
W = Parameter("Width", 3, LINEAR);
```

```

H = Parameter("Height", 1, LINEAR);
T = Parameter("Thickness", 0.2, LINEAR);
g0 = Box(0,0,0,L,W,H);
g1 = G3Shell(g0, Point(L/2, W/2, H), T);
Output(g1);
//After inserting a shelled object in the drawing, shell thickness can be
edited in the Selection Info palette (as well as Length, Width and Height
parameters)

```

## G3Bend

The G3Bend function is used for bending 3D objects.

### Format:

```
G3Bend(<Object>, <Line>, <Angle>, <Radius>, <Depth> );
```

<Object>	Defines the 3D object which is to be bent
<Line>	Defines a line about which the solid object will be bent. It is defined by 2 Points: Point(x1, y1, z1), Point(x2, y2, z2). The line must lie on the solid face selected for bending.
<Angle>	Defines the bending angle. The angle is measured from the plane of the bent face.
<Radius>	Defines the bending radius
<Depth>	Defines the Neutral Depth to set the distance into the depth of material along which there will be no tension or compression

### Bend Example:

```
G3Bend(Part3, Point(x1, y1, z1), Point(x2, y2, z2), Angle, R, 0);
```

### Another Example:

```

P1=Thickness(Rectangle(10,20),3);
B0 = G3Bend(P1, Point(3, 3, 0),
            Point(3,8,0), 90, 2, 0);
Output(B0);

```

## Setting and Changing Object Properties

The SetProperty function is used to set the properties of objects.

### Format:

```

SetProperties(<Object>, <PropertyName> = PropertyValue, <PropertyName> =
PropertyValue, ...);

```

<Object>	Defines the object to be used as the base for the new object with set properties
----------	--

<b>&lt;PropertyName&gt;</b>	Defines the name of the property to be set. The name should be surrounded with quotation marks
<b>&lt;PropertyValue&gt;</b>	Defines the value to be assigned to the property

**Example of SetPropertyies:**

```
BlueRect=Rectangle(10,5);
RedRect = SetPropertyies(BlueRect, "PenColor" = 0xff, "PenWidth" = 0.2);
Output(RedRect);
```

**Another Example:**

```
Side2M = SetPropertyies(Side2, "Material" = "Wood\Pine", "PenColor" = 0xff);
```

**Another Example:**

```
PL1 = SetPropertyies(PL0, "Brush" = "SOLID");
```

**Another Example:**

```
SetPlastic = ("Material" = "Plastics\Plain white");
BoxMaterial = SetPropertyies(MyBox,SetPlastic);
```

In the Parametric Part manager there is a special tool to choose the required value for such properties as Material, Pen Color and Brush Style. To activate it, right-click on the property name. This will open the Local Menu either for Material table or PenColor table or BrushStyle table. The appropriate table will appear where the desired value can be chosen.

## Nesting Functions

Functions can be nested within a single expression to optimize scripting efficiency.

**For Example:**

```
BF = BooleanSub-
tract(B1,Move(RotateZ(RotateY(Box(-5,-5,-5,5,5,5),45),45),-1,-1,-1));
```

**Example Used in a Small Script:**

```
B1 = Box(0,0,0,10,10,10);
BF = BooleanSub-
tract(B1,Move(RotateZ(RotateY(Box(-5,-5,-5,5,5,5),45),45),-1,-1,-1));
Output(BF);
```

## Functions for Creating Text

### ***Text***

The Text function defines the text string itself and its characteristics, including fonts, style, effects, etc. Acceptable font values are dependent upon those installed on your machine.

**Format:**

*Text(<Text object>, <Text Font>, <Text Style>);*

<b>&lt;Text object&gt;</b>	Defines the text string. Text string can be specified either directly here (with quotation marks) or via an identifier of text object
<b>&lt;Text Font&gt;</b>	Defines the text font
<b>&lt;Text Style&gt;</b>	Defines the text style

**Example:**

```
bsb = Text("BS(b)", Tfont, Tstyle);
```

### ***TextFont***

The TextFont function sets the text font, size, and the angle of text line location.

**Format:**

*TextFont(<mode>, <Height>, <Angle>, <font>);*

<b>&lt;mode&gt;</b>	Defines the mode of the text: Standard (when mode=0) or Scalable (when mode=1 or any other value different from 0)
<b>&lt;Height&gt;</b>	Defines the text font size
<b>&lt;Angle&gt;</b>	Defines the angle of text line
<b>&lt;font&gt;</b>	Defines the text font

**Example:**

```
Tfont = TextFont(0,2, 45, "Arial");
```

Where:

- 0 — means that text is Standard
- 2 — text height
- 45 — text line is located at 45 degrees
- Arial — font



## TextStyle

The TextStyle function sets the text style including justification, text effects and styles.

**Format:**

```
TextStyle(<list of characteristics>);
```

**<list of characteristics>**

Defines the text characteristics separated with commas.

The following values of characteristics are allowed:

For Justification:

LEFT, CENTER, RIGHT, TOP, MIDDLE, BASELINE, BOTTOM

For Text Effects:

BOX, UNDERLINE, STRIKETHROUGH, ALLCAPS

For Style:

BOLD, ITALIC

**Example:**

```
Tstyle = TextStyle(LEFT, TOP, UNDERLINE);
```

**Another Example:**

```
//Standard text of Times New Roman font with 5in of font size,
//with Left,Top justification, with TextBox effect, Bold, Italic, at 45
degrees of Angle
ht=5;
font_name = "Times New Roman";
Tfont = TextFont(0, ht, 45, font_name);
Tstyle = TextStyle(LEFT, TOP, BOX,BOLD, ITALIC);
bsb = Text("BS(b)", Tfont, Tstyle);
Output(bsb);
```

## Auxiliary Functions

### Extents

The ExtentsX1, ExtentsX2, ExtentsY1, ExtentsY2, ExtentsZ1 and ExtentsZ2 functions are used to calculate the extents of graphic objects.

**Format:**

```
ExtentsX1(<Object>);
ExtentsX2(<Object>);
ExtentsY1(<Object>);
ExtentsY2(<Object>);
ExtentsZ1(<Object>);
```

```
ExtentsZ2(<Object>);
```

**<Object>** Defines the object to be used

The presence of X, Y or Z characters in the function name determines axis along which the extents will be calculated.  
1 or 2 index—indicates whether minimum or maximum value should be calculated.

#### Example of Extents:

```
xmin = ExtentsX1(PartA);
xmax = ExtentsX2(PartA);
ymin = ExtentsY1(PartA);
ymax = ExtentsY2(PartA);
zmin = ExtentsZ1(PartA);
zmax = ExtentsZ2(PartA);
P1 = Box(xmin, ymin, zmin, xmax, ymax, zmax);
```

#### Another Example of Extents:

```
A0=Thickness(Rectangle(H-3/4,D), 3/4);
A1=RotateY(A0,90);
xmin = ExtentsX1(A1);
xmax = ExtentsX2(A1);
ymin = ExtentsY1(A1);
ymax = ExtentsY2(A1);
zmin = ExtentsZ1(A1);
zmax = ExtentsZ2(A1);
P1 = Box(xmin, ymin, zmin, xmax, ymin+3, zmin+4);
```

## ParameterPoint

The ParameterPoint function defines a parametric point with number and coordinates.

#### Format:

```
ParameterPoint (<N>,<xc>,<yc>,<zc>);
```

**<N>** Defines the number of the parametric point

**<xc>,<yc>,<zc>** Defines the coordinates of parametric point

#### Example of ParameterPoint:

```
P0 = ParameterPoint(0, 1, -1, 0);
P1 = ParameterPoint(1, 0, 0, 0);
```

## PointX, PointY, PointZ functions

The PointX, PointY, PointZ are used to calculate the coordinates of parametrical point. The PointX function calculates X-coordinate of parametrical point. The PointY function calculates Y-coordinate of parametrical point. The PointZ function calculates Z-coordinate of parametrical point.

### Format:

```
PointX (<point>);
PointY(<point>);
PointZ(<point>);
```

**<point>** Defines the parametrical point from which the X or Y or Z coordinate will be extracted

### Examples of Point:

```
x0 = PointX(P0); // x0=1 for P0 = ParameterPoint(0, 1, -1, 0);
y1 = PointY(P1); //y1=0 for P1 = ParameterPoint(1, 0, 0, 0);
z1 = PointZ(P1); //z1=0 for P1 = ParameterPoint(1, 0, 0);
```

## Special functions and operators

### IF

The IF function allows various actions to be performed depending upon whether the specified condition is fulfilled or not fulfilled. It plays the role of a conditional operator, and can be used to create branches in the logic of building a parametric part.

### Format:

```
IF(<Condition>, <ExprOnTRUE>, <ExprOnFALSE>);
```

**<Condition>** Defines the condition under test using the following comparison operations:

```
== (equal)
< (less than)
> (greater than)
<= (not greater than)
>= (not less than)
```

**<ExprOnTRUE>** Defines the value of the IF function when the value of <Condition> is TRUE;

**<ExprOnFALSE>** Defines the value of the IF function when the value of <Condition> is FALSE;

### IF Example:

```
A = IF(L >= H, Rectangle(L, H), Rectangle(H, L));
//Regardless of the specified size of L and H, the created rectangle A will be
positioned //horizontally (the longer side will be along the X axis).

/* In this example "Rectangle(L, H)" is the TRUE result and "Rectangle(H, L"
is the FALSE result. */
```

**Another Example:**

```
Tstyle = IF(dir > 0, TextStyle(MIDDLE, RIGHT), TextStyle(MIDDLE, LEFT));
//Regardless of the specified size of dir, Text Style will be specified with
Right or Left justification.
```

**UNITS**

The UNITS function defines the units that will be used in the script. It defines the System, Space Units and Scale of dimensions used while creating objects. This function allows loading parts correctly in drawings with different specified units.

**Format:**

```
Units(<N>[<units of dimension>]);
```

<N>	Defines object scale
<units of dimension>	Defines the units in the English or Metric systems

**For Example:**

```
Units(1[in]); // means that default unit of drawing is inches
Units(1[mm]); // means that default unit of drawing is millimeters
```

Units (1[in]) — this means that the main units of measurement are inches. It is the default unit of the script. All geometrical values are dimensioned in ‘inches’ without any mention of units.

It is possible to use other units for some particular values even when the entire drawing is created with the default unit. In order to use millimeters for particular values while inches are default units, you can explicitly declare the desired unit for these values.

For example, you can use value M=5[mm]; and Units(1[in]) in the same script. It means that only M value is measured in mm while all others are measured in inches.

Moreover, this function allows for scaling the created objects down (when N<1) or up (when N>1).

**For Example:**

```
Units(2[in]); // created object is scaled up 2 times compared with the case of
Units(1[in]);
Units(0.5[in]); // created object is scaled ½ as large as compared with the
case of Units(1[in]);
```

**RefPoint**

The RefPoint function sets the location of the Reference Point for the parametric part. When the Reference Point is one of the output values of a script, it is inserted in the drawing along with the part. This enables precise insertion of the parametric object into the drawing.

**Format:**

```
RefPoint(<Point>);
```

**<Point>** Defines the (x,y,z) coordinates for location of Reference Point

**For Example:**

```
xArrow = PointX(P0);
yArrow = PointY(P0);
rf = RefPoint(xArrow, yArrow, 0); //-> RefPoint is placed on the point (xAr-
row,yArrow, 0)
Output(rf);
```

## ***Input and Output***

The Input and Output functions are used for inputting initial values or objects into the script and outputting result objects from the script.

**Format:**

```
Input(<list of variable identifiers, separated with commas>);
Output(<list of variable identifiers, separated with commas>);
```

**<list of variable identifiers, separated with commas>** Defines the list of variables or objects for input or a list of results for output

**For Example:**

```
Input(H, W, D, A, Dis);
Output(SideA_L,Bottom_B,Back_I, Facel, FalseDl, E1,E2,E3,E4, Nl, Tl, Door,
FF,
SideA_R);
```

**Example of the Output with Conditional Output:**

```
Sw = Parameter("Switch", 1, CHECKBOX);
Pl = Thickness(Rectangle(5,5), 3);
S1= Thickness(Circle(2.5),4);
Output(IF(Sw,Pl,S1));
//Here is either cylinder or box inserted on the drawing
//depending on checkbox Sw value
```

## ***min and max***

The min and max functions are used for choosing the minimum or maximum values within a set of values.

**Format:**

```
min(<set of values>);
max(<set of values>);
```

**<set of values>** Defines the set of numerical values, identifiers of variables or Array of variables

**For Example:**

```
r=min(2,5,1,7,9);//r=1
R=max(2,5,1,7,9);//R=9
```

**For Example:**

```
A=2; B=5; C=1; D=7; E=9;
A1=2; B1=5; C1=1; D1=7; E1=9;
r=min(A,B,C,D,E);//r=1
R=max(A1,B1,C1,D1,E1);//R=9
```

**Example of using Array of Values:**

```
A=2; B=5; C=1; D=7; E=9;
r=min(Array(A,B,C,D,E));//r=1
```

---

*Note: A Group of objects cannot be used as argument of these functions, because a Group is a collection of graphic objects, rather than a collection of numbers.*

---

## Mod

The Mod function is used for finding the remainder of the integer division. For example, Mod(5,4) is 1, because  $5/4 = 1$ , with a remainder of 1. Mod(7,4) is 3, because  $7/4 = 1$ , with a remainder of 3. Mod(7,3) = 1, because  $7/3 = 2$ , remainder 1.

---

*Note: The Mod function is often used to determine if a number is odd or even, because  $\text{Mod}(\text{AnyOddNumber}, 2) = 1$ , while  $\text{Mod}(\text{AnyEvenNumber}, 2) = 0$ .*

---

**Format:**

```
Mod(<value1, value2>);
```

**<value1 >** Defines the expression or identifier that represents the dividend

**<value2>** Defines the expression or identifier that represents the divisor

**For Example:**

```
A = 7;
B = 4;
C = Rectangle(A, Mod(A,B));
Output(C);
```

## Div

The Div function is used to perform division.

**Format:**

```
Div(<value1>,<value2>);
```

<b>&lt;value1&gt;</b>	Defines the dividend
<b>&lt;value2&gt;</b>	Defines the divisor

**For Example:**

```
A=7;
B=3;
result1 = A/B;
result2 = Div(A, B);
rect = Rectangle(result1, result2)
Output(rect);
```

**Additional Math Functions****sqrt**

Calculates the square root of a specified number

```
P = sqrt(b);
```

**asin**

Calculates the arcsine. Returns the angle in radians

```
P = asin(0.5);
```

**acos**

Calculates the arcsine. Returns the angle in radians

```
P = acos(0.5);
```

**Array**

The Array function defines an array of values, or an array of Points, by directly listing the elements of the array. In other words the Array function collects geometric objects or values into an Array object.

**Format:**

```
Array(<list of objects>)
```

**<list of objects>**

list of numerical values or geometric objects

An *<object>* can be represented by either a value, or the *<identifier>* of a value, or by a Point(x,y,z) function.

**For Example:**

```
Array(Point(L/2, 0, 0), Point(0, W/2, 0), Point(L/2, W, 0), Point(L, W/2, 0))
// It is the array of points defining the edges for G3Fillet.

Array(R, R, R, R, R, R, R, R, R)
//It is the array of radius values for filleting the array of edges.
```

**Another Example: Can**

```
txt = Parameter("text", "Simple text example", TEXT);
a = Array(TextFont(0,10,"Arial"), TextStyle(CENTER, MIDDLE, ITALIC));
//Array of 2 items: TextFont and TextStyle)
s0 = Text(txt, a);
Output(s0);
```

**Group**

The Group function collects multiple graphic objects into a group and assigns an identifier name to the result. It allows the script to work with multiple objects as if they were a single object. Also a Group can be the output value of a script. Groups of objects can take part in different operations: Move, Rotate, etc.

**Format:**

*Group (<list of objects>);*

**<list of objects>**

Defines the list of graphic objects, separated with commas. The <object> may be any graphic objects

**For Example:**

```
bse = Group(bse_below, bse_above); //group of 2 graphic objects
Br2 = Group(Br0, Br1);
```

**For Example:**

```
Bx = Group(Move(BxL, -Dis*1.5), Move(BxR, Dis*1.5));
ShelfFBx = BooleanSubtract(ShelfF, Bx);
Output(ShelfFBx, Bx);_
```

**Special Functions without Parameters****PI**

The PI function calculates the value of Pi = 3.14159...

**Creating custom functions**

When scripts of the same type are created, which describe a particular class of parametric parts, it can be convenient to have the sequence of repeated actions as a separate specialized function. To achieve this, the repeated actions can be put into a separate <name>.ppm file.

In this case, all input variables should be listed in the Input operator:

**Format:**

*Input(<list of variable identifiers, separated with commas>);*

**For Example:**

```
Input(x0,y0,z0,x1,y1,z1);
```

The Output operator should also be defined.



**Format:**

```
Output(<list of variable identifiers, separated with commas>);
```

A custom function created in this manner must be placed in a Macro folder, which is always located inside the folder of the calling script. When the custom function is used, the script's file name (without the .ppm extension) is used just as if it was a built-in function.

**Format:**

```
<file name>(<list of input parameters>)
```

Below is an example of a custom function. The file box.ppm can be found in the PPM Documentation Samples/Macro folder:

```
// box.ppm -- defines a custom
Box function.
// The custom function is called
in this way:
//      B = Box(Xmin, Ymin, Zmin,
Xmax, Ymax, Zmax);
// The function creates a 3D box
with given min/max values
Input(x0,y0,z0,x1,y1,z1);
R = Rectangle(x1-x0, y1-y0,      //
Rectangle with Xmin = x0, Xmax=
x1
(x0+x1)/2,
```

```
(y0+y1)/2); // Ymin = y0, Ymax =
y1
T = Thickness(R, z1-z0); //
depth = Zmax - Zmin
Output(Move(T, 0, 0, z0)); //
move result along z to Zmin
```

The script below is box\_blend.ppm, which calls the custom function box.ppm

```
//box_blend.ppm uses the custom
Box.ppm function in the Macro
folder.
x = Parameter("size", 5, LINEAR,
GreaterThan(0));
r1 = Parameter("r1", 0.5, LINEAR,
GreaterThan(0));
b0 = Box(0, 0, 0, x, x, x);
b1 = G3Fillet(b0, Point(x/2, 0,
0), Array(r1, r1*2));
Output(b1);
```

File location is crucial when using parametric scripts as custom functions. In the example above, if blend\_box.ppm lies in the folder D:/Symbols, then it can only find the box.ppm script if box.ppm is located in the folder D:/Symbols/Macro.

# Parametric Parts Reserved Word List

PI	LINEAR	TEXT
ANGULAR	MATERIAL	FONT
COLOR	CHECKBOX	ITALIC
BOLD	UNDERLINE	BOX
ALLCAPS	STRICKETHROUGH	TOP
MIDDLE	BOTTOM	BASELINE
LEFT	CENTER	RIGHT
Call	Array	+
-	*	Div

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atan	min	max
**	=	==
!=	<	>
<=	>=	&
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