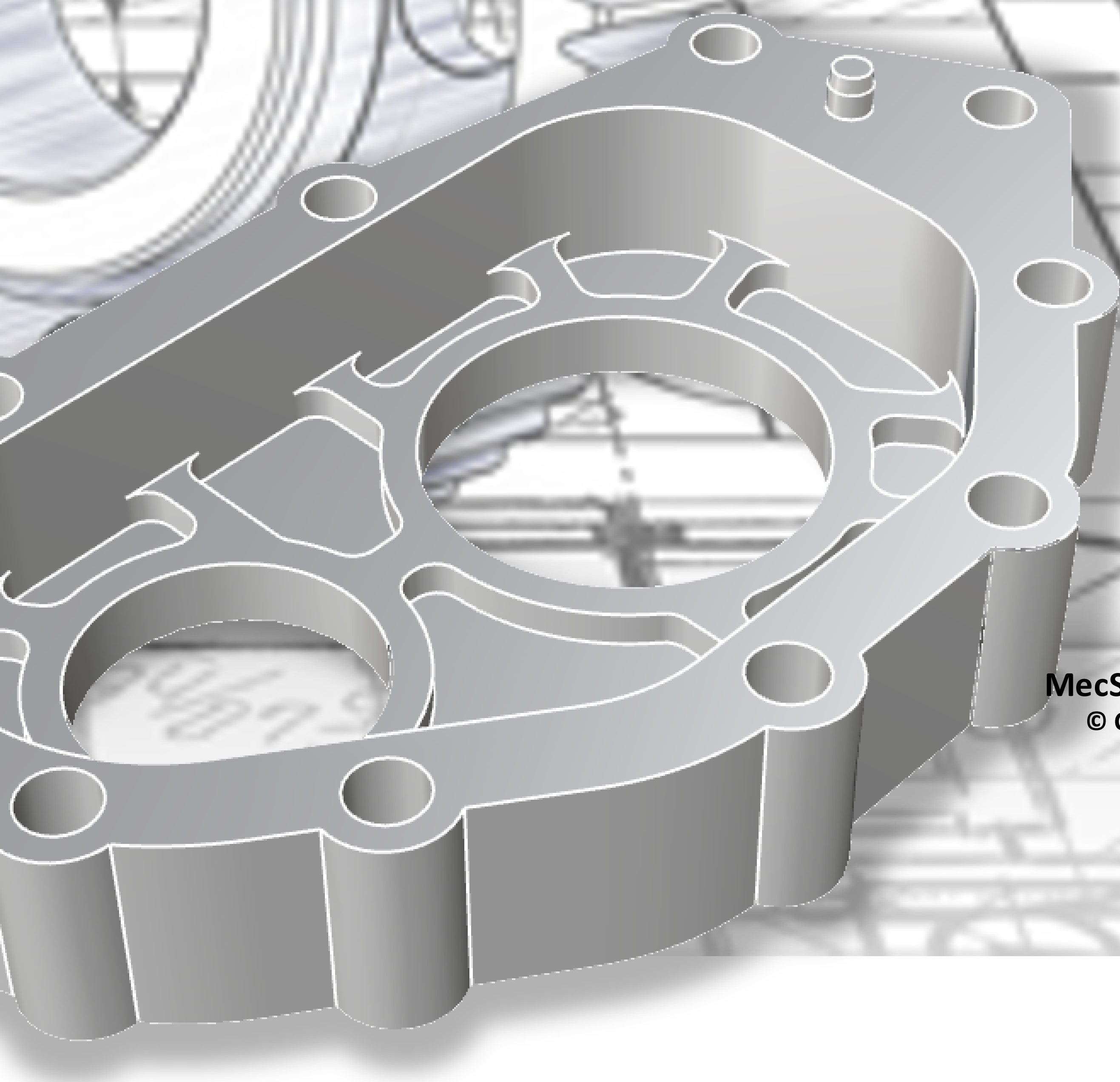
VisualCAD Companion 2026

Thursday, March 12, 2026



MecSoft Corporation

Table of Contents

Welcome to VisualCAD!

VisualCAD 2026

Prefer Printed Documentation? Click Here!

What's New | Quick Start Play List

VisualCAD is a CAD platform that can be used to design a part file. A detailed description of the product features, User Interface, File Translators, CAD Features, Selection and Transformation tools is described in the sections below.

Related Topics

The VisualCAD Screen Layout

VisualCAD Features Overview

Default Shortcut Keys

Customize VisualCAD Display

1.1 Features Overview

Here is an overview of the features you will find in VisualCAD:

General Features

- Ribbon-based menu interface
- Multiple viewport layouts.
- Create, Split, Save, Load and Delete Viewports
- Large display icons for context sensitive toolbars
- Rearrange and dock multiple toolbars inside each other
- (lockable Toolbars - Command Bar, Layer Manager, Property Page, Plug-in browsers - MOPS Browser, Machining Objects Browser)
- Auto Hide control for all lockable toolbars
- Print Preview and File > Print

- File > Open ... (Browse for VisualCAD part files)
- Translators include:

  VisualCAD Part Files (*.vcp)
  VisualMILL Part Files (*.vmp)
  VisualTurn Part Files (*.vct)
  3D Studio Files (*.3ds)
  Adobe Illustrator Files (*.ai)
  AMF Files (*.amf)
  AutoCAD (*.dxf;*.dwg)
  IGES Files (*.igs;*.iges)
  LightWave Files (*.lwo)
  MecSoft Region Files (*.mrg)
  OBJ Files (*.obj)
  Parasolid Files (*.x_t;*.x_b)
  Point Cloud Files (*.cvs;*.txt;*.asc)
  RAW Triangle Files (*.raw)
  RHINO 3DM Files (*.3dm)
  SAT Files (*.sat)
  SketchUP Files (*.skp)
  SLC Files (*.slc)
  STEP Files (*.stp;*.step)
  Stereo-lithography files (*.sla;*.stl)
  Universal 3D Files (*.u3d)
  VRML Files (*.wrl)
- Save and load C-planes
- Construction Plane (C-Plane) tools - C-Plane to a 3D Face, C-Plane by X-axis, WCS and more.
- Supports 3D mouse devices from 3Dconnexion
- Auto file save with an option to set the time interval for auto save
- Recovery of file in the event of application crash
- Dynamic view rotates with animation
- Save Viewport layout in the part file
- Assign hot keys for CAD tools on ribbon bar
- Crash Recovery support

Selection Tools

- Selections made easy. Users can select by dragging a window across the geometry in any of the viewports
- Select by Layer, Color, Type and Name

- Invert Selection
- Select Last Created and Previous
- Select by geometry types (Points, Lines/Polylines, Arcs/Circles, Polycurves, Dimensions, Surfaces)
- Rectangular Select
- Selection Mask – To select only certain types of geometries from the viewport
- Selection command to select duplicate objects

Transformation Tools

- Dynamic Move, Rotate
- Array Geometry - Rectangular and Polar
- Scaling part geometry on units change
- Graphical transformation of objects using Graphical Manipulator
- Allow graphical scaling of geometry.
- Scaling using fit to rectangle

CAD Tools

- Coordinate input for geometry creation in World and Construction planes
- Create Points, Lines, Arcs, Curves and curve editing tools
- Point cloud objects
- Create NURBS curves
- Create Text and Text on a Curve
- Curve creation visual aids
- Curve editing tools
- Surface creation tools.
- Surface Edit Tools (Trim, Project curves to surface, Wrap, Reverse Normal, Unify Normals, Explode)
- Mesh creation Tools (box, sphere, cylinder, cone, tubes, extrude, revolve)
- Mesh Boolean Tools (Unite, Subtract and Intersect)
- Solid creation tools (box, sphere, cylinder cone, torus, extrude, revolve, rectangular/circular tube)
- Solid editing tools (Unify Normals, Explode, Stitch)

- Curve extraction tools – Flat area regions, Create Surface Boundary and section curves.
- Dimensioning tools (Horizontal, Vertical, Oblique, Radius, Diameter, Angular, Leader and Annotation text)
- Delete holes/cap holes
- Tool to detect open loops in curve
- Multiple line text input in create Text dialog
- Spell check in text edit box
- Project to C-plane
- Auto Fillet
- Creation of boundary curve when a topologically connected set of surfaces are selected
- Extend Curve using a distance
- Arc fits on a poly-line
- Corner rounding of a poly-line
- Create silhouette curve around a part or selected geometries parallel to the c-plane
- Lock layers in layer manager
- Copy one layer to another including all geometry
- Reorder layers
- Hide and un hide objects using Ctrl H and Ctrl Alt H
- Lock and unlock objects using Ctrl L and Ctrl Alt L

Related Topics

Welcome to VisualCAD

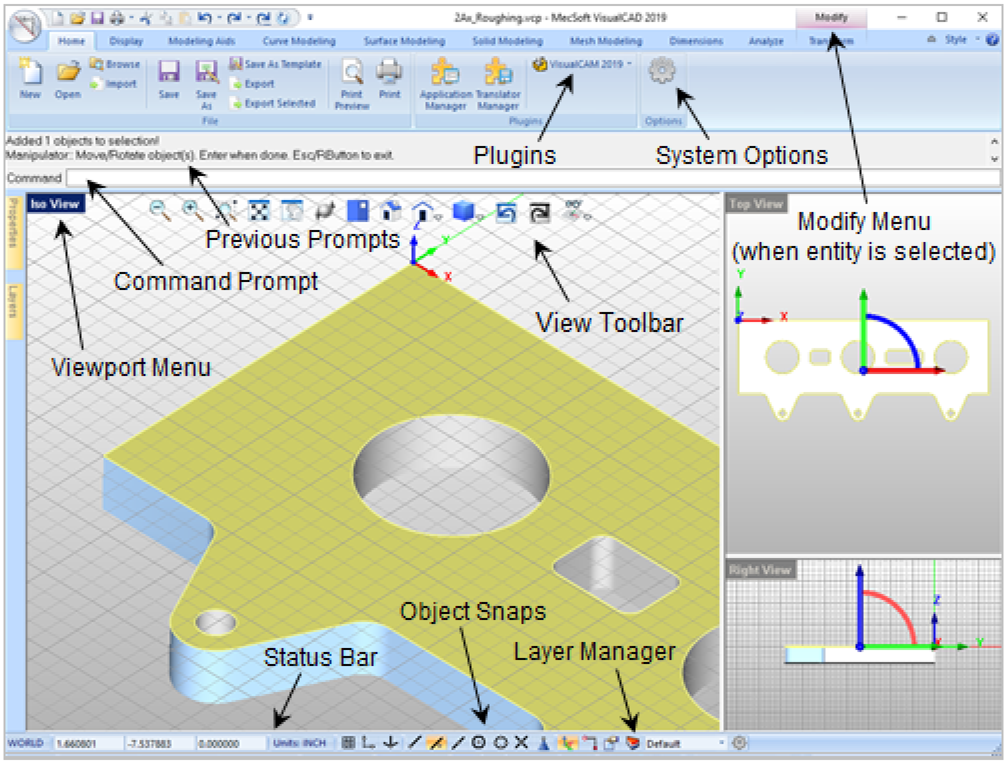
The VisualCAD Screen Layout

Default Shortcut Keys

Customize VisualCAD Display

1.2 Screen Layout

The VisualCAD screen layout is shown below. For high resolution displays (4K) the icons are automatically scaled to match the screen resolution.



VisualCAD Screen Layout

Related Topics

[Welcome to VisualCAD]()

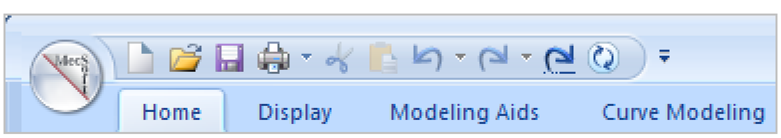
[VisualCAD Features Overview]()

[Default Shortcut Keys]()

[Customize VisualCAD Display]()

1.3 Quick Access Toolbar

The Quick Access Toolbar is located at the top of the VisualCAD display. It contains commands that you use on a regular basis. You can add and remove commands to the toolbar to suite your drawing needs.



The Quick Access Toolbar

Quick Access Toolbar's Default Commands

File New / Ctrl+N

File Open / Ctrl+O

File Save / Ctrl+S

Print... / Ctrl+P

Print Preview

Print Setup

Cut / Ctrl+X

Copy / Ctrl+C

Paste / Ctrl+V

Undo / Ctrl+Z

Undo Multiple

Redo / Ctrl+Y

Redo Multiple

Command Recall

Toggle Continuous Command Recall

Customize the Quick Access Toolbar

Related Topics

Customize Quick Access Toolbar

VisualCAD Features Overview

Default Shortcut Keys

Preset <Alt> Accelerator Keys

Customize VisualCAD Display

1.4 Default Shortcut Keys

VisualCAD installs with the following keyboard shortcuts preset. You can customize your keyboard to quickly execute many other VisualCAD commands using the Customize Quick Access Toolbar and Customize Keyboard dialogs.

Default Keyboard Shortcuts

Point / Ctrl+Alt+P

New / Ctrl+N

Open / Ctrl+O

Save / Ctrl+S

Print... / Ctrl+P

Undo / Ctrl+Z

Redo / Ctrl+Y

Cut / Ctrl+X

Copy / Ctrl+C

Paste / Ctrl+V

Lock / Ctrl+L

Unlock / Ctrl+Alt+L

Hide

Hide Non-Selected / Ctrl+Shift+H

Show / Ctrl+Alt+H

Select All / Ctrl+A

Select None / Ctrl+D

Chain Select / Shift +

Related Topics

Customize Quick Access Toolbar

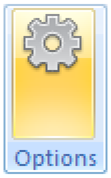
Customize Keyboard

Customize the Display

Preset <Alt> Accelerator Keys

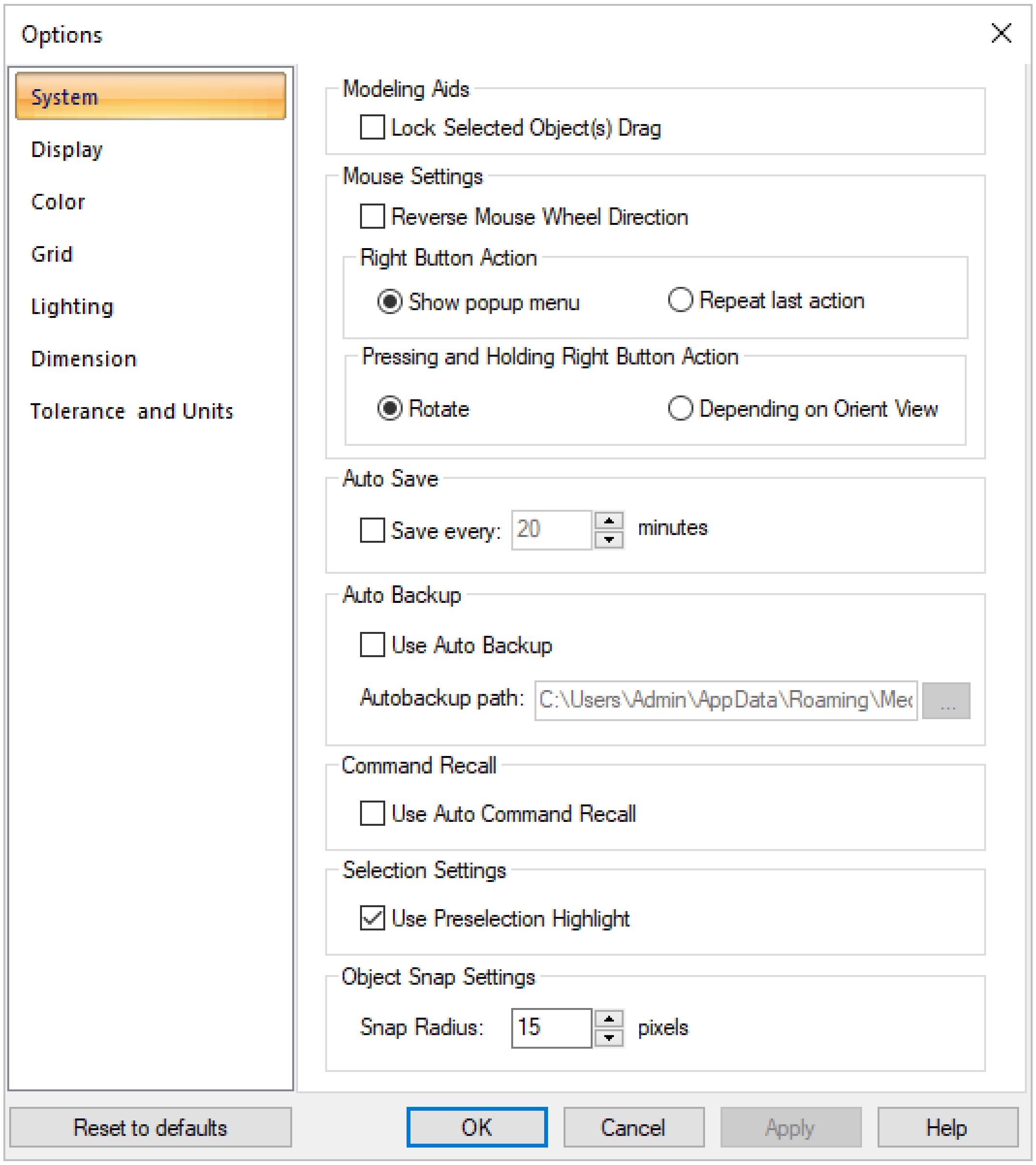
The VisualCAD Screen Layout

1.5 Options



The Home Ribbon Bar contains the Options pane. Selecting the icon will display the Options dialog shown below. This dialog allows you to set global parameters which include System, Display, Color, Grid, Lighting, Dimension, Tolerance and Units preferences. This dialog can also be accessed from the Status Toolbar.

Dialog Box: Options



Dialog Box: Options

Related Topics

[System]

[Display]

[Color]

[Grid]

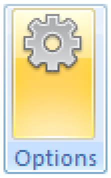
[Lighting]

[Dimension]

[Tolerance and Units]

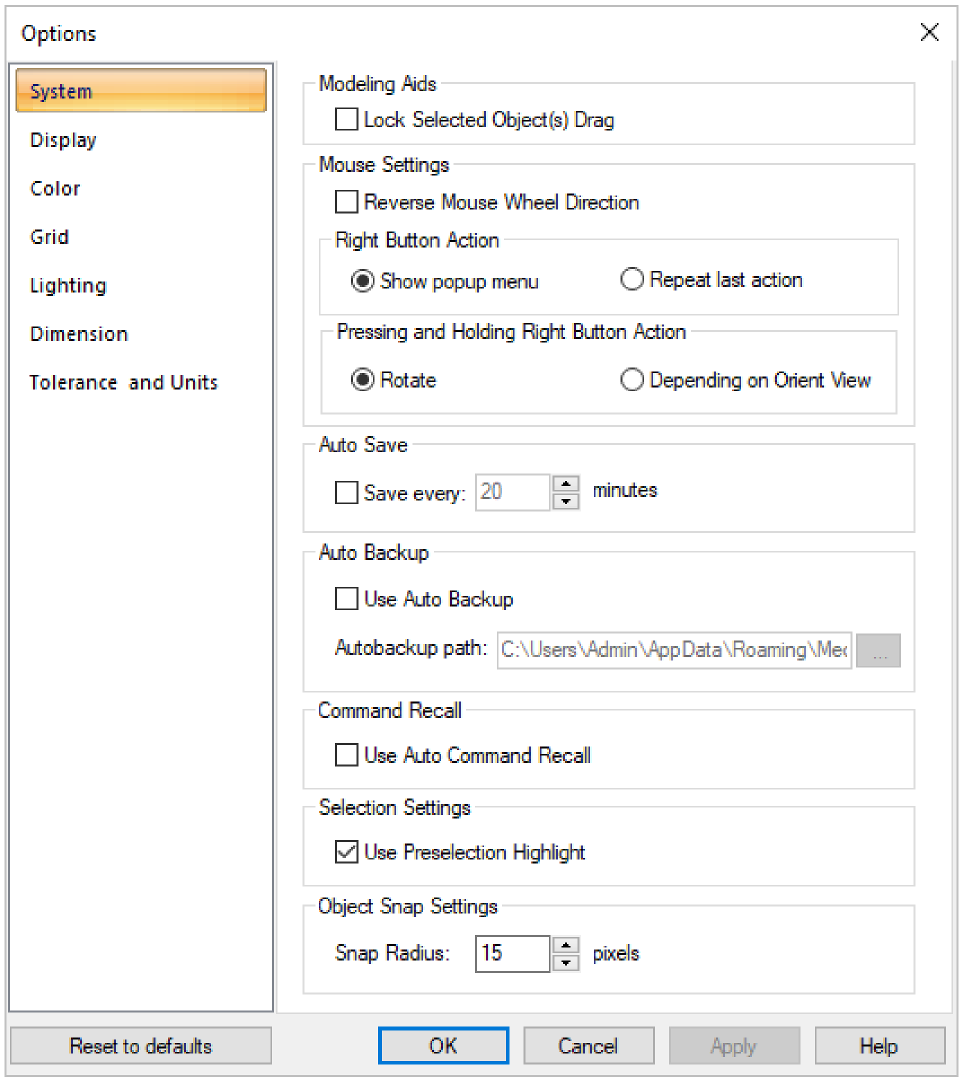
Home Ribbon Bar

1.5.1 System



Allows user to set preferences for modeling aids, mouse settings, auto save and command recall.

Dialog Box: Options > System



Dialog Box: Options > System

Lock Selected Object(s) Drag

If the box is not checked, you can select a geometry object and drag it around the C-Plane. When the box is checked, the objects cannot be dragged.

Mouse Settings

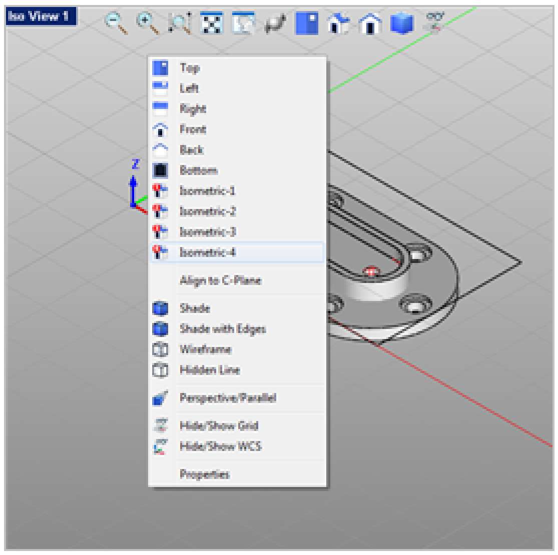
The following Mouse options are supported:

Reverse Mouse Wheel Direction

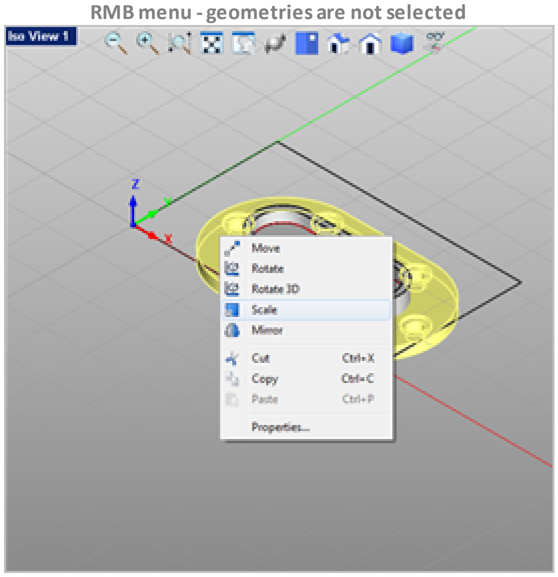
Reverses the direction of view zooming when using the mouse wheel.

Right Button Action

When Show pop-up menu is selected, performing a right mouse button click in the view port displays the commands in the pop-up menu.



RMB menu - geometries are not selected



RMB menu - geometries are selected

When Repeat last action is selected, this recalls the previous command.

Pressing and Holding Right Button Action

This option tells VisualCAD what to do when the right mouse button is pressed and held.

- Rotate
  When selected, pressing and holding down the right mouse button from all views performs a Rotate on the display.
- Depending on Orient View
  When depending on orient view is selected, pressing & holding down the right mouse button from Top, Front, Right, Left, Back, Bottom views does a Dynamic Pan of the geometries instead of rotate. For Iso views (Iso1,2,3 & 4) it performs a Rotate when pressing and holding down right mouse button down.

AutoSave

The AutoSave function can be used to automatically save your active part file at user-specified time intervals. AutoSave always saves to the current file name. This saved file is removed when you successfully close your model.

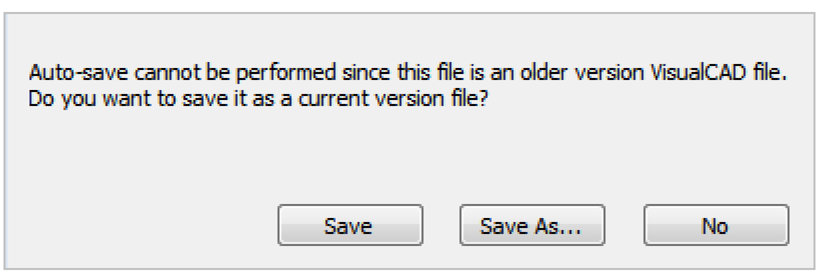
> **NOTE**: Once you activate AutoSave, you will need to Exit VisualCAD and then launch VisualCAD again before AutoSave will begin saving your part files!

Under the AutoSave function, check the box next to 'Save every xx minutes' to activate the AutoSave function. The time interval can be entered in minutes.

> AutoSave saves the file under: *<APPDATA>\MecSoft\VisualCAD 2018\AutoSave* where APPDATA is a logical pointing to a folder set up by your system administrator. By default it points to: *C:\Users\<username>\AppData\Roaming*.

Opening an Older Version *.vcp File

Opening an older version vcp file, displays the following dialog allowing you the opportunity to save the file to the current version.



AutoSave Message

When does AutoSave Prompt me to Save a File?

Under the following conditions VisualCAD will prompt you to save your file:

1. When you have opened a non-native VisualCAD file.
2. When you have imported a file.
3. When you have created a new part and has not saved the file yet.

When is the AutoSave File Deleted?

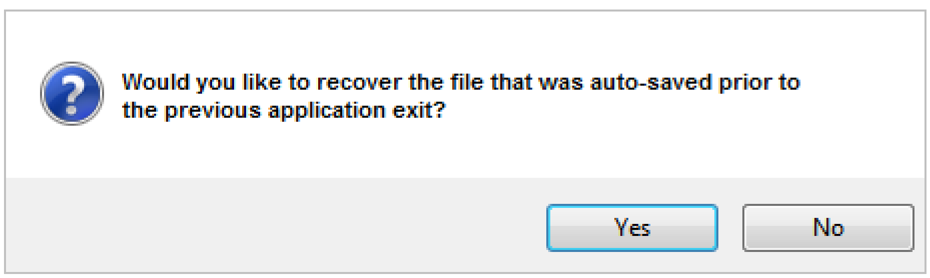
Auto save file is deleted under the following scenarios:

1. When the application closes normally.
2. When you close the file either explicitly or by opening another file.
3. When you select not to recover an AutoSave file that was detected on startup

File Recovery

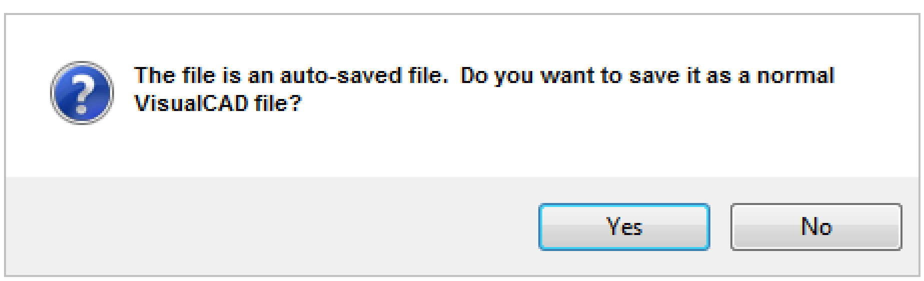
In the event of a computer crash or if VisualCAD is terminated abruptly by ending the process from task manager, VisualCAD will recover your file if AutoSave was active and your part file was automatically saved at least once.

If an AutoSave file has been created prior to the crash or termination, VisualCAD will detect the file located in the AutoSave folder the next time it is loaded. You are then prompted with the following dialog:



AutoSave Message

Selecting Yes, displays the following dialog, prompting you to save your file:



AutoSave Message

This is the only opportunity to recover the part file. If you choose not to recover it, the AutoSave file will be removed.

Use Auto Backup

Use this option to back up your VisualCAD files automatically. You can change the Auto Backup Path destination directory by selecting the 'browse' button.

If you have the check box checked, then the file is saved ONCE (as soon as the file is loaded) to the Auto Backup path with the same name as the original path file. This file is not deleted on exit so it can be recovered later if necessary. By default the check box to Auto Backup is unchecked.

Use Auto Command Recall

When this option is selected, VisualCAD automatically repeats the last command without having to go back & reselect the command from the toolbar or use the right mouse button click or enter option to repeat the previous command.

Use Preselection Highlight

When this option is selected, VisualCAD automatically highlights selections as the cursor moves over them. See Pre-Selection Highlight for more information.

Object Snap Settings

You can control the pixel radius of the object snaps located on the Status Toolbar. Enter the Snap Radius desired.

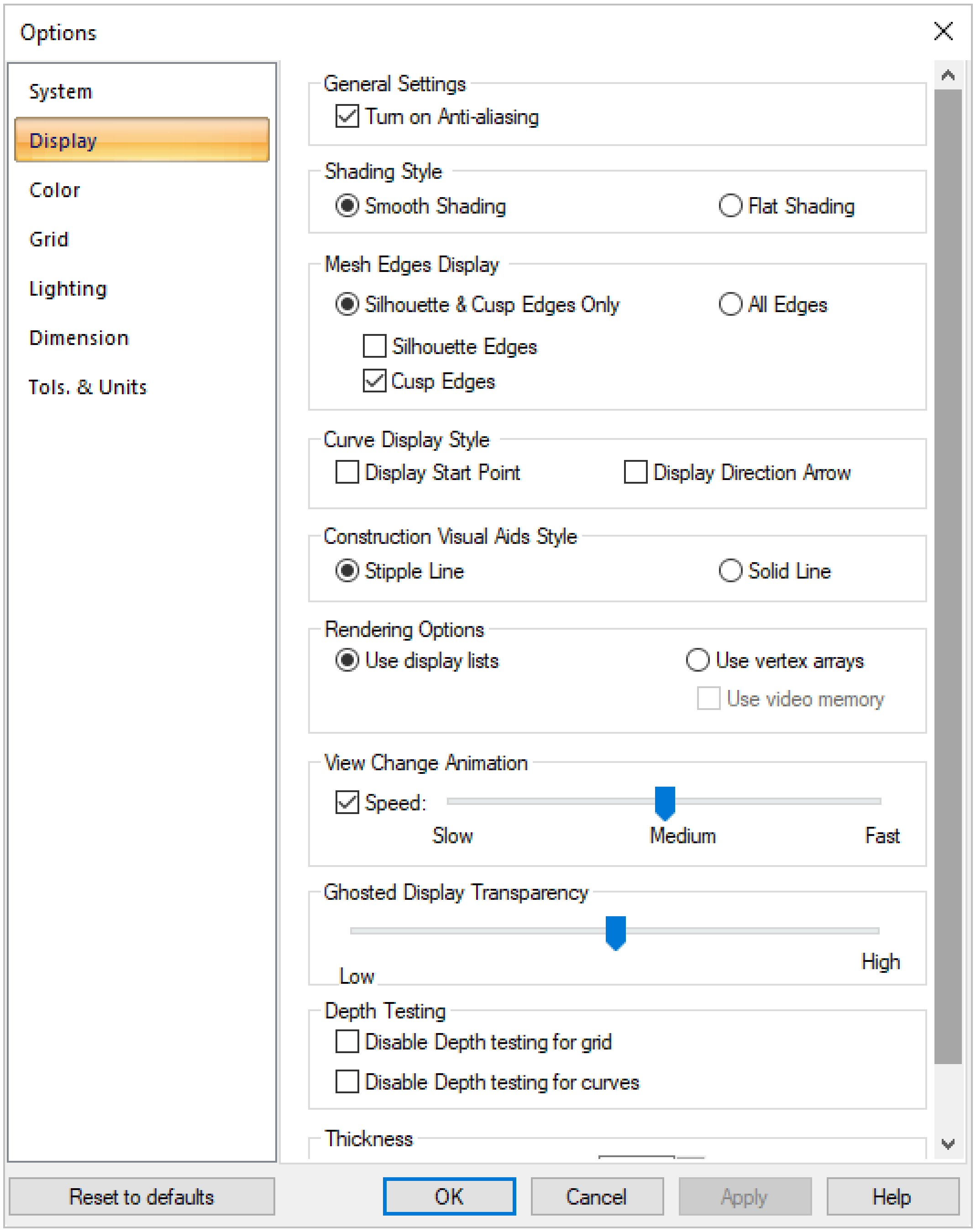
Related Topics

Options

1.5.2 Display

Controls how geometry is displayed. Select from the following:

Dialog Box: Options > Display



Dialog Box: Options > Display

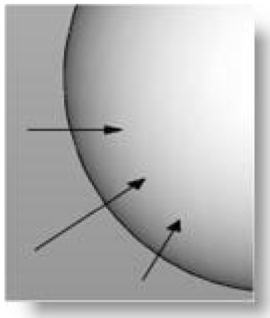
General Settings

Turn on Anti-aliasing

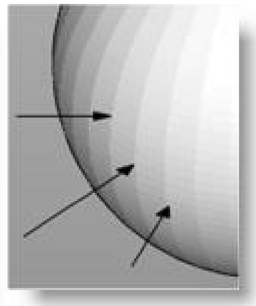
A method of smoothing the jagged edges along the lines and curves of text or graphics. Aliasing is caused by limited display resolution. Selecting this option turns on Anti-aliasing.

Shading Style

This controls how smooth surfaces are shaded.



Smooth Shading

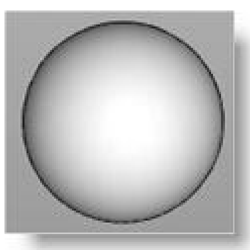


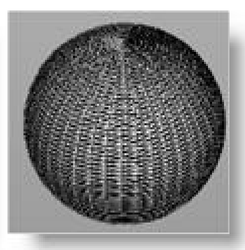
Flat Shading

The pictures demonstrate the difference between smooth and flat shading. The left picture smoothly changes the amount of shading as the surface curves. The right picture just displays the shading for a flat facet on the surface. Since all surfaces (including smooth ones) are internally represented as flat facets, flat shading can affect any surface. The default is smooth shading.

Mesh Edges Display Style

This changes how smooth surfaces are displayed. Since all surfaces are internally represented as flat facets, these facets join up along edges. On smooth surfaces, these edges are not sharp and are not normally displayed. An example of a sharp edge is the edge of a cube.

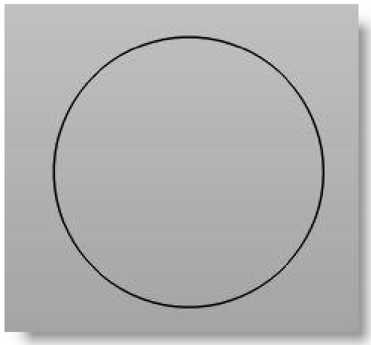




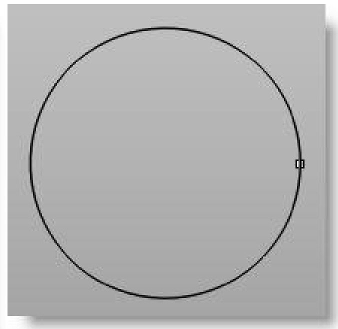
The pictures demonstrate the difference between silhouette & cusp edges only and all edge display. The left picture displays only sharp edges (there are none on a sphere); meanwhile, the right picture displays all the edges. The default is Silhouette & Cusp Edges Only. When selected, this allows you to select Silhouette Edges, Cusp Edges or All Edges to display.

Curve Display Style

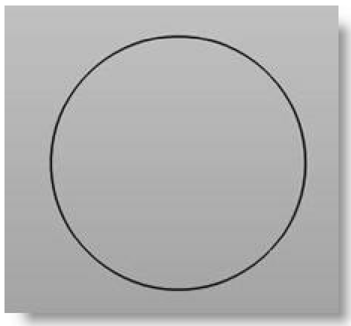
This provides an option to turn on or off display of curve start point and direction arrow.



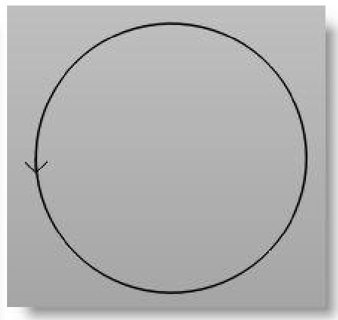
Start Point display OFF



Start Point display ON



Curve Direction arrow OFF

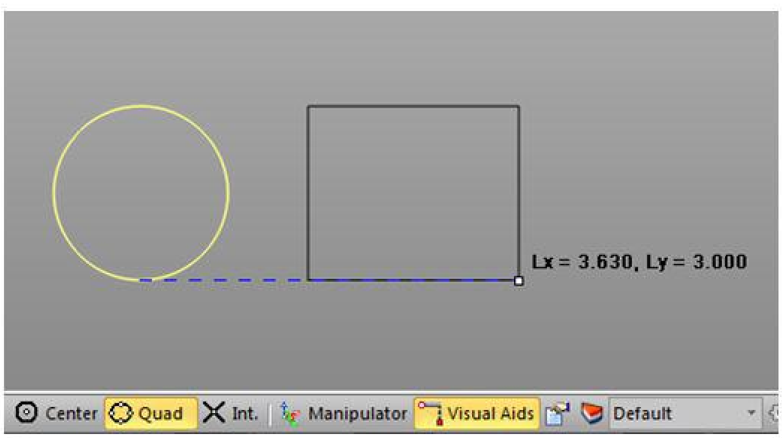


Curve Direction arrow ON

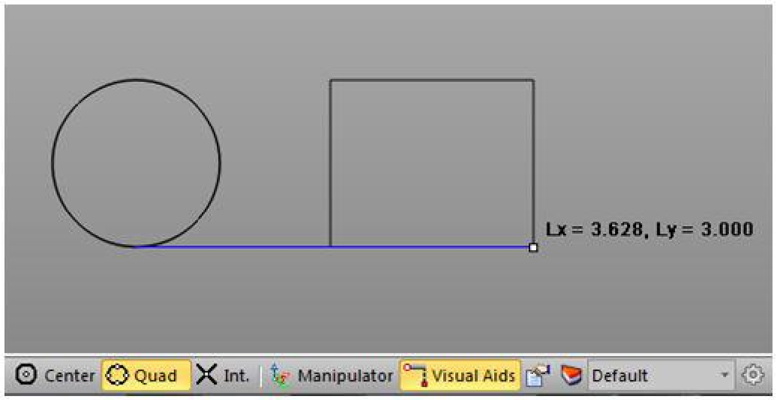
Selecting Start Point and Curve Direction arrow display under options displays both start point and curve direction arrow.

Construction Visual Aids Style

This provides an option to choose the display style between Stipple line and Solid line for visual aids when the Visual Aids is set to active under the status bar.



Stipple line display



Solid line display

Rendering Options

This can be set to Use Display Lists or Vertex Arrays. The Vertex Arrays option also provides the option to Use Video Memory from the video card that is installed on the computer. The speed and performance varies depending on the operating system type and video card installed.

View Change Animation

Whenever the view orientation is changed within the active viewport, VisualCAD can make that orientation change instantly or through smooth animation. If animation is used, the speed of the animation can be set.

Under the View Change Animation function, check the box next to Speed to activate the animation. The animation speed can be controlled using the slider bar.

Ghosted Display Transparency

Whenever the view mode is set to Ghosted Display, you can use this slider to control the transparency level of the hidden (ghosted) geometry.

Depth Testing

These options allow you to fully display certain objects regardless of the view mode. For example, checking the box to Disable Depth Testing for curves, will display all curves even if the view is set to shaded model. You can also Disable Depth testing for grid. If enabled, the grid lines will always display.

Thickness

Use this parameter to set the display line thickness for all wireframe geometry. Set this to 1 for the default line thickness. Higher for thicker lines.

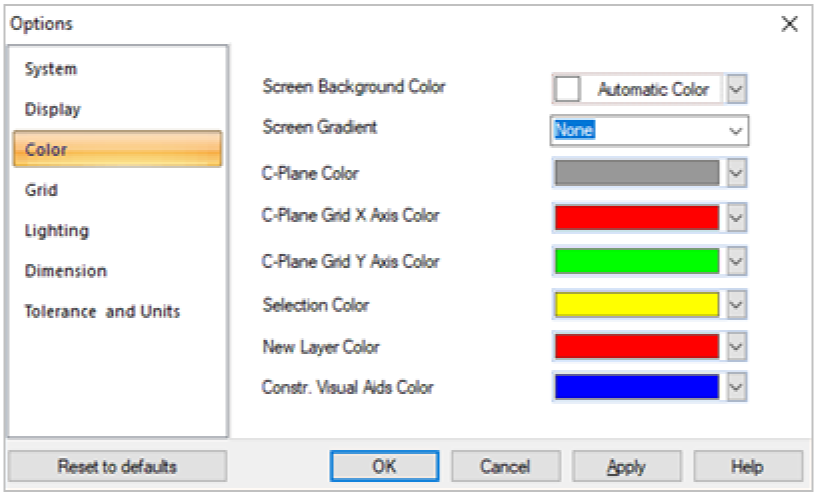
Related Topics

[Options](#)

1.5.3 Color

The default colors are set using this dialog.

Dialog Box: Options > Color



Dialog Box: Options > Color

- Screen Background Color
  This allows user to set the viewport background color. This is the color displayed when no objects are at that position.
- Screen Gradient
  This is a smooth change of lighting. The direction specified is the lighter side. It starts slightly darker on the opposite side of the screen and gradually gets lighter until it reaches the specified side.
- C-Plane Color
  This changes the color of the construction plane grid.
- C-Plane Grid X Axis Color
  This changes the horizontal axis line of the construction plane.
- C-Plane Grid Y Axis Color
  This changes the vertical axis line of the construction plane.
- Selection Color
  This is the highlight color of objects that have been selected.
- New Layer Color
  This is the default color of objects created on new (not Default) layers.

- Constr. Visual Aids Color
  This is the default color of construction aids when Visual Aids is set to active in the status bar.

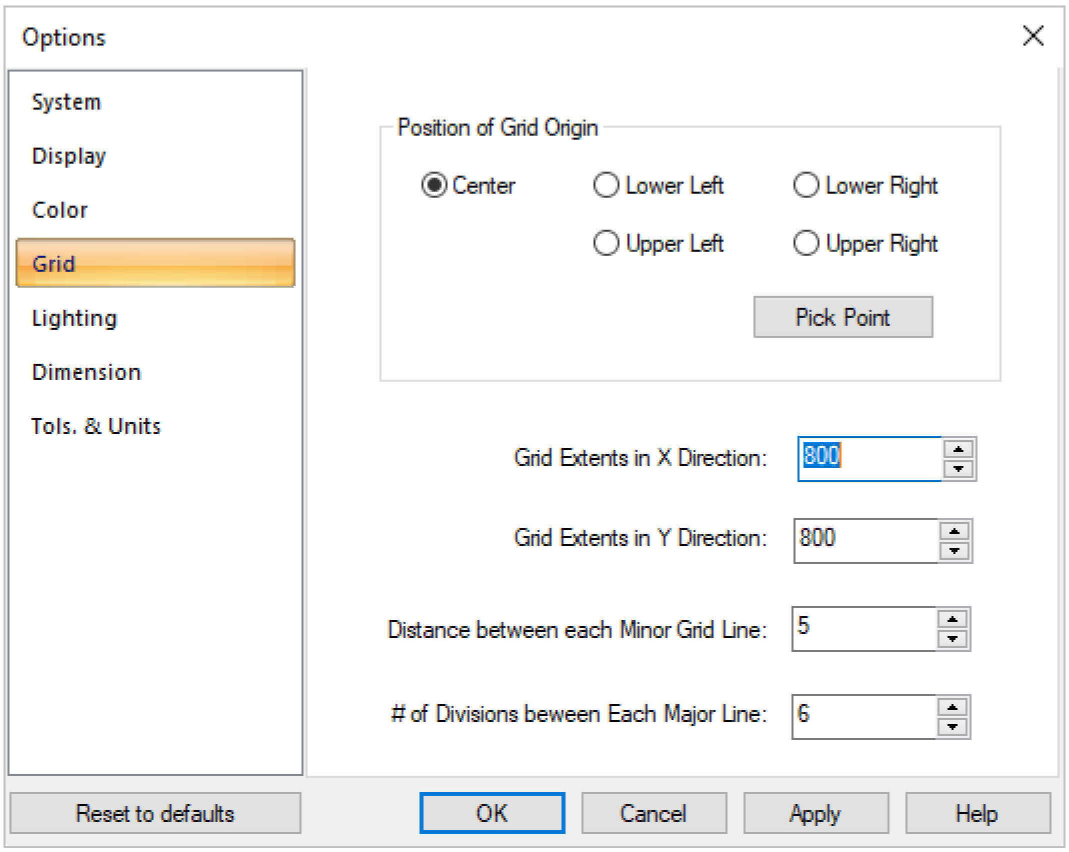
Related Topics

[Options]

1.5.4 Grid

This controls how the construction plane grid is displayed.

Dialog Box: Options > Grid



Dialog Box: Options > Grid

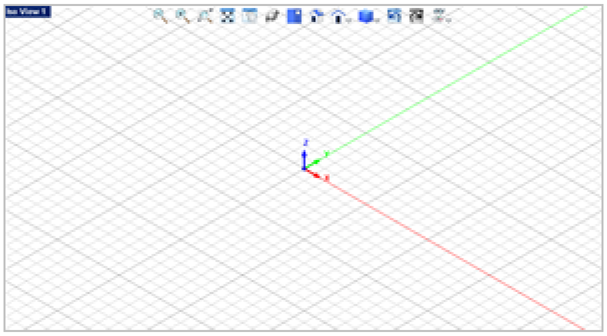
Position of Grid Origin

These options will control where the grid is positioned relative to the World Coordinate System (WCS).

The grid orientation also carries over to whichever view is active, making construction easier.

Center

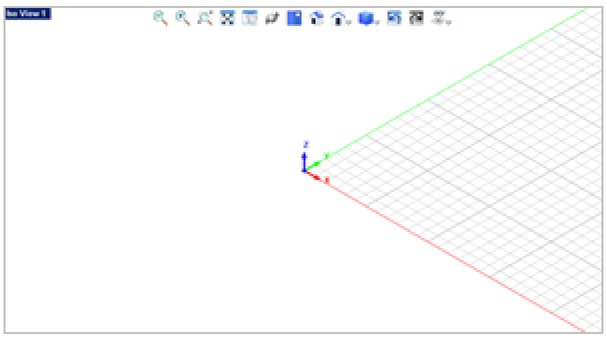
Move the center of the grid to the WCS.



Move the center of the grid to the WCS.

Lower Left

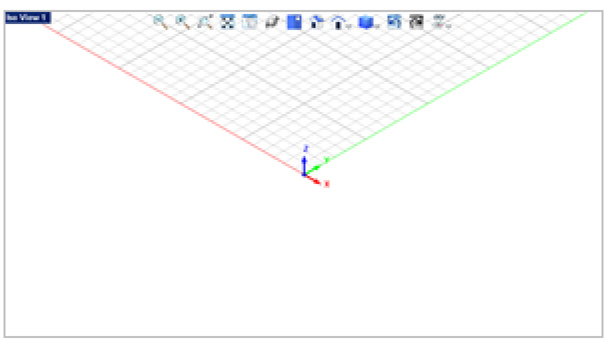
Move the lower left corner of the grid to the WCS.



Move the lower left corner of the grid to the WCS.

Lower Right

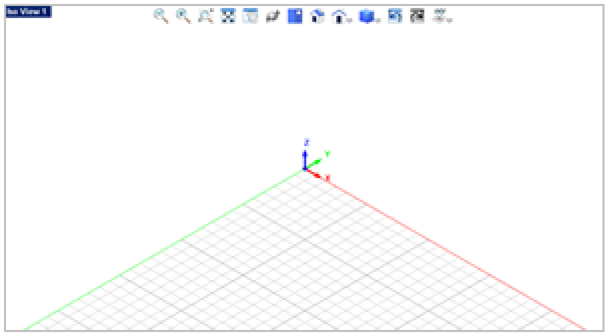
Move the lower right corner of the grid to the WCS.



Move the lower right corner of the grid to the WCS.

Upper Left

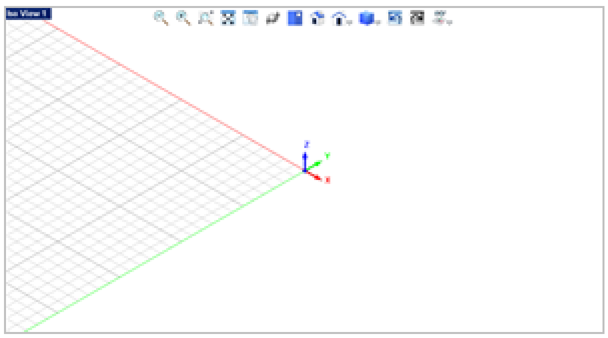
Move the upper left corner of the grid to the WCS.



Move the upper left corner of the grid to the WCS.

Upper Right

Move the upper right corner of the grid to the WCS.



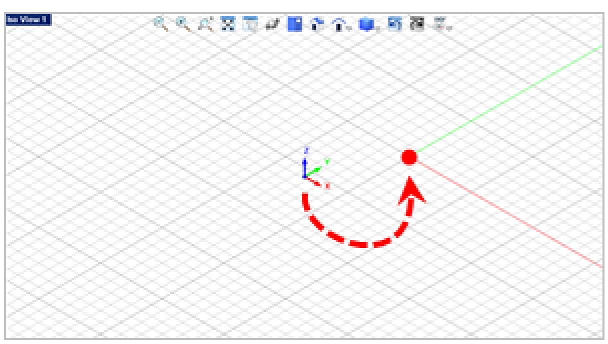
Move the upper right corner of the grid to the WCS.

Pick

Use the button to pick a point on the screen and the origin of the grid will move to this location.

Notes:

- The point you pick is always on the default XY plane.
- If you use the Upper/Lower options above after using this Pick option, the offset distance between the WCS and the new origin of the grid remains in effect.



Move the the origin of the grid to this location.

Other Grid Options

Grid Extents in X Direction

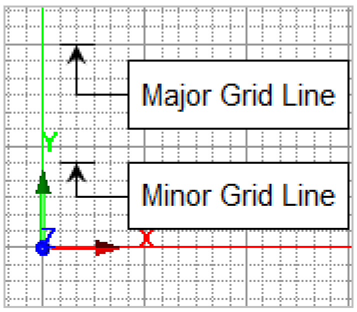
This controls how many major divisions are displayed along the X axis direction.

Grid Extents in Y Direction

This controls how many major divisions are displayed along the Y axis direction.

Distance between each Minor Grid Line

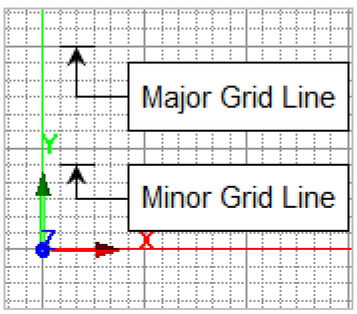
This controls the distance between each minor grid line.



Major/Minor Grid Lines

of Divisions between Each Major Line

This controls the distance between each major grid line. the number of minor rid line multiplied by the number of major grid lines will determine the overall extent of the grid.



Major/Minor Grid Lines

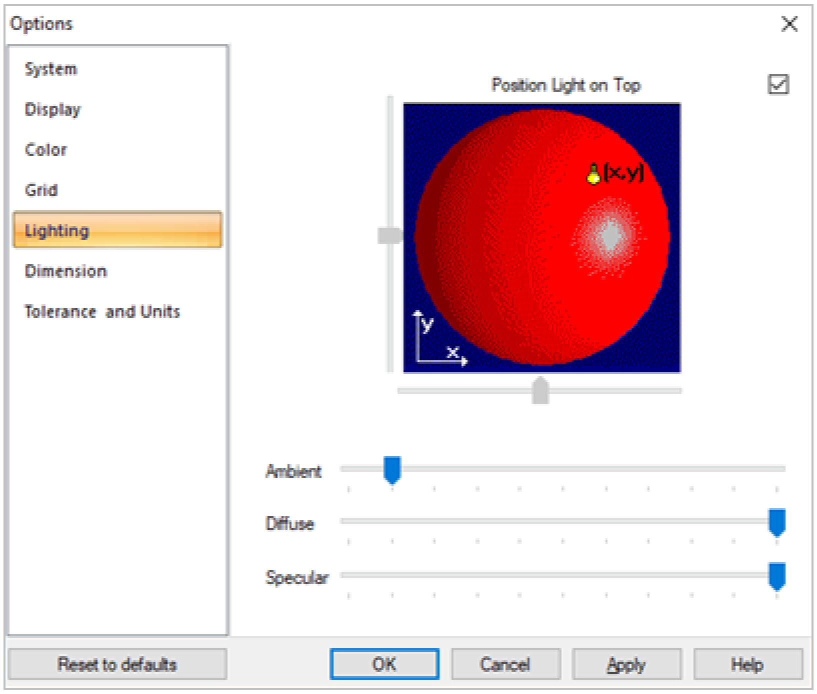
Related Topics

Options

1.5.5 Lighting

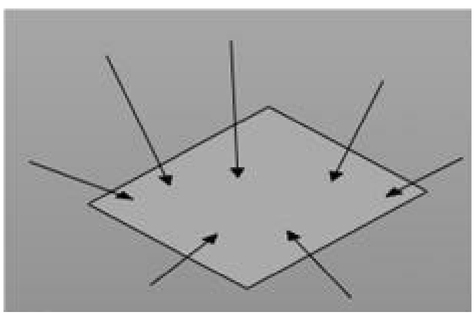
This controls the characteristics of lighting for the model.

Dialog Box: Options > Lighting



Dialog Box: Options > Lighting

There are three types of lighting: ambient, diffuse, and specular.

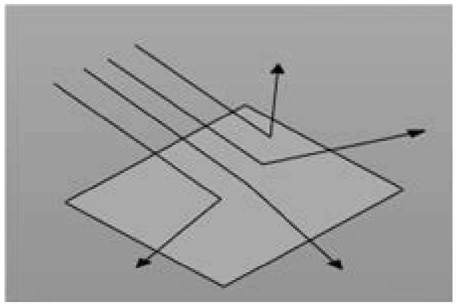


Position Light on Top

The scene has a single light source and there are controls for setting its position. Not all lighting in the model comes directly from that one source; there is some ambient light coming in from all over. The **Position Light on Top** checkbox must be unchecked to be able to move the light source.

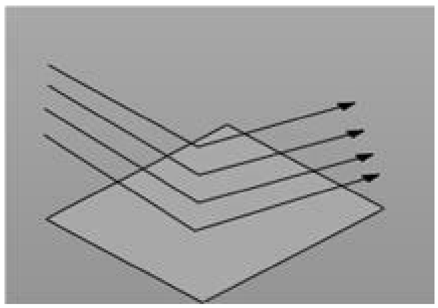
Ambient

Ambient light comes in from all different directions equally. Light sources have no effect on ambient light. Shadows don't appear with only ambient light.



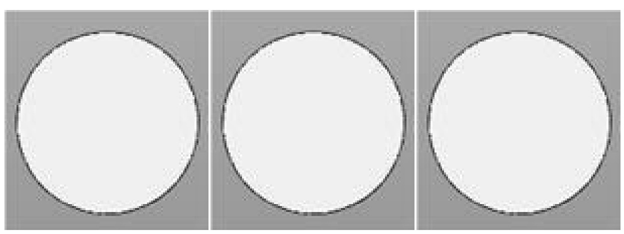
Diffuse

Diffuse light comes from a single source but is scattered in all directions. Shadows appear, but there are no "shiny" spots.

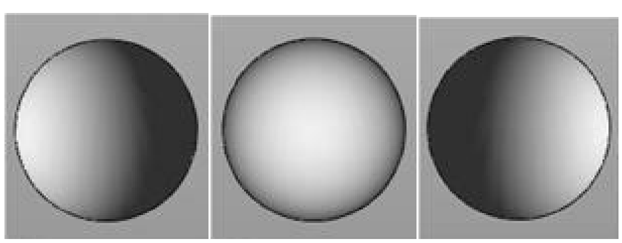


Specular

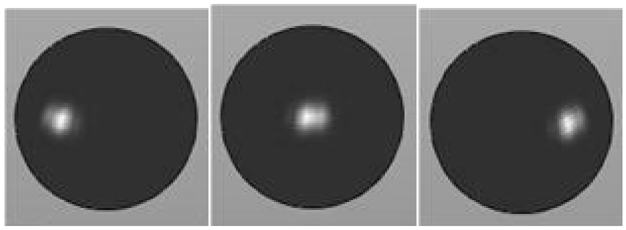
Specular light comes from a single source and is scattered in a single direction. It is reflective light. Shiny metal objects demonstrate good specular reflection. Shading is not a significant consideration with specular light.



The three pictures above demonstrate ambient light as the light source moves from left to right. There is no effect since ambient light comes from all sides equally.



The pictures above show the effects of the motion of the light source using only diffuse lighting. There are no "shiny" spots but yet there is a direction from the light.



The last set of pictures uses only specular lighting. As the light source moves from left to right, the "shiny" spot follows it. However, there is nothing else lit up, so the effect is not just reflective metal.

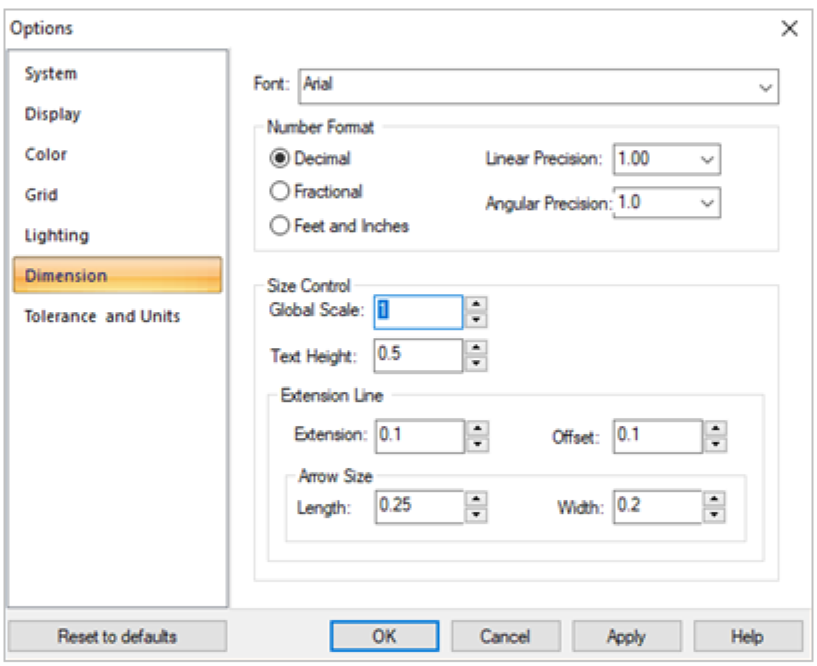
Related Topics

Options

1.5.6 Dimension

This controls the appearance of dimensioning labels, arrows, and leader lines.

Dialog Box: Options > Dimension



Dialog Box: Options > Dimension

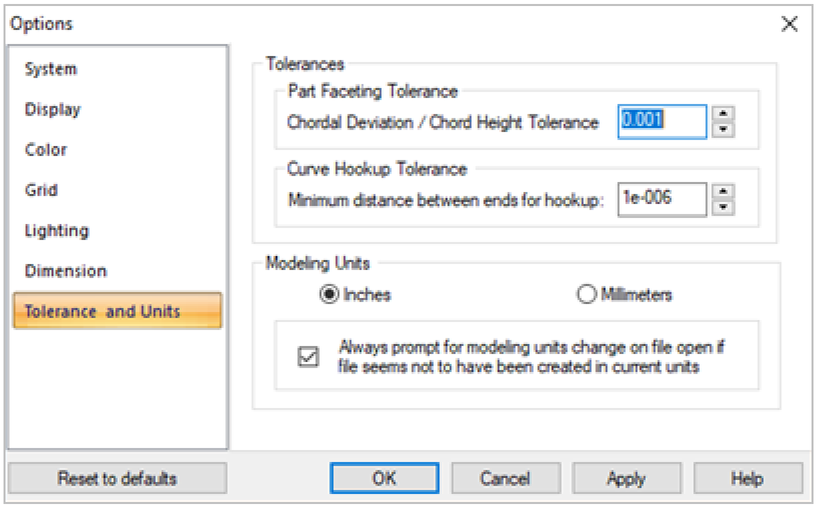
Related Topics

Options

1.5.7 Tolerance and Units

This controls the accuracy of the model and part units.

Dialog Box: Options > Tolerance and Units



Dialog Box: Options > Tolerance and Units

Tolerances

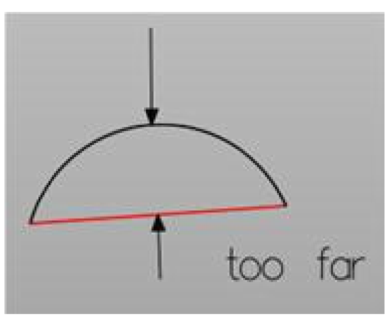
This controls the accuracy of the models. Smaller tolerance numbers often result in significantly big files and slower processing time but are more accurate.

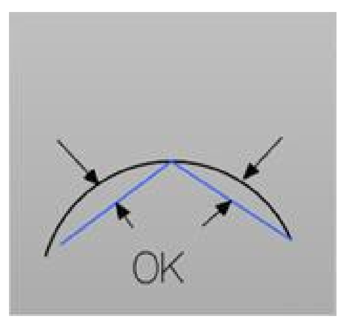
The tolerances are set in a dialog.

Part Faceting Tolerance

Chordal Deviation/Chord Height Tolerance

The chord height tolerance is used in commands that require approximation of smooth curves and surfaces.



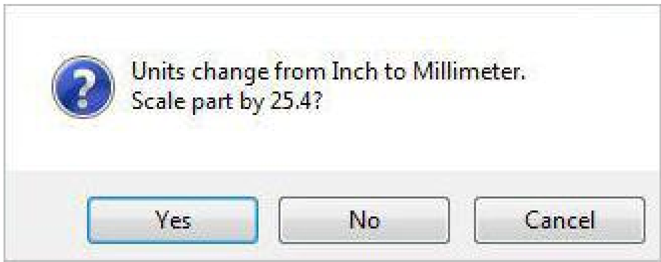


Curve Hookup Tolerance

This specifies maximum separation between the endpoints of different curves when joining them into a single curve.

Units

This converts units from inches to millimeters or from millimeters to inches. If there are existing objects, a dialog will appear confirming if you really want to convert the units.



The objects remain the original size but the coordinate values change.

You can check the box to always prompt you to change the units when files are opened if they appear not to be in the correct units. For example, if the system sees a coordinate value that is quite large such as 500.5, then this dialog will display giving you the opportunity to change the units to Millimeters.

Related Topics

[Options]

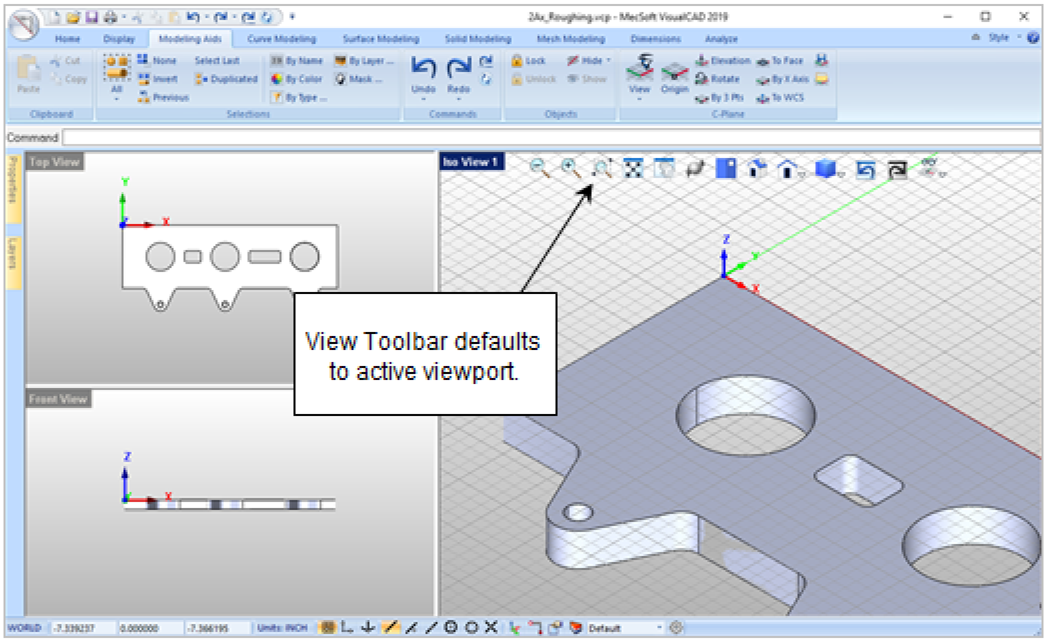
1.6 View Toolbar

The View Toolbar contains easily accessible commands that control the appearance of the currently active view. This includes motion of the camera as well as how objects are displayed. When a view change is invoked from the View Toolbar toolbar, the existing view magnification factor is maintained. The View Toolbar is displayed in the currently active [Viewport].

The View Toolbar

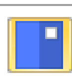
The View Toolbar

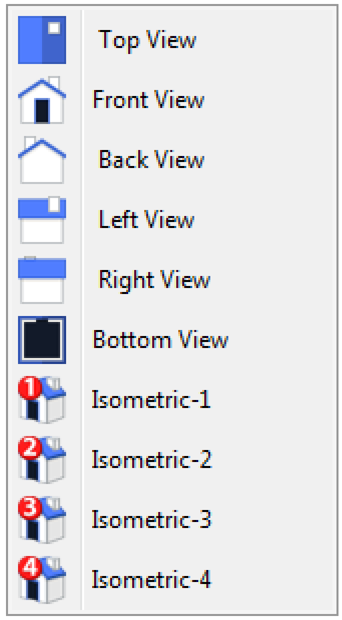
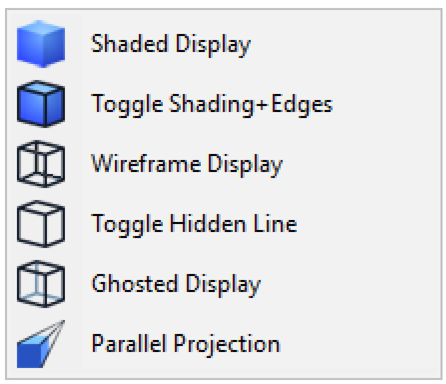
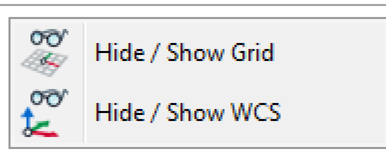
The View Toolbar Displayed in the Active Viewport



View Toolbar Displays in the Active Viewport by Default

The View Toolbar Commands

Icon	Command/Menu	Command
	Zoom Out	
	Zoom In	
	Zoom Box	
	Fit View	-
	Dynamic Pan	
	Dynamic Rotate	
	Top View	
	Iso View	

	 View Menu	Top View Front View Back View Left View Right View Bottom View Isometric-1,2,3,4
	 Display Menu	Shaded Display Toggle Shaded+Edges Wireframe Display Toggle Hidden Lines Ghosted Display Toggle Parallel/Perspective Projection
	Previous View	These are Undo/Redo View commands.
	Next View	
	 Show/Hide Menu	Hide/Show Grid Hide/Show WCS

Related Topics

Zoom Out

Zoom In

Zoom Box

Fit View

Dynamic Pan View

Dynamic Rotate View

Top View

Iso View

Front View

Back View

Left View

Right View

Bottom View

Shaded Display

Toggle Shading + Edges

Wireframe Display

Hidden Line Display

Ghosted Display

Toggle Parallel/Perspective Projection

Hide/Show Grid

Hide/Show WCS

1.6.1 3D Mouse

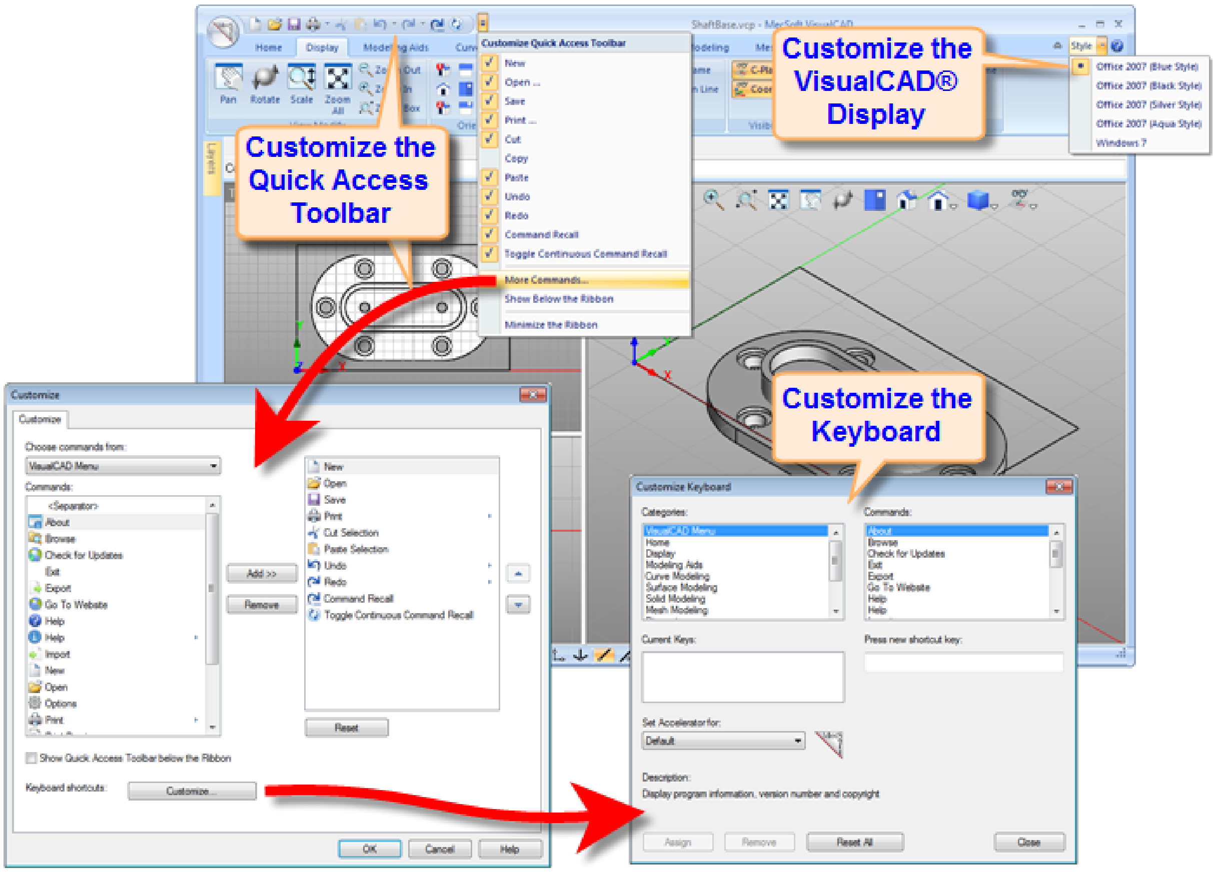
VisualCAD supports 3D devices from 3Dconnexion®. It can be customized in the 3Dconnexion® Properties dialog by selecting the Button Configuration tab. Button configurations can be set for either the Left mouse button, the Right mouse button, or both and other buttons depending on the type of 3D device you are using.

Related Topics

View Tool Bar

1.7 Customize VisualCAD

The VisualCAD display can be easily customized to suit your drawing needs.



Customize VisualCAD®

Related Topics

[Customize Quick Access Toolbar]

[Minimize Ribbon Bar]

[Preset <Alt> Accelerator Keys]

[Customize Dialog]

[Customize the Keyboard]

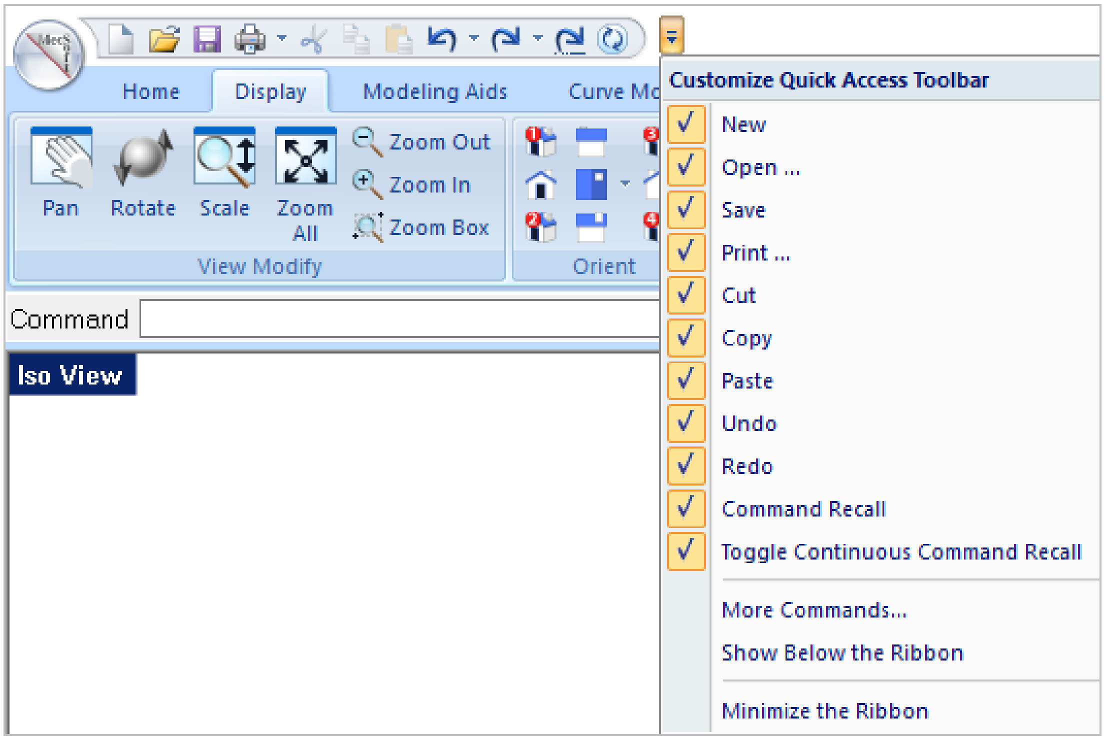
[Styles Menu]

1.7.1 Customize Quick Access Toolbar

You can add/remove commands to the Quick Access Toolbar and perform other User Interface customization from the Customize Quick Access Toolbar menu.

Menu: Customize Quick Access Toolbar

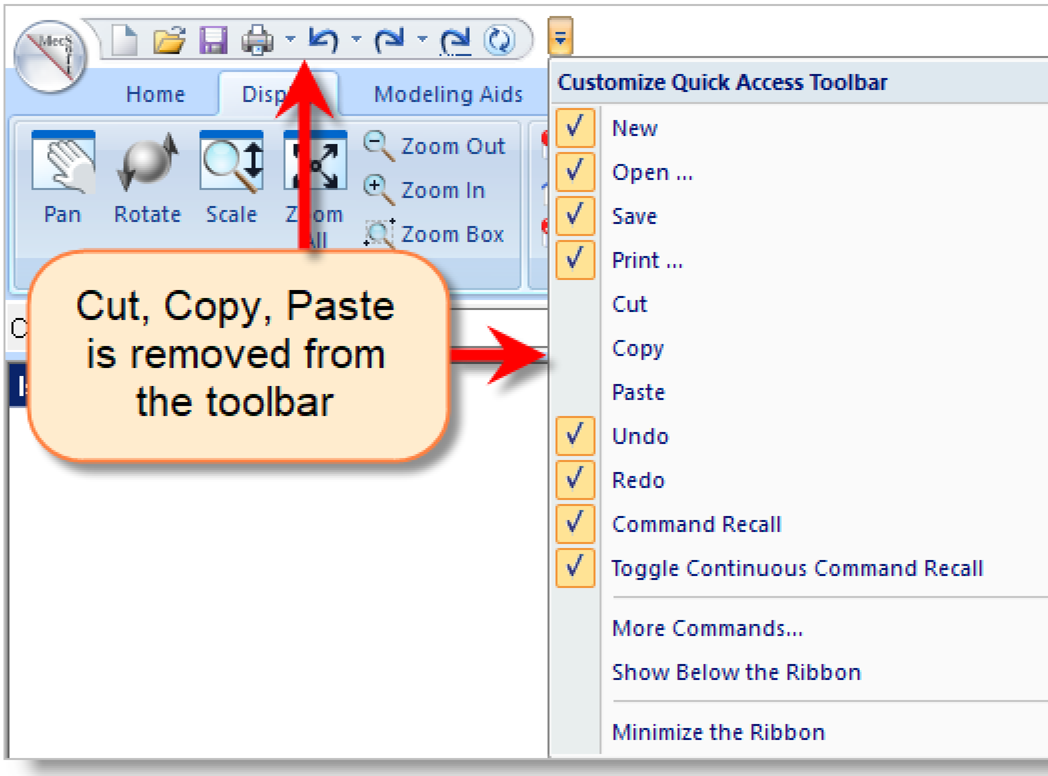
Select the indicator to the right of the Quick Access Toolbar to display the menu:



Displaying the Quick Access Toolbar Manu

Quickly Turn Commands On/Off

Select a command from the menu to "uncheck" it and remove it from the Quick Access Toolbar. Selecting it again will add it back to the toolbar.



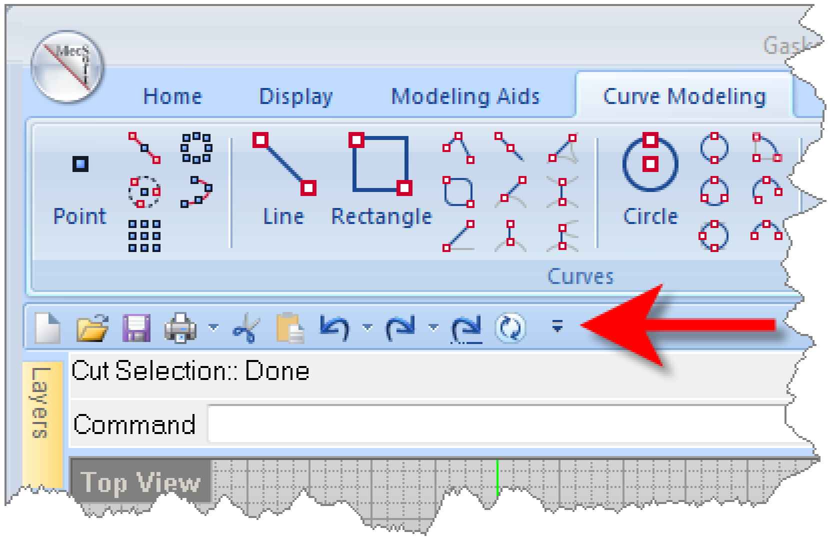
Quickly Turn Commands On/Off from the Quick Access Toolbar

More Commands...

You can add more commands to the toolbar by selecting More Commands ... from the Quick Access Toolbar Menu. This will display the Customize dialog. From this dialog you can also Customize the Keyboard.

Show below the Ribbon

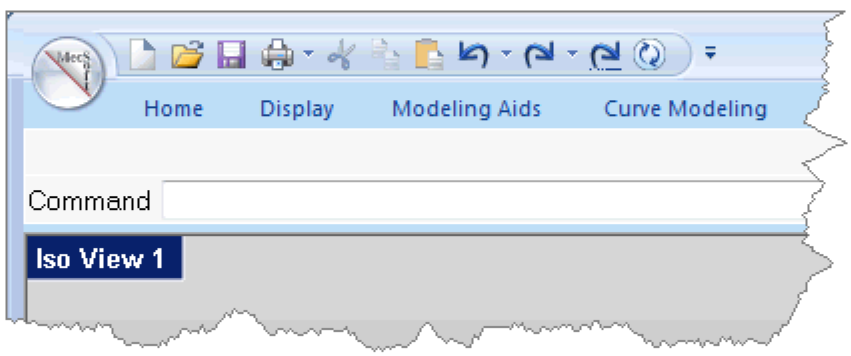
You can move the Quick Access Toolbar to reside below the Ribbon bar by selecting Show Below Ribbon from the Customize Quick Access Toolbar menu. You can also do this by selecting appropriate check box in the Customize dialog.



Show the Quick Access Toolbat below the Ribbon Bar

Minimize the Ribbon Bar

You can minimize the Ribbon Bar by selecting Minimize the Ribbon from the Customize Quick Access Toolbar menu. You can then select one of the tabs to show the Ribbon Bar as needed.



The Ribbon Bar is shown Minimized

Related Topics

[Customize VisualCAD]()

[Customize Quick Access Toolbar]()

[Preset <Alt> Accelerator Keys]()

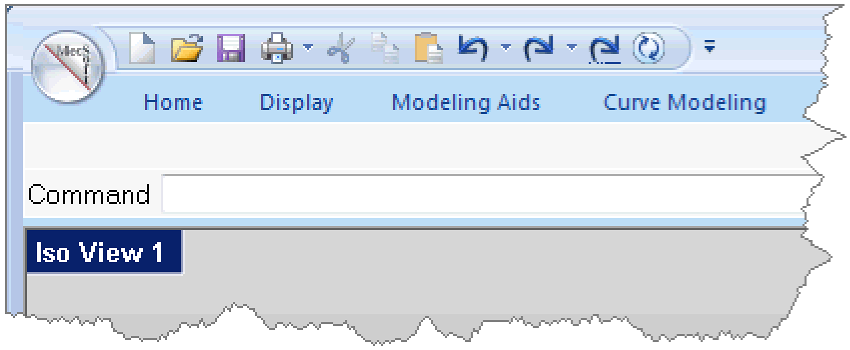
[Customize Dialog]()

[Customize the Keyboard]()

[Styles Menu]()

1.7.2 Minimize Ribbon Bar

You can minimize the Ribbon Bar by selecting Minimize the Ribbon from the Customize Quick Access Toolbar menu. You can then select one of the tabs to show the Ribbon Bar as needed.



The Ribbon Bar is shown Minimized

Related Topics

Customize VisualCAD

Customize Quick Access Toolbar

Preset <Alt> Accelerator Keys

Customize Dialog

Customize the Keyboard

Styles Menu

1.7.3 Preset <Alt> Accelerator Keys

VisualCAD installs with Preset <Alt> Key combinations that you can use to quickly execute the various commands located on each Ribbon Bar. Refer to topic below for more information.

Using the Preset <Alt> Keys

1. Press the <Alt> key on your keyboard and VisualCAD will display (and activate) <Alt> key assignments for the VisualCAD Menu, Quick Access Toolbar and each Ribbon Bar.
2. With the <Alt> key pressed, you can then press one of the assigned keys to execute the command or display the menu or Ribbon Bar.
3. To display (and activate) the <Alt> keys assigned to commands on one of the Ribbon Bars, first press the <Alt> key combination for the desired Ribbon Bar and then with the <Alt> key still pressed, press the key assigned to the desired command.

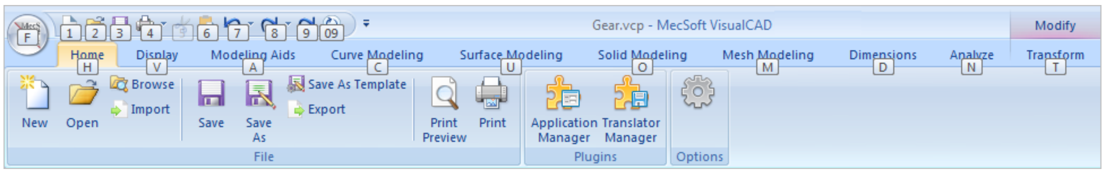
4. For example, to execute the Single Line command, press <Alt+C> to display the Curve Modeling Ribbon Bar and then press <LS> with the <Alt> key still pressed.

The preset <Alt> key assignments for each menu and command are shown below.

Preset <Alt> Keys: Menus

The following preset <Alt> Accelerator key combinations are assigned to commands on the VisualCAD Menu, Quick Access Toolbar and Ribbon Bars as shown below. These can be displayed in VisualCAD by pressing the <Alt> key.

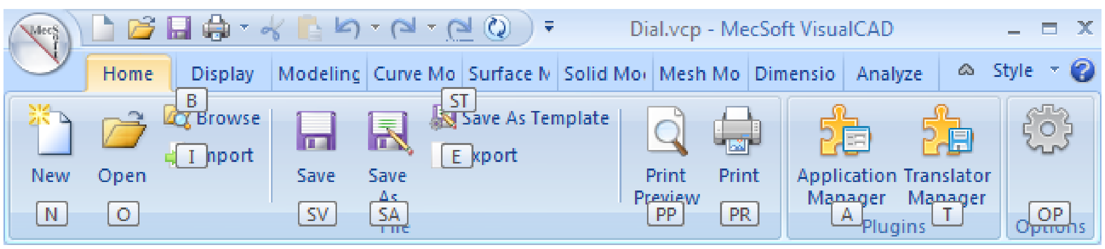
For example, pressing <Alt+A> will automatically display the Modeling Aids Ribbon Bar.



Preset <Alt> Keys: Menus

Preset <Alt> Keys: Home Ribbon Bar

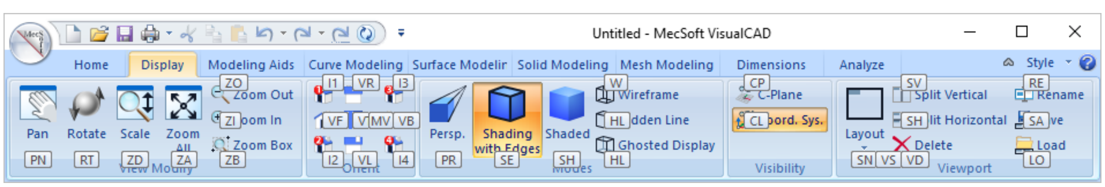
The following preset <Alt> Accelerator key combinations are assigned to commands on the Home Ribbon Bar.



Preset <Alt> Keys: Home Ribbon Bar

Preset <Alt> Keys: Display Ribbon Bar

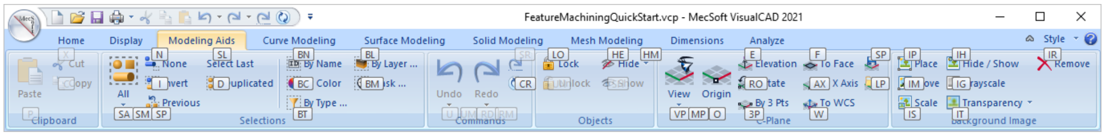
The following preset <Alt> Accelerator key combinations are assigned to commands on the Display Ribbon Bar.



Preset <Alt> Keys: Display Ribbon Bar

Preset <Alt> Keys: Modeling Aids Ribbon Bar

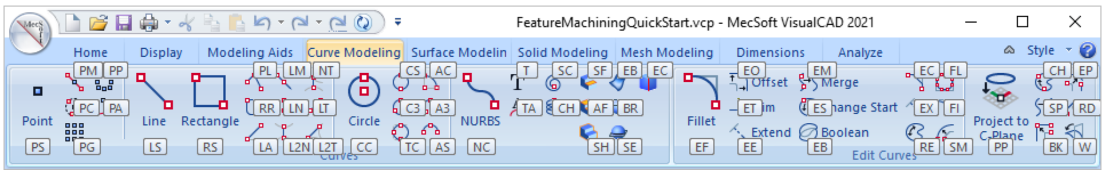
The following preset <Alt> Accelerator key combinations are assigned to commands on the Modeling Aids Ribbon Bar.



Preset <Alt> Keys: Modeling Aids Ribbon Bar

Preset <Alt> Keys: Curve Modeling Ribbon Bar

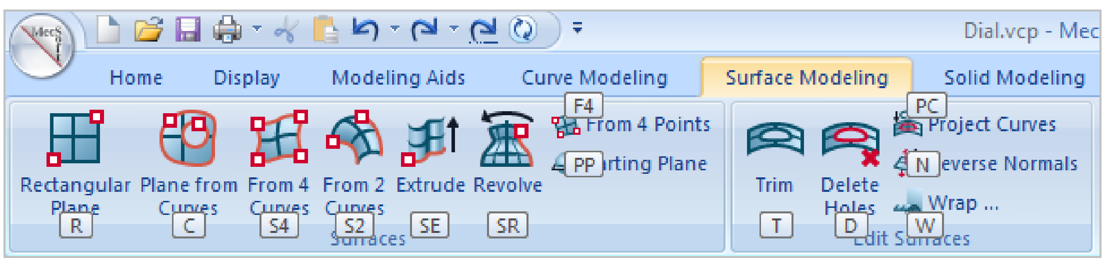
The following preset <Alt> Accelerator key combinations are assigned to commands on the Curve Modeling Ribbon Bar.



Preset <Alt> Keys: Curve Modeling Ribbon Bar

Preset <Alt> Keys: Surface Modeling Ribbon Bar

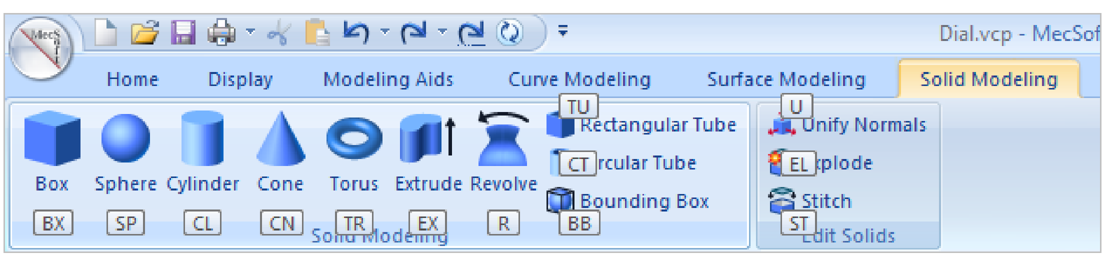
The following preset <Alt> Accelerator key combinations are assigned to commands on the Surface Modeling Ribbon Bar.



Preset <Alt> Keys: Surface Modeling Ribbon Bar

Preset <Alt> Keys: Solid Modeling Ribbon Bar

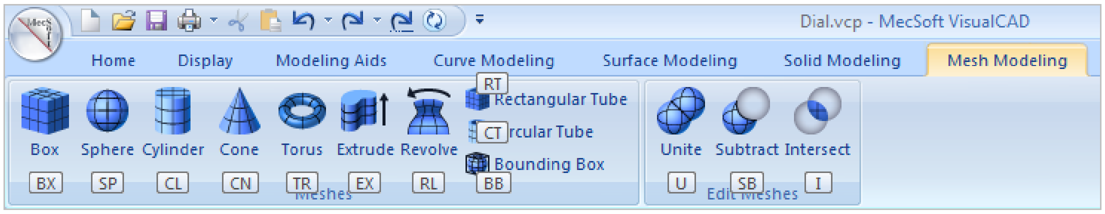
The following preset <Alt> Accelerator key combinations are assigned to commands on the Solid Modeling Ribbon Bar.



Preset <Alt> Keys: Solid Modeling Ribbon Bar

Preset <Alt> Keys: Mesh Modeling Ribbon Bar

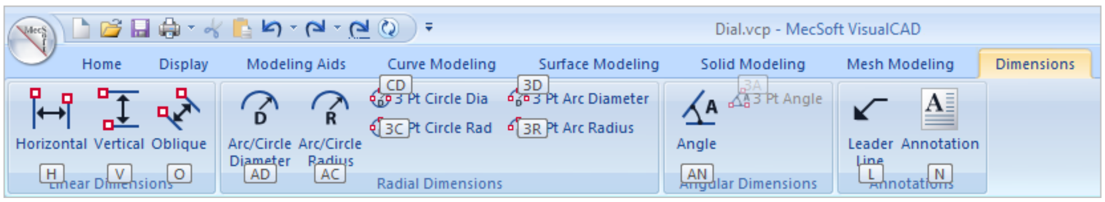
The following preset <Alt> Accelerator key combinations are assigned to commands on the Mesh Modeling Ribbon Bar.



Preset <Alt> Keys: Mesh Modeling Ribbon Bar

Preset <Alt> Keys: Dimensions Ribbon Bar

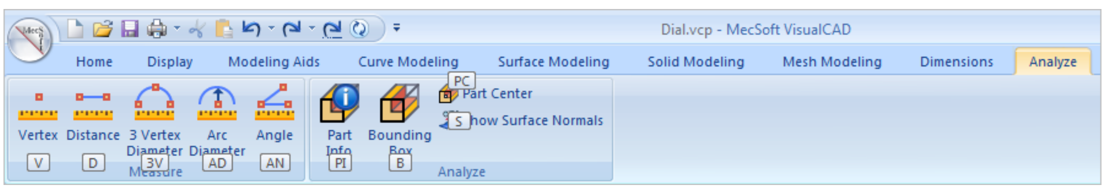
The following preset <Alt> Accelerator key combinations are assigned to commands on the Dimensions Ribbon Bar.



Preset <Alt> Keys: Dimensions Modeling Ribbon Bar

Preset <Alt> Keys: Analyze Ribbon Bar

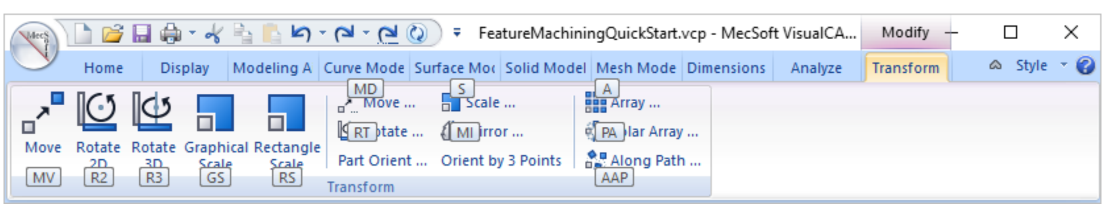
The following preset <Alt> Accelerator key combinations are assigned to commands on the Analyze Ribbon Bar.



Preset <Alt> Keys: Analyze Ribbon Bar

Preset <Alt> Keys: Transform Ribbon Bar

The following preset <Alt> Accelerator key combinations are assigned to commands on the Transform Ribbon Bar.



Preset <Alt> Keys: Transform Ribbon Bar

Related Topics

Customize VisualCAD

Customize Quick Access Toolbar

Minimize Ribbon Bar

Customize Dialog

Customize the Keyboard

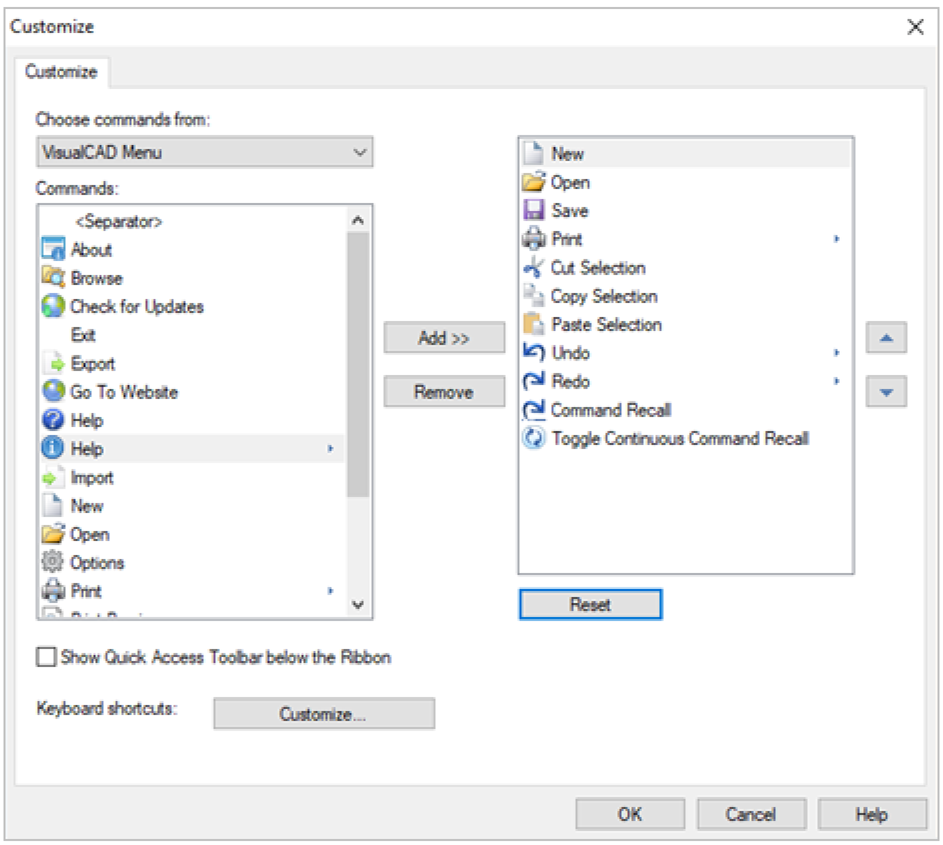
Styles Menu

1.7.4 Customize Dialog

The Customize dialog shown below allows you to add new command icons to the Quick Access Toolbar. From this dialog you can also Customize the Keyboard to execute VisualCAD commands quickly. Refer to the following topics for information.

Dialog Box: Customize

In the dialog shown below, the command icons currently assigned to the Quick Access Toolbar are listed on the right. The command icons from the VisualCAD Menu and each Ribbon bar are shown on the left.



Dialog Box: Customize

Choose commands from:

Select a Ribbon bar from the drop-down list. The commands on the selected Ribbon will be listed in the Commands: window.

Commands:

This window lists all of the commands on the Ribbon bar selected from the Choose commands from: list. Select a command from the list and then pick the Add >> button to add it to the list on the right.

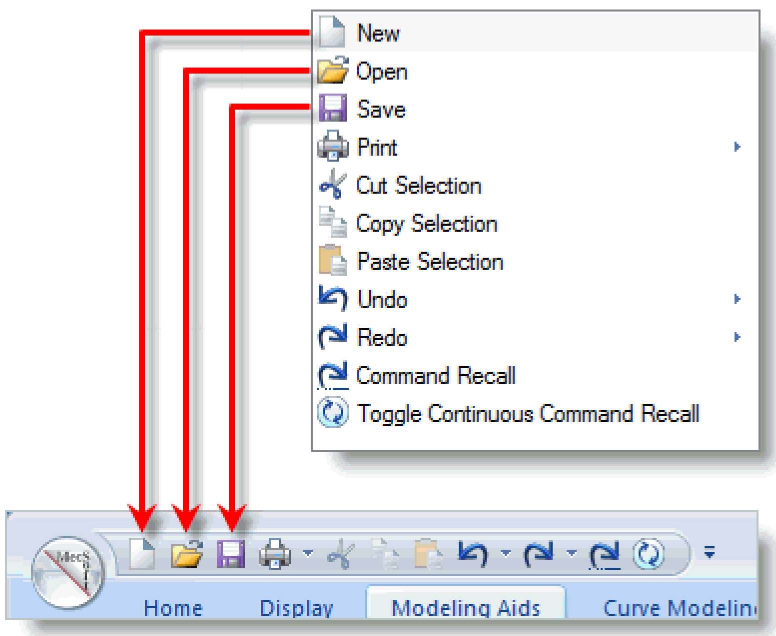
Add >> / Remove

The Add >> button adds a selected command from the Commands: list on the left to the list on the right. The Remove button removes a selected command from the list on the right.

Selected Commands List

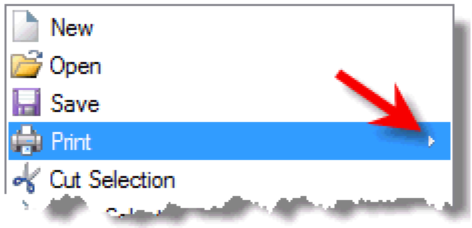
The list on the right shows all of the commands that are currently being displayed on the Quick Access Toolbar. The order of the commands in the list determines the order of the command icons in on the Quick Access Toolbar.

Customize Dialog



Quick Access Toolbar

The icon ‣ to the left of a command in the list indicates that this command has a sub menu that will also be included on the Quick Access Toolbar.



Submenu Indicator

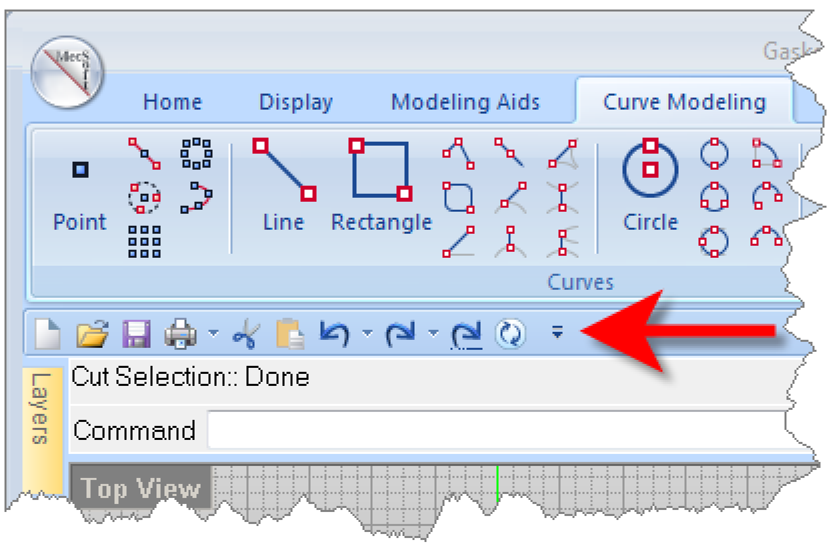
 Pick the Up or Down buttons to move a selected command higher (to the left) or lower (to the right) on the Quick Access Toolbar.

Reset

Select the Reset button to return the list on the right to the Quick Access Toolbar's default list of commands.

Show Quick Access Toolbar below Ribbon

Checking this box will force the Quick Access Toolbar to display below the Ribbon bar. You can also do this by selecting Show Below the Ribbon from the Customize Quick Access Toolbar menu.



Show the Quick Access Toolbat below the Ribbon Bar

Keyboard Shortcuts / Customize

Select the Customize... button to display the Customize Keyboard dialog. From this dialog you can then reassign shortcut keys to any command on the various Ribbon bars.

Related Topics

Customize VisualCAD

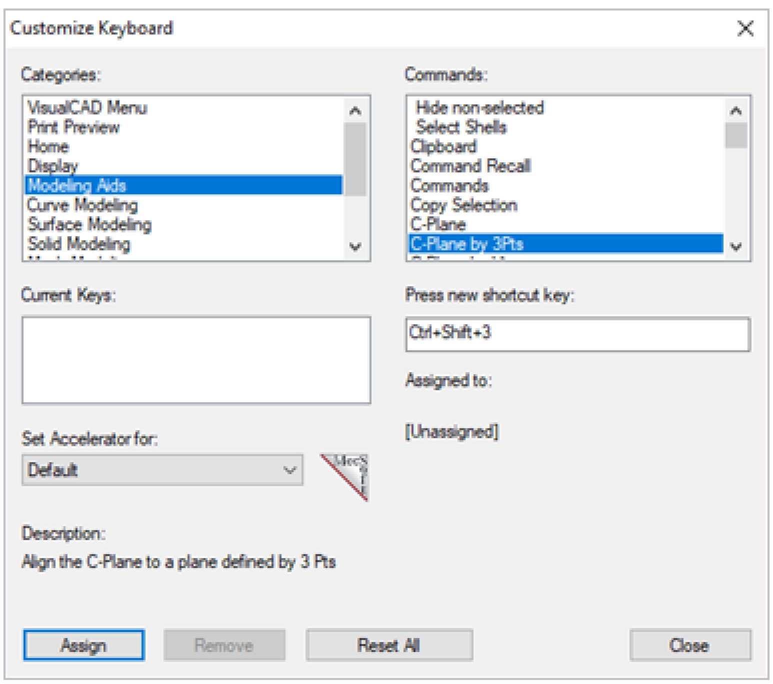
1.7.5 Customize the Keyboard

Invoked by the Customize dialog, the Customize Keyboard dialog shown below allows you to redefine the Default Shortcut Keys assigned to your keyboard. Keyboard shortcuts are combinations of keys that can be pressed to execute a VisualCAD command. Pressing the shortcut keys does the same thing as moving the cursor to the toolbar / menu item and selecting the command, thus saving time and promotes the continuous flow of your work.

Keyboard shortcuts are also called hot-keys.

Dialog Box: Customize Keyboard

In the dialog shown below, the C-Plane by 3 Points command located on the Modeling Aids Ribbon bar is being assigned to the keyboard short keys Ctrl+Shift+3. The dialog also informs you that those keyboard shortcut keys have not previously been assigned. Each section of the dialog is described in detail below.



Dialog Box: Customize Keyboard

Categories:

This section lists all of the individual panes on each ribbon bars as well as each ribbon bar. Select a category and the commands located on that ribbon bar/pane will be listed on the right within the Commands list.

Commands:

This section lists all of the commands located on the Ribbon bar or Pane selected from the Categories section on the left. Select a command from the list to assign a new shortcut keys or to reassign its current shortcut keys.

Current Keys:

If the command selected from the Commands list has a shortcut key assigned to it, it will be displayed in the Current Keys: windows. This window is for information purposes and cannot be edited.

Press new shortcut key:

After selecting a command from the Commands:, place the mouse cursor in this field to activate it. Then from the keyboard, press the shortcut keys that you want to assign to the command.

> If the keyboard shortcut keys are currently assigned to another command, that command is listed under the Assigned to: section directly below this field. * If you pick the Assign button, the shortcut keys will be reassigned!

Assigned to:

If the keyboard shortcut keys you entered in the Press new shortcut key: fields are currently assigned to another command, that command is listed under the Assigned to: section.

Set Accelerator for:

This sets the keyboard accelerator. Currently only Default is available on the list which sets the accelerator for VisualCAD.

Description:

After you select a command from the Commands: list a description of the command is shown under this section. Refer to the example dialog shown above as a reference.

Assign / Remove / Reset All

Once you are certain about the selected command and the new shortcut keys, pick the Assign button to associate them. VisualCAD will then execute the command when those shortcut keys are pressed.

Related Topics

[Customize VisualCAD]

[Customize Quick Access Toolbar]

[Minimize Ribbon Bar]

[Preset <Alt> Accelerator Keys]

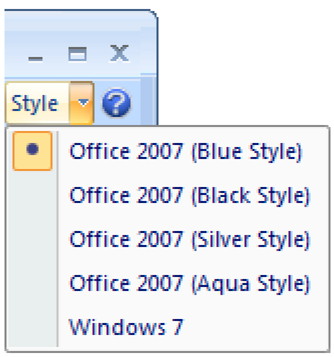
[Customize Dialog]

[Styles Menu]

1.7.6 Styles Menu

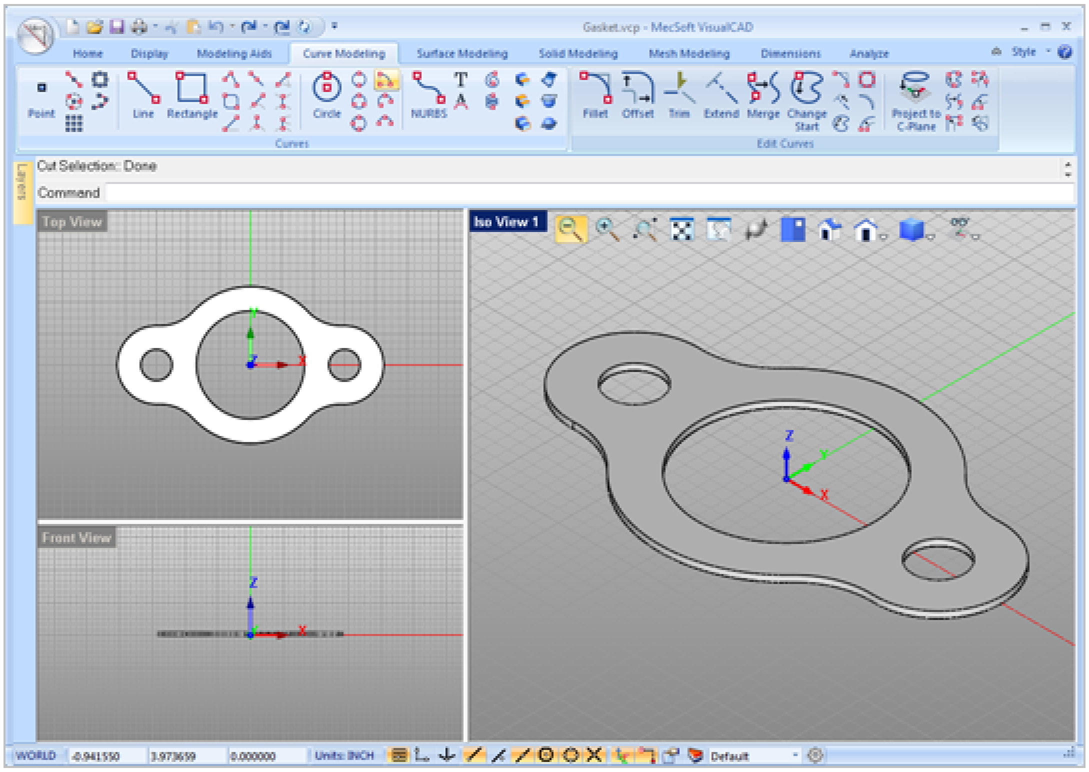
You can select different themes to change how the windows appear. The borders, colors, highlighting, and shadowing of standard buttons, dialogs, and windows are controlled by which theme is selected.

The Styles Menu



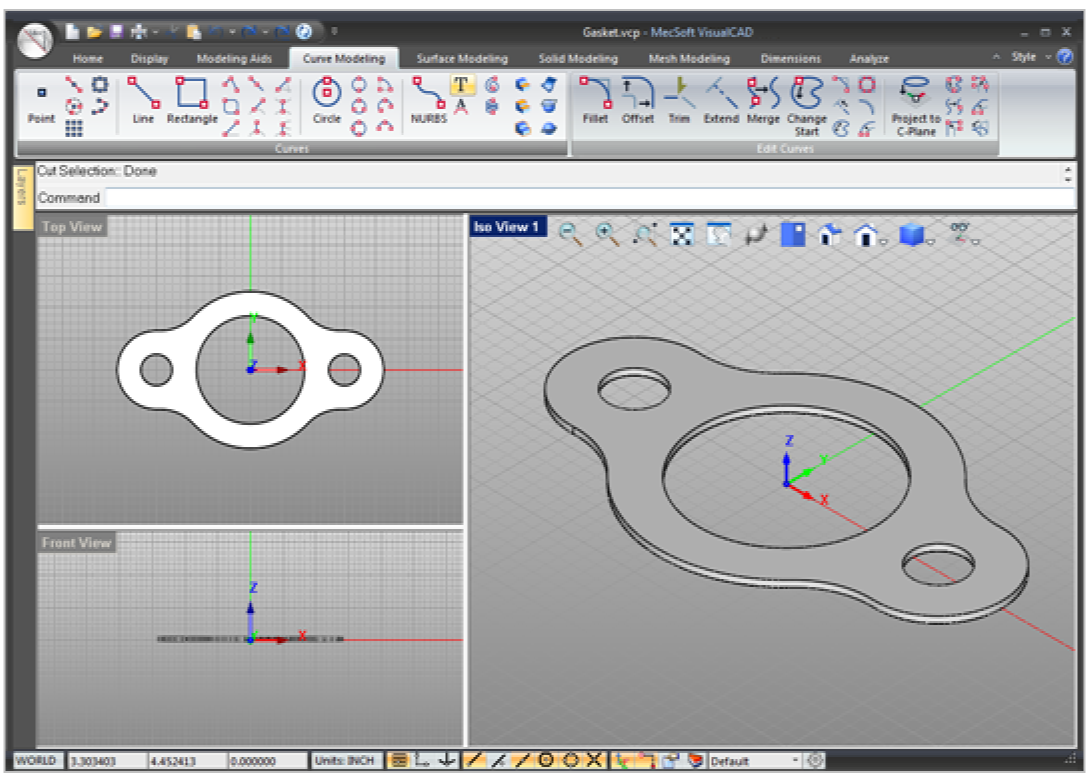
Office 2007 Blue

Office 2007 (Blue Style)



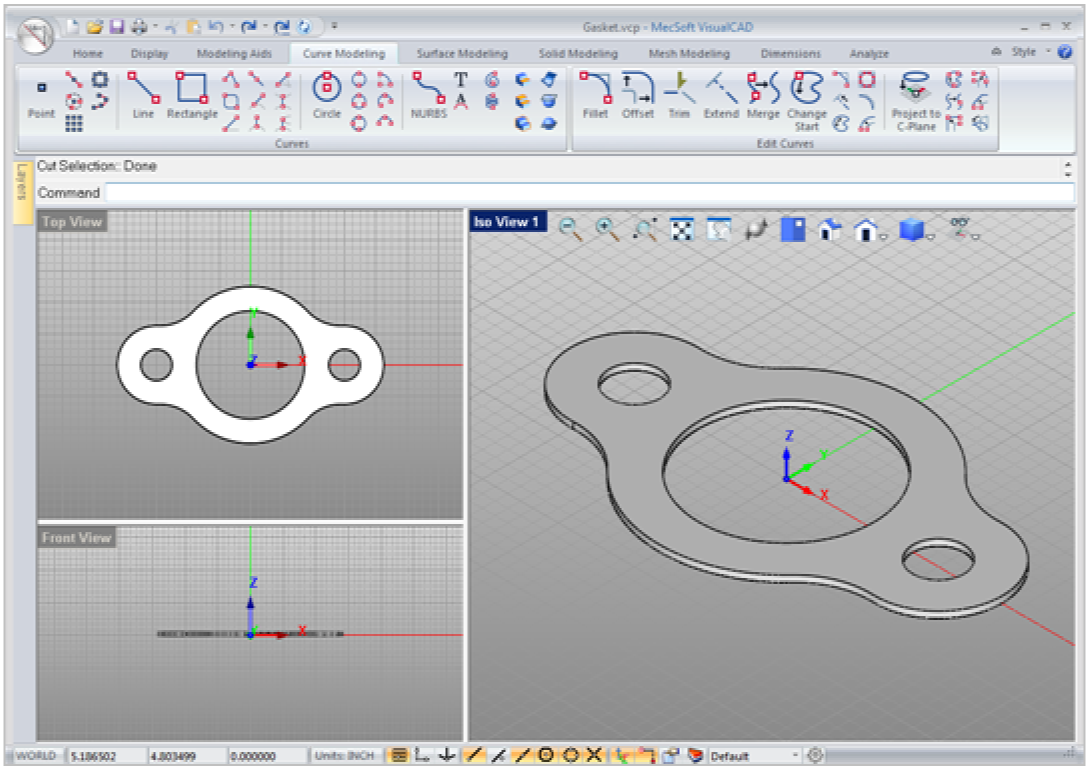
Office 2007 (Blue Style)

Office 2007 (Black Style)



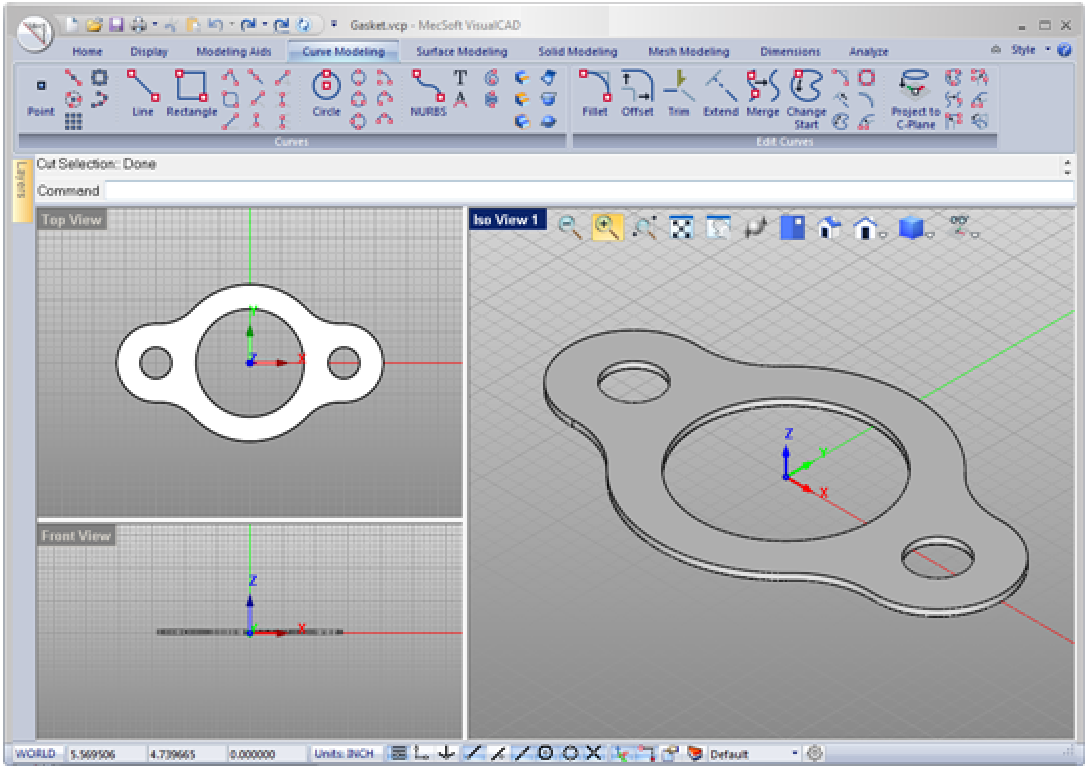
Office 2007 (Black Style)

Office 2007 (Silver Style)



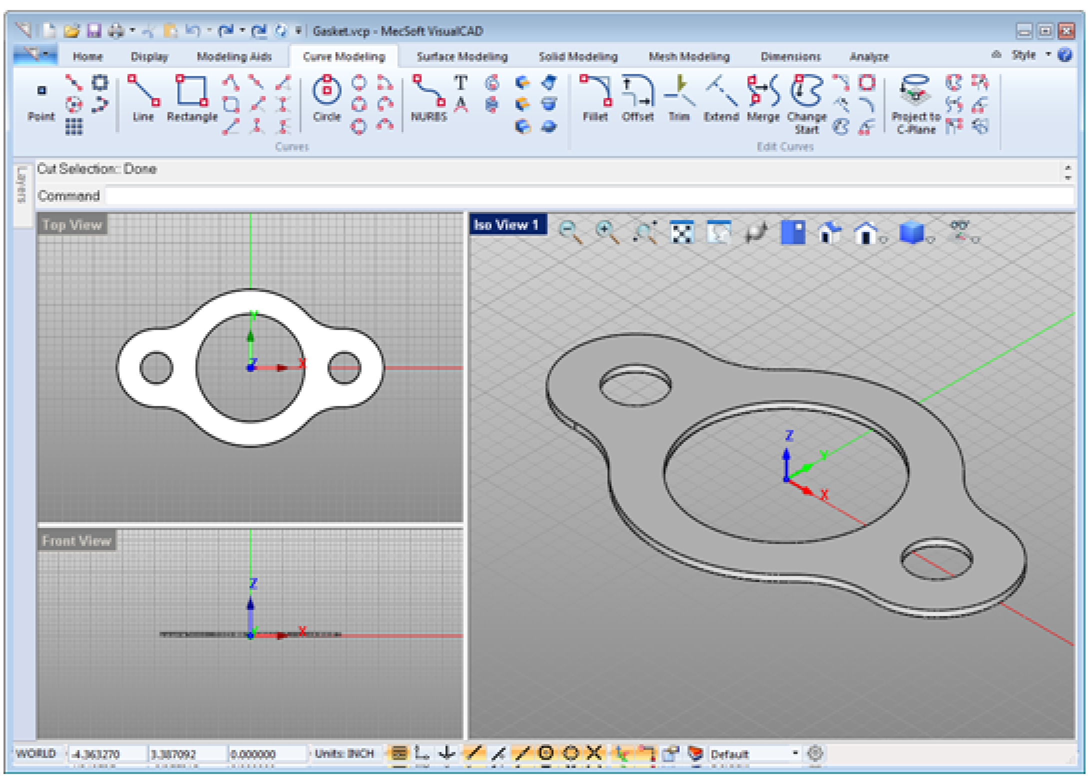
Office 2007 (Silver Style)

Office 2007 (Aqua Style)



Office 2007 (Aqua Style)

Windows 7



Windows 7

Related Topics

[Customize VisualCAD]

[Customize Quick Access Toolbar]

[Minimize Ribbon Bar]

[Preset <Alt> Accelerator Keys]

[Customize Dialog]

[Customize the Keyboard]

Quick Start

MecSoft Corporation
Your CAM Partner

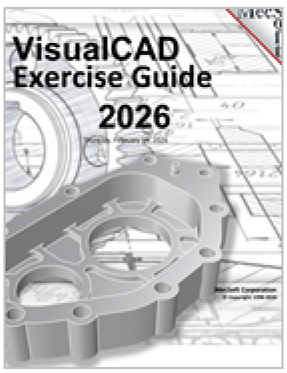
VisualCAD 2026

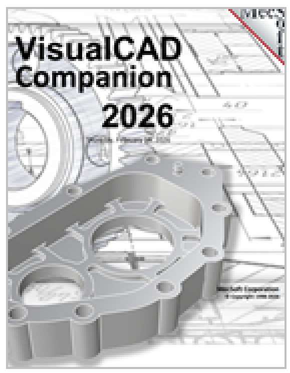
Prefer Printed Documentation? Click Here!

What's New | Quick Start Play List

Quick Start Guides for each VisualCAD module are available in both PDF and Video format. Refer to the following information to access these resources:

Training Guides (Click on Image to Download)





What's New!

What's New in VisualCAD/CAM 2026

The Complete Quick Start Video Play List

Here is a link to the complete 2026 Video Play List

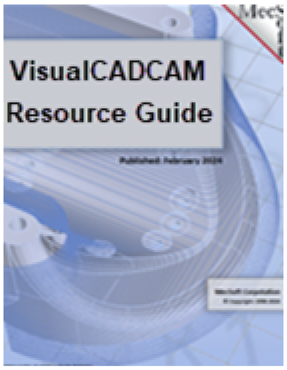
Related Topics

Find More Resources

Resource Guide

Download this PDF Guide for a list of the available VisualCAD Resources.

2025 VisualCAD Resource Guide



The 2026 VisualCAD Resource Guide!

18 Pages

Lists PDF downloads and Online resources including Quick Start Guides, Reference Guides, Exercise Guides, Tutorials and More.

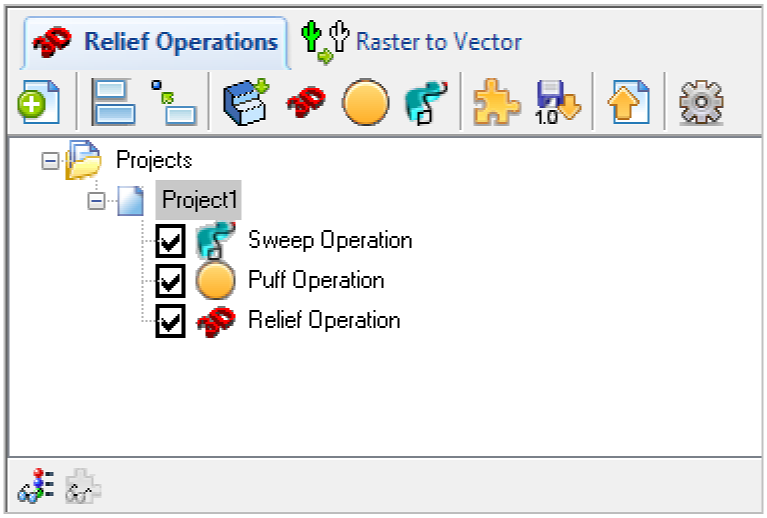
Prefer Printed Documentation? Click Here!

What's New | Quick Start Play List

VisualCAD Menu

The VisualCAD Menu controls for manipulating files, printing, and exiting the program. The most recently opened VCP file names are listed for easy access. If one of those names is selected, that file will be loaded. All current work will be overwritten by the loaded file.

Show me the File Menu



File Menu

Related Topics

- New
- Open ...
- Browse ...
- Import ...
- Save
- Save As ...
- Export ...
- Print ...
  - Print Preview
  - Print Setup ...
- Options ...
- Help & Info
  - Help Topics
  - Check for Updates

[Go To Website]

[About VisualCAD]

[Exit]

4.1 New

Create a new VCP (VisualCAD Part) file. The file holds the geometry, layers, orientation, and preferences. If there are unsaved objects, a dialog will ask you if they want to save them before creating the new VCP file. All current work will be overwritten by the new file.

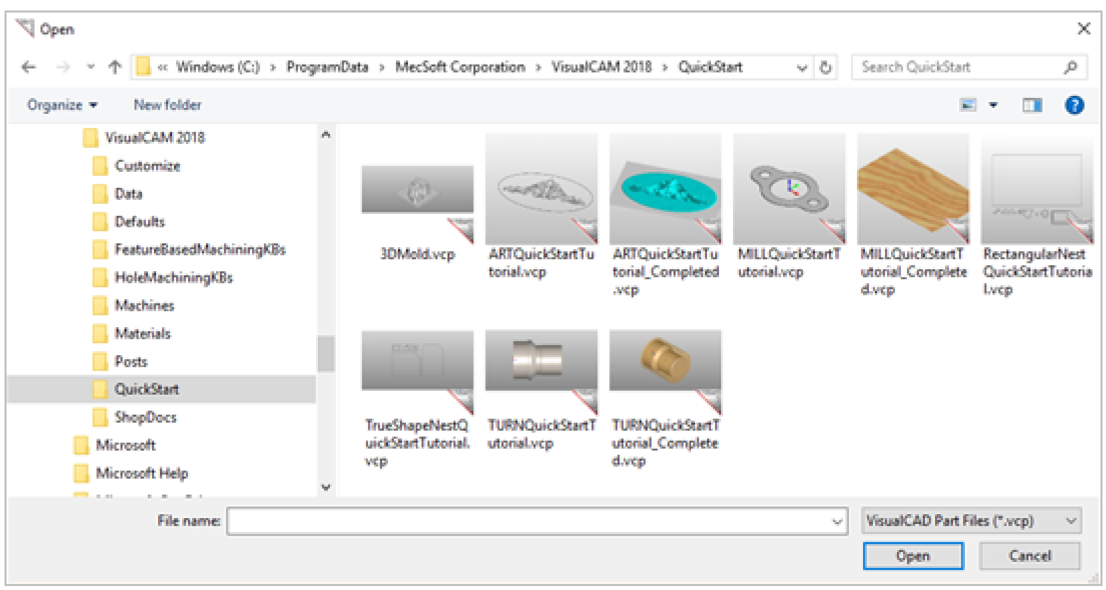
Related Topics

[File]

4.2 Open ...

Open an existing VCP (VisualCAD Part) file or one of the supported file types. The file holds the geometry, layers, orientation, and preferences. If there are existing unsaved objects, a dialog will ask you if you want to save them before opening the VCP file. All current work will be overwritten by the loaded file.

Dialog Box: Open



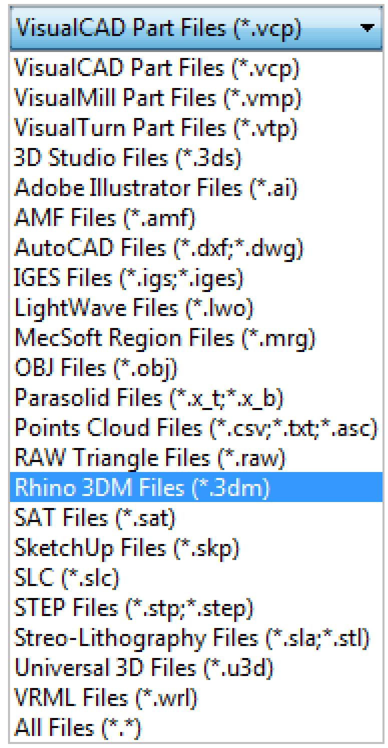
Dialog Box: Open

Standard Windows® File Browser supports preview of VisualCAD *.vcp drawings and models.

Files of Type Supported

The following file types are supported by VisualCAD. See File Open and File Import for more information

- VisualCAD Part Files (*.vcp)
- VisualMILL Part Files (*.vmp)
- VisualTurn Part Files (*.vct)
- 3D Studio Files (*.3ds)
- Adobe Illustrator Files (*.ai)
- AMF Files (*.amf)
- AutoCAD (*.dxf;*.dwg)
- IGES Files (*.igs;*.iges)
- LightWave Files (*.lwo)
- MecSoft Region Files (*.mrg)
- OBJ Files (*.obj)
- Parasolid Files (*.x_t;*.x_b)
- Point Cloud Files (*.cvs;*.txt;*.asc)
- RAW Triangle Files (*.raw)
- RHINO 3DM Files (*.3dm)
- SAT Files (*.sat)
- SketchUP Files (*.skp)
- SLC Files (*.slc)
- STEP Files (*.stp;*.step)
- Stereo-lithography files (*.sla;*.stl)
- Universal 3D Files (*.u3d)
- VRML Files (*.wrl)



File Types Supported

Related Topics

File

Import ...

Save

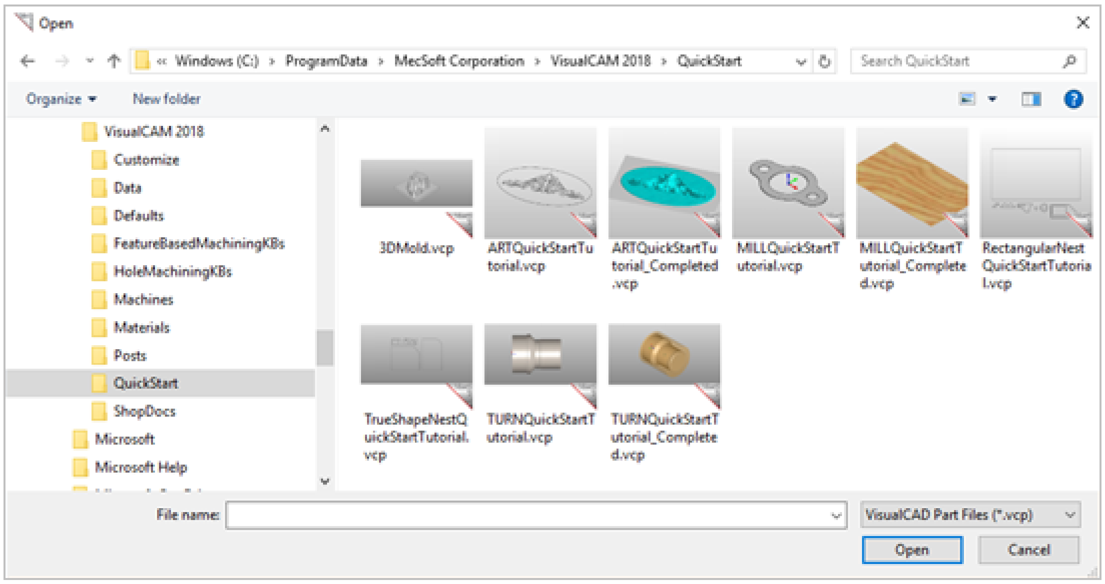
Save As ...

Browse...

4.3 Browse ...

Visually browse files on the computer. VCP files can be opened. A preview window will show the last view displayed when the original file was saved.

Dialog Box: VisualCAD® File Browser



Dialog Box: VisualCAD® File Browser

Related Topics

[File Menu](#)

[Import ...](#)

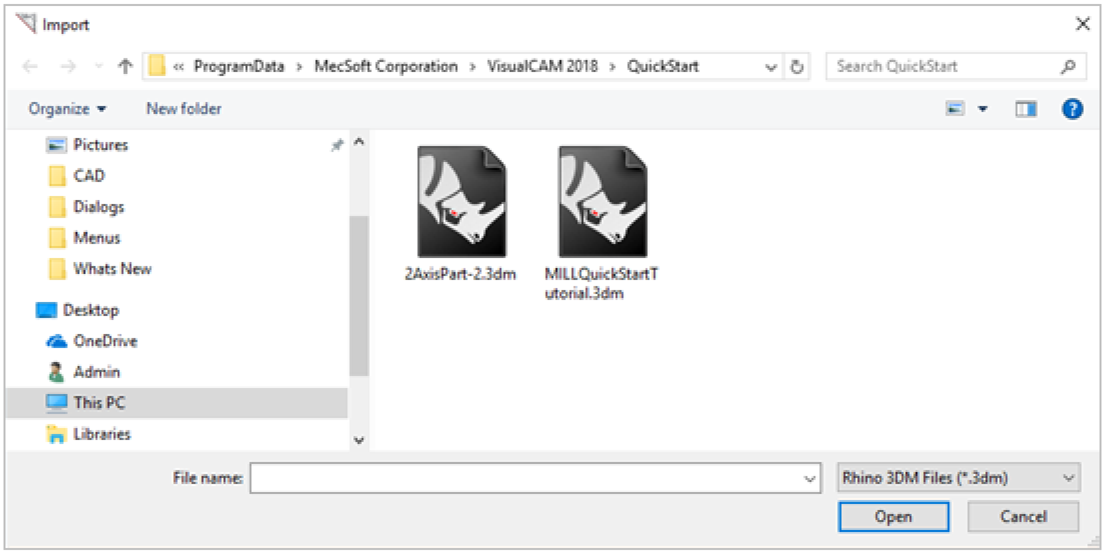
[Save](#)

[Save As ...](#)

4.4 Import ...

Open existing files made by other programs. This is used to transfer information, normally geometric models. Currently, VisualCAD can import the following formats:

Dialog Box: Import

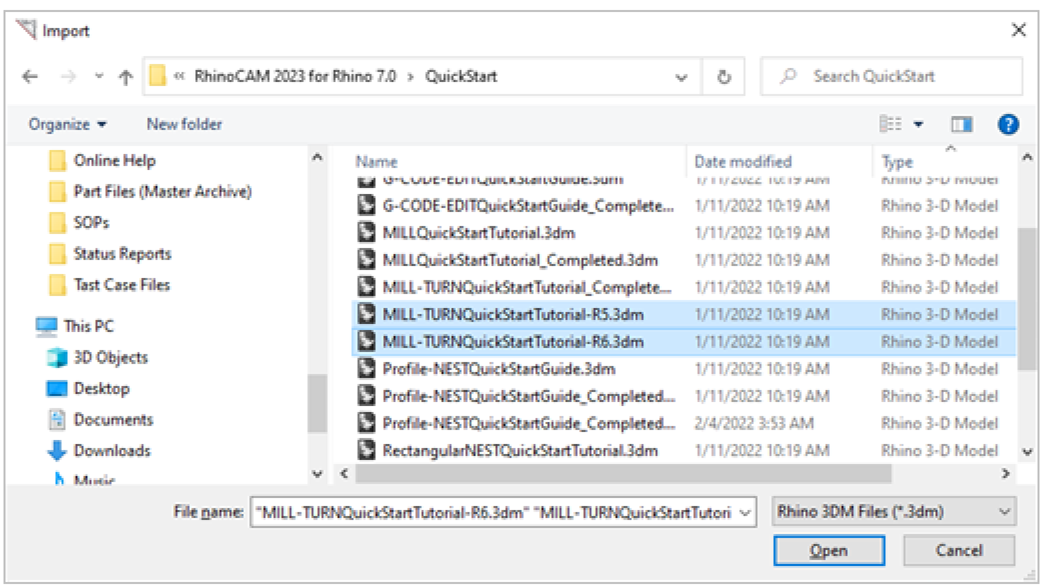


Dialog Box: Import

Standard Windows® File Browser supports preview of VisualCAD *.vcp drawings and models.

Import Multiple Files

You can Import multiple files at the same time. To do so, first press and hold the <Ctrl> key while selecting the files from the Import dialog. Each file will highlight and also be listed together in the File name field at the bottom of the dialog. When done selecting files, pick the Open button.



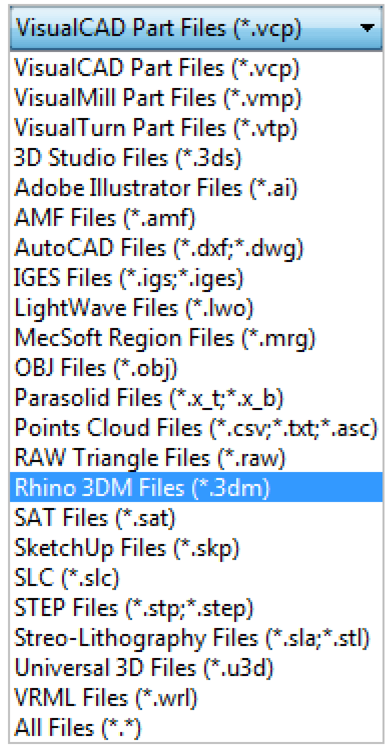
Dialog Box: Import (When importing multiple files at once)

Standard Windows® File Browser supports preview of VisualCAD *.vcp drawings and models.

Files of Type Supported

The following file types are supported by VisualCAD. See File Open and File Import for more information

- VisualCAD Part Files (*.vcp)
- VisualMILL Part Files (*.vmp)
- VisualTurn Part Files (*.vct)
- 3D Studio Files (*.3ds)
- Adobe Illustrator Files (*.ai)
- AMF Files (*.amf)
- AutoCAD (*.dxf;*.dwg)
- IGES Files (*.igs;*.iges)
- LightWave Files (*.lwo)
- MecSoft Region Files (*.mrg)
- OBJ Files (*.obj)
- Parasolid Files (*.x_t;*.x_b)
- Point Cloud Files (*.cvs;*.txt;*.asc)
- RAW Triangle Files (*.raw)
- RHINO 3DM Files (*.3dm)
- SAT Files (*.sat)
- SketchUP Files (*.skp)
- SLC Files (*.slc)
- STEP Files (*.stp;*.step)
- Stereo-lithography files (*.sla;*.stl)
- Universal 3D Files (*.u3d)
- VRML Files (*.wrl)



File Types Supported

Related Topics

File

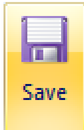
Export

Save

Save As

Browse...

4.5 Save

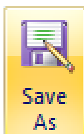


Save geometry, layers, orientation, and preferences in a VCP file. If saving for the first time, you can name the file and select where on the computer to save the file. If the file had been saved previously, the current information overwrites the original information.

Related Topics

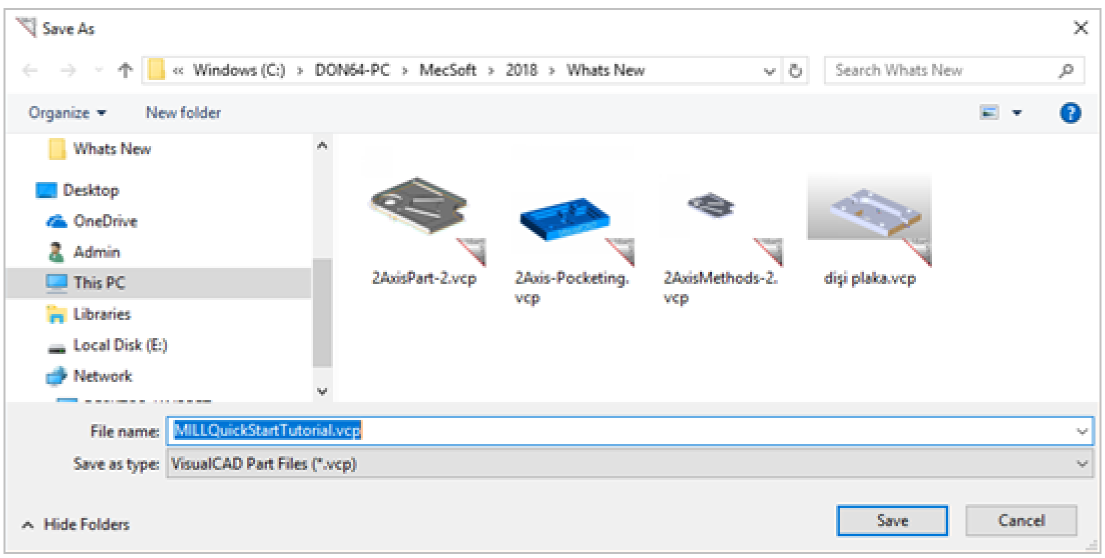
File

4.6 Save As ...



Save geometry, layers, orientation, and preferences in a VCP file. Name the file and select where on the computer to save it. If selecting an existing file, the current information will overwrite the original information. See Export ... to save as different file types.

File Save As dialog box



File Save As dialog box

Save as Files Types Supported

- VisualCAD Part Files (*.vcp)

Related Topics

File

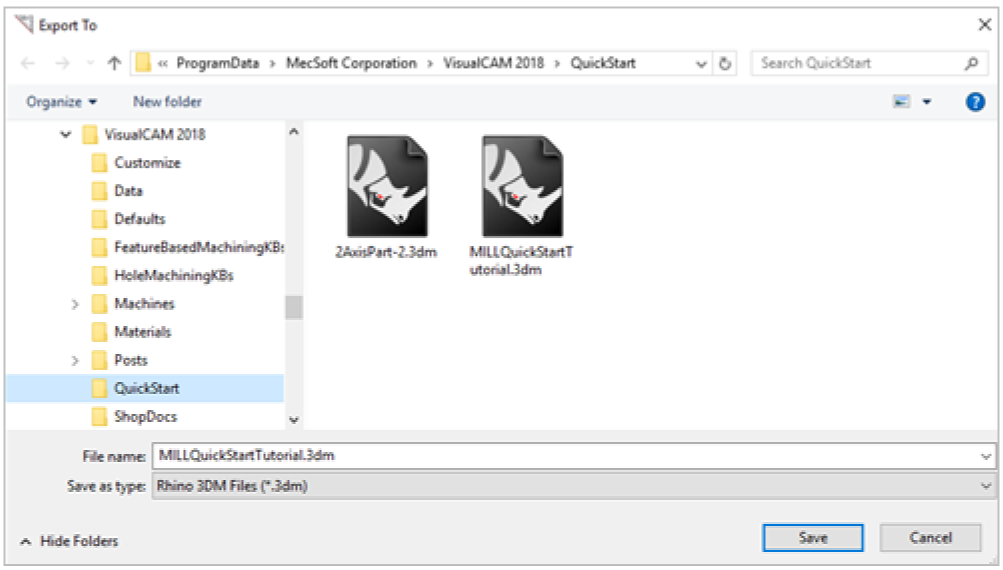
4.7 Save As Template

Use this to save VisualCAD® preferences (System, Display, Color, Grid, Lighting, Dimension, Tolerance and Units) in the **.vcp* file. This template file can then be used to begin new VisualCAD drawings.

4.8 Export ...

Save geometry in a neutral file format (can be read by many different programs). The following export translators are available. Binary formats are usually much smaller in size, but ASCII formats are more easily read. Ordinary text programs can read the ASCII files.

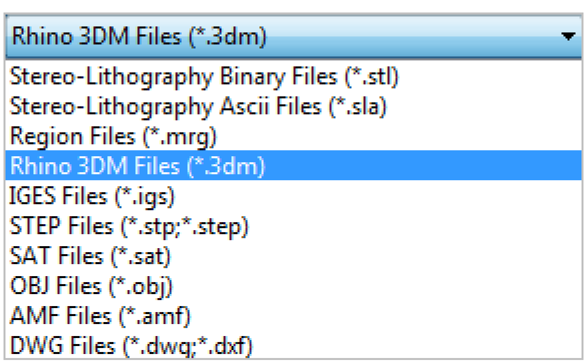
Dialog Box: Export To



Dialog Box: Export To

Export File Types Supported

The following file types are supported by VisualCAD during Export:

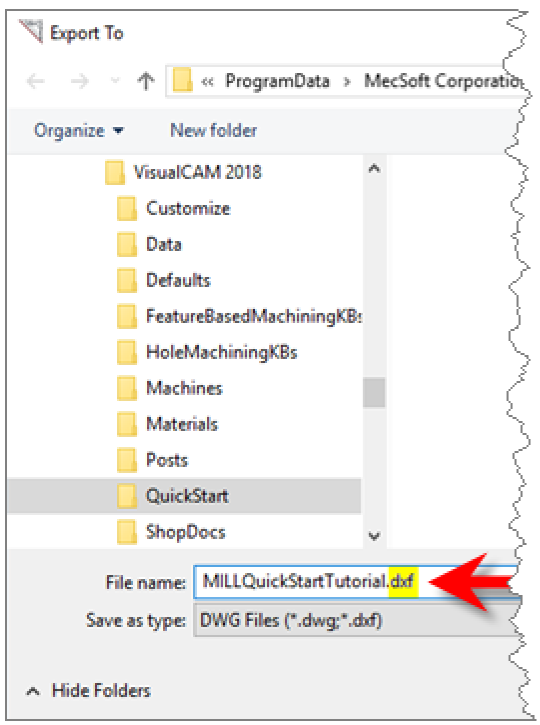


File Types Supported during Export

- Stereo-Lithography Binary (**.stl*)
- Stereo-Lithography ASCII (**.sla*)
- Region Files (**.mrg*)
- Rhino (*.3dm)
- IGES (*.igs)
- STEP Files (**.stp*; **.step*)
- SAT Files (**.sat*)
- OBJ Files (**.obj*)
- AMF Files (*.amf)
- DWG Files *(*.dxf; *.dwg)*

Exporting to Dwg/Dxf formats

When you select the Dwg/Dxf export File Type, the file is exported to *.dwg format by default. If you would like to export the file in *.dxf format, you will need to enter the file extension to the end of the file name (i.e., Example.dxf).



Exporting to the *.dxf File Type

Related Topics

File

Import ...

Save

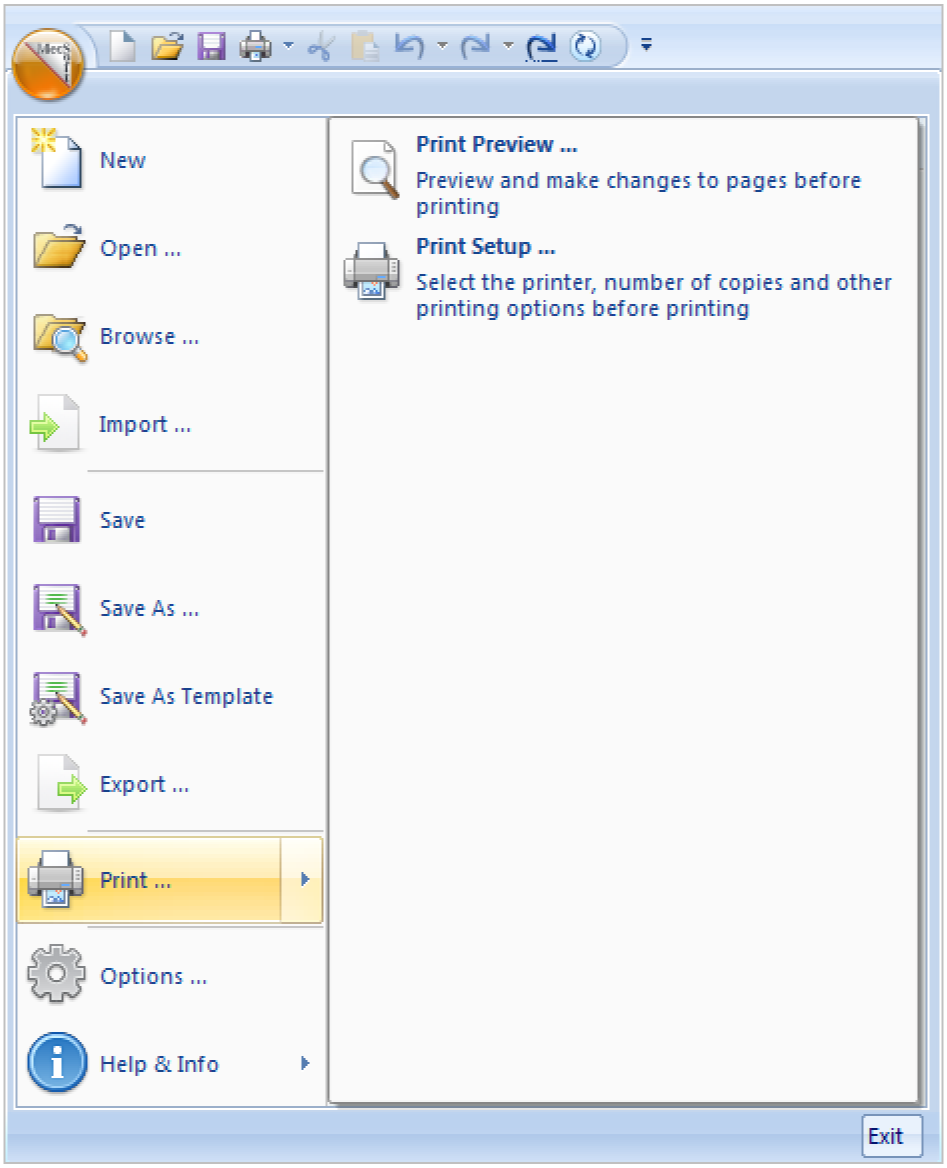
Save As ...

Browse...

4.9 Print Menu

Moving the mouse cursor over the Print ... button from the VisualCAD Menu will display additional print options. You can also select (left-click) on this button to execute the Print command.

VisualCAD Menu > Print ...



VisualCAD® Menu > Print ...

Related Topics

[VisualCAD Menu]

[Print ...]

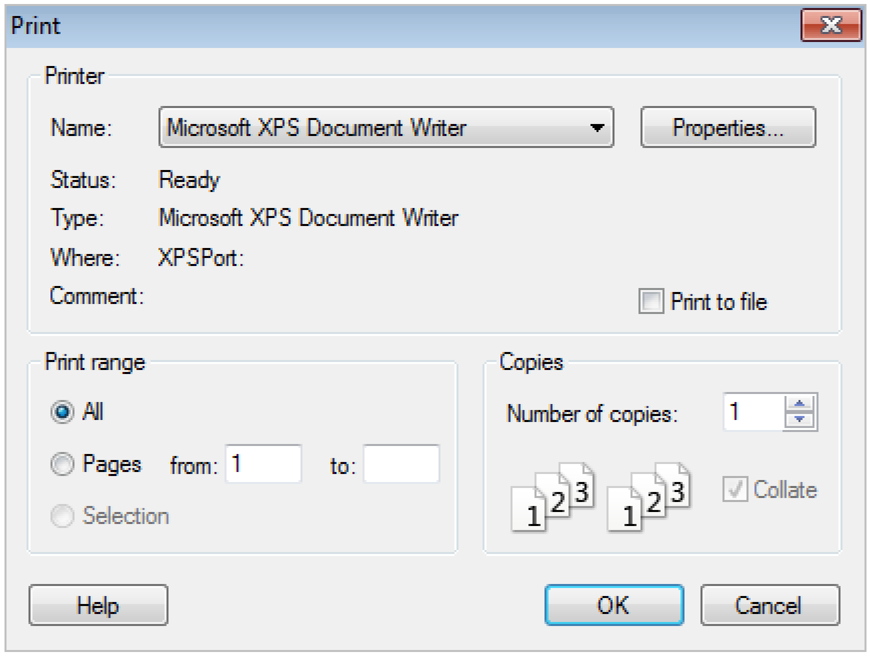
[Print Preview ...]

[Print Setup ...]

4.9.1 Print

Prints the objects displayed. If multiple views are displayed, the currently selected view will be printed. The standard Windows® Print dialog box is displayed.

The standard Windows® Print dialog



The standard Windows® Print dialog

Related Topics

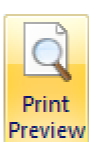
[VisualCAD Menu]

[Print Menu]

[Print Setup ...]

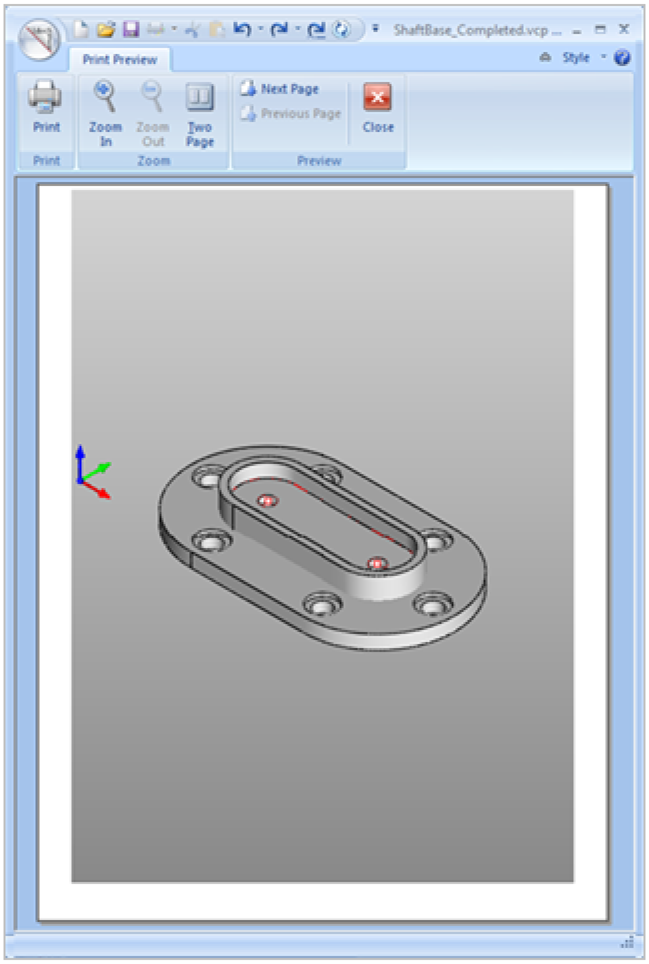
[Print Preview ...]

4.9.2 Print Preview ...



This shows a picture of what will be printed. The picture can be printed from here along with the additional controls listed below.

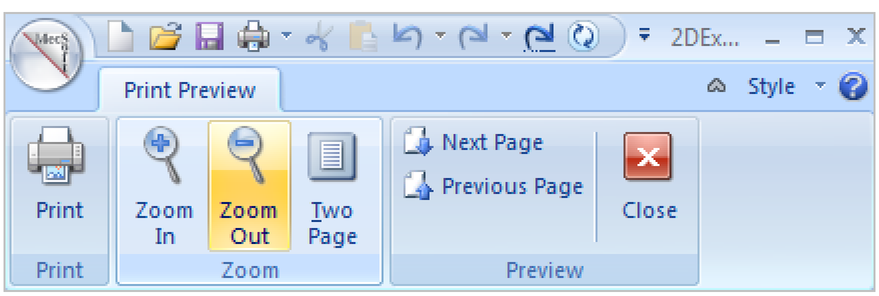
Print Preview Display



File > Print Preview Display

Print Preview Controls

The following controls are available from the Print Preview ribbon bar:



Print Preview Ribbon Bar

- Print
- Zoom In/Out
- Two Page
- Next Page

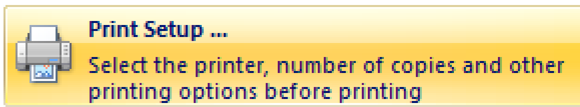
- Prev Page
- Close

Related Topics

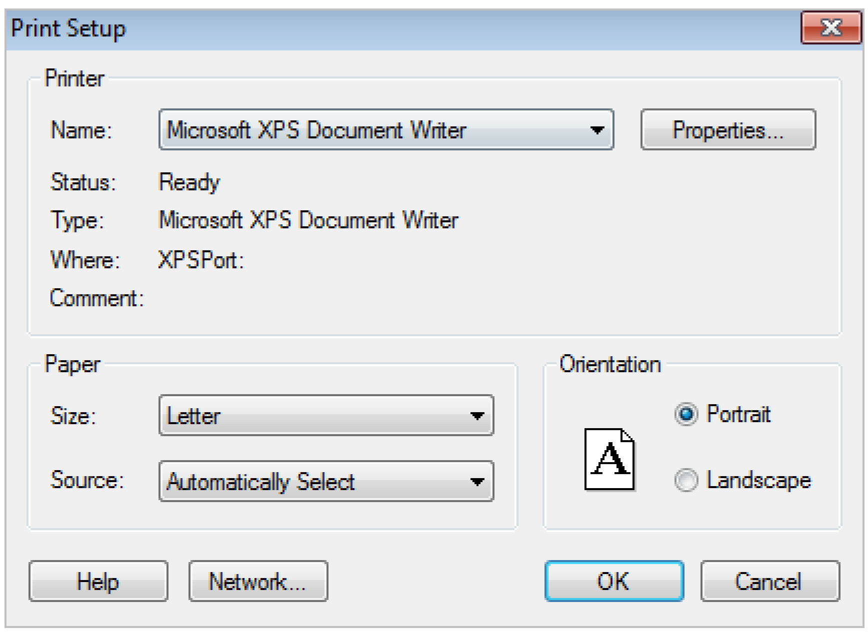
File

4.9.3 Print Setup ...

From the VisualCAD Menu moving the mouse cursor over the Print ... option will display the Print Menu with this Print Setup ... option available. Use this to select the printer used to print. The orientation (landscape or portrait), paper size and additional printer properties can be specified. The standard Windows® Print Setup dialog box is displayed.



The standard Windows® Print Setup dialog



The standard Windows® Print Setup dialog

Related Topics

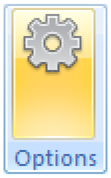
VisualCAD Menu

Print Menu

Print ...

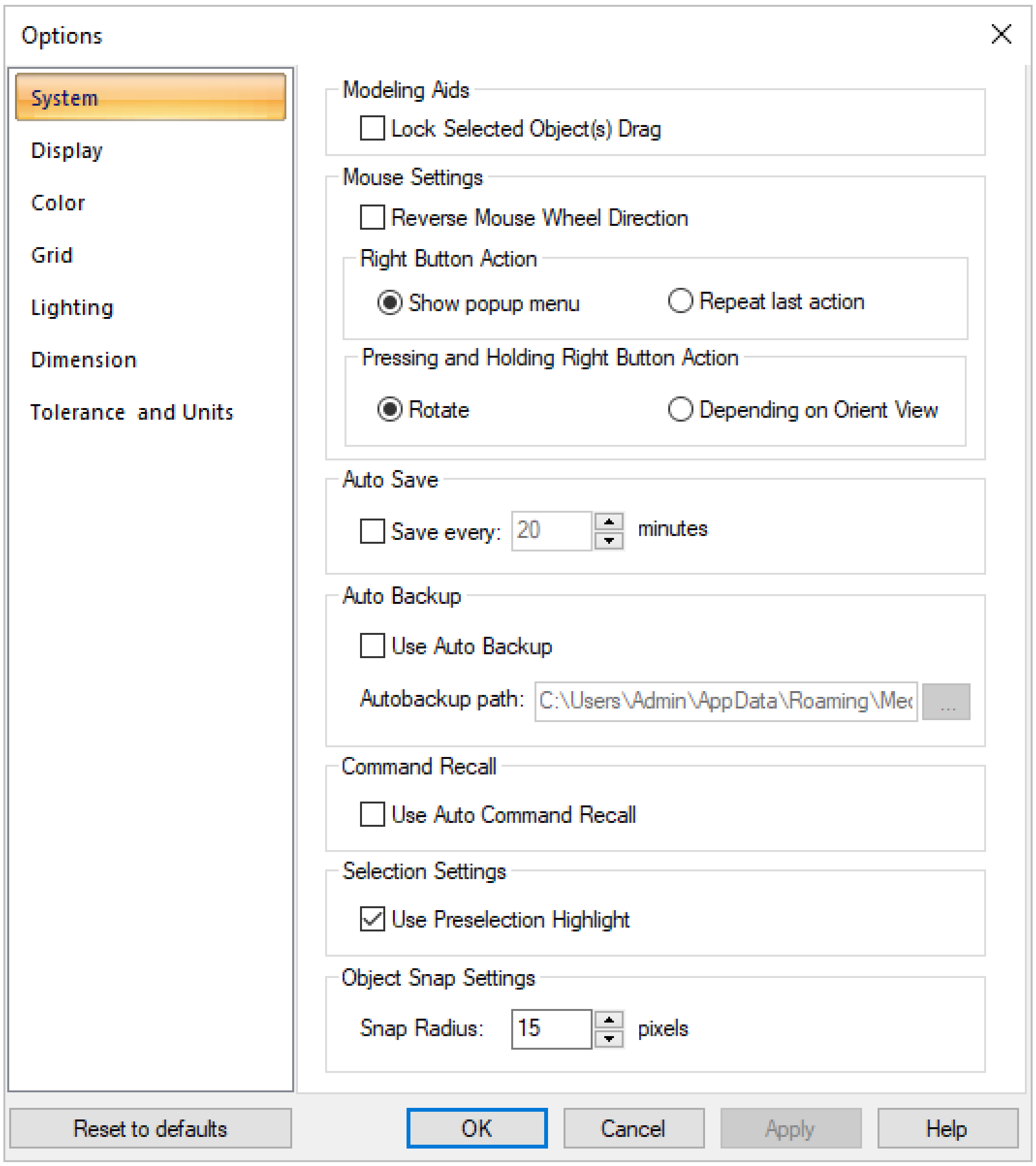
Print Preview ...

4.10 Options



The Home Ribbon Bar contains the Options pane. Selecting the icon will display the Options dialog shown below. This dialog allows you to set global parameters which include System, Display, Color, Grid, Lighting, Dimension, Tolerance and Units preferences. This dialog can also be accessed from the Status Toolbar.

Dialog Box: Options



Dialog Box: Options

Related Topics

System

Display

Color

Grid

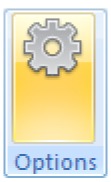
Lighting

Dimension

Tolerance and Units

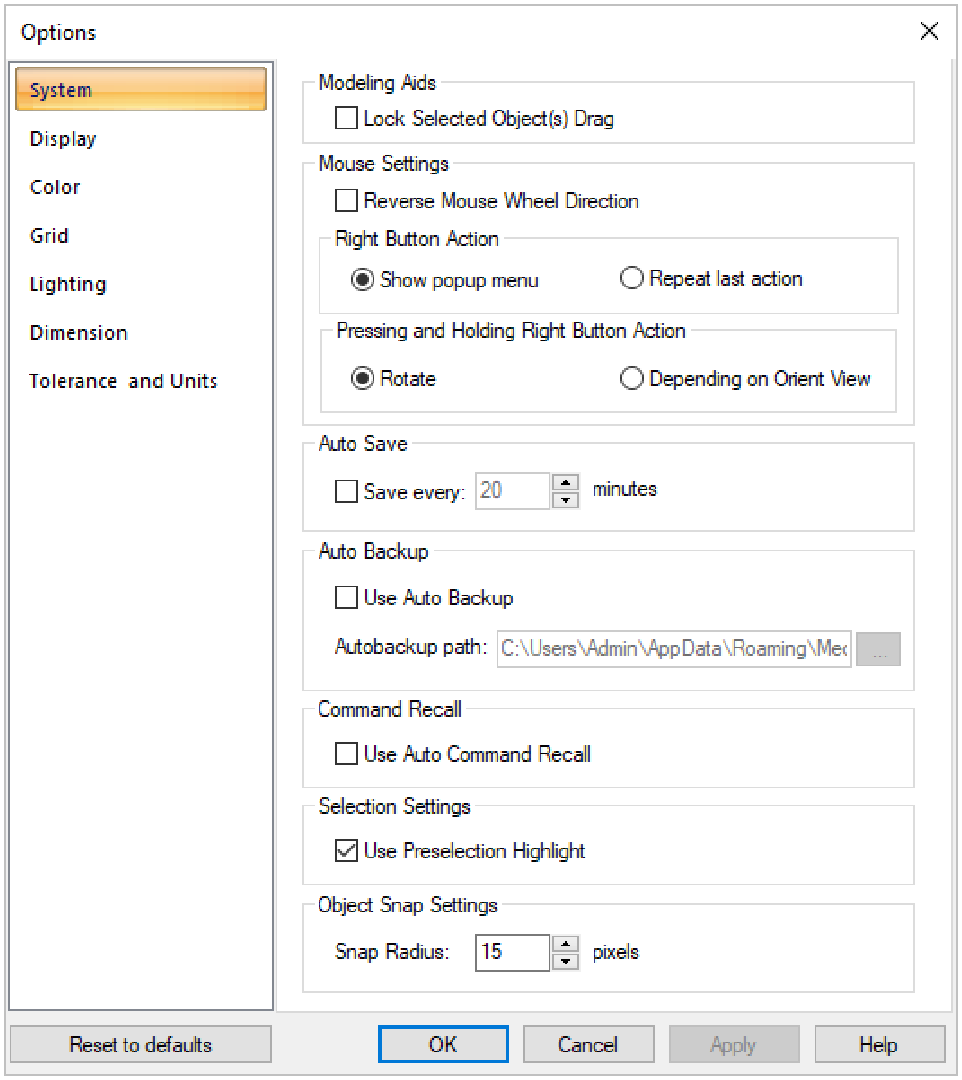
Home Ribbon Bar

4.10.1 System



Allows user to set preferences for modeling aids, mouse settings, auto save and command recall.

Dialog Box: Options > System



Dialog Box: Options > System

Lock Selected Object(s) Drag

If the box is not checked, you can select a geometry object and drag it around the C-Plane. When the box is checked, the objects cannot be dragged.

Mouse Settings

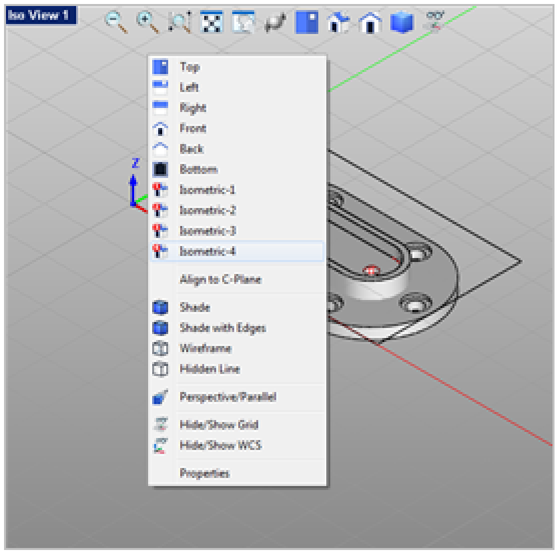
The following Mouse options are supported:

Reverse Mouse Wheel Direction

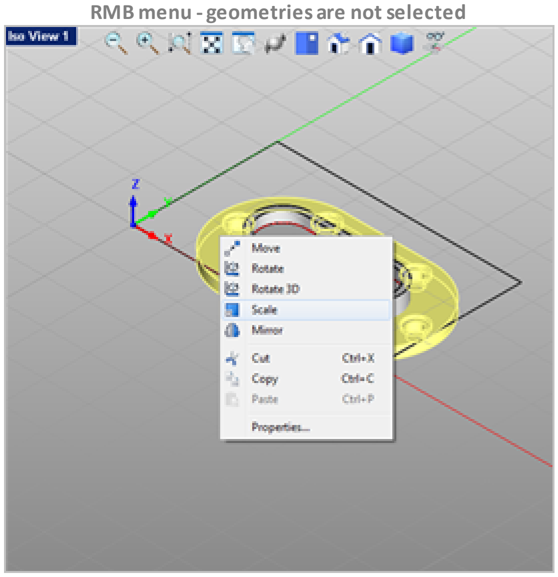
Reverses the direction of view zooming when using the mouse wheel.

Right Button Action

When Show pop-up menu is selected, performing a right mouse button click in the view port displays the commands in the pop-up menu.



RMB menu - geometries are not selected



RMB menu - geometries are selected

When Repeat last action is selected, this recalls the previous command.

Pressing and Holding Right Button Action

This option tells VisualCAD what to do when the right mouse button is pressed and held.

- Rotate
  When selected, pressing and holding down the right mouse button from all views performs a Rotate on the display.

- Depending on Orient View
  When depending on orient view is selected, pressing & holding down the right mouse button from Top, Front, Right, Left, Back, Bottom views does a Dynamic Pan of the geometries instead of rotate. For Iso views (Iso1,2,3 & 4) it performs a Rotate when pressing and holding down right mouse button down.

AutoSave

The AutoSave function can be used to automatically save your active part file at user-specified time intervals. AutoSave always saves to the current file name. This saved file is removed when you successfully close your model.

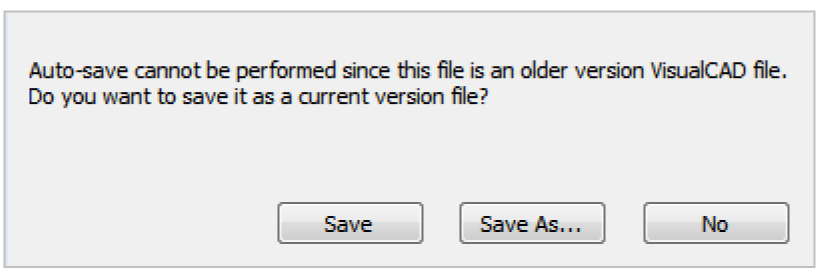
> **NOTE**: Once you activate AutoSave, you will need to Exit VisualCAD and then launch VisualCAD again before AutoSave will begin saving your part files!

Under the AutoSave function, check the box next to 'Save every xx minutes' to activate the AutoSave function. The time interval can be entered in minutes.

> AutoSave saves the file under: *<APPDATA>\MecSoft\VisualCAD 2018\AutoSave* where APPDATA is a logical pointing to a folder set up by your system administrator. By default it points to: *C:\Users\<username>\AppData\Roaming*.

Opening an Older Version *.vcp File

Opening an older version vcp file, displays the following dialog allowing you the opportunity to save the file to the current version.



AutoSave Message

When does AutoSave Prompt me to Save a File?

Under the following conditions VisualCAD will prompt you to save your file:

1. When you have opened a non-native VisualCAD file.
2. When you have imported a file.

3. When you have created a new part and has not saved the file yet.

When is the AutoSave File Deleted?

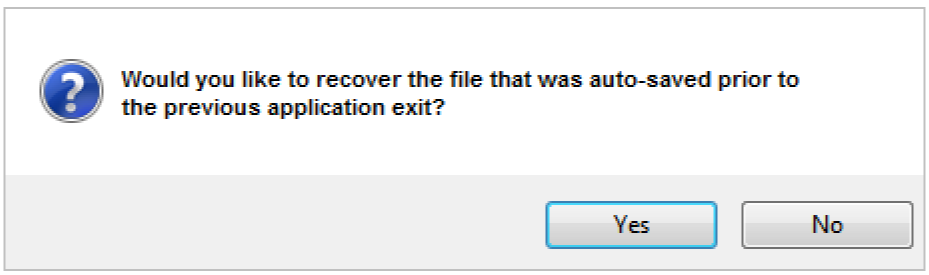
Auto save file is deleted under the following scenarios:

1. When the application closes normally.
2. When you close the file either explicitly or by opening another file.
3. When you select not to recover an AutoSave file that was detected on startup

File Recovery

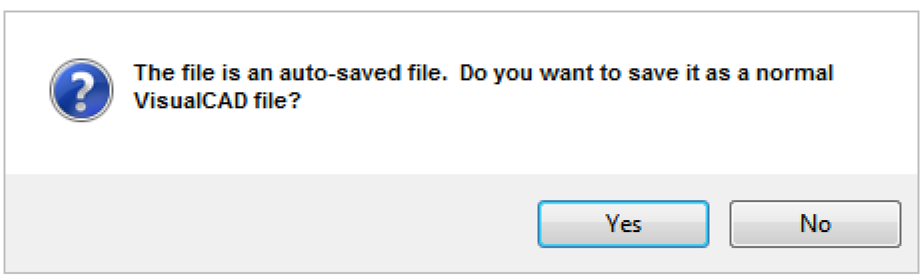
In the event of a computer crash or if VisualCAD is terminated abruptly by ending the process from task manager, VisualCAD will recover your file if AutoSave was active and your part file was automatically saved at least once.

If an AutoSave file has been created prior to the crash or termination, VisualCAD will detect the file located in the AutoSave folder the next time it is loaded. You are then prompted with the following dialog:



AutoSave Message

Selecting Yes, displays the following dialog, prompting you to save your file:



AutoSave Message

> **!** This is the only opportunity to recover the part file. If you choose not to recover it, the AutoSave file will be removed.

Use Auto Backup

Use this option to back up your VisualCAD files automatically. You can change the Auto Backup Path destination directory by selecting the 'browse' button.

If you have the check box checked, then the file is saved ONCE (as soon as the file is loaded) to the Auto Backup path with the same name as the original path file. This file is not deleted on exit so it can be recovered later if necessary. By default the check box to Auto Backup is unchecked.

Use Auto Command Recall

When this option is selected, VisualCAD automatically repeats the last command without having to go back & reselect the command from the toolbar or use the right mouse button click or enter option to repeat the previous command.

Use Preselection Highlight

When this option is selected, VisualCAD automatically highlights selections as the cursor moves over them. See Pre-Selection Highlight for more information.

Object Snap Settings

You can control the pixel radius of the object snaps located on the Status Toolbar. Enter the Snap Radius desired.

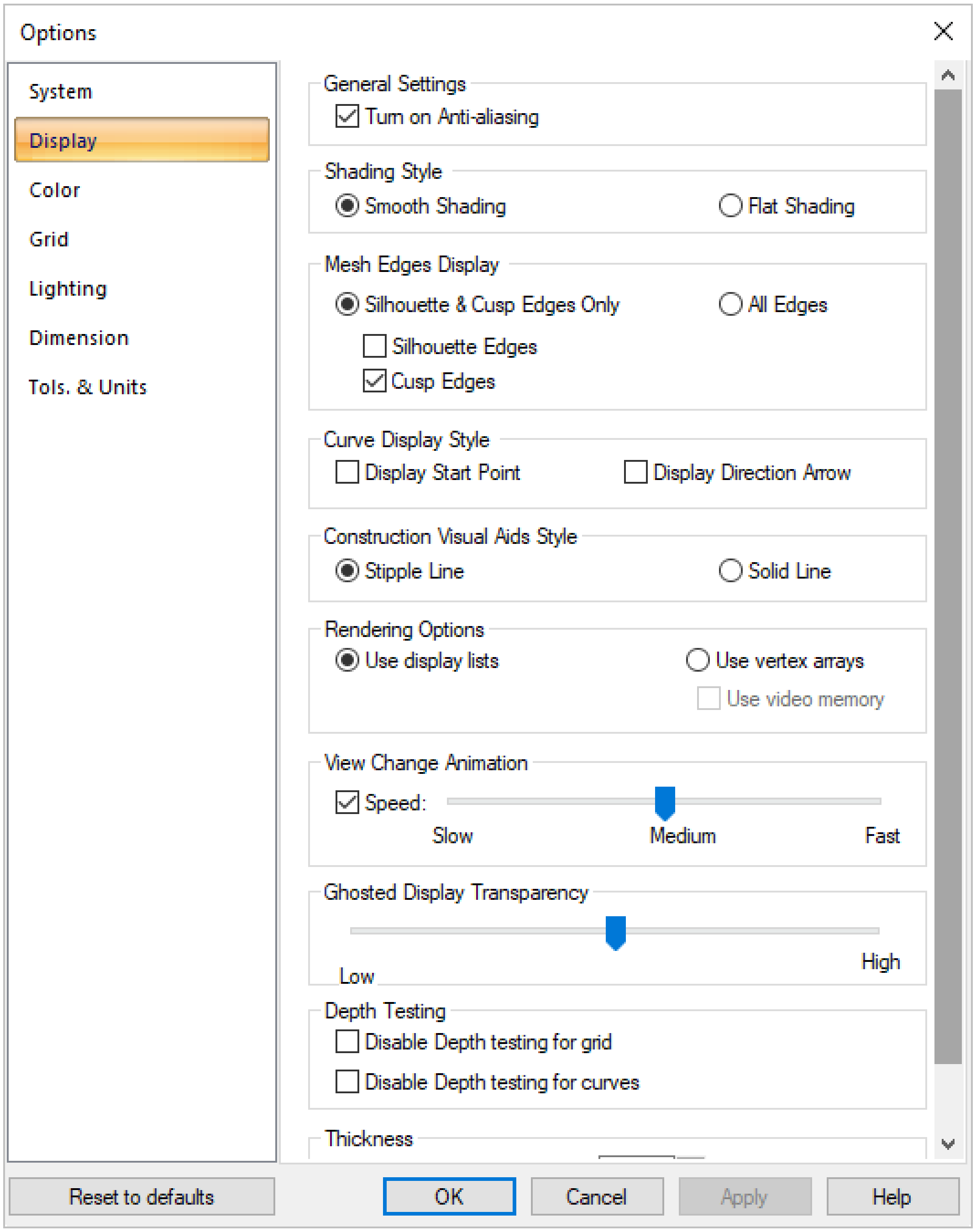
Related Topics

Options

4.10.2 Display

Controls how geometry is displayed. Select from the following:

Dialog Box: Options > Display



Dialog Box: Options > Display

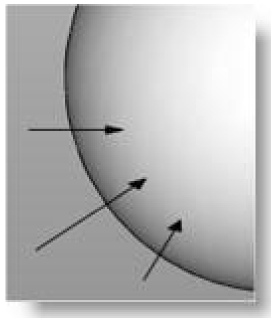
General Settings

Turn on Anti-aliasing

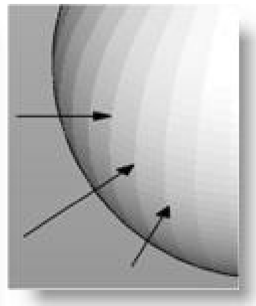
A method of smoothing the jagged edges along the lines and curves of text or graphics. Aliasing is caused by limited display resolution. Selecting this option turns on Anti-aliasing.

Shading Style

This controls how smooth surfaces are shaded.



Smooth Shading

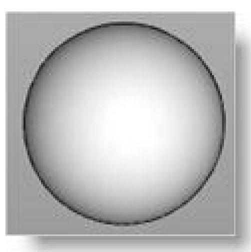


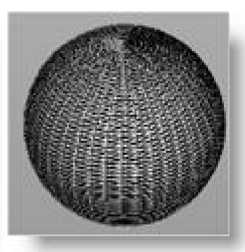
Flat Shading

The pictures demonstrate the difference between smooth and flat shading. The left picture smoothly changes the amount of shading as the surface curves. The right picture just displays the shading for a flat facet on the surface. Since all surfaces (including smooth ones) are internally represented as flat facets, flat shading can affect any surface. The default is smooth shading.

Mesh Edges Display Style

This changes how smooth surfaces are displayed. Since all surfaces are internally represented as flat facets, these facets join up along edges. On smooth surfaces, these edges are not sharp and are not normally displayed. An example of a sharp edge is the edge of a cube.

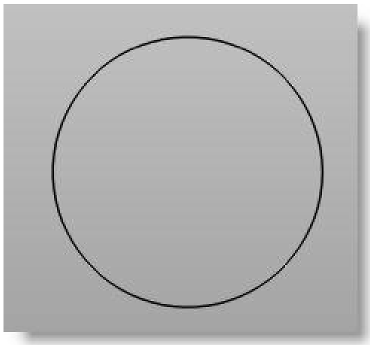




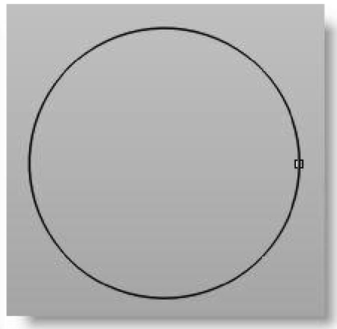
The pictures demonstrate the difference between silhouette & cusp edges only and all edge display. The left picture displays only sharp edges (there are none on a sphere); meanwhile, the right picture displays all the edges. The default is Silhouette & Cusp Edges Only. When selected, this allows you to select Silhouette Edges, Cusp Edges or All Edges to display.

Curve Display Style

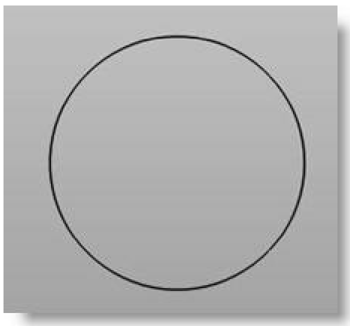
This provides an option to turn on or off display of curve start point and direction arrow.



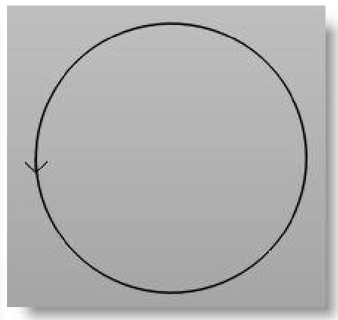
Start Point display OFF



Start Point display ON



Curve Direction arrow OFF

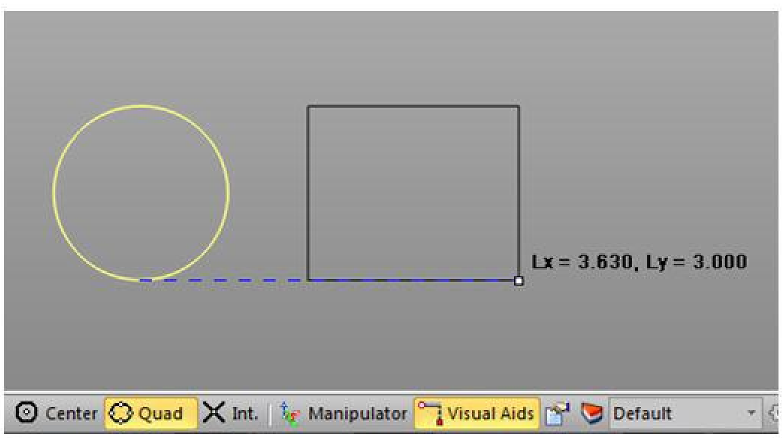


Curve Direction arrow ON

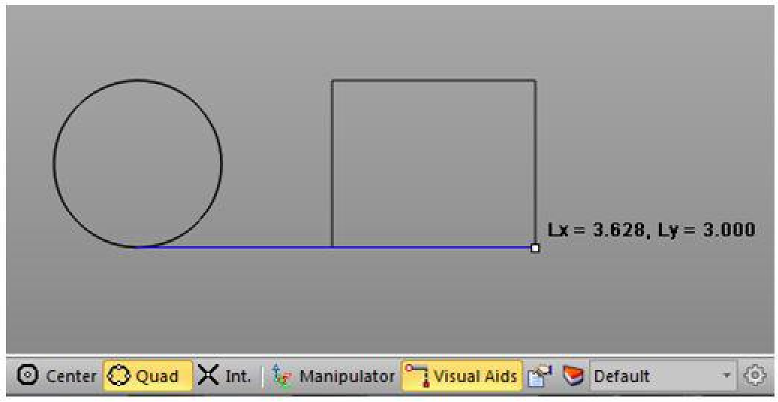
Selecting Start Point and Curve Direction arrow display under options displays both start point and curve direction arrow.

Construction Visual Aids Style

This provides an option to choose the display style between Stipple line and Solid line for visual aids when the Visual Aids is set to active under the status bar.



Stipple line display



Solid line display

Rendering Options

This can be set to Use Display Lists or Vertex Arrays. The Vertex Arrays option also provides the option to Use Video Memory from the video card that is installed on the computer. The speed and performance varies depending on the operating system type and video card installed.

View Change Animation

Whenever the view orientation is changed within the active viewport, VisualCAD can make that orientation change instantly or through smooth animation. If animation is used, the speed of the animation can be set.

Under the View Change Animation function, check the box next to Speed to activate the animation. The animation speed can be controlled using the slider bar.

Ghosted Display Transparency

Whenever the view mode is set to Ghosted Display, you can use this slider to control the transparency level of the hidden (ghosted) geometry.

Depth Testing

These options allow you to fully display certain objects regardless of the view mode. For example, checking the box to Disable Depth Testing for curves, will display all curves even if the view is set to shaded model. You can also Disable Depth testing for grid. If enabled, the grid lines will always display.

Thickness

Use this parameter to set the display line thickness for all wireframe geometry. Set this to 1 for the default line thickness. Higher for thicker lines.

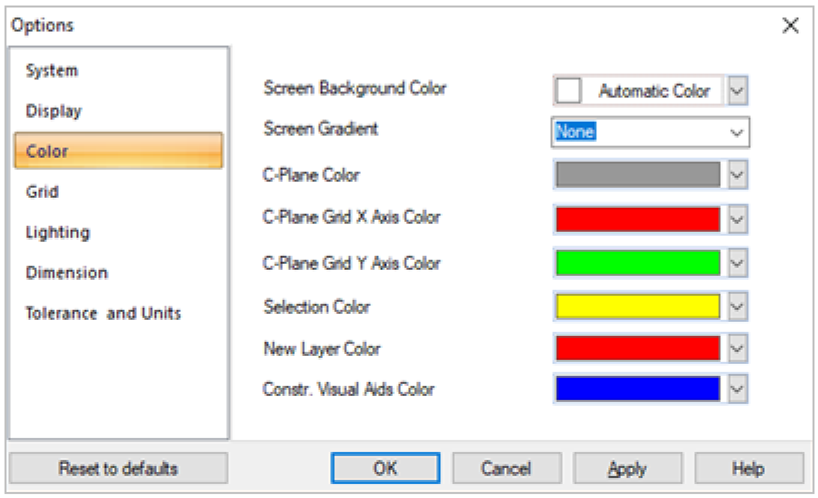
Related Topics

Options

4.10.3 Color

The default colors are set using this dialog.

Dialog Box: Options > Color



Dialog Box: Options > Color

- Screen Background Color
  This allows user to set the viewport background color. This is the color displayed when no objects are at that position.
- Screen Gradient
  This is a smooth change of lighting. The direction specified is the lighter side. It starts slightly darker on the opposite side of the screen and gradually gets lighter until it reaches the specified side.
- C-Plane Color
  This changes the color of the construction plane grid.
- C-Plane Grid X Axis Color
  This changes the horizontal axis line of the construction plane.
- C-Plane Grid Y Axis Color
  This changes the vertical axis line of the construction plane.
- Selection Color
  This is the highlight color of objects that have been selected.
- New Layer Color
  This is the default color of objects created on new (not Default) layers.

- Constr. Visual Aids Color
  This is the default color of construction aids when Visual Aids is set to active in the status bar.

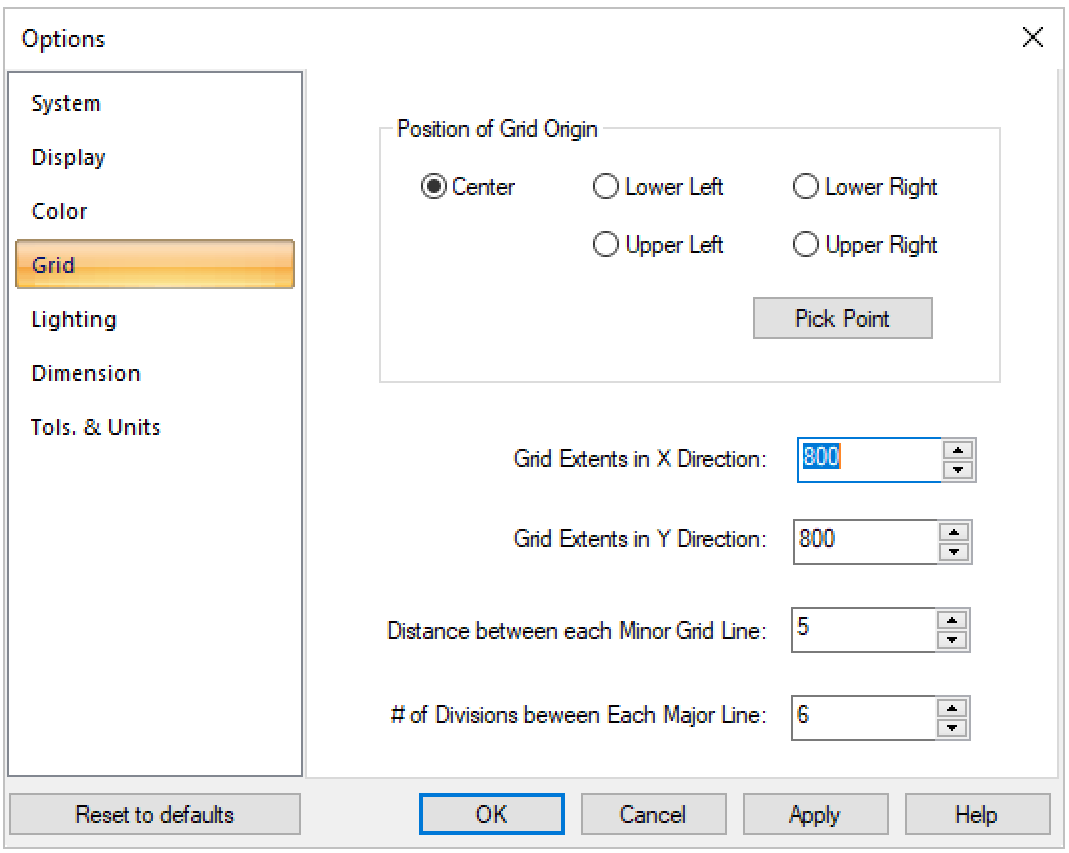
Related Topics

Options

4.10.4 Grid

This controls how the construction plane grid is displayed.

Dialog Box: Options > Grid



Dialog Box: Options > Grid

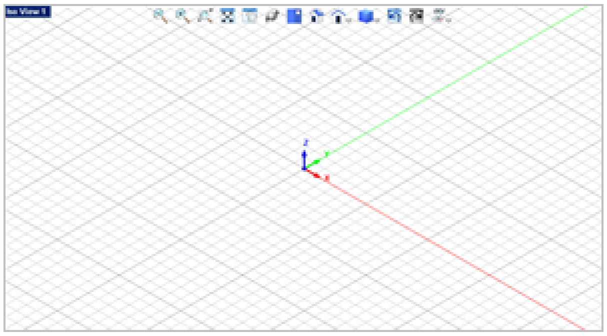
Position of Grid Origin

These options will control where the grid is positioned relative to the World Coordinate System (WCS).

The grid orientation also carries over to whichever view is active, making construction easier.

Center

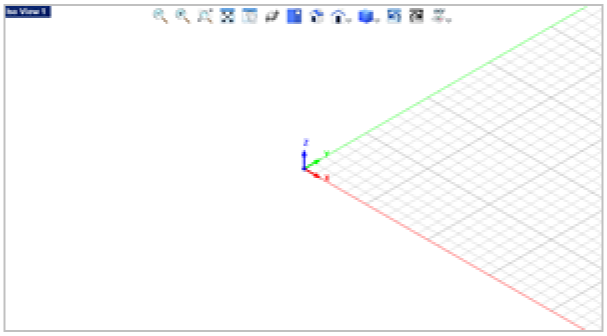
Move the center of the grid to the WCS.



Move the center of the grid to the WCS.

Lower Left

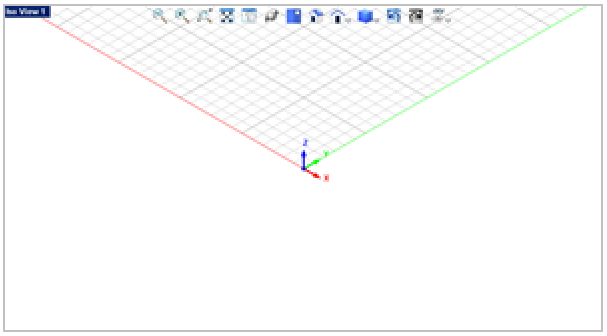
Move the lower left corner of the grid to the WCS.



Move the lower left corner of the grid to the WCS.

Lower Right

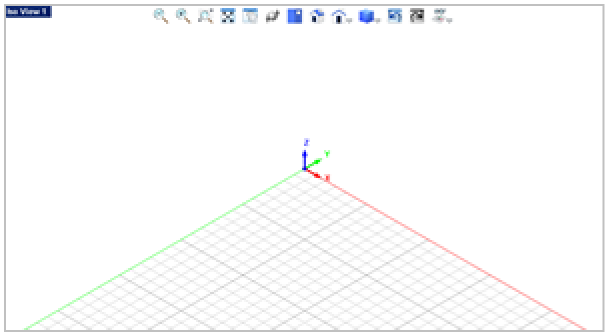
Move the lower right corner of the grid to the WCS.



Move the lower right corner of the grid to the WCS.

Upper Left

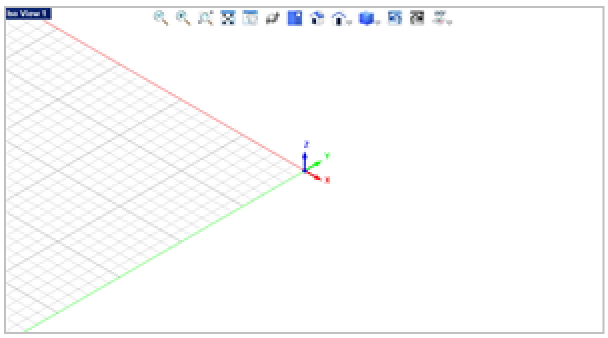
Move the upper left corner of the grid to the WCS.



Move the upper left corner of the grid to the WCS.

Upper Right

Move the upper right corner of the grid to the WCS.



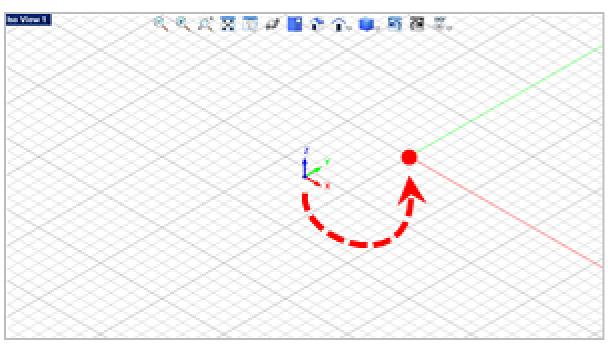
Move the upper right corner of the grid to the WCS.

Pick

Use the button to pick a point on the screen and the origin of the grid will move to this location.

Notes:

- The point you pick is always on the default XY plane.
- If you use the Upper/Lower options above after using this Pick option, the offset distance between the WCS and the new origin of the grid remains in effect.



Move the the origin of the grid to this location.

Other Grid Options

Grid Extents in X Direction

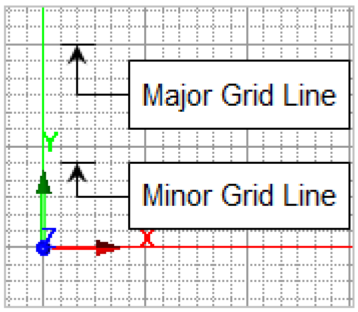
This controls how many major divisions are displayed along the X axis direction.

Grid Extents in Y Direction

This controls how many major divisions are displayed along the Y axis direction.

Distance between each Minor Grid Line

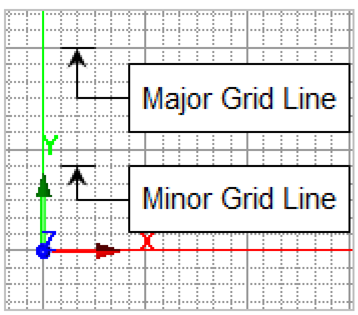
This controls the distance between each minor grid line.



Major/Minor Grid Lines

of Divisions between Each Major Line

This controls the distance between each major grid line. the number of minor rid line multiplied by the number of major grid lines will determine the overall extent of the grid.



Major/Minor Grid Lines

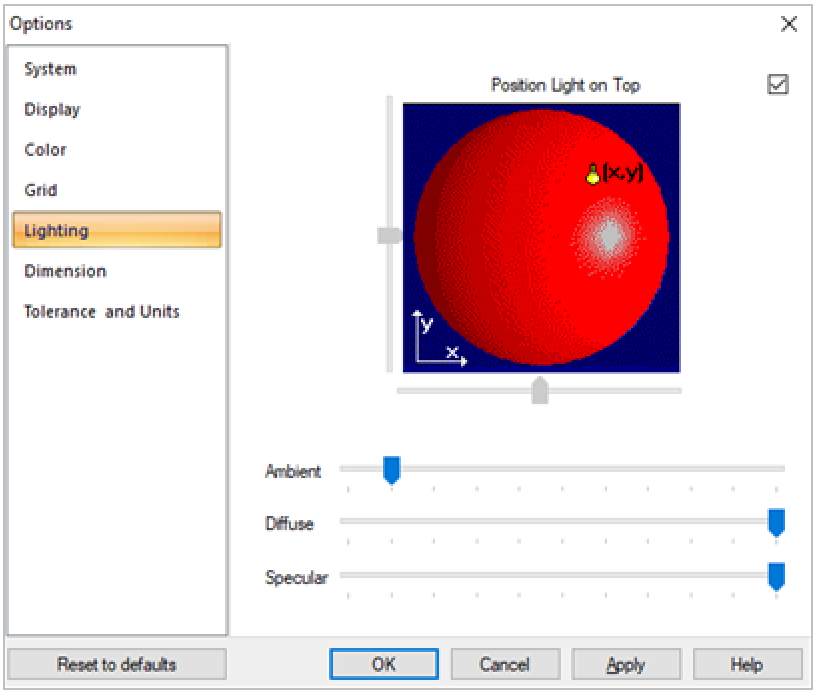
Related Topics

Options

4.10.5 Lighting

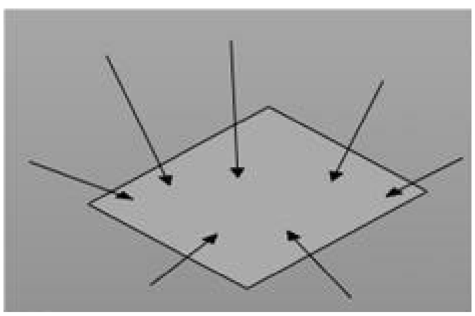
This controls the characteristics of lighting for the model.

Dialog Box: Options > Lighting



Dialog Box: Options > Lighting

There are three types of lighting: ambient, diffuse, and specular.

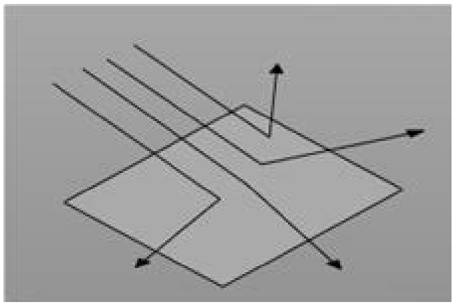


Position Light on Top

The scene has a single light source and there are controls for setting its position. Not all lighting in the model comes directly from that one source; there is some ambient light coming in from all over. The **Position Light on Top** checkbox must be unchecked to be able to move the light source.

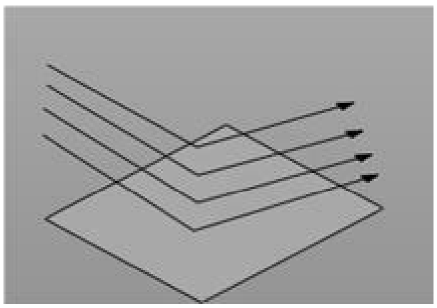
Ambient

Ambient light comes in from all different directions equally. Light sources have no effect on ambient light. Shadows don't appear with only ambient light.



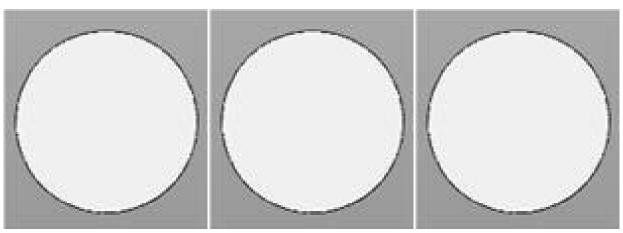
Diffuse

Diffuse light comes from a single source but is scattered in all directions. Shadows appear, but there are no “shiny” spots.

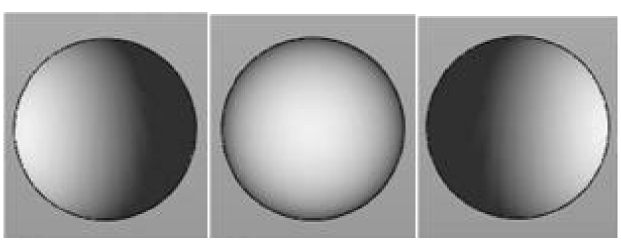


Specular

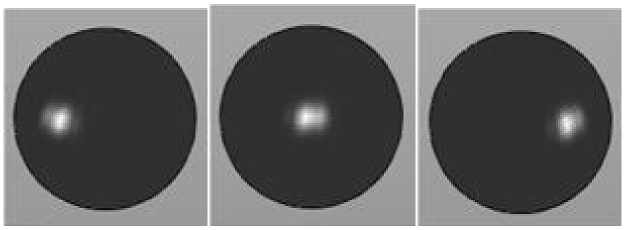
Specular light comes from a single source and is scattered in a single direction. It is reflective light. Shiny metal objects demonstrate good specular reflection. Shading is not a significant consideration with specular light.



The three pictures above demonstrate ambient light as the light source moves from left to right. There is no effect since ambient light comes from all sides equally.



The pictures above show the effects of the motion of the light source using only diffuse lighting. There are no “shiny” spots but yet there is a direction from the light.



The last set of pictures uses only specular lighting. As the light source moves from left to right, the “shiny” spot follows it. However, there is nothing else lit up, so the effect is not just reflective metal.

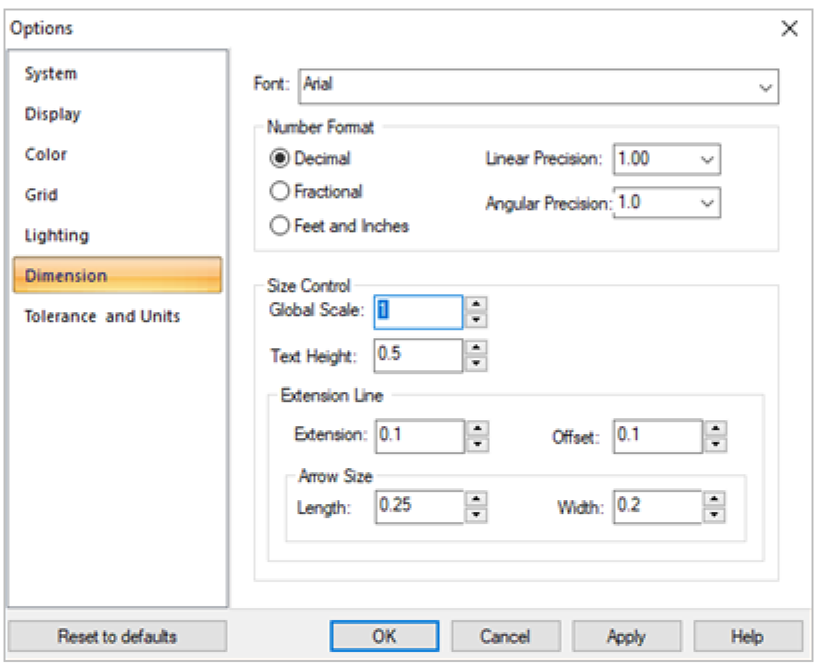
Related Topics

[Options](#)

4.10.6 Dimension

This controls the appearance of dimensioning labels, arrows, and leader lines.

Dialog Box: Options > Dimension



Dialog Box: Options > Dimension

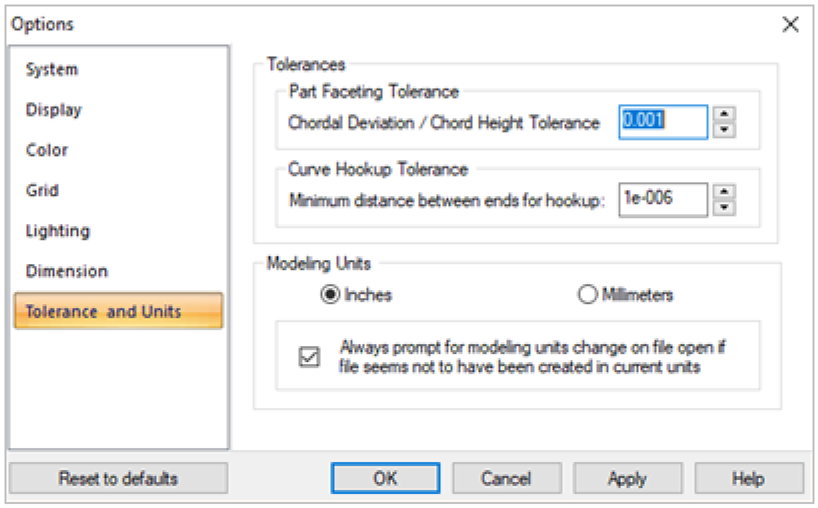
Related Topics

[Options](#)

4.10.7 Tol. & Units

This controls the accuracy of the model and part units.

Dialog Box: Options > Tolerance and Units



Dialog Box: Options > Tolerance and Units

Tolerances

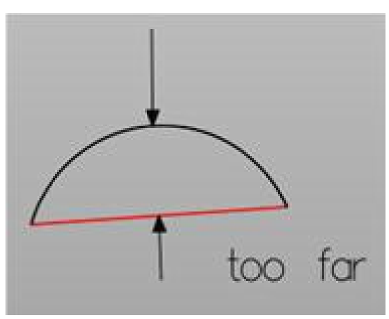
This controls the accuracy of the models. Smaller tolerance numbers often result in significantly big files and slower processing time but are more accurate.

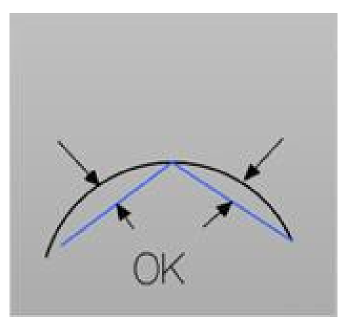
The tolerances are set in a dialog.

Part Faceting Tolerance

Chordal Deviation/Chord Height Tolerance

The chord height tolerance is used in commands that require approximation of smooth curves and surfaces.



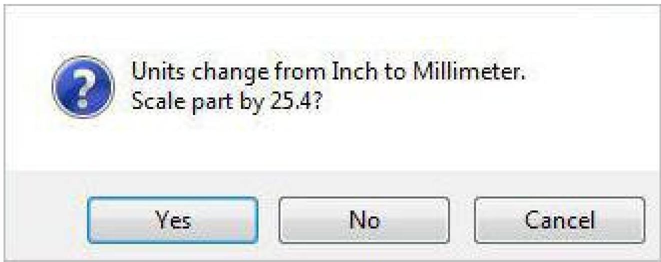


Curve Hookup Tolerance

This specifies maximum separation between the endpoints of different curves when joining them into a single curve.

Units

This converts units from inches to millimeters or from millimeters to inches. If there are existing objects, a dialog will appear confirming if you really want to convert the units.



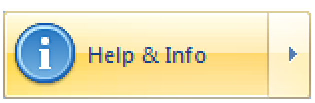
The objects remain the original size but the coordinate values change.

You can check the box to always prompt you to change the units when files are opened if they appear not to be in the correct units. For example, if the system sees a coordinate value that is quite large such as 500.5, then this dialog will display giving you the opportunity to change the units to Millimeters.

Related Topics

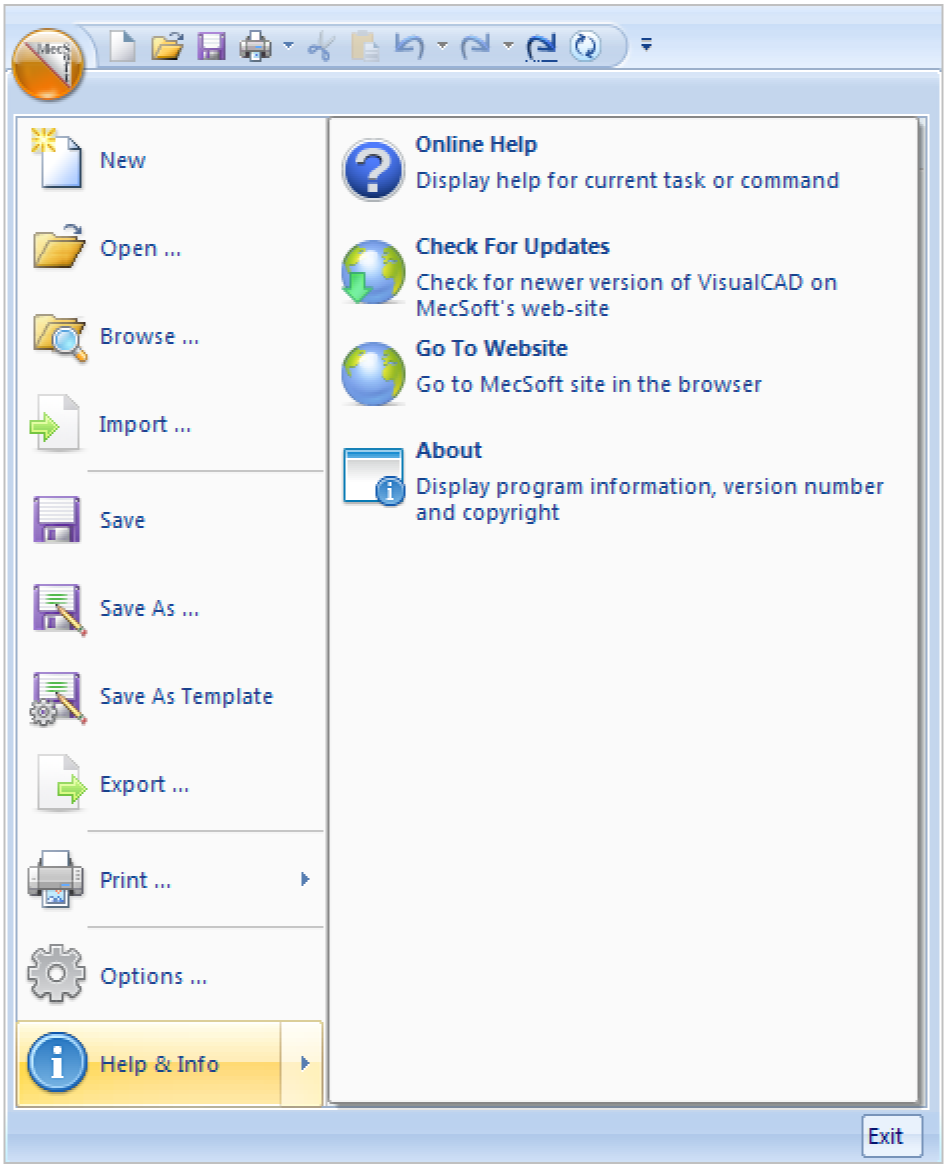
Options

4.11 Help & Info Menu



Select Help & Info from the VisualCAD Menu to access information about VisualCAD and how to use it. You can also select (left-click) on this button to display the VisualCAD Online Help.

The Help & Info Menu



VisualCAD® Menu > Help & Info

Related Topics

[Online Help]

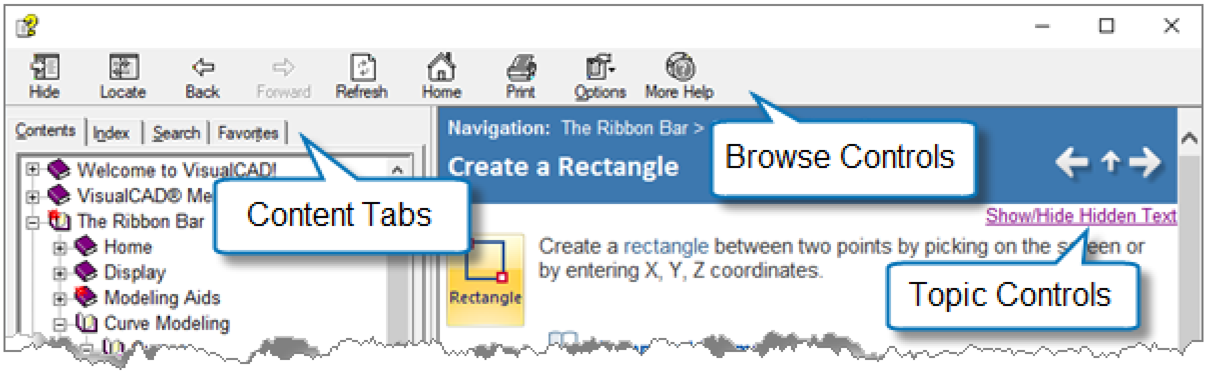
[Check for Updates]

[Go To Website]

[About]

4.11.1 Help Topics ...

VisualCAD is accompanied with a comprehensive online help system. When a command is active you can press the <F1> key to display context-sensitive help for that command. Many dialogs also have a Help button that you can click to display help for that dialog. Whenever you see the Help "?" icon, select it to display the Online Help system.



VisualCAD Online Help Browser Controls

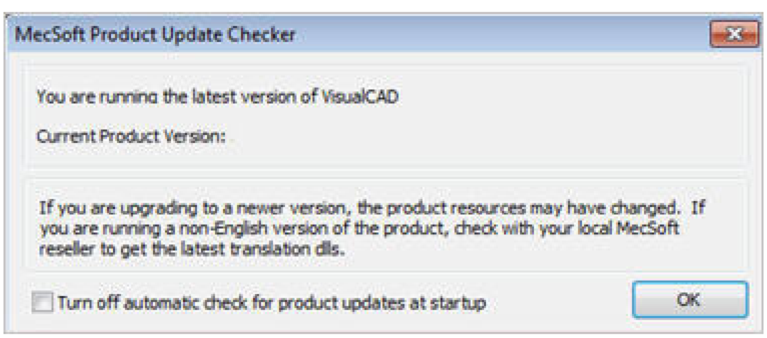
Related Topics

[Help]

4.11.2 Check for Updates ...

This checks for product updates. Internet connection is required to perform this task.

Dialog Box: Check for Updates



Dialog Box: Check for Updates

- Turn off automatic check for product updates at startup
  Check this box if do not want VisualCAD to check for updates automatically.

Related Topics

[Help]

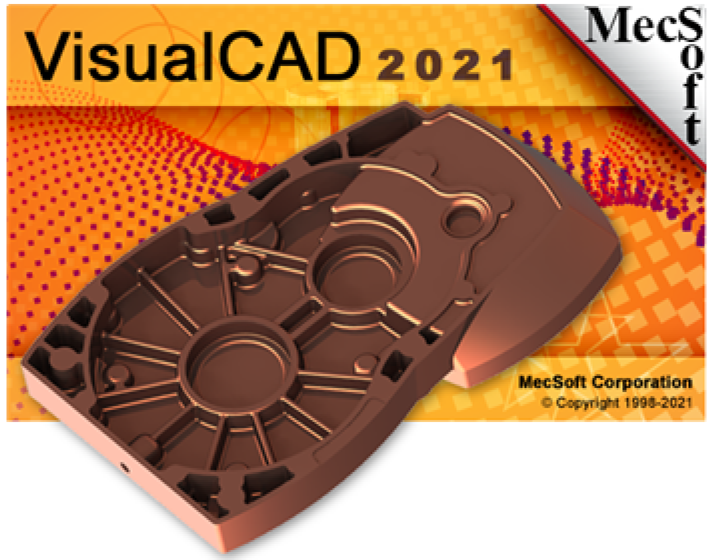
4.11.3 Go To Website

Selecting this option displays the MecSoft.com website in your default web browser. Note that and Internet connection is required. From the website you can access free downloads, videos, tutorials and keep up to date on the latest innovations from MecSoft Corporation.

Go to the MecSoft.com Website

4.11.4 About VisualCAD

This brings up a dialog, which shows the product version number.



Dialog Box: About VisualCAD

Related Topics

[Help]

[Help Topics ...]

[Check for Updates ...]

4.12 Exit

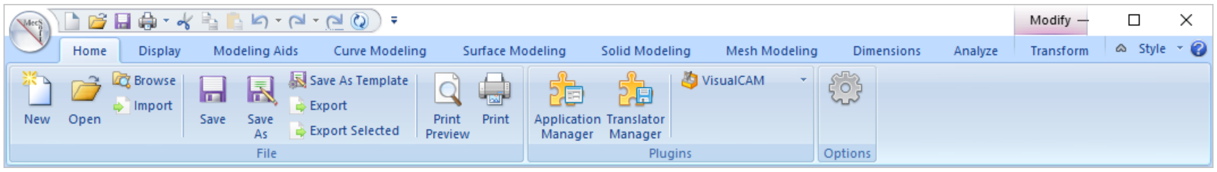
Exit Quits VisualCAD. If there is unsaved work, you will be asked if you want to save it before quitting. If the unsaved work is not saved, it will be lost.

Related Topics

[File]

The Ribbon Bar

The Ribbon Bar displays all of the features available in VisualCAD. It conforms to the standard for Ribbon Bars set by Microsoft® Corp.



VisualCAD® Ribbon Bar

Tabs along the top of the Ribbon Bar allow selection of major categories of functionality, such as creation, modification, analysis, dimensioning, etc. Within each tab are subordinate panes of related functionality that are labeled along the bottom of the group. The Home Ribbon bar shown above has three sub panes of commands labeled File, Plugins and Options.

Ribbon Bars

Home

Display

Modeling Aids

Curve Modeling

Surface Modeling

Solid Modeling

Mesh Modeling

Dimensions

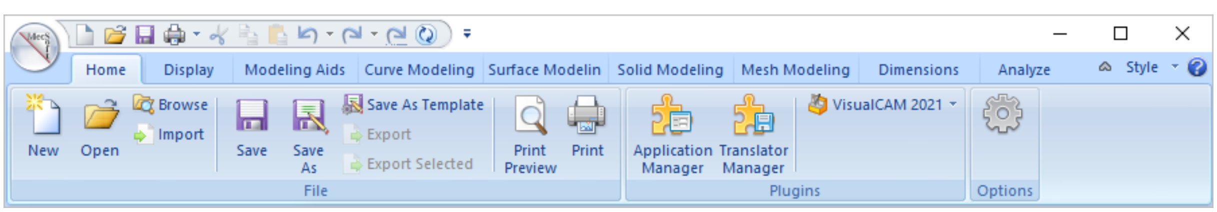
Analyze

Transform

Print Preview

5.1 Home

The Home Ribbon Bar contains File, Plugins, and Options panes and associated functions.



The Home Ribbon Bar

Related Topics

[File Pane, Home Ribbon Bar](#)

[Plugins Pane, Home Ribbon Bar](#)

[Options Pane, Home Ribbon Bar](#)

[The Ribbon Bar](#)

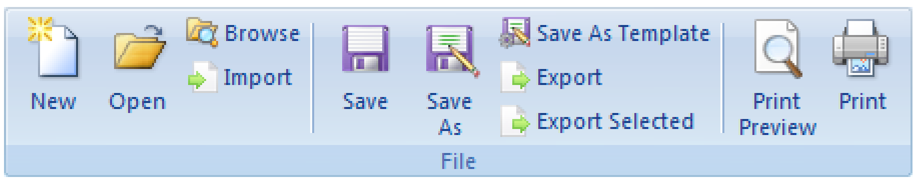
[Customize Quick Access Toolbar](#)

[Customize Dialog](#)

[VisualCAD Options](#)

5.1.1 File

The Home Ribbon Bar contains the following File pane with associated functions.



Home Ribbon Bar
File Pane

Related Topics

[New](#)

[Open ...](#)

[Browse ...](#)

[Import ...](#)

[Save](#)

[Save As ...](#)

[Save As Template](#)

[Export ...](#)

[Print Preview ...](#)

[Print ...](#)

[Home Ribbon Bar](#)

5.1.1.1 New

Create a new VCP (VisualCAD Part) file. The file holds the geometry, layers, orientation, and preferences. If there are unsaved objects, a dialog will ask you if they want to save them before creating the new VCP file. All current work will be overwritten by the new file.

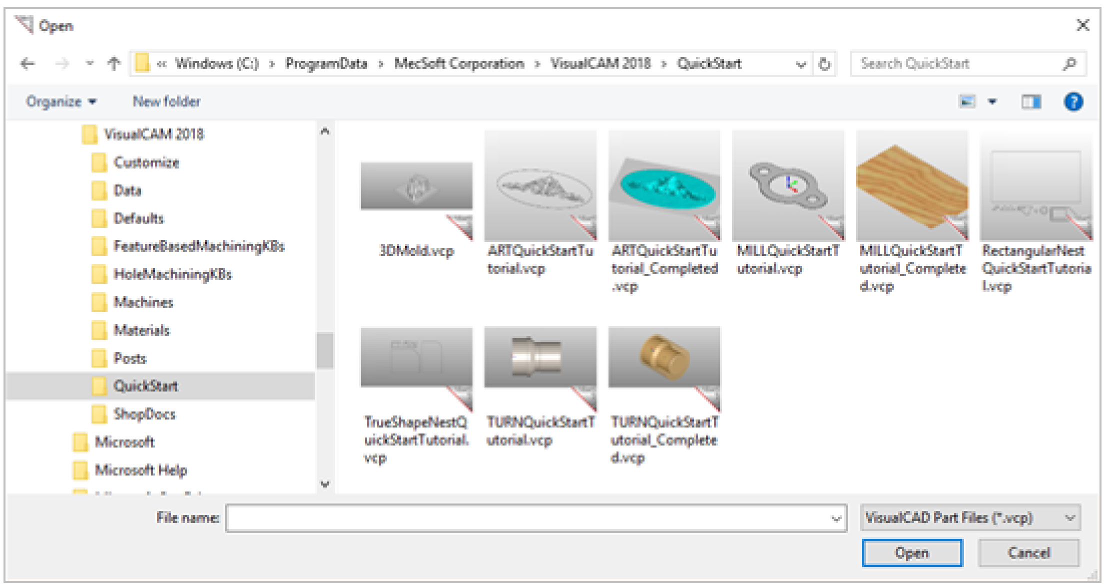
Related Topics

[File](#)

5.1.1.2 Open ...

Open an existing VCP (VisualCAD Part) file or one of the [supported file types](#). The file holds the geometry, layers, orientation, and preferences. If there are existing unsaved objects, a dialog will ask you if you want to save them before opening the VCP file. All current work will be overwritten by the loaded file.

Dialog Box: Open



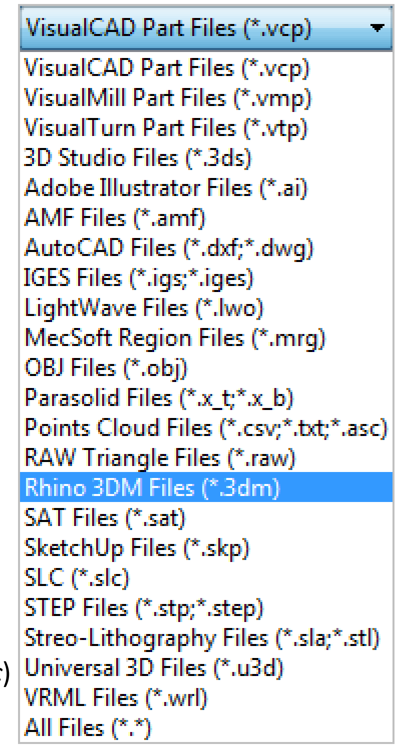
Dialog Box: Open

Standard Windows® File Browser supports preview of VisualCAD *.vcp drawings and models.

Files of Type Supported

The following file types are supported by VisualCAD. See [File Open](#) and [File Import](#) for more information

- VisualCAD Part Files (*.vcp)
- VisualMILL Part Files (*.vmp)
- VisualTurn Part Files (*.vct)
- 3D Studio Files (*.3ds)
- Adobe Illustrator Files (*.ai)
- AMF Files (*.amf)
- AutoCAD (*.dxf;*.dwg)
- IGES Files (*.igs;*.iges)
- LightWave Files (*.lwo)
- MecSoft Region Files (*.mrg)
- OBJ Files (*.obj)
- Parasolid Files (*.x_t;*.x_b)
- Point Cloud Files (*.cvs;*.txt;*.asc)
- RAW Triangle Files (*.raw)
- RHINO 3DM Files (*.3dm)
- SAT Files (*.sat)
- SketchUP Files (*.skp)
- SLC Files (*.slc)
- STEP Files (*.stp;*.step)
- Stereo-lithography files (*.sla;*.stl)
- Universal 3D Files (*.u3d)
- VRML Files (*.wrl)



File Types Supported

Related Topics

File

Import ...

Save

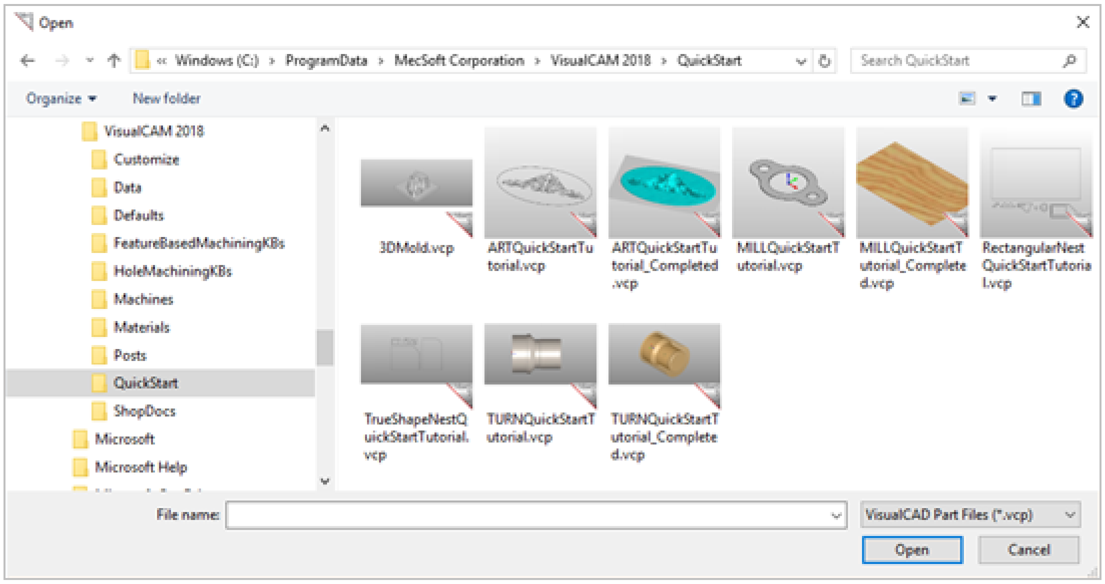
Save As ...

Browse...

5.1.1.3 Browse ...

Visually browse files on the computer. VCP files can be opened. A preview window will show the last view displayed when the original file was saved.

Dialog Box: VisualCAD® File Browser



Dialog Box: VisualCAD® File Browser

Related Topics

[File Menu]

[Import ...]

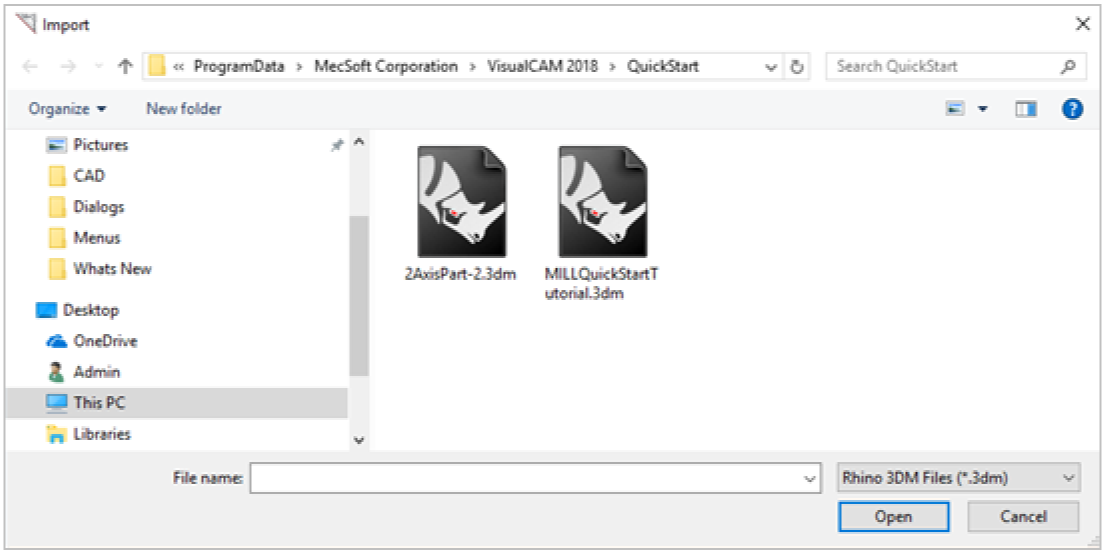
[Save]

[Save As ...]

5.1.1.4 Import ...

Open existing files made by other programs. This is used to transfer information, normally geometric models. Currently, VisualCAD can import the following formats:

Dialog Box: Import

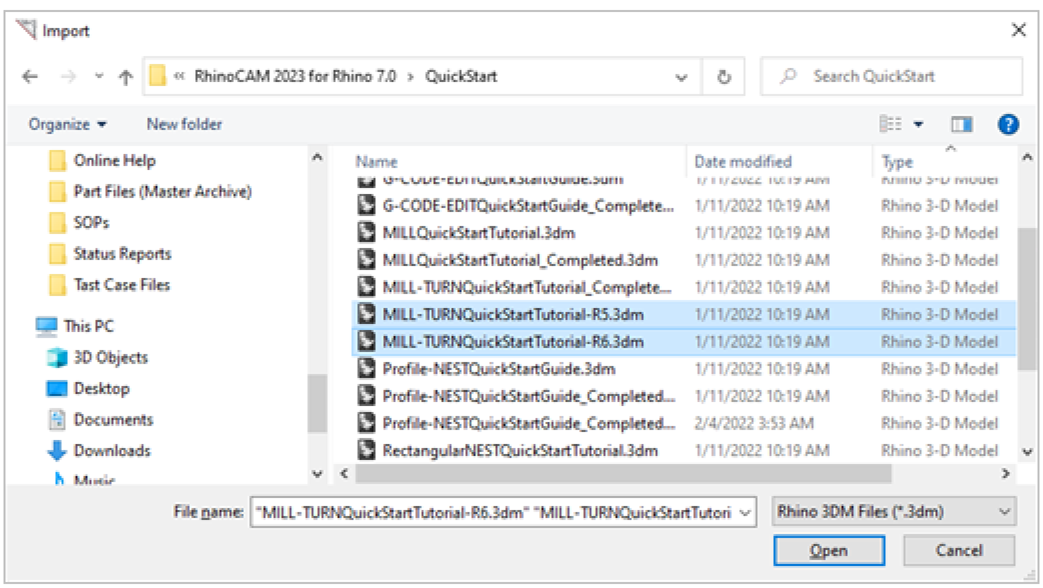


Dialog Box: Import

Standard Windows® File Browser supports preview of VisualCAD *.vcp drawings and models.

Import Multiple Files

You can Import multiple files at the same time. To do so, first press and hold the <Ctrl> key while selecting the files from the Import dialog. Each file will highlight and also be listed together in the File name field at the bottom of the dialog. When done selecting files, pick the Open button.



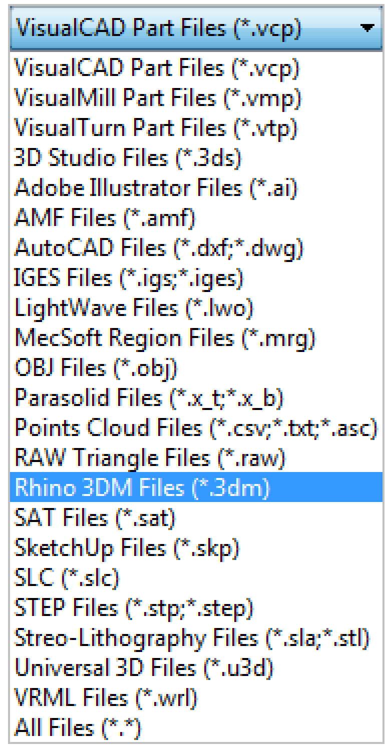
Dialog Box: Import (When importing multiple files at once)

Standard Windows® File Browser supports preview of VisualCAD *.vcp drawings and models.

Files of Type Supported

The following file types are supported by VisualCAD. See File Open and File Import for more information

- VisualCAD Part Files (*.vcp*)
- VisualMILL Part Files (*.vmp*)
- VisualTurn Part Files (*.vct*)
- 3D Studio Files (*.3ds*)
- Adobe Illustrator Files (*.ai*)
- AMF Files *(*.amf)*
- AutoCAD (*.dxf;*.dwg*)
- IGES Files (*.igs;*.iges*)
- LightWave Files (*.lwo*)
- MecSoft Region Files (*.mrg*)
- OBJ Files (*.obj*)
- Parasolid Files (*.x_t;*.x_b*)
- Point Cloud Files (*.cvs;*.txt;*.asc*)
- RAW Triangle Files (*.raw*)
- RHINO 3DM Files (*.3dm*)
- SAT Files (*.sat*)
- SketchUP Files *(*.skp)*
- SLC Files (*.slc*)
- STEP Files (*.stp;*.step*)
- Stereo-lithography files (*.sla;*.stl*)
- Universal 3D Files (*.u3d*)
- VRML Files (*.wrl*)



File Types Supported

Related Topics

File

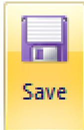
Export

Save

Save As

Browse...

5.1.1.5 Save

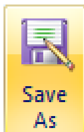


Save geometry, layers, orientation, and preferences in a VCP file. If saving for the first time, you can name the file and select where on the computer to save the file. If the file had been saved previously, the current information overwrites the original information.

Related Topics

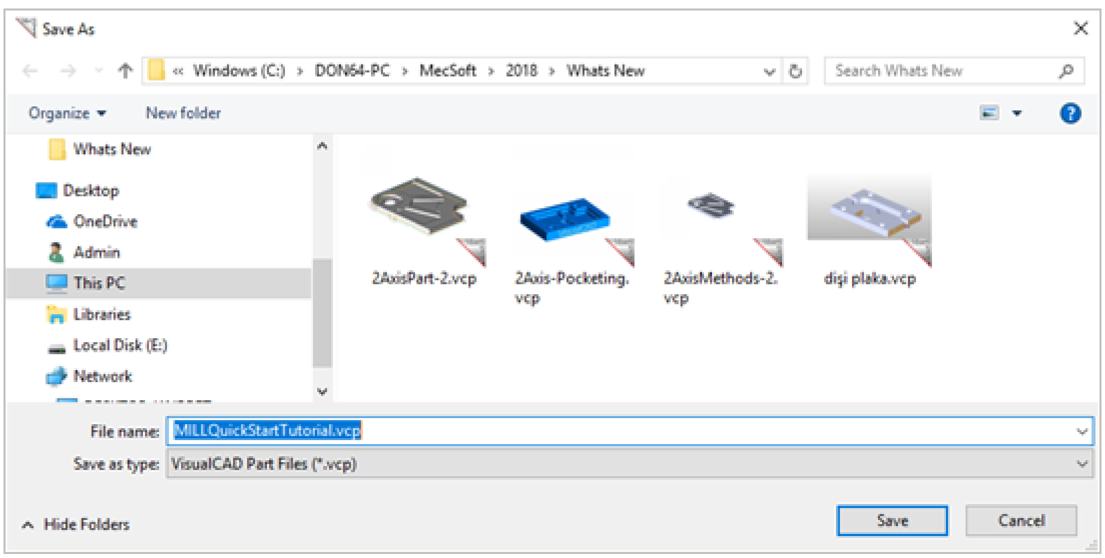
File

5.1.1.6 Save As ...



Save geometry, layers, orientation, and preferences in a VCP file. Name the file and select where on the computer to save it. If selecting an existing file, the current information will overwrite the original information. See Export ... to save as different file types.

File Save As dialog box



File Save As dialog box

Save as Files Types Supported

- VisualCAD Part Files (*.vcp)

Related Topics

File

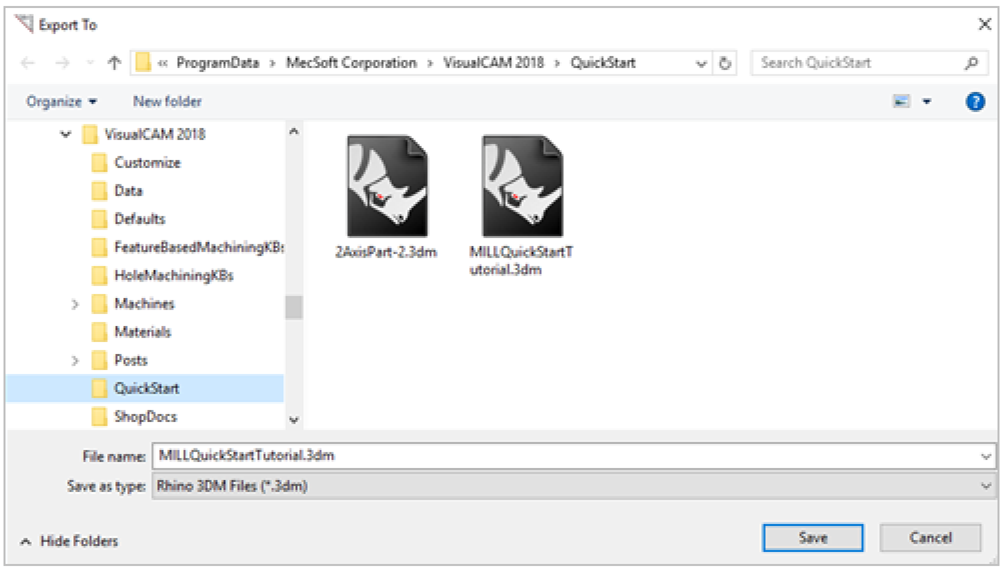
5.1.1.7 Save As Template

Use this to save VisualCAD® preferences (System, Display, Color, Grid, Lighting, Dimension, Tolerance and Units) in the **.vcp* file. This template file can then be used to begin new VisualCAD drawings.

5.1.1.8 Export ...

Save geometry in a neutral file format (can be read by many different programs). The following export translators are available. Binary formats are usually much smaller in size, but ASCII formats are more easily read. Ordinary text programs can read the ASCII files.

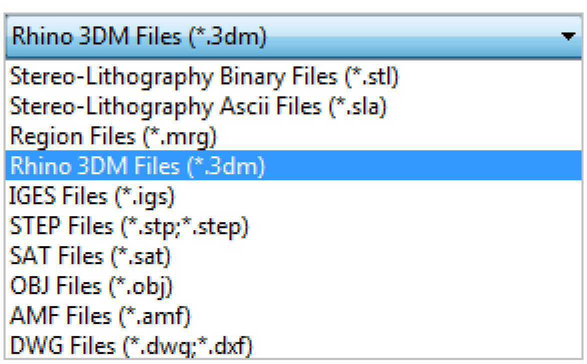
Dialog Box: Export To



Dialog Box: Export To

Export File Types Supported

The following file types are supported by VisualCAD during Export:



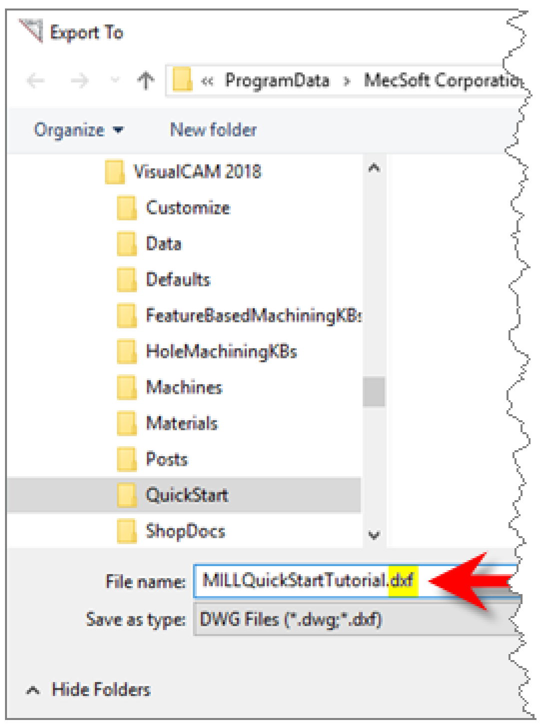
File Types Supported during Export

- Stereo-Lithography Binary (**.stl*)

- Stereo-Lithography ASCII (*.sla*)
- Region Files (*.mrg*)
- Rhino (*.3dm)
- IGES (*.igs)
- STEP Files (*.stp*; *.step*)
- SAT Files (*.sat*)
- OBJ Files (*.obj*)
- AMF Files (*.amf)
- DWG Files *(*.dxf; *.dwg)*

Exporting to Dwg/Dxf formats

When you select the Dwg/Dxf export File Type, the file is exported to *.dwg format by default. If you would like to export the file in *.dxf format, you will need to enter the file extension to the end of the file name (i.e., Example.dxf).



Exporting to the *.dxf File Type

Related Topics

File

Import ...

Save

[Save As ...](#)

[Browse...](#)

5.1.1.9 Export Selected

This command is similar to the File Export command but only exports the currently selected entities. The entities MUST be pre-selected before executing this command. All of the file types supported by the File Export command are also supported by this command.

Basic Procedure

	Screen Entry	Command Input
Step 1	Select the entities that you wish to export.	
Step 2	 Pick the Export Selected icon from the Home Ribbon Bar.	
	Use the Export To dialog to create the exported file type required. See File Export ... for more information about using the Export To dialog.	

Related Topics

File Export ...

File

Import ...

Save

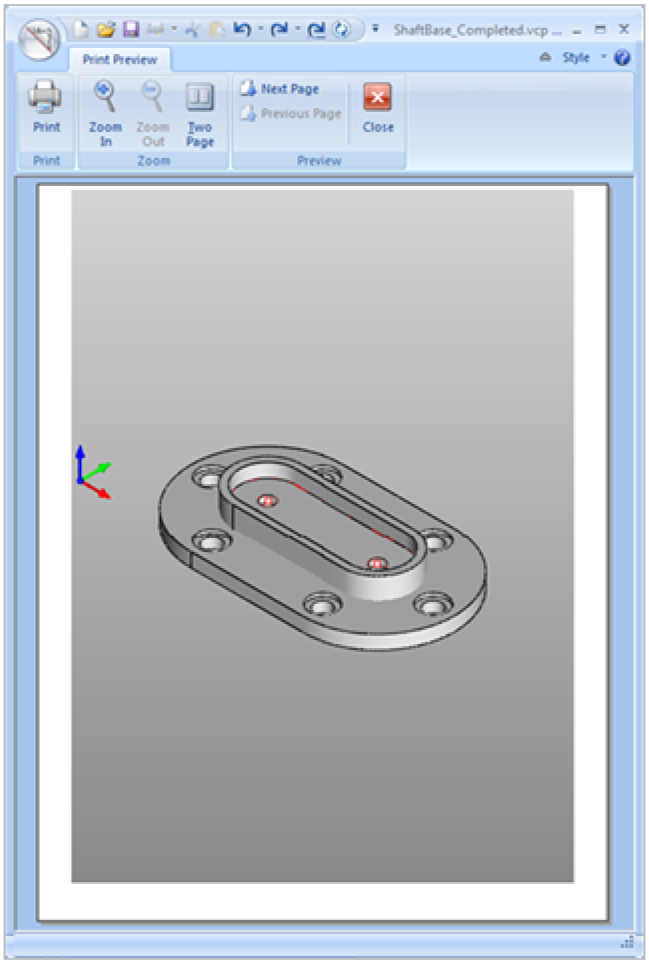
Save As ...

Browse...

5.1.1.10 Print Preview

This shows a picture of what will be printed. The picture can be printed from here along with the additional controls listed below.

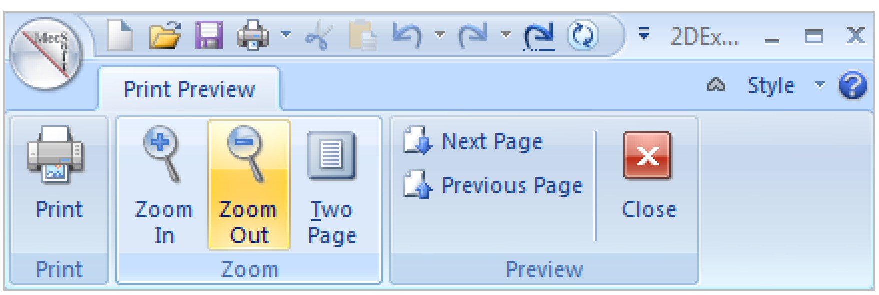
Print Preview Display



File > Print Preview Display

Print Preview Controls

The following controls are available from the Print Preview ribbon bar:



Print Preview Ribbon Bar

- Print
- Zoom In/Out
- Two Page
- Next Page

- Prev Page
- Close

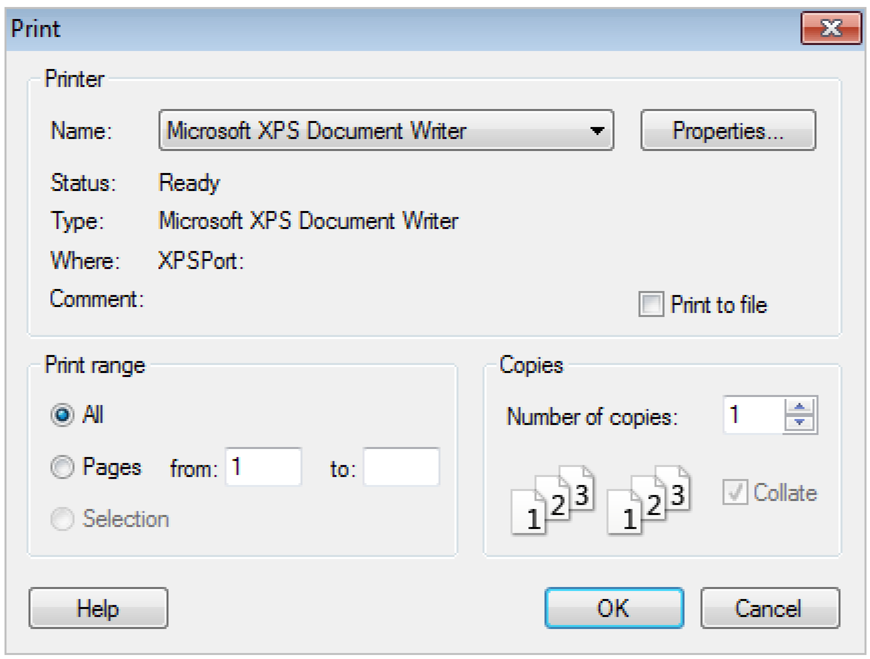
Related Topics

File

5.1.1.11 Print ...

Prints the objects displayed. If multiple views are displayed, the currently selected view will be printed. The standard Windows® Print dialog box is displayed.

The standard Windows® Print dialog



The standard Windows® Print dialog

Related Topics

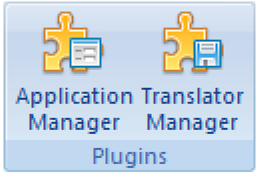
VisualCAD Menu

Print Menu

Print Setup ...

Print Preview ...

5.1.2 Plugins



The Home Ribbon Bar contains the following Plugins pane with associated functions.

Related Topics

Application Plug-ins ...

Translator Plug-ins ...

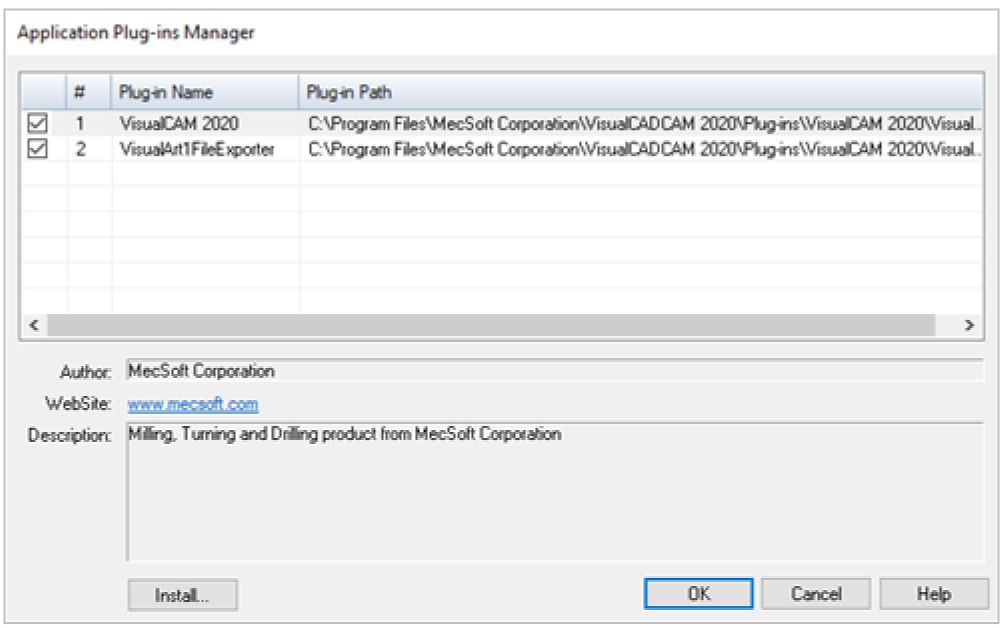
Home Ribbon Bar

5.1.2.1 Application Plug-ins ...

This allows additional functionality to be added to VisualCAD after installation.

Select from the following for more information:

Dialog Box: Application Plug-ins Manager



Dialog Box: Application Plug-ins Manager

Enable/Disable Application Plug-ins

Active application plug-ins are listed at the top of the manager. The Plug-in Name and Plug-in Path are listed.

Check the box to the left of the application plug-in to enable or disable the plug-in.

Install New Application Plug-ins

1. Select the Install button to display the File Browser.
2. Separate files, normally with an extension *.vca, are stored in the Plug-ins folder under the installation directory for VisualCAD®.
3. Select a Plug-in file and then pick Open.
4. The plug-in is added to the Application Plug-in Manager along with Author, Website and a Description.

Related Topics

Options

Export

Import

Save

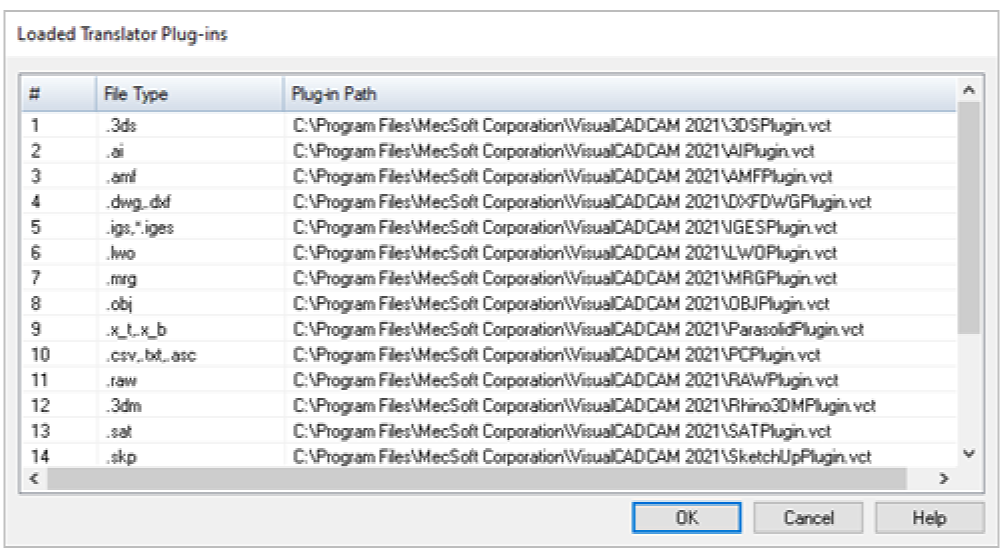
Save As

Browse...

5.1.2.2 Translator Plug-ins ...

This allows different file types to be read, imported, and exported. VisualCAD can read and write its own file types (*.vcp) with no additional plug-ins. There are some translators that come already pre-installed such as things to read Dxf, Dwg, STEP, IGES, SAT, Parasolids and STL file formats.

Dialog Box: Loaded Translator Plug-ins



#	File Type	Plug-in Path
1	.3ds	C:\Program Files\MecSoft Corporation\VisualCADCAM 2021\3DSPlugin.vct
2	.ai	C:\Program Files\MecSoft Corporation\VisualCADCAM 2021\AIPlugin.vct
3	.amf	C:\Program Files\MecSoft Corporation\VisualCADCAM 2021\AMFPlugin.vct
4	.dwg,.dxf	C:\Program Files\MecSoft Corporation\VisualCADCAM 2021\DXFDWGPlugin.vct
5	.igs,*.iges	C:\Program Files\MecSoft Corporation\VisualCADCAM 2021\IGESPlugin.vct
6	.lwo	C:\Program Files\MecSoft Corporation\VisualCADCAM 2021\LWOPlugin.vct
7	.mrg	C:\Program Files\MecSoft Corporation\VisualCADCAM 2021\MRGPlugin.vct
8	.obj	C:\Program Files\MecSoft Corporation\VisualCADCAM 2021\OBJPlugin.vct
9	.x_t,.x_b	C:\Program Files\MecSoft Corporation\VisualCADCAM 2021\ParasolidPlugin.vct
10	.csv,.txt,.asc	C:\Program Files\MecSoft Corporation\VisualCADCAM 2021\PCPlugin.vct
11	.raw	C:\Program Files\MecSoft Corporation\VisualCADCAM 2021\RAWPlugin.vct
12	.3dm	C:\Program Files\MecSoft Corporation\VisualCADCAM 2021\Rhino3DMPlugin.vct
13	.sat	C:\Program Files\MecSoft Corporation\VisualCADCAM 2021\SATPlugin.vct
14	.skp	C:\Program Files\MecSoft Corporation\VisualCADCAM 2021\SketchUpPlugin.vct

Dialog Box: Loaded Translator Plug-ins

As more translators become available, they can be plugged in and used.

Related Topics

Options

Export ...

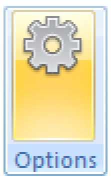
Import ...

Save

Save As ...

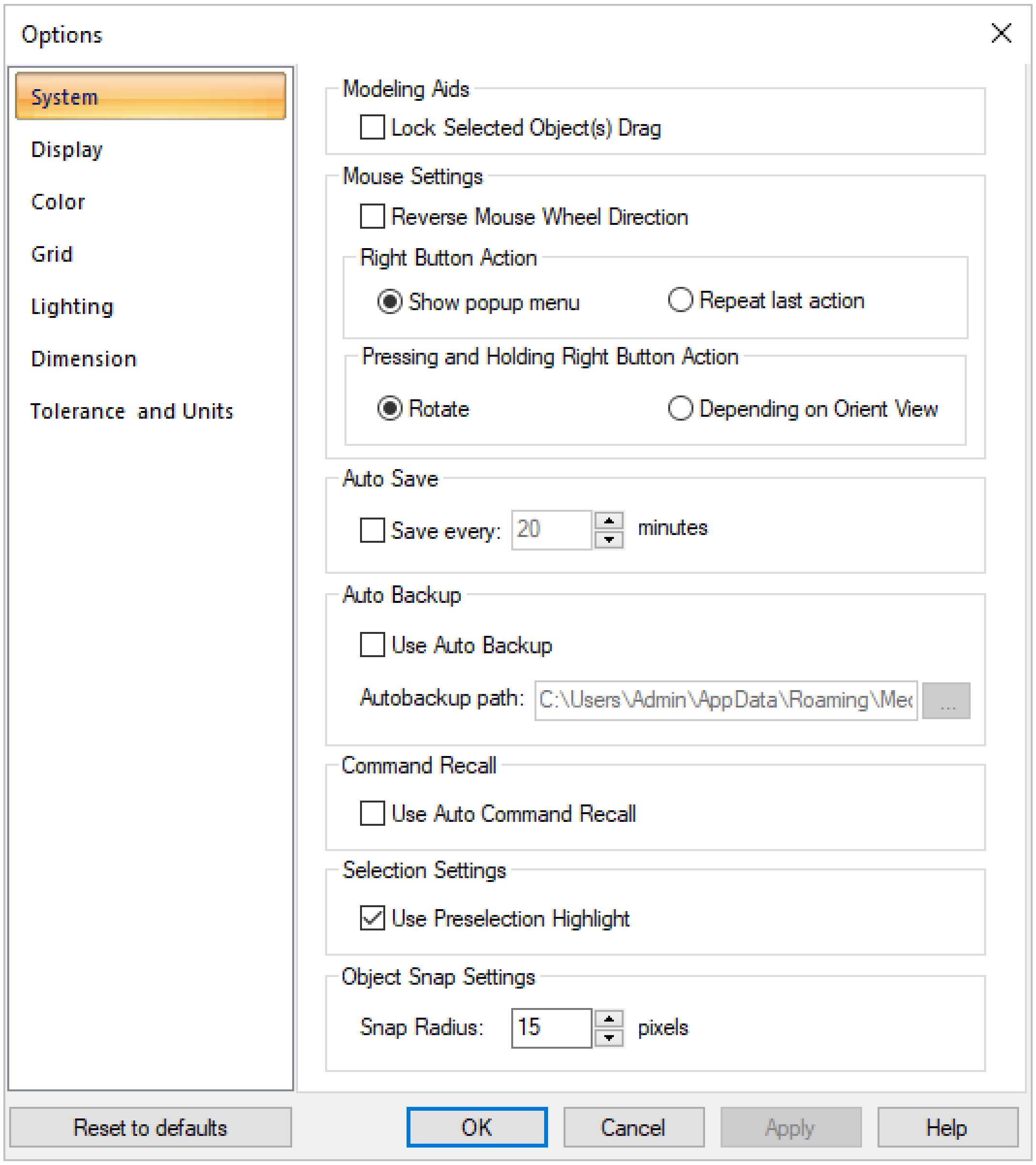
Browse...

5.1.3 Options



 The Home Ribbon Bar contains the Options pane. Selecting the icon will display the Options dialog shown below. This dialog allows you to set global parameters which include System, Display, Color, Grid, Lighting, Dimension, Tolerance and Units preferences. This dialog can also be accessed from the Status Toolbar.

Dialog Box: Options



Dialog Box: Options

Related Topics

[System](#)

[Display](#)

[Color](#)

[Grid](#)

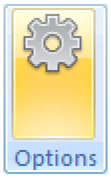
[Lighting](#)

[Dimension](#)

[Tolerance and Units](#)

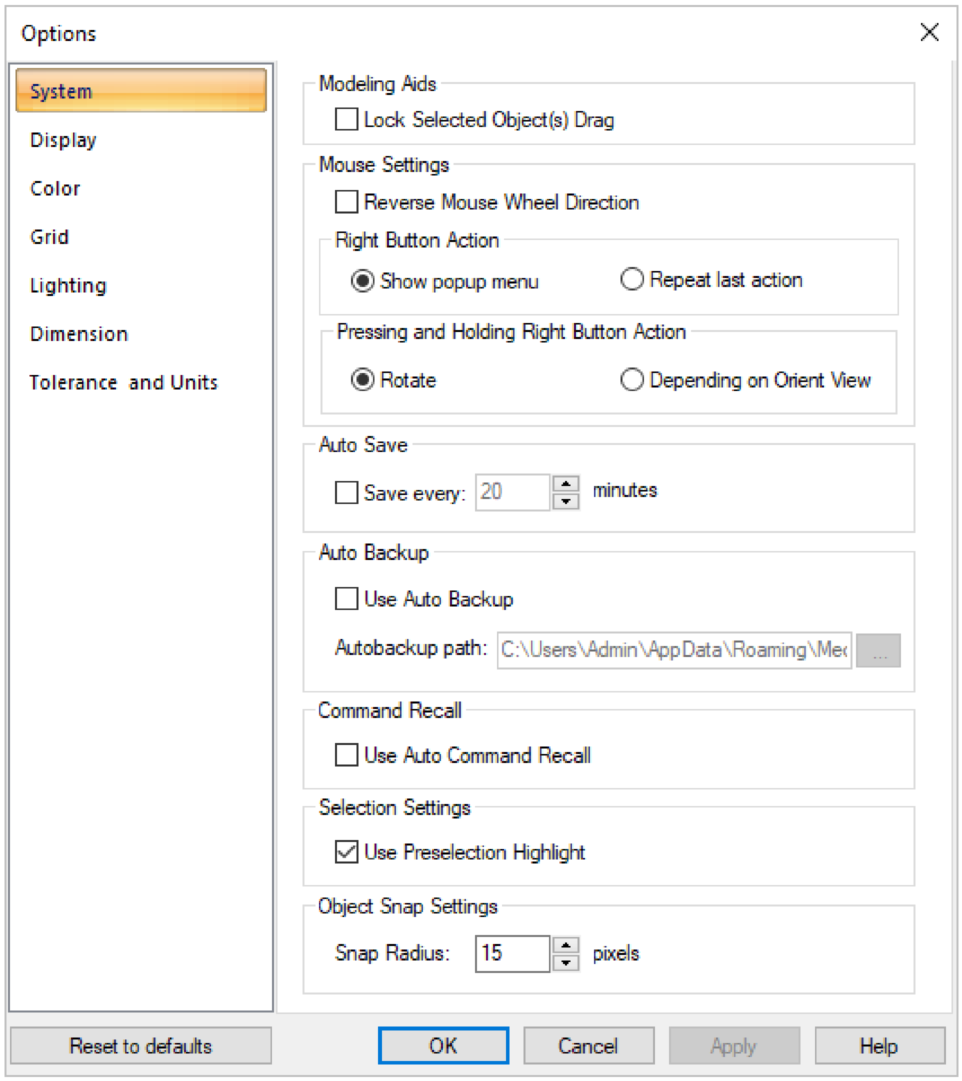
Home Ribbon Bar

5.1.3.1 System



Allows user to set preferences for modeling aids, mouse settings, auto save and command recall.

Dialog Box: Options > System



Dialog Box: Options > System

Lock Selected Object(s) Drag

If the box is not checked, you can select a geometry object and drag it around the C-Plane. When the box is checked, the objects cannot be dragged.

Mouse Settings

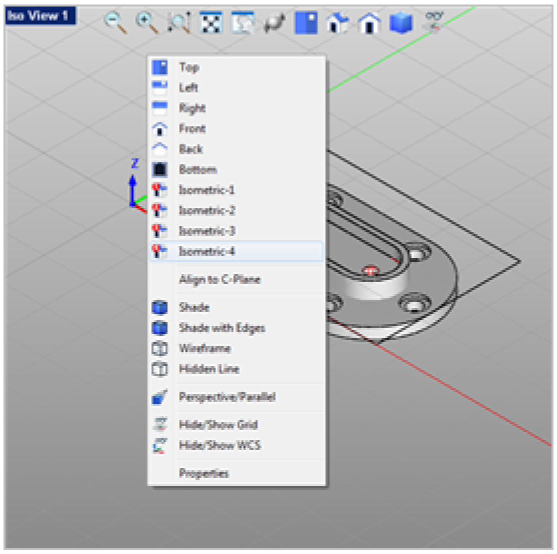
The following Mouse options are supported:

Reverse Mouse Wheel Direction

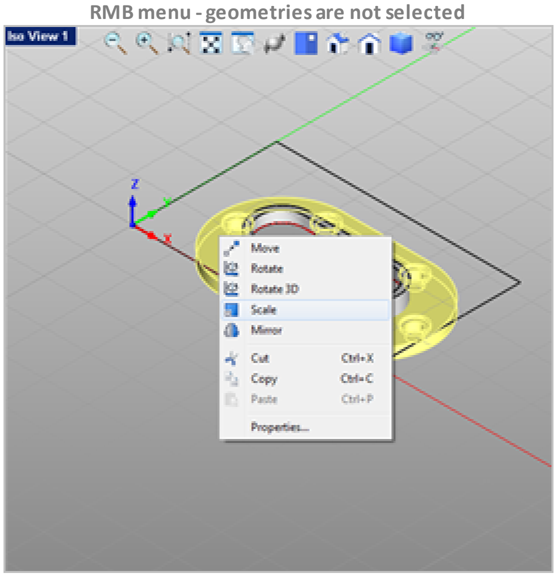
Reverses the direction of view zooming when using the mouse wheel.

Right Button Action

When Show pop-up menu is selected, performing a right mouse button click in the view port displays the commands in the pop-up menu.



RMB menu - geometries are not selected



RMB menu - geometries are selected

When Repeat last action is selected, this recalls the previous command.

Pressing and Holding Right Button Action

This option tells VisualCAD what to do when the right mouse button is pressed and held.

- Rotate
  When selected, pressing and holding down the right mouse button from all views performs a Rotate on the display.
- Depending on Orient View
  When depending on orient view is selected, pressing & holding down the right mouse button from Top, Front, Right, Left, Back, Bottom views does a Dynamic Pan of the geometries instead of rotate. For Iso views (Iso1,2,3 & 4) it performs a Rotate when pressing and holding down right mouse button down.

AutoSave

The AutoSave function can be used to automatically save your active part file at user-specified time intervals. AutoSave always saves to the current file name. This saved file is removed when you successfully close your model.

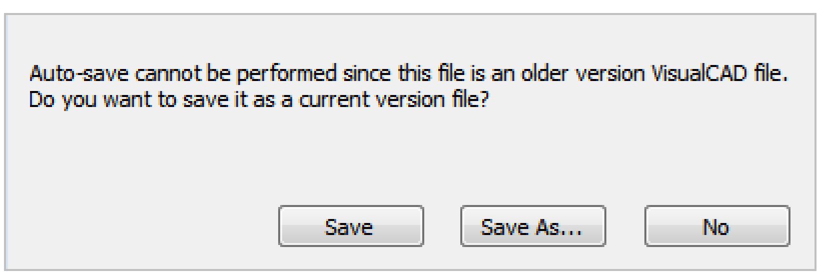
> **NOTE**: Once you activate AutoSave, you will need to Exit VisualCAD and then launch VisualCAD again before AutoSave will begin saving your part files!

Under the AutoSave function, check the box next to 'Save every xx minutes' to activate the AutoSave function. The time interval can be entered in minutes.

> AutoSave saves the file under: *<APPDATA>\MecSoft\VisualCAD 2018\AutoSave* where APPDATA is a logical pointing to a folder set up by your system administrator. By default it points to: *C:\Users\<username>\AppData\Roaming*.

Opening an Older Version *.vcp File

Opening an older version vcp file, displays the following dialog allowing you the opportunity to save the file to the current version.



AutoSave Message

When does AutoSave Prompt me to Save a File?

Under the following conditions VisualCAD will prompt you to save your file:

1. When you have opened a non-native VisualCAD file.
2. When you have imported a file.
3. When you have created a new part and has not saved the file yet.

When is the AutoSave File Deleted?

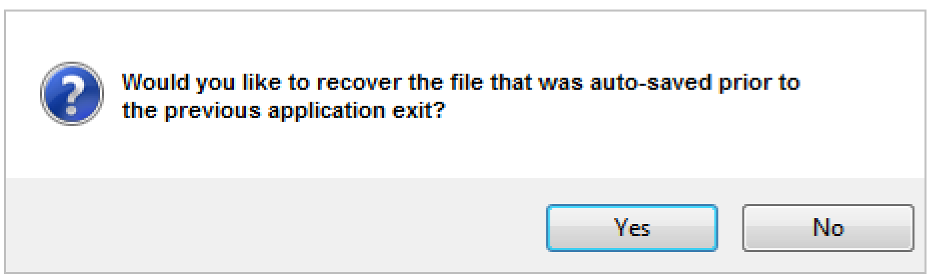
Auto save file is deleted under the following scenarios:

1. When the application closes normally.
2. When you close the file either explicitly or by opening another file.
3. When you select not to recover an AutoSave file that was detected on startup

File Recovery

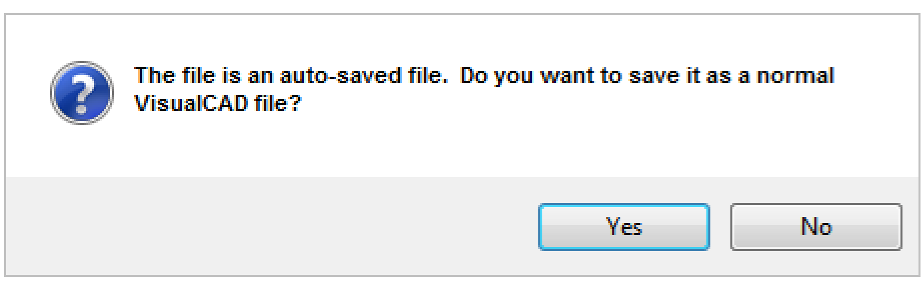
In the event of a computer crash or if VisualCAD is terminated abruptly by ending the process from task manager, VisualCAD will recover your file if AutoSave was active and your part file was automatically saved at least once.

If an AutoSave file has been created prior to the crash or termination, VisualCAD will detect the file located in the AutoSave folder the next time it is loaded. You are then prompted with the following dialog:



AutoSave Message

Selecting Yes, displays the following dialog, prompting you to save your file:



AutoSave Message

This is the only opportunity to recover the part file. If you choose not to recover it, the AutoSave file will be removed.

Use Auto Backup

Use this option to back up your VisualCAD files automatically. You can change the Auto Backup Path destination directory by selecting the 'browse' button.

If you have the check box checked, then the file is saved ONCE (as soon as the file is loaded) to the Auto Backup path with the same name as the original path file. This file is not deleted on exit so it can be recovered later if necessary. By default the check box to Auto Backup is unchecked.

Use Auto Command Recall

When this option is selected, VisualCAD automatically repeats the last command without having to go back & reselect the command from the toolbar or use the right mouse button click or enter option to repeat the previous command.

Use Preselection Highlight

When this option is selected, VisualCAD automatically highlights selections as the cursor moves over them. See Pre-Selection Highlight for more information.

Object Snap Settings

You can control the pixel radius of the object snaps located on the Status Toolbar. Enter the Snap Radius desired.

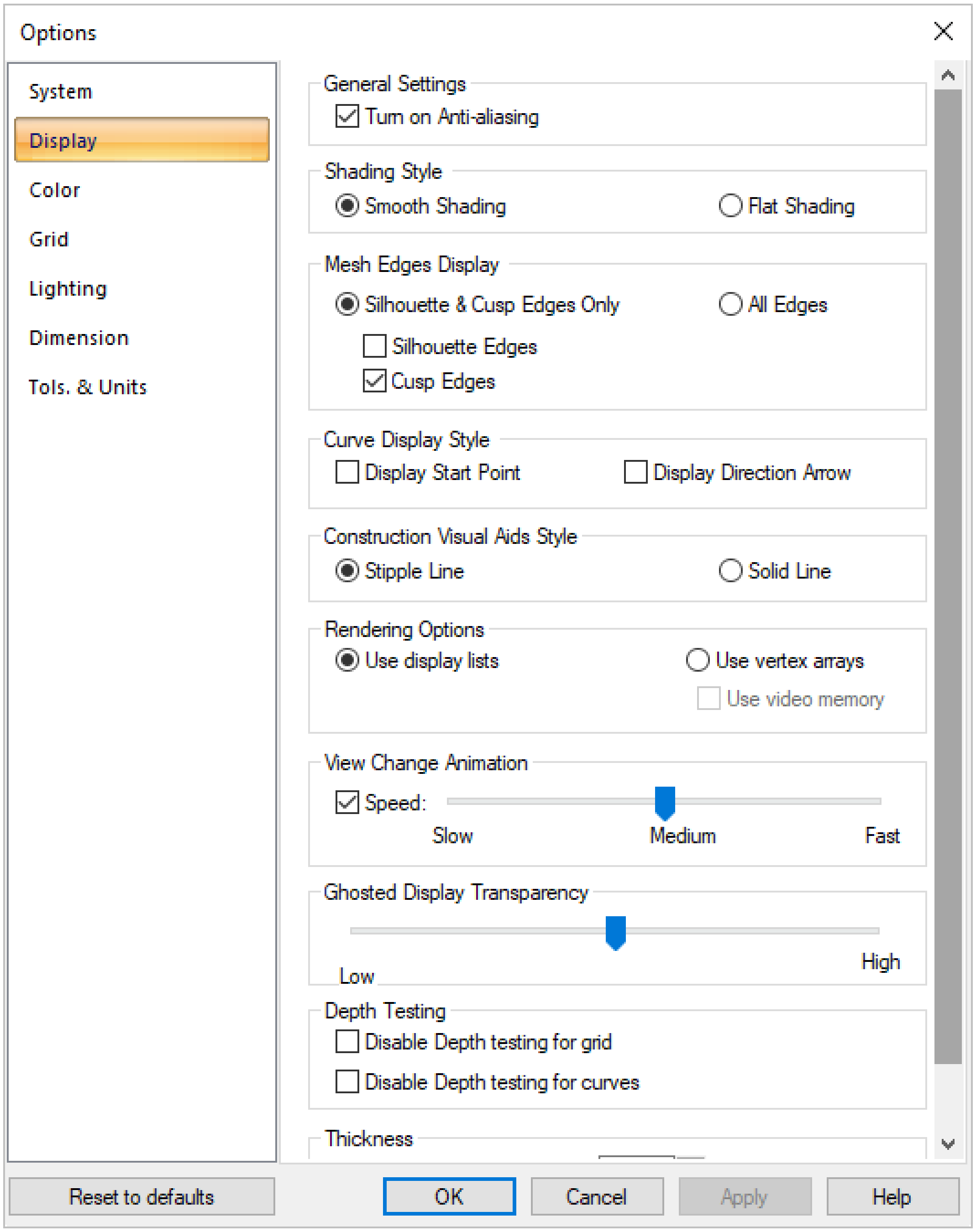
Related Topics

Options

5.1.3.2 Display

Controls how geometry is displayed. Select from the following:

Dialog Box: Options > Display



Dialog Box: Options > Display

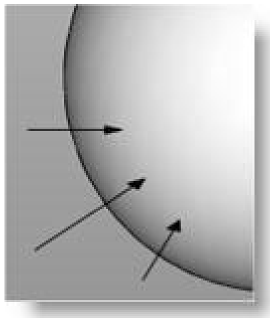
General Settings

Turn on Anti-aliasing

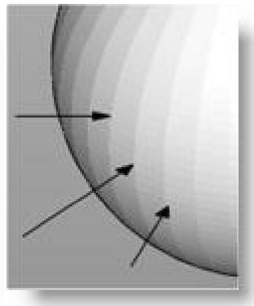
A method of smoothing the jagged edges along the lines and curves of text or graphics. Aliasing is caused by limited display resolution. Selecting this option turns on Anti-aliasing.

Shading Style

This controls how smooth surfaces are shaded.



Smooth Shading

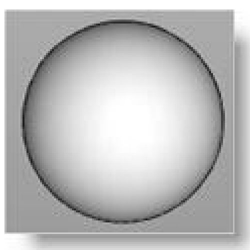


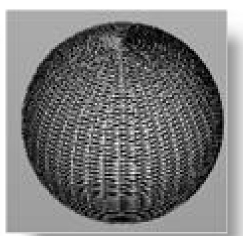
Flat Shading

The pictures demonstrate the difference between smooth and flat shading. The left picture smoothly changes the amount of shading as the surface curves. The right picture just displays the shading for a flat facet on the surface. Since all surfaces (including smooth ones) are internally represented as flat facets, flat shading can affect any surface. The default is smooth shading.

Mesh Edges Display Style

This changes how smooth surfaces are displayed. Since all surfaces are internally represented as flat facets, these facets join up along edges. On smooth surfaces, these edges are not sharp and are not normally displayed. An example of a sharp edge is the edge of a cube.

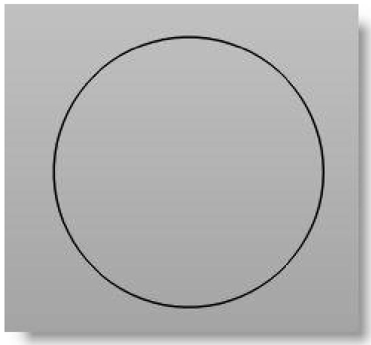




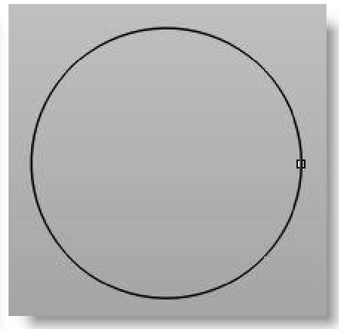
The pictures demonstrate the difference between silhouette & cusp edges only and all edge display. The left picture displays only sharp edges (there are none on a sphere); meanwhile, the right picture displays all the edges. The default is Silhouette & Cusp Edges Only. When selected, this allows you to select Silhouette Edges, Cusp Edges or All Edges to display.

Curve Display Style

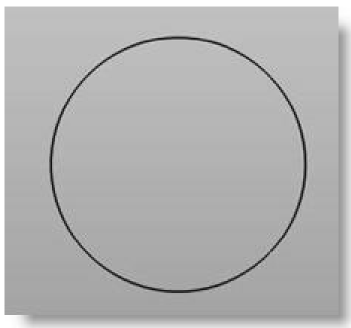
This provides an option to turn on or off display of curve start point and direction arrow.



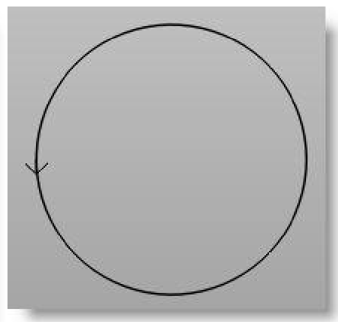
Start Point display OFF



Start Point display ON



Curve Direction arrow OFF

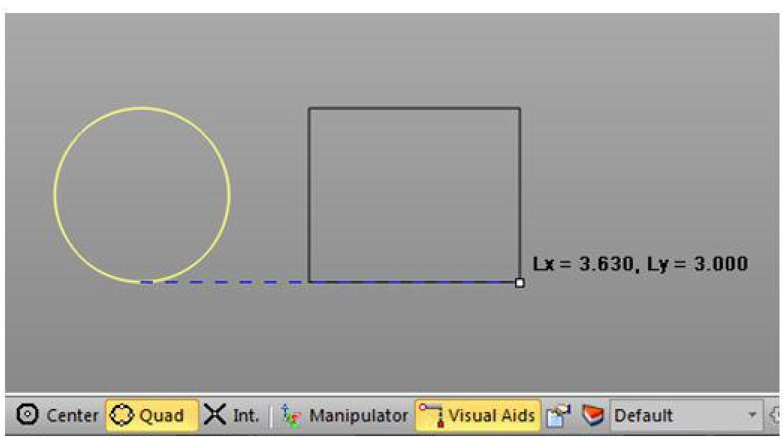


Curve Direction arrow ON

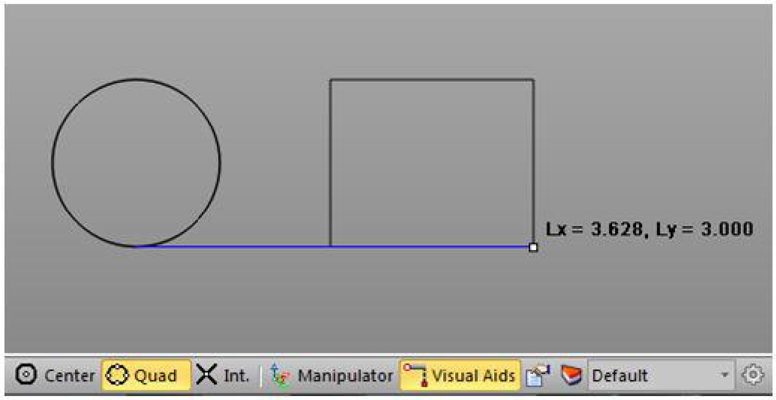
Selecting Start Point and Curve Direction arrow display under options displays both start point and curve direction arrow.

Construction Visual Aids Style

This provides an option to choose the display style between Stipple line and Solid line for visual aids when the Visual Aids is set to active under the status bar.



Stipple line display



Solid line display

Rendering Options

This can be set to Use Display Lists or Vertex Arrays. The Vertex Arrays option also provides the option to Use Video Memory from the video card that is installed on the computer. The speed and performance varies depending on the operating system type and video card installed.

View Change Animation

Whenever the view orientation is changed within the active viewport, VisualCAD can make that orientation change instantly or through smooth animation. If animation is used, the speed of the animation can be set.

Under the View Change Animation function, check the box next to Speed to activate the animation. The animation speed can be controlled using the slider bar.

Ghosted Display Transparency

Whenever the view mode is set to Ghosted Display, you can use this slider to control the transparency level of the hidden (ghosted) geometry.

Depth Testing

These options allow you to fully display certain objects regardless of the view mode. For example, checking the box to Disable Depth Testing for curves, will display all curves even if the view is set to shaded model. You can also Disable Depth testing for grid. If enabled, the grid lines will always display.

Thickness

Use this parameter to set the display line thickness for all wireframe geometry. Set this to 1 for the default line thickness. Higher for thicker lines.

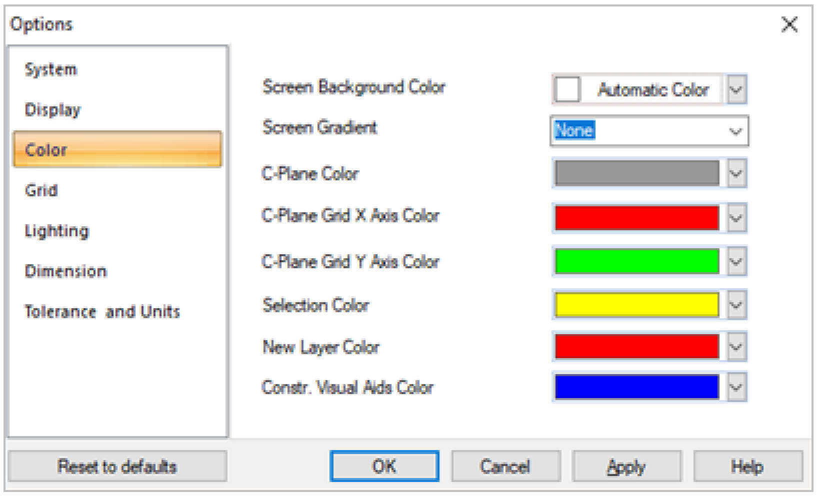
Related Topics

Options

5.1.3.3 Color

The default colors are set using this dialog.

Dialog Box: Options > Color



Dialog Box: Options > Color

- Screen Background Color
  This allows user to set the viewport background color. This is the color displayed when no objects are at that position.
- Screen Gradient
  This is a smooth change of lighting. The direction specified is the lighter side. It starts slightly darker on the opposite side of the screen and gradually gets lighter until it reaches the specified side.
- C-Plane Color
  This changes the color of the construction plane grid.
- C-Plane Grid X Axis Color
  This changes the horizontal axis line of the construction plane.
- C-Plane Grid Y Axis Color
  This changes the vertical axis line of the construction plane.
- Selection Color
  This is the highlight color of objects that have been selected.
- New Layer Color
  This is the default color of objects created on new (not Default) layers.

- Constr. Visual Aids Color
  This is the default color of construction aids when Visual Aids is set to active in the status bar.

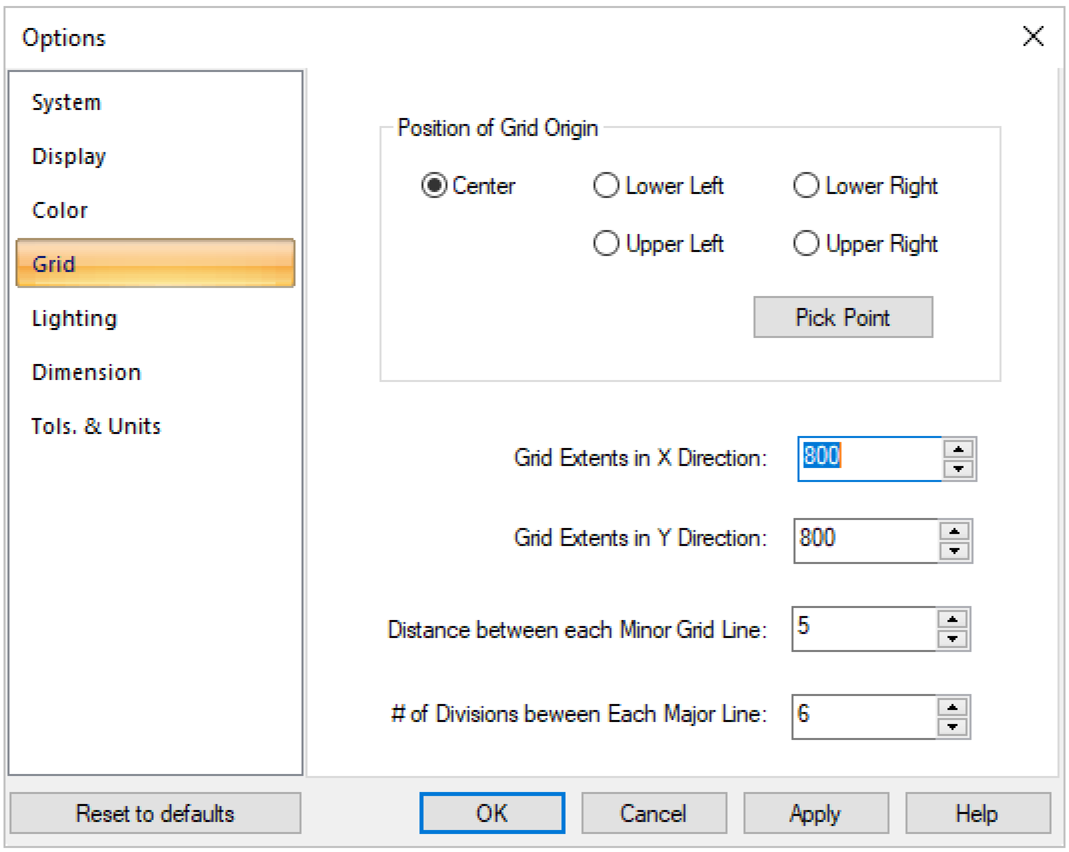
Related Topics

Options

5.1.3.4 Grid

This controls how the construction plane grid is displayed.

Dialog Box: Options > Grid



Dialog Box: Options > Grid

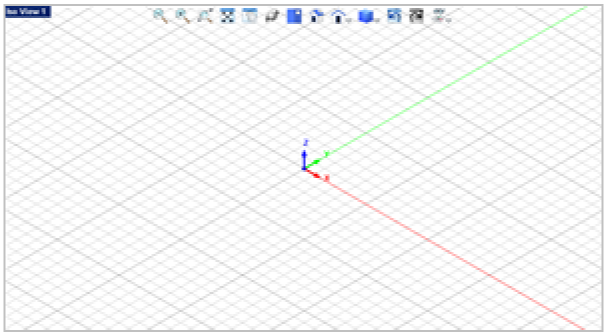
Position of Grid Origin

These options will control where the grid is positioned relative to the World Coordinate System (WCS).

The grid orientation also carries over to whichever view is active, making construction easier.

Center

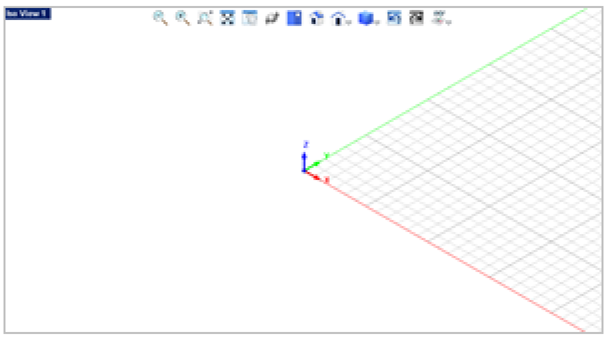
Move the center of the grid to the WCS.



Move the center of the grid to the WCS.

Lower Left

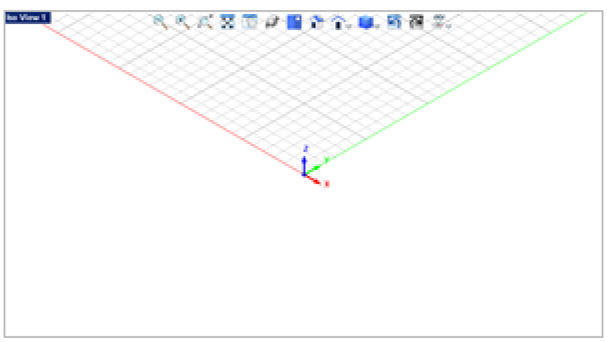
Move the lower left corner of the grid to the WCS.



Move the lower left corner of the grid to the WCS.

Lower Right

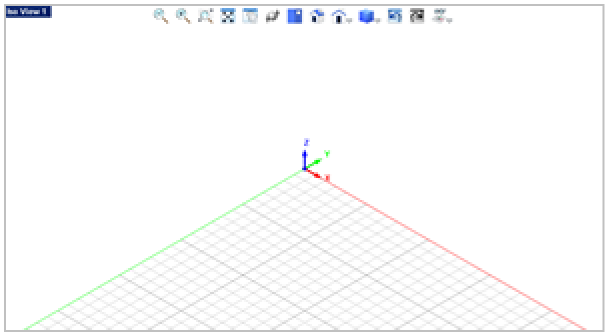
Move the lower right corner of the grid to the WCS.



Move the lower right corner of the grid to the WCS.

Upper Left

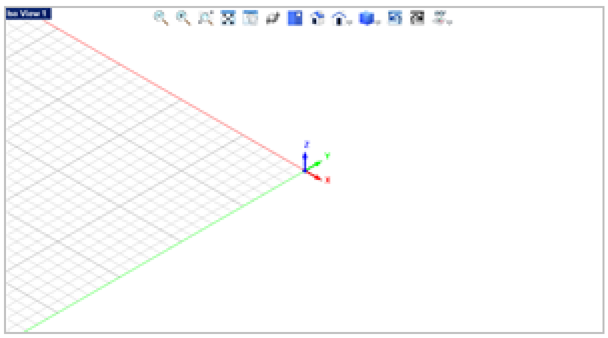
Move the upper left corner of the grid to the WCS.



Move the upper left corner of the grid to the WCS.

Upper Right

Move the upper right corner of the grid to the WCS.



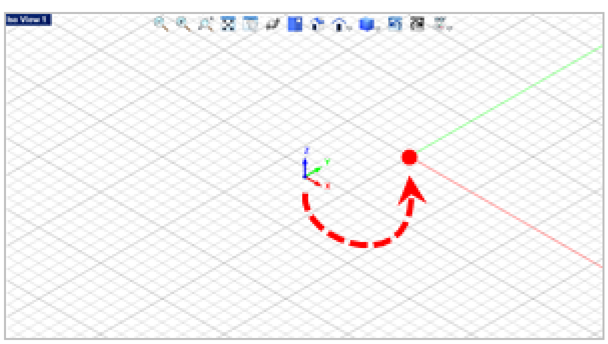
Move the upper right corner of the grid to the WCS.

Pick

Use the button to pick a point on the screen and the origin of the grid will move to this location.

Notes:

- The point you pick is always on the default XY plane.
- If you use the Upper/Lower options above after using this Pick option, the offset distance between the WCS and the new origin of the grid remains in effect.



Move the the origin of the grid to this location.

Other Grid Options

Grid Extents in X Direction

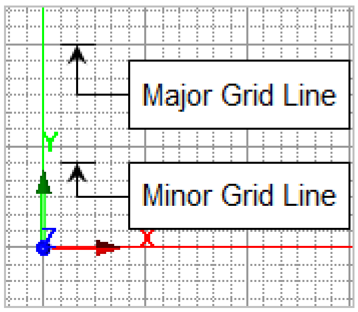
This controls how many major divisions are displayed along the X axis direction.

Grid Extents in Y Direction

This controls how many major divisions are displayed along the Y axis direction.

Distance between each Minor Grid Line

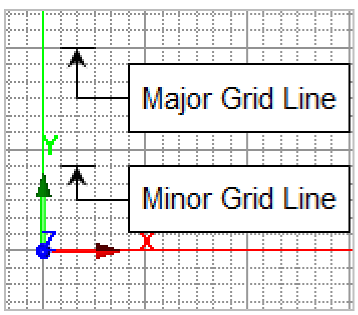
This controls the distance between each minor grid line.



Major/Minor Grid Lines

of Divisions between Each Major Line

This controls the distance between each major grid line. the number of minor rid line multiplied by the number of major grid lines will determine the overall extent of the grid.



Major/Minor Grid Lines

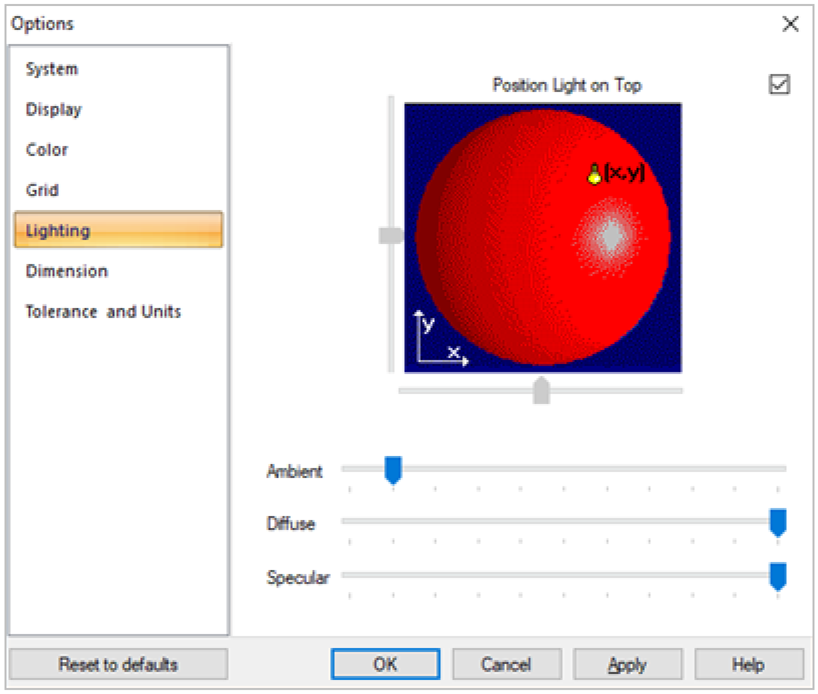
Related Topics

Options

5.1.3.5 Lighting

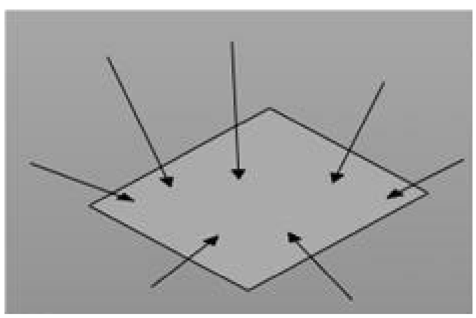
This controls the characteristics of lighting for the model.

Dialog Box: Options > Lighting



Dialog Box: Options > Lighting

There are three types of lighting: ambient, diffuse, and specular.

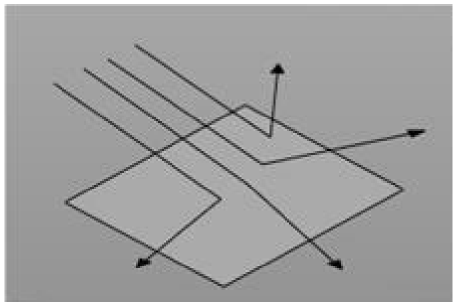


Position Light on Top

The scene has a single light source and there are controls for setting its position. Not all lighting in the model comes directly from that one source; there is some ambient light coming in from all over. The **Position Light on Top** checkbox must be unchecked to be able to move the light source.

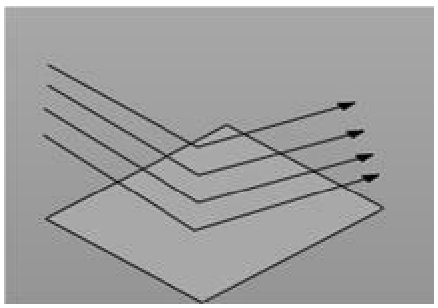
Ambient

Ambient light comes in from all different directions equally. Light sources have no effect on ambient light. Shadows don't appear with only ambient light.



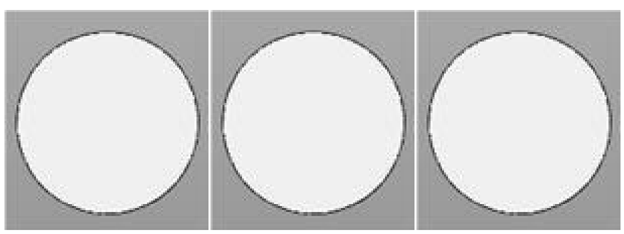
Diffuse

Diffuse light comes from a single source but is scattered in all directions. Shadows appear, but there are no “shiny” spots.

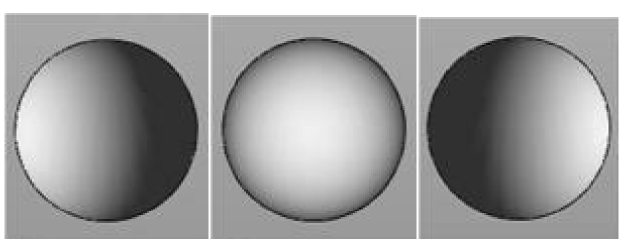


Specular

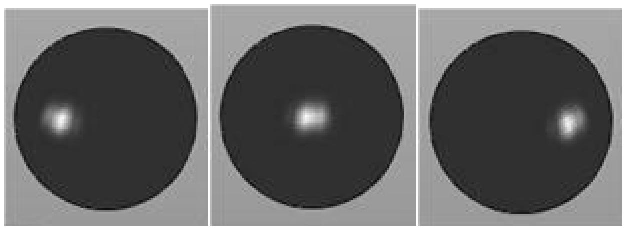
Specular light comes from a single source and is scattered in a single direction. It is reflective light. Shiny metal objects demonstrate good specular reflection. Shading is not a significant consideration with specular light.



The three pictures above demonstrate ambient light as the light source moves from left to right. There is no effect since ambient light comes from all sides equally.



The pictures above show the effects of the motion of the light source using only diffuse lighting. There are no “shiny” spots but yet there is a direction from the light.



The last set of pictures uses only specular lighting. As the light source moves from left to right, the "shiny" spot follows it. However, there is nothing else lit up, so the effect is not just reflective metal.

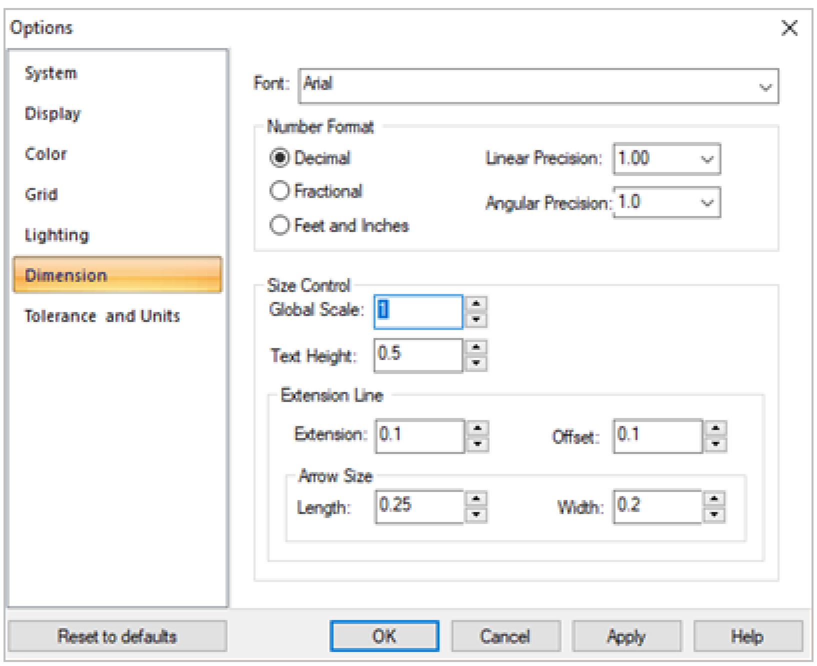
Related Topics

Options

5.1.3.6 Dimension

This controls the appearance of dimensioning labels, arrows, and leader lines.

Dialog Box: Options > Dimension



Dialog Box: Options > Dimension

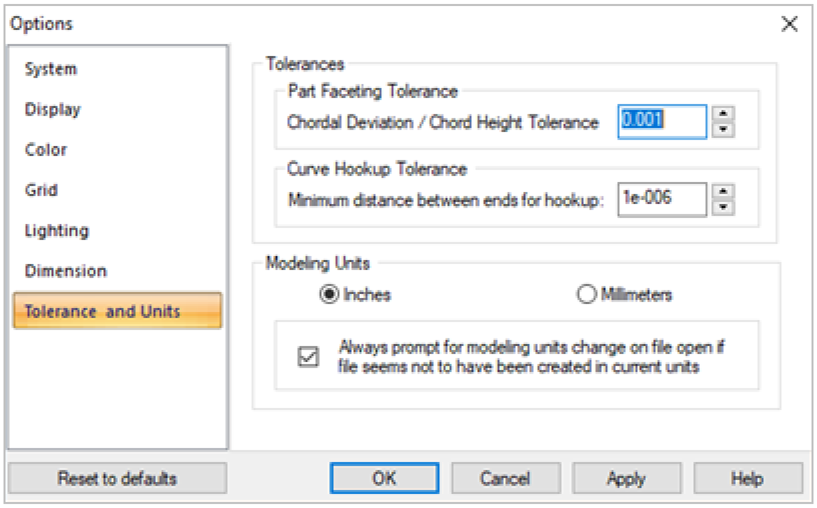
Related Topics

Options

5.1.3.7 Tol. & Units

This controls the accuracy of the model and part units.

Dialog Box: Options > Tolerance and Units



Dialog Box: Options > Tolerance and Units

Tolerances

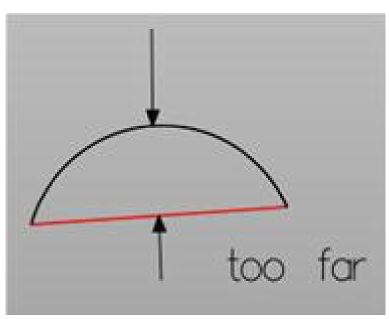
This controls the accuracy of the models. Smaller tolerance numbers often result in significantly big files and slower processing time but are more accurate.

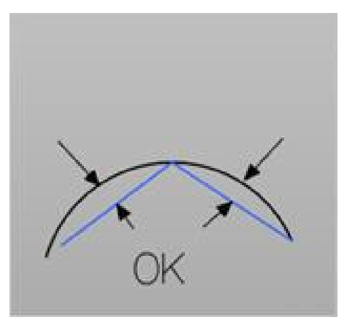
The tolerances are set in a dialog.

Part Faceting Tolerance

Chordal Deviation/Chord Height Tolerance

The chord height tolerance is used in commands that require approximation of smooth curves and surfaces.



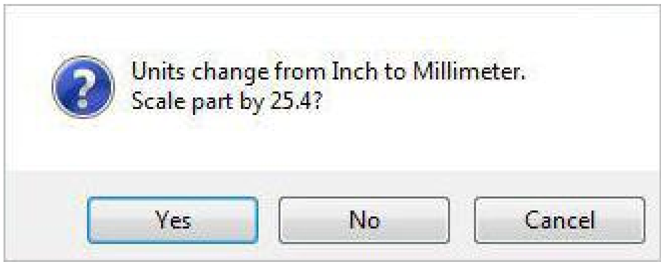


Curve Hookup Tolerance

This specifies maximum separation between the endpoints of different curves when joining them into a single curve.

Units

This converts units from inches to millimeters or from millimeters to inches. If there are existing objects, a dialog will appear confirming if you really want to convert the units.



The objects remain the original size but the coordinate values change.

You can check the box to always prompt you to change the units when files are opened if they appear not to be in the correct units. For example, if the system sees a coordinate value that is quite large such as 500.5, then this dialog will display giving you the opportunity to change the units to Millimeters.

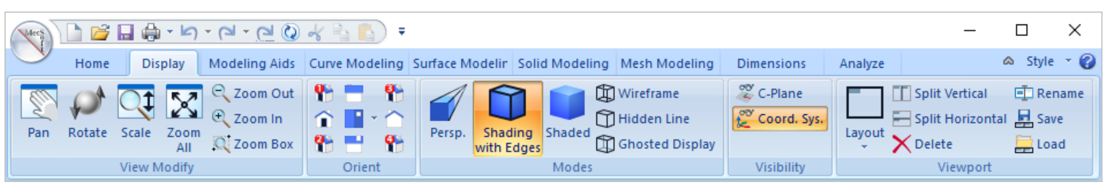
Related Topics

[Options]

5.2 Display

The Display Ribbon Bar contains View Modify, Orient, Modes, Visibility and Viewport panes and associated functions.

The Display Ribbon Bar



Display Ribbon Bar

Related Topics

[View Modify Pane, Display Ribbon Bar]

[Orient Pane, Display Ribbon Bar]

[Modes Pane, Display Ribbon Bar]

[Visibility Pane, Display Ribbon Bar]

[Viewport Pane, Display Ribbon Bar]

[The Ribbon Bar](#)

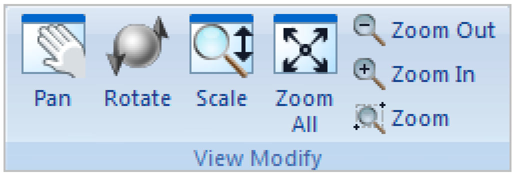
[Customize Quick Access Toolbar](#)

[Customize Dialog](#)

[VisualCAD Options](#)

5.2.1 View Modify

The Display Ribbon Bar contains the following View Modify pane with associated functions.



Display Ribbon Bar
View Modify Pane

Related Topics

[Dynamic Pan View](#)

[Dynamic Rotate View](#)

[Dynamic Zoom](#)

[Fit View](#)

[Zoom Out](#)

[Zoom In](#)

[Zoom Box](#)

[Display Ribbon Bar](#)

5.2.1.1 Dynamic Pan View

The mouse moves the camera left, right, up, or down. It does not change the orientation or rotate.

You can perform dynamic pan by holding down the shift key on your keyboard and using right mouse button click.

Related Topics

[View Bar Options](#)

5.2.1.2 Dynamic Rotate View

The mouse moves the camera around the model. The center of the view is the center of rotation. You can perform dynamic rotate using right mouse button click.

Related Topics

View Bar Options

5.2.1.3 Dynamic Zoom

This moves the camera towards or away from the center of view with the mouse. Hold down the left mouse-button while moving the mouse forward (zooms in – makes things appear larger) and backward (zooms out – makes things appear smaller).

You can perform dynamic zoom by holding down the control key on your keyboard and using right mouse button click.

Related Topics

View Bar Options

5.2.1.4 Fit View

This changes the position of the camera to so that all objects are in view. It does not change the orientation or rotation. If a 3D object (surface, solid or mesh) is selected, the Fit View command will fit the view to the selected objects.

Note: With the VisualMILL plug-in module loaded, Fit View will also account for the Stock model if Stock display is toggled on.

Related Topics

View Bar Options

5.2.1.5 Zoom Out

The camera backs up to make the objects appear twice as small. The dimensions of the objects don't change, only the position of the camera.

Related Topics

View Bar Options

5.2.1.6 Zoom In

The camera moves forward to make the objects appear twice as large. The dimensions of the objects don't change, only the position of the camera.

Related Topics

View Bar Options

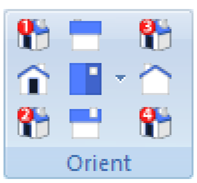
5.2.1.7 Zoom Box

This creates a rubber-banded rectangle surrounding an area. The camera is moved so that the selected area fills the view. If the area's aspect ratio doesn't match the view's ratio, the area is expanded so that the entire selected area is in view. The dimensions of the objects don't change, only the position of the camera.

Related Topics

View Bar Options

5.2.2 Orient



The Display Ribbon Bar contains the following Orient pane with associated functions.

Related Topics

Top View

Bottom View

Front View

Back View

Left View

Right View

Iso View 1-4

Display Ribbon Bar

5.2.2.1 Top View

Changes the currently selected view to look down to the origin. The camera is positioned at a positive z-height on the z-axis. The x-axis points to the right, the y-axis points up, and the z-axis points out of the screen towards you. If geometry is not at the origin, it might be out of view after top view is selected.

Related Topics

View Bar Options

5.2.2.2 Bottom View

Changes the currently selected view to look up to the origin. The camera is positioned at a negative z-height on the z-axis. The x-axis points to the right, the y-axis points down, and the z-axis points into the screen away from you. If geometry is not at the origin, it might be out of view after bottom view is selected.

Related Topics

View Bar Options

5.2.2.3 Front View

Changes the currently selected view to look at the origin from the front. The camera is position at a negative y-coordinate on the y-axis. The x-axis points to the right, the y-axis points away from you, and the z-axis points up. If geometry is not at the origin, it might be out of view after front view is selected.

Related Topics

View Bar Options

5.2.2.4 Back View

Changes the currently selected view to look at the origin from the back. The camera is position at a positive y-coordinate on the y-axis. The x-axis points to the left, the y-axis points towards you, and the z-axis points up. If geometry is not at the origin, it might be out of view after back view is selected.

Related Topics

View Bar Options

5.2.2.5 Left View

Changes the currently selected view to look at the origin from the left. The camera is position at a negative x-coordinate on the x-axis. The x-axis points away from you, the y-axis points to the left, and the z-axis points up. If geometry is not at the origin, it might be out of view after left view is selected.

Related Topics

View Bar Options

5.2.2.6 Right View

Changes the currently selected view to look at the origin from the right. The camera is position at a positive x-coordinate on the x-axis. The x-axis points towards you, the y-axis points to the right, and the z-axis points up. If geometry is not at the origin, it might be out of view after right view is selected.

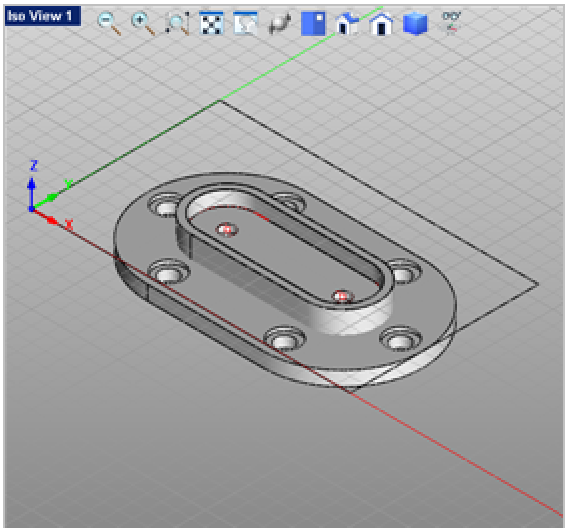
Related Topics

View Bar Options

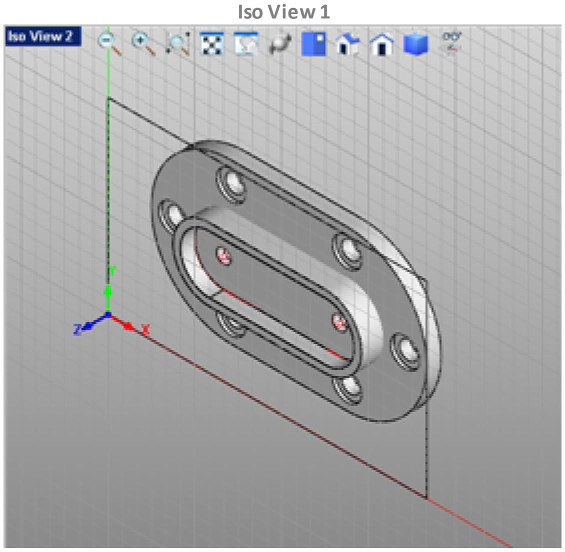
5.2.2.7 Iso View 1-4

There are 4 preset Iso Views to choose from. Selecting one of the 4 icons changes the currently active viewport to the assigned Iso View direction. Note that if geometry is not at the origin, it might be out of view after an Iso View is selected.

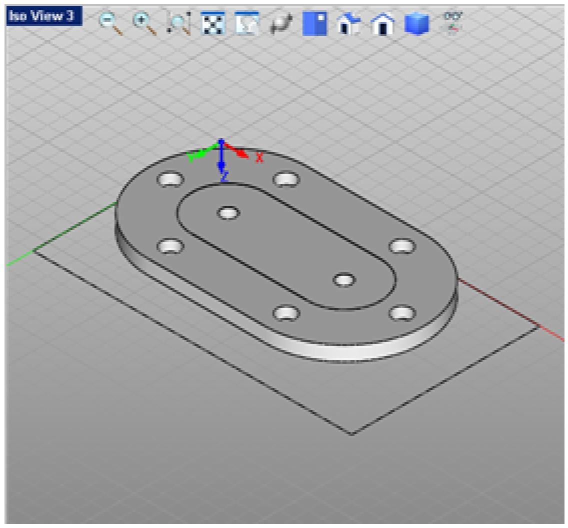
Iso View Examples



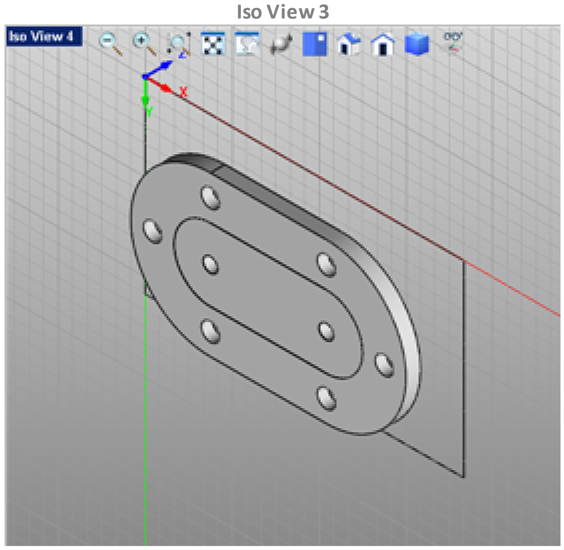
Iso View 1



Iso View 2



Iso View 3



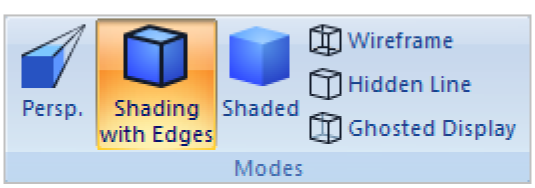
Iso View 4

Related Topics

View Bar Options

5.2.3 Modes

The Display Ribbon Bar contains the following Modes pane with associated functions.



Display Ribbon Bar
Modes Pane

Related Topics

[Perspective / Parallel]

[Toggle Shading + Edges]

[Shaded Display]

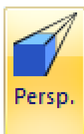
[Wireframe Display]

[Toggle Hidden Lines]

[Toggle Ghosted Display]

[Display Ribbon Bar]

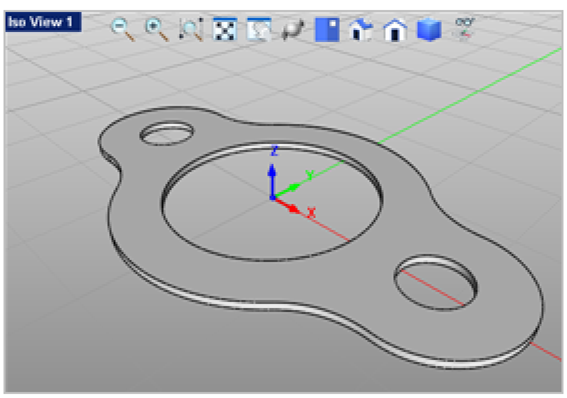
5.2.3.1 Perspective / Parallel



Switch the currently selected view between a perspective view and a parallel view. The default is parallel view. A perspective view shows nearer objects as bigger than normal while objects that are farther away are shown smaller than normal. Even nearer faces of volumes will be bigger than normal while the farther away faces are smaller. This is a standard are technique to convey distance but distorts the appearance of objects.

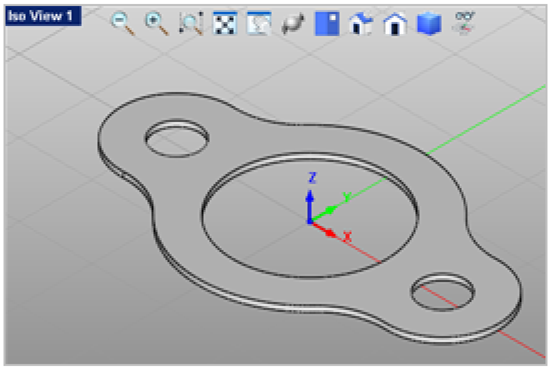
A parallel view maintains the size of all objects, regardless of apparent distance.

Perspective Enabled



Perspective Enabled

Parallel Enabled



Parallel Enabled

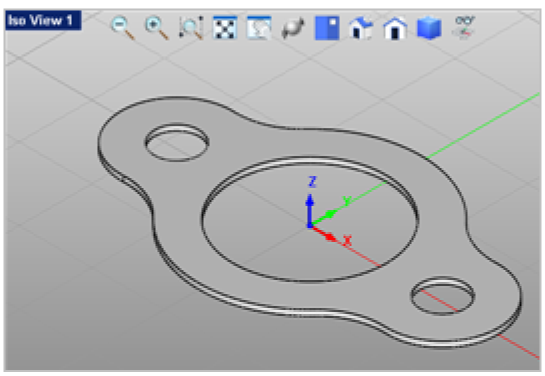
Related Topics

View Bar Options

5.2.3.2 Toggle Shading + Edges

All surfaces and volumes in the currently selected view are filled in with colors. The edges of surfaces and volumes are outlined in black. The surface color is modified to display shading and lighting effects. There is a light source that can cause shadows and glossy appearances depending on the position of the geometry and how it is viewed.

Shading + Edges Enabled



Shading + Edges Enabled

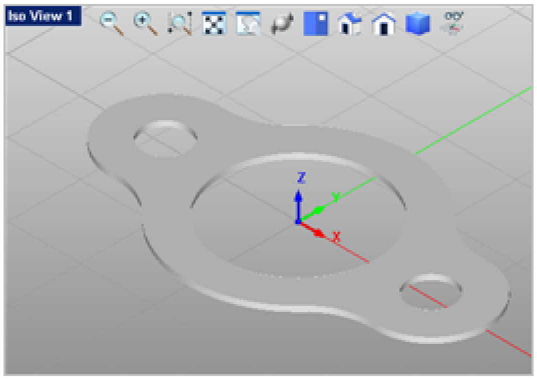
Related Topics

View Bar Options

5.2.3.3 Shaded Display

All surfaces and volumes in the currently selected view are filled in with colors. The color is modified to display shading and lighting effects. There is a light source that can cause shadows and glossy appearances depending on the position of the geometry and how it is viewed.

Shading Display Enabled



Shading Display Enabled

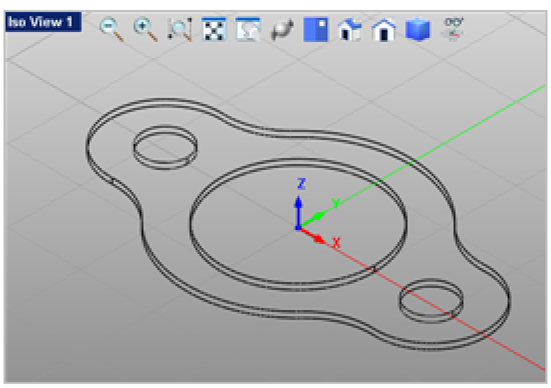
Related Topics

[View Bar Options]

5.2.3.4 Wireframe Display

All surfaces and volumes in the currently selected view are transparent. Only the edges are displayed. Even the edges that would normally be on the backside of a volume and not normally be displayed show up. If one object is behind another, the farther away object is visible through the nearer object.

Wireframe Display Enabled



Wireframe Display Enabled

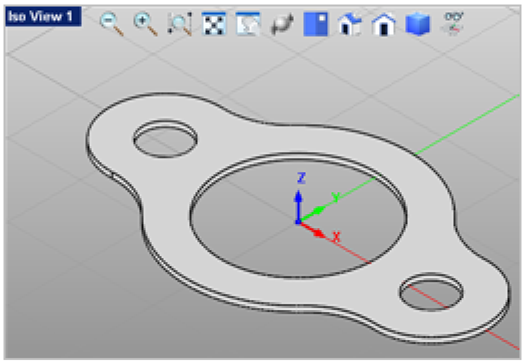
Related Topics

[View Bar Options]

5.2.3.5 Hidden Line Display

All surfaces and volumes in the currently selected view are filled in with a neutral gray. The edges of surfaces and volumes are displayed in the normal color for that layer. No lighting or shading effects are displayed.

Hidden Line Display Enabled



Hidden Line Display Enabled

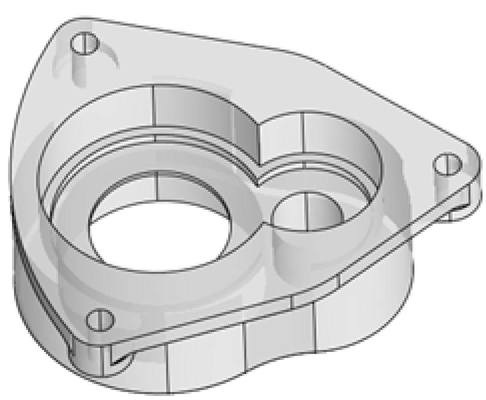
Related Topics

[View Bar Options]

5.2.3.6 Ghosted Display

This display mode is a combination of Shaded and Hidden Line where the hidden areas of the part are displayed in a sub-tone color of the part's primary layer color. This display mode give you an understanding the hidden areas of the part. You can control the transparency level (low to high) from the Display Options dialog.

Ghosted Display Enabled



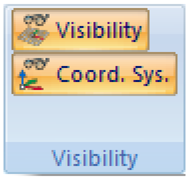
Ghosted Display Enabled

Related Topics

View Bar Options

Display Options

5.2.4 Visibility



The Display Ribbon Bar contains the following Visibility pane with associated functions.

Related Topics

Hide/Show Grid

Hide/Show WCS

Display Ribbon Bar

5.2.4.1 Hide/Show Grid

Switches on and off the display of the construction plane in the currently selected view. The default is to show the construction plane (grid).

Related Topics

View Bar Options

5.2.4.2 Hide/Show WCS

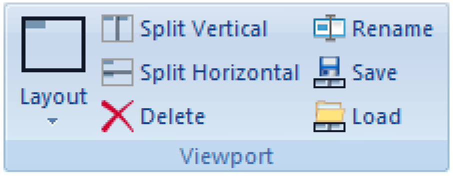
Switches on and off the display of the World Coordinate System in the currently selected view. The World Coordinate System is the three vectors at the origin, which show the positive x, y, and z axes. The default is to show the WCS.

Related Topics

View Bar Options

5.2.5 Viewport

The Display Ribbon Bar contains the following Viewport pane with associated functions.



Display Ribbon Bar
Viewport Pane

Related Topics

[Viewport Layouts]

[Viewport Menu]

[Split Viewport]

[Delete Viewport]

[Rename Viewport]

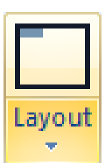
[Save Viewport]

[Load Viewport]

[Display Ribbon Bar]

5.2.5.1 Viewport Layouts

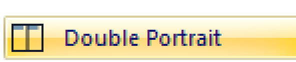
The next eight options control the viewport layout on the screen according to the following table. These are various useful arrangements of the views.



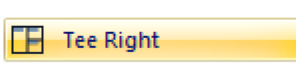
Single Viewport

Top View

Top View
Iso View



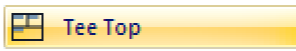
Top View	Iso View



Iso View	Top View
	Right View

Top View	Iso View
Front View	

Iso View	
Top View	Right View



Top View	Right View
Iso View	

Top View	Iso View
Front View	Right View

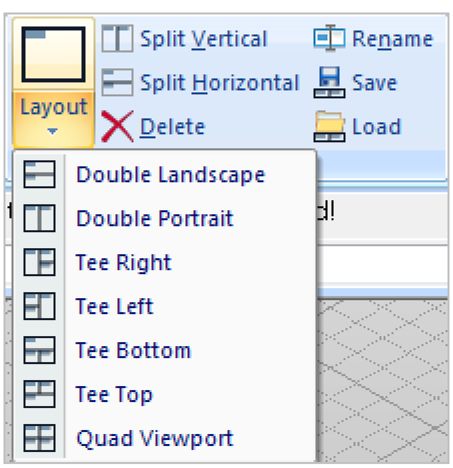
Related Topics

[Viewport]

5.2.5.1.1 Viewport Menu

Use this menu to change the configuration of viewport layouts. A viewport is the section of the screen that displays what a camera would see if it were positioned at a particular location and orientation. The camera will see all the geometry and other objects currently in view with correct lighting and color. The view can be changed by panning or rotating.

Show me the Viewport Menu



Viewport Menu

Right-click Menu from Viewport Title

More about Viewports

Viewports have titles like Top, Right, Left, and Iso (Isometric). The way that the construction plane is aligned is also associated with each different title. For example, both Top and Iso have the construction plane in the z-plane; meanwhile, Left and Right have the construction plane in the y-plane, and Front and Back have the construction plane in the x-plane.

Each of the named views has a starting position and orientation. These positions correspond to the name (i.e. Top is on top of the model, at a large z position (0, 0, big +z). Both the icons and the names suggest looking at a model (a house) from the top, left, right, front, and back. To distinguish between left and right, notice the little chimney on top of the house. It is visible from one side but not the other. The viewport will always start in the same location.

It is possible that objects may be visible in one view but out of view in another. When you change views, some objects won't appear, but they are still there.

Multiple viewports can be displayed at the same time. Commonly used configurations can be selected with the toolbar button.

Selecting one of these buttons immediately changes the view port configuration. While some objects might suddenly become out-of-view, they are still there. Panning or rotating any view should allow them to be seen.

Viewport menu can also be invoked when you right mouse button click on any viewport title.

Related Topics

Split Viewport

Delete Viewport

Viewport Layouts

Rename Viewport

Align to C-Plane

5.2.5.2 Split Viewport

The first two options under Viewport are used to split the currently active view into two views, either vertically or horizontally. Any other view that exists in the Viewport layout will remain unchanged.

Top View
Iso View

Top View	Iso View

Related Topics

[Viewport]

5.2.5.3 Delete Viewport

The third option will delete the currently active view from the viewport layout. Instantly, one of the adjacent views will fill in the vacated space in the viewport layout. The Delete Viewport option will cause no action if the current viewport layout is a single view.

Related Topics

[Viewport]

5.2.5.4 Rename Viewport

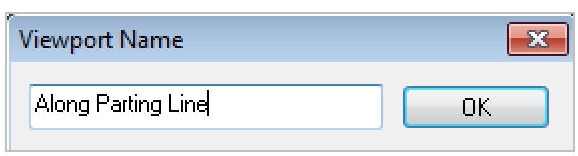
This option allows the currently active view to be renamed. The name change is temporary and neither the name nor the viewport are saved for later reference.

Related Topics

[Viewport]

5.2.5.5 Save Viewport

This option allows the currently active Viewport to be Saved for later recall. Complete the Viewport Name dialog and then pick OK to save it. Then at any time you can use the [Load Viewport] command to reload it as needed.



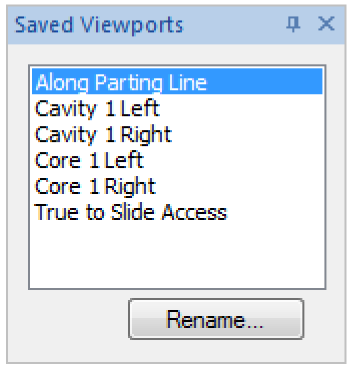
Viewport Name Dialog

Related Topics

[Load Viewport]

5.2.5.6 Load Viewport

This option allows you to Load a Viewport that was previously saved using the Save Viewport command. When you select the Load Viewport icon the following Saved Viewports dialog is displayed. Select a saved viewport from the list and it will be displayed in the currently active viewport. To rename a saved viewport, select it and then pick the Rename ... button.



Saved Viewport Dialog

Related Topics

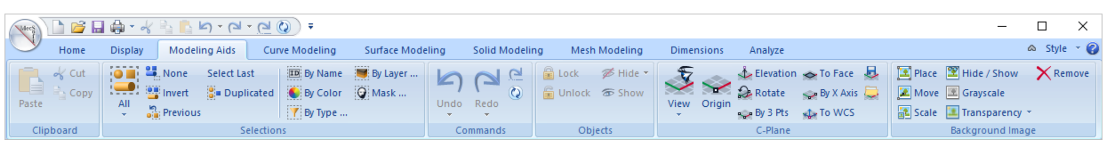
Viewport

Save Viewport

5.3 Modeling Aids

The Modeling Aids Ribbon Bar contains Clipboard, Selections, Commands, Objects and C-Plane panes and associated functions.

The Modeling Aids Ribbon Bar



Modeling Aids Ribbon Bar

Related Topics

Clipboard Pane, Modeling Aids Ribbon Bar

Selections Pane, Modeling Aids Ribbon Bar

Commands Pane, Modeling Aids Ribbon Bar

Objects Pane, Modeling Aids Ribbon Bar

C-Plane Pane, Modeling Aids Ribbon Bar

[Background Image Pane, Modeling Aids Ribbon Bar]

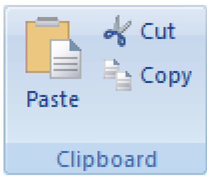
[The Ribbon Bar]

[Customize Quick Access Toolbar]

[Customize Dialog]

[VisualCAD Options]

5.3.1 Clipboard



The Modeling Aids Ribbon Bar contains the following Clipboard pane with associated functions.

Related Topics

[Cut Selection]

[Copy Selection]

[Paste Selection]

[Modeling Aids Ribbon Bar]

5.3.1.1 Cut Selection

Objects will be deleted, but will remain on the clipboard until replaced by the next items moved/copied to the clipboard. If no objects are selected when the command is executed, objects can be picked.

Related Topics

[Copy Selection]

[Paste Selection]

5.3.1.2 Copy Selection

Objects will be copied onto the clipboard. If no objects are selected when the command is executed, objects can be picked.

Related Topics

[Cut Selection]

[Paste Selection]

5.3.1.3 Paste Selection

All objects on the clipboard will be put at the cursor's position when the command was executed.

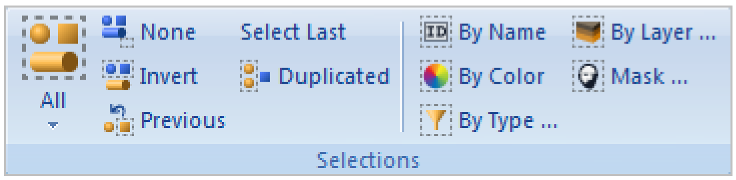
Related Topics

Cut Selection

Copy Selection

5.3.2 Selections

The Modeling Aids Ribbon Bar contains the following Selections pane with associated functions.



Modeling Aids Ribbon Bar
Selections Pane

These commands allow you to select certain geometry to apply commands to as a group. Commands such as Delete or Scale will apply to all of the currently selected geometry in the same way and at the same time. Selected geometry will be highlighted with the highlight color. The default highlight color is yellow. Selected items can also be on multiple layers.

Multiple items can be selected by holding down the control key (on the keyboard) and left-clicking on the item with the mouse.

Selections do not apply to geometries that area hidden, locked and on layers that are not visible and or locked.

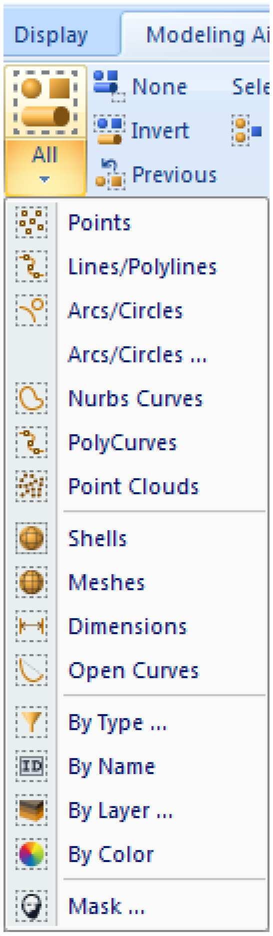
Related Topics

Select Menu

- Select Points
- Select Lines/Polylines
- Select Arcs/Circles
- Select Arcs/Circles ...
- Select Nurbs Curves
- Select Polycurves

5.3.2.1 Select Menu

Pick the indicator to drop down the Select Menu.



Select Menu

5.3.2.1.1 Select Points

All points will be selected. All other geometry will be deselected.

Related Topics

Select Menu

Selections Pane, Modeling Aids Ribbon Bar

5.3.2.1.2 Select Lines/Polylines

All lines and polylines will be selected. All other geometry will be deselected.

Related Topics

[Select Menu]

[Selections Pane, Modeling Aids Ribbon Bar]

5.3.2.1.3 Select Arcs/Circles

All arcs and circles will be selected. All other geometry will be deselected.

Related Topics

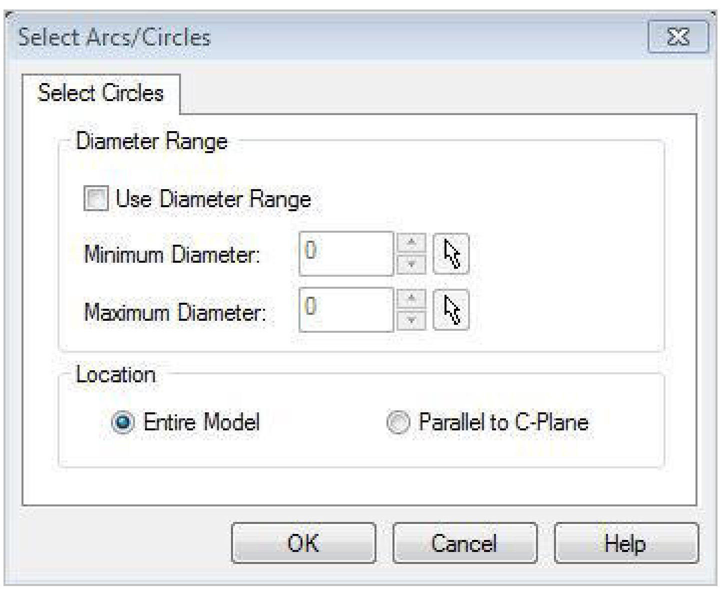
[Select Menu]

[Selections Pane, Modeling Aids Ribbon Bar]

5.3.2.1.4 Select Arcs/Circles ...

This function selects arcs and circles only, but gives the option to be selective to a range of diameter values and orientation.

Select Arcs/Circles dialog box



Select Arcs/Circles dialog box

- Diameter Range
  If the diameter range is used, the diameters may be entered directly or existing arcs may be picked on the screen to set these values.

- Location
  Another option restricts the selection to only arcs and circles that are Parallel (co-planar) to the C-Plane.

Related Topics

Select Menu

Selections Pane, Modeling Aids Ribbon Bar

5.3.2.1.5 Select Nurbs Curves

All NURBS will be selected. All other geometry will be deselected.

Related Topics

Select Menu

Selections Pane, Modeling Aids Ribbon Bar

5.3.2.1.6 Select Polycurves

All curves made of sub curves (e.g. chained curves) will be selected. All other geometry will be deselected.

Related Topics

Select Menu

Selections Pane, Modeling Aids Ribbon Bar

5.3.2.1.7 Select Point Clouds

Use this command to select Point Cloud objects. Currently, the Point Cloud object can only be created from corresponding data during Import.

Related Topics

Select Menu

Selections Pane, Modeling Aids Ribbon Bar

5.3.2.1.8 Select Shells

All surfaces & poly-surface objects (not meshes) will be selected. All other geometry will be deselected.

Related Topics

[Select Menu]

[Selections Pane, Modeling Aids Ribbon Bar]

5.3.2.1.9 Select Meshes

All meshes will be selected. All other geometry will be deselected.

Related Topics

[Select Menu]

[Selections Pane, Modeling Aids Ribbon Bar]

5.3.2.1.10 Select Dimensions

All Dimension labels and leader lines will be selected. All other geometry will be deselected.

Related Topics

[Select Menu]

[Selections Pane, Modeling Aids Ribbon Bar]

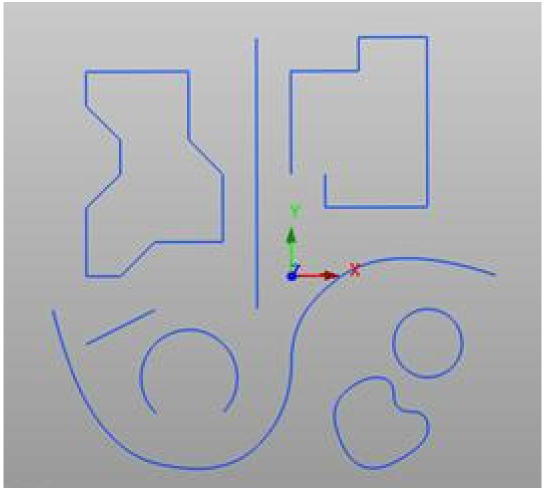
5.3.2.1.11 Select Open Curves

All curves including polylines, poly-curves, NURBS curves that are open will be selected. All other geometry will be deselected.

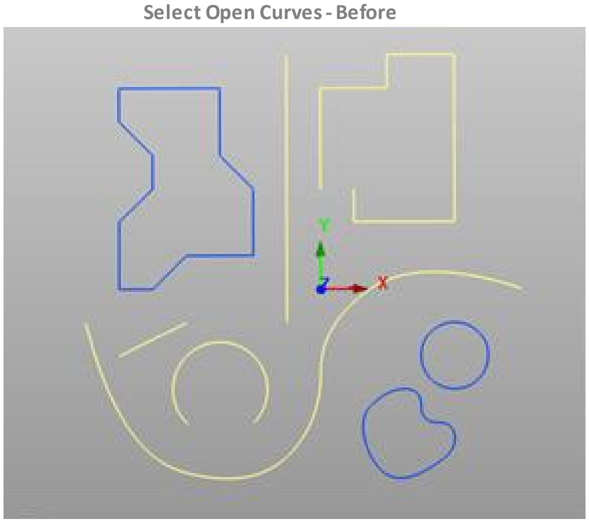
The Open Curves selection function is found under Select on the Menu Bar.

This function will automatically identify all curves on the screen that have an 'open' end that is not connected to another curve. These curves will be selected and highlighted.

Select Open Curves example



Select Open Curves - Before



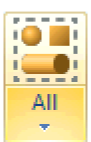
Select Open Curves - Before

Related Topics

Select Menu

Selections Pane, Modeling Aids Ribbon Bar

5.3.2.2 Select All



All geometry on all layers is selected.

Related Topics

Select Menu

Selections Pane, Modeling Aids Ribbon Bar

5.3.2.3 Select None

Everything is deselected.

Related Topics

Select Menu

Selections Pane, Modeling Aids Ribbon Bar

5.3.2.4 Invert Selections

All geometry currently selected will be deselected; meanwhile, currently unselected items will be selected and highlighted.

Related Topics

Select Menu

Selections Pane, Modeling Aids Ribbon Bar

5.3.2.5 Select Previous

Geometries that were previously selected will be re-selected. All currently selected items will be deselected.

Related Topics

Select Menu

Selections Pane, Modeling Aids Ribbon Bar

5.3.2.6 Select Last

The geometry created last will be selected.

Related Topics

Select Menu

Selections Pane, Modeling Aids Ribbon Bar

5.3.2.7 Select Duplicated

Selects objects that are geometrically identical with another object and visible, regardless of other object properties.

Related Topics

[Select Menu]

[Selections Pane, Modeling Aids Ribbon Bar]

5.3.2.8 Select by Name

Geometry will be selected whose name matches a name that you enter. All other geometry will be deselected.

Related Topics

[Select Menu]

[Selections Pane, Modeling Aids Ribbon Bar]

5.3.2.9 Select By Color

Select an entity and then all geometry with the same color is also selected automatically. All other geometry will be deselected.

Related Topics

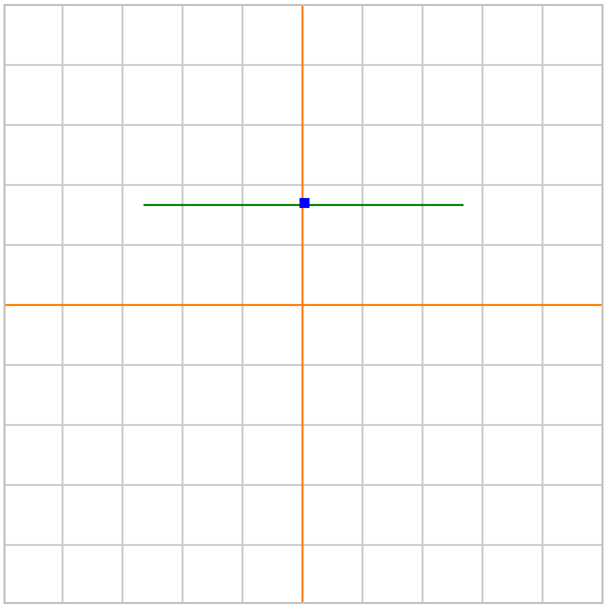
[Select Menu]

[Selections Pane, Modeling Aids Ribbon Bar]

5.3.2.10 Select By Type ...

Geometry of the desired type will be selected. All other geometry will be deselected.

Object Selection Filters dialog box



Object Selection Filters dialog box

- Selectable Types
  Check the box next to the object types that you wish to select.
- Select All / Clear All
  Selects or Clears all types selected.

Related Topics

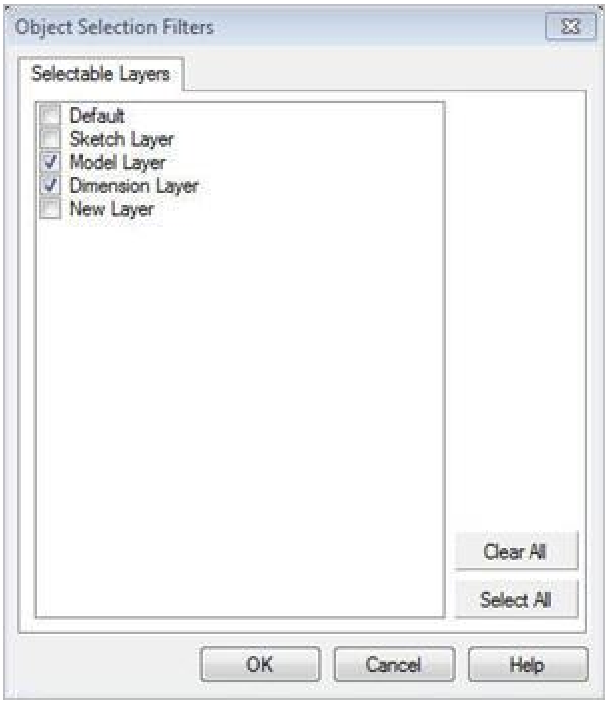
[Select Menu]

[Selections Pane, Modeling Aids Ribbon Bar]

5.3.2.11 Select By Layer ...

All geometry on the desired layer will be selected. All other geometry will be deselected.

Object Selection Filters dialog box (by layer)



Object Selection Filters dialog box (by layer)

- Selectable Types
  Check the box next to the object types that you wish to select.
- Select All / Clear All
  Selects or Clears all types selected.

Related Topics

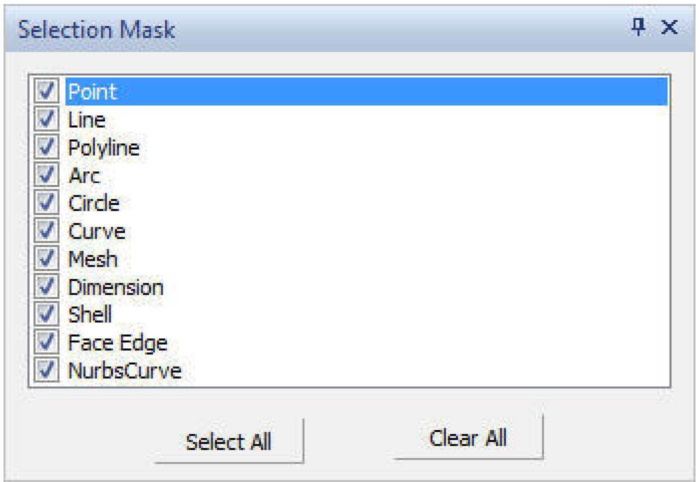
Select Menu

Selections Pane, Modeling Aids Ribbon Bar

5.3.2.12 Selection Mask ...

Use this to set masks based on object types for subsequent selection operations. All other geometry will be deselected. This can be selected from the Menu bar by clicking on Select, then choosing Selection Mask from the list. Geometries can be selected either by clicking on Select from Menu bar and selecting All or dragging a window with mouse to select the geometry in the viewport.

Selection Mask dialog box



Selection Masks

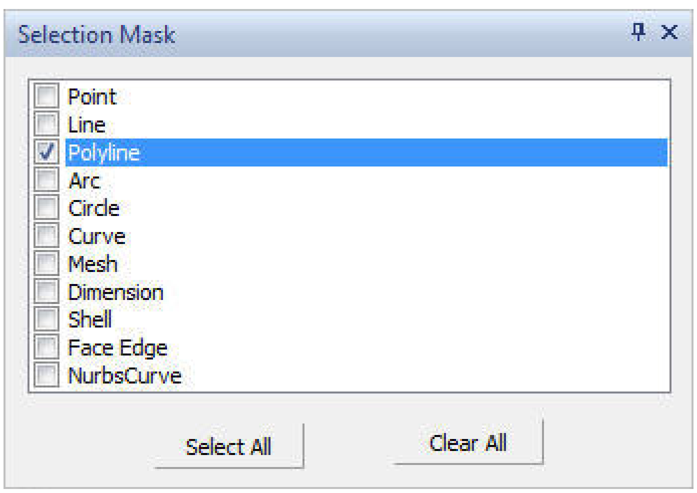
Check the box next to the mask types that you wish to select.

Select All / Clear All

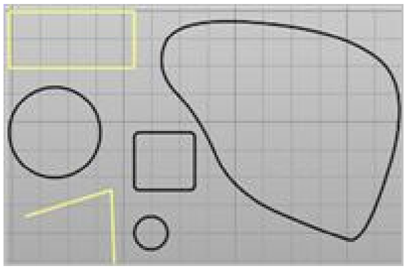
Selects or Clears all types selected.

Selection Mask dialog box example

Example shown displays the Selection Mask for Polyline only and the only geometry selected in the viewport are the polylines.



Geometry can be selected either by clicking on Select from Menu bar and selecting All or dragging a window by holding left mouse to select the geometry.



Selection Mask for Polyline results

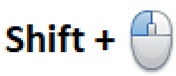
Related Topics

[Select Menu]

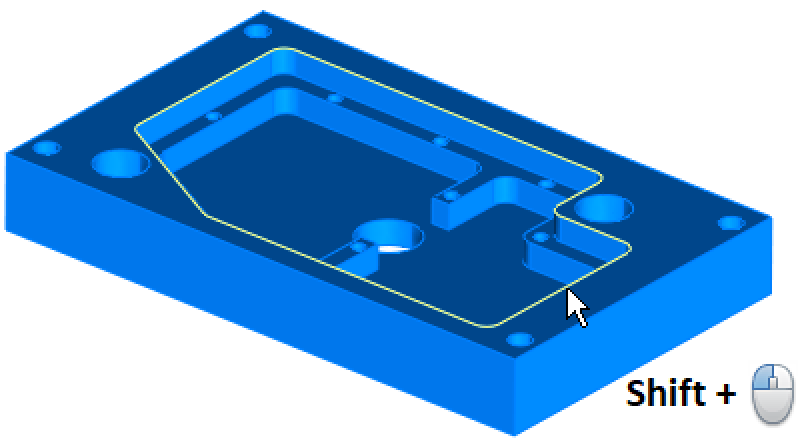
[Selections Pane, Modeling Aids Ribbon Bar]

5.3.2.13 Other Selection Techniques

5.3.2.13.1 Select Chain



You can automatically select a chain of curves or edges by pressing the <Shift> key while performing a left-click selection. This works with any curves or edges that are connected end-to-end. If you want to create a Curve Chain entity see [Create a Curve Chain]. Note that this is not a command and is not located on a menu.



Chain Select (<Shift> + Left-click)

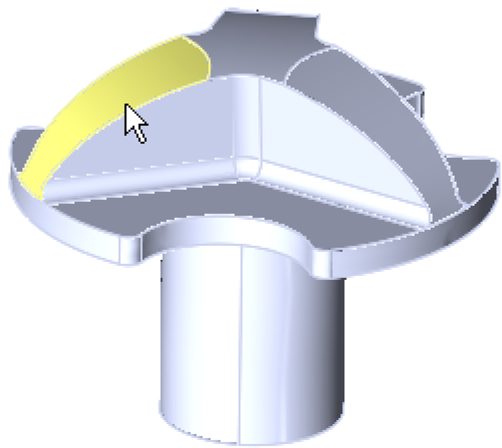
Related Topics

[Select Menu]

[Create a Curve Chain]

5.3.2.13.2 Highlights

Objects that are available for selection are highlighted when the cursor moves over them. This functionality is not a command and is not located on a menu. This functionality is enabled/disabled from the [System Options] dialog.



Pre-Selection Highlight

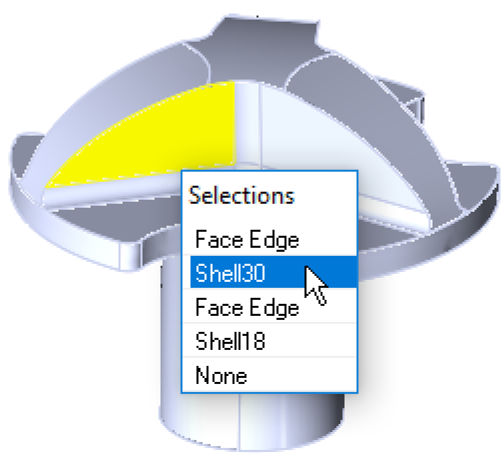
Related Topics

[Select Menu](#)

System Options

5.3.2.13.3 Multi-Selections

When you make a selection and there are multiple objects available, the Multi-Selection Dialog will display listing the available objects. You can select an object from the list. If you select None or pick outside of the list, the dialog will close automatically.



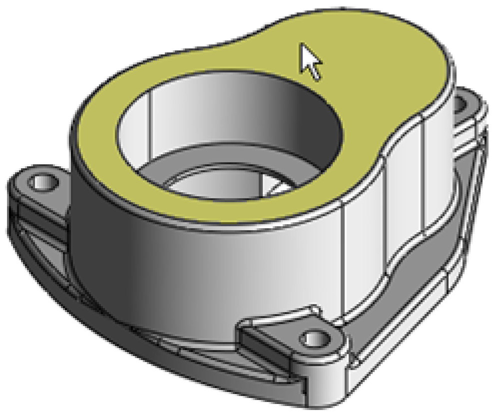
Multi-Selection Dialog

Related Topics

[Select Menu](#)

5.3.2.13.4 from Solid

If your part is a solid model (also referred to as a poly-surface), you can highlight and select one or more surfaces. To select one surface just move the cursor over it and then left-click to select it. To select multiple surface, press and hold the <Ctrl> key when selecting.



Select Surface from Solid

Related Topics

[Select Menu]

5.3.3 Commands

The Modeling Aids Ribbon Bar contains the following Commands pane with associated functions.

Related Topics

Undo

Undo Multiple

Redo

Redo Multiple

Command Recall

Continuous Command Recall

Modeling Aids Ribbon Bar

5.3.3.1 Undo

Undoes the previous construction or edit command. Everything goes back to how it was before the last command was executed. Geometry objects that were previously selected (highlighted) will remain selected when Undo is used.

Related Topics

[Undo](#)

[Undo Multiple](#)

[Redo](#)

[Redo Multiple](#)

[Commands Pane, Modeling Aids Ribbon Bar](#)

5.3.3.2 Redo

Redoes a previously undone construction or edit command. Everything returns to the state after the command was executed.

Related Topics

[Undo](#)

[Undo Multiple](#)

[Redo Multiple](#)

[Commands Pane, Modeling Aids Ribbon Bar](#)

5.3.3.3 Undo Multiple

This command will Undo multiple commands at the same time. A list of recent commands is displayed. A command is selected, and all commands after that one including it are undone. Everything returns to the state before the selected command was executed.

The commands are listed in reverse order with the most recent at the top. The last command executed is number one, and the second to last command executed is number two. Only one command needs to be picked, and all commands higher (more recent or occurred after that one) will automatically be selected.

Dialog Box: Undo Multiple

Dialog Box: Undo Multiple

Related Topics

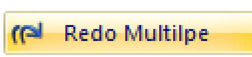
[Undo]()

[Redo]()

[Redo Multiple]()

[Commands Pane, Modeling Aids Ribbon Bar]()

5.3.3.4 Redo Multiple

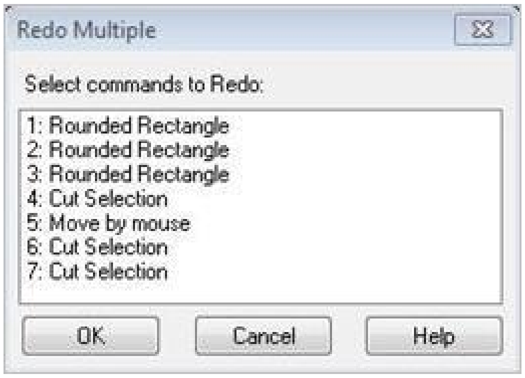


Redo Multiple commands that were previously undone. A list of undone commands is displayed. A command is selected, and all commands after that one including it are redone. Everything returns to the state after the selected command was executed.

The commands are listed in the order that they were originally executed. The first command executed is number one, and the second command executed is number two. Only one command needs to be picked, and all commands higher will automatically be selected.

Once commands have been undone, they will appear in the multiple redo list until another command is executed. Then the multiple redo list is emptied. Redo will have no effect.

Dialog Box: Redo Multiple



Dialog Box: Redo Multiple

Related Topics

[Undo]

[Undo Multiple]

[Redo]

[Commands Pane, Modeling Aids Ribbon Bar]

5.3.3.5 Command Recall


Redo the previous command.

Related Topics

[Commands Pane, Modeling Aids Ribbon Bar]

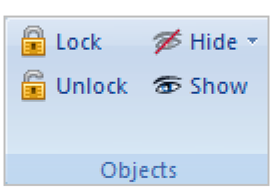
5.3.3.6 Continuous Command Recall


Automatically repeats the last command without having to go back & reselect the command from the toolbar or use the right mouse button click or enter option to repeat the previous command.

Related Topics

[Commands Pane, Modeling Aids Ribbon Bar]

5.3.4 Objects



The Modeling Aids Ribbon Bar contains the following Objects pane with associated functions.

Related Topics

[Lock Selection]
[Unlock All]
[Hide Selection]
[Hide Non-Selected]
[Show All]
[Modeling Aids Ribbon Bar]

5.3.4.1 Lock Selection

Locks all selected objects. Sets the state of selected objects so they can be seen but cannot be selected for editing.

Related Topics

Unlock

5.3.4.2 Unlock All

Unlocks all locked objects.

Related Topics

Lock

5.3.4.3 Hide Selection

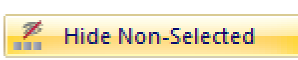
Hides or Conceals selected objects from view.

Related Topics

Show

Hide Non-Selected

5.3.4.4 Hide Non-Selected



Hides or Conceals non selected objects from view.

Related Topics

Hide

Show

5.3.4.5 Show All

Displays all hidden objects.

Related Topics

Hide

Hide Non-Selected

5.3.5 C-Plane

The Modeling Aids Ribbon Bar contains the following C-Plane pane with associated functions.

C-Plane Pane, Modeling Aids Ribbon Bar

Related Topics

Set C-Plane to View

- Set C-Plane to Top View
- Set C-Plane to Bottom View
- Set C-Plane to Front View
- Set C-Plane to Back View
- Set C-Plane to Left View
- Set C-Plane to Right View

Set C-Plane Origin

Set C-Plane Elevation

Rotate the C-Plane

Set C-Plane by 3 Points

Set C-Plane to 3D Face

Set C-Plane to Planar Curve

Set C-Plane by X Axis

[Set C-Plane to WCS](#)

[Save C-Plane](#)

[Load a C-Plane](#)

[Modeling Aids Ribbon Bar](#)

5.3.5.1 Set to View

This option immediately sets the orientation of the C-Plane to be parallel to the plane of the currently active view. The origin remains the same.

Command Prompts

C-Plane by View:: Done

Related Topics

C-Plane Pane, Modeling Aids Ribbon Bar

5.3.5.2 Set to Top View

Move the construction plane to the origin and in the z-plane.

Related Topics

C-Plane Pane, Modeling Aids Ribbon Bar

5.3.5.3 Set to Bottom View

Move the construction plane to the origin and in the z-plane.

Related Topics

C-Plane Pane, Modeling Aids Ribbon Bar

5.3.5.4 Set to Front View

Move the construction plane to the origin and in the y-plane.

Related Topics

C-Plane Pane, Modeling Aids Ribbon Bar

5.3.5.5 Set to Back View

Move the construction plane to the origin and in the y-plane.

Related Topics

C-Plane Pane, Modeling Aids Ribbon Bar

5.3.5.6 Set to Left View

Move the construction plane to the origin and in the x-plane.

Related Topics

C-Plane Pane, Modeling Aids Ribbon Bar

5.3.5.7 Set to Right View

Move the construction plane to the origin and in the x-plane.

Related Topics

C-Plane Pane, Modeling Aids Ribbon Bar

5.3.5.8 Set Origin

Moves the origin of the C-Plane to a selected point or entered coordinates. The current configuration of the construction plane is saved whenever the model is saved into the VCP (VisualCAD Part) file.

Command Prompts

C-Plane Origin:: Pick origin point or enter coordinates x, y and z

Related Topics

C-Plane Pane, Modeling Aids Ribbon Bar

5.3.5.9 Set Elevation

Moves the elevation of the C-Plane along its normal (perpendicular to the plane). Input is a single distance and may be positive or negative. A positive number moves the plane towards the positive end of the coordinate axes even if the viewing location is along the negative axis. All input values are treated as an incremental distance relative to the current location of the plane.

Command Prompts

C-Plane Elevation:: Enter C-Plane elevation

Related Topics

C-Plane Pane, Modeling Aids Ribbon Bar

5.3.5.10 Rotate

Rotates the C-Plane about one of the principle axes of the current C-Plane, X, Y, or Z. The center of rotation is the center of the plane and not necessarily the origin. Input is 'axis,degrees'.

Command Prompts

Rotate C-Plane:: Enter axis to rotate about and angle to rotate by (eg: X,45)

Related Topics

C-Plane Pane, Modeling Aids Ribbon Bar

5.3.5.11 Set by 3 Pts

Three points are selected (origin, x-axis and y-axis) or entered to re-orient the C-Plane. The y-axis point defines the C-Plane and the general direction of the positive y-axis.

Command Prompts

C-Plane by 3Pts:: Pick origin point or enter coordinates x, y and z

C-Plane by 3Pts:: Pick x axis point or enter coordinates x,y and z

C-Plane by 3Pts:: Pick y axis point or enter coordinates x,y and z

Notes

1. The true y-axis will lie in the plane and be perpendicular to the x-axis and will pass through the origin point.
2. The third point cannot be collinear with the other two points.

Related Topics

C-Plane Pane, Modeling Aids Ribbon Bar

5.3.5.12 C-Plane to 3D Face

The C-Plane will be oriented to a selected flat plane of a geometry object. The pick point will become the C-Plane origin.

Command Prompts

C-Plane to 3D Face:: Pick flat area:

Basic Procedure

	Screen Pick	Command Input
Step 1	Select the 3D face to align the C-Plane to.	None

Related Topics

C-Plane Pane, Modeling Aids Ribbon Bar

5.3.5.13 C-Plane to Planar Curve

The C-Plane will be oriented to a selected planar curve. The pick point will become the C-Plane origin.

Command Prompts

C-Plane to Planar Curve:: Pick a planar curve

Basic Procedure

	Screen Pick	Command Input
Step 1	Select the 3D planar curve to align the C-Plane to.	None

Related Topics

C-Plane Pane, Modeling Aids Ribbon Bar

5.3.5.14 Set by X Axis

Two points are entered to re-orient the x-axis of the C-Plane:

Command Prompts

C-Plane by X-Axis:: Pick origin point or enter coordinates x, y and z

C-Plane by X-Axis:: Pick x axis point or enter coordinates x,y and z

Basic Procedure

	Screen Pick	Command Input
Step 1	Pick the new origin point or enter the x,y,z coordinates.	Command 1,1,0.75
Final	Pick a point on the new x-axis. The new C-Plane is parallel to the prior orientation.	

Related Topics

C-Plane Pane, Modeling Aids Ribbon Bar

5.3.5.15 Set to WCS

Resets the C-Plane back to its default orientation relative to the WCS and the currently active viewport. The origin will be the WCS origin.

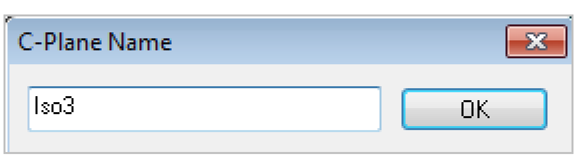
Related Topics

C-Plane Pane, Modeling Aids Ribbon Bar

5.3.5.16 Save ...

This option saves the C-Plane orientation from the currently active view. A name must be provided for later reference when loading.

C-Plane Name dialog box



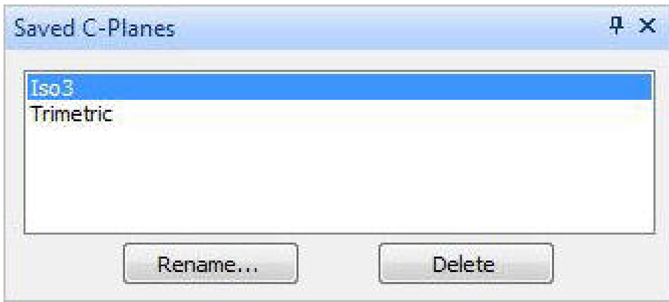
Related Topics

C-Plane Pane, Modeling Aids Ribbon Bar

5.3.5.17 Load ...

This option allows the retrieval of a saved C-Plane into the currently active view. A list of previously saved C-Planes is presented. These saved C-Planes may also be deleted in this option or renamed.

Saved C-Planes dialog box

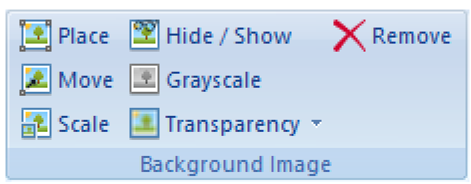


Related Topics

C-Plane Pane, Modeling Aids Ribbon Bar

5.3.6 Background Image

The Modeling Aids Ribbon Bar contains the following Background Image pane with associated functions.



Background Image Pane, Modeling Aids Ribbon Bar

Related Topics

Place Background Image

Move Background Image

Scale Background Image

Toggle Display of Background Image

Gray Scale Background Image

Background Image Transparency

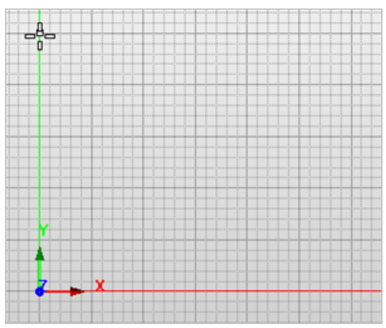
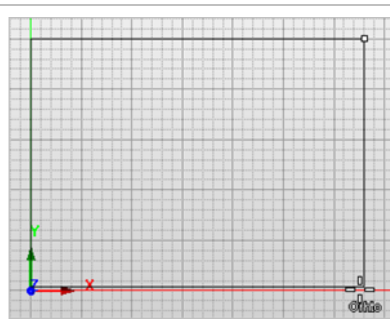
5.3.6.1 Place Image

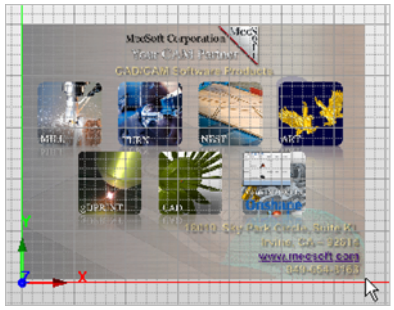
Use this command to place an image onto the background of the C-Plane.

Command Prompts

1. Select an image using the file browser.
2. Pick first point of enter x,y and z
3. Pick second point or enter coordinates x, y and z

Basic Procedure

	Screen Pick	Command Input
Step 1	none	Select image file.
Step 2	 Pick First Point	Pick first point of enter x,y and z.
Step 3	 Pick Second Point	Pick first point of enter x,y and z.

Final	 Background Image  Background Image Full Viewport	

Related Topics

[Background Image](#)

[Move Background Image](#)

[Scale Background Image](#)

[Toggle Background Image On/Off](#)

[Gray Scale Background Image](#)

[Background Image Transparency](#)

[Delete Background Image](#)

5.3.6.2 Move Image

Use this command to Move the existing background image from one point to another on the C-Plane.

Command Prompts

1. Pick first point or enter coordinates x,y and z
2. Pick first point or enter coordinates x,y and z

Basic Procedure

	Screen Pick	Command Input
Step 1	Pick a point to move from. You can use the Object Snaps available from the Status Toolbar.  Pick First Point	Pick first point or enter coordinates x,y and z
Step 2	Pick a point to move to. You can use the Object Snaps available from the Status Toolbar.  Pick Second Point	Pick first point or enter coordinates x,y and z

Related Topics

Background Image

Place Background Image

Scale Background Image

Toggle Background Image On/Off

Gray Scale Background Image

Background Image Transparency

Delete Background Image

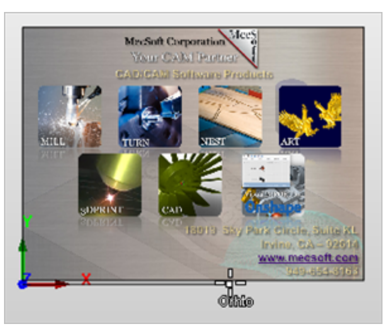
5.3.6.3 Scale Image

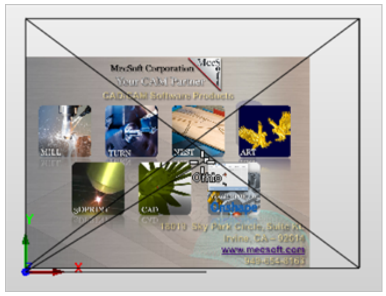
Scale Use this command to Scale the existing background image on the C-Plane.

Command Prompts

1. Enter scale center point
2. Enter first reference point or scale factor
3. Enter second reference point

Basic Procedure

	Screen Pick	Command Input
Step 1	Pick a point to serve as the scaling center point. You can use the Object Snaps available from the Status Toolbar.  Enter scale center point	Enter scale center point
Step 2	Pick a reference point or enter a scale factor. You can use the Object Snaps available from the Status Toolbar.  Enter first reference point or scale factor	Enter first reference point or scale factor

Step 3	Pick a second reference point. You can use the Object Snaps available from the Status Toolbar.  Enter second reference point	Enter second reference point

Related Topics

Background Image

Place Background Image

Move Background Image

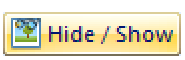
Toggle Background Image On/Off

Gray Scale Background Image

Background Image Transparency

Delete Background Image

5.3.6.4 Toggle Image On/Off



Use this command to toggle the display of the background image on and off.

Command Prompts

1. None

Basic Procedure

	Screen Pick	Command Input
Step 1	Pick the command icon to toggle the display of the background image off.	None

<table>
<tr><td></td><td>

Background Image On</td><td></td></tr>
<tr><td>Step 2</td><td>Pick the command icon to toggle the display of the background image on.


Background Image Off</td><td>None</td></tr>
</table>

Related Topics

[Background Image]

[Place Background Image]

[Move Background Image]

[Scale Background Image]

[Gray Scale Background Image]

[Background Image Transparency]

[Delete Background Image]

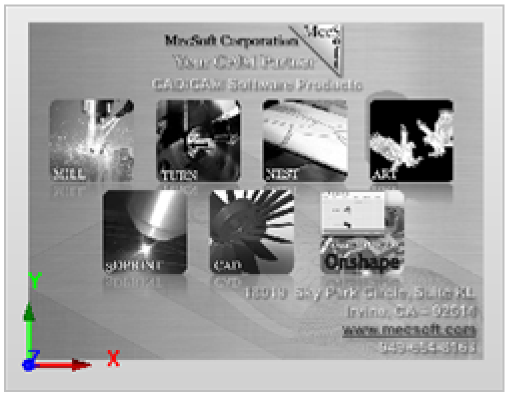
5.3.6.5 Gray Scale Image

Use this command to toggle the display of the background image to and from gray scale.

Command Prompts

1. None

Basic Procedure

	Screen Pick	Command Input
Step 1	Pick the command icon to toggle the display of the background to gray scale (On)  Gray Scale On	None
Step 2	Pick the command icon to toggle the display of the background from gray scale (Off)  Gray Scale Off	None

Related Topics

[Background Image]

[Place Background Image]

[Move Background Image]

[Scale Background Image](#)

[Toggle Background Image On/Off](#)

[Background Image Transparency](#)

[Delete Background Image](#)

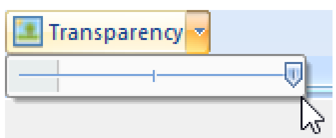
5.3.6.6 Image Transparency

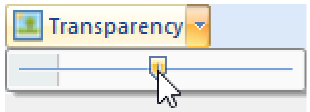
Use this command to adjust the transparency of the background image.

Command Prompts

1. None

Basic Procedure

	Screen Pick	Command Input
Step 1	Pick the dropdown menu indicator to the right of the command icon to display the transparency slider. Move the slider to the right for no transparency   Transparency Slider full right	None
Step 2	Move the slider to the middle for half transparency.	None

	  Transparency half	

Related Topics

[Background Image]

[Place Background Image]

[Move Background Image]

[Scale Background Image]

[Toggle Background Image On/Off]

[Gray Scale Background Image]

[Delete Background Image]

5.3.6.7 Delete Image

Use this command to delete the currently active background image.

Command Prompts

1. None

Basic Procedure

	Screen Pick	Command Input
Step 1	Pick the command icon to delete the background image.	None

Related Topics

[Background Image](#)

[Place Background Image](#)

[Move Background Image](#)

[Scale Background Image](#)

[Toggle Background Image On/Off](#)

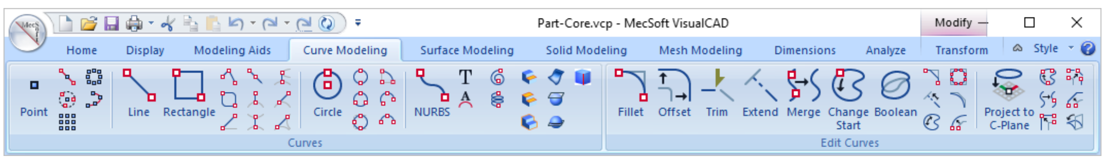
[Gray Scale Background Image](#)

[Background Image Transparency](#)

5.4 Curve Modeling

The Curve Modeling Ribbon Bar contains Curves and Edit Curves panes and associated functions.

The Curve Modeling Ribbon Bar



Curve Modeling Ribbon Bar

Related Topics

[Curves Pane, Curve Modeling Ribbon Bar](#)

[Edit Curves Pane, Curve Modeling Ribbon Bar](#)

[The Ribbon Bar](#)

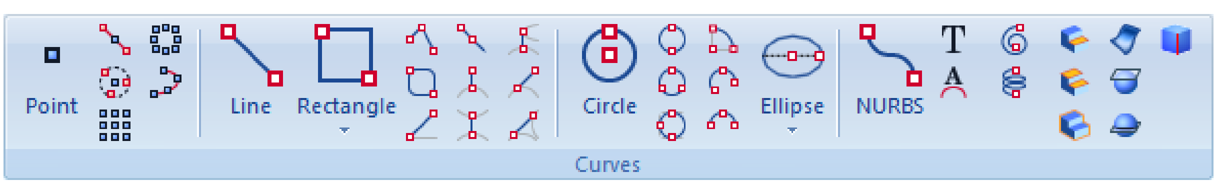
[Customize Quick Access Toolbar](#)

[Customize Dialog](#)

[VisualCAD Options](#)

5.4.1 Curves

The Curve Modeling Ribbon Bar contains the following Curves pane with associated functions.



Curve Modeling Ribbon Bar
Curves Pane

Related Topics

Point

Mid Point

Center Point

Point Grid

Polar Grid of Points

Points along a Curve

Line

Rectangle, 2 Points

Rectangle, Center

Rectangle, 3 Points

Rectangle, Vertical

Rectangle

Polyline

Rounded Rectangle

Line at an Angle

Mid-Point Line

Line Normal to a Curve

Line Normal to 2 Curves

Line Normal & Tangent

Line Tangent to a Curve

Line Tangent to 2 Curves

Circle at Center and a Point

Circle by 2 Dia. Points

Circle with 3 Dia Points

Circle Tangent to 3 Curves

Arc with Center, Start and Angle

3-Point Arc

Arc with Center, Start and On Point

Ellipse, Diameter

Ellipse, Foci

Nurbs Curve

5.4.1.1 Point

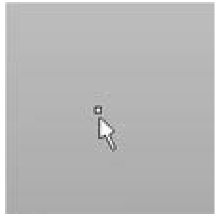
Create a point by picking on the screen or by entering X, Y, Z coordinates.

Command Prompts

Point:: Pick point or enter coordinates x,y and z

Basic Procedure

	Screen Pick	Command Input
Step 1	Create a point by picking a location on the screen. 	(Optional) Enter the X, Y, Z coordinates of the point.  Also: Other valid entries are...  X, Y coordinates only. Z is 0.000 **Note**: X, Y, Z coordinates are relative to the Construction Plane in the Active View.

Final		

Related Topics

[Curves Pane, Curve Modeling Ribbon Bar]

5.4.1.2 Point, Mid

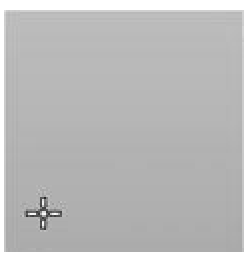
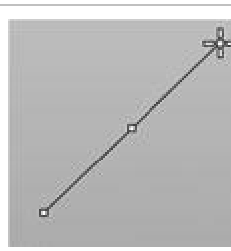
Create a point mid-way between two screen picks or by entering X, Y, Z coordinates for either point.

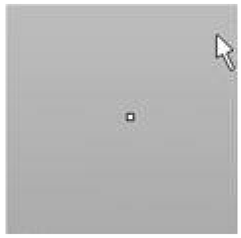
Command Prompts

Mid:: Pick first point or enter coordinates x,y and z

Mid:: Pick second point or enter coordinates x, y and z

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point. 
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point. 

Final		

Related Topics

[Curves Pane, Curve Modeling Ribbon Bar]()

5.4.1.3 Point, Center

 Create a point at the center of a circle made from three construction points.

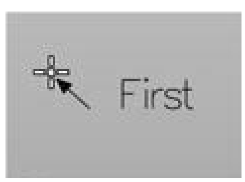
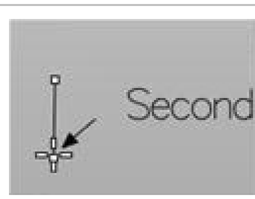
Command Prompts

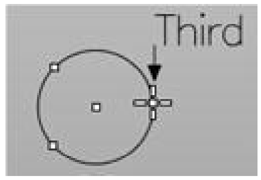
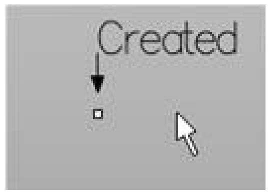
Center :: Pick first point or enter coordinates x,y and z

Center :: Pick second point or enter coordinates x, y and z

Center :: Pick third point or enter coordinates x,y and z

Basic Procedure

	Screen Pick	Command Input
Step 1	Pick three points to define a circle. 	(Optional) Enter X, Y, [Z] coordinates for any of the three points. 0,1,0
Step 2		(Optional) Enter X, Y, [Z] coordinates for any of the three points. 0,0,0

Step 3		(Optional) Enter X, Y, [Z] coordinates for any of the three points. 1,0,0
Final		

Notes

1. All construction points and circle are on the Construction Plane and are not kept.

Related Topics

[Curves Pane, Curve Modeling Ribbon Bar]()

5.4.1.4 Point, Grid

Create points in a rectangular grid array.

Command Prompts

Grid :: Enter number of points(circles) in the first and normal direction:: X=4, Y=4, [D=0.000000]

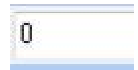
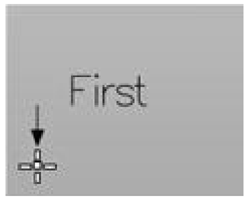
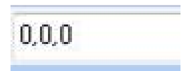
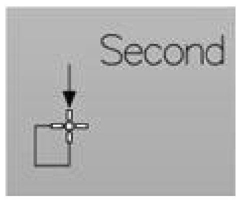
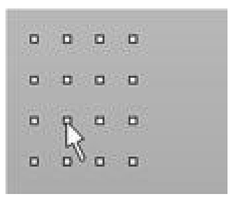
Grid :: Enter angle of first row about X axis:: [A=0.000000]

Grid :: Pick first corner point of spacing

Grid :: Pick second corner point of spacing or enter spacing values

Basic Procedure

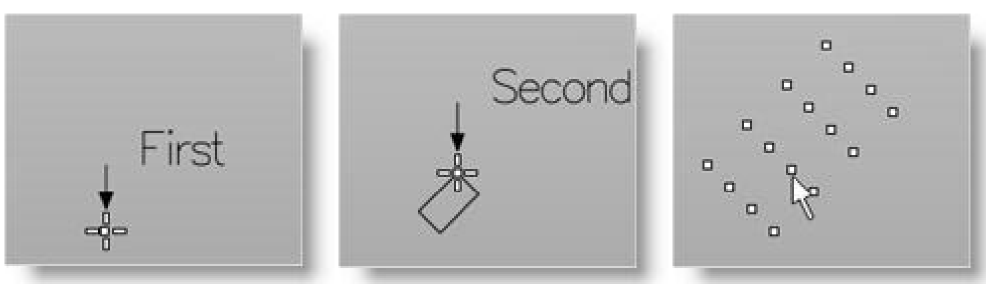
	Screen Pick	Command Input
Step 1		Enter the number of columns (along X) and rows (along Y). The default values are shown in the Prompt. 4,4

Step 2		Enter the angle that the array makes with the X axis. The Default value is shown in the Prompt. 
Step 3		(Optional) Enter X, Y, [Z] coordinates for the first point. 
Step 4		(Optional) Enter X, Y, [Z] coordinates for the second point. 
Final		

Notes

1. The incremental distance between the two points (in X and Y) determines the incremental distance between columns and rows respectively.
2. Since the construction plane changes between the different views, the directions “First” and “Normal” will also change.

Example with a 45-degree rotation



Related Topics

[Curves Pane, Curve Modeling Ribbon Bar]

5.4.1.5 Point, Polar Grid

Create points in a circular grid array.

Command Prompts

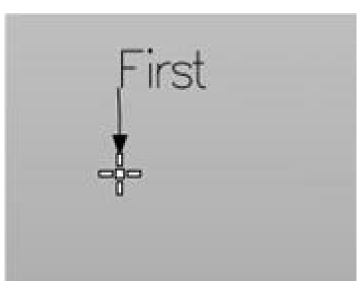
Polar Grid :: Enter number of points along circumference:: N=4, [D=0.000000]

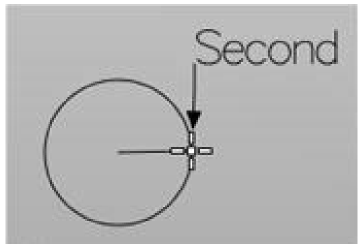
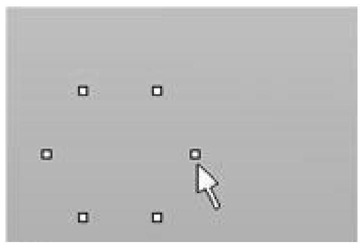
Polar Grid :: Enter angle of first row about X axis:: [A=0.000000]

Polar Grid :: Pick center point or enter coordinates x,y and z

Polar Grid :: Enter radius or pick radius point:: [R=4.000000]

Basic Procedure

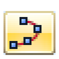
	Screen Pick	Command Input
Step 1		Enter the number of points in the circular array. The default value is shown in the Prompt. 6
Step 2		Enter the angle that a radial line through the first circumference point makes with the X axis. The Default value is shown in the Prompt. 0
Step 3		(Optional) Enter X, Y, [Z] coordinates for the first point. 0,0,0

Step 4		(Optional) Enter X, Y, [Z] coordinates for the second point (determines the radius). 1,1,0 or... (Optional) Enter the radius value directly. 9.50
Final		

Related Topics

[Curves Pane, Curve Modeling Ribbon Bar]

5.4.1.6 Point, On Curve

 Create points equidistant along a curve.

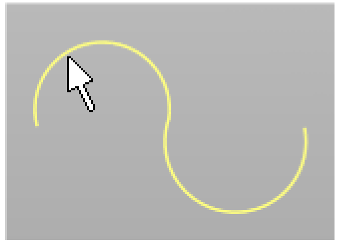
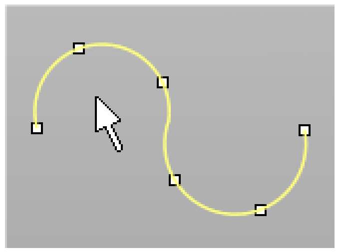
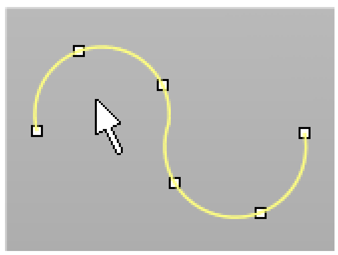
Command Prompts

On Curve:: Pick the curve(s) to draw the points on. Hit Right Mouse Button/Enter when done.

On Curve :: Enter number of points along the curve(s):: N=4

Basic Procedure

	Screen Pick	Command Input

Step 1		
Step 2		Enter the number of points to be created. The default value is shown in the Prompt. 
Final		

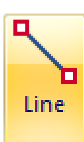
Notes

1. If multiple curves are selected, a full set of points will be created on each curve.
2. The curves can be 3D spatial curves.

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.7 Line



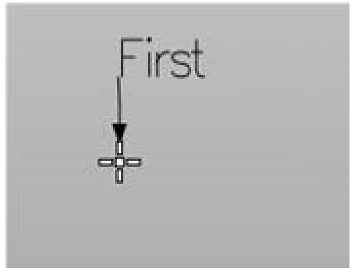
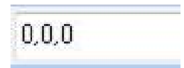
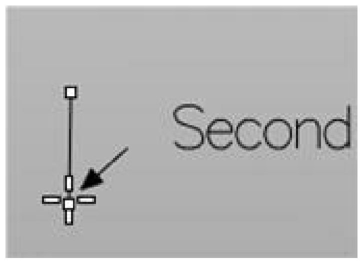
Create a line between two points by picking on the screen or by entering X, Y, Z coordinates.

Command Prompts

Line :: Pick first point or enter coordinates x,y and z

Line :: Pick second point or enter coordinates x, y and z

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point. 
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point. 

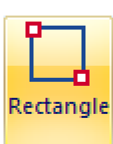
Notes

1. Coordinate entry can be in absolute or relative coordinates. An example of absolute coordinate is shown above.
2. Example of coordinate entry Input for relative coordinates:
   a. Pick first point or enter coordinates x,y and Z: 2,0
   b. Pick second point or enter coordinates x,y and Z: R5,0
3. For the second point, the coordinate input can also be entered as @5,0. This creates a horizontal line that is 5 units in length from 2,0 to 7,0 in X Y coordinates.
4. To create a vertical line, the coordinate entry for second point can be entered as @0,5. This creates a vertical line that is 5 units in length from 2,0 to 2,5 in X Y coordinates.
5. To create a line at an angle the coordinate entry for the second point can be entered as @5<45. This creates a line at 45 degrees which is 5 units in length.

Related Topics

[Curves Pane, Curve Modeling Ribbon Bar]

5.4.1.8 Rectangle, 2 Points



Create a rectangle between two points by picking on the screen or by entering X, Y, Z coordinates.

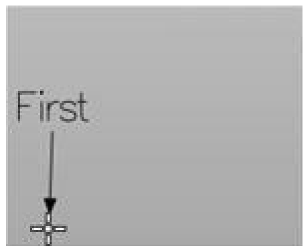
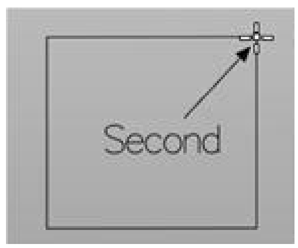
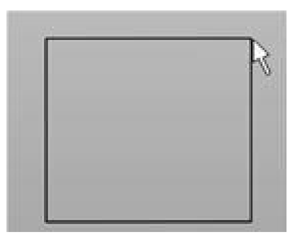
Command Prompts

Rectangle:: Pick first corner point or enter coordinates x,y and z

Rectangle:: Pick second corner point or enter coordinates x,y and z

Rectangle:: Done

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point. 
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point (opposite corner). 
Final		

Notes

1. The rectangle is aligned with the axes of the Construction Plane.
2. The final rectangle will lie in a plane parallel to the Construction Plane and with one corner at the first point selected.
3. If the second point selected is not in that same plane, it will be projected onto the destination plane of the final rectangle.
4. Coordinate entry can be in absolute coordinates or in relative coordinates. An example of absolute coordinate is shown above.
5. Example of coordinate entry Input for relative coordinates:

a. Pick first corner point or enter coordinates x,y and Z: 2,0

b. Pick second corner point or enter coordinates x,y and Z: R6,4

6. For the second point, the coordinate input can also be entered as @6,4.
This creates a rectangle line that is 6 units in length and 4 units in width from 2,0 to 8,4 in X Y coordinates.

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.9 Rectangle, Center

Create a rectangle about a center point. Picking points or entering coordinates.

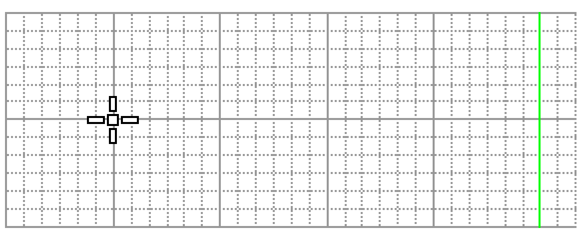
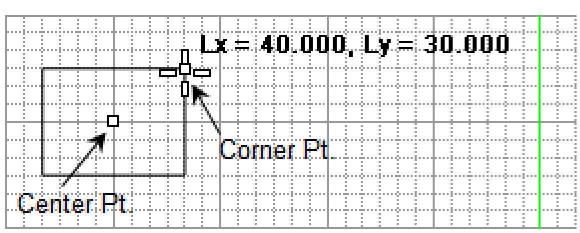
Command Prompts

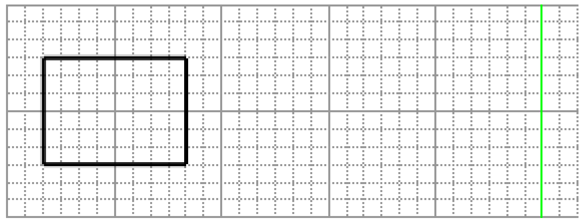
Rectangle:: Pick center point or enter coordinates x,y and z

Rectangle:: Pick other corner point or enter coordinates x,y and z

Rectangle:: Done

Basic Procedure

	Screen Pick
Step 1	Rectangle:: Pick center point or enter coordinates x,y and z 
Step 2	Rectangle:: Pick other corner point or enter coordinates x,y and z 
Final	Rectangle:: Done



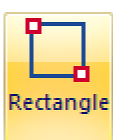
Notes

1. The rectangle is aligned with the axes of the Construction Plane.
2. The final rectangle will lie in a plane parallel to the Construction Plane and with one corner at the first point selected.
3. If the second point selected is not in that same plane, it will be projected onto the destination plane of the final rectangle.
4. Coordinate entry can be in absolute coordinates or in relative coordinates.
5. Example of coordinate entry Input for relative coordinates:
   a. Pick first corner point or enter coordinates x,y and Z: 2,0
   b. Pick second corner point or enter coordinates x,y and Z: R6,4
6. Coordinate input can also be entered as @6,4.
   This is 6 units in length and 4 units in width from 2,0 to 8,4 in X Y coordinates.

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.10 Rectangle, 2 Points 2



Create a rectangle between two points by picking on the screen or by entering X, Y, Z coordinates.

Command Prompts

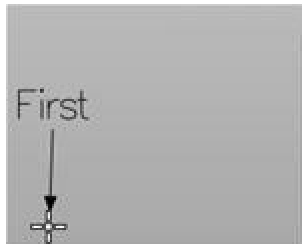
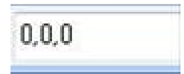
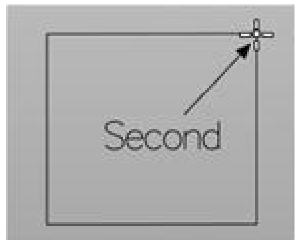
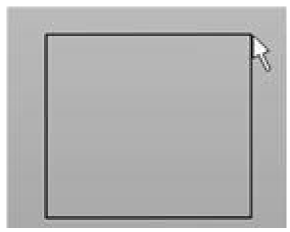
Rectangle:: Pick first corner point or enter coordinates x,y and z

Rectangle:: Pick second corner point or enter coordinates x,y and z

Rectangle:: Done

Basic Procedure

	Screen Pick	Command Input

Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point. 
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point (opposite corner). 
Final		

Notes

1. The rectangle is aligned with the axes of the Construction Plane.
2. The final rectangle will lie in a plane parallel to the Construction Plane and with one corner at the first point selected.
3. If the second point selected is not in that same plane, it will be projected onto the destination plane of the final rectangle.
4. Coordinate entry can be in absolute coordinates or in relative coordinates. An example of absolute coordinate is shown above.
5. Example of coordinate entry Input for relative coordinates:
    a. Pick first corner point or enter coordinates x,y and Z: 2,0
    b. Pick second corner point or enter coordinates x,y and Z: R6,4
6. For the second point, the coordinate input can also be entered as @6,4.
   This creates a rectangle line that is 6 units in length and 4 units in width from 2,0 to 8,4 in X Y coordinates.

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.11 Rectangle, 3 Points

Create a rectangle between two base points and a height point. Picking points or entering coordinates.

Command Prompts

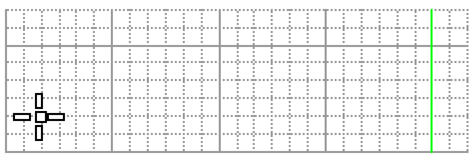
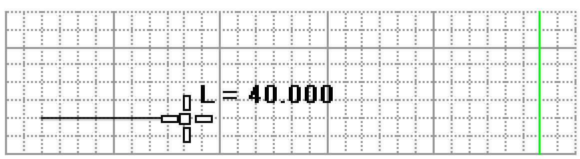
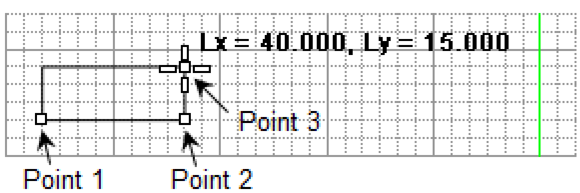
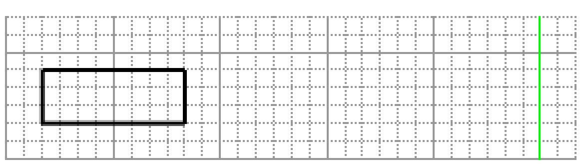
Rectangle:: Pick edge start point or enter coordinates x,y and z

Rectangle:: Pick edge end point or enter coordinates x,y and z

Rectangle:: Pick point or enter coordinates x,y and z or enter width

Rectangle:: Done

Basic Procedure

	Screen Pick
Step 1	Rectangle:: Pick edge start point or enter coordinates x,y and z 
Step 2	Rectangle:: Pick edge end point or enter coordinates x,y and z 
Step 3	Rectangle:: Pick point or enter coordinates x,y and z or enter width 
Final	Rectangle:: Done 

Notes

1. The rectangle is aligned with the axes of the Construction Plane.
2. The final rectangle will lie in a plane parallel to the Construction Plane and with one corner at the first point selected.
3. If the second point selected is not in that same plane, it will be projected onto the destination plane of the final rectangle.
4. Coordinate entry can be in absolute coordinates or in relative coordinates. An example of absolute coordinate is shown above.
5. Example of coordinate entry Input for relative coordinates:
    a. Pick first corner point or enter coordinates x,y and Z: 2,0
    b. Pick second corner point or enter coordinates x,y and Z: R6,4
6. For the second point, the coordinate input can also be entered as @6,4.
This creates a rectangle line that is 6 units in length and 4 units in width from 2,0 to 8,4 in X Y coordinates.

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.12 Rectangle, Vertical

Create a rectangle between two base points and a height point. You can pick points or enter coordinates (See notes below).

Command Prompts

Rectangle:: Pick edge start point or enter coordinates x,y and z

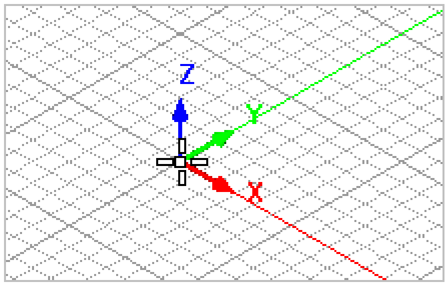
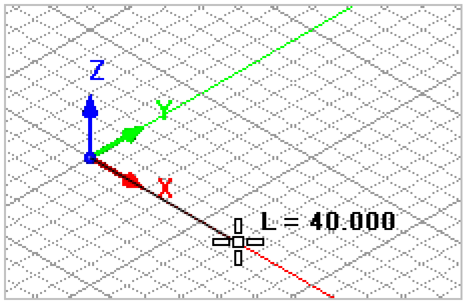
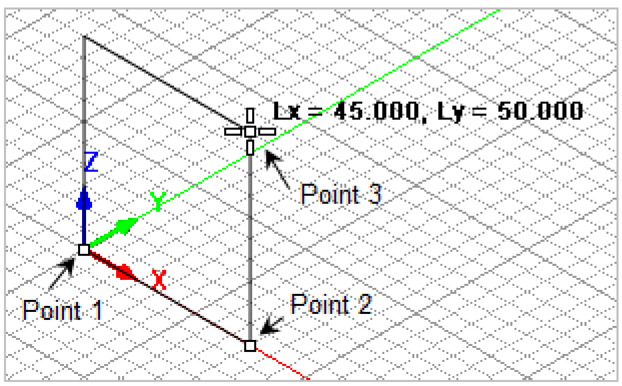
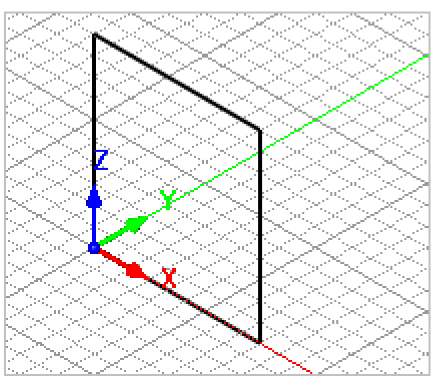
Rectangle:: Pick edge end point or enter coordinates x,y and z

Rectangle:: Pick point or enter coordinates x,y and z or enter width

Rectangle:: Done

Basic Procedure

	Screen Pick
Step 1	Rectangle:: Pick edge start point or enter coordinates x,y and z

<table>
<tr><td></td><td>
</td></tr>
<tr><td>Step 2</td><td>Rectangle:: Pick edge end point or enter coordinates x,y and z

</td></tr>
<tr><td>Step 3</td><td>Rectangle:: Pick point or enter coordinates x,y and z or enter width

</td></tr>
<tr><td>Final</td><td>Rectangle:: Done

</td></tr>
</table>

Notes

1. The rectangle is aligned with the axes of the Construction Plane.
2. The final rectangle will lie in a plane parallel to the Construction Plane and with one corner at the first point selected.
3. If the second point selected is not in that same plane, it will be projected onto the destination plane of the final rectangle.
4. Coordinate entry can be in absolute coordinates or in relative coordinates. An example of absolute coordinate is shown above.
5. Example of coordinate entry Input for relative coordinates:
    a. Pick first corner point or enter coordinates x,y and Z: 2,0
    b. Pick second corner point or enter coordinates x,y and Z: R6,4
6. For the second point, the coordinate input can also be entered as @6,4.
   This creates a rectangle line that is 6 units in length and 4 units in width from 2,0 to 8,4 in X Y coordinates.

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.13 Polyline

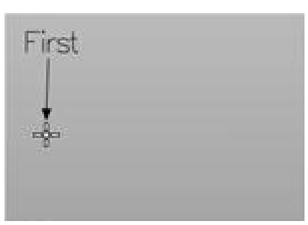
Create a polyline between consecutive points by picking on the screen or by entering X, Y, Z coordinates. Polylines are continuous line segments connected end-to-end.

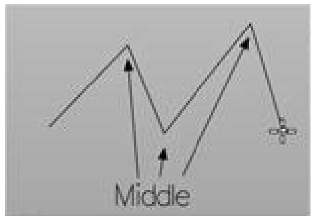
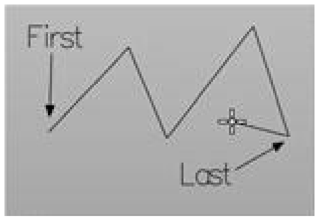
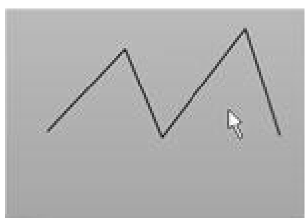
Command Prompts

Polyline :: Pick point or enter coordinates x,y and z

The Prompt (and point selection) is repeated until the right mouse button is clicked.

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point. Input: 0,0,0

Step 2		(Optional) Enter X, Y, [Z] coordinates for any of the intermediate points.
Step 3	 After the last point is selected, click the right mouse button to end the selection.	
Final		

Notes

1. Coordinate entry can be in absolute coordinates or in relative coordinates. An example of absolute coordinate is shown above.
2. Example of coordinate entry Input for relative coordinates:
    a. Pick point or enter coordinates x,y and Z: 2,0
    b. Pick point or enter coordinates x,y and Z: R5,0
3. For the second point, the coordinate input can also be entered as @5,0. This creates a horizontal line that is 5 units in length from 2,0 to 7,0 in X Y coordinates.
4. To create a vertical line, the coordinate entry for next point can be entered as @0,5. This creates a vertical line that is 5 units in length from 2,0 to 2,5 in X Y coordinates.
5. To create a line at an angle the coordinate entry for the next point can be entered as @5<45. This creates a line at 45 degrees which is 5 units in length.

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.14 Rounded Rectangle

Create a rounded rectangle between two points by picking on the screen or by entering X, Y, Z coordinates.

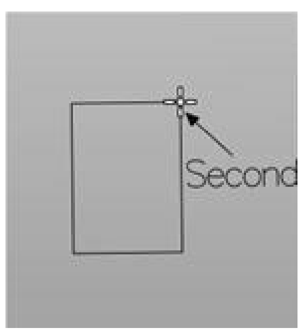
Command Prompts

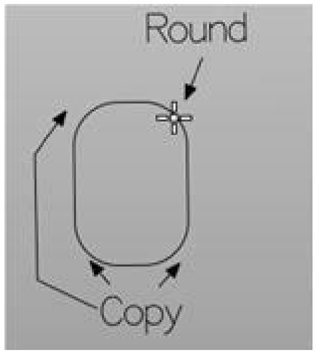
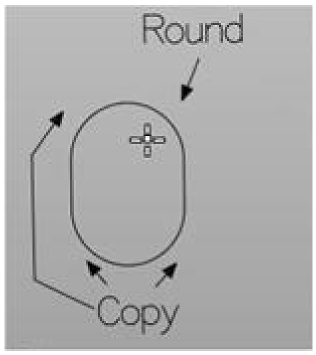
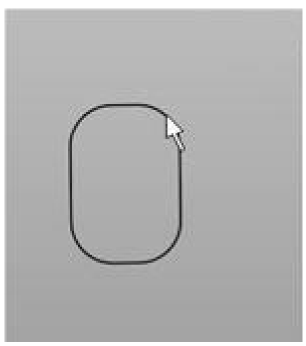
Rounded Rect. :: Pick first corner point or enter coordinates x,y and z

Rounded Rect. :: Pick second corner point or enter coordinates x,y and z

Rounded Rect. :: Enter radius or point for rounded corner to pass through::[R=0.250000]

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point. 0,0,0
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point (opposite corner). 1,1,0

Step 3	 	Enter the radius of the rounded corners.  or... (Optional) Enter X, Y, [Z] coordinates that the arc of the rounded corner should pass through.  If the point specified is too far inside the rectangle, the maximum radius will be maintained.
Final		

Notes

1. The rules for alignment and positioning are the same as for Rectangle.
2. All four rounded corners will have the same radius.
3. A radius value of 0.00 will produce a standard rectangle with sharp corners.
4. If the entered radius equals or exceeds half the width of the rectangle, then a full-radius rectangle will result.

Example of coordinate entry Input for relative coordinates.

1. Pick first corner point or enter coordinates x,y and Z: 2,0
2. Pick second corner point or enter coordinates x,y and Z: R6,4
   For the second point, the coordinate input can also be entered as @6,4

This creates a rounded rectangle line that is 6 units in length and 4 units in width from 2,0 to 8,4 in X Y coordinates.

3. Enter radius: 0.25

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.15 Line at Angle

Create a line at an angle to a base line, and either through a point or you can input an angle and a line length.

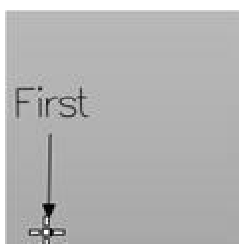
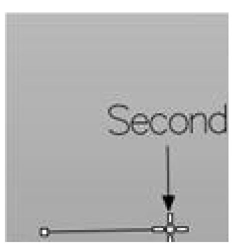
Command Prompts

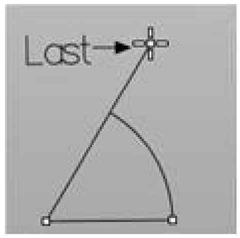
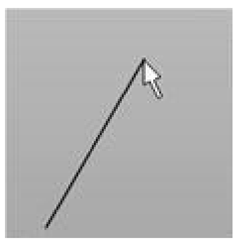
Line at Angle :: Select first point of Base Line

Line at Angle :: Select second point of Base Line

Line at Angle :: Enter point on line or specify an angle::[A=0.000000]

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point. 0,0,0
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point. 1,0,0

Step 3		(Optional) Enter X, Y, [Z] coordinates for the third point. 1,1,0 or... (Optional) Enter the angle of the line from this base line. 45 and... (optional) Enter the length of the line to be created. 4
Final		

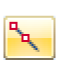
Notes

1. The first point selected is the pivot point about which the angle is measured.
2. If the second point is defined by picking on the screen or by entering coordinates, then the distance between those two points becomes the length of the created line, unless the length is entered through Input.

Related Topics

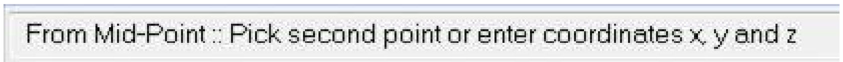
Curves Pane, Curve Modeling Ribbon Bar

5.4.1.16 Line, from Mid-Point

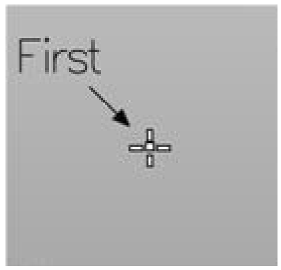
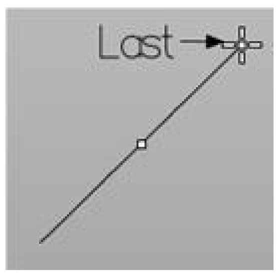
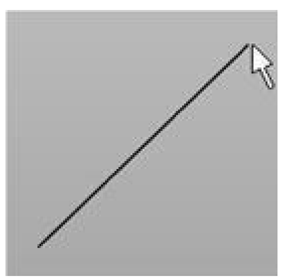
 Create a line by specifying its mid-point and an end-point.

Command Prompts

From Mid-Point :: Pick first point or enter coordinates x,y and z



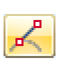
Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point (mid-point). 
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point (end point). 
Final		

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.17 Line, Normal to



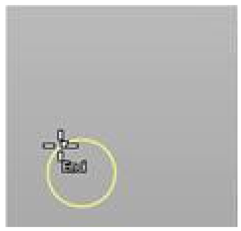
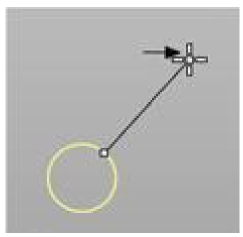
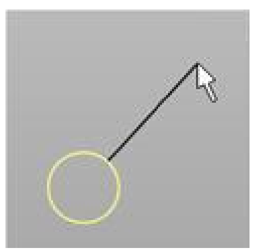
Create a line that is normal (perpendicular) to a curve. Line segments, polylines, and rectangles can also be selected.

Command Prompts

Tangent to:: Pick the curve to draw the tangent to

Tangent to:: Pick end point of Tangent line. You can use "S" key to change tangent line (BLACK line will be created)

Basic Procedure

	Screen Pick	Command Input
Step 1		
Step 2		(Optional) Enter X, Y, [Z] coordinates for the far end point. 
Final		

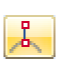
Notes

1. If the normal line cannot be calculated, no line will appear on the screen.
2. In the case of multiple solutions, the nearest normal point will be used.

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.18 Line, Normal to 2

 Create a line that is normal (perpendicular) to two curves. Line segments, polylines, and rectangles can also be selected.

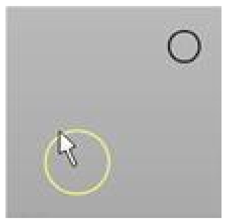
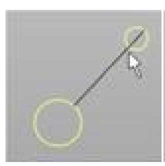
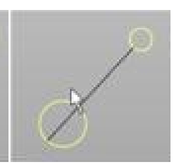
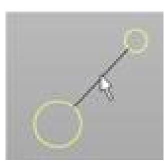
Command Prompts

Create Line Normal to 2 :: Pick first curve to draw the normal to

Create Line Normal to 2 :: Pick second curve to draw the normal to

Create Line Normal to 2 :: Pick the candidate solution

Basic Procedure

	Screen Pick	Command Input
Step 1		none
Step 2		
Step 3	   In step 3 above where there are multiple solutions, move the cursor along the line and left click when the desired solution appears.	

Notes

1. If the normal line cannot be calculated, no line will appear on the screen.

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.19 Line, Normal & Tangent

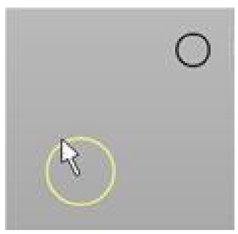
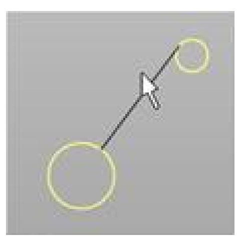
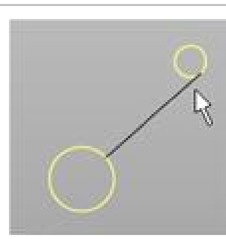
Create a line that is normal (perpendicular) to the first curve and tangent to the second curve. Line segments, polylines, and rectangles can also be selected.

Command Prompts

Normal & Tangent:: Pick the curve to draw the normal to

Normal & Tangent:: Pick curve to draw the tangent to

Basic Procedure

	Screen Pick	Command Input
Step 1		none
Step 2		
Step 3		
Final		

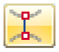
Notes

1. If the line cannot be calculated, a line will not appear on the screen.

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.20 Line, Tangent to



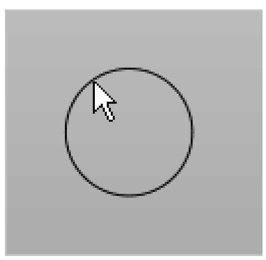
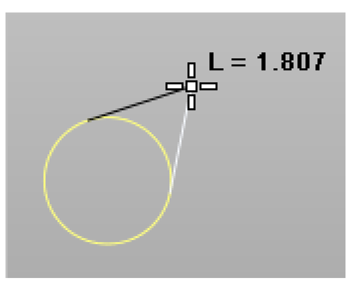
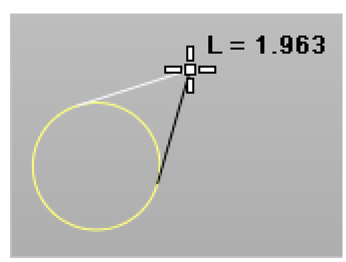
Create a line that is tangent to a curve. Line segments, polylines, and rectangles can also be selected.

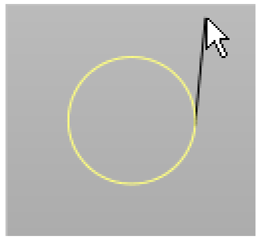
Command Prompts

Tangent to:: Pick the curve to draw the tangent to

Tangent to:: Pick end point of Tangent line. You can use "S" key to change tangent line (BLACK line will be created)

Basic Procedure

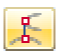
	Screen Pick	Command Input
Step 1		
Step 2		(Optional) Enter X, Y, [Z] coordinates for the non-tangent end point.  (Optional) Pick the S key to switch the tangent direction of the line: 

Final		

Related Topics

[Curves Pane, Curve Modeling Ribbon Bar]

5.4.1.21 Line, Tangent to 2

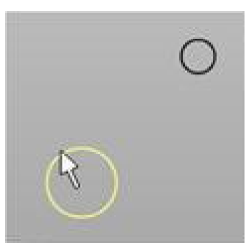
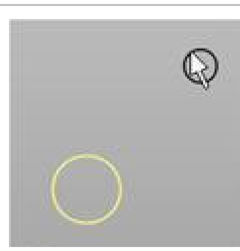
 Create a line that is tangent to a two curves. Line segments, polylines, and rectangles can also be selected.

Command Prompts

Tangent to 2:: Pick first curve to draw the tangent to

Tangent to 2:: Pick second curve to draw the tangent to

Basic Procedure

	Screen Pick	Command Input
Step 1		none
Step 2		

Final		

Notes

1. If multiple curves are pre-selected at the time that this function is used, only one of the pre-selected curves is retained as the first, picked curve.
2. If no solution is possible, then no line will appear on the screen.
3. Moving the cursor around will reveal the other solutions of tangency.

Related Topics

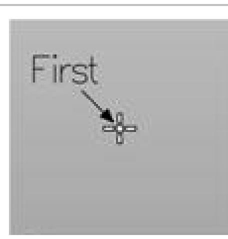
[Curves Pane, Curve Modeling Ribbon Bar]

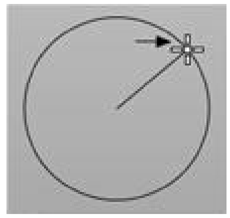
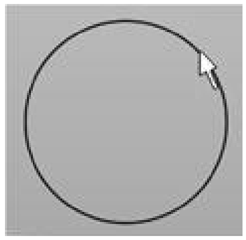
5.4.1.22 Circle, Center, On Pt

Create a circle by picking a center point and a point on the circle.

Command Prompts

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the center point. 

Step 2		(Optional) Enter X, Y, [Z] coordinates for the point (diameter) on the circle. 
Final		

Notes

1. The circle will be created in a plane that passes through the specified center point and that is parallel to the Construction Plane of the currently selected view.
2. If the circumference point is not on the plane of the circle, it will be projected normal onto that plane.

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.23 Circle, Start, Diameter Pt

Create a circle by picking two opposite points on the circumference that determine the diameter of the circle.

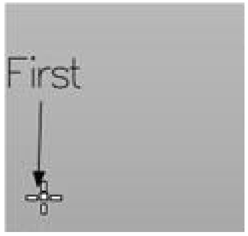
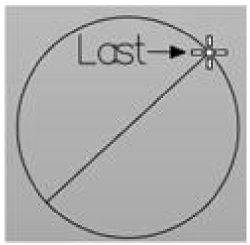
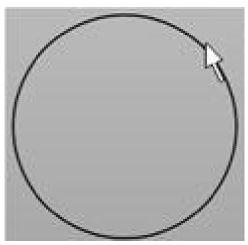
Command Prompts

Start, Diameter Pt :: Pick diameter start point or enter coordinates x,y and z

Start, Diameter Pt :: Pick diameter end point or enter coordinates x,y and z

Basic Procedure

	Screen Pick	Command Input

Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point. 
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point. 
Final		

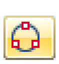
Notes

1. The circle will be created in a plane that passes through the first circumference point and that is parallel to the Construction Plane of the currently selected view.
2. If the second circumference point is not on the plane of the circle, it will be projected normal onto that plane.

Related Topics

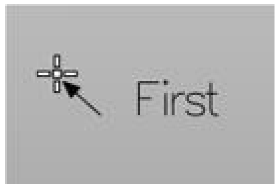
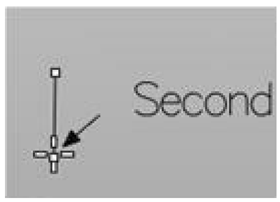
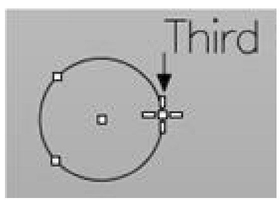
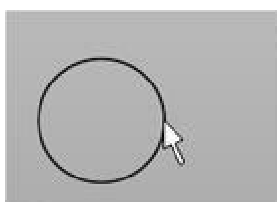
Curves Pane, Curve Modeling Ribbon Bar

5.4.1.24 Circle, 3 Pts

 Create a circle by picking three points on the circumference of the circle.

Command Prompts

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point. 
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point. 
Step 3		(Optional) Enter X, Y, [Z] coordinates for the third point. 
Final		

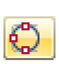
Notes

1. The circle will be created in a plane that passes through all three points. These points may not be collinear.
2. The start of the circle will be at the first circumference point specified.

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.25 Circle, Tangent to 3

 Create a circle that will be mutually tangent to three curves or locations. The geometry selected may be points, lines, or curves.

Command Prompts

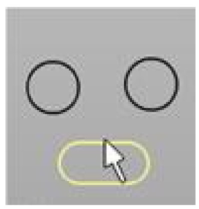
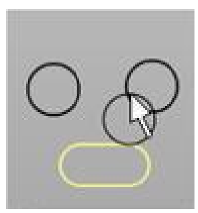
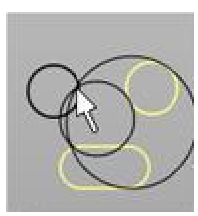
Tangent to 3 :: Pick the first point, line or circle entity

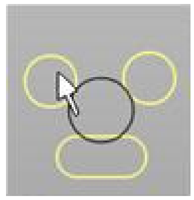
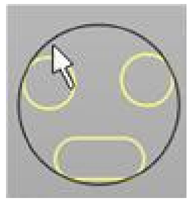
Tangent to 3 :: Pick the second point, line or arc entity or hit enter for mouse point

Tangent to 3 :: Pick the third point, line or arc entity or hit enter for mouse point

Tangent to 3 :: Pick the candidate solution

Basic Procedure

	Screen Pick	Command Input
Step 1		none
Step 2		
Step 3		

Step 4	 or 	

Notes

1. If line segments are used, they are extended to lines for the purpose of computing the tangency.

Related Topics

[Curves Pane, Curve Modeling Ribbon Bar]

5.4.1.26 Arc, Center, Start, Angle Pts

Create an arc by giving a center point, a starting point which determines the radius, and an end point, which determines the angle of the arc.

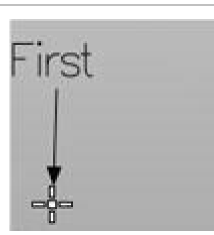
Command Prompts

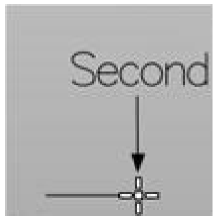
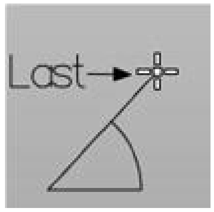
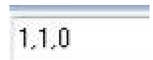
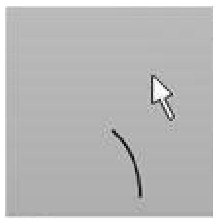
Center,Start,Angle Pts :: Pick center point or enter coordinates x,y and z

Center,Start,Angle Pts :: Pick arc start point or enter coordinates x,y and z

Center,Start,Angle Pts :: Pick arc end point or enter coordinates x,y and z::[A=45.000000]

Basic Procedure

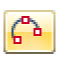
	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point (center of circle). 0,0,0

Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point (determines radius and arc start). 
Step 3		(Optional) Enter X, Y, [Z] coordinates for the third point (determines angle and arc end). 
Final		

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.27 Arc, 3 Pts



Create an arc by selecting three points on the circumference: a start point, an intermediate point, and an end point (in that order of selection).

Command Prompts

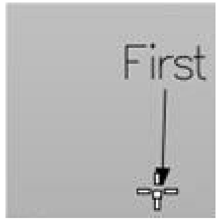
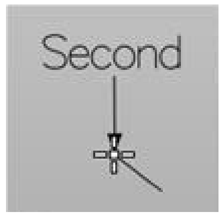
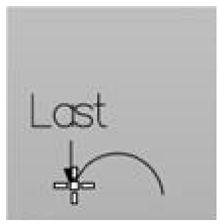
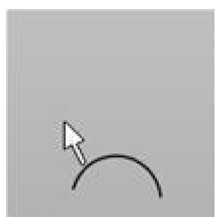
3 Pts :: Pick first point or enter coordinates x,y and z

3 Pts :: Pick second point or enter coordinates x, y and z

3 Pts :: Pick second point or enter coordinates x, y and z

Basic Procedure

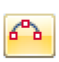
	Screen Pick	Command Input

Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point (start of arc). 1,0,0
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point (intermediate point on arc). 1,1,0
Step 3		(Optional) Enter X, Y, [Z] coordinates for the third point (end of arc). 0,0,0
Final		

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.28 Arc, Start, End, On

 Create an arc by selecting three points on the circumference: a start point, an end point, and then an intermediate point (in that order of selection).

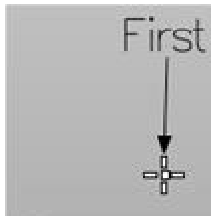
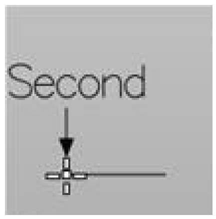
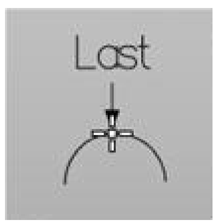
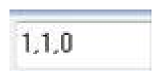
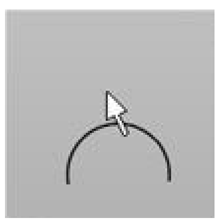
Command Prompts

Start, End, On :: Pick first point or enter coordinates x,y and z

Start, End, On :: Pick second point or enter coordinates x, y and z

Start, End, On :: Pick third point or enter coordinates x,y and z

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point (start of arc). 
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point (end of arc). 
Step 3		(Optional) Enter X, Y, [Z] coordinates for the third point (intermediate point on arc). 
Final		

Related Topics

[Curves Pane, Curve Modeling Ribbon Bar]

5.4.1.29 Ellipse, Center, Diameter, End Pt.

Create an Ellipse by selecting three points. The first two points define the major axis while the third point defines the minor axis.

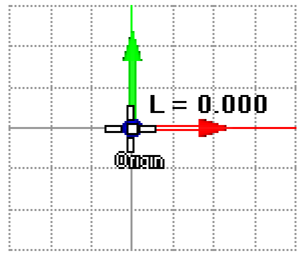
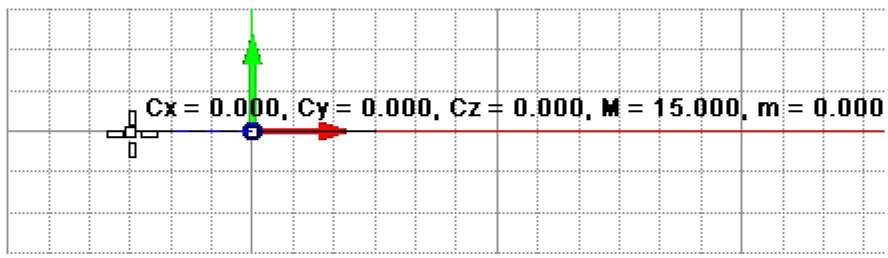
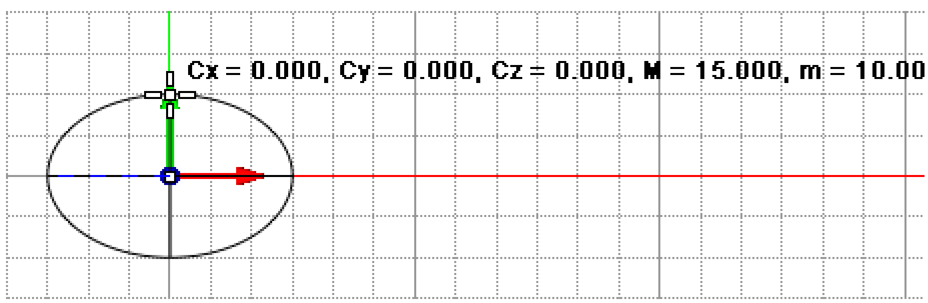
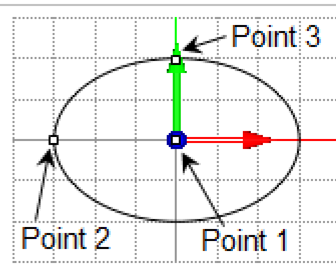
Command Prompts

Ellipse:: Pick major axis start point or enter coordinates x,y and z

Ellipse:: Pick major axis end point or enter coordinates x,y and z

Ellipse:: Pick minor axis end point or enter coordinates x,y and z or Enter width value

Basic Procedure

	Screen Pick / Command Input
Step 1	 Ellipse:: Pick major axis start point or enter coordinates x,y and z
Step 2	 Ellipse:: Pick major axis end point or enter coordinates x,y and z
Step 3	 Ellipse:: Pick minor axis end point or enter coordinates x,y and z or Enter w value.
Final	

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.30 Ellipse, Diameter, End Pt.

Create an Ellipse by selecting two diameter points and a third point to define the axis end point.

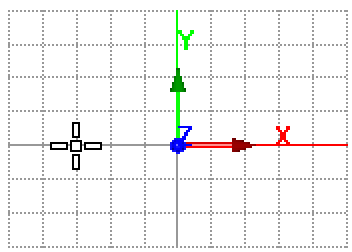
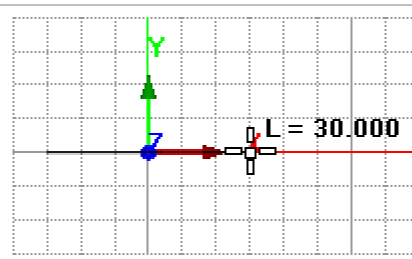
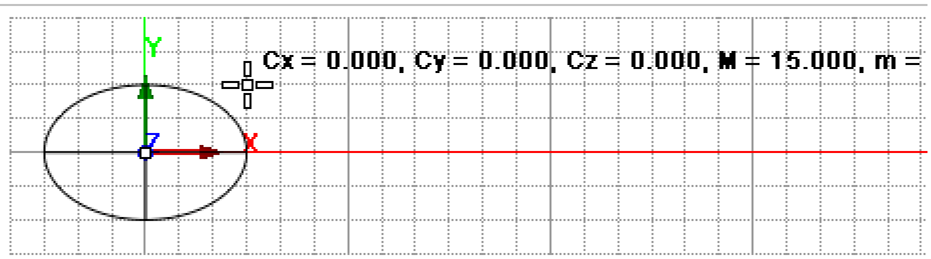
Command Prompts

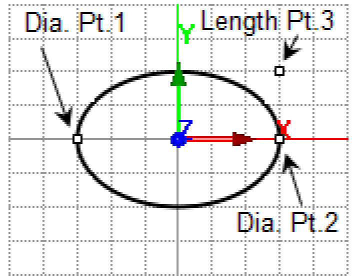
Ellipse:: Pick major axis start point or enter coordinates x,y and z

Ellipse:: Pick major axis end point or enter coordinates x,y and z

Ellipse:: Pick minor axis end point or enter coordinates x,y and z or Enter width value

Basic Procedure

	Screen Pick / Command Input
Step 1	 Ellipse:: Pick major axis start point or enter coordinates x,y and z
Step 2	 Ellipse:: Pick major axis end point or enter coordinates x,y and z
Step 3	 Ellipse:: Pick minor axis end point or enter coordinates x,y and z or Enter w value.

Final	

Related Topics

[Curves Pane, Curve Modeling Ribbon Bar]()

5.4.1.31 Ellipse, Foci

Create an Ellipse by defining a focal point about which the ellipse is positioned. The first two points define the ellipse major axis and are the focal points about which the ellipse is located. The minor axis of the ellipse passes through a third point defining the minor axis. Refer to the steps below.

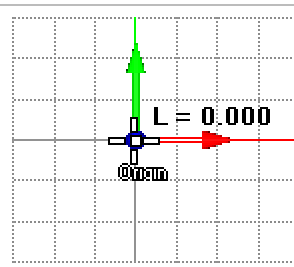
Command Prompts

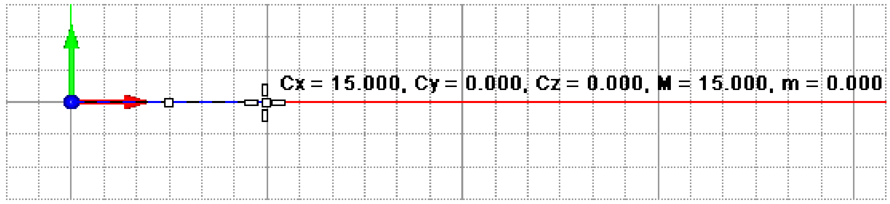
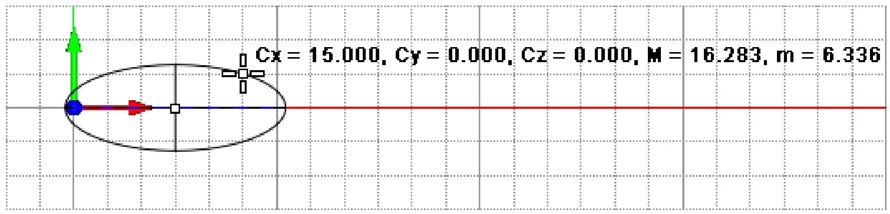
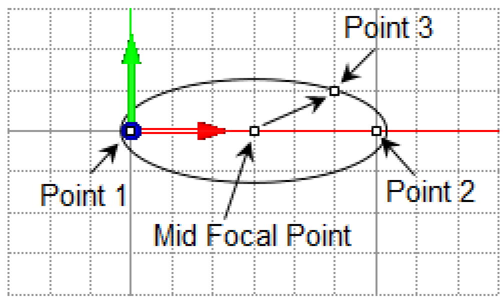
Ellipse:: Pick first foci point or enter coordinates x,y and z

Ellipse:: Pick second foci point or enter coordinates x,y and z

Ellipse:: Pick point or enter coordinates x,y and z

Basic Procedure

	Screen Pick / Command Input
Step 1	 Ellipse:: Pick first foci point or enter coordinates x,y and z

Step 2	 Ellipse:: Pick second foci point or enter coordinates x,y and z
Step 3	 Ellipse:: Pick point or enter coordinates x,y and z
Final	

Related Topics

[Curves Pane, Curve Modeling Ribbon Bar]

5.4.1.32 Nurbs Curve

Non-uniform rational basis-spline (NURBS) is a mathematical way of defining curves. The control points determine the shape of the curve. Typically, each point of the curve is computed by taking a weighted sum of a number of control points. The degree of the Nurbs curve can be set between 1 and 11.

Create a NURBS Curve by first entering the degree of B-Spline and then pick point or enter the X, Y, Z coordinates. Open and closed curves can be created using NURBS Curve. NURBS Curve icon can be selected in the Geometry Ribbon Bar under the Curves tab.

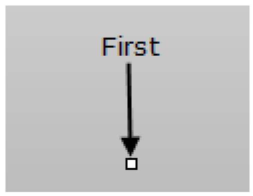
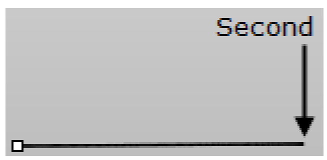
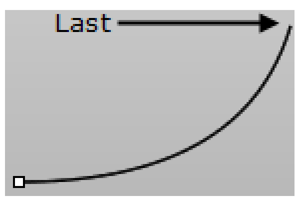
Command Prompts

Nurbs curve:: Enter degree of BSpline (1 < degree < 11)::[degree=2]

Nurbs curve:: Pick point or enter coordinates x,y and z

Nurbs curve:: Pick point or enter coordinates x,y and z

Basic Procedure

	Screen Pick	Command Input
Step 1		Enter degree of B Spline (1 < degree < 11):: [degree=2] Command 2
Step 2		(Optional) Pick point or enter coordinates x,y and z
Step 3		(Optional) Pick point or enter X, Y, [Z] coordinates for the second point.
Step 4		(Optional) Pick point or enter X, Y, [Z] coordinates for the last point.

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.33 Text Curves

Create text curves from keyed-in text. The outlines of letters are created as a series of separate curves.

Command Prompts

Enter parameters and text in dialog.

Text:: Enter start point of text

Dialog Box: Text to Create

Dialog Box: Text to Create

Choose Font

VisualCAD supports over 300 different text fonts. Select one from the list.

Auto Kerning

Kerning is the process of adjusting the spacing between characters in a proportional font, usually to achieve a visually pleasing result. Kerning adjusts the space between individual letter forms, while tracking (letter-spacing) adjusts spacing uniformly over a range of characters. In a well-kerned font, the two-dimensional blank spaces between each pair of characters all have a visually similar area.

- Use the slider to adjust Kerning or check the box to Auto Kerning.

Other Options

- Bold/Italic
- Text Size
- Allow Single Line Font

Basic Procedure

	Screen Action	Comments

Step 1		Font: font style, size, bold, and italic can be specified. Kerning slider bar: The spacing between the characters can be set between 0 (closest) to 1000 (farthest). Enter text.
Step 2		(Optional) Enter X, Y, [Z] coordinates for the text start point 
Final		Top line kerning = 0 Middle line kerning = 250 Bottom line kerning = 500

Notes

1. The text curves will be created on a plane parallel to the Construction Plane.
2. After the text is created, the curves can be independently manipulated.

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

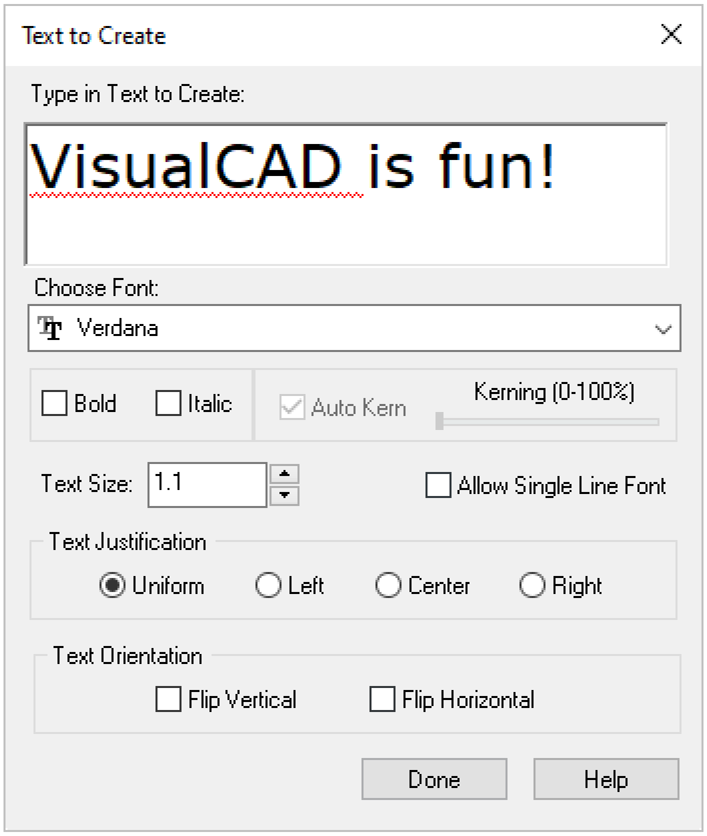
5.4.1.34 Text on Curve

Create text curves that follow a curve. The outlines of letters are created as a series of separate curves.

Command Prompts

Enter parameters and text in dialog.

Dialog Box: Text to Create



Text to Create (on Curve)

Choose Font

VisualCAD® supports over 300 different text fonts. Select one from the list.

Auto Kerning

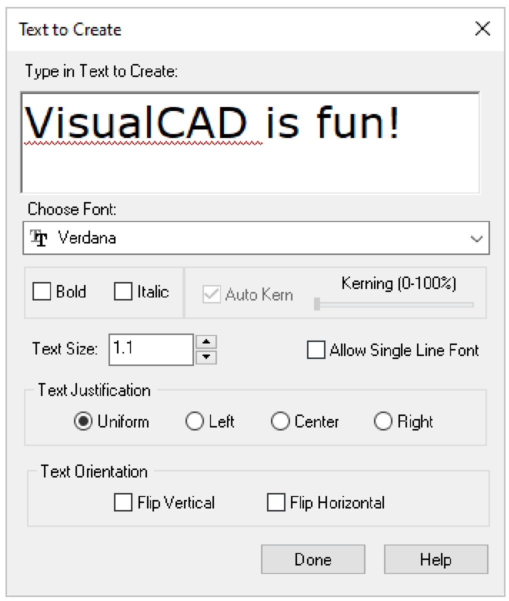
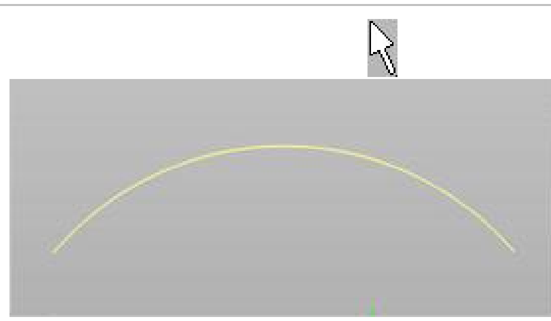
Kerning is the process of adjusting the spacing between characters in a proportional font, usually to achieve a visually pleasing result. Kerning adjusts the space between individual letter forms, while tracking (letter-spacing) adjusts spacing uniformly over a range of characters. In a well-kerned font, the two-dimensional blank spaces between each pair of characters all have a visually similar area.

- Use the slider to adjust Kerning or check the box to Auto Kerning.

Other Options

- Bold/Italic
- Text Size
- Allow Single Line Font
- Text Justification
- Text Size
- Text Spacing
- Text Orientation

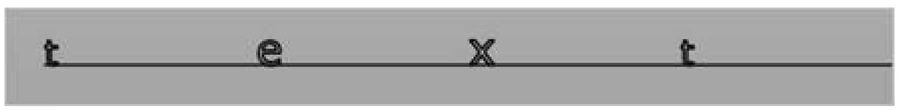
Basic Procedure

	Screen Activity	Comments
Step 1	 Text to Create (on Curve)	See Dialog Box: Text to Create below. Font: font style, size, bold, and italic can be specified. Kerning slider: The spacing between the characters can be set between 0 (closest) to 1000 (farthest). Enter text.
Step 2		

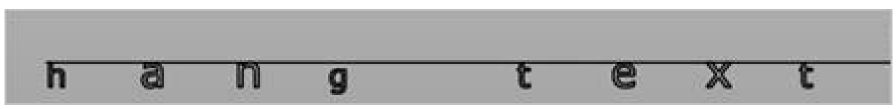
Final		

Notes

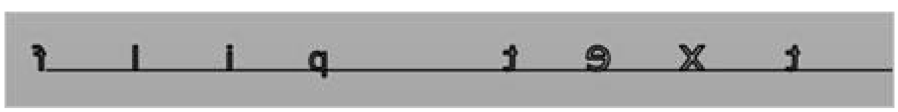
1. The text curves will be created along the curve in oriented to the current view.
2. After the text is created, the curves can be independently manipulated.
3. The text to appear is typed into the box. It will appear in the box in the selected font, bold, and italic applied. Changes in size do not appear in the text in the box but do take effect when the text is actually created.



4. Hanging text has the top of the letters on the curve; whereas, the default is the bottom of the letters are on the curve.



5. Flip text flips each individual letter while keeping the sequence of letters the same.



6. The plane of the text is the viewing plane of the currently selected view. It doesn't have to be a construction plane.

Related Topics

[Curves Pane, Curve Modeling Ribbon Bar]

5.4.1.35 Curve, Spiral

 Create a spiral curve in a plane.

Command Prompts

Spiral :: Enter the number of revolutions::[N=4]

Spiral :: Pick center point or enter coordinates x,y and z

Spiral :: Enter radius or pick radius point::[R=4.000000]

Basic Procedure

	Screen Pick	Command Input
Step 1		Enter the number of revolutions of the spiral. The Prompt shows the default. 4
Step 2		(Optional) Enter X, Y, [Z] coordinates for the center point. 0,0,0
Step 3		(Optional) Enter the radius. 2.25 or... (Optional) Enter X, Y, [Z] coordinates for the second point which determines the radius. 1,1,0

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.36 Curve, Helix

Create a helix curve.

Command Prompts

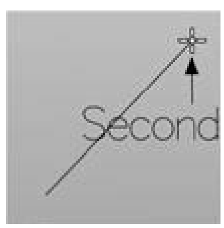
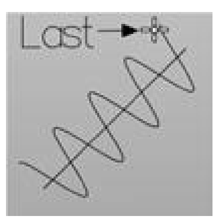
Helix :: Enter the number of revolutions::[N=4]

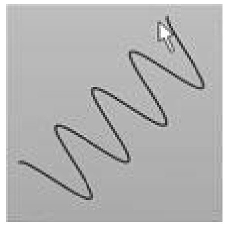
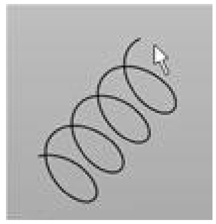
Helix :: Pick center point or enter coordinates x,y and z

Helix :: Enter the height of the helix::[H=2.000000]

Helix :: Enter radius or pick radius point::[R=2.000000]

Basic Procedure

	Screen Pick	Command Input
Step 1		Enter the number of revolutions of the helix. The Prompt shows the default. 4
Step 2		(Optional) Enter X, Y, [Z] coordinates for the center point. 0,0,0
Step 3		(Optional) Enter X, Y, [Z] coordinates for the second point which determines the axis and height of the helix. 1,1,0
Step 4		(Optional) Enter the radius. 2.25 or... (Optional) Enter X, Y, [Z] coordinates for the third point which determines the radius of the helix. 1,0,0

Final	 	

Notes

1. Intermediate geometry is discarded and only the helix is retained.

Related Topics

[Curves Pane, Curve Modeling Ribbon Bar]()

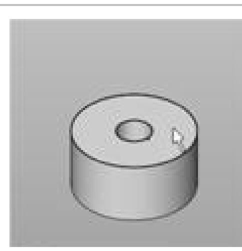
5.4.1.37 Curve, Flat Area

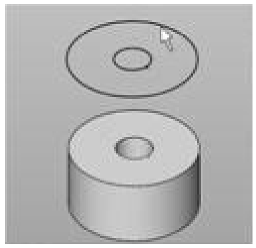
Create a curve(s) forming the boundary of a flat-area region. The region could be a face of a mesh or a plane.

Command Prompts

Flat Area :: Pick flat area:

Basic Procedure

	Screen Pick	Command Input
Step 1		none

Final		

Notes

1. If there are holes in the area, multiple curves will be made, one for the outer boundary, and one curve each for the inner boundary of the holes.

Related Topics

[Curves Pane, Curve Modeling Ribbon Bar]

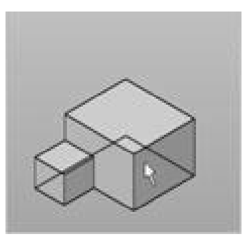
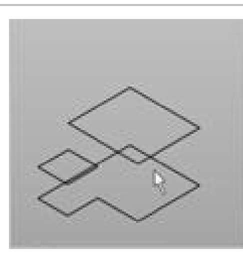
5.4.1.38 Curve, All Flat Areas

Create a set of curves enclosing flat-area regions. All flat regions parallel to the construction plane will generate curves including surfaces on multiple meshes.

Command Prompts

Click on the function 'All Flat Areas'

Basic Procedure

	Screen Pick	Command Input
Select Function		none
Final		

Notes

1. Each of the generated curves is independent after it is created.

Related Topics

[Curves Pane, Curve Modeling Ribbon Bar]

5.4.1.39 Silhouette Curves

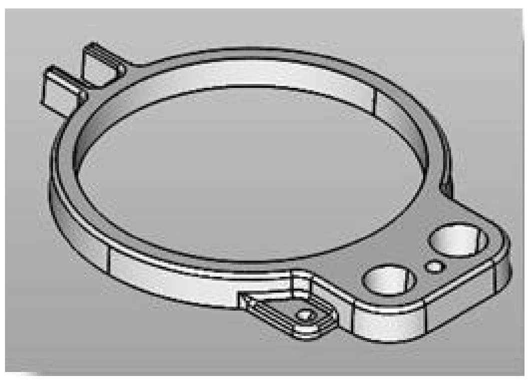
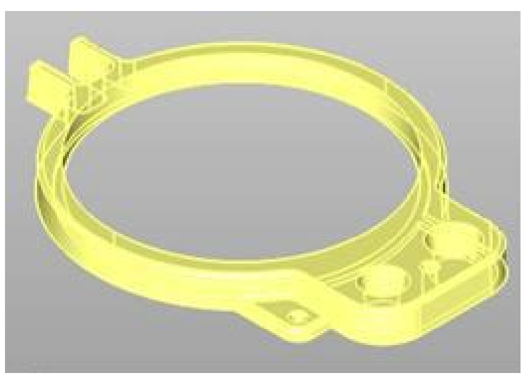
 Creates a 2D curve that represents the boundaries of the part for all selected solids and surface geometries on the screen. The curves will be independent once created and will be parallel to the active construction.

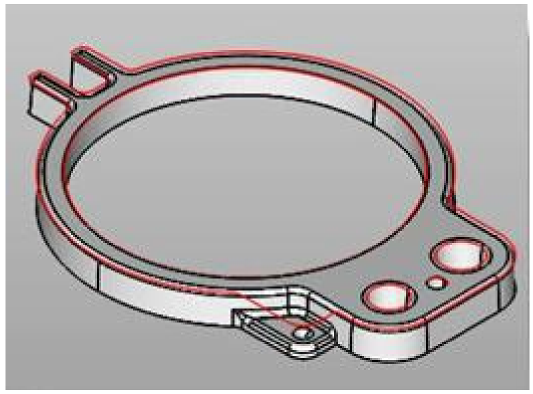
Reminder: You might want to set a new Layer as the destination for the newly created curves.

Command Prompts

Silhouette:: Pick surface(s) for creating silhouette area curves

Basic Procedure

	Screen Entry	Command Input
Step 1	 	Select Surfaces by single pick or rectangular selection. Hit the ENTER key when done.

Final		

Related Topics

[Curves Pane, Curve Modeling Ribbon Bar]

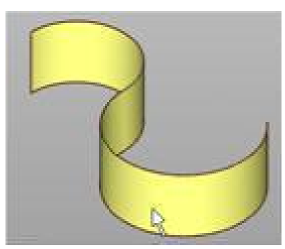
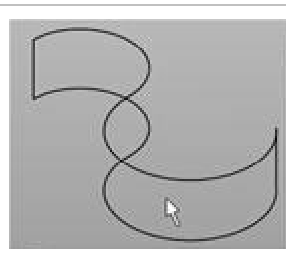
5.4.1.40 Curve, Surface Boundary

 Create a single curve from the outer boundary of a surface.

Command Prompts

Surface Boundary :: Pick a surface to extract edges from

Basic Procedure

	Screen Pick	Command Input
Step 1		none
Final		

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.41 Curve, Bounding Rect.

Create a rectangular curve that is derived from a volumetric box that surrounds selected geometry objects.

The volumetric box and rectangle are oriented to either the Construction Plane or the World Coordinate System (your choice). The bounding rectangle will then be created at the top, middle, or bottom of that volumetric box (your choice).

Command Prompts

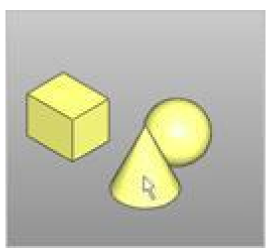
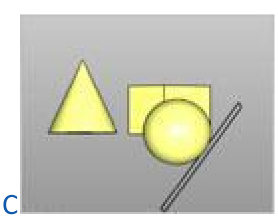
Bounding Rect:: Pick Objects for command. Hit Right Mouse Button/Enter when done.

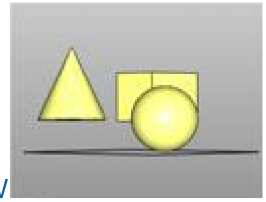
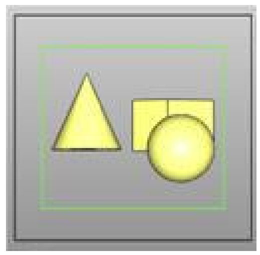
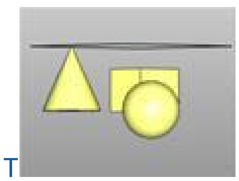
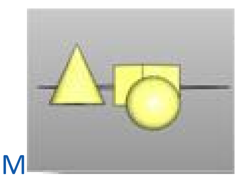
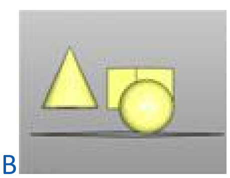
Bounding Rect:: Enter Orientation [CPlane=C,World=W]

Bounding Rect:: Enter offsets in X & Y Direction [X=0.000000,Y=0.000000]

Bounding Rect:: Enter Location [Bottom=B,Mid=M,Top=T]

Basic Procedure

	Screen Pick or Screen Results	Command Input
Step 1	 Select all objects to be included in the volumetric box. Hit ENTER to end the selection.	
Step 2	 C	Enter 'C' or 'W'. 'C' orients the volumetric box to the Construction Plane. C

	 W	'W' orients the volumetric box to the World Coordinate System. 
Step 3		Enter X and Y offsets. These are incremental offsets that will enlarge the volumetric box in the X and Y directions respectively. Negative values are not allowed. 
Step 4	 T  M  B	Enter 'T', 'M', or 'B'. 'T' creates the rectangle at the top of the volumetric box. 'M' creates the rectangle at the middle of the volumetric box. 'B' creates the rectangle at the bottom of the volumetric box. 

Related Topics

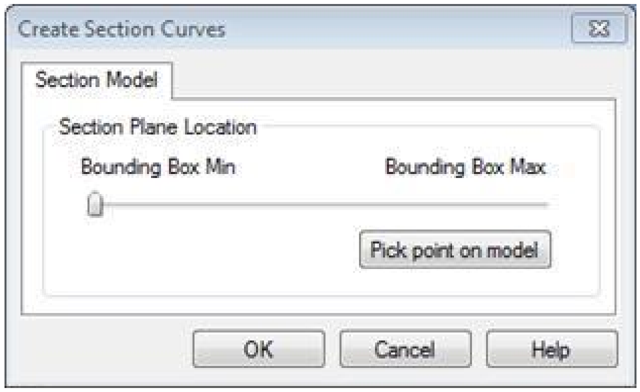
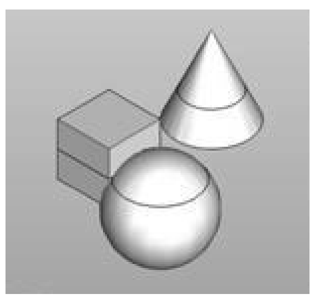
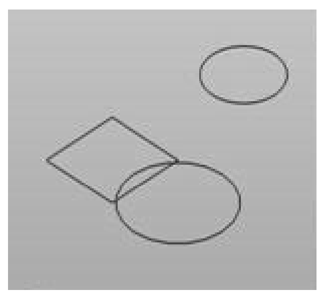
Curves Pane, Curve Modeling Ribbon Bar

5.4.1.42 Section Curves

Creates cross-section curves through all solids, surface and mesh geometries on the screen. The curves will be independent once created.

Reminder: You might want to set a new Layer as the destination for the newly created curves.

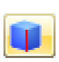
Basic Procedure

	Screen Entry	Command Input
Step 1	 Adjust the Bounding Box slider to raise or lower the cutting plane (dynamically shown on the models). When the cutting plane is positioned correctly, hit OK on the dialog to create the curves. or... ...Pick a point on the model to determine where the cutting plane is positioned.	(Optional) Enter X, Y, and Z values that define the point of the cutting plane.  Hit the ENTER key when done.
Final	 	

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.1.43 Extract Curve



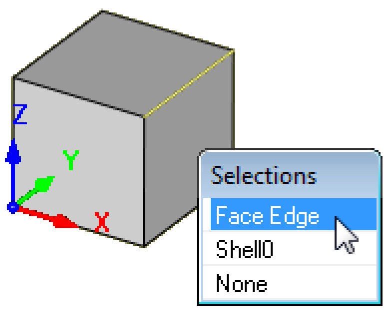
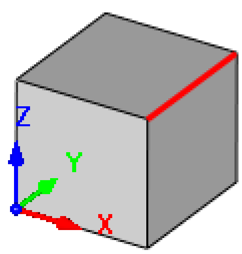
Creates (extracts) a curve from a surface edge. The curve will be independent of the surface once created.

Reminder: You might want to set a new Layer as the destination for the newly created curves.

Command Prompts

Extract curve from face edge:: Select face edge(s) and hit Right Mouse Button/Enter when done

Basic Procedure

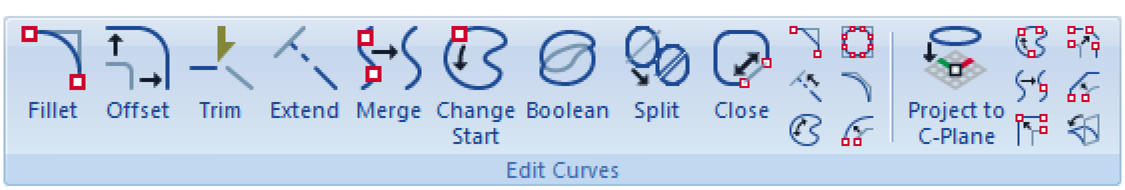
	Screen Entry	Command Input
Step 1	Select the surface edge to create a curve from. You can pre-select the surface edge before executing this command. 	Right-click or press <ENTER> when done.
Final		

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.2 Edit Curves

The Curve Modeling Ribbon Bar contains the following Edit Curves pane with associated functions.



Curve Modeling Ribbon Bar
Edit Curves Pane

Related Topics

- Fillet 2 Curves
- Create Offset Curves
- Trim a Curve
- Extend Curve
- Merge Curves
- Curve Boolean
- Split by Curve
- Close Curve
- Change Curve Start Point
- Chamfer Curves
- Extend Curve by Distance
- Reverse Curve Direction
- Automatically Fillet Curves of a Polyline
- Fit an Arc to a Polyline
- Smooth a Curve
- Project Curves to a C-Plane
- Create a Curve Chain
- Split Curves
- Break Curves at Sharp Corners
- Explode Curves
- Reduce a Curve
- Wrap a Curve onto a Cylinder
- Curve Modeling Ribbon Bar

5.4.2.1 Fillet

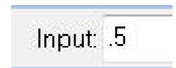
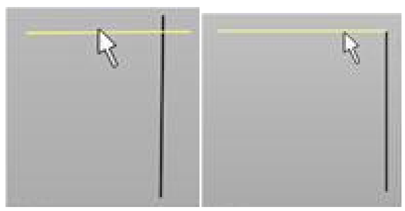
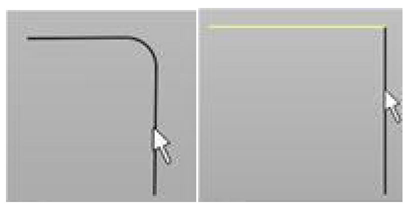
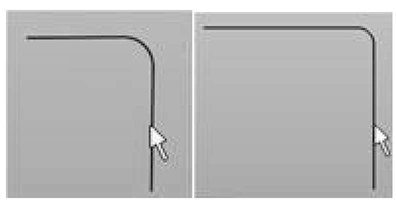
Create a tangent arc blend (fillet) at the intersection of two curves. The radius of the fillet is user specified. **Zero radius fillets can be created with this command by entering 0 as the fillet radius.**

Command Prompts

Fillet:: Enter fillet radius [R=0.500000]

Fillet:: Pick first curve

Basic Procedure

	Screen Pick	Command Input
Step 1		Enter the radius of the desired corner fillet. 
Step 2		
Step 3		
Final		

Notes

1. If necessary, the ends of the target curves will be trimmed back to meet the ends of the fillet arc.
2. Curve ends will not be extended to meet the intended fillet arc. If the ends are too short, the fillet will not be created.

Related Topics

Edit Curves Pane, Curve Modeling Ribbon Bar

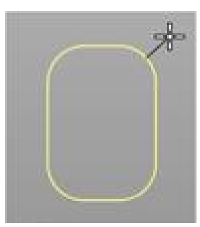
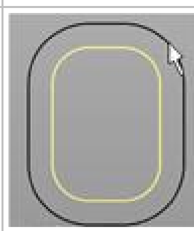
5.4.2.2 Offset

Create a new curve that is offset from a selected curve. The curve may be open or closed. **Corners may be rounded or sharp.**

Command Prompts

1. Offset:: Select convex corner type [r - rounded; s - sharp] or Pick curve to offset [convex corners will be rounded]
2. Pick curve to offset
3. Enter offset distance [D=1.000000]

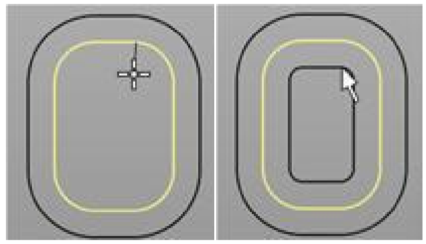
Basic Procedure

	Screen Pick	Command Input
Step 1		Select convex corner type [r - rounded; s - s Pick curve to offset [convex corners will be rounded]
Step 2		
Step 3	Use the cursor to indicate the offset side and offset distance. 	(Optional) Enter the offset distance.  Note: Negative offset distances are not allo To offset toward the inside, position the cu inside the curve and enter a positive offset distance.
Final		

Notes

1. The existing curve may be pre-selected.

2. Sharp, outside corners in the original curve will result in radius offset corners in the new curve.



An example of offsetting on the inside of the curve using the cursor pick position.

Related Topics

[Edit Curves Pane, Curve Modeling Ribbon Bar]

5.4.2.3 Trim

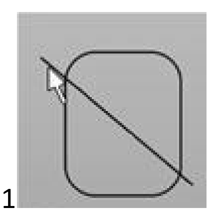
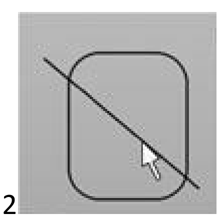
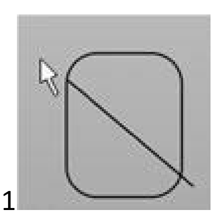
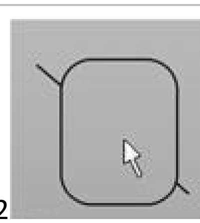
Modify a curve by trimming away part of it. The curve can be trimmed in the following two cases:

1. Between its end point and an intersection with another curve, or
2. Between two intersections with other curves.

Command Prompts

Prompt: Trim:: Pick curve to trim

Basic Procedure

	Screen Pick	Command Input
Step 1	1  or 2 	none
Final	1  or 2 	

Notes

1. Where you select the curve also selects the segment to be trimmed away.

Related Topics

[Edit Curves Pane, Curve Modeling Ribbon Bar]

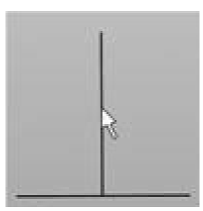
5.4.2.4 Extend

Extend a curve along its tangency to its nearest intersection with another geometry object.

The curve is extended only on the end where it is picked. To extend the other end, the EXTEND function must be used again by picking the other end.

Command Prompts

Basic Procedure

	Screen Pick	Command Input
Step 1		none
Final		

Related Topics

[Edit Curves Pane, Curve Modeling Ribbon Bar]

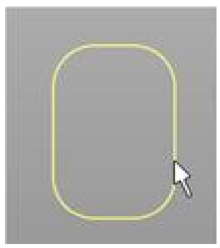
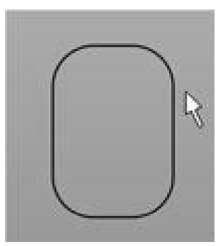
5.4.2.5 Merge

Convert two or more curves into one curve. The adjacent endpoints must be within VisualCAD tolerance. See **Note 1** below.

Command Prompts

Merge:: Pick curves to merge. Hit Right Mouse Button/Enter when done.

Basic Procedure

	Screen Pick	Command Input
Step 1		none
Step 2		
Final		

Notes

1. To set the system tolerance for curve hookup, go to: Home > Options > Tolerance and Units > Curve Hookup Tolerance.
2. If curves do not merge as expected, check to see if their end point proximity exceeds the system hookup tolerance.

Related Topics

Edit Curves Pane, Curve Modeling Ribbon Bar

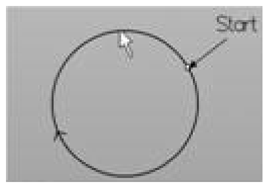
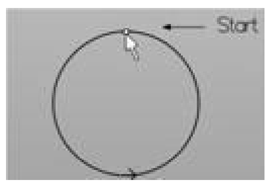
5.4.2.6 Change Start

Move the start/end position on a closed curve.

Command Prompts

Change Start:: Pick closed curve to change start point

Basic Procedure

	Screen Pick	Command Input
Step 1		none
Final		

Notes

1. This function has no effect on open curves.
2. To show curve start point, go to Home > Options > Display > Curve Display Style. Check the box Display Curve / Region Start Point.

Related Topics

Edit Curves Pane, Curve Modeling Ribbon Bar

5.4.2.7 Boolean

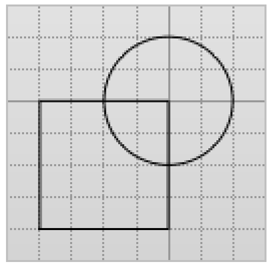
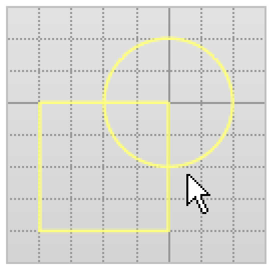
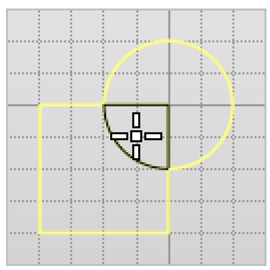
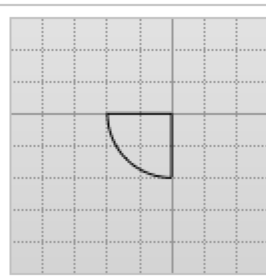
Use this command to perform boolean operations on selected curves. The curves must intersect each other and lie on the construction plane. Also be sure to read the notes section below.

Command Prompts

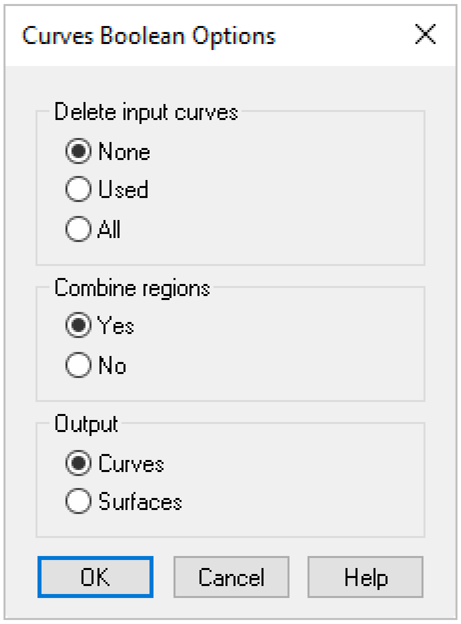
Boolean:: Pick curves to boolean and hit Right Mouse Button/Enter when done.

Boolean:: Pick point or enter coordinates x,y and z.

Basic Procedure

	Screen Pick	Command Input
Step 1	 	Select the curves to boolean and hit Right Mouse Button/Enter when done.
Step 2		Make desired selections from the Curve Boolean options dialog. (see below)
Final		Delete input curves: All Combine Regions: Yes Output: Curves

Curve Boolean Options Dialog



Corve Booleam Dialog

Delete input curves

- None: Do not delete the input curves.
- Used: Delete only the affected input curves. Leave all other selected curves intact.
- All: Delete ALL selected input curves, even those that are unaffected by the operation.

Combine regions

- Yes: Merge all affected curves into one curve region.
- No: Do not merge the affected curves.

Output

- Curves: The resulting regions should be curves.
- Surfaces: The resulting regions should be planar surfaces.

Notes

1. All selected curves must be planar and lie on the construction plane.
2. Experiment with each dialog option until you understand the expected results.

Related Topics

Edit Curves Pane, Curve Modeling Ribbon Bar

5.4.2.8 Split by Curve

Split a curve(s) by another curve. If the curves to split are nested, you can choose to honor the nest (Y) or not (N).

Command Prompts

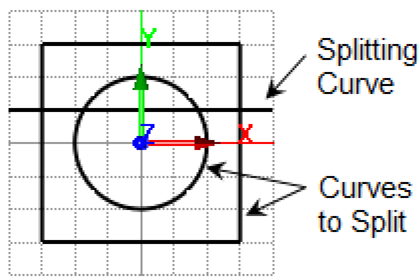
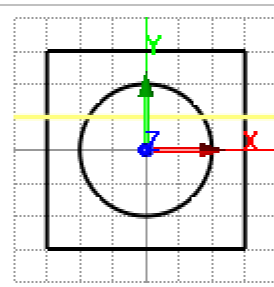
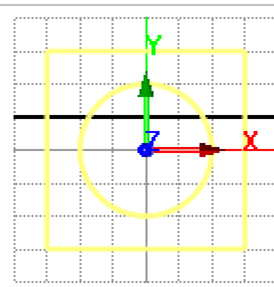
Split:: Pick Splitting curve

Split:: Pick curves to split. Hit Right Mouse Button/Enter when done.

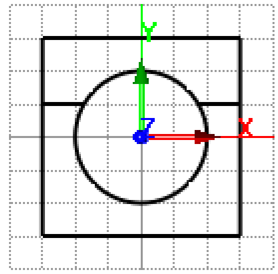
Split:: Honor Nesting option(Y/N)?

Split:: Done

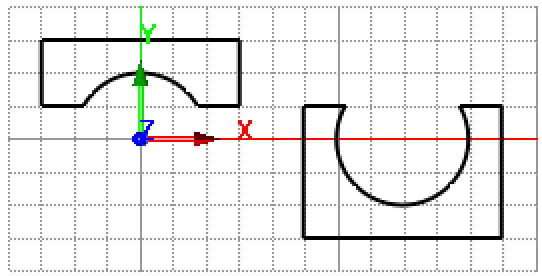
Basic Procedure

	Screen Pick / Command Input
Geometry to split	
Step 1	 Split:: Pick Splitting curve
Step 2	 Split:: Pick curves to split. Hit Right Mouse Button/Enter when done.

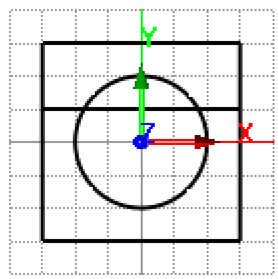
Step 3



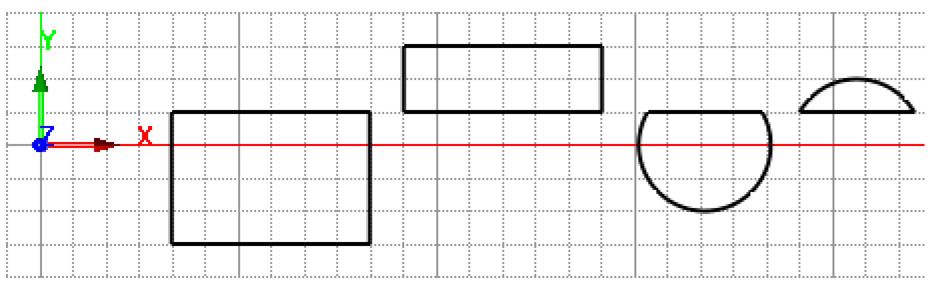
Split:: Honor Nesting option(Y/N)? Y (Results)



Split:: Honor Nesting option(Y/N)? Y (Results separated)



Split:: Honor Nesting option(Y/N)? N (Results)



Split:: Honor Nesting option(Y/N)? N (Results separated)

Related Topics

[Curves Pane, Curve Modeling Ribbon Bar]

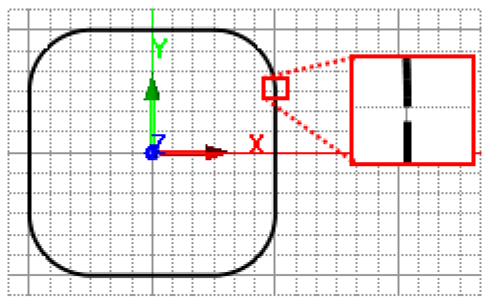
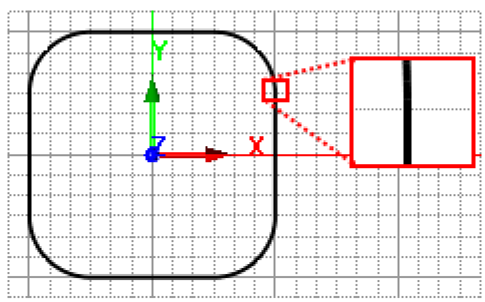
5.4.2.9 Close Curve

Use this command to close a curve by automatically connecting its endpoints. Note that this command will not close multiple curves. There needs to be a single curve and multiple curves that have been merged into a single curve. Also **Note**: You can use the Merge Curve command to close multiple curves as long as the Curve Hookup Tolerance is larger than the gap you are attempting to close. You can find the Curve Hookup Tolerance parameter on the Tols. & Units tab of the VisualCAD Options dialog.

Command Prompts

Close:: Pick curve to close and Hit Right Mouse Button/Enter when done.

Basic Procedure

	Screen Pick / Command Input
Curve to Close	
Step 1 **Final**	 Close:: Pick curve to close and Hit Right Mouse Button/Enter when done.

Related Topics

Curves Pane, Curve Modeling Ribbon Bar

5.4.2.10 Chamfer

Create a chamfer at the intersection of two curves. The length of the chamfer is user specified.

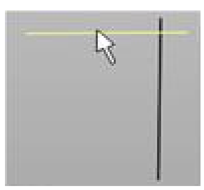
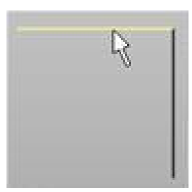
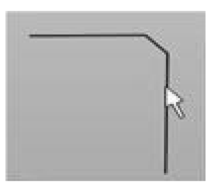
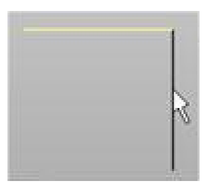
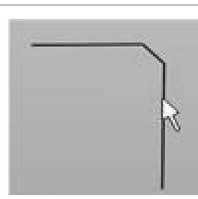
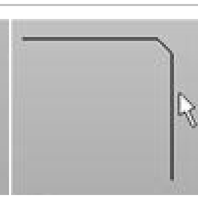
Command Prompts

Chamfer:: Enter chamfer length [C=0.500000]

Chamfer:: Pick first curve

Chamfer:: Pick second curve

Basic Procedure

	Screen Pick	Command Input
Step 1		Enter the length of the desired chamfer. .5
Step 2	 	
Step 3	 	
Final	 	

Notes

1. If necessary, the ends of the target curves will be trimmed back to meet the ends of the chamfer line.
2. Curve ends will not be extended to meet the intended chamfer. If the ends are too far apart, the chamfer will not be created.

Related Topics

[Edit Curves Pane, Curve Modeling Ribbon Bar]

5.4.2.11 Extend by Dist

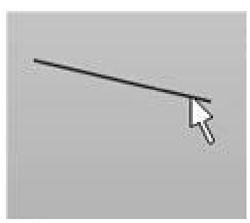
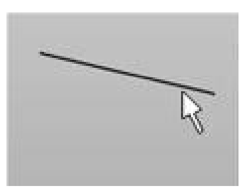
Extend a curve along its tangency by the specified distance. The curve is extended only on the end where it is picked. To extend the other end, the EXTEND function must be used again by picking the other end.

Command Prompts

Extend by Dist:: Enter extension distance::[L=1.000000]

Extend by Dist:: Pick Curve to Extend

Basic Procedure

	Screen Pick	Command Input
Step 1		Enter extension distance .5
Final	 	

Notes

1. This command can be used with arcs, polylines, poly-curves and Nurbs curve.

Related Topics

[Edit Curves Pane, Curve Modeling Ribbon Bar]

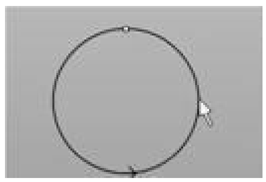
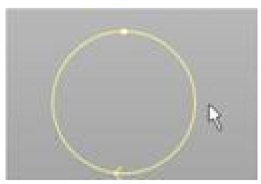
5.4.2.12 Reverse

 Reverse the definition direction of curves.

Command Prompts

Reverse:: Pick curves to reverse and hit Right Mouse Button/Enter when done

Basic Procedure

	Screen Pick	Command Input
Step 1		none
Final		

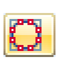
Notes

1. To show curve direction arrow: Home > Options > Display > Curve Display Style. Check the box Display Curve / Region Direction Arrow.

Related Topics

[Edit Curves Pane, Curve Modeling Ribbon Bar]()

5.4.2.13 Auto Fillet

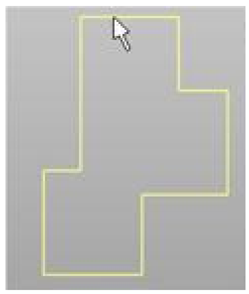
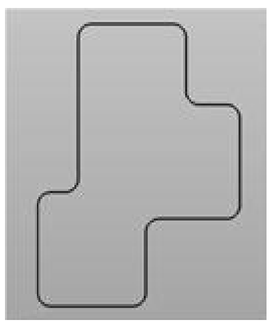
 Rounds the corners of Polycurves and Polylines with arcs of a specified radius at the intersection of two curves.

Command Prompts

Auto Fillet:: Enter fillet radius::[0.250000]

Auto Fillet:: Pick polylines for corners rounding . Hit Right Mouse Button/Enter when done.

Basic Procedure

	Screen Pick	Command Input
Step 1		Enter the radius of the desired corner fillet. .5
Step 2		
Final		

Notes

1. The prompt displays the filleted corner count.

Related Topics

[Edit Curves Pane, Curve Modeling Ribbon Bar]

5.4.2.14 Arc Fit

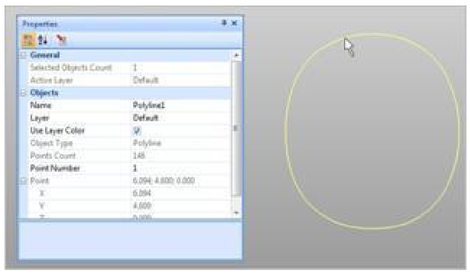
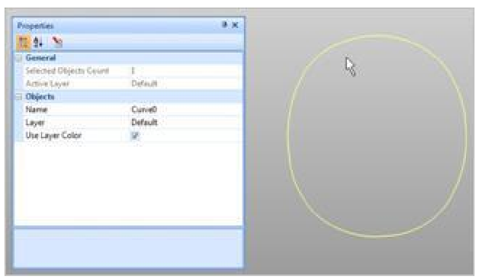
 Fit arcs to a polyline.

Command Prompts

Arc Fit:: Enter fillet radius::[0.010000]

Arc Fit:: Pick polylines for corners rounding . Hit Right Mouse Button/Enter when done.

Basic Procedure

	Screen Pick	Command Input
Step 1		Enter the fillet radius and press enter. 0.001
Step 2		
Final		

Notes

1. The properties dialog is displayed to show the geometry type as polyline before arc fit and as a curve after the arc fit command is executed.

Related Topics

[Edit Curves Pane, Curve Modeling Ribbon Bar]

5.4.2.15 Smooth

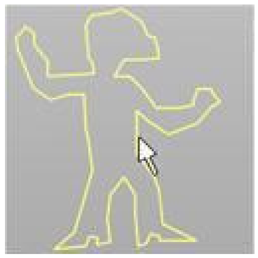
Modify a curve to smooth sharp corners.

Command Prompts

Smooth:: Pick curve(s) to smooth and hit Right Mouse Button/Enter when done

Smooth:: Enter curve smoothing tolerance::[0.002000]

Basic Procedure

	Screen Pick	Command Input
Step 1		
Step 2		Enter the smoothing tolerance. A smaller number produces greater smoothing. .020
Final		

Notes

1. Curves may be pre-selected.
2. Highly smoothed curves will contain many underlying segments. This increases computation when these curves are used in calculations.

Related Topics

[Edit Curves Pane, Curve Modeling Ribbon Bar](#)

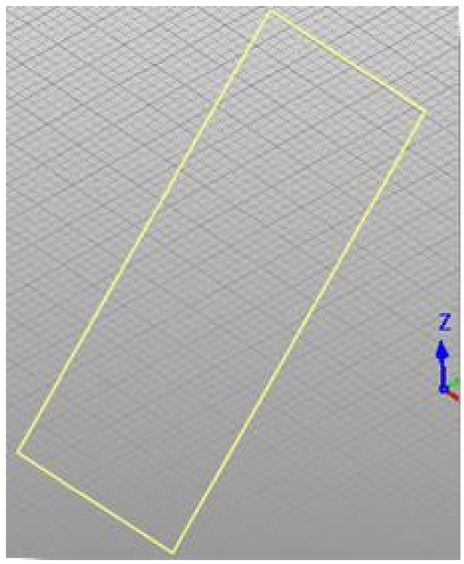
5.4.2.16 Project to CPlane

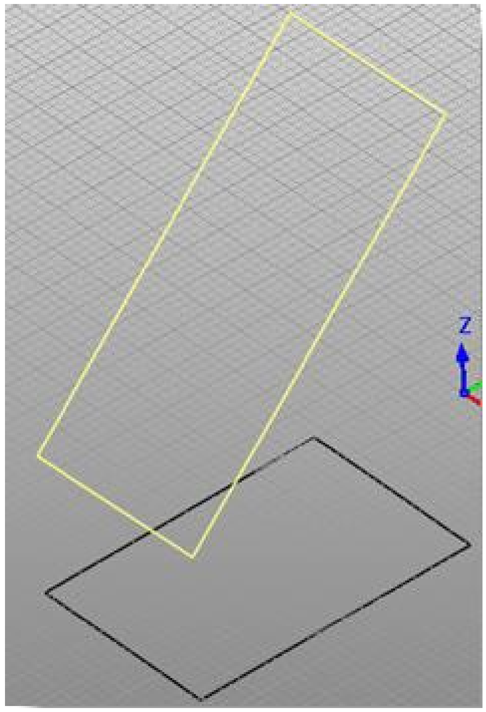
Projects the selected curves to active construction plane. Split an existing curve into two segments at the pick point.

Command Prompts

Project to CPlane:: Pick curves to be projected. Hit Right Mouse Button/Enter when done.

Basic Procedure

	Screen Pick	Command Input
Step 1	Pick curves to be projected with left mouse click selection or dragging a window with left mouse hold. 	none
Final	Hit right mouse button or hit Enter on keyboard to accept selection. Selected geometry is now projected to the C-Plane coordinate.	

Related Topics

[Edit Curves Pane, Curve Modeling Ribbon Bar]

5.4.2.17 Chain

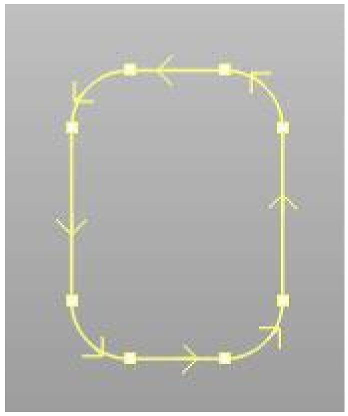
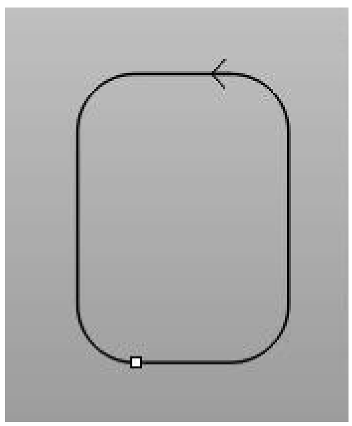
Create a single curve out of multiple curves that form a closed loop. You can automatically select a chain of curves or edges using the <Shift> key while selecting. The curve chain does not have to be created using this command. See Chain Select for more information.

Command Prompts

Chain:: Pick curve to start chain

Basic Procedure

	Screen Pick	Command Input

Step 1	 Select any one curve of a set of curves that form a closed region.	none
Final		

Notes

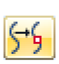
1. To show curve direction arrow and start point: Home > Options > [Display](#) > Curve Display Style. Check the box Display Curve / Region Direction Arrow and Display Curve / Region Start Point.

Related Topics

[Edit Curves Pane, Curve Modeling Ribbon Bar](#)

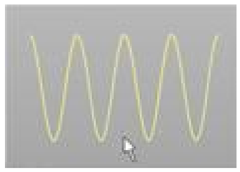
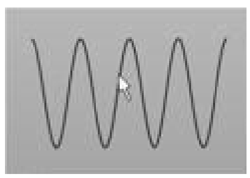
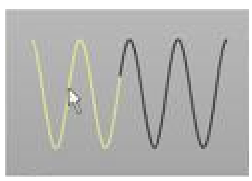
[Chain Select](#)

5.4.2.18 Split

 Split an existing curve into two segments at the pick point.

Command Prompts

Basic Procedure

	Screen Pick	Command Input
Step 1		none
Step 2	Move the cursor to determine the split position. Left click causes split. 	
Final		

Notes

1. If the curve is pre-selected, the function will begin at Step 2 above.
2. If the curve is not pre-selected, then the pick point becomes the split point.
3. If the curve is picked very near a sharp corner, the split will be at the corner.

> To create breaks in curves at multiple corners that have a given angle size, see the Break command.

Related Topics

Edit Curves Pane, Curve Modeling Ribbon Bar

5.4.2.19 Break

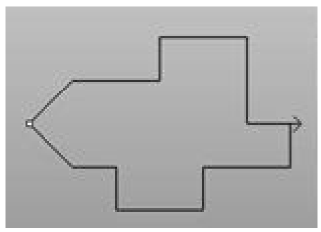
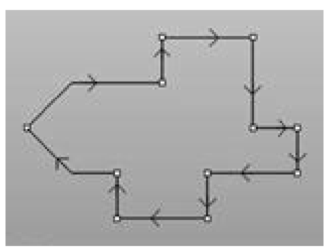
Break a curve into segments at its sharp corners, where the corners are equal or less than a user-entered angle value.

Command Prompts

Prompt: Break:: Pick curve(s) to break and hit Right Mouse Button/Enter when done

Prompt: Break:: Enter limiting angle for breaking curves::[90.000000]

Basic Procedure

	Screen Pick	Command Input
Step 1		
Step 2		Enter the angle where breaks should occur. Input: 90
Final		

Notes

1. Curves may be pre-selected.
2. **Tip**: If you wish to break a curve at selected corners and are not concerned about the corner angle, then use the Curve Modeling > Edit Curves > Split command to directly select a corner of the curve where you want the curve break to occur.

Related Topics

Edit Curves Pane, Curve Modeling Ribbon Bar

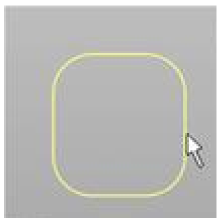
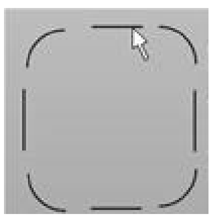
5.4.2.20 Explode

 Convert one curve into multiple curve segments.

Command Prompts

Explode:: Pick curves to explode. Hit Right Mouse Button/Enter when done.

Basic Procedure

	Screen Pick	Command Input
Step 1		none
Final	Gaps are magnified to show separation into independent curves. 	

Related Topics

[Edit Curves Pane, Curve Modeling Ribbon Bar]

5.4.2.21 Reduce

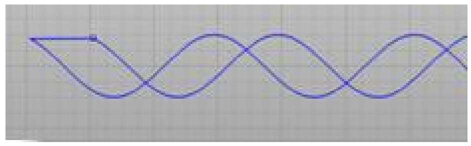
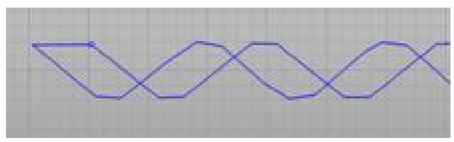
 Modify a curve by reducing the number of underlying segments which make up the curve.

Command Prompts

Reduce:: Pick curve(s) to reduce and hit Right Mouse Button/Enter when done

Reduce:: Enter curve reduction tolerance::[0.020000]

Basic Procedure

	Screen Pick	Command Input
Step 1		
Step 2		Enter the reduction tolerance. A larger number produces fewer underlying curves. 
Final		

Notes

1. Curves may be pre-selected.
2. This function will improve the speed of computation for curves that have had their underlying segments reduced.
3. Reducing the number of underlying segments of a curve will result in less accurate calculations with that curve.

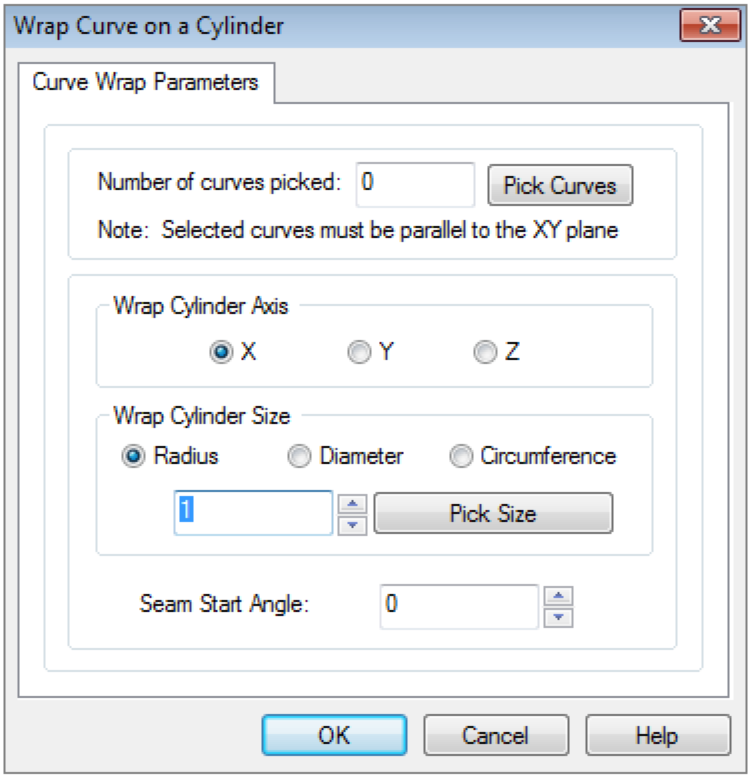
Related Topics

Edit Curves Pane, Curve Modeling Ribbon Bar

5.4.2.22 Wrap

Wrap curves onto a construction cylinder whose center-line is aligned with one of the principle axes of the CPlane.

Dialog Box: Wrap Curve on a Cylinder

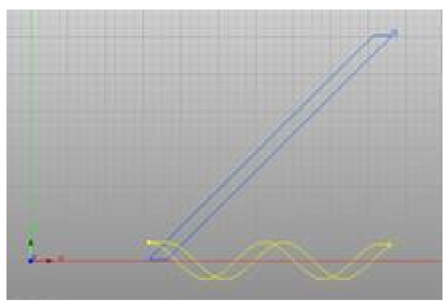
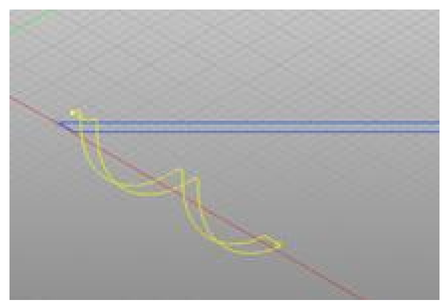


Dialog Box: Wrap Curve on a Cylinder

- Number of curves picked
  The number of curves selected is displayed. Select the Pick Curves button to add more curves to the selection.
- Wrap Cylinder Axis
  Select X, Y or Z to determine the cylinder axis.
- Wrap Cylinder Size
  Select Radius, Diameter or Circumference to determine the size of the cylinder to wrap the selected curves on. Then enter the appropriate value or select the Pick Size button to select two points to set the size value.
- Seam Start Angle
  Specify the start angle for the seam of the cylinder according to the right-hand rule (counter-clockwise).

Basic Procedure

	Dialog Interaction

Initial	
Step 1	Select the Pick Curves button and pick the curves to be wrapped. The curv may be pre-selected.
Step 2	Select one of the C-Plane axes that is used as the construction cylinder's a
Step 3	Choose the method to input the size of the construction cylinder.
Step 4	Enter the dimension size of the construction cylinder. You may pick a geo object to determine that size.
Step 5	Enter the start angle on the cylinder where wrapping will begin.
Step 6	Hit the OK button
Final	 

Notes

1. The curves to be wrapped must be co-planar and parallel to the CPlane.
2. The curves will be wrapped in a positive direction around the construction cylinder according to the right-hand rule (counter-clockwise).

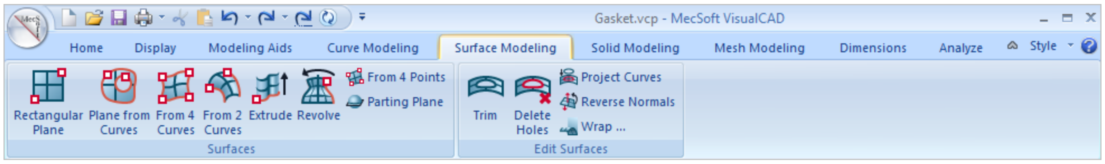
Related Topics

Edit Curves Pane, Curve Modeling Ribbon Bar

5.5 Surface Modeling

The Surface Modeling Ribbon Bar contains Surfaces and Edit Surfaces panes and associated functions.

The Surface Modeling Ribbon Bar



Surface Modeling Ribbon Bar

Related Topics

[Surfaces Pane, Surface Modeling Ribbon Bar]

[Edit Surfaces Pane, Surface Modeling Ribbon Bar]

[The Ribbon Bar]

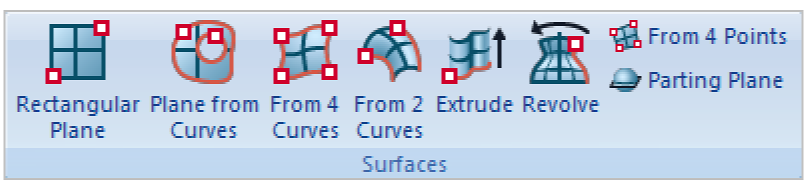
[Customize Quick Access Toolbar]

[Customize Dialog]

[VisualCAD Options]

5.5.1 Surfaces

The Surface Modeling Ribbon Bar contains the following Surfaces pane with associated functions.



Surface Modeling Ribbon Bar
Surfaces Pane

Related Topics

[Create a Rectangular Plane]

[Create a Plane from Curves]

[Create a Surface from 4 Curves]

[Create a Surface from 2 Planar Curves]

[Create an Extruded Surface]

[Create a Revolved Surface]

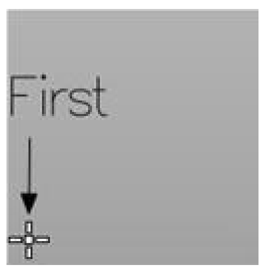
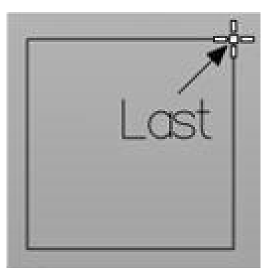
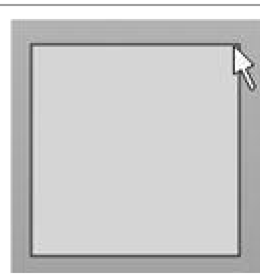
[Create a Surface from 4 Points]
[Create a Parting Line Surface]
[Surface Modeling Ribbon Bar]

5.5.1.1 Rectangular Plane

Create a rectangular plane parallel to the Construction Plane.

Command Prompts

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point.  Note: This point will determine a level (parallel to the Construction Plane), on which the final rectangular plane will be created.
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point.  Note: This point will be projected onto the parallel level of the first point.
Final		

Notes

1. The created plane will be parallel to the Construction Plane and through the first point.
   Coordinate entry can be in absolute coordinates or in relative coordinates.
   An example of absolute coordinate is shown above.

   Example of coordinate entry Input for relative coordinates.

   a) Pick first point or enter coordinates x,y and Z: 2,0

   b) Pick second point or enter coordinates x,y and Z: R6,4

2. For the second point, the coordinate input can also be entered as @6,4
   This creates a rectangular plane that is 6 units in length and 4 units in width from 2,0 to 8,4 in X Y coordinates.

Related Topics

Surfaces Pane, Surface Modeling Ribbon Bar

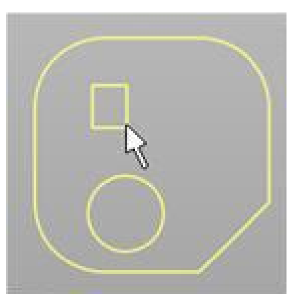
5.5.1.2 Plane from Curves

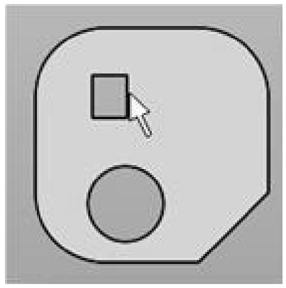
Create a plane from curves (or surface edges) that define a closed region. Internal closed regions may also be selected to create voids or holes in the plane.

Command Prompts

Plane from Curves:: Pick planar curve(s) and hit Right Mouse Button/Enter when done

Basic Procedure

	Screen Pick	Command Input
Step 1		none

Final		

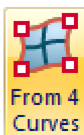
Notes

1. If the curves (or surface edges) are pre-selected, the plane will automatically be created from those curves.

Related Topics

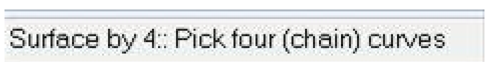
Surfaces Pane, Surface Modeling Ribbon Bar

5.5.1.3 Surface by 4

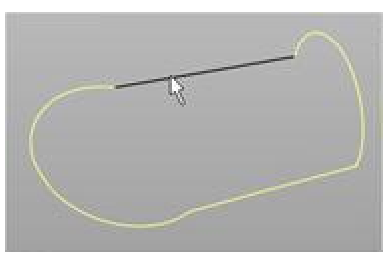


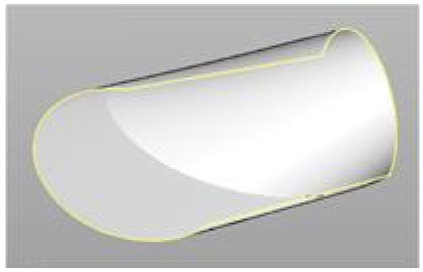
Create a surface from four chain curves (or surface edges) that form a closed region.

Command Prompts



Basic Procedure

	Screen Pick	Command Input
Step 1		none

Final		

Notes

1. The curves (or surface edges) may be pre-selected.
2. The curves (or surface edges) do not need to be co-planar.
3. The endpoints of adjacent curves (or surface edges) must be within the hookup tolerance. See Home > Options > Tolerance and Units > Curve Hookup Tolerances.

Related Topics

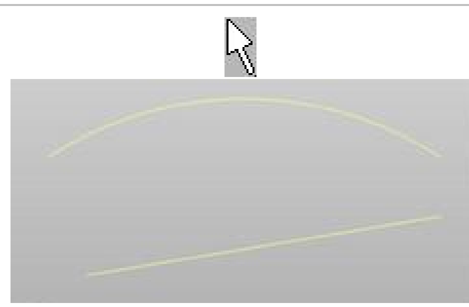
Surfaces Pane, Surface Modeling Ribbon Bar

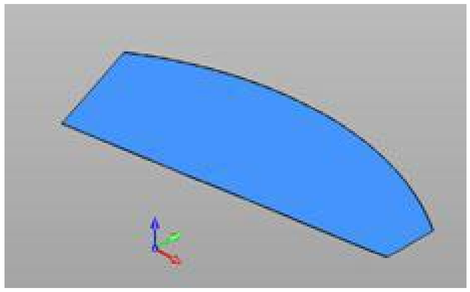
5.5.1.4 Surface by 2

Create a surface between two planar curves (or surface edges).

Command Prompts

Basic Procedure

	Screen Pick	Command Input
Step 1		none

	Pick two planar curves (or surface edges).	
Final		

Related Topics

Surfaces Pane, Surface Modeling Ribbon Bar

5.5.1.5 Surface of Extrusion

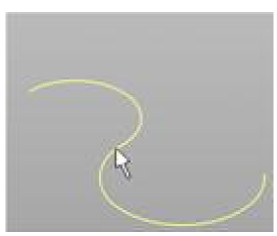
Create a surface by extruding a curve (or surface edge). Multiple curves (or surface edges) may be selected, which will create multiple surfaces.

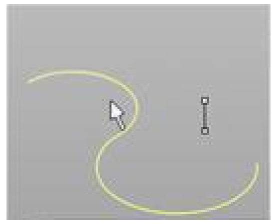
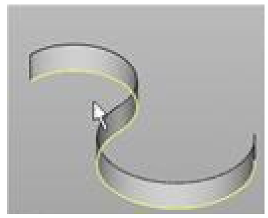
Command Prompts

Surface of Extrusion:: Pick curves to extrude and hit right mouse when done

Surface of Extrusion:: Enter extrusion distance::[L=1.000000]

Basic Procedure

	Screen Pick	Command Input
Step 1	 Hit right mouse button when done.	

Step 2		(Optional) Enter an extrusion distance. 2.25
Final		

Notes

1. The curves (or surface edges) may be pre-selected.

Related Topics

[Surfaces Pane, Surface Modeling Ribbon Bar]()

5.5.1.6 Surface of Revolution

Create a surface by revolving a curve (or surface edge) around an axis. Start and end angles may be entered according to the right-hand rule of positive rotation, as determined by the direction that the axis line is defined.

Command Prompts

Surface of Revolution:: Pick curve to revolve

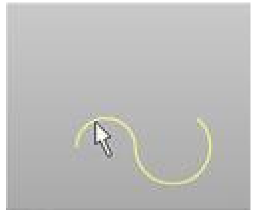
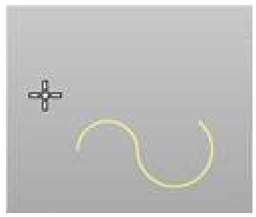
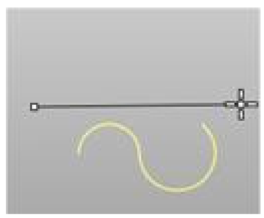
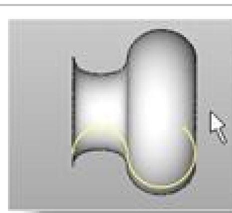
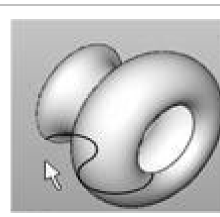
Surface of Revolution:: Pick start point of axis of revolution or enter coordinates x,y and z

Surface of Revolution:: Enter start angle of revolution::[sA=0.000000]

Surface of Revolution:: Enter end angle of revolution::[eA=360.000000]

Basic Procedure

	Screen Pick	Command Input

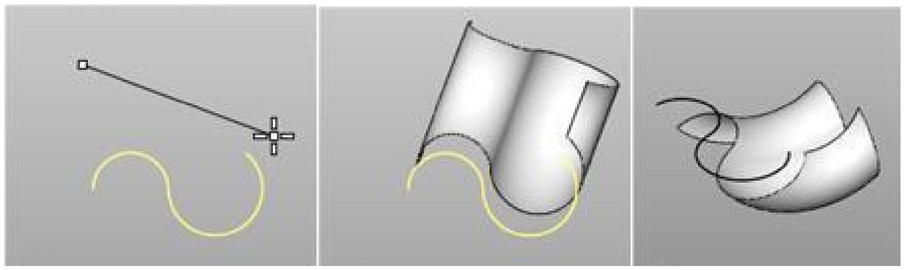
Step 1	 The curves (or surface edges) may be pre-selected.	
Step 2		(Optional) Enter X, Y, [Z] coordinates for the first point (start of axis line). 0,0,0
Step 3		(Optional) Enter X, Y, [Z] coordinates for the second point (end of axis line). 1,0,0
Step 4		Enter the start angle of revolution (according to the right-hand rule of positive rotation). 0
Step 5		Enter the end angle of revolution. 360
Final	 	

Notes

1. The direction of definition of the axis line is important because it determines the right-hand rule of positive rotation for the start and end angles of the revolved

surface.

2. The curve's (or surfaces edge's) existing position defines the zero degree position for angles.
   Here is an example doing a partial rotation from 20 to 100 degrees.



Related Topics

[Surfaces Pane, Surface Modeling Ribbon Bar]

5.5.1.7 Bi-linear Surface from 4 Points

Create a surface by specifying four corner points.

Command Prompts

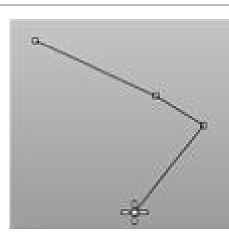
Bilinear Surface from 4 Points:: Pick first corner point or enter coordinates x,y and z

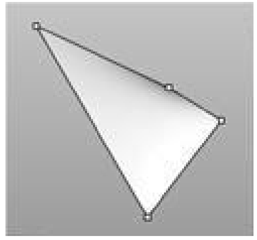
Bilinear Surface from 4 Points:: Pick second corner point or enter coordinates x,y and z

Bilinear Surface from 4 Points:: Pick third corner point or enter coordinates x,y and z

Bilinear Surface from 4 Points:: Pick fourth corner point or enter coordinates x,y and z

Basic Procedure

	Screen Pick	Command Input
Steps 1-4		(Optional) Enter X, Y, [Z] coordinates for any of the points 1-4. 0,1,0

Final		

Notes

1. Pre-selected points are not allowed.

Related Topics

Surfaces Pane, Surface Modeling Ribbon Bar

5.5.1.8 Parting Plane

Create a parting plane positioned relative to a volumetric box around selected surfaces or meshes. VisualCAD creates a volumetric box around the surfaces and/or meshes. The box is oriented to either the Construction Plane of the World Coordinate System (your choice). The parting plane will then be created at the top, middle, or bottom of that volumetric box (your choice).

Command Prompts

Parting Plane:: Pick Objects for command. Hit Right Mouse Button/Enter when done.

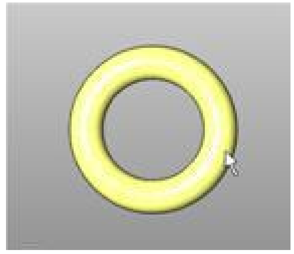
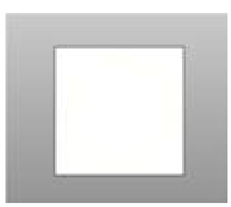
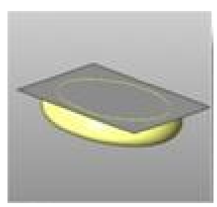
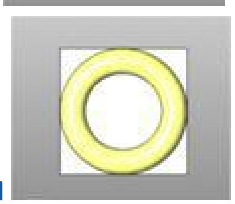
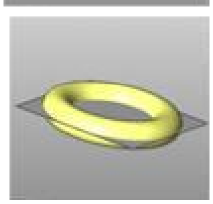
Parting Plane:: Enter Orientation [CPlane=C,World=W]

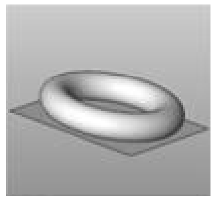
Parting Plane:: Enter offsets in X & Y Direction [X=0.000000,Y=0.000000]

Parting Plane:: Enter Location [Bottom=B,Mid=M,Top=T]

Basic Procedure

	Screen Pick	Command Input

Step 1		
Step 2		Enter 'C' or 'W'. 'C' orients the parting plane parallel to the Construction Plane. 'W' orients the parting plane parallel to the World Coordinate System. 
Step 3		Enter X and Y offsets into the Command Input Bar. These are incremental offsets that will enlarge the parting plane in the X and Y directions respectively. Negative values are not allowed. 
Step 4	T   M  	Enter 'T', 'M', or 'B'. 'T' creates the parting plane at the top of the volumetric box. 'M' creates the parting plane at the middle of the volumetric box. 'B' creates the parting plane at the bottom of the volumetric box. 

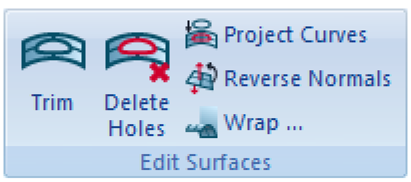
	B  	

Related Topics

[Surfaces Pane, Surface Modeling Ribbon Bar]

5.5.2 Edit Surfaces

The Surface Modeling Ribbon Bar contains the following Edit Surfaces pane with associated functions.



Surface Modeling Ribbon Bar
Edit Surfaces Pane

Related Topics

Trim Surfaces with Curves

Delete Holes from Surfaces

Project Curves onto a Surface

Reverse Surface Normals

Wrap a Surface

5.5.2.1 Trim

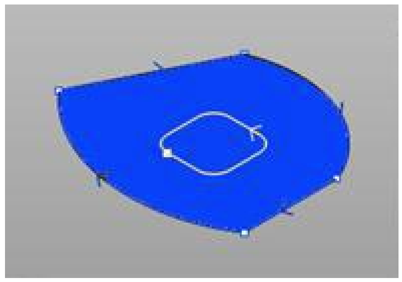
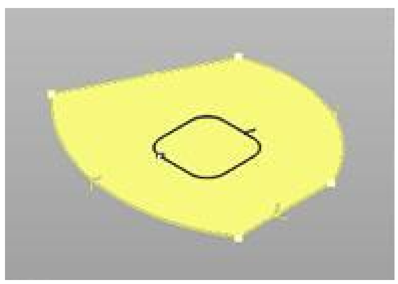
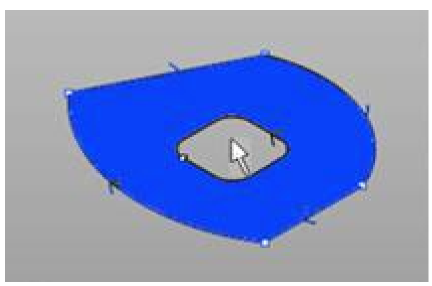
Trim away sections of surfaces by selecting closed curves that bound those sections.

Command Prompts

Trim:: Select cutting objects. Hit Right Mouse Button/Enter when done.

Trim:: Pick an object to be trimmed.

Basic Procedure

	Screen Pick	Command Input
Step 1		None
Step 2		
Final		

Notes

1. When you pick the surface to be trimmed, select the piece of the surface that you want trimmed away. The pick point determines which surface region is trimmed away.

Related Topics

[Edit Surfaces Pane, Surface Modeling Ribbon Bar]

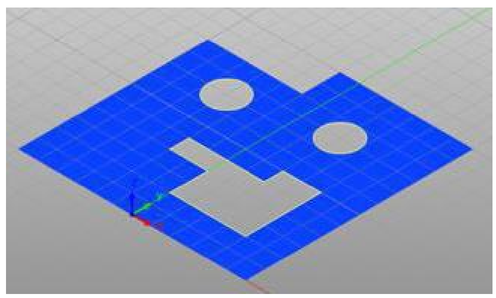
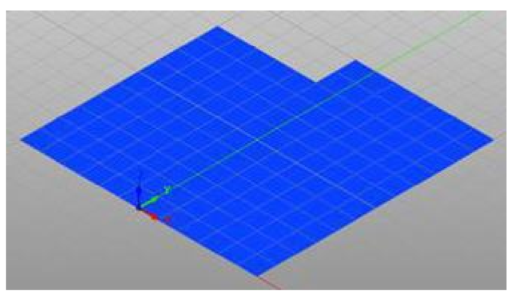
5.5.2.2 Delete Holes

This function will remove holes from a surface.

Command Prompts

Delete Holes:: Pick hole edges to delete. Hit Right Mouse Button/Enter when done.

Basic Procedure

	Screen Pick	Command Input
Step 1	Select the edges of the holes that you want removed. 	None
Step 2	When the selection is complete, hit the Right-Mouse Button or Enter to remove the holes. 	

Notes

1. This function will not remove holes in mesh surfaces.

Related Topics

Edit Surfaces Pane, Surface Modeling Ribbon Bar

5.5.2.3 Project Curves

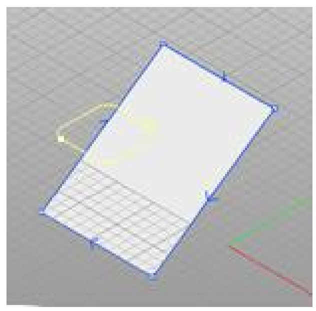
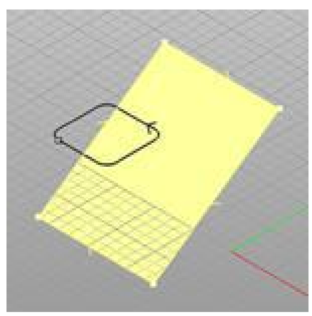
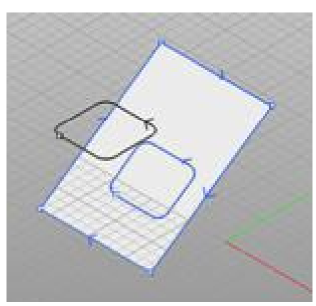
Project curves along the surface normals onto a surface.

Command Prompts

Project Curves:: Pick curves to be projected. Hit Right Mouse Button/Enter when done.

Project Curves:: Pick surface for projection.

Basic Procedure

	Screen Pick	Command Input
Step 1		none
Step 2		
Final		

Notes

1. Meshes are not allowed as the projection surface.

Related Topics

Edit Surfaces Pane, Surface Modeling Ribbon Bar

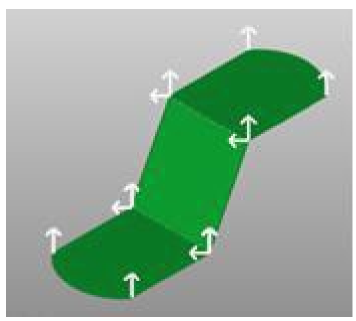
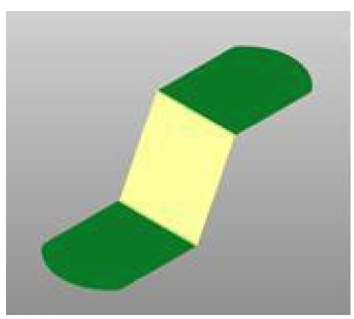
5.5.2.4 Reverse Normals

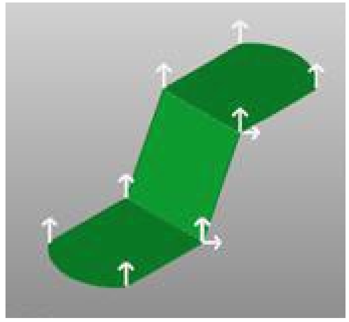
This function will reverse the normal vectors for each surface selected.

Command Prompts

Reverse Normals:: Pick surfaces to reverse and hit Right Mouse Button/Enter when done

Basic Procedure

	Screen Pick	Command Input
Initial		None
Step 1		

Final		

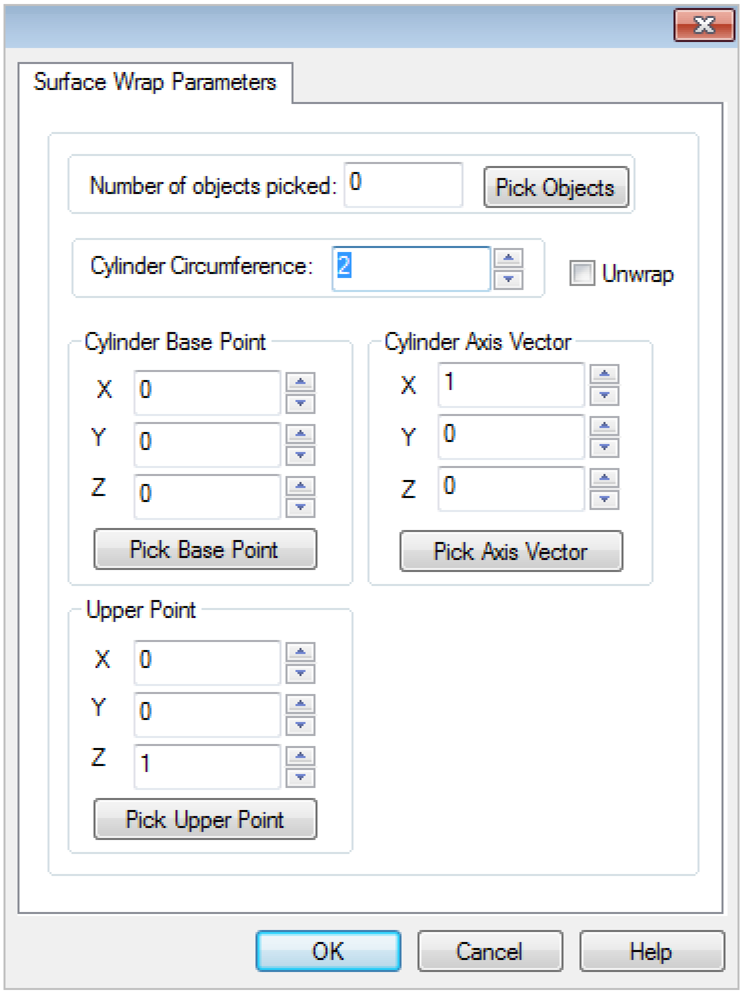
Related Topics

[Edit Surfaces Pane, Surface Modeling Ribbon Bar]()

5.5.2.5 Wrap

Wrap — Wraps a surface onto a construction cylinder whose center line is aligned with one of the principle axes of the CPlane.

Dialog Box: Edit Surface Wrap



Dialog Box: Edit Surface Wrap

- Number of objects picked
  The number of curves selected is displayed. Select the Pick Curves button to add more curves to the selection.
- Cylinder Circumference
  Enter a circumference for the wrapped cylinder.
- Unwrap
  Check this box to unwrap the selected surface.
- Cylinder Base Point
  Enter the Cylinder Base Point XYZ coordinates or select the Pick Base Point button and then select a point to locate the base of the cylinder.
- Cylinder Axis Vector
  Enter the Cylinder Axis Vector XYZ coordinates or select the Pick Axis Vector button and then select a start point and end point to define the axis.

- **Upper point**
  Enter the Cylinder Upper Point XYZ coordinates or select the Pick Upper Point button and then select a point to locate the upper point of the cylinder.

Basic Procedure

	Dialog Interaction
Step 1	Select the Pick Objects button and pick the surface to be wrapped. The su may be pre-selected.
Step 2	Specify the cylinder circumference.
Step 3	Select one of the C-Plane axes that is used as the construction cylinder's a
Step 4	Set the cylinder base point
Step 5	Set the cylinder upper point
Step 6	Hit the OK button

Notes

1. The surface to be wrapped must be co-planar and parallel to the C-Plane.
2. The surface will be wrapped in a positive direction around the construction cylinder according to the right-hand rule (counter-clockwise).

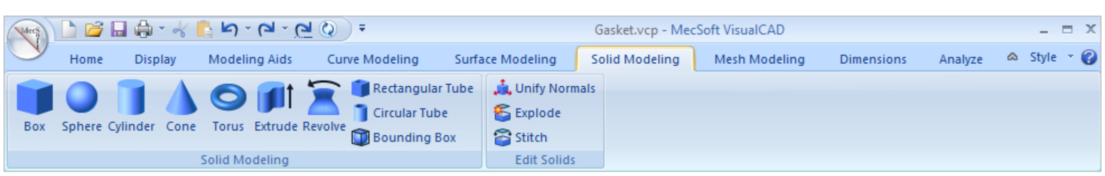
Related Topics

Edit Surfaces Pane, Surface Modeling Ribbon Bar

5.6 Solid Modeling

The Solid Modeling Ribbon Bar contains Solid Modeling and Edit Solids panes and associated functions.

The Solid Modeling Ribbon Bar



Solid Modeling Ribbon Bar

Related Topics

Solid Modeling Pane, Solid Modeling Ribbon Bar

Edit Solids Pane, Solid Modeling Ribbon Bar

The Ribbon Bar

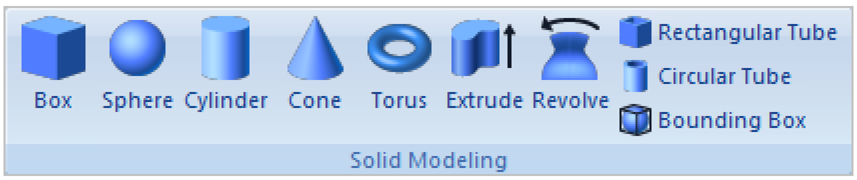
Customize Quick Access Toolbar

Customize Dialog

VisualCAD Options

5.6.1 Solid Modeling

The Solid Modeling Ribbon Bar contains the following Solid Modeling pane with associated functions. See What is a Solid? for information about Meshes and Solids in VisualCAD.



Surface Modeling Ribbon Bar
Solid Modeling Pane

Related Topics

What is a Solid?

Create a Solid Box

Create a Solid Sphere

Create a Solid Cylinder

Create a Solid Cone

Create a Solid Torus

Create an Extruded Solid

Create a Revolved Solid

Create a Solid Rectangular Tube

Create a Solid Circular Tube

Create a Solid Bounding Box

Solid Modeling Ribbon Bar

5.6.1.1 Solid Box

Create a Mesh Solid in the shape of a rectangular box. See What is a Solid? for information about Meshes and Solids in VisualCAD.

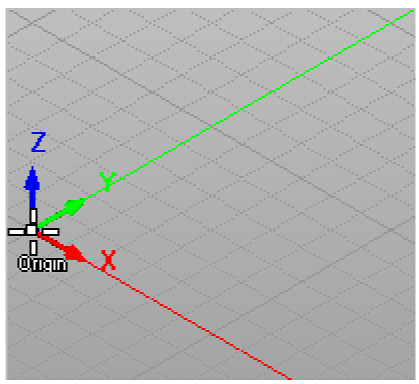
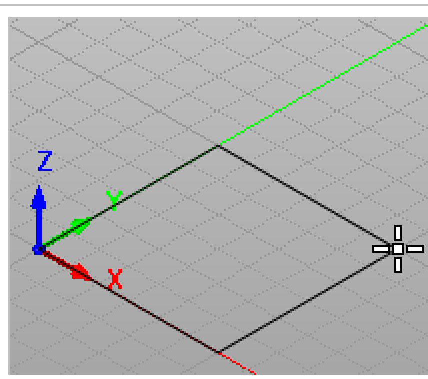
Command Prompts

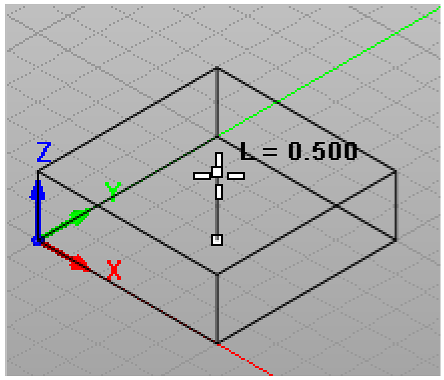
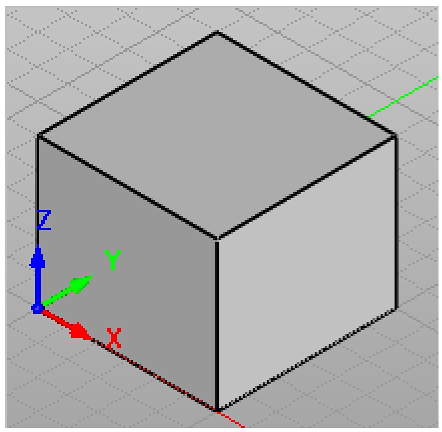
Box:: Pick first corner point or enter coordinates x,y and z::[0.000,0.000,0.000]

Box:: Pick second corner point or enter coordinates x,y and z

Box:: Enter extrusion distance::[L=1.000000]

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point. 0,0,0
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point (opposite corner 1,1,0

Step 3		(Optional) Enter X, Y, [Z] coordinates for the extrusion Length.  or... (Optional) Enter the Length. 
Final		

Notes

1. The bottom of the box will lie in a base plane (parallel to the Construction Plane of the active view) with its corner at the first point defined.
2. If the second point does not lie on the base plane, it will be projected onto that base plane.

Related Topics

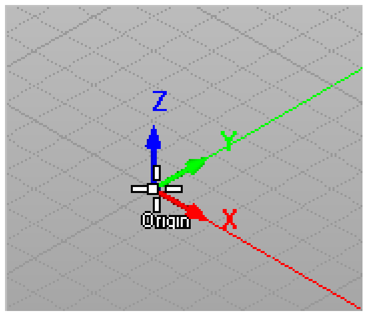
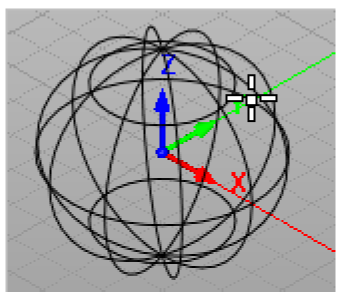
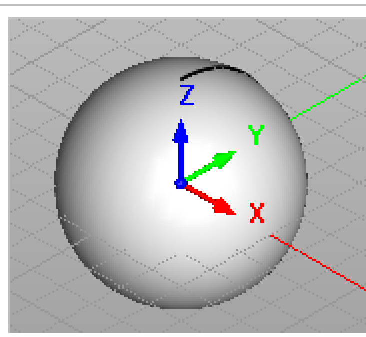
Solid Modeling Pane, Solid Modeling Ribbon Bar

5.6.1.2 Solid Sphere

Create a solid in the shape of a sphere. See What is a Solid? for information about Meshes and Solids in VisualCAD.

Command Prompts

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point (center). 
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point (radius point).  or... (Optional) Enter the radius. 
Final		

Notes

1. In Step 2, if a single number is entered, it will be interpreted as the radius. If 2 or 3 numbers are entered, they will be interpreted as the coordinates of the radius point.

Related Topics

[Solid Modeling Pane, Solid Modeling Ribbon Bar]

5.6.1.3 Solid Cylinder

Create a solid in the shape of a cylinder. See What is a Solid? for information about Meshes and Solids in VisualCAD.

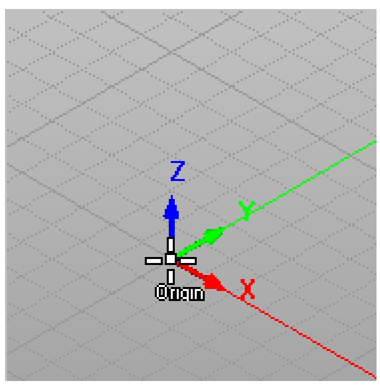
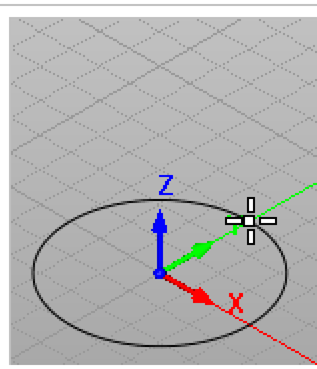
Command Prompts

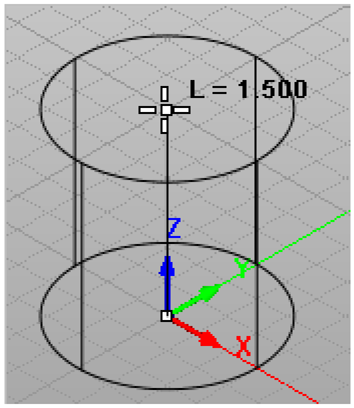
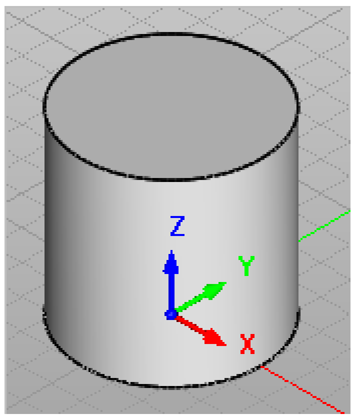
Cylinder:: Pick center point or enter coordinates x,y and z::[0.000,0.000,0.000]

Cylinder:: Enter radius or pick radius point::[R=1.000000]

Cylinder:: Enter extrusion distance::[L=1.000000]

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point (center). 0,0,0
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point (radius point). 1,1,0 or... (Optional) Enter the radius. 2.25

Step 3		(Optional) Enter X, Y, Z coordinates for the third point (height). 1,1,1 or... (Optional) Enter the height. 4.5
Final		

Notes

1. The axis of the cylinder will be normal (perpendicular) to the Construction Plane.

Related Topics

Solid Modeling Pane, Solid Modeling Ribbon Bar

5.6.1.4 Solid Cone

Create a mesh that is the shape of a cone. See What is a Solid? for information about Meshes and Solids in VisualCAD.

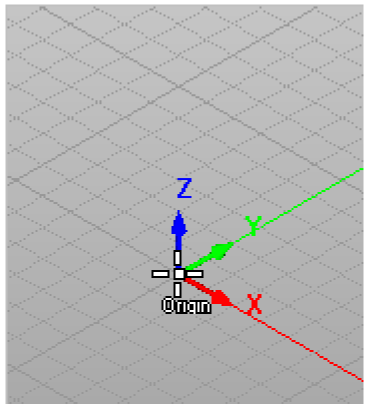
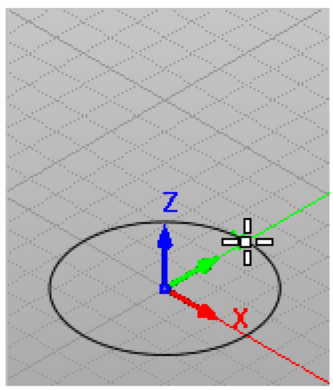
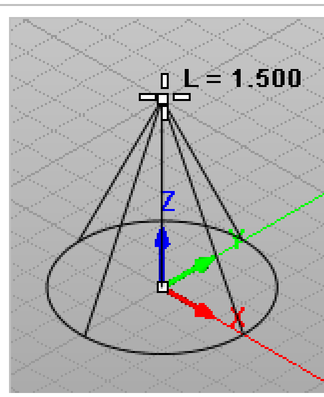
Command Prompts

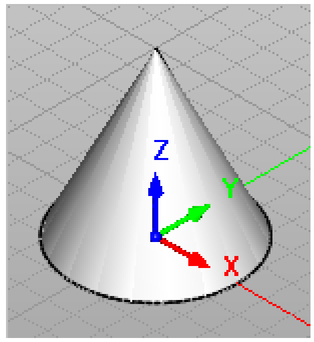
Cone:: Pick center point or enter coordinates x,y and z::[0.000,0.000,0.000]

Cone:: Enter radius or pick radius point::[R=1.000000]

Cone:: Enter extrusion distance::[L=1.000000]

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point (center). 0,0,0
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point (radius point). 1,1,0 or... (Optional) Enter the radius. 2.25
Step 3		(Optional) Enter X, Y, Z coordinates for the third point (height). 1,1,1 or... (Optional) Enter the height. 4.5

Final		

Notes

1. The base plane for the cone will be parallel to the Construction Plane of the active view when the second point (radius point) is defined.
2. The axis of the cone will be normal (perpendicular) to the base plane.
3. The height of the cone is measured from the base along the axis of the cone.

Related Topics

[Solid Modeling Pane, Solid Modeling Ribbon Bar]

5.6.1.5 Solid Torus

Create a mesh that is the shape of a ring torus. See What is a Solid? for information about Meshes and Solids in VisualCAD.

Command Prompts

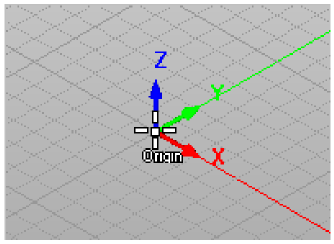
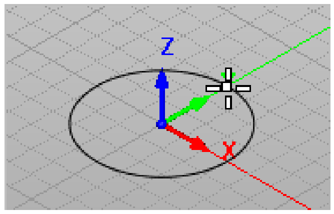
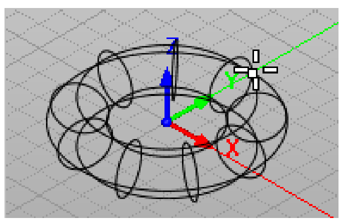
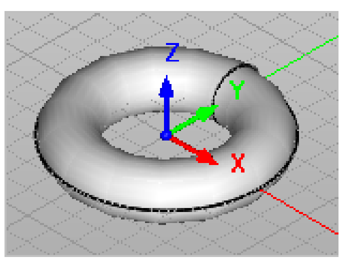
Torus:: Pick center point or enter coordinates x,y and z::[0.000,0.000,0.000]

Torus:: Enter radius or pick radius point::[R=1.000000]

Torus:: Enter extrusion distance::[L=1.000000]

Basic Procedure

	Screen Pick	Command Input

Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point (hole center). 0,0,0
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point (center-line radius point). 1,1,0 or... (Optional) Enter the radius. 2.25
Step 3		(Optional) Enter the radius of the tube section (tube radius or outer diameter point). 2.25 or... (Optional) Enter X, Y, [Z] coordinates for the third point on the surface (tube radius or outer diameter point). 1,1,0
Final		

Notes

1. The plane of the circular center line of the torus will be parallel to the Construction Plane of the active view when the second point (center line radius point) is defined.

Related Topics

Solid Modeling Pane, Solid Modeling Ribbon Bar

5.6.1.6 Solid Extrusion

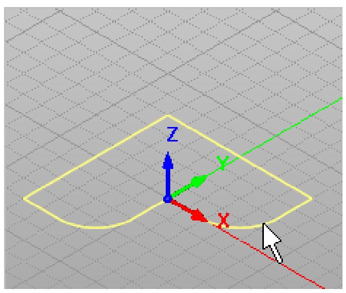
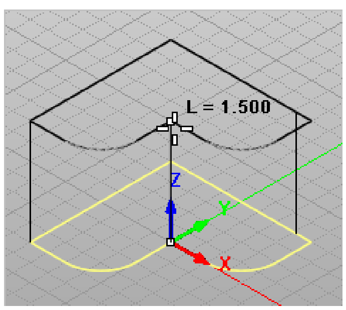
Create a solid by extruding a closed curve (or surface edge) set. See What is a Solid? for information about Meshes and Solids in VisualCAD.

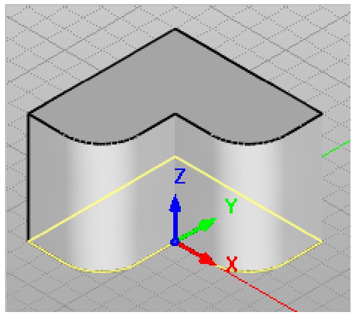
Command Prompts

Extrude:: Pick curves to extrude and hit right mouse when done

Extrude:: Enter extrusion distance::[L=1.000000]

Basic Procedure

	Screen Pick	Command Input
Step 1		none
Step 2		(Optional) Enter the extrusion distance. 2.25 or.. (Optional) Enter X, Y, Z coordinates for the extrusion distance point. 1,1,1

Step Final		

Notes

1. The curve (or surface edge) set may be pre-selected.
2. If the curve (or surface edge) set is not coplanar, each curve (or surface edge) is projected onto the Construction Plane.
3. The extrusion is performed normal to the Construction Plane.

Related Topics

Solid Modeling Pane, Solid Modeling Ribbon Bar

5.6.1.7 Solid Revolve

Create a solid by revolving a closed curve (or surface edge) around an axis. See What is a Solid? for information about Meshes and Solids in VisualCAD.

Command Prompts

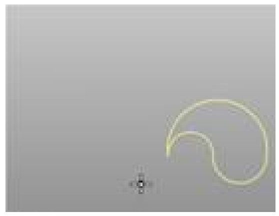
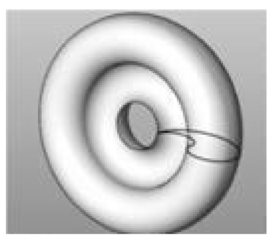
Revolve:: Pick curve to revolve

Revolve:: Pick start point of axis of revolution or enter coordinates x,y and z

Revolve:: Pick end point of axis of revolution or enter coordinates x,y and z

Basic Procedure

	Screen Pick	Command Input

Step 1		none
Step 2		
Step 3		
Final		

Related Topics

Solid Modeling Pane, Solid Modeling Ribbon Bar

5.6.1.8 Solid Rectangular Tube

Create a solid that is the shape of a rectangular tube with constant wall thickness. See What is a Solid? for information about Meshes and Solids in VisualCAD.

Command Prompts

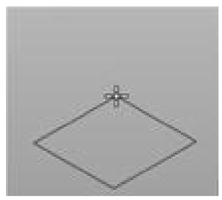
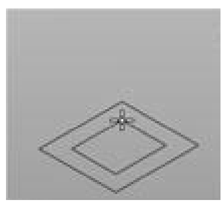
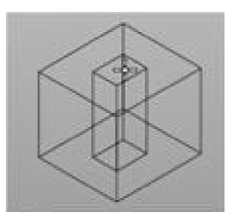
Rectangular Tube:: Pick first corner point or enter coordinates x,y and z::[0.000,0.000,0.000]

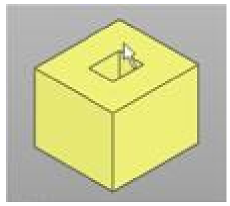
Rectangular Tube:: Pick second corner point or enter coordinates x,y and z

Rectangular Tube:: Enter offset distance::[O=0.000000]

Rectangular Tube:: Enter extrusion distance::[L=1.000000]

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point. 0,0,0
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point (opposite corner). 1,1,0
Step 3		(Optional) Enter X, Y, Z coordinates for the third point (wall thickness). .25,.25,0 or... (Optional) Enter the wall thickness. .5
Step 4		(Optional) Enter X, Y, Z coordinates for the fourth point (height). 1,1,1 or... (Optional) Enter the extrusion distance. 4.5

Final		

Related Topics

Solid Modeling Pane, Solid Modeling Ribbon Bar

5.6.1.9 Solid Circular Tube

Create a solid that is the shape of a circular tube (pipe). See What is a Solid? for information about Meshes and Solids in VisualCAD.

Command Prompts

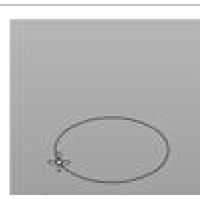
Circular Tube:: Pick center point or enter coordinates x,y and z::[0.000,0.000,0.000]

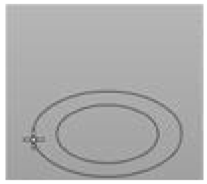
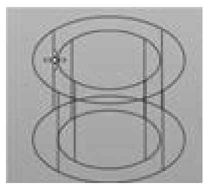
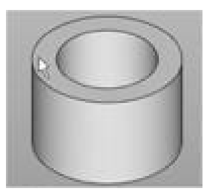
Circular Tube:: Enter radius or pick radius point::[R=1.000000]

Circular Tube:: Enter radius or pick radius point::[R=1.250000]

Circular Tube:: Enter extrusion distance::[L=1.000000]

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point (hole center). 0,0,0
Step 2		(Optional) Enter the first radius of the tube. 2.25 or... (Optional) Enter X, Y, [Z] coordinates for the second point (first radius point).

		1,0,0
Step 3		(Optional) Enter the other radius of the tube. .5 or... (Optional) Enter X, Y, [Z] coordinates for the third point (other radius point). 1.5,0,0
Step 4		(Optional) Enter X, Y, Z coordinates for the fourth point (height). 1,1,1 or... (Optional) Enter the extrusion distance. 4.5
Final		

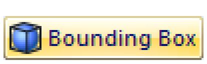
Notes

1. The base plane for the circular tube will be parallel to the Construction Plane of the active view when the second point (radius point) is defined.
2. The axis of the circular tube will be normal (perpendicular) to the base plane.

Related Topics

Solid Modeling Pane, Solid Modeling Ribbon Bar

5.6.1.10 Solid Bounding Box



Create a rectangular box solid that encompasses selected geometry objects. See What is a Solid? for information about Meshes and Solids in VisualCAD.

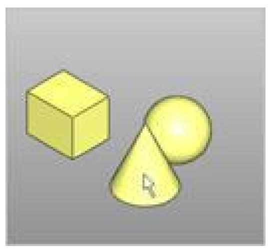
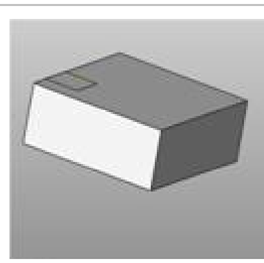
Command Prompts

Bounding Box:: Pick Objects for command. Hit Right Mouse Button/Enter when done.

Bounding Box:: Enter Orientation [CPlane=C,World=W]

Bounding Box:: Enter offsets in X & Y Direction [X=0.000000,Y=0.000000,Z=0.000000]

Basic Procedure

	Screen Pick	Command Input
Step 1	Select all objects to be included in the volumetric box. Hit Right Mouse Button or ENTER to end the selection. 	
Step 2		Enter 'C' or 'W'. 'C' orients the volumetric box to the Construction Plane. 'W' orients the volumetric box to the World Coordinate System.
Step 3		Enter X, Y and Z offsets. These are incremental offsets that will enlarge the volumetric box in the X, Y and Z directions respectively. Negative values are not allowed.
Final		

Notes

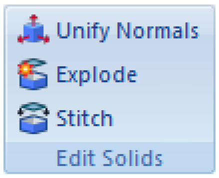
1. All geometry types are permitted.

Related Topics

Solid Modeling Pane, Solid Modeling Ribbon Bar

5.6.2 Edit Solids

The Solid Modeling Ribbon Bar contains the following Edit Solids pane with associated functions. See What is a Solid? for information about Meshes and Solids in VisualCAD.



Related Topics

What is a Solid?

Unify Surface Normals

Explode Surfaces

Stitch Surfaces into a Solid

Solid Modeling Ribbon Bar

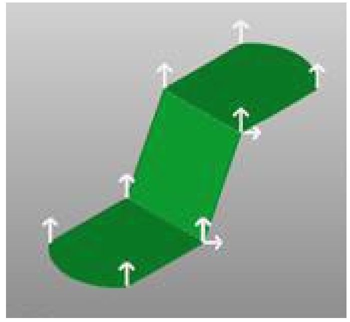
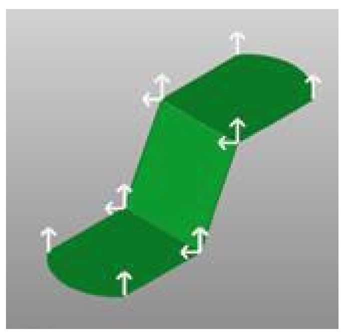
5.6.2.1 Unify Normals

For a set of surfaces that are adjacent and share edges, this function will make all surface normals consistent in direction, relative to the other surfaces.

Command Prompts

Basic Procedure

	Screen Pick	Command Input

Initial		none
Step 1		
Final		

Related Topics

Edit Solids Pane, Solid Modeling Ribbon Bar

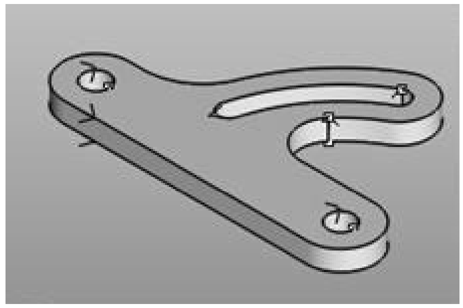
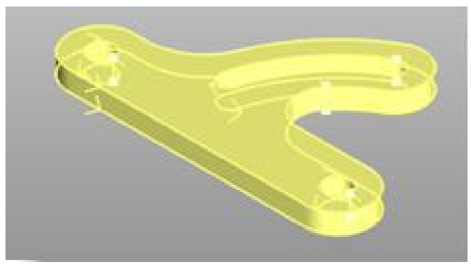
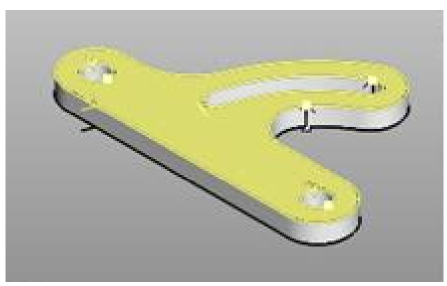
5.6.2.2 Explode

This function will separate all the surfaces of a shell or poly-surface into independent surfaces with no topological relationships to each other.

Command Prompts

Explode:: Pick surfaces to explode. Hit Right Mouse Button/Enter when done.

Basic Procedure

	Screen Pick	Command Input
Initial		none
Step 1	Select poly-surface part. 	
Final	After Explode, each surface is independent from others. 	

Notes

1. Meshes are not allowed for the Explode function.

Related Topics

Edit Solids Pane, Solid Modeling Ribbon Bar

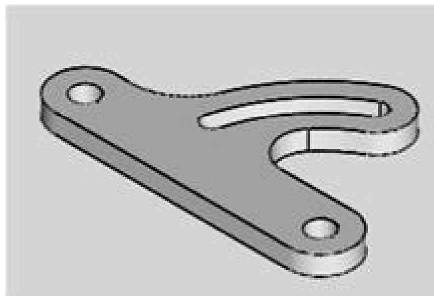
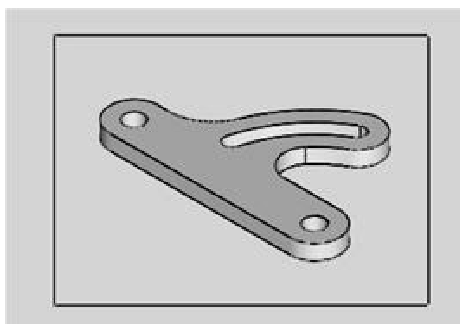
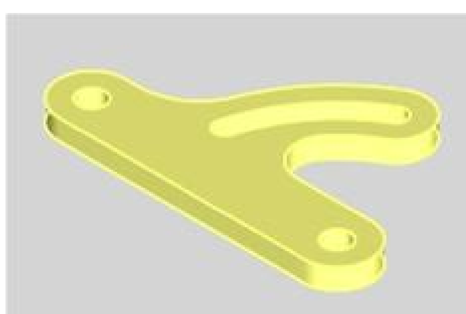
5.6.2.3 Stitch

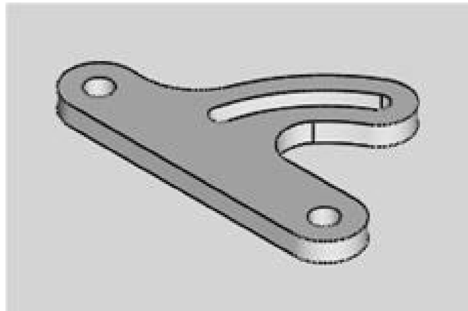
This function will join selected surfaces to form a shell or poly-surface.

Command Prompts

Stitch:: Pick surfaces to Stitch. Hit Right Mouse Button/Enter when done.

Basic Procedure

	Screen Pick	Command Input
Initial		none
Step 1	Select surface(s) part. You can either use single select and pick individual surfaces or draw a window to select all surfaces. 	
	Selected surface(s) are now highlighted. 	
Final	After Stitch, selected surfaces are merged to shell or poly-surface.	

	 .	

Notes

1. Meshes are not allowed for the Stitch function.

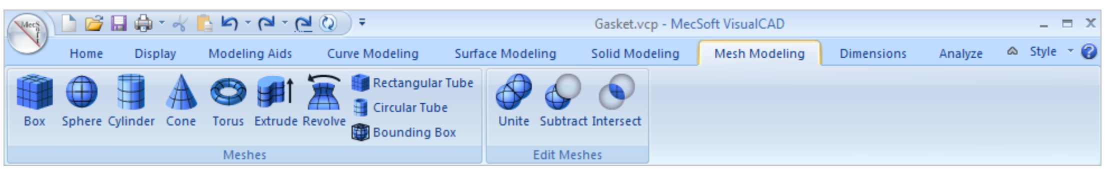
Related Topics

[Edit Solids Pane, Solid Modeling Ribbon Bar]

5.7 Mesh Modeling

The Mesh Modeling Ribbon Bar contains Meshes and Edit Meshes panes and associated functions.

The Mesh Modeling Ribbon Bar



Mesh Modeling Ribbon Bar

Related Topics

Meshes Pane, Mesh Modeling Ribbon Bar

Edit Meshes Pane, Mesh Modeling Ribbon Bar

The Ribbon Bar

Customize Quick Access Toolbar

Customize Dialog

VisualCAD Options

5.7.1 Meshes

The Mesh Modeling Ribbon Bar contains the following Meshes pane with associated functions.

Mesh Modeling Ribbon Bar
Meshes Pane

Related Topics

[Create a Mesh Box]

[Create a Mesh Sphere]

[Create a Mesh Cylinder]

[Create a Mesh Cone]

[Create a Mesh Torus]

[Create a Rectangular Tube Mesh]

[Create a Mesh Circular Tube]

[Create Extruded Mesh]

[Create a Revolved Mesh]

[Create a Mesh Bounding Box]

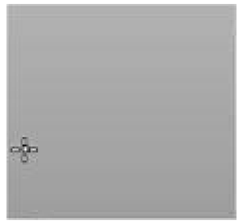
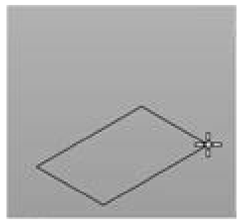
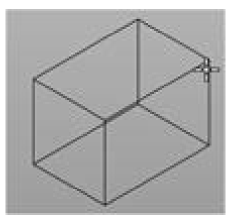
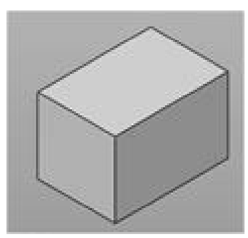
[Mesh Modeling Ribbon Bar]

5.7.1.1 Box

Create a mesh that is the shape of a rectangular box. The bottom of the box will lie in a base plane (parallel to the Construction Plane of the selecting view) with its corner at the first point defined. If the second point does not lie on the base plane, it will be projected onto that base plane.

Basic Procedure

	Screen Pick	Command Input
Step 1	Pick the first corner of the base rectangle.	(Optional) Enter X, Y, Z coordinates for the first point. Example: 0,0,0

		
Step 2	Pick the diagonal corner of the base rectangle. 	(Optional) Enter X, Y, Z coordinates for the second point (opposite corner). Example: 1,1,0
Step 3	Pick a point to define the height of the box. 	(Optional) Enter X, Y, Z coordinates for the third point (height). Example: 1,1,1 *or...* (Optional) Enter the height. Example: 4.5
Final		

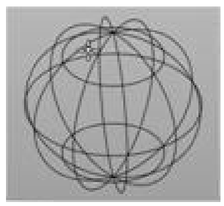
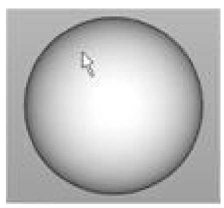
Related Topics

Meshes Pane, Mesh Modeling Ribbon Bar

5.7.1.2 Sphere

Create a mesh that is the shape of a sphere. In Step 2 (see below), if a single number is entered, it will be interpreted as the radius. If 2 or 3 numbers are entered, they will be interpreted as the coordinates of the radius point.

Basic Procedure

	Screen Pick	Command Input
Step 1	Pick the center point for the sphere. 	(Optional) Enter X, Y, Z coordinates for the first point (center). Example: 0,0,0
Step 2	Pick the radius point for the sphere. 	(Optional) Enter X, Y, Z coordinates for the second point (radius point). Example: 1,1,0 or... (Optional) Enter the radius. Example: 2.25
Final		

Related Topics

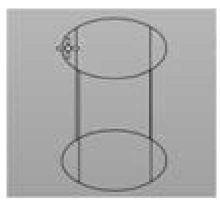
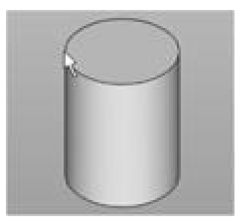
Meshes Pane, Mesh Modeling Ribbon Bar

5.7.1.3 Cylinder

Create a mesh that is the shape of a cylinder. The axis of the cylinder will be normal (perpendicular) to the Construction Plane.

Basic Procedure

	Screen Pick	Command Input
Step 1	Pick the center point of the base circle.	(Optional) Enter X, Y, [Z] coordinates for the first point (center). Example: 0,0,0

		
Step 2	Pick the radius point of the base circle. 	(Optional) Enter X, Y, [Z] coordinates for the second point (radius point). Example: 1,1,0 or... (Optional) Enter the radius. Example: 2.25
Step 3	Pick the length point of the cylinder. 	(Optional) Enter X, Y, [Z] coordinates for the third point (height). Example: 1,1,1 or... (Optional) Enter the height. Example: 4.5
Final		

Related Topics

Meshes Pane, Mesh Modeling Ribbon Bar

5.7.1.4 Cone

Create a mesh that is the shape of a cone. The base plane for the cone will be parallel to the Construction Plane of the active view when the second point (radius point) is defined. The axis of the cone will be normal (perpendicular) to the base plane. The height of the cone is measured from the base along the axis of the cone.

Basic Procedure

	Screen Pick	Command Input
Step 1	Pick the center point of the base circle. 	(Optional) Enter X, Y, Z coordinates for the first point (center). Example: 0,0,0
Step 2	Pick the radius point of the base circle. 	(Optional) Enter X, Y, Z coordinates for the second point (radius point). Example: 1,1,0 or... (Optional) Enter the radius. Example: 2.25
Step 3	Pick the length point of the cone. 	(Optional) Enter X, Y, Z coordinates for the third point (height). Example: 1,1,1 or... (Optional) Enter the height. Example: 4.5
Final		

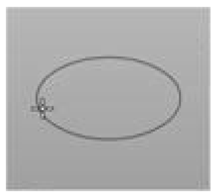
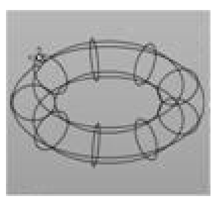
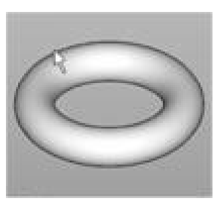
Related Topics

Meshes Pane, Mesh Modeling Ribbon Bar

5.7.1.5 Torus

Create a mesh that is the shape of a ring torus. The plane of the circular center line of the torus will be parallel to the Construction Plane of the active view when the second point (center line radius point) is defined.

Basic Procedure

	Screen Pick	Command Input
Step 1	Pick the center point of the torus. 	(Optional) Enter X, Y, Z coordinates for the first point (hole center). Example: 0,0,0
Step 2	Pick the radius point of the torus. 	(Optional) Enter X, Y, Z coordinates for the second point (center line radius point). Example: 1,1,0 or... (Optional) Enter the radius. Example: 2.25
Step 3	Pick the second radius (i.e., cross-section radius) point of the torus. 	(Optional) Enter the radius of the tube section. Example: 2.25 or... (Optional) Enter X, Y, Z coordinates for the third point on the surface (tube radius point). Example: 1,1,0
Final		

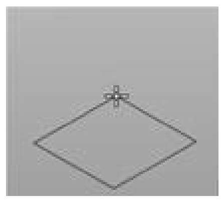
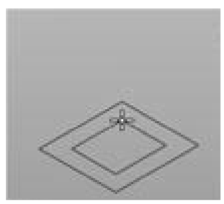
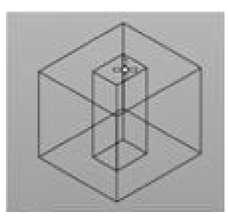
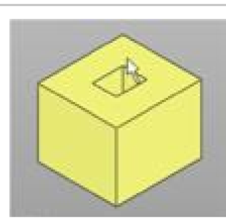
Related Topics

[Meshes Pane, Mesh Modeling Ribbon Bar]

5.7.1.6 Rectangular Tube

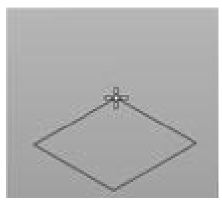
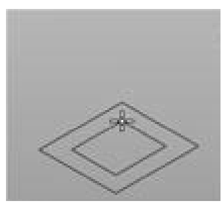
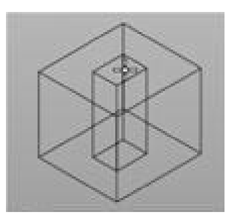
Create a mesh that is the shape of a rectangular tube with constant wall thickness.

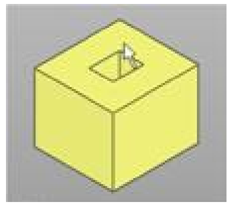
Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point. 0,0,0
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point (opposite corner). 1,1,0
Step 3		(Optional) Enter X, Y, [Z] coordinates for the third point (wall thickness). .25,.25,0 or... (Optional) Enter the wall thickness. .5
Step 4		(Optional) Enter X, Y, [Z] coordinates for the fourth point (height). 1,1,1 or... (Optional) Enter the extrusion distance. 4.5
Final		

Create a mesh that is the shape of a rectangular tube with constant wall thickness.

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point. 0,0,0
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point (opposite corner). 1,1,0
Step 3		(Optional) Enter X, Y, [Z] coordinates for the third point (wall thickness). .25,.25,0 or... (Optional) Enter the wall thickness. .5
Step 4		(Optional) Enter X, Y, [Z] coordinates for the fourth point (height). 1,1,1 or... (Optional) Enter the extrusion distance. 4.5

Final		

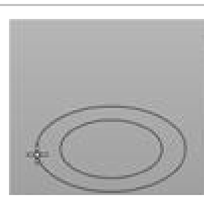
Related Topics

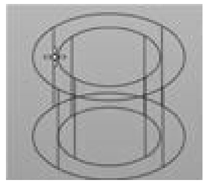
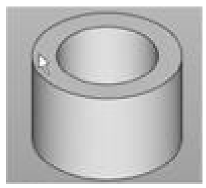
[Meshes Pane, Mesh Modeling Ribbon Bar]

5.7.1.7 Circular Tube

Circular Tube Create a mesh that is the shape of a circular tube (pipe).

Basic Procedure

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point (hole center). 0,0,0
Step 2		(Optional) Enter the first radius of the tube. 2.25 or... (Optional) Enter X, Y, [Z] coordinates for the second point (first radius point). 1,0,0
Step 3		(Optional) Enter the other radius of the tube. .5 or... (Optional) Enter X, Y, [Z] coordinates for the third point (other radius point).

		1.5,0,0
Step 4		(Optional) Enter X, Y, [Z] coordinates for the fourth point (height). 1,1,1 or... (Optional) Enter the extrusion distance. 4.5
Final		

Notes

1. The base plane for the circular tube will be parallel to the Construction Plane of the active view when the second point (radius point) is defined.
2. The axis of the circular tube will be normal (perpendicular) to the base plane.

Related Topics

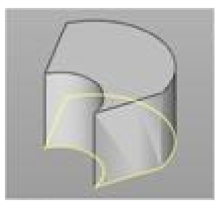
Meshes Pane, Mesh Modeling Ribbon Bar

5.7.1.8 Extrude

Create a mesh by extruding a closed curve (or surface edge) set.

Basic Procedure

	Screen Pick	Command Input
Step 1	Pick curves (or surface edges) to extrude and hit right mouse when done	

		
Step 2		(Optional) Enter the extrusion distance.  or.. (Optional) Enter X, Y, [Z] coordinates for the extrusion distance point. 
Final		

Notes

1. The curves (or surface edges) may be pre-selected.
2. If the curves (or surface edges) are not coplanar, they are projected onto the Construction Plane.
3. The extrusion is performed normal to the Construction Plane.

Related Topics

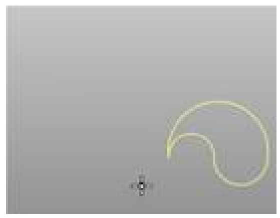
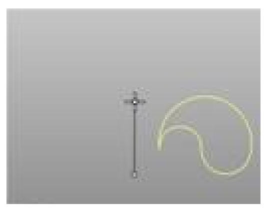
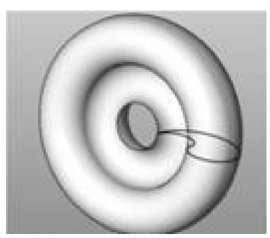
Meshes Pane, Mesh Modeling Ribbon Bar

5.7.1.9 Revolve

Create a mesh by revolving a closed curve (or surface edge) around an axis.

Basic Procedure

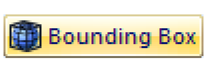
	Screen Pick	Command Input

Step 1	Pick curve to revolve 	none
Step 2	Pick start point of axis of revolution or enter coordinates x,y and z 	
Step 3	Pick end point of axis of revolution or enter coordinates x,y and z 	
Final		

Related Topics

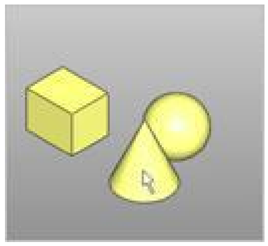
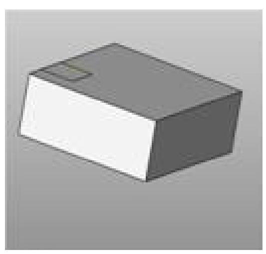
[Meshes Pane, Mesh Modeling Ribbon Bar]()

5.7.1.10 Bounding Box



Create a rectangular box mesh that encompasses selected geometry objects.

Basic Procedure

	Screen Pick	Command Input
Step 1	Select all objects to be included in the volumetric box. Hit Right Mouse Button or ENTER to end the selection. 	
Step 2		Enter 'C' or 'W'. 'C' orients the volumetric box to the Construction Plane. 'W' orients the volumetric box to the World Coordinate System.
Step 3		Enter X, Y and Z offsets. These are incremental offsets that will enlarge the volumetric box in the X, Y and Z directions respectively. Negative values are not allowed.
Final		

Notes

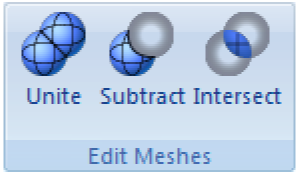
1. All geometry types are permitted.

Related Topics

Meshes Pane, Mesh Modeling Ribbon Bar

5.7.2 Edit Meshes

The Mesh Modeling Ribbon Bar contains the following Edit Meshes pane with associated functions.



Related Topics

[Unite Two Mesh Solid Bodies]

[Subtract one Mesh Solid Body from another]

[Intersect 2 Mesh Solids]

[Mesh Modeling Ribbon Bar]

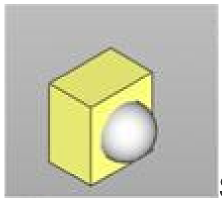
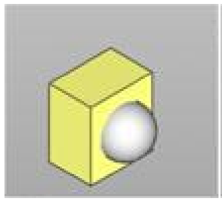
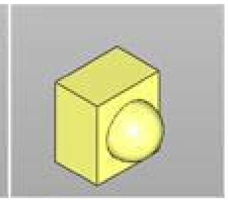
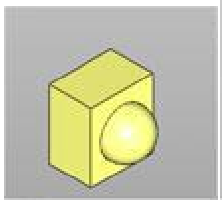
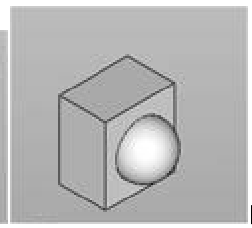
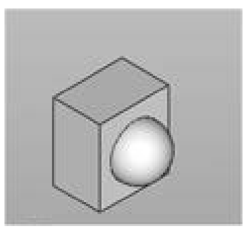
5.7.2.1 Unite

Unite two mesh objects into one mesh object. The first mesh object may be pre-selected. Unite two or more mesh objects into one mesh object. The first set of mesh objects may be pre-selected. A Mesh Revolve that is open-ended (not a full 360 degrees) cannot be used in this function.

> **Meshes Must be Closed**: If any selected mesh does not form closed volume (i.e., there are any open edges) the Mesh Boolean operations (i.e., Unite, Subtract or Intersect) will fail. This can sometimes occur with imported data files. If this occurs, use the Auto Fix command on each mesh before using the Unit, Subtract or Intersect Mesh commands. You can also use the Diagnose command to check for mesh errors.

Basic Procedure

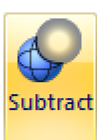
	Screen Pick	Command Input
Step 1	Pick the first mesh to unite.	none

	Select meshes to unite. 	
Step 2	Pick the second mesh solid. The Unite is performed automatically. Right-click or press <Enter> to perform the unite. 	

Related Topics

[Edit Meshes Pane, Mesh Modeling Ribbon Bar]

5.7.2.2 Subtract

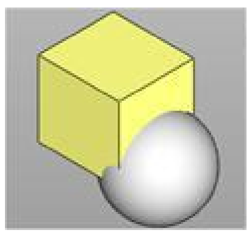
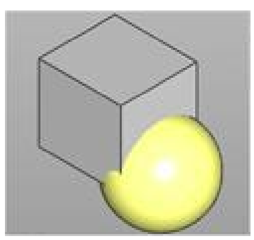
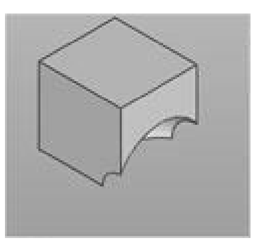


Subtract one mesh volume from another. The two mesh objects must intersect. The first mesh may be pre-selected. The second mesh will be deleted after the subtraction.Subtract one or more mesh volumes from another. At least one mesh in each selection must intersect. The first set of meshes may be pre-selected. The second set of meshes will be deleted after the subtraction. A Mesh Revolve that is open-ended (not a full 360 degrees) cannot be used in this Subtract function.

> **Meshes Must be Closed**: If any selected mesh does not form closed volume (i.e., there are any open edges) the Mesh

Boolean operations (i.e., Unite, Subtract or Intersect) will fail. This can sometimes occur with imported data files. If this occurs, use the Auto Fix command on each mesh before using the Unit, Subtract or Intersect Mesh commands. You can also use the Diagnose command to check for mesh errors.

Basic Procedure

	Screen Pick	Command Input
Step 1	Select the mesh object to subtract from.Select one or more mesh objects to subtract from. Right-click or press <Enter> when done. 	none
Step 2	Select the mesh object to subtract.Select one or more mesh objects to subtract. Right-click or press <Enter> when done. 	
Final	The Subtract is performed automatically. 	

Related Topics

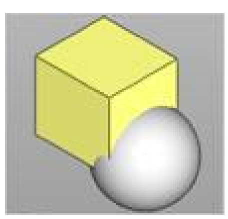
Edit Meshes Pane, Mesh Modeling Ribbon Bar

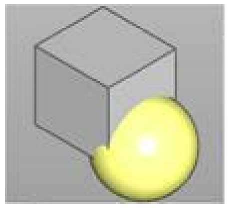
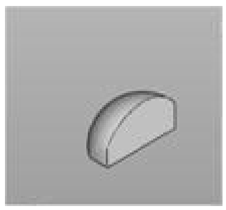
5.7.2.3 Intersect

Create a mesh that represents the volume that is shared between two intersecting meshes. The first mesh may be pre-selected. Both of the original meshes will be deleted after the Intersect function is completed. Only their shared volume will remain.Create a mesh that represents the volume that is shared between two or more intersecting meshes. The first set of meshes may be pre-selected. Each of the original meshes will be deleted after the Intersect function is completed. Only their one shared volume mesh will remain. A Mesh Revolve that is open-ended (not a full 360 degrees) cannot be used in this Intersect function.

> **Meshes Must be Closed**: If any selected mesh does not form closed volume (i.e., there are any open edges) the Mesh Boolean operations (i.e., Unite, Subtract or Intersect) will fail. This can sometimes occur with imported data files. If this occurs, use the Auto Fix command on each mesh before using the Unit, Subtract or Intersect Mesh commands. You can also use the Diagnose command to check for mesh errors.

Basic Procedure

	Screen Pick	Command Input
Step 1	Select the first mesh object to intersect. Select the first set of mesh objects to intersect. Right-click or press <Enter> when done. 	none
Step 2	Select the second mesh object.	

	Select the second set of mesh objects. Right-click or press <Enter> when done. 	
Final	The resulting mesh represents their shared volume. 	

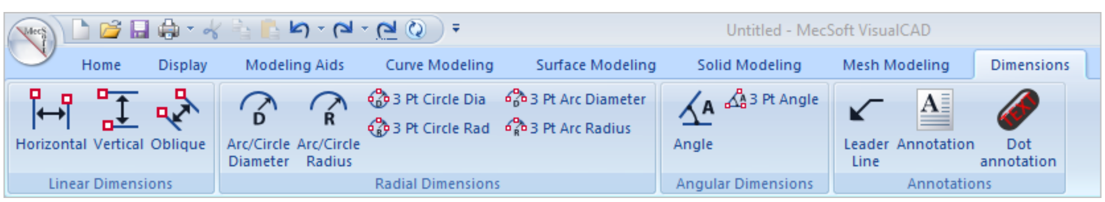
Related Topics

[Edit Meshes Pane, Mesh Modeling Ribbon Bar]()

5.8 Dimensions

The Dimensions Ribbon Bar contains Linear Dimensions, Radial Dimensions, Angular Dimensions and Annotations panes and associated functions.

The Dimensions Ribbon Bar



Dimensions Ribbon Bar

Related Topics

[Linear Dimensions Pane, Dimensions Ribbon Bar]()

[Radial Dimensions Pane, Dimensions Ribbon Bar]()

[Angular Dimensions Pane, Dimensions Ribbon Bar]()

Annotations Pane, Dimensions Ribbon Bar

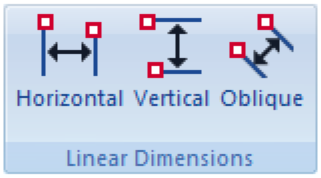
The Ribbon Bar

Customize Quick Access Toolbar

Customize Dialog

VisualCAD Options

5.8.1 Linear Dimensions



The Dimensions Ribbon Bar contains the following Linear Dimensions pane with associated functions.

Related Topics

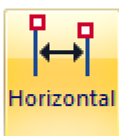
Create a Horizontal Dimension

Create a Vertical Dimension

Create an Oblique Dimension

Dimensions Ribbon Bar

5.8.1.1 Horizontal Dimension



Creates a linear dimension label horizontally.

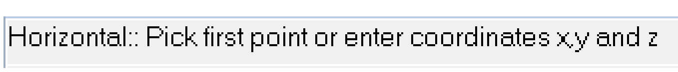
More Information

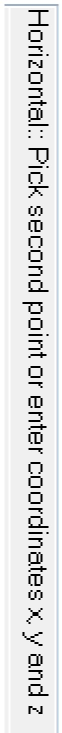
Depending on the view, the horizontal will be parallel to different principle axes (x, y, or z). The horizontal direction is whichever axis is horizontal when a view is first started. For example, the Iso-view has the z-plane as a construction plane and the horizontal direction is parallel to the x-axis. Meanwhile, the right-view has the x-plane as a construction plane with the horizontal direction parallel to the y-axis. If the view is rotated, the horizontal stays with the construction plane. If the construction plane is modified, then horizontal direction is then modified.

Two points are selected and they are projected onto a horizontal line. The dimension is measured from the separation of these projected points. In other words, only the distance along the horizontal direction is measured.

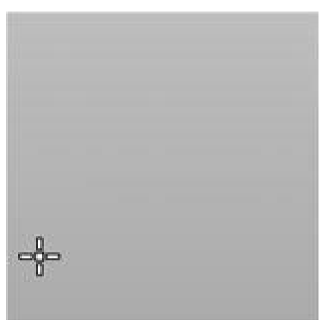
The label can be slid vertically so that its projection line intersects the mouse. The picked point determines where the label appears. The units are included.

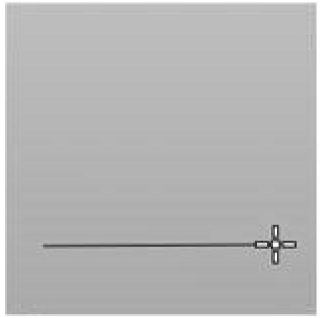
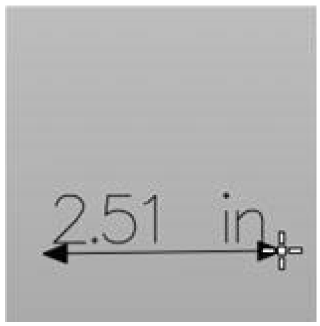
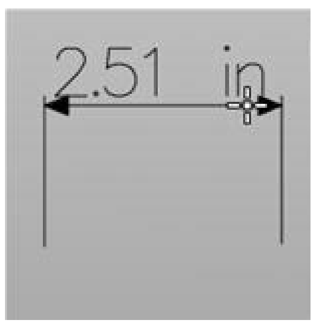
Command Prompts





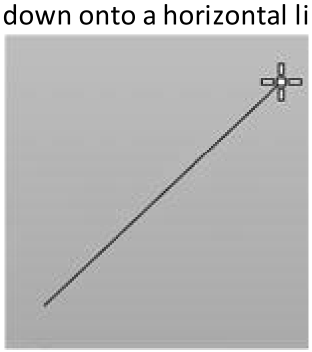
Basic Procedure

	Screen Pick	Command Input
Step 1	Pick the first point or type in the coordinates into the Command Input Bar. 	None
Step 2	Pick the second point or type in the coordinates.	

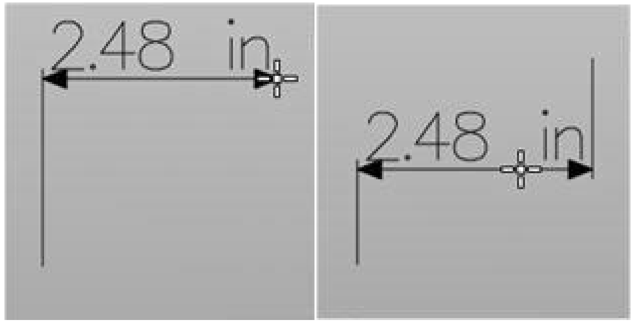
		
Step 3	Pick a point to determine the vertical position of the label.   	

Notes

1. Even if the picked points are not at the same level, the points will be projected down onto a horizontal line.



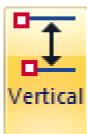
2. Here is the result.



Related Topics

Linear Dimensions Pane, Dimensions Ribbon Bar

5.8.1.2 Vertical Dimension



Creates a linear dimension label vertically.

More Information

Depending on the view, the vertical will be parallel to different principle axes (x, y, or z). The vertical direction is whichever axis is vertical when a view is first started. For example, the Iso-view has the z-plane as a construction plane and the vertical direction is parallel to the y-axis. Meanwhile, the right-view has the x-plane as a construction plane with the vertical direction parallel to the z-axis. If the view is rotated, the vertical stays with the construction plane. If the construction plane is modified, then vertical direction is then modified.

Two points are selected and they are projected onto a vertical line. The dimension is measured from the separation of these projected points. In other words, only the distance along the horizontal direction is measured.

The label can be slid horizontally so that its projection line intersects the mouse. The picked point determines where the label appears. The units are included.

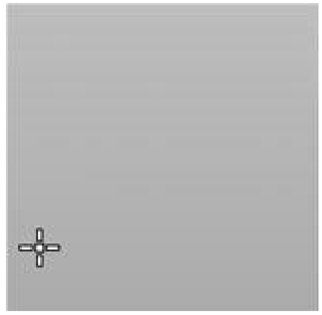
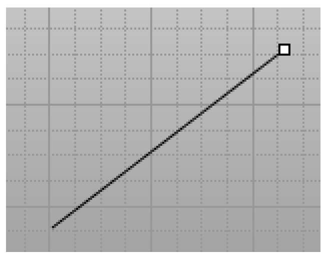
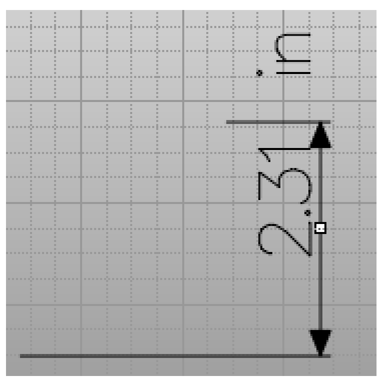
Command Prompts

Vertical:: Pick first point or enter coordinates x,y and z

Vertical:: Pick second point or enter coordinates x, y and z

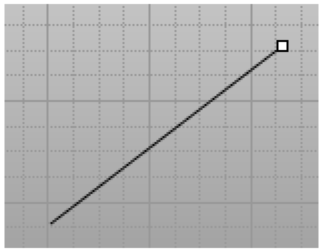
Basic Procedure

	Screen Pick	Command Input

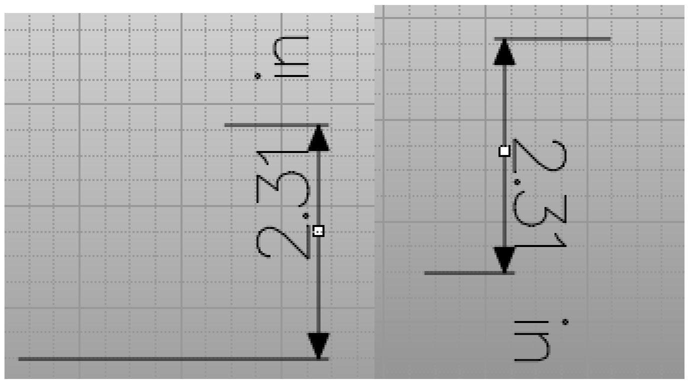
Step 1	Pick the first point or type in the coordinates into the Command Input Bar. 	None
Step 2	Pick the second point or type in the coordinates. 	
Step 3	Pick a point to determine the vertical position of the label. 	

Notes

1. Even if the picked points are not at the same level, the points will be projected down onto a horizontal line.



Here is the result.



Related Topics

[Linear Dimensions Pane, Dimensions Ribbon Bar]

5.8.1.3 Oblique Dimension

Creates a linear dimension label at an angle. Two points are selected. The three-dimensional distance between them is computed and is the number displayed. The label can be slid normally (perpendicular) to the line connecting the two points. The units are included.

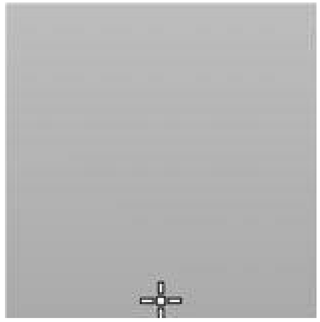
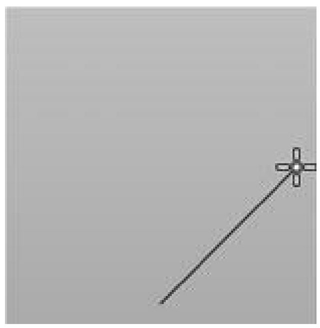
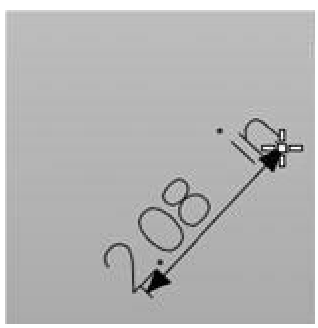
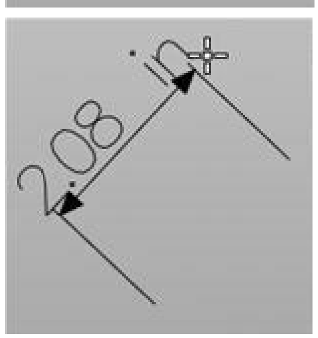
Command Prompts

Oblique:: Pick first point or enter coordinates x,y and z

Oblique:: Pick second point or enter coordinates x, y and z

Basic Procedure

	Screen Pick	Command Input
Step 1	Pick the first point or type in the coordinates into the Command Input Bar.	None

		
Step 2	Pick the second point or type in the coordinates. 	
Step 3	Pick a point to determine the offset position of the label.  	

Related Topics

Linear Dimensions Pane, Dimensions Ribbon Bar

5.8.2 Radial Dimensions

The Dimensions Ribbon Bar contains the following Radial Dimensions pane with associated functions.

Dimensions Ribbon Bar
Radial Dimensions Pane

Related Topics

Create an Arc Circle/Diameter Dimension

Create an Arc Circle/Radius Dimension

Create a 3-Point Circular Diameter Dimension

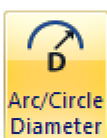
Create a 3-Point Circular Radius Dimension

Create a 3-Point Arc Diameter Dimension

Create a 3 Point Arc Radius Dimension

Dimensions Ribbon Bar

5.8.2.1 Arc/Circle Diameter



Create a dimension label for an arc. It measures the diameter of the circle of which the arc is a part. The letter ‘D’ appears in the label to show that it is measuring the diameter.

An arrow specifies which arc is labeled. The text is always horizontal. Both are always in the plane of the arc. Both can be slid around the arc. The units are included.

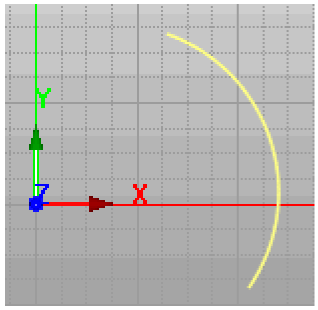
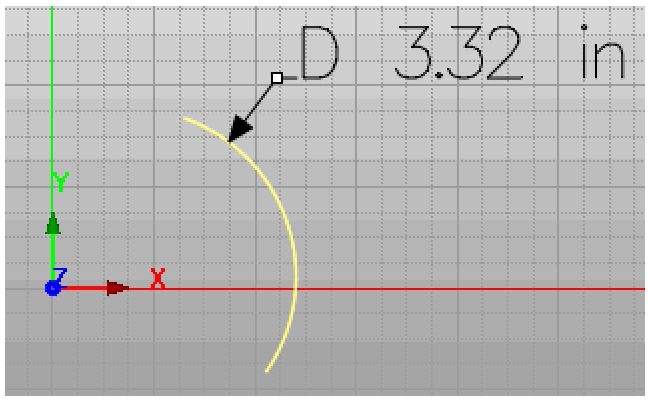
Command Prompts

Circle:: Pick the arc to dimension

Circle:: Enter dimension point

Basic Procedure

	Screen Pick	Command Inpu

Step 1	Pick the arc or circle to dimension. 	None
Step 2	Select a point to locate the radial dimension. 	

Related Topics

Radial Dimensions Pane, Dimensions Ribbon Bar

5.8.2.2 Arc/Circle Radius

Create a dimension label for an arc. It measures the radius of the circle of which the arc is a part. The letter ‘R’ appears in the label to show that it is measuring the radius.

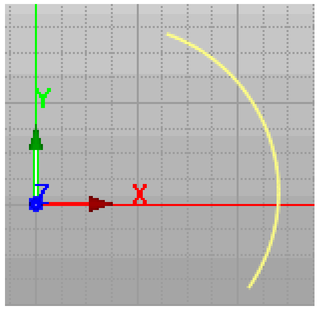
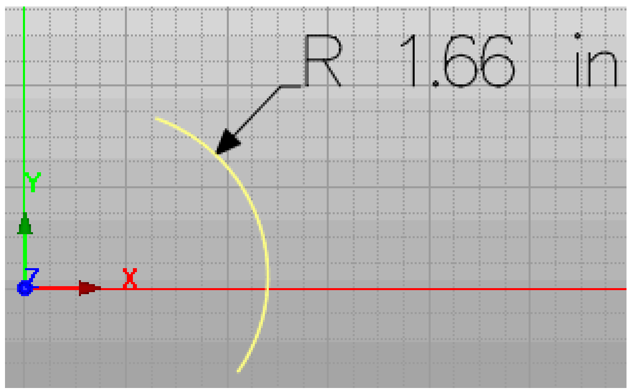
An arrow specifies which arc is labeled. The text is always horizontal. Both are always in the plane of the arc. Both can be slid around the arc. The units are included.

Command Prompts

Arc Diameter:: Pick the arc to dimension

Arc Diameter:: Enter dimension point

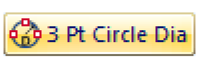
Basic Procedure

	Screen Pick	Command Input
Step 1	Pick the arc or circle to dimension. 	None
Step 2	Select a point to locate the radial dimension. 	

Related Topics

Radial Dimensions Pane, Dimensions Ribbon Bar

5.8.2.3 3 Pt Circle Dia



Create a dimension label for a circle. It measures the diameter of the circle. The letter ‘D’ appears in the label to show that it is measuring the diameter.

An arrow specifies which circle is labeled and is always radial. The text is always horizontal. Both are always in the plane of the circle. Both can be slid around the circle. The units are included. **Note:** You can use points as input to this command.

Command Prompts

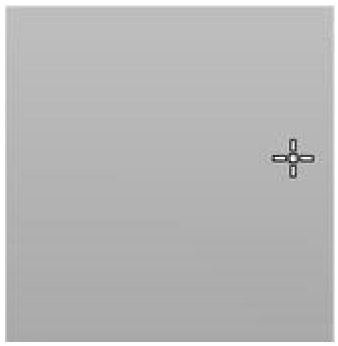
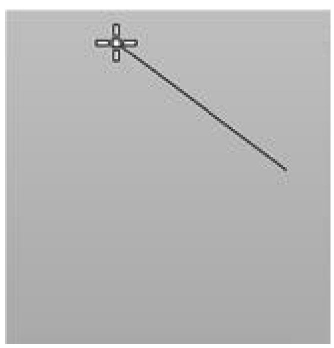
3 Pt Circle Dia:: Pick first point or enter coordinates x,y,z

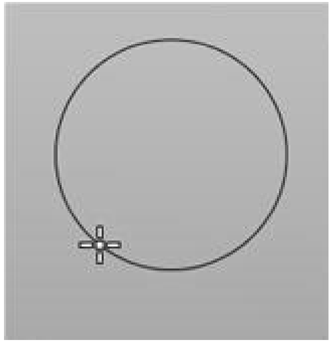
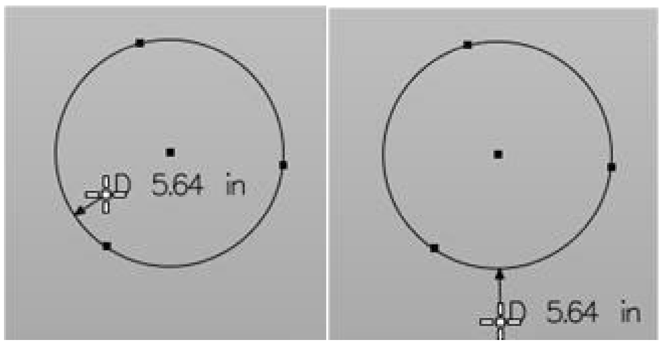
3 Pt Circle Dia:: Pick second point or enter coordinates x,y,z

3 Pt Circle Dia:: Pick third point or enter coordinates x,y,z

3 Pt Circle Dia:: Pick third point or enter the dimension point

Basic Procedure

	Screen Pick	Command Inpu
Step 1	Pick the first point on the circle or type in the coordinates into the Command Input Bar. 	None
Step 2	Pick the second point on the circle or type in coordinates. 	
Step 3	Pick the third point on the circle or type in coordinates.	

		
Step 4	Pick the endpoint of the arrow which points at the circle. This is also where the text appears. 	

Related Topics

Radial Dimensions Pane, Dimensions Ribbon Bar

5.8.2.4 3 Pt Circle Rad

Create a dimension label for a circle. It measures the radius of the circle. The letter 'R' appears in the label to show that it is measuring the radius.

An arrow specifies which circle is labeled and is always radial. The text is always horizontal. Both are always in the plane of the circle. Both can be slid around the circle. The units are included. **Note:** You can use points as input to this command.

Command Prompts

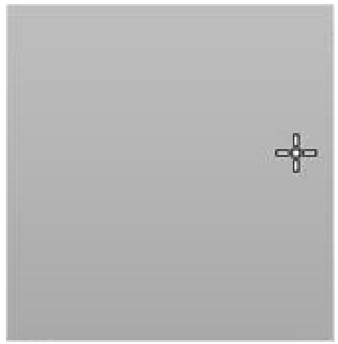
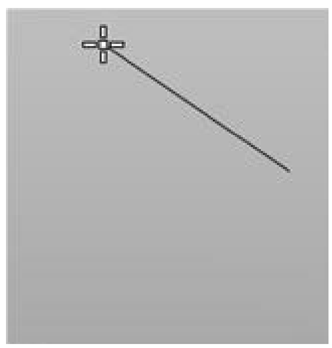
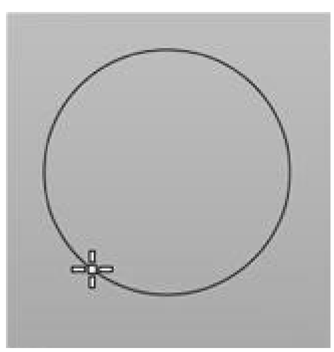
3 Pt Circle Rad:: Pick first point or enter coordinates x,y,z

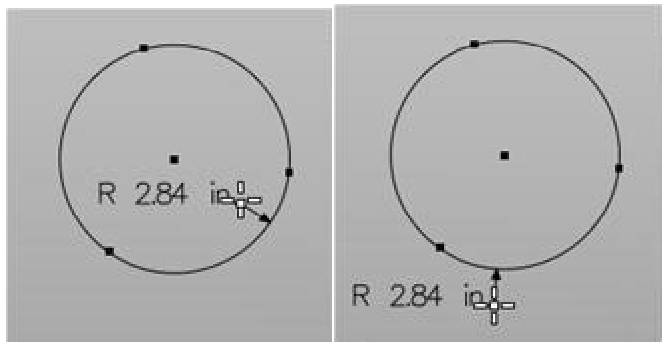
3 Pt Circle Rad:: Pick second point or enter coordinates x,y,z

3 Pt Circle Rad:: Pick third point or enter coordinates x,y,z

3 Pt Circle Rad:: Pick third point or enter the dimension point

Basic Procedure

	Screen Pick	Command Inpu
Step 1	Pick the first point on the circle or type in the coordinates into the Command Input Bar. 	None
Step 2	Pick the second point on the circle or type in coordinates. 	
Step 3	Pick the third point on the circle or type in coordinates. 	
Step 4	Pick the endpoint of the arrow which points at the circle. This is also where the text appears.	

Related Topics

[Radial Dimensions Pane, Dimensions Ribbon Bar]

5.8.2.5 3 Pt Arc Diameter

Create a dimension label for an arc. It measures the diameter of the circle of which the arc is a part. The letter ‘D’ appears in the label to show that it is measuring the diameter.

An arrow specifies which arc is labeled. The text is always horizontal. Both are always in the plane of the arc. Both can be slid around the arc. The units are included. **Note:** You can use points as input to this command.

Command Prompts

3 Pt Arc Diameter:: Pick first point or enter coordinates x,y,z

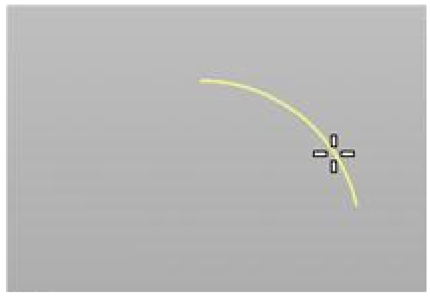
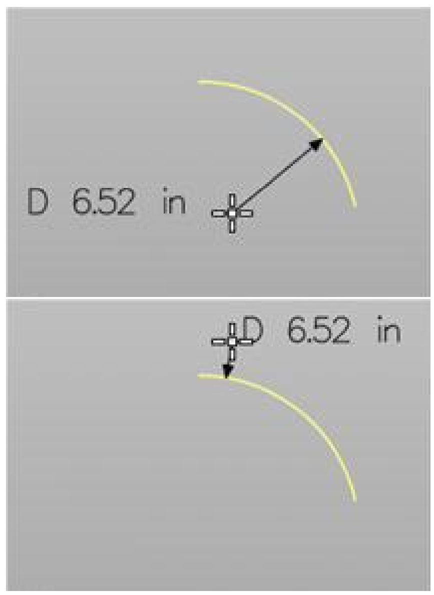
3 Pt Arc Diameter:: Pick second point or enter coordinates x,y,z

3 Pt Arc Diameter:: Pick third point or enter coordinates x,y,z

3 Pt Arc Diameter:: Pick third point or enter the dimension point

Basic Procedure

	Screen Pick	Command Input
Step 1	Pick the arc. It can be pre-selected.	None

		
Step 2	Pick the endpoint of the arrow which points at the arc. This is also where the text appears.  	

Related Topics

Radial Dimensions Pane, Dimensions Ribbon Bar

5.8.2.6 3 Pt Arc Radius

Create a dimension label for an arc. It measures the radius of the circle of which the arc is a part. The letter ‘R’ appears in the label to show that it is measuring the radius.

An arrow specifies which arc is labeled. The text is always horizontal. Both are always in the plane of the arc. Both can be slid around the arc. The units are included. **Note:** You can use points as input to this command.

Command Prompts

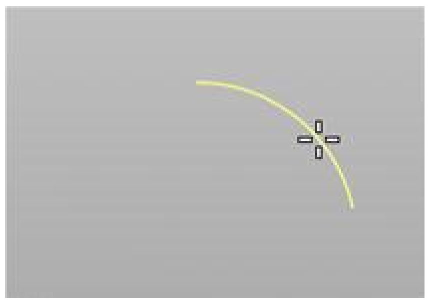
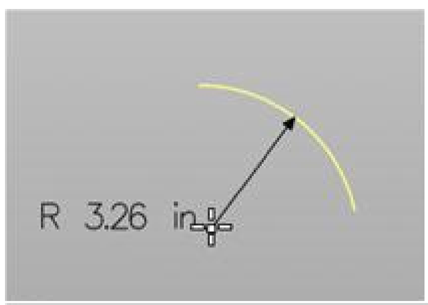
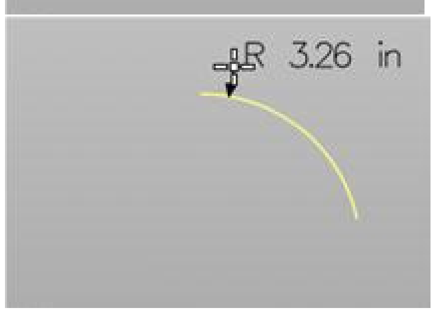
3 Pt Arc Radius:: Pick first point or enter coordinates x,y,z

3 Pt Arc Radius:: Pick second point or enter coordinates x,y,z

3 Pt Arc Radius:: Pick third point or enter coordinates x,y,z

3 Pt Arc Radius:: Pick third point or enter the dimension point

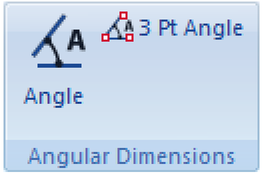
Basic Procedure

	Screen Pick	Command Input
Step 1	Pick the arc. It can be pre-selected. 	None
Step 2	Pick the endpoint of the arrow which points at the arc. This is also where the text appears.  	

Related Topics

[Radial Dimensions Pane, Dimensions Ribbon Bar]()

5.8.3 Angular Dimensions



The Dimensions Ribbon Bar contains the following Angular Dimensions pane with associated functions.

Related Topics

Create an Angle Dimension

Create a 3 Point Angle Dimension

Dimensions Ribbon Bar

5.8.3.1 Angle

Create a dimension label for the angle between line segments. The segments do not have to intersect; they will be extended into lines for the purposes of calculation.

The text is always along the dimension label and centered. The units are degrees and are included.

There are four possible arcs. Pick a point which will define the quarter of the labeling plane and how far away from the vertex to put the label.

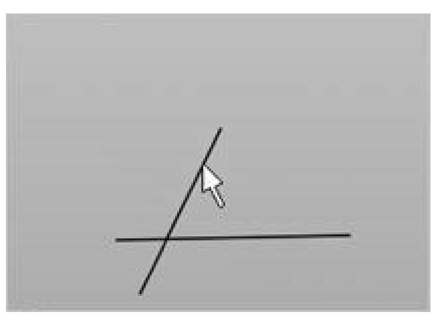
Command Prompts

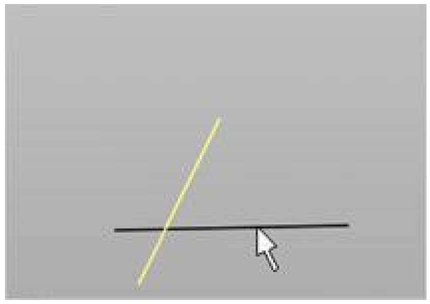
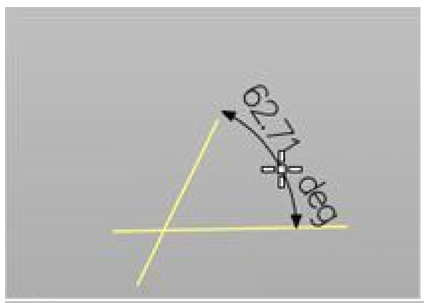
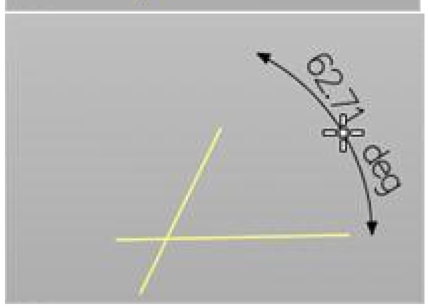
Angle:: Pick first curve

Angle:: Pick second curve

Angle:: Enter dimension point

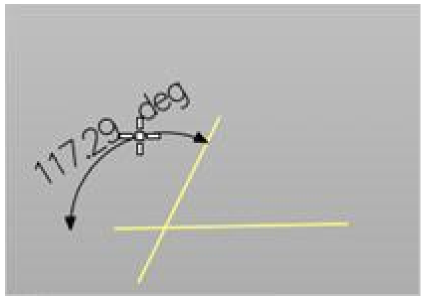
Basic Procedure

	Screen Pick	Command Input
Step 1	Pick the first line segment. It can be pre-selected. 	None

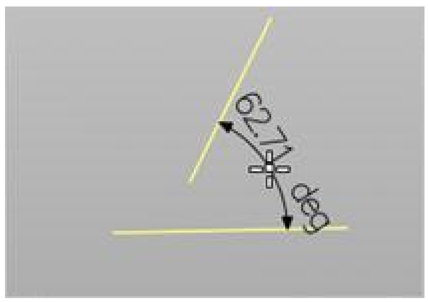
Step 2	Pick the second line segment. 	
Step 3	Pick the radial distance where the label should be displayed. Arrows go along the arc and point at the two rays. The label appears on the outside of the arc.  	

Notes

1. It is possible to choose an arc not inside the angle that was just made. For example, the defined angle was the 63 degree angle, but the supplementary 117 degree angle can also be dimensioned.



2. It is not necessary for the segments to intersect, the segments will be extended to intersection for the purpose of calculation.



Related Topics

[Angular Dimensions Pane, Dimensions Ribbon Bar]

5.8.3.2 3 Pt Angle

Create a dimension label for an arc. It measures the angle subtended.

An arrow specifies which arc is labeled. The text is always along the dimension label and centered. Both are always in the plane of the arc. Both can be slid radially. The units are degrees and are included.

Pick three points, the first of which is the vertex of the arc and two additional points which define lines going through the vertex point. There are now four possible arcs, one in each quarter of these intersecting lines. Pick another point which defines which quadrant and how far away from the vertex the label should be. **Note:** You can use points as input to this command.

Command Prompts

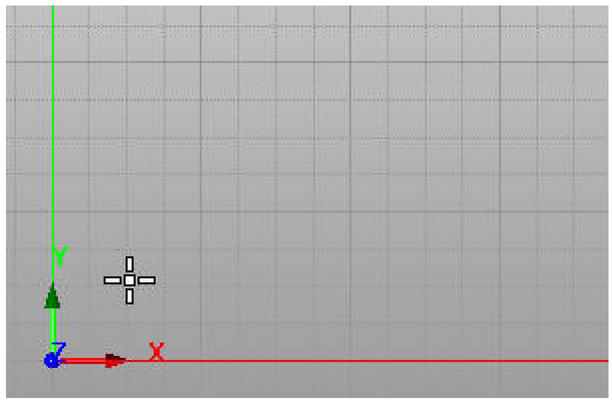
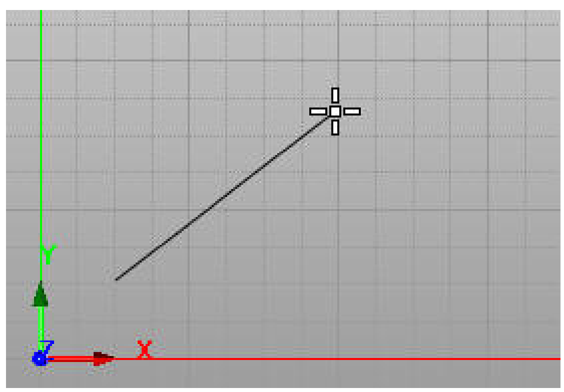
3 Pt Angle:: Pick first point or enter coordinates x,y and z

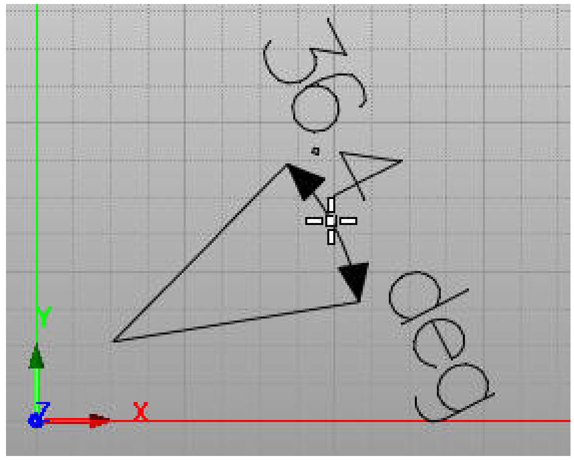
3 Pt Angle:: Pick second point or enter coordinates x, y and z

3 Pt Angle:: Pick third point or enter coordinates x,y and z

3 Pt Angle:: Enter dimension point

Basic Procedure

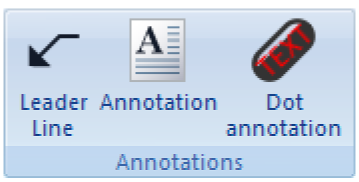
	Screen Pick	Command Input
Step 1	Pick the first point on the arc to dimension or enter x,y,z coordinates. 	None
Step 2	Pick the second point on the arc to dimension or enter x,y,z coordinates. 	
Step 3	Enter the dimension point.	



Related Topics

[Angular Dimensions Pane, Dimensions Ribbon Bar]

5.8.4 Annotations



The Dimensions Ribbon Bar contains the following Annotations pane with associated functions.

Related Topics

Create a Leader Line Dimension

Create a Text Annotation

Dimensions Ribbon Bar

5.8.4.1 Leader Line

Create a text label with an arrow pointing in a direction. It can be used to label various pieces of geometry or other items of interest.

More Information

Type in the text into the Command Input Bar.

Create a dimension label for the angle between line segments. The segments do not have to intersect; they will be extended into lines for the purposes of calculation.

The text is always along the dimension label and centered. The units are degrees and are included.

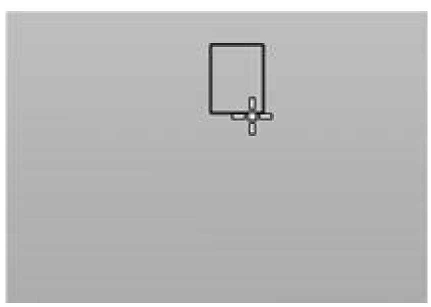
There are four possible arcs. Pick a point which will define the quarter of the labeling plane and how far away from the vertex to put the label.

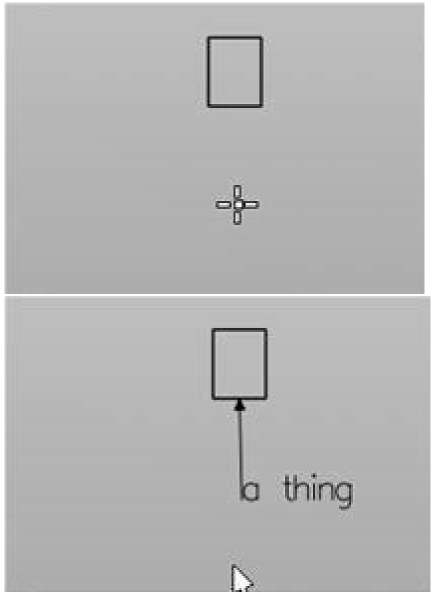
Command Prompts

Leader Line:: Enter text string

Leader Line:: Pick point or enter coordinates x,y and z

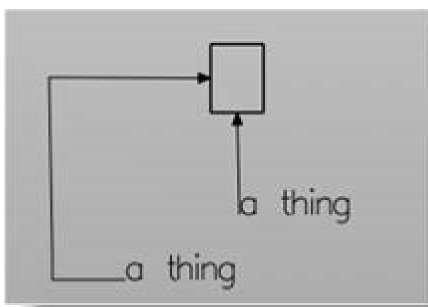
Basic Procedure

	Screen Pick	Command Input
Step 1	Pick the point or type in coordinates. This is where the head of the arrow should go. The arrow is a polyline and so can have multiple sections at angles. This can be useful if having to label something in the middle of a collection of other items. The labeling arrow can bend its way through the collection to point at the correct item. 	None
Step 2	Right-click the mouse to stop adding segments to the polyline. The text will appear at the last point selected.	

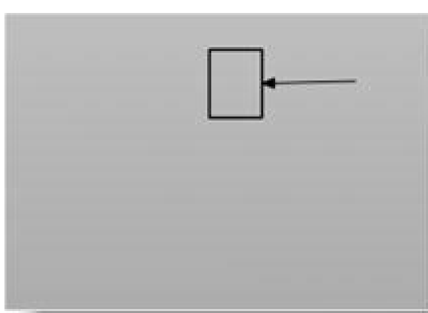
		

Notes

1. Here is an example of a multiple segment polyline leader.



2. Text does not have to be included. This results in only an arrow. When asked for text, start picking the endpoints of the leader.



Related Topics

[Annotations Pane, Dimensions Ribbon Bar]

5.8.4.2 Annotation

Create annotation text. It can be used to put in notes or other information in your drawing. The text Font Height can be edited individually for each annotation. See Editing Annotation Text Font Height below for more information on how to do this.

Command Prompts

Leader Line:: Enter text string

Leader Line:: Pick point or enter coordinates x,y and z

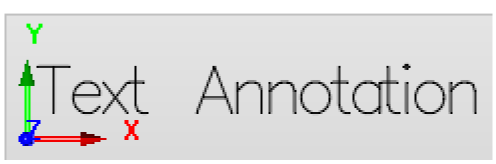
Basic Procedure

	Screen Pick	Command Input
Step 1	Type in the text into the Command Input Bar.	None
Step 2	Pick the point or type in coordinates. This is where the text will appear.  	

Editing Annotation Font Text Height

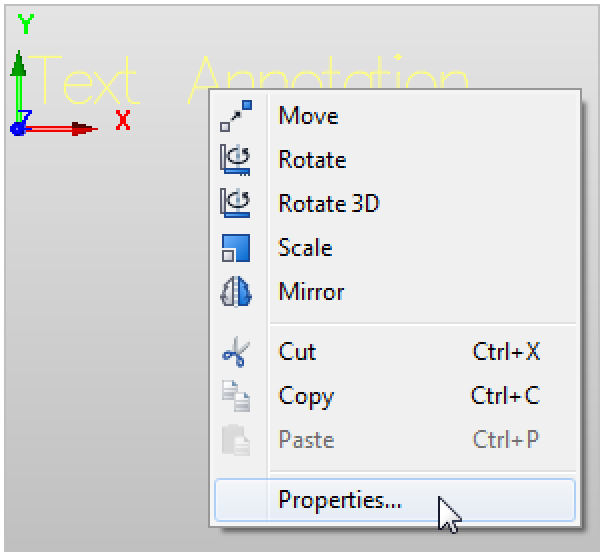
The height of annotation text can be edited individually for each annotation entity. Here's how its done:

1. Select the Text Annotation that you wish to edit.



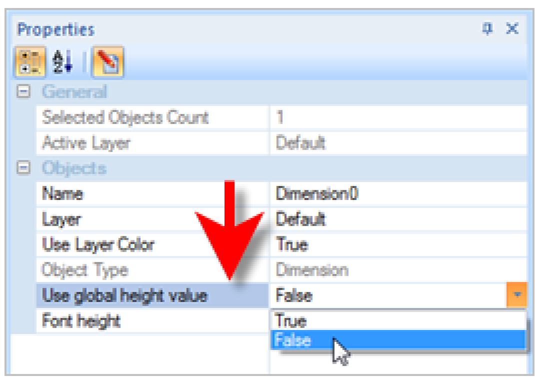
Annotation Text Example

2. Right-click and select Properties.



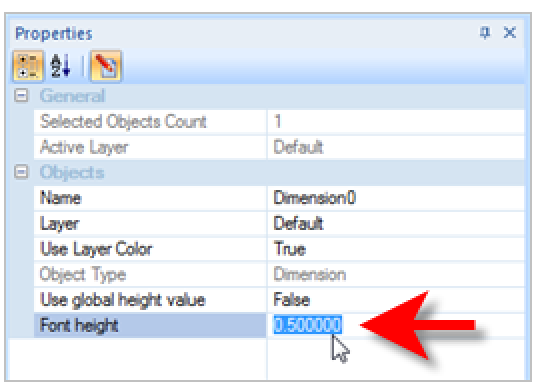
Right-click & select Properties

3. From the Properties dialog, change the Use global height value to False.



The Properties dialog

4. Then edit the Font Height.



The Properties dialog

Related Topics

[Annotations Pane, Dimensions Ribbon Bar](#)

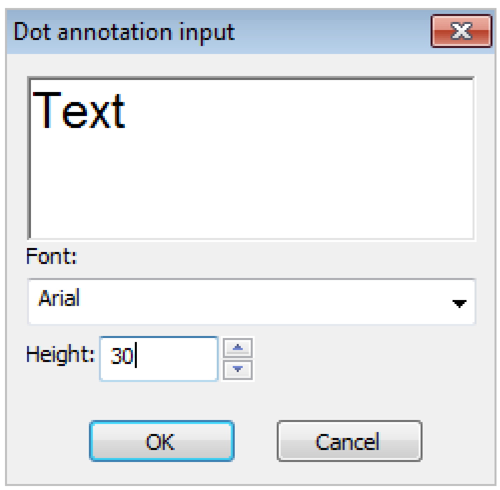
5.8.4.3 Dot Annotation

Create graphical dot text at a selected point. This can be used to identify geometry in VisualCAD without using a geometric element.

Command Prompts

Dot annotation:: Enter start point of text

Basic Procedure

	Screen Pick	Command Input
Step 1	Use the Dot Annotation Input dialog to enter your Text, Font and Height and then pick OK. 	None
Step 2	Select a location in the drawing window to place the dot annotation. 	

Related Topics

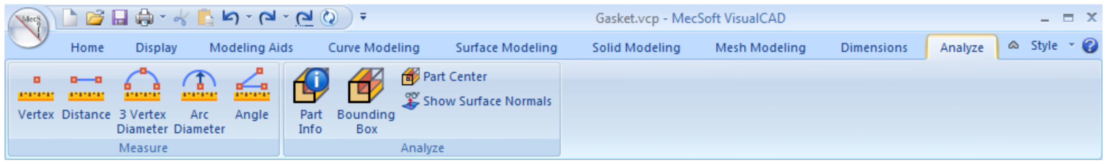
[Annotations Pane, Dimensions Ribbon Bar](#)

[Annotation](#)

5.9 Analyze

The Analyze Ribbon Bar contains Measure and Anal panes and associated functions.

The Analyze Ribbon Bar



Analyze Ribbon Bar

Related Topics

[Measure Pane, Analyze Ribbon Bar]

[Analyze Pane, Analyze Ribbon Bar]

[The Ribbon Bar]

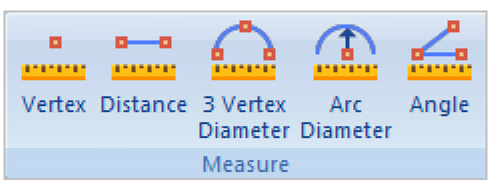
[Customize Quick Access Toolbar]

[Customize Dialog]

[VisualCAD Options]

5.9.1 Measure

The Analyze Ribbon Bar contains the following Measure pane with associated functions.



Related Topics

[Measure a Vertex]

[Measure a Distance]

[Measure a 3 Vertex Diameter]

[Measure an Arc Diameter]

[Measure an Angle]

[Analyze Ribbon Bar]

5.9.1.1 Vertex

Get the X, Y, Z coordinates of a selected point (vertex). Move the cursor over a point with one of the object snaps active. The coordinates of the point are displayed dynamically. After you select the point, the coordinates are displayed in the command prompt window.

Command Prompts

Co-ordinates:: Pick vertex point

Vertex: {0.500000,1.250000,0.000000}

Basic Procedure

	Screen Pick	Command Input
Step 1		Pick a vertex point.

Notes

1. Unless a point on an existing object is selected, the picked point will be in the Construction Plane of the currently selected view.

Related Topics

[Measure Pane, Analyze Ribbon Bar]

5.9.1.2 Distance

Get the 3D distance between two points. The point coordinates and the distance between them are displayed dynamically. The distance is also displayed in the command prompt window.

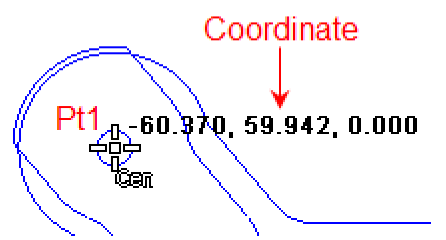
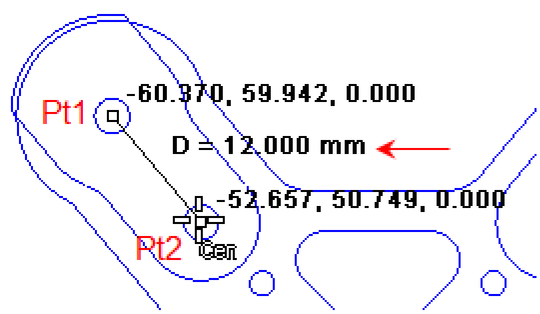
Command Prompts

Distance:: Pick first point or enter coordinates x,y and z

Distance:: Pick second point or enter coordinates x, y and z

D: 1.414214 in, Dz: 0.000000 in; {-2.000000,1.000000,0.000000} - {-1.000000,2.000000,0.000000}

Basic Procedure

	Screen Pick	Command Input
Step 1		Pick first point or enter x,y
Step 2		(Optional) Enter X, Y, [Z] coordinates for the second

Notes

1. The distance is displayed with a letter 'D' and the units. The 3D coordinates of the two picked points will also be displayed after the distance.
2. Unless points on an existing object are selected, picked points will be in the construction plane of the currently selected view.

Related Topics

Measure Pane, Analyze Ribbon Bar

5.9.1.3 3 Vertex Diameter

Get the diameter and center coordinates of a temporary arc through three points. The coordinates of each of the three points are dynamically displayed. The calculated diameter between the three points is also dynamically displayed on the arc as well as in the command prompt window. The arc center coordinates are displayed in the command prompt window. **Note**: You can select arcs, Circles or surface edges.

Command Prompts

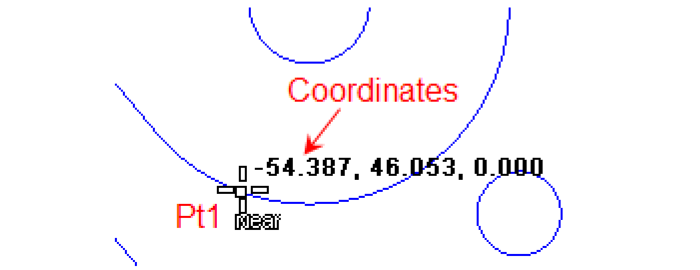
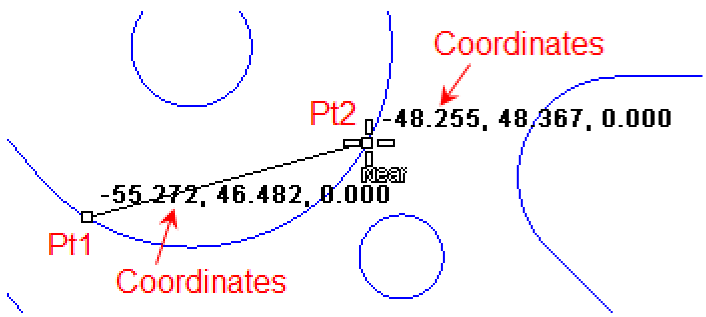
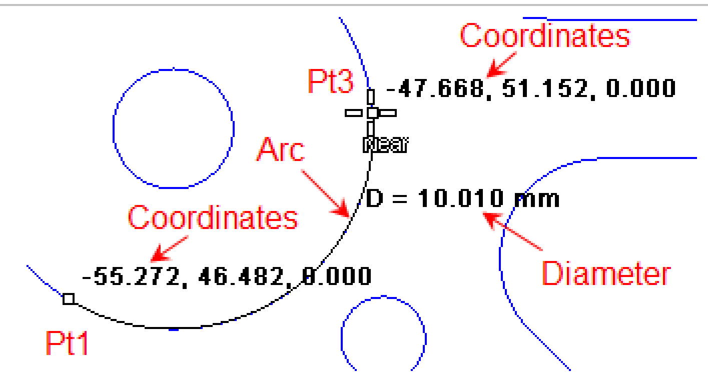
(Optional) Enter X, Y, [Z] coordinates for the second point (intermediate point on arc).

(Optional) Enter X, Y, [Z] coordinates for the second point (intermediate point on arc).

(Optional) Enter X, Y, [Z] coordinates for the second point (intermediate point on arc).

Arc Diameter: 10.010000 mm, Center: {-52.656830,50.749151,0.000000}

Basic Procedure

	Screen Pick
Step 1	
Step 2	
Step 3	

Notes

1. The diameter (with units) and center information is from the circle that contains the temporary arc.

2. Unless points on an existing object are selected, picked points will be in the construction plane of the currently selected view.

Related Topics

[Measure Pane, Analyze Ribbon Bar]

5.9.1.4 Arc Diameter

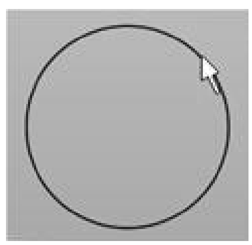
Get the diameter of an arc or circle. Pick an arc or circle and the diameter and center will be displayed in the Prompt Input Bar. **Note**: You can select arcs, Circles or surface edges.

Command Prompts

Arc Diameter:: Pick Arc/Closed Curve to measure

Arc Diameter: 6.000000 in, Center: {-12.000000,7.000000,0.000000}

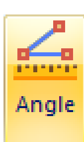
Basic Procedure

	Screen Pick	Command Input
Step 1	Select an arc or circle. 	none

Related Topics

[Measure Pane, Analyze Ribbon Bar]

5.9.1.5 Angle



Get the angle formed between a vertex point and two other points.

The coordinates of the points are dynamically displayed along with the included angle.

Command Prompts

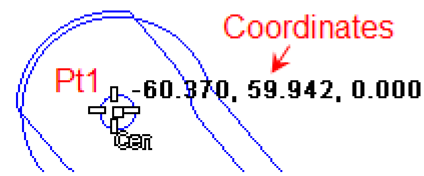
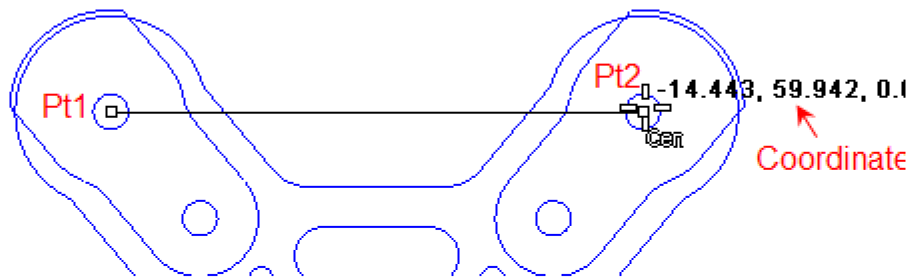
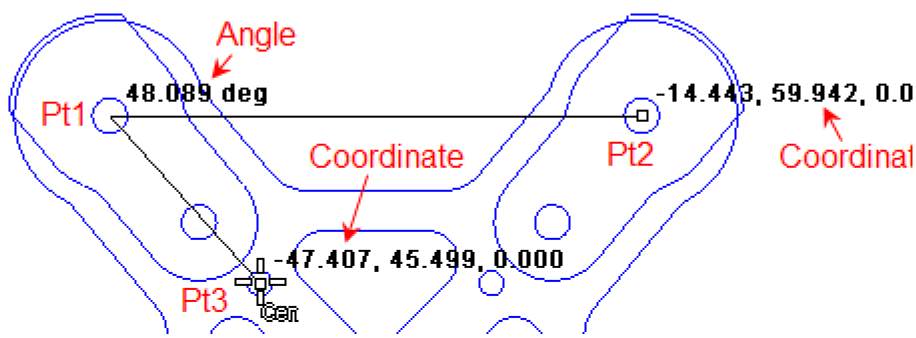
Pick first point or enter coordinates x,y and z

Pick second point or enter coordinates x,y and z

Pick third point or enter coordinates x,y and z

Angle: 48.089217 deg

Basic Procedure

<table>
<tr><th colspan="2">Screen Pick</th></tr>
<tr><td>Step 1</td><td>
</td></tr>
<tr><td>Step 2</td><td>
</td></tr>
<tr><td>Step 3</td><td>
</td></tr>
</table>

Notes

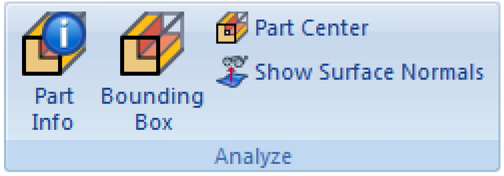
1. The resultant angle is always between 0 and 180 degrees.
2. The picked points will be in the Construction Plane of the currently selected view.

Related Topics

Measure Pane, Analyze Ribbon Bar

5.9.2 Analyze

The Analyze Ribbon Bar contains the following Analyze pane with associated functions.



Analyze Ribbon Bar
Analyze Pane

Related Topics

Analyze the Part Information

Analyze the Bounding Box

Analyze the Part Center

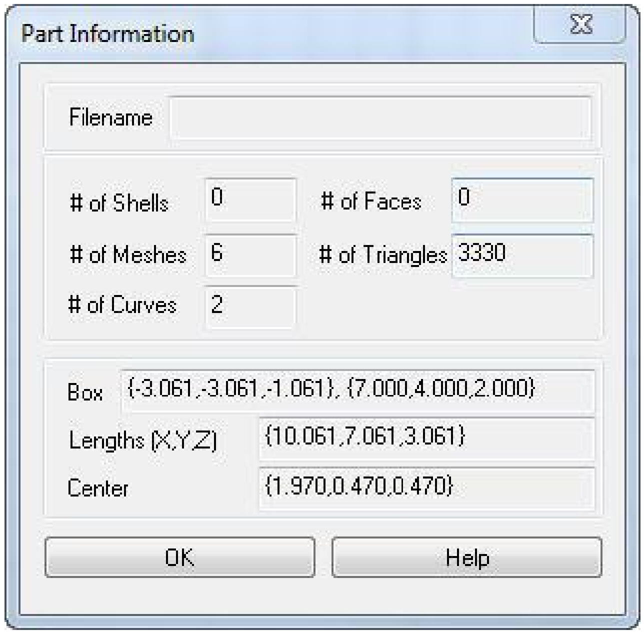
Show Surface Normals

Analyze Ribbon Bar

5.9.2.1 Part Info.

When this function is selected, all geometry is evaluated and important information is reported in a dialog, as shown below:

Dialog Box: Part Information



Part Information dialog box

- Filename
  Shows the saved location of the current part file. This field will be empty if the part file has not yet been saved.
- Shells
  Refers to poly-surface geometry objects.
- Faces
  Refers to surface geometry objects.
- Meshes
  Refers to a Box, Sphere, Cylinder, Cone Torus, Rectangular Tube, Circular Tube, Extrude, and Revolve.
- Curves
  Refers to the number of points, lines, polylines, arcs, circles, and other curves in the part geometry.
- Faces
  Refers to the number of surfaces in the part geometry.
- Triangles
  Refers to the number of triangles in all mesh geometry.
- Box, Lengths, Center
  This information is the same as described in the Bounding Box function and Part Center function.
- OK
  Closes the dialog.

- Help
  Brings up the on-line Help.

Related Topics

Analyze Pane, Analyze Ribbon Bar

5.9.2.2 Bounding Box

When this function is selected, VisualCAD will analyze all existing geometry, will create an internal minimum, volumetric box that contains all that geometry, and will return the following information about the box:

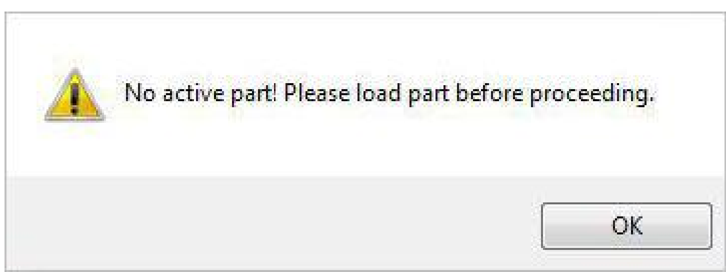
- {minimum X, Y, Z} equals the lower left bottom vertex of the box.
- {maximum X, Y, Z} equals the upper right top vertex of the box.
- {lengths along X, Y, Z} of the box.

This information is oriented to the World Coordinate System (WCS).

Command Prompts

Notes

1. Existing geometry is required for the Analyze functions. If none exists, a message is returned.



Analyze Bounding Box Message

2. Geometry objects do not need to be selected since the bounding box will automatically contain all existing objects.
3. All geometry is evaluated, including points, curves, surfaces, meshes, etc.
4. Geometry on all layers, visible or invisible, is contained in the box.

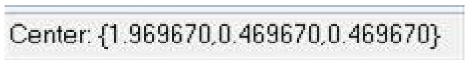
Related Topics

Analyze Pane, Analyze Ribbon Bar

5.9.2.3 Part Center

This function will return the coordinates of the center of the bounding box that contains all the part geometry, as referred to in the Bounding Box function above.

Command Prompts



Notes

1. Refer to the Notes in the Bounding Box function above.

Related Topics

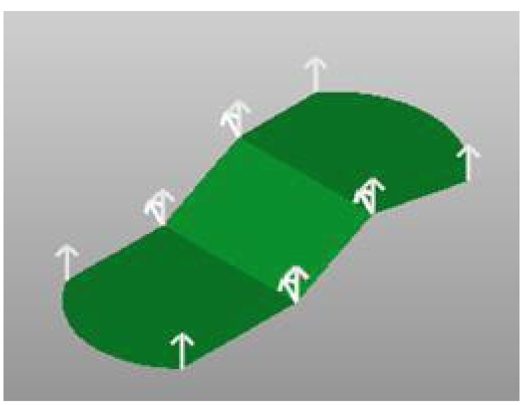
Analyze Pane, Analyze Ribbon Bar

5.9.2.4 Show Surface Normals

Selecting this function will cause normal vector arrows to display on every surface on the screen. Geometry objects do not need to be selected. Mesh objects do not apply.

Command Prompts

Basic Procedure



Press the ESC key to abort the display of surface normals.

Notes

1. Surface normals are also displayed for objects that are locked. If no valid surfaces are found, the following error message will be displayed.

Related Topics

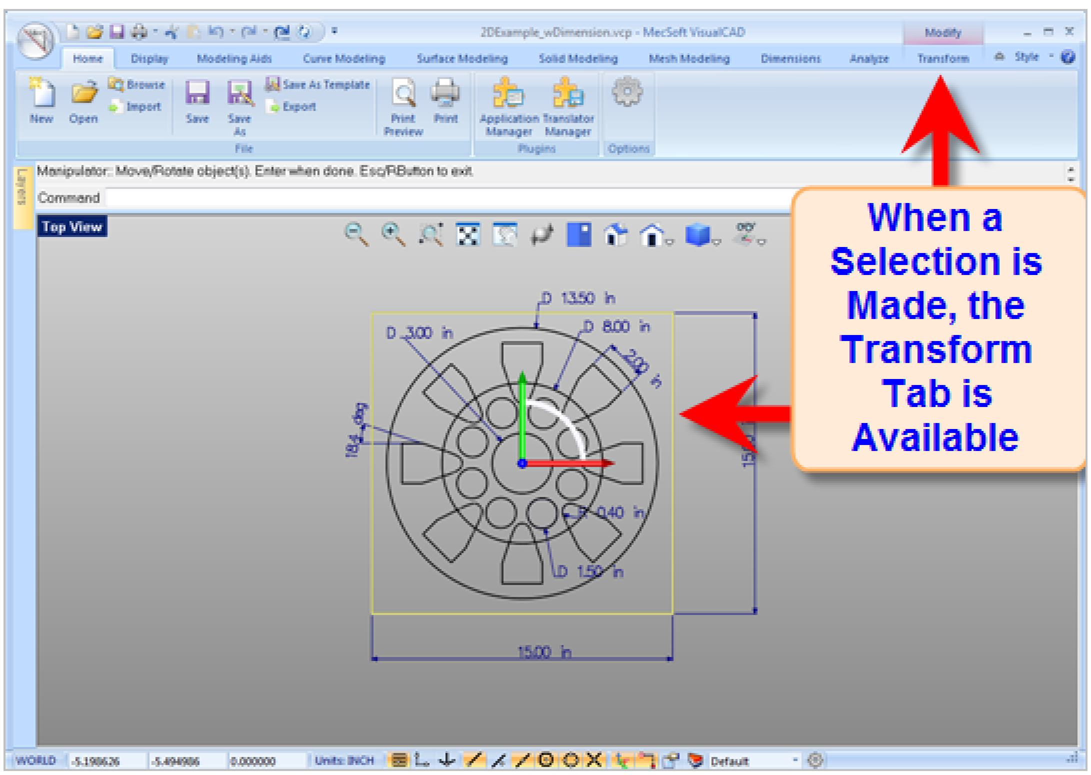
[Analyze Pane, Analyze Ribbon Bar]

5.10 Transform

The Transform Ribbon Bar contains a Transform pane and associated functions.

Enabling the Transform Ribbon Bar

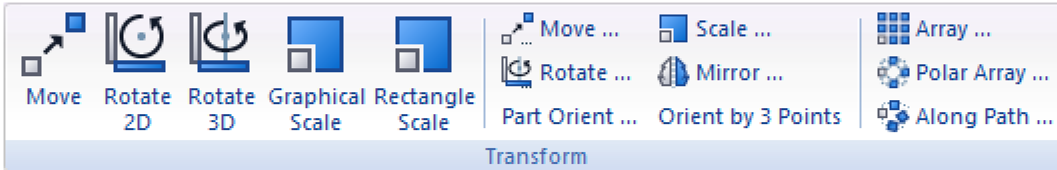
When a geometry selection is made VisualCAD enter Edit Mode and allows access to the Transform Ribbon Bar.



The Transform Ribbon Bar is Enabled

Selecting Transform will display the Transform Ribbon Bar.

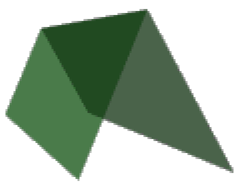
Selecting a Transform command will act upon the selected geometry.



Transform Ribbon Bar

Context Transform Menu (Right-click)

Selecting geometry in a viewport and by using right mouse button click also provides access to transformation tools.


Context Transform Menu (Right-click)

If you do not see the above options when you Right mouse button click, enable Show pop-up menu under Home > Options.

Related Topics

Move Objects by Mouse

Rotate Selected Object in 2D

Rotate Selected Object in 3D

Graphically Scale Selected Objects

Rectangle Scale selected Objects

Move Objects in 3D

Rotate Selected Objects

Orient the Entire Part

Scale Selected Objects

Mirror Selected Objects

Array Selected Objects

Polar Array Selected Objects

Graphical Manipulator

Transform Ribbon Bar

5.10.1 Move by Mouse

All selected items will be moved in 3D. The direction and distance to move is specified by picking a starting and ending point. All selected items will be moved that direction and distance.

Command Prompts

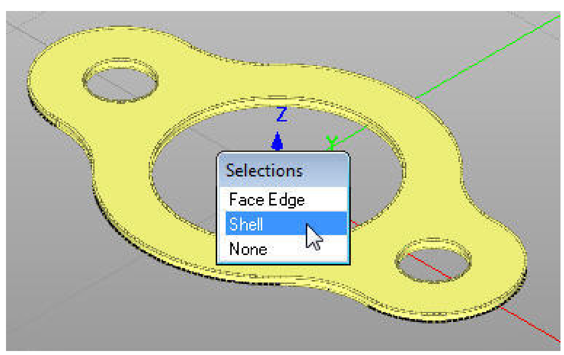
Select objects and hit Right Mouse Button/Enter when done

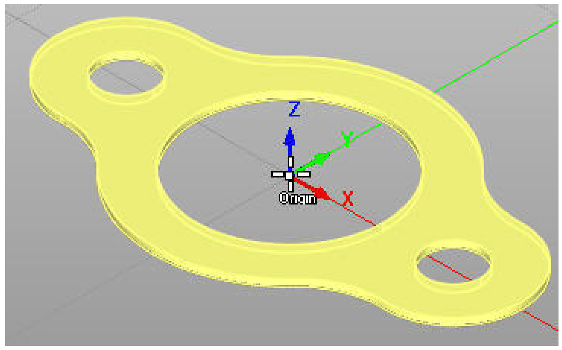
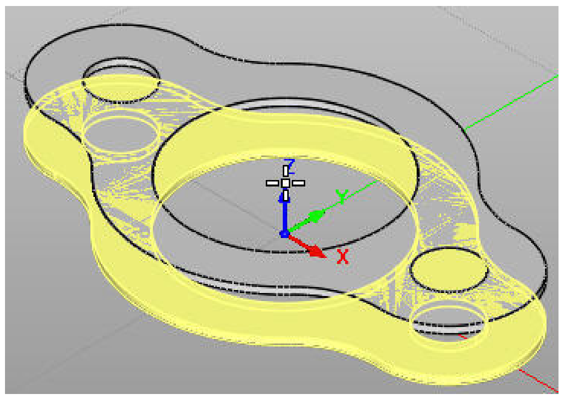
Move by mouse:: Pick from point or enter coordinates x,y and z

Move by mouse:: Pick to point or enter coordinates x,y and z

Added # objects to selection!

Basic Procedure

	Screen Activity	Comments
Step 1	Select items to move. The items could be pre-selected. If a mesh is selected, choose from the pop-up selection options provided.  	
Step 2	Pick a starting point or type in coordinates into the **Command Bar**.	

		
Step 3	Pick an ending point or type in coordinates. 	

Related Topics

Transform Ribbon Bar

5.10.2 Rotate 2D

Rotate selected objects around a picked point a given angle. The plane of rotation is parallel to the construction plane in the currently selected view.

Command Prompts

2D Rotate by mouse:: Select objects - hit right mouse to end

2D Rotate by mouse:: Enter rotation center point

2D Rotate by mouse:: Enter first reference point or rotation angle

2D Rotate by mouse:: Enter second reference point

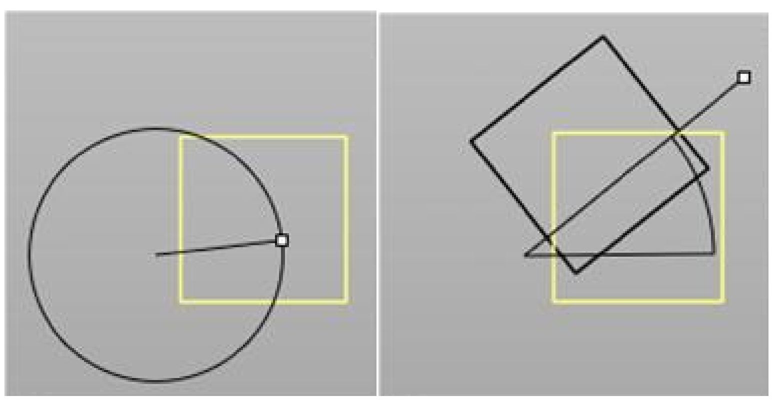
Basic Procedure

	Screen Activity	Comments

Step 1	Pick objects to be rotated. They can be pre-selected.	
Step 2	Pick a center of rotation or type in the coordinates into the Command Input Bar.	
Step 3	Pick a starting point or type in the coordinates. A temporary ray from the center of rotation going through the starting point will be made.	
Step 4	Pick an ending point or type in the coordinates. A temporary ray from the center of rotation going through the ending point will be made. The angle of rotation will be measured from the starting ray.	

Example

In the example below, a rectangle was selected and is highlighted yellow. A center point of rotation was picked to the left of the rectangle. The picture on the right side shows that the starting point was picked inside the rectangle. The cursor has moved to just above the rectangle and its position is marked by a white dot. A temporary line segment goes from the center of rotation to that point. The effects of the rotation on the selected geometry (rectangle) are displayed in black.

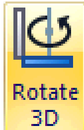


Once the final point is selected, the geometry is rotated and all the temporary line segments, circles, rays, and points disappear. The final result is a rotated rectangle.

Related Topics

Transform Ribbon Bar

5.10.3 Rotate 3D



Rotate selected objects around a picked point a given angle.

Command Prompts

3D Rotate by mouse:: Enter number of copies or Pick rotation axis base point

3D Rotate by mouse:: Pick axis head point

3D Rotate by mouse:: Enter angle or reference point

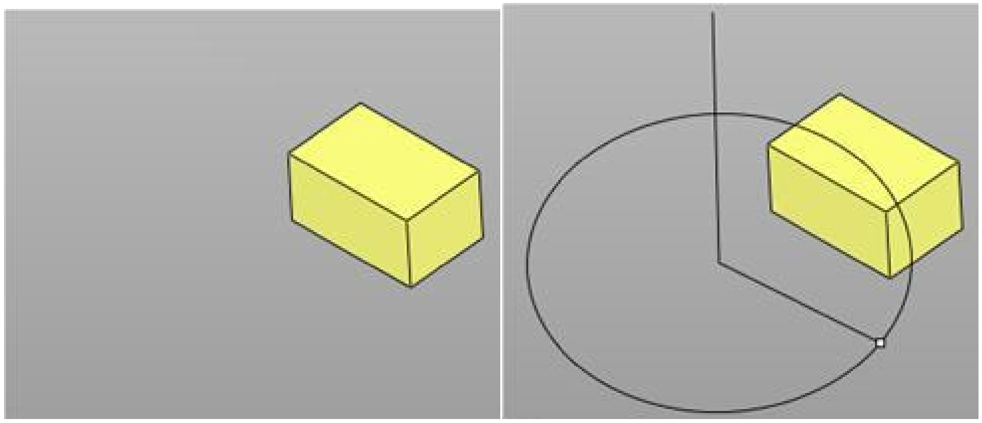
3D Rotate by mouse:: Enter second reference point

Basic Procedure

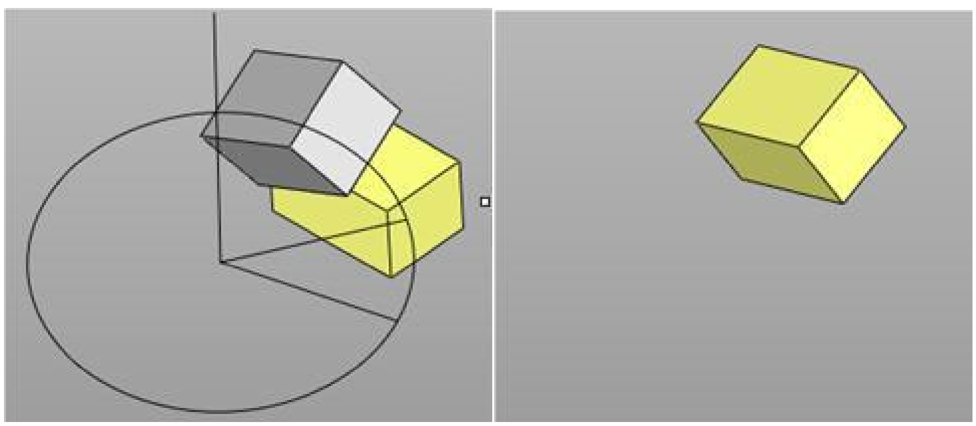
	Screen Activity	Comments
Step 1	Pick objects to be rotated. They can be pre-selected.	
Step 2	Type in the number of copies to be made. If no copies are to be made, just go to step 3.	
Step 3	Pick the axis basis point or type in the coordinates into the Command Input Bar.	
Step 4	Pick the axis head point or type in the coordinates. A temporary segment will connect the axis base and axis head points.	
Step 5	Pick a starting point or type in the coordinates. A temporary ray from the center of rotation going through the starting point will be made.	
Step 6	Pick an ending point or type in the coordinates. A temporary ray from the center of rotation going through the ending point will be made. The angle of rotation will be measured from the starting ray	

Example

In the example below, a box was selected and is highlighted yellow. The axis base point and head point were picked. The axis is the vertical line to the left of the box. The starting point for the angle is being picked in the second picture. The cursor location is the white dot below the box at the intersection of the rotary circle and a ray from the axis base point.



In the first image below, the starting point was picked. The current cursor position is to the right of the selected box and is marked by a white dot. The effects of the rotation are displayed by the gray box. Once the endpoint is picked the geometry is rotated and all the temporary line segments, circles, rays, and points disappear.



Related Topics

[Transform Ribbon Bar]

5.10.4 Scale

This function will interactively scale selected 2D/3D geometry about a point. The scale factor can be directly keyed-in to the Command Bar or will be determined by the ratio of two user-defined radial distances. This method of scaling will maintain a constant aspect ratio of the part model size.

Command Prompts

Graphical Scale:: Enter scale center point

Graphical Scale:: Enter first reference point or scale factor

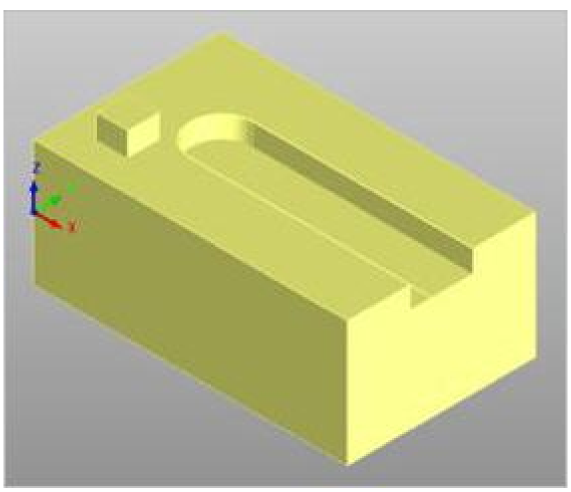
Graphical Scale:: Enter second reference point

Basic Procedure (also see Example below)

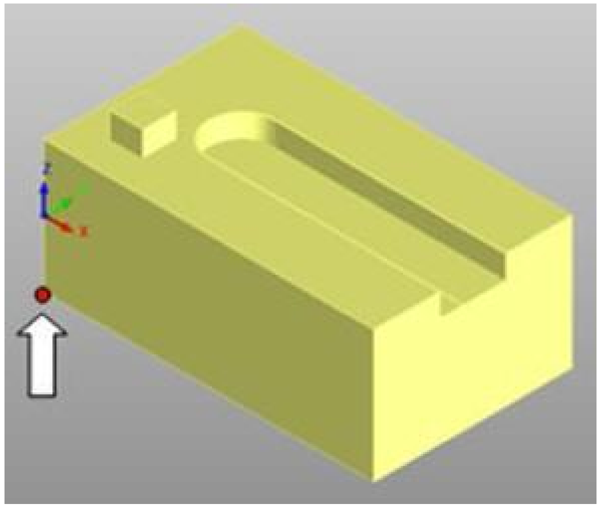
	Screen Activity	Comments
Step 1	Select Graphical Scale from Transform menu.	None
Step 2	Select the 2D and/or 3D geometry objects that you want to be scaled.	
Step 3	Enter the scale reference point (or scale focus point).	
Step 4	Determine the scaling factor (factor or distance ratios).	
Step 5	Select a second point for scaling.	
Step 6	ESC or Right Mouse Button to quit.	

Example

1. Select the 2D and/or 3D geometry objects that you want to be scaled. They may also be pre-selected. After the desired geometry is selected and right-click to close the selection.

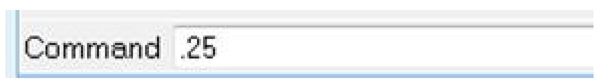


2. Enter the scale reference point (or scale focus point). This can be any point on the model or the grid or you can key-in World coordinates, etc. Scaling will be done toward or away from this point.

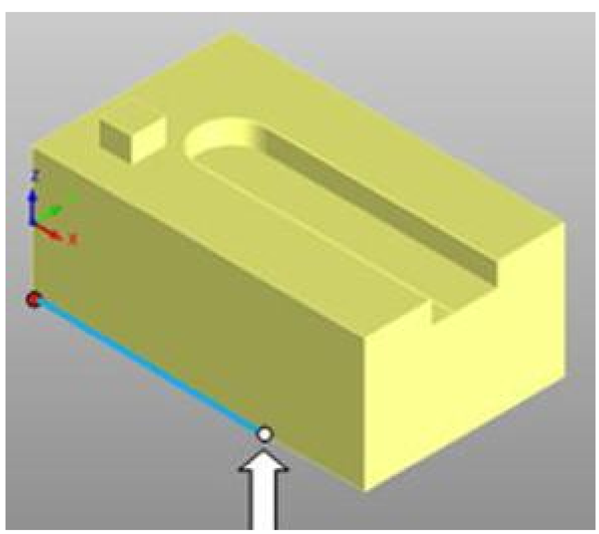


3. Determine the scaling factor in one of two ways described below:

   a. *Enter scale factor*: A scale factor (decimal value) may be directly entered into the Command Bar. When this value is accepted, the geometry will immediately be scaled and the function will end.

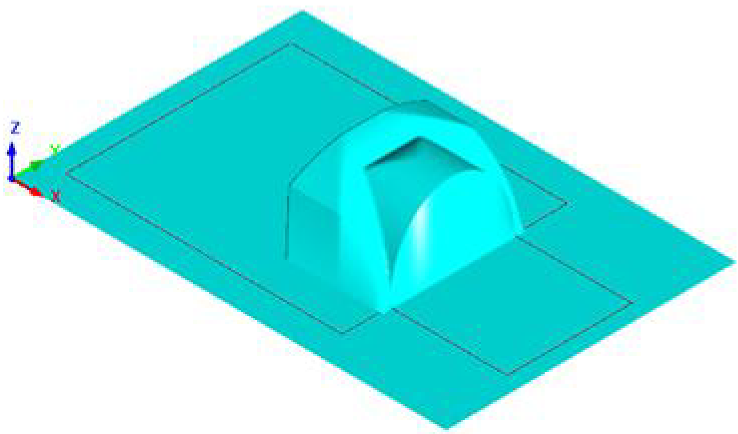


   b. *Distance ratios*: Select a point away from the scale reference point (from **Step 3).** The distance between this point and the scale reference point represents 100% or the full scale of the selected geometry.



4. Select a second point for scaling. The distance between this second point and the scale reference point (from **Step 3**) is compared to the first distance and this ratio determines the scale factor for the geometry. If this second point is selected using

the cursor, you will see the geometry being scaled dynamically until this second point is finally determined.



5. Finally, after the second point is picked, the geometry will be scaled and the function will end. Scaling larger or smaller are both permitted. The function can be canceled at any time before completion by using ESC or the Right Mouse Button.

Related Topics

Transform Ribbon Bar

5.10.5 Rectangular Scale

This function will scale selected 2D/3D geometry by interactively changing the size of a 'minimum rectangle' around the geometry. This scaling method performs 2D scaling oriented to the C-Plane and does not maintain a constant aspect ratio of the part model size.

Command Prompts

Rectangle Scale:: Select objects - hit right mouse to end

Rectangle Scale:: Move corners of rectangle to scale. Enter when done. ESC/Right-Mouse to quit

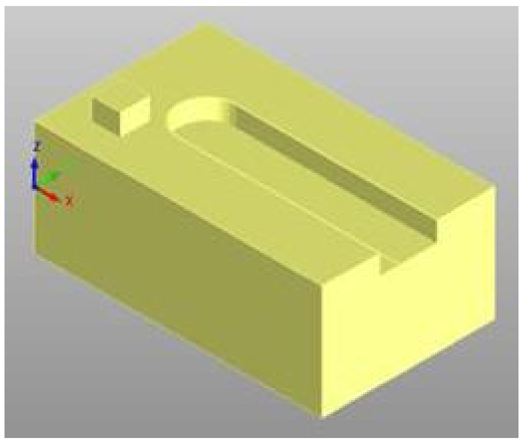
Basic Procedure

	Screen Activity	Comments
Step 1	Select Rectangular Scale from Transform menu.	None
Step 2	Select the geometry objects to scale.	

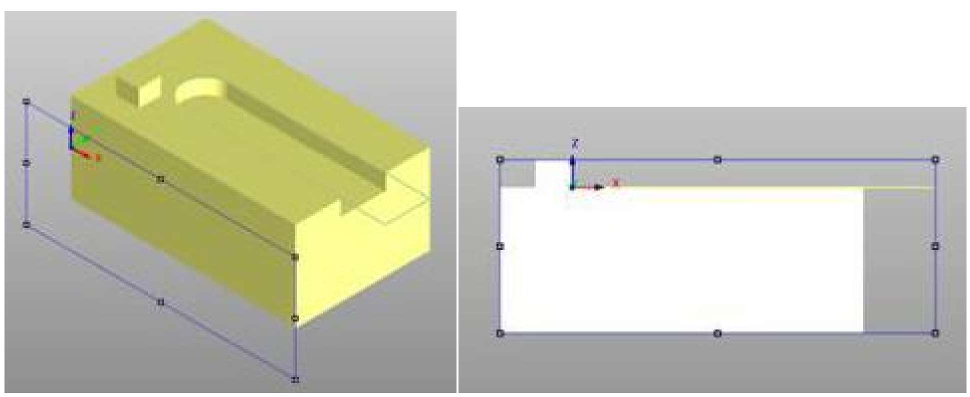
Step 3	Interactively move the rectangle corners/mid-points to reshape the rectangle.	
Step 4	ESC or Right-click to quit.	

Example

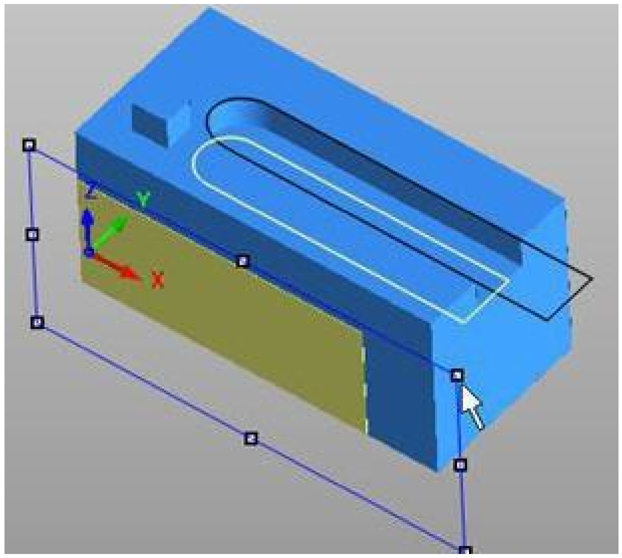
1. Select the 2D and/or 3D geometry objects that you want to be scaled. They may also be pre-selected. After the desired geometry is selected, hit the Right Mouse Button to close the selection.



2. VisualCAD will immediately create a minimum rectangle in the C-Plane that surrounds the selected geometry.



3. Interactively move the rectangle corners/mid-points to reshape the rectangle. As the rectangle is changed, a temporary duplicate of the geometry will be shown to help you visualize the resultant model change. When all changes are made to the rectangle, select ENTER to accept the changes or ESC/Right Mouse Button to cancel the function.



Related Topics

Transform Ribbon Bar

5.10.6 Move ...

All selected items will be moved in 3D. The direction and distance to move is specified by picking a starting and ending point. All selected items will be moved that direction and distance. A dialog will appear where you can input parameters for the move.

Command Prompts

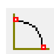
Select objects and hit Right Mouse Button/Enter when done

Basic Procedure

	Screen Activity	Comments
Step 1	From the Move select text by mouse dialog box, select Pick Objects and select the objects to move.	
Step 2	Enter the Move From XYZ coordinates or select the Pick From Point button and then select a point.	

Step 3	Enter the Move To XYZ coordinates or select the Pick To Point button and then select a point.	
Step 4	Select the Pick Translation Axis button to define the translation axis.	Optional
Step 5	Enter the Number of Copies and then pick OK to complete the Move.	

Dialog Box: Move select text by mouse



Number of objects picked

The number of items to be moved is displayed. If more items need to be moved, use the Pick Objects button.

Move From / Move To

The starting point is defined in the Move From area, and the ending point is defined in the Move To area. The coordinates can be entered or a point can be picked. All picked points are projected down into the view's construction plane.

Pick Translation Axis

The Pick Translation Axis can be used to define the starting and ending points. Once the button is pushed, you make a temporary line segment: the endpoints of that segment are the start and end points of the move.

Number of Copies

The Number of Copies allows you to make additional copies of the objects, each separated by the move vector. For example, a point at the origin is moved from (0,0,0) to (5,0,0) with three copies. The result is four points at (0,0,0), (5,0,0), (10,0,0), and (15,0,0). The original point is unaffected, and the other points were copied using the movement vector.

Related Topics

Transform Ribbon Bar

5.10.7 Rotate ...

All selected objects will be rotated in 3D. The axis and angle of rotation are specified, and all selected items will be that angle about the axis. Refer to the Basic Procedure below. Note additional information under the Dialog Box below.

Command Prompts

Select objects and hit Right Mouse Button/Enter when done

Basic Procedure

	Screen Activity	Comments
Step 1	From the Rotate selected objects dialog box, select Pick Objects and select the objects to rotate.	
Step 2	Enter the Rotate About XYZ coordinates or select the Pick About Point button and then select a point.	Default is 0,0,0 origin.
Step 3	Enter the Axis Vector XYZ coordinates or select the Pick Rotation Axis button and then select a start point and end point to define the axis.	
Step 4	Enter the Rotation Angle.	
Step 5	You can Copy and Rotate at the same time if desired. Enter the Number of Copies and then pick OK to complete the Rotate.	Optional

Dialog Box: Rotate selected objects

Number of objects picked

The number of items to be rotated is displayed. If more items need to be rotated, use the Pick Objects button.

Rotate About

The point of reference is defined in the Rotate About section. The coordinates can be entered or an About Point can be picked.

Axis Vector

The axis of rotation is defined with a point and a vector. The point allows the axis to be offset from the origin. The vector is parallel to the axis, and the direction of the vector tells which rotation is a positive angle. This is the Right Hand Rule. For example, an axis of rotation at the origin (point is (0,0,0)) and a vector (0,0,1) has positive angle going counterclockwise from the top view (from positive x-axis to positive y-axis). Alternatively, if the axis of rotation was at the origin (0,0,0) but went the other way with a vector (0,0,-1) has positive angle going clockwise (from positive x-axis to negative y-axis).

Rotation Angle

The rotation angle is measured from the current location.

Number of Copies

This allows you to make additional copies of the objects, each separated by the rotation angle. The original object is unaffected, and the other points were copied using the rotation angle.

Related Topics

Transform Ribbon Bar

5.10.8 Part Orient ...

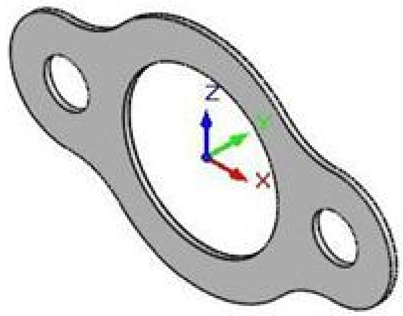
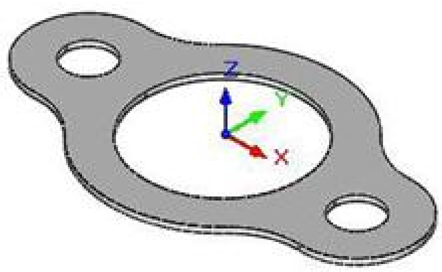
This dialog allows user to orient the part to have it aligned parallel to World Coordinate System (WCS). The dialog offers a convenient way of selecting each of the six principal directions to set the cutting direction.

Command Prompts

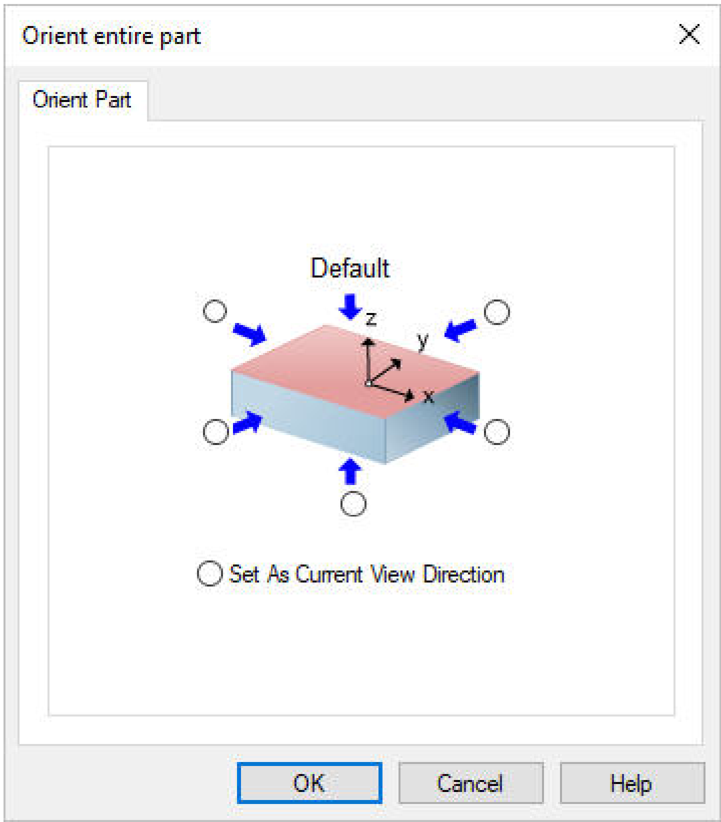
None.

Basic Procedure

	Screen Activity	Comments
Step 1	Open part file.	

		
Step 2	Select Part Orient... from Transform menu to display the Orient Entire Part dialog box.	
Step 3	From the dialog box, select the WCS direction to orient the part to.	
Step 4	Select to Set as Current View Direction if desired.	Optional
Step 5	Pick OK and the part is now oriented as shown below. 	

Dialog Box: Rotate selected objects



World Coordinate System (WCS)

Once you select a WCS direction, all of the geometry will be rotated to an orientation where the selected cutting direction becomes the negative Z axis.

Set as Current View Direction

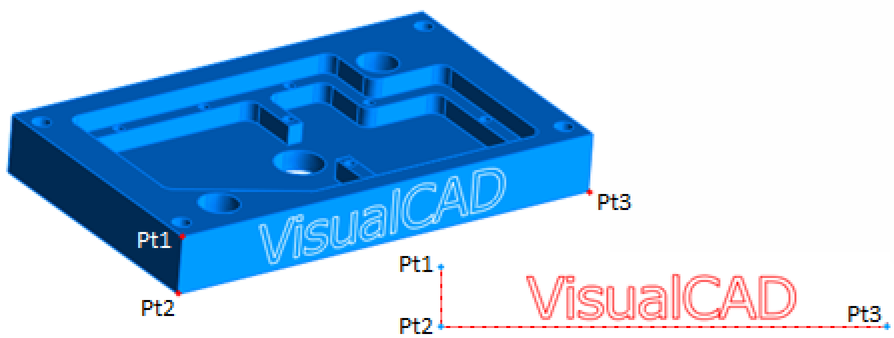
Orients the part parallel to the current view orientation.

Related Topics

Transform Ribbon Bar

5.10.9 Orient by 3 Pts

This command allows you to transform the currently selected objects using three reference points and three target points.



Command Prompts

Pick or enter first reference point.

Pick or enter second reference point.

Pick or enter third reference point.

Pick or enter first target point.

Pick or enter second target point.

Pick or enter third target point.

Basic Procedure

	Screen Activity	Comments
Step 1	Pick three reference points in relation to the selected objects.	The selected objects are re-oriented.
Step 2	Pick three target points to re-orient the objects.	

Related Topics

[Transform Ribbon Bar]

5.10.10 Scale ...

All selected objects will be magnified, shrunk, or stretched the same way. A dialog will appear where you can input parameters for the scaling.

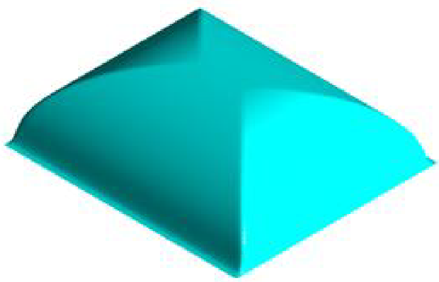
Command Prompts

None.

Basic Procedure

	Screen Activity	Comments
Step 1	Select Scale... from the Transform menu to display the Scale selected objects dialog box.	
Step 2	The number of objects is displayed. Use Pick Objects to add more to the selection.	
Step 3	Select either Uniform or Non-Uniform Scaling and enter the XYZ scale factors.	
Step 4	Pick OK to Scale the selected objects.	Optional

Dialog Box: Scale selected objects



Scale Selected Objects

Number of objects picked

The number of objects to be scaled is displayed. If more items need to be scaled, use the Pick Objects button.

Scale Units

This converts the units associated with the geometry to a different set of units. Conversions can go from millimeters to inches or from inches to millimeters. The object's size remains constant, but the numbers associated with the size change. A one-inch object will be scaled into a 25.4 mm object. The units of the display view are not affected, so the object appears to grow or shrink. When the units of the display are changed, the object will appear as it did before the scaling.

Uniform scaling

This will magnify or shrink all dimensions equally. A scaling of 1.0 is no change. A scaling of 0.5 will make the object half as small, and a scaling of 3 will make the object 3 times bigger.

Non-Uniform Scaling

This will stretch the object. The coordinates of the object are what are scaled. This will cause an object not at the origin to move. For example, if a point is at (5,0,0) and is uniformly scaled by a factor of 3, the point will be at (15, 0, 0).

Related Topics

Transform Ribbon Bar

5.10.11 Mirror ...

Reflects all selected objects in a plane parallel to one of the principal planes. A dialog will appear where you can input parameters for the mirroring.

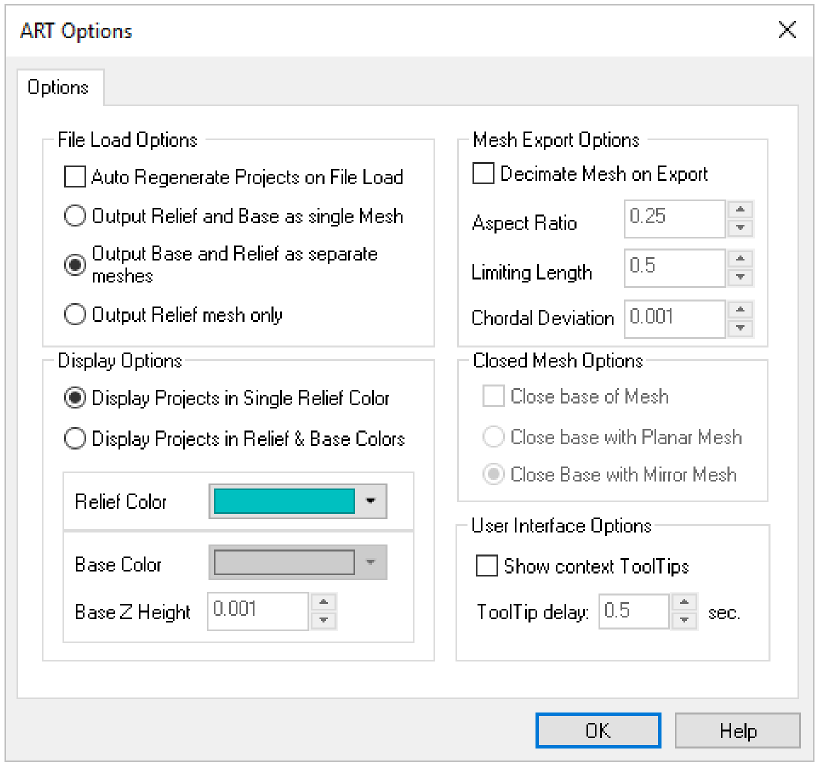
Command Prompts

None.

Basic Procedure

	Screen Activity	Comments
Step 1	Select Mirror... from Transform menu.	
Step 2	The number of objects is displayed. Use Pick Objects to add more to the selection.	
Step 3	Define the Mirror Plane and Pick (P) point.	
Step 4	Check the box to mirror a copy if desired.	Optional

Dialog Box: Mirror selected objects



Mirror Selected Objects

Number of objects picked

The number of objects to be mirrored is displayed. If more items need to be scaled, use the Pick Objects button.

Mirror Plane / Point (P)

Mirror Plane is parallel to one of the principal planes: XY Plane, XZ Plane, or YZ Plane.

Point (P) is defined to allow the planes to be offset. This point is on the mirroring plane. You may also select Pick Plane Point.

Create Copy

This allows a copy to be made and leaves the original object in place and puts a copy at the mirrored location.

Related Topics

[Transform Ribbon Bar]

5.10.12 Array ...

All selected objects will be copied into a uniform grid of objects offset from each other specified distances in x, y, and z. A dialog will appear where you can input parameters.

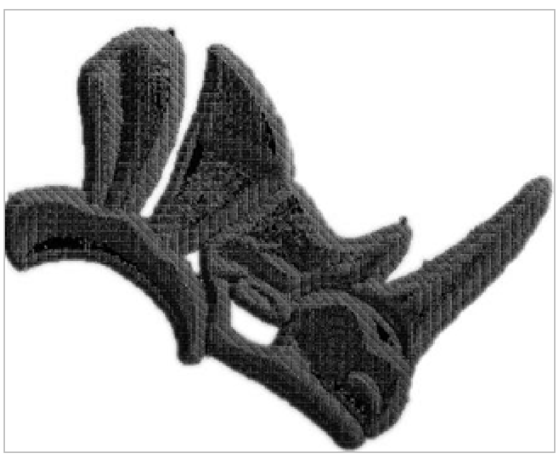
Command Prompts

None.

Basic Procedure

	Screen Activity	Comments
Step 1	Select Array... from Transform menu.	
Step 2	The number of objects is displayed. Use Pick Objects to add more to the selection.	
Step 3	Define the XYZ Spacing and # of Copies.	
Step 4	Pick OK to complete the Mirror command.	

Dialog Box: Array selected objects



Array selected objects

Number of objects picked

The number of objects to be arrayed is displayed. If more items need to be scaled, use the Pick Objects button.

XYZ Spacing

The XYZ Spacing is the separation of a given point on an object and the corresponding point on the copied objects. The object's size is not considered during this command.

of Copies

There can be a different Number of X, Y or Z Copies made in each direction.

Notes

1. If the spacing numbers are all positive, the original object will be the one with the lowest (x, y, z) coordinates.

Related Topics

Transform Ribbon Bar

5.10.13 Polar Array ...

All selected objects will be copied into a uniform ring of objects around a circle. A dialog will appear where you can input parameters.

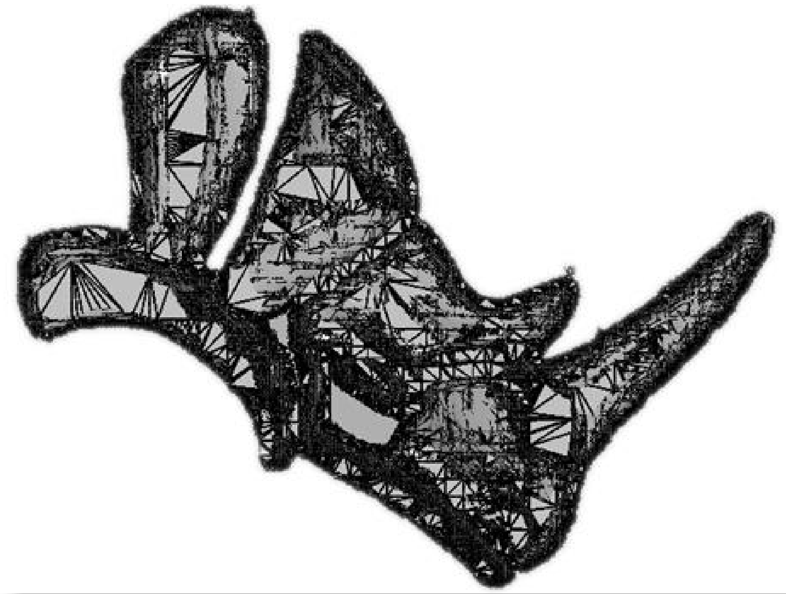
Command Prompts

None.

Basic Procedure

	Screen Activity	Comments
Step 1	Select Polar Array... from Transform menu.	
Step 2	Enter or pick the point to Rotate About.	
Step 3	Enter the Angle to Fill. and total Number of Copies to array.	
Step 4	Pick OK to complete the Polar Array command.	

Dialog Box: Polar array selected objects



Polar Array

Number of objects picked

The number of objects to be polar arrayed is displayed. If more items need to be scaled, use the Pick Objects button.

Rotate About

This is defined with a point and a vector. Pick About Point allows the axis to be offset from the origin.

Axis Vector

This is parallel to the axis, and the direction of the vector tells which rotation is a positive angle. This is the Right Hand Rule. For example, an axis of rotation at the origin (point is (0,0,0)) and a vector (0,0,1) has positive angle going counterclockwise from the top view (from positive x-axis to positive y-axis). Alternatively, if the axis of rotation was at the origin (0,0,0) but went the other way with a vector (0,0,-1) has positive angle going clockwise (from positive x-axis to negative y-axis).

Angle to Fill

The rotation **Angle to Fill** measured from the current location.

Number of Copies

The **Number of Copies** is evenly distributed about the Angle to Fill.

Related Topics

Transform Ribbon Bar

5.10.14 Along Path ...

All selected objects will be copied along a path curve. A dialog will appear where you can input parameters.

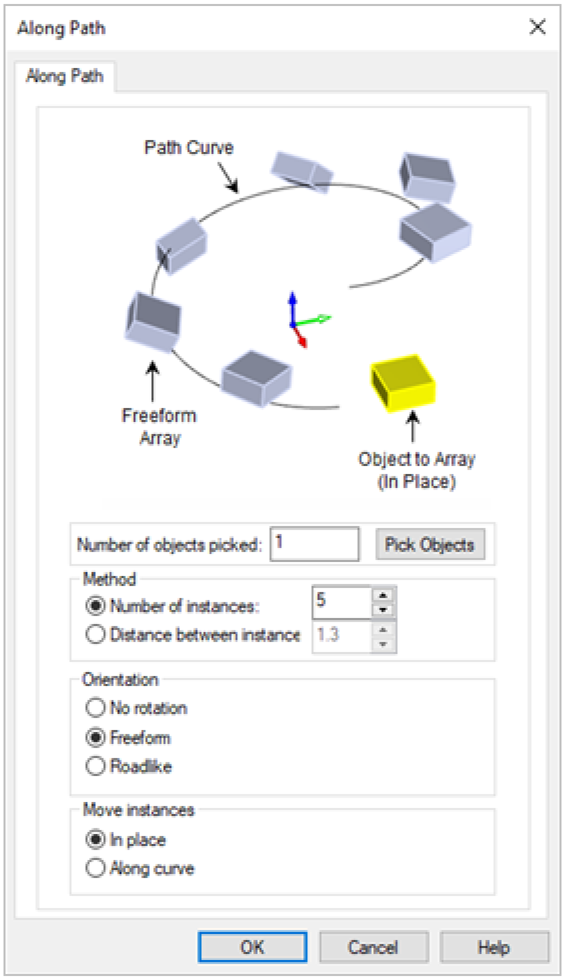
Command Prompts

None.

Basic Procedure

	Screen Activity	Comments
Step 1	Select Along Path... from Transform menu.	
Step 2	Select options from the dialog for Method, Orientation and Move Instances.	
Step 3	Pick OK to complete the Polar Array command.	

Dialog Box: Polar array selected objects



Trabsform Along Path

Number of objects picked

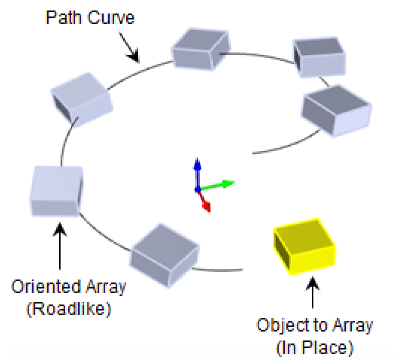
The number of objects to be transformed along the path curve. If more items need to be scaled, use the Pick Objects button.

Method

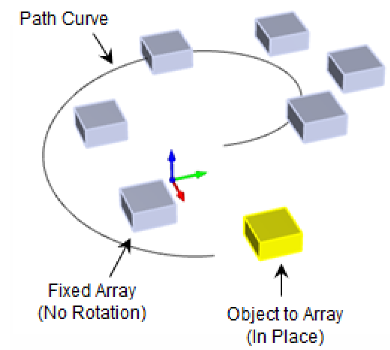
This option allows you to specify the distance between the copied objects. You either specify the number of instances or specify the number of objects.

Orientation

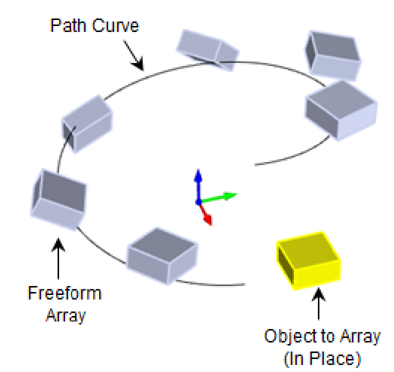
This specifies how the object instances should be oriented. The illustrations below show the different options. There can be No Rotation where the objects are fixed in place. There can also be a Freeform or a Road like orientation. Again the illustrations below provides some guidance on how the transformation will apply.



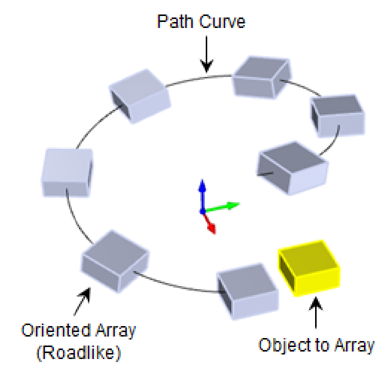
Array In Place Road Like



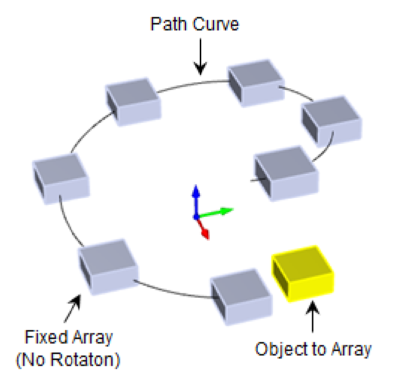
Array In Place No Rotation



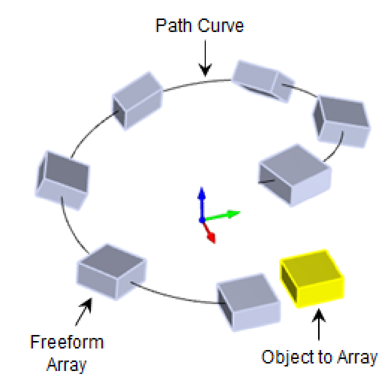
Array In Place Freeform



Array Along Path Roadlike



Aray Along Path No Rotation



Array Along Path Freeform

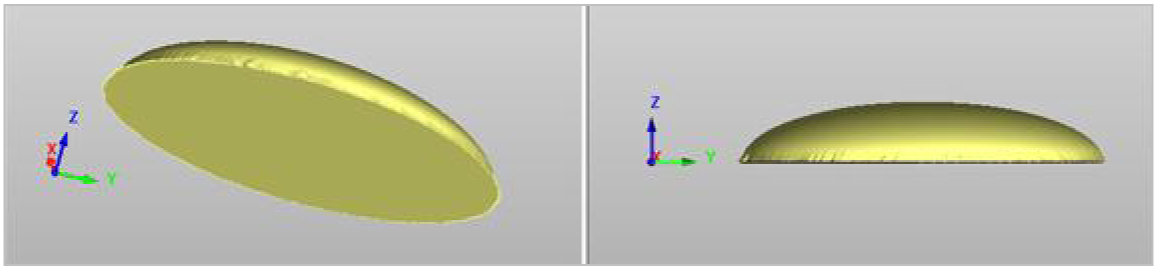
Move Instances

This specifies how the objects are moved. Select either In Place or Along Curve. Refer to the illustrations above for guidance.

Related Topics

Transform Ribbon Bar

5.10.15 Graphical Manipulator



Displays the graphical manipulator widget on selected object(s) facilitating move, scale, and rotate transformations around the graphical manipulator origin. This can be toggled on and off by selecting Manipulator on the Status bar.


Manipulator On


Manipulator Off

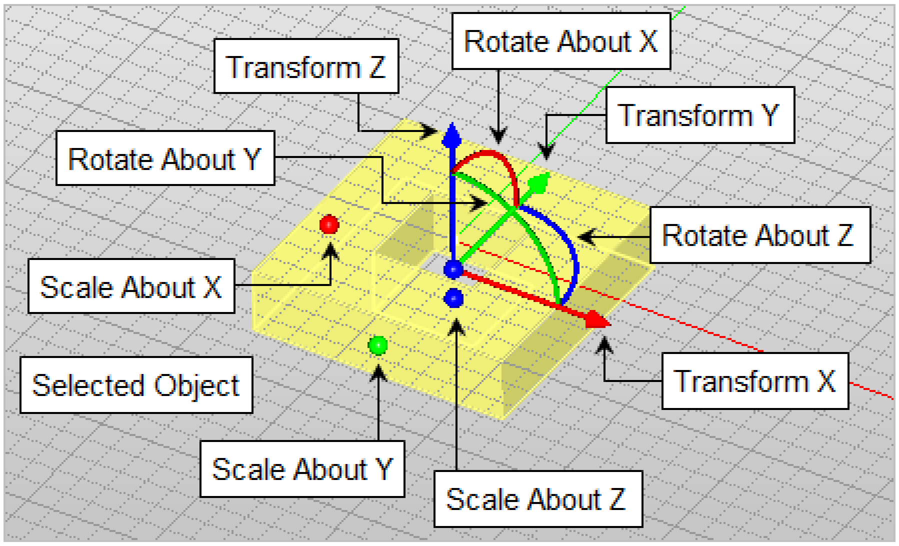
About the Graphical Manipulator

Graphical Manipulator can be used for graphical transform of objects by selecting the icon on the Status Toolbar. This feature can be used to move or rotate objects. The orientation of the graphical manipulator is in the same orientation as the World Coordinate System (WCS).

The arrows displayed on the graphical manipulator can be selected and a numerical value placed to move the geometry in either X, Y or Z direction. Or the arrows can be selected and simply dragged in either the X, Y or Z direction.

To rotate the geometry, use the arcs on display of the graphical manipulator. The arcs displayed on the graphical manipulator can be selected and a numerical value placed to rotate the geometry in either X, Y or Z direction. Or the arcs can be selected and simply rotated by dragging with the left hold of mouse.

Graphical Manipulator can also be enabled by selecting Transform on the Menu bar and then selecting Manipulator.



Graphical Manipulator

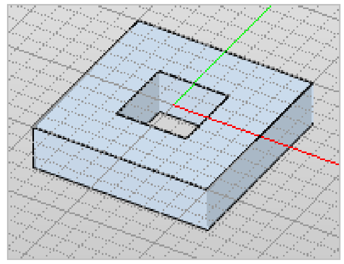
Command Prompts

Manipulator:: Move/Rotate object(s). Enter when done. Esc/RButton to exit.

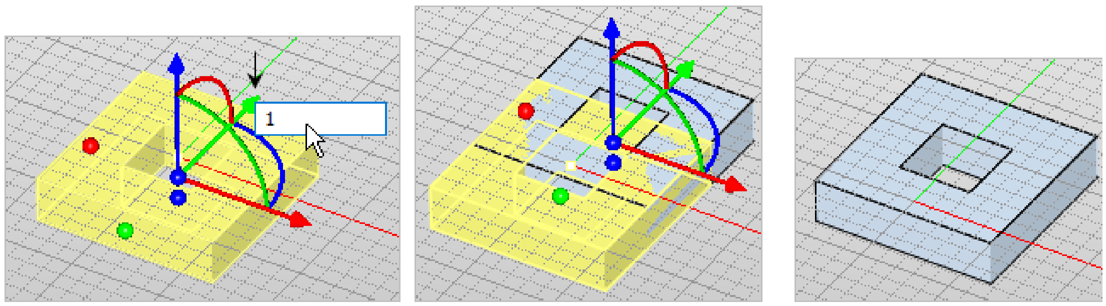
Basic Procedure

	Screen Activity	Comments
Step 1	Select an object to manipulate.	
Step 2	Select Manipulator from Transform menu or from the Status Toolbar. to display the manipulator widget.	
Step 3	Left-click on one of the manipulator widgets. There are three for each axis. They are Scale, Transform and Rotate. Refer to the illustration above.	
Step 4	Enter a value in the pop-up window and press ENTER. Alternately you can drag the widget to dynamically transform the object.	
	Left click on the origin of the Manipulator to display additional options. See below.	Optional

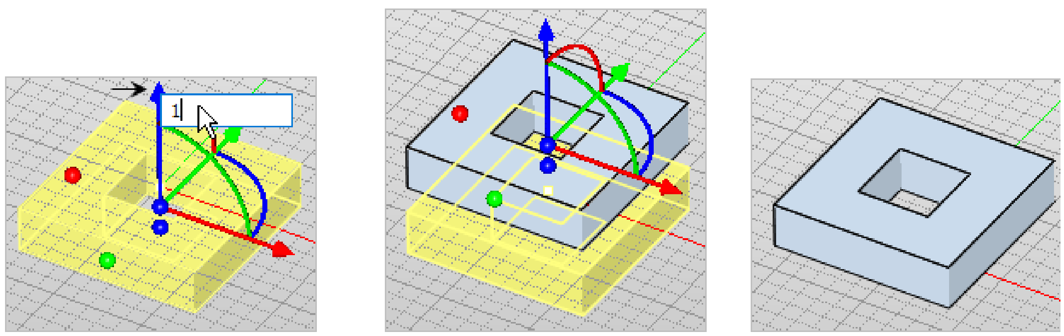
Examples



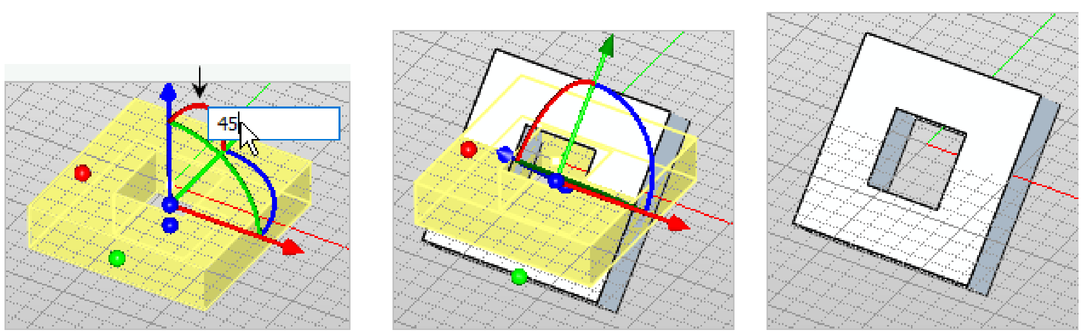
Original Geometry



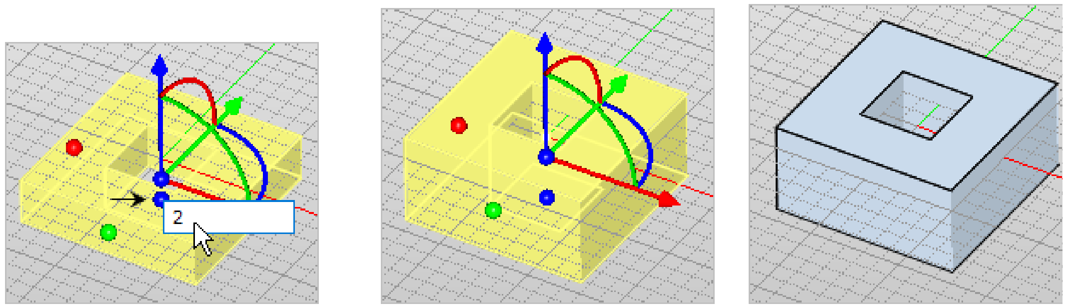
Move 1" along the positive Y Axis



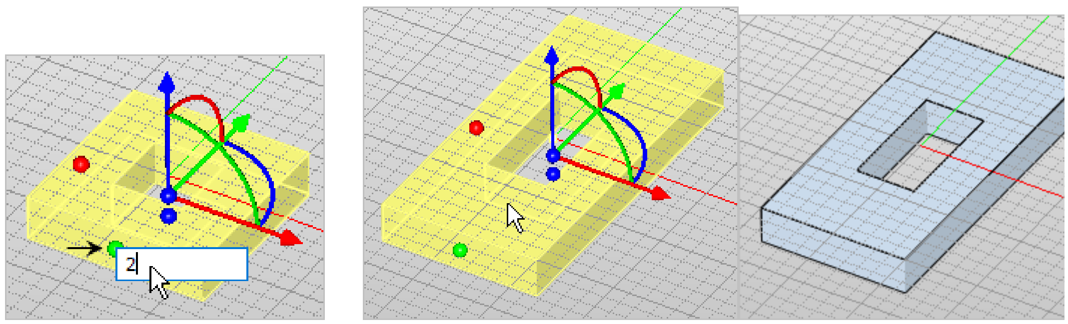
Move 1" along the positive Z Axis



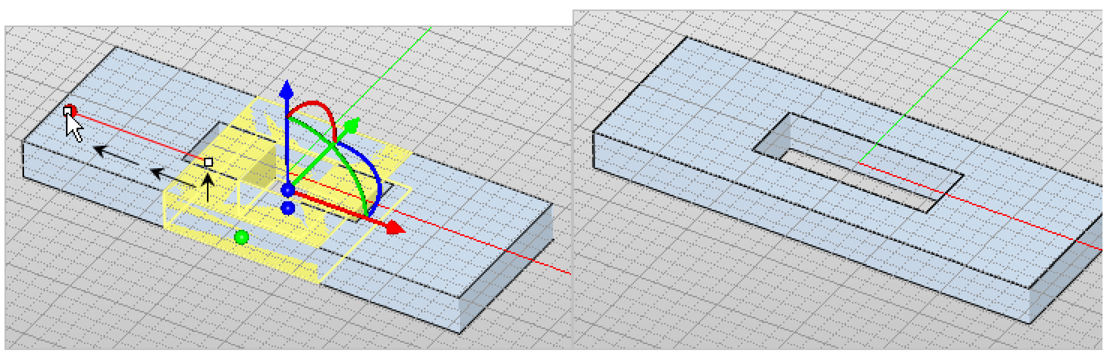
Rotate 45 degrees about the positive X Axis



Scale by 2 about the positive Z Axis



Scale by 2 about the positive Y Axis



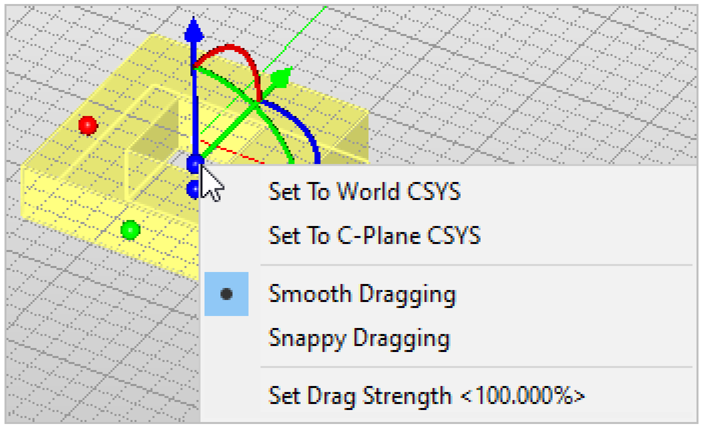
Scale by Dragging equal distance about the X Axis

Graphical Manipulator options

Graphical Manipulator options can be invoked by left clicking on the origin of the manipulator.

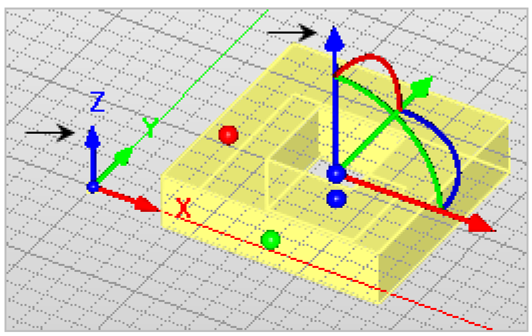
This allows for the selection of additional manipulator options such as Set to World CSYS, Set to CPlane CSYS, Smooth Dragging, Snappy Dragging and Set Drag Strength<100.000%>

The below example shows the options after left click of mouse on origin of the manipulator.



Set to World CSYS

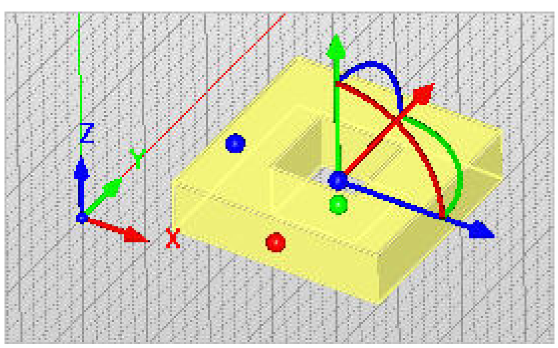
Set to World CSYS - In the below example, the Graphical Manipulator is set to the World Coordinate System (WCS).



Set to C-Plane CSYS

This sets the Graphical Manipulator to the Construction Plane Coordinate System

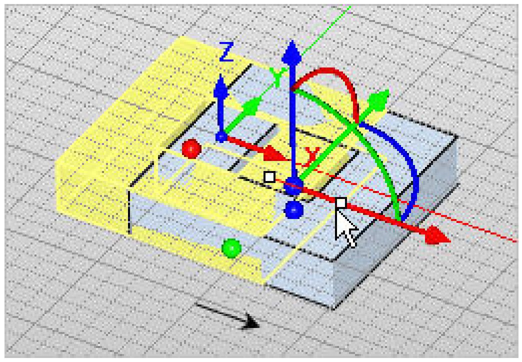
In the below example, the C-Plane is set to 45 degrees on the X axis coordinate and the manipulator is also set to that CPlane.



Smooth Dragging

This allows the Graphical Manipulator to glide freely when being manipulated.

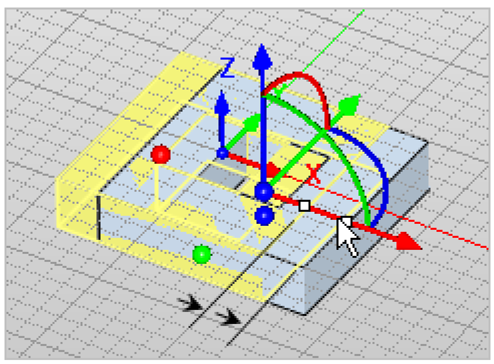
In the below example, please note that the geometry is gliding freely in between cells.



Snappy Dragging

This allows the Graphical Manipulator to move in increments by snapping from cell to cell for each move.

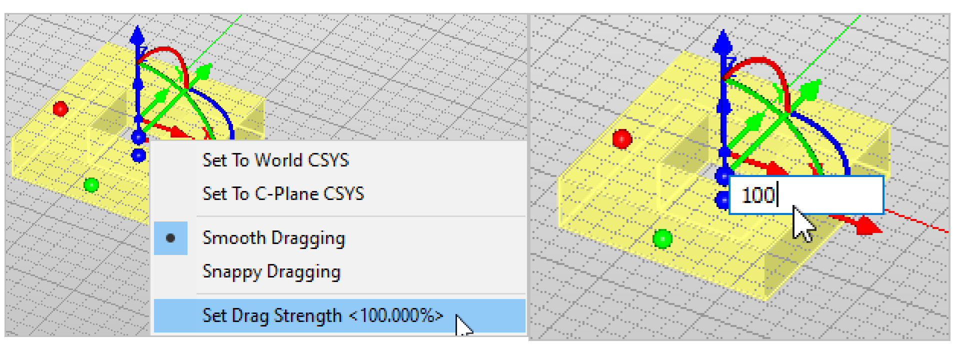
In the below example, please note that the geometry is snapping directly on to the cells



Set Drag Strength<100.000%>

This value by default is set to 100%. This sets the percentage of the speed by which the Graphical Manipulator moves when being dragged or rotated. The lower the value, the slower the speed.

In the below example the drag strength is set at 100%.



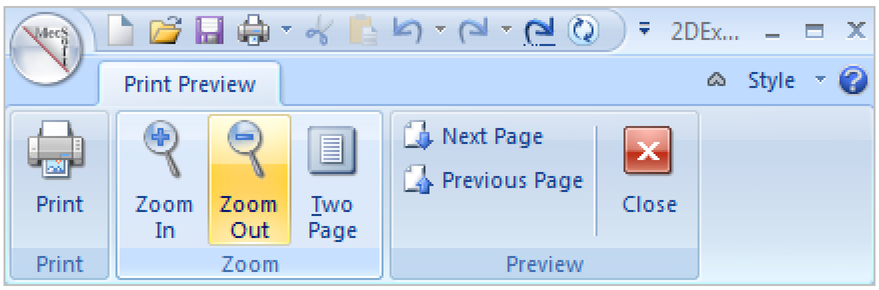
Related Topics

[Transform Ribbon Bar]

5.11 Print Preview

The Print Preview Ribbon Bar contains Print, Zoom, and Preview panes and associated functions. This Ribbon Bar is only displayed when the Print Preview command is executed.

The Print Preview Ribbon Bar



Print Preview Ribbon Bar

Related Topics

The Ribbon Bar

VisualCAD Menu

Print ...

Print Preview ...

Print Setup ...

Status Toolbar

Status Tool Bar

Related Topics

[Coordinate input]()

[Current Position]()

[Part Units]()

[Object Snap Control]()

[Graphical Manipulator]()

[Visual Aids]()

[Properties]()

[Layer Manager]()

[Options]()

[Background Properties]()

6.1 Coordinate input

Coordinate input for geometry creation can be set to World coordinates or Construction Plane (C-Plane coordinates). To toggle between WORLD and C-PLANE, click WORLD on status bar.

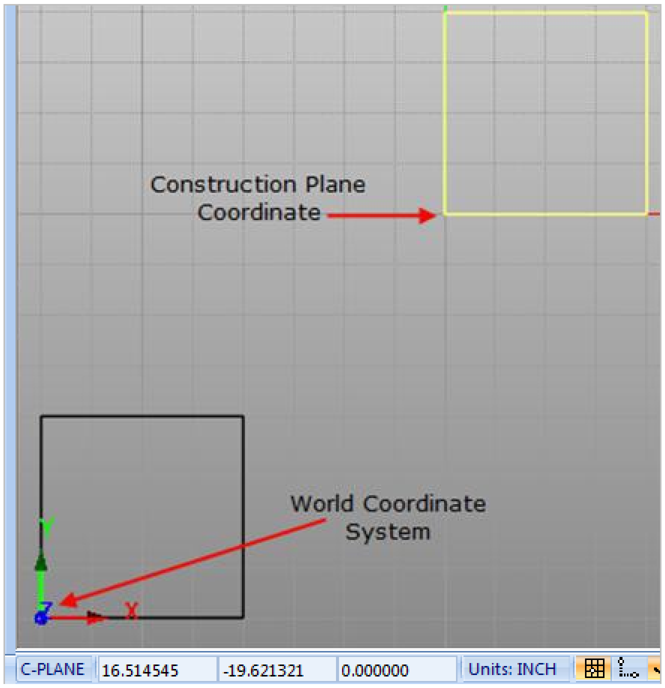
WORLD World Coordinates
C-PLANE Construction Plane Coordinates

Construction Plane coordinates

Construction Plane coordinates can be used for inputting coordinates for geometry creation. This comes in handy when the C-plane is transformed from the WCS orientation. User can toggle between C-PLANE and WORLD coordinates by selecting C-PLANE or WORLD icon located on the Status bar.

Example: Construction Plane coordinates

Example above shows the construction plane transformed from the World origin.



Example: Construction Plane coordinates

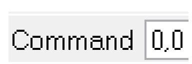
Basic Procedure

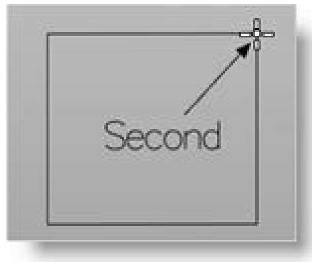
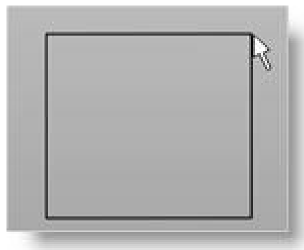
To input coordinates in C-Plane, activate the C-Plane from the status bar by toggling this from WORLD to C-Plane.

Under the Curve pane of the Curve Modeling Ribbon Bar, select the Rectangle icon.

Step 1

Step 2

	Screen Pick	Command Input
Step 1		(Optional) Enter X, Y, [Z] coordinates for the first point. 

Step 2		(Optional) Enter X, Y, [Z] coordinates for the second point (opposite corner). 
Final		

Related Topics

[Status Tool Bar]

6.2 Current Position

Displays information on current settings. It also has controls for the picking of points.

Current Position (X, Y, Z)

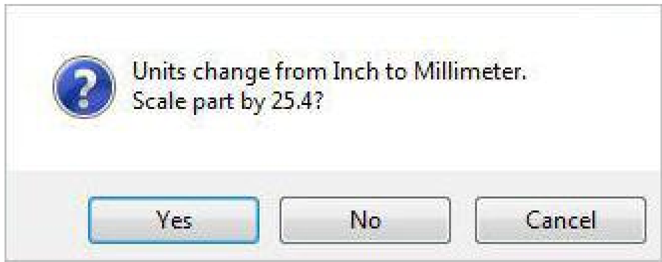
Displays the coordinates of the current cursor position in the view. The cursor is projected down into the current view's construction plane.

Related Topics

[Status Tool Bar]

6.3 Part Units

Displays the current model units. If it is picked, it will attempt to change units from inches to millimeters or from millimeter to inches. If objects are already in the model, a dialog will verify if you really want to change units. The objects keep the same size but their measured dimensions change.

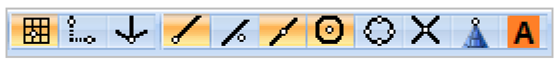


Related Topics

[Status Tool Bar]

6.4 Object Snap Control

The Object Snap controls are located on the [Status Tool Bar]. They allow you to select objects by their snap locations. The available Object Snaps are listed in the Related Topics section below.



Object Snaps

Related Topics

[Status Tool Bar]

[Grid Snap]

[Ortho Snap]

[Origin Point Snap]

[End Point Snap]

[Near Point Snap]

[Mid Point Snap]

[Center Point Snap]

[Quad Point Snap]

[Intersection Point Snap]

Vertex Point Snap

Project C-Plane Snap

6.4.1 Grid Snap

 Points picked near construction plane grid intersection will be snapped to that grid point. This can be toggled on and off by selecting Grid Snap on the Status bar.

Related Topics

Object Snap Control

6.4.2 Ortho Snap

Switches on and off the display of the construction plane in the currently selected view. The default is to show the construction plane (grid). This can be toggled on and off by selecting Ortho Snap on the Status bar.

Ortho Snap On　　Ortho Snap Off

Related Topics

Object Snap Control

6.4.3 Origin Point Snap

Points picked near the origin will be snapped to the origin. This can be toggled on and off by selecting Origin Snap on the Status bar.

Origin Point Snap On　　Origin Point Snap Off

Related Topics

Object Snap Control

6.4.4 End Point Snap

Points picked near the end points of objects will be snapped to that end point. This can be toggled on and off by selecting End Point Snap on the Status bar.

Related Topics

Object Snap Control

6.4.5 Near Point Snap

Points picked near objects will be snapped to the nearest point. The nearest point is the closest point on the object to the cursor. This can be toggled on and off by selecting Near Point Snap on the Status bar.

Near Point Snap On　　Near Point Snap Off

Related Topicscs

[Object Snap Control]

6.4.6 Mid Point Snap

Points picked near the midpoints of a curve will be snapped to that point. This can be toggled on and off by selecting Mid Point Snap on the Status bar.

Mid Point Snap On

Mid Point Snap Off

Related Topicsics

[Object Snap Control]

6.4.7 Center Point Snap

Points picked near the center point of a circle, arc, or closed polygon will be snapped to that point. This can be toggled on and off by selecting Center Point Snap on the Status bar.

Related Topics

[Object Snap Control]

6.4.8 Quad Point Snap

Points picked near the quadrant points of circles and arc will be snapped to that point. Quadrant points are the points on the circle at 0, 90, 180, and 270 degrees. This can be toggled on and off by selecting Quad Point Snap on the Status bar.

Quad Point Snap On

Quad Point Snap Off

Related Topics

[Object Snap Control]

6.4.9 Intersection Point Snap

Points picked near the intersection of curves will be snapped to that point. This can be toggled on and off by selecting Intersection Point Snap on the Status bar.

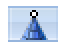
Related Topics

Object Snap Control

6.4.10 Vertex Point Snap

Points picked near the vertex of a mesh will be snapped to that point.


Quad Point Snap On


Quad Point Snap Off

Related Topics

Object Snap Control

6.4.11 Project to C-Plane Snap

All selected points will be projected to the construction plane.


Quad Point Snap On

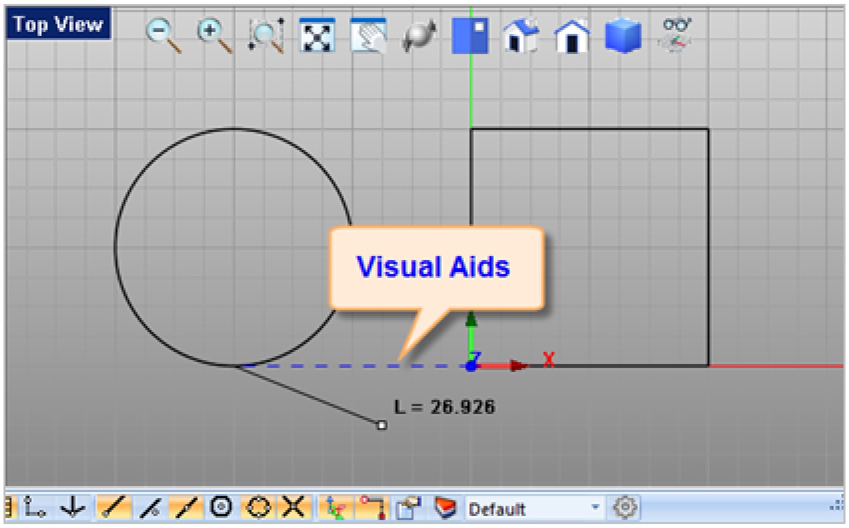

Quad Point Snap Off

Related Topics

Object Snap Control

6.5 Visual Aids

 This provides tools for the construction of geometry.



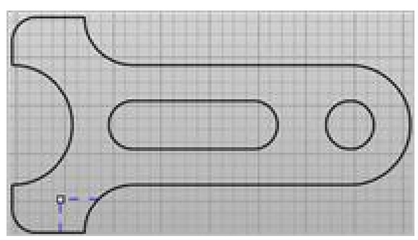
Visual Aids Enabled

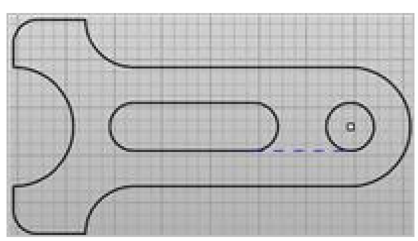
This can be toggled on and off by selecting Visual Aids on the Status bar.


Visual Aids On


Visual Aids Off

This feature can serve as a construction aid or a tool for referencing geometry. With Visual Aids enabled, it will allow viewing of construction lines as it references geometry. This allows for easier visual placement of new geometry. Curve creation using Visual Aids can be enabled by selecting the Visual Aids icon on the Status Toolbar.


The above example shows intersection reference of 2 curves


The above example shows tangent reference to 2 circles

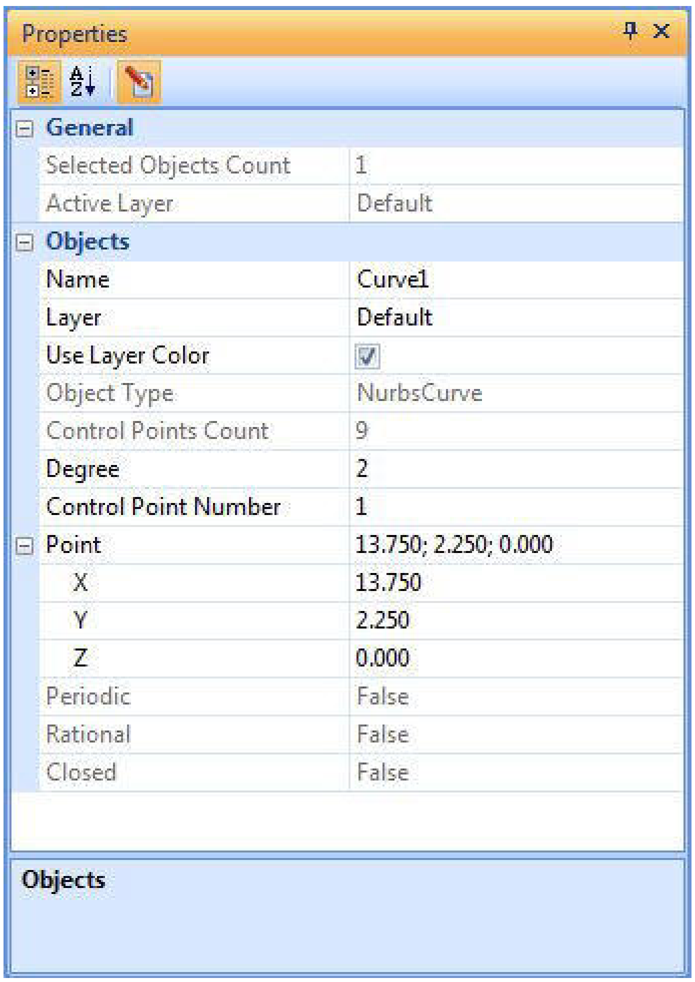
Related Topics

[Status Tool Bar]

6.6 Properties ...

Information about a selected object will be displayed. This includes the name, layer that it's on, the item's color, number of selected objects, object type, and attributes related to the selected object. These properties can be modified.

Object Properties Dialog



Object Properties Dialog

The dialog shows the object name, layer that it's on, the item's color, object type, and attributes related to the selected object. It also shows the number of selected objects and the name of active layer.

The object name is automatically generated when the object was created but can be changed. Points, lines, arcs, circles, curves default to the geometry-type name (i.e.

points, lines, ...) with a number at the end. The number has no significance; it is just an internally generated number. Surface and poly-surfaces are named as Shells and polygonal meshes as Mesh.

The layer that the object is on is displayed.

The color of the object defaults to the construction color of the layer; however, it can be changed. Uncheck the "Use Layer Color" box and click the color bar underneath it. A color-picking dialog will appear. This color change only applies to that object.

This selects which of the existing layers to make the active layer. If that layer was not visible before being selected, it and all of the objects on that layer become visible. Any objects created after that will be the default color for that layer.

The information displayed in the properties dialog can be grouped or listed in alphabetical order.

When 2 or more objects are selected, the property manager displays the selected object count and the active layer information.

Selecting Edit mode allows editing of curve geometry.

Related Topics

Status Tool Bar

Edit Mode

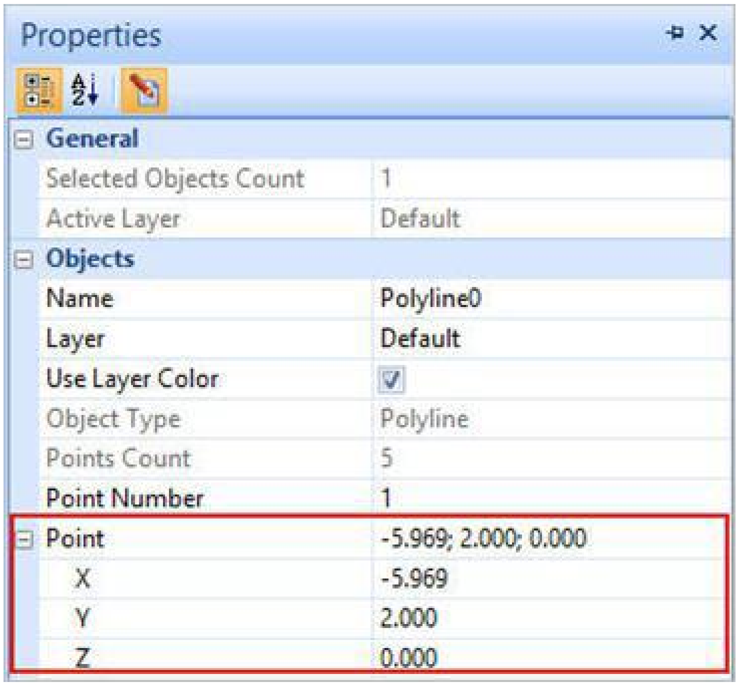
6.6.1 Edit Mode

 Edit Mode can be enabled by clicking on the Edit Mode icon in the Properties dialog and can be toggled on and off by clicking it. With Edit Mode active, the points are visible in the viewport and coordinates displayed in the properties dialog. They are grayed out in the Properties tab if Edit Mode is not active.

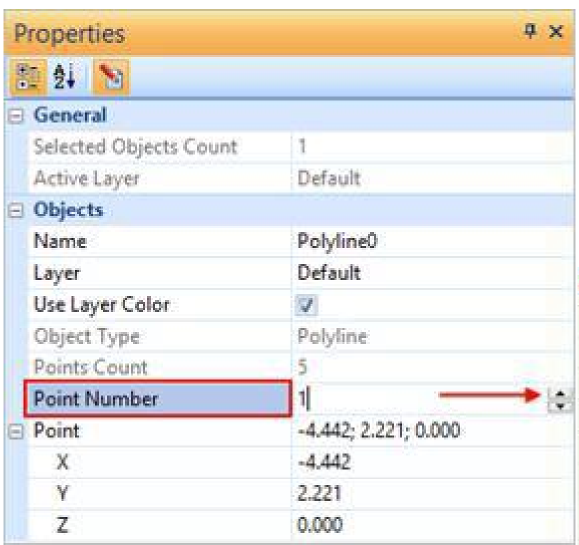
You can use the Properties dialog to parametrically edit certain types of geometry such as the radius of a circle or the length and width of a rectangle. This allows you to quickly change geometry dimension values!

The Properties tab can be located in the Status Toolbar. The X, Y, and Z coordinates of the geometry can be modified by changing the Point values within the Properties tab under Objects or graphically by dragging them in the viewport. You can cycle through all the edit points and modify the X, Y and Z coordinates for each point by using the spinner increment. Cycling through these points highlights the points on the geometry in the viewport.

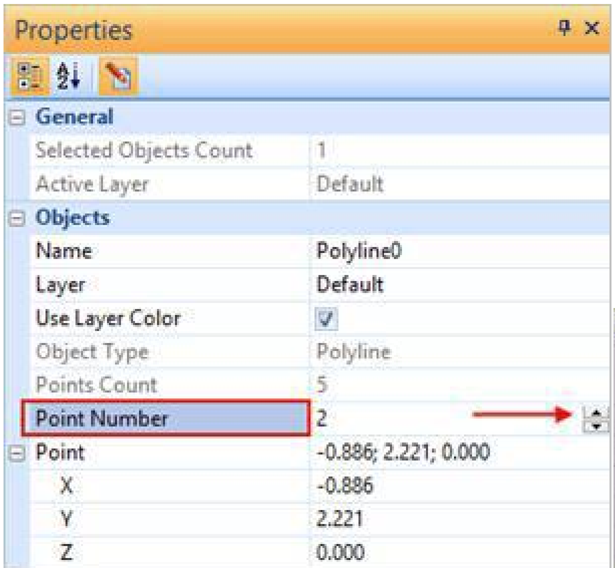
Properties Dialog

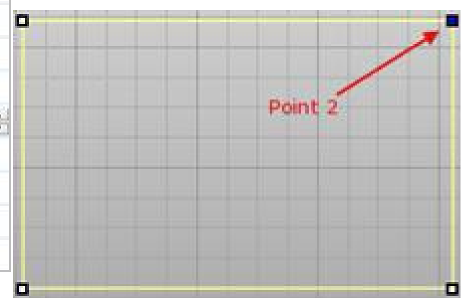


Properties tab



Properties tab - Point Number 1

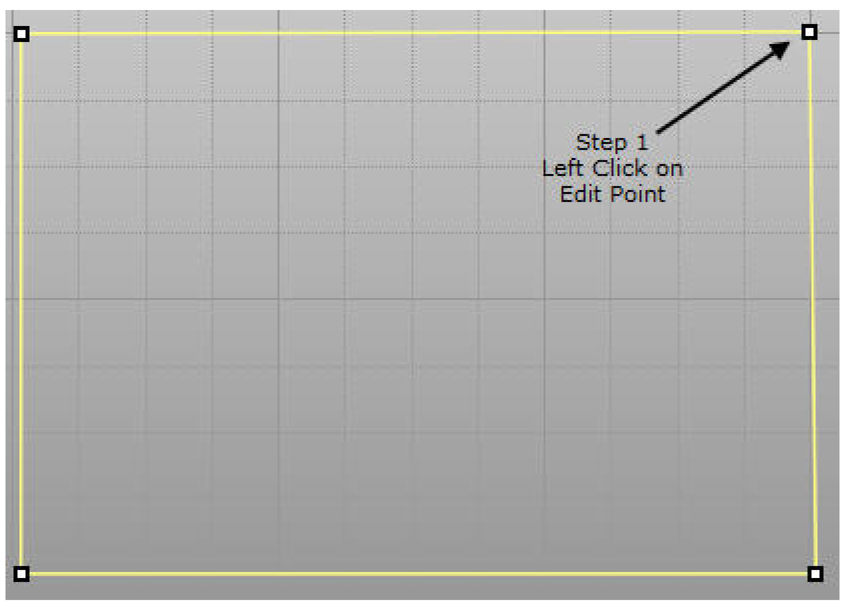




Properties tab - Point Number 2

Basic Procedure

Edit points can also be modified by picking the corner points graphically in the viewport with drag and drop.

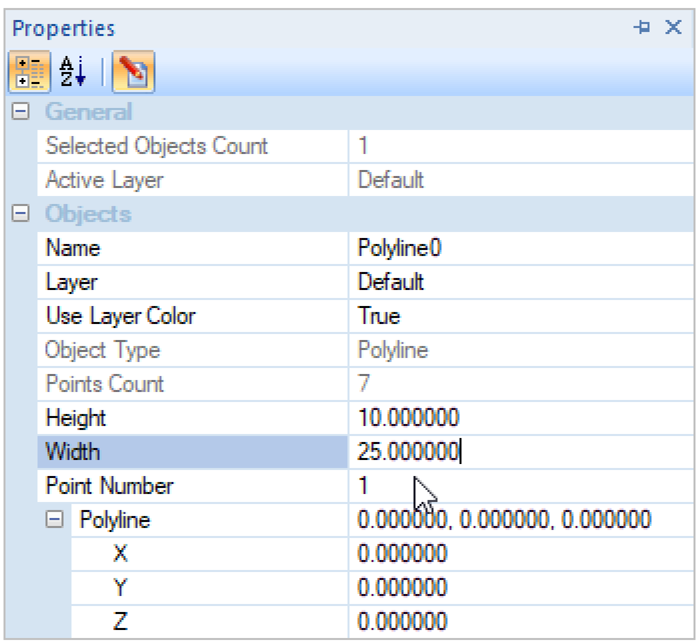
	Screen Pick
Step 1	 Left-click on Edit Point

Step 2	 Drag & Drop the Edit Point to new coordinates

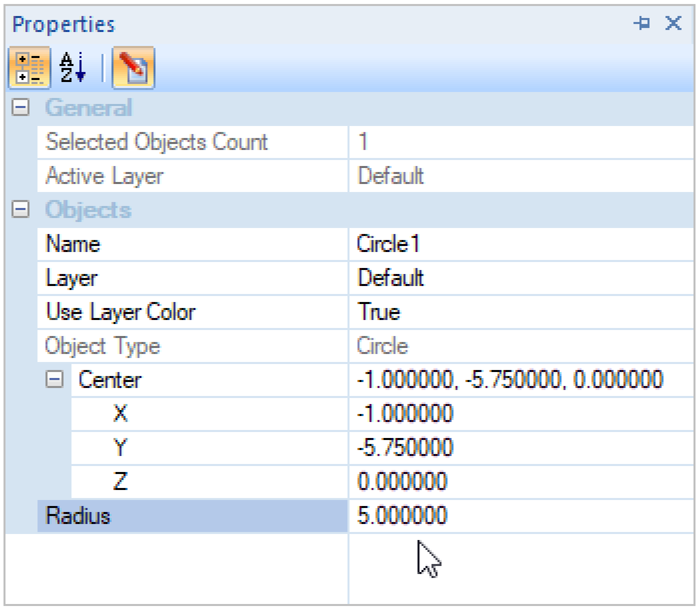
Edit Geometry Parametrically

You can parametrically edit the dimensions of geometry using the Properties dialog.

Just select the Edit icon until the dimension values are activated. When you edit a dimension vales and close the Properties dialog the geometry to update automatically.



Edit the Length & Width of a Rectangle

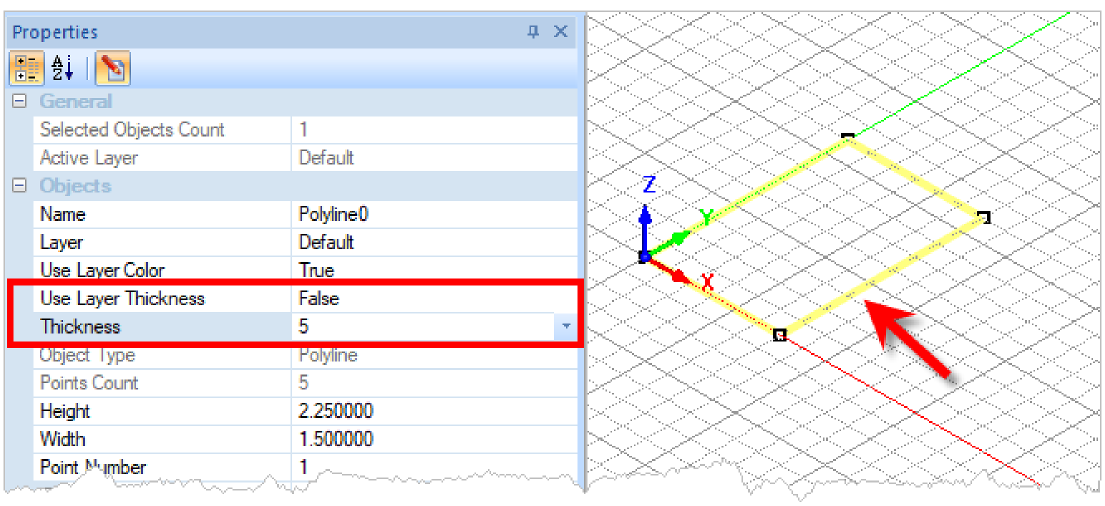


Edit the Radius of a Circle

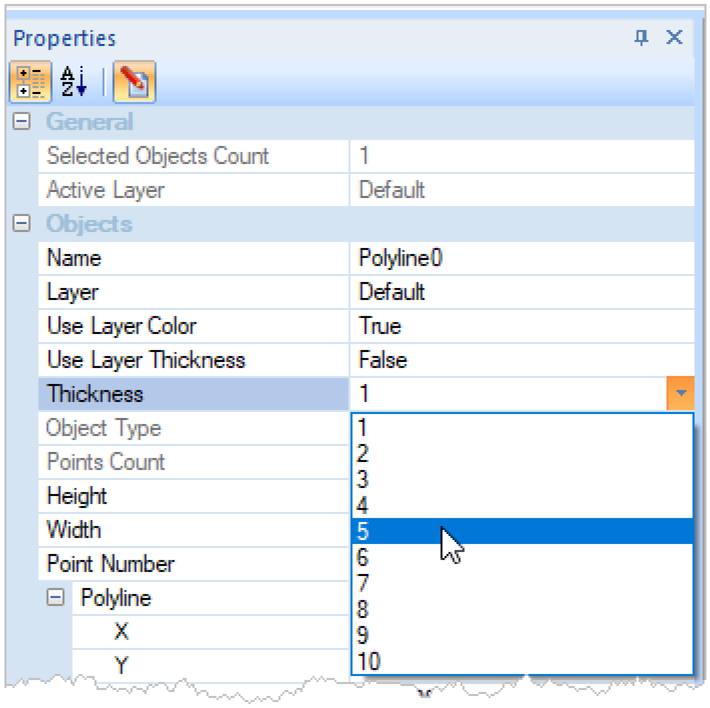
Editing Line Widths

You can edit the width of a line or curve using the Properties dialog.

Just select the Edit icon until the object properties are activated. Then set the Use Layer Thickness to False. This will activate the Thickness field. Drop down the Thickness menu and select a line width. When you edit a dimension vales and close the Properties dialog the geometry to update automatically.



Edit the Width of a line or curve



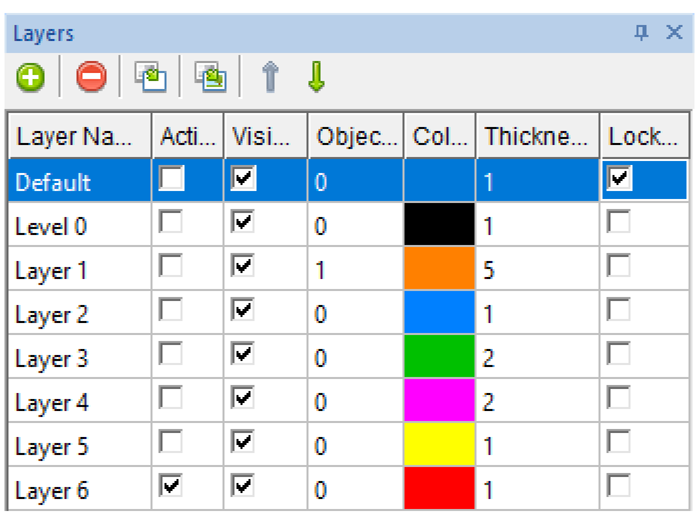
Line Width Property Selector

Related Topics

Properties

6.7 Layer Manager

This button calls up a dialog to manage the layers.



Layer Manager

Each layer has a name and color associated with it. There can be 0 or more geometric objects on any layer. A layer and all the geometry on it can be made invisible if you uncheck the **Visible** checkbox. The layer can be turned back on by rechecking the box. All layers are visible by default. Layers can also have a thickness.

Related Topics

[Status Tool Bar]

[Layer Visibility]

[Lock Layer]

[New Layer]

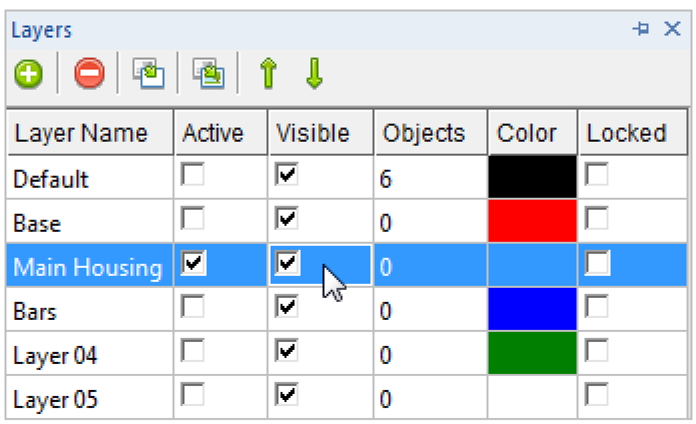
[Delete Layer]

[Duplicate Layer]

[Duplicate Layer and Objects]

6.7.1 Layer Visibility

Visibility of layers is useful in the construction of complicated models. Create multiple different layers, one for each subsection of the model. Then turn off all but the layer for the subsection currently being modified. This makes very complicated models much easier to see and work with. Otherwise, it can be difficult to see whatever was just created or picking particular items in the middle of lots of other objects from different subsections.

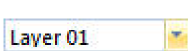


Layer Visibility

The visibility of the other layers can be periodically turned on and off to check the positioning of the current subsection with regards to the rest of the model. Visibility of active layer cannot be turned off.

Only one layer at a time is active. Any new objects created will be created on this layer and with the default layer color. Selecting objects by layer will select only the objects on the current layer.

You can change the active layer through this dialog or through the [Status Toolbar] drop-down list.



Active Layer

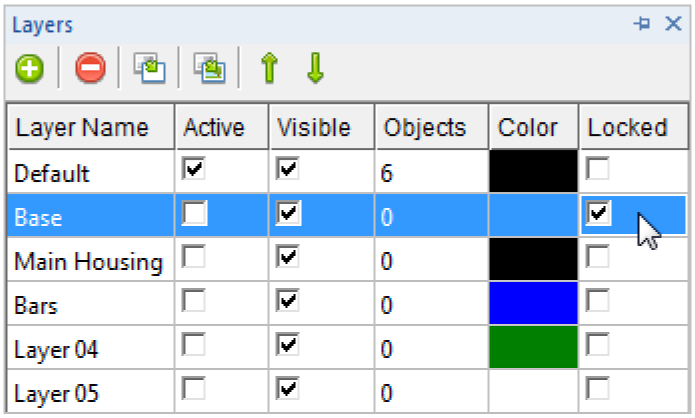
The current number of objects on each layer is also displayed.

Related Topics

[Layer Manager]

6.7.2 Lock Layer

Locks a layer when the Locked check box is selected.



Lock Layer

Locking a layer does not allow selection, editing and snapping to objects. This would be useful where you would like to have the geometries visible but locked for selection and editing. The layer that is active cannot be locked.

Related Topics

[Layer Manager]

6.7.3 New Layer

 Creates a new layer. The default name is New Layer, but the name can be changed.

Related Topics

[Layer Manager]

6.7.4 Delete Layer

 Deletes the selected layer.

Related Topicspics

Layer Manager

6.7.5 Duplicate Layer

Copies the layer selected with its attributes.

Related Topics

Layer Manager

6.7.6 Duplicate Layer and Objects

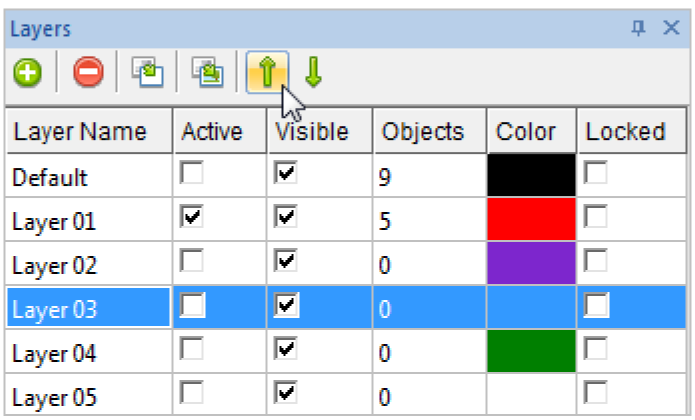
Copies the layer selected with its attributes and all of the objects on the layer.

Related Topics

Layer Manager

6.7.7 Reorder Layers

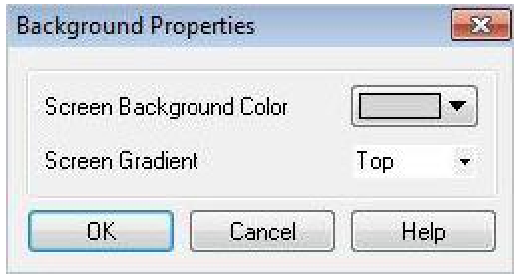
From the Layer Manager, select these icons to Reorder Layers within the Layer Manager list. First select a layer and then pick the Move Layer Up or Move Layer Down icon from the Layer Manager toolbar.



Layer Manager - Reorder Layer

6.8 Background Properties

Performing a Right mouse click in any of the view ports when no geometries are selected and selecting Properties from the menu display the Background Properties dialog.



Background Properties

Screen Background Color

Allows user to set the viewport background color. This is the color displayed when no objects are at that position.

Screen Gradient

This is a smooth change of lighting. The direction specified is the lighter side. It starts slightly darker on the opposite side of the screen and gradually gets lighter until it reaches the specified side.

Related Topics

Layer Manager

Exercises

7.1 #1: VisualCAD Preferences

In this exercise you will learn about many of the most commonly used system options in VisualCAD including how to access the online help, set display options, tolerances and units, system options, viewports and other VisualCAD display functionality.

What you will learn:

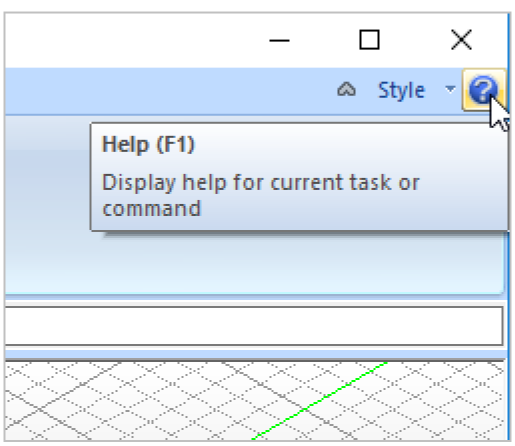
In this exercise you will perform the following VisualCAD tasks:

1. Access the online help.
2. Set the Display Style.
3. Set the Units to Inches.
4. Set System Options.
5. Set to Quad Viewports.
6. Viewing the Command Prompts.
7. The Status Bar.

7.1.1 Accessing the Online Help

Accessing the Online Help

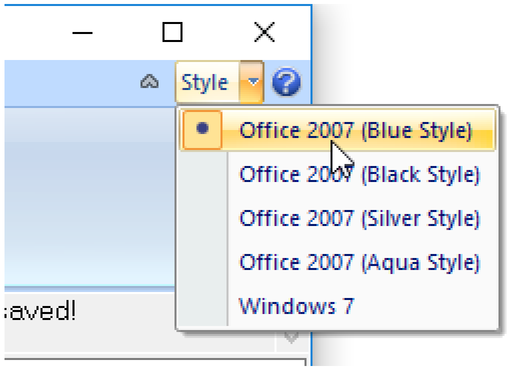
At any time you can display the VisualCAD Online Help by selecting this icon or pressing the <F1> key during any command prompt or by selecting the Help icon shown below.



7.1.2 Set the Display Style

Set the Display Style

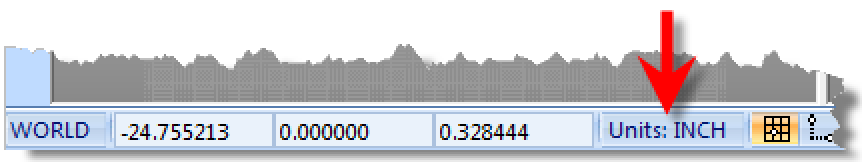
VisualCAD allows you to change the Style of the display. At the very top right of the VisualCAD display window, select the Style drop-down menu and select Office 2007 (Blue Style).



7.1.3 Set the Units to Inches

Set the Units to Inches

Open VisualCAD and check to see if the part unit is set to Inch.



Status Toolbar

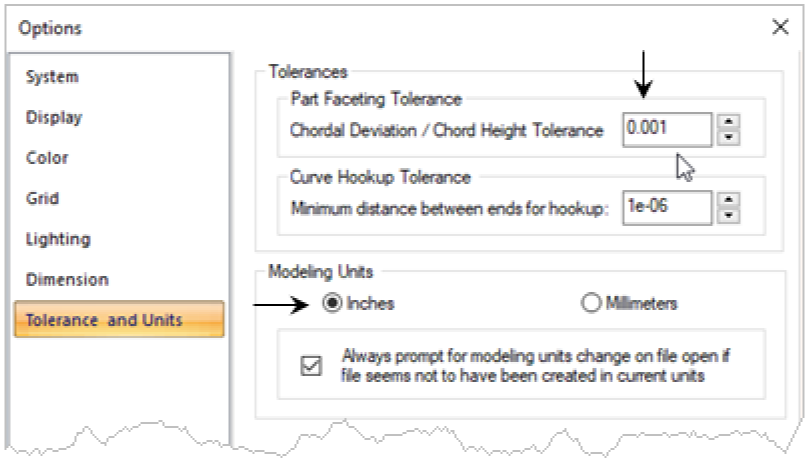
If Units is set to MM, double-click on it to change to INCH.

7.1.4 Set Systems Options

Set Systems Options

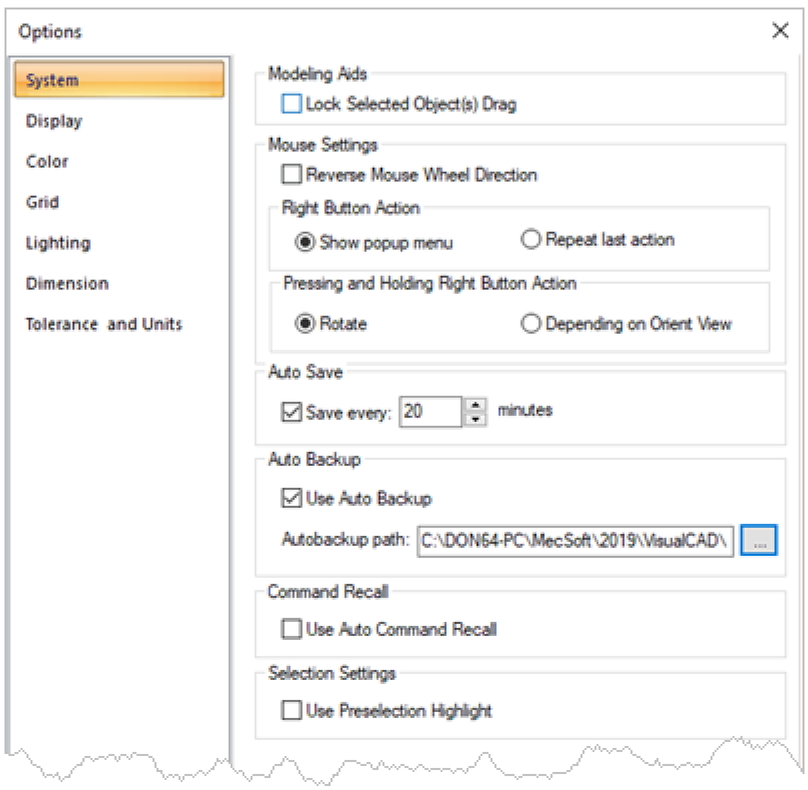
In this exercise you will learn how to set some of the systems options in VisualCAD.

1. Select the Home tab.
2. Now select the Options icon to display the Options dialog. This icon is also located on the Status Bar.
3. Select Tolerance and Units from the left side of the dialog.
4. Under Part Faceting Tolerance set the Chordal Deviation / Chord Height Tolerance to 0.001".
5. Set the Units to Inches.



System Options Dialog

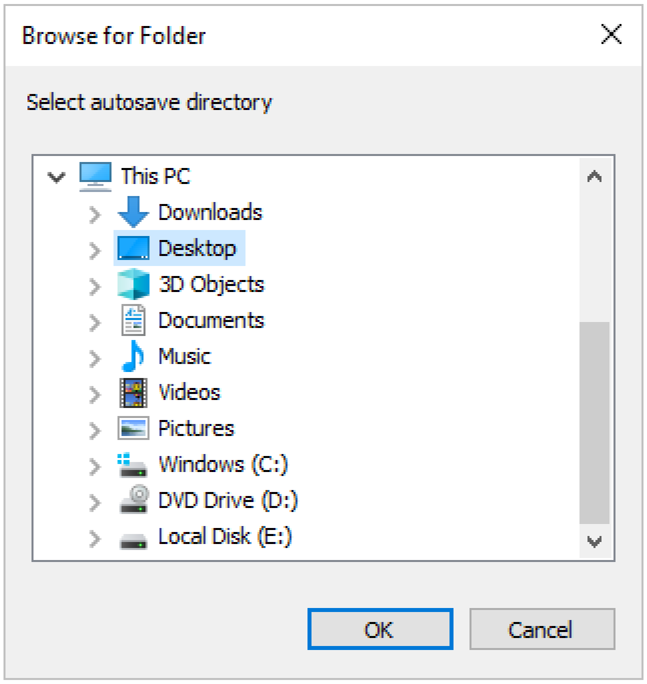
6. Select System from the left side of the dialog.



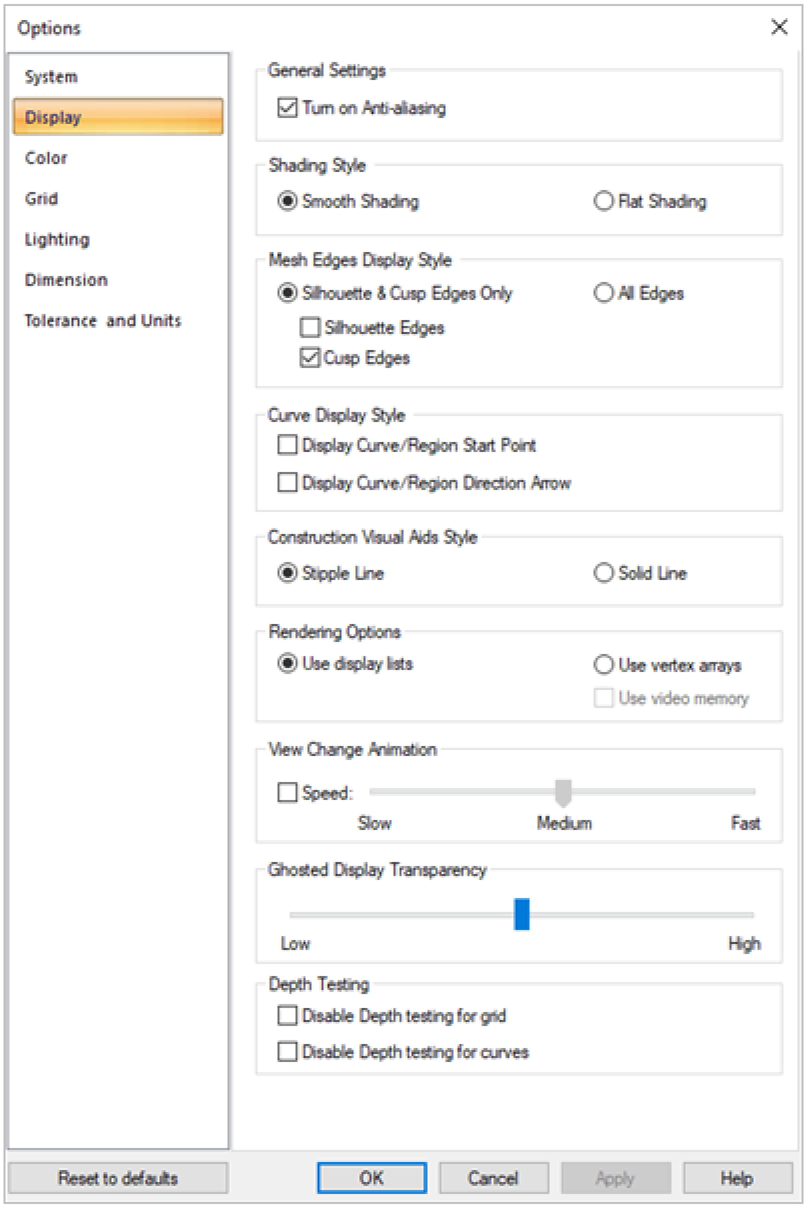
7. Under Auto Save, check the box and then set the value to save every 10 minutes.

8. Check the box to use Auto Backup and then select the ... box to display the Browse for Folder dialog.

9. Select the folder where you want backup files to be saved and then pick OK.

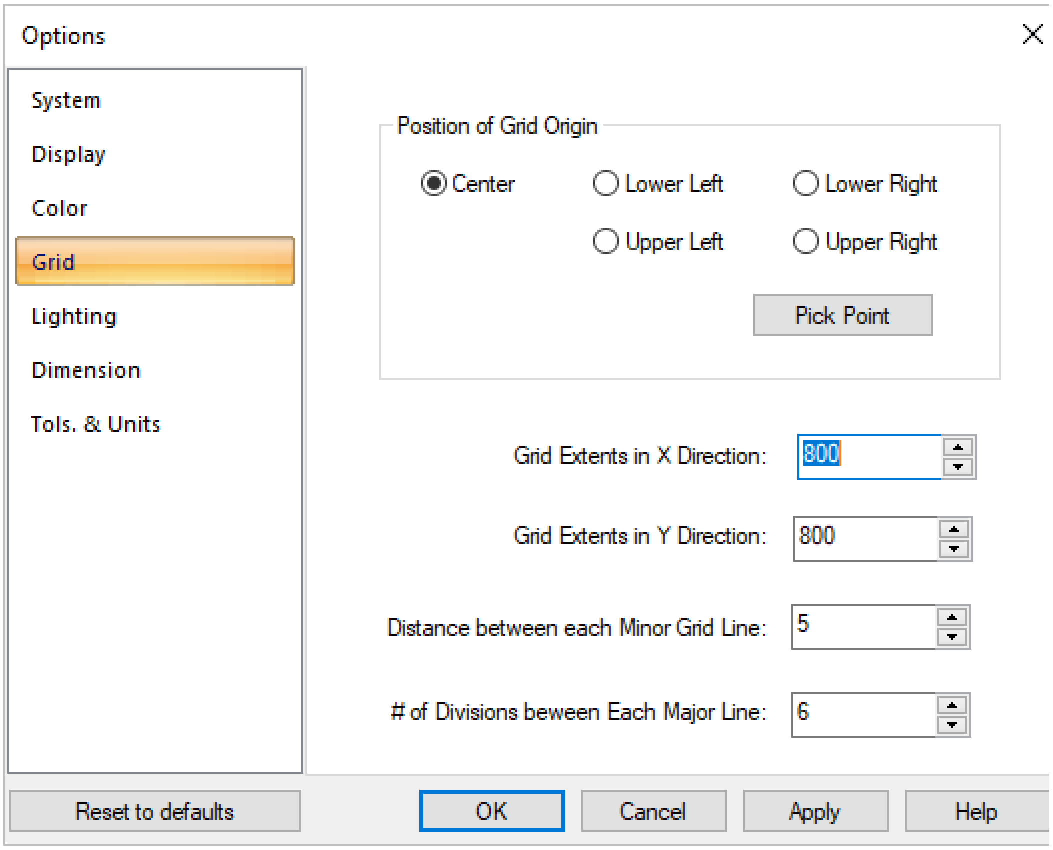


10. Set the remaining System Options as shown in the dialog above.
11. Now select Display from the left side of the dialog.
12. Under View Change Animation uncheck the box next to Speed.



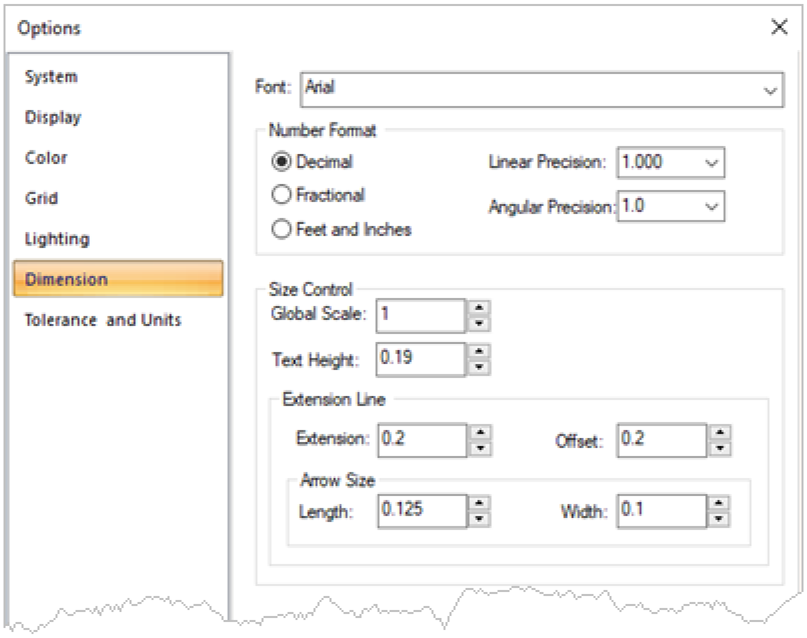
13. Set the remaining Display Options as shown in the dialog above.

14. Select Grid from the left side of the dialog. In these exercises we use the following Grid options.



System Options Dialog

15. Select Dimension from the left side of the dialog.
16. Set the Dimension Preferences to those shown in the dialog below.
17. Pick OK to close the Options dialog.



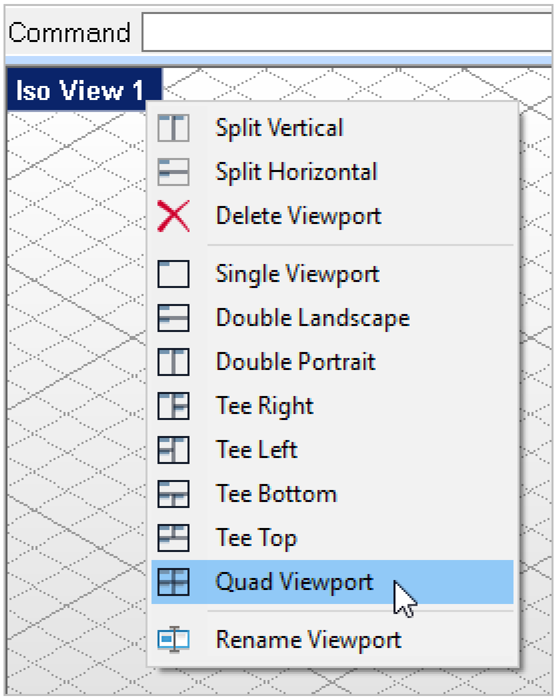
7.1.5 Set to Quad Viewports

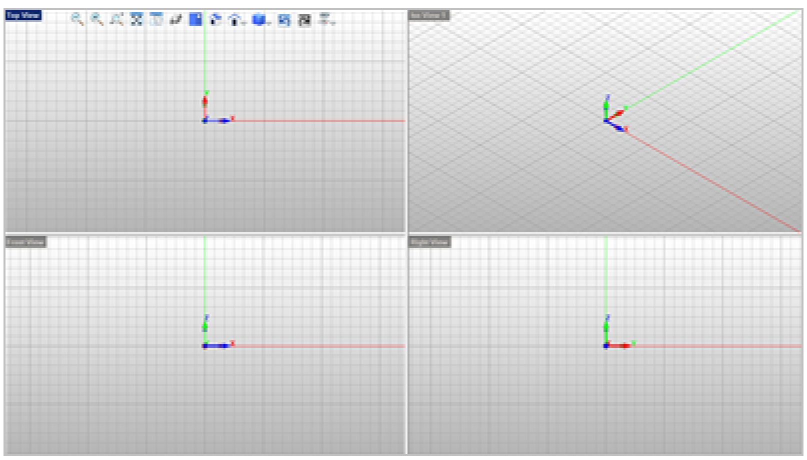
Set to Quad Viewports

It is much faster to draw and model in VisualCAD if you have multiple Viewports displayed.

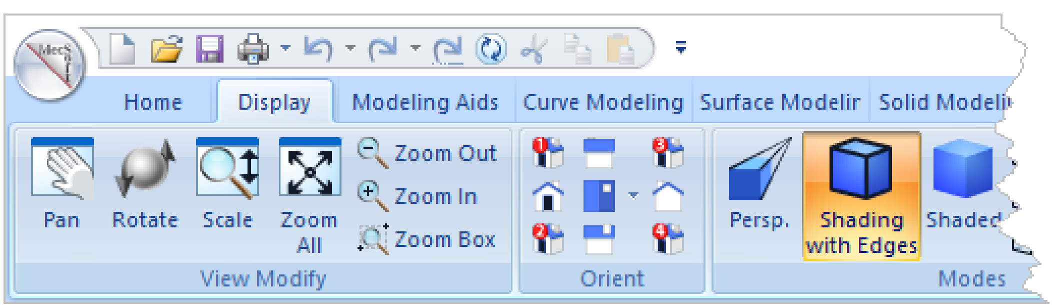
Select the Home tab.

1. Right-click on the Viewport tab located in the upper left side of the window and select Quad Viewport.





2. You can also select Display and Viewport commands from the Display tab of the VisualCAD Ribbon Bar.

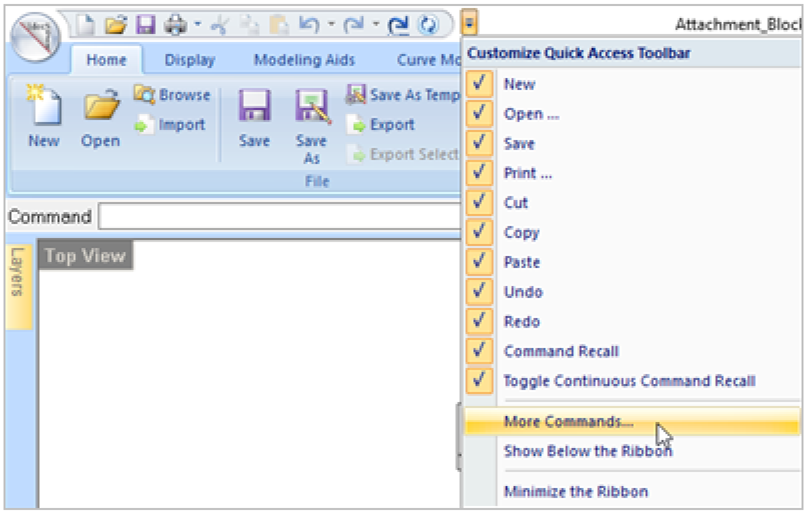


7.1.6 The Quick Access Toolbar

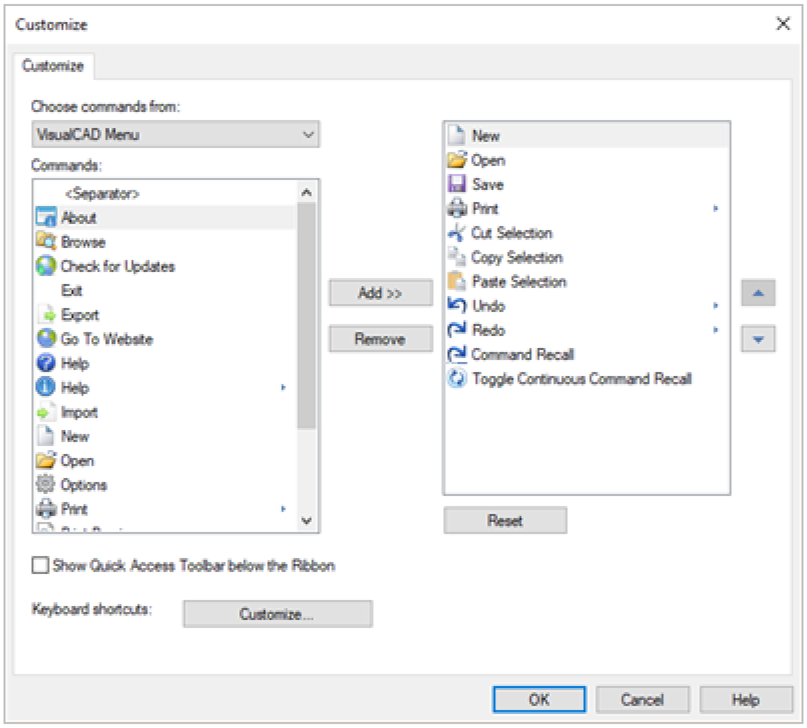
The Quick Access Toolbar

Located at the very top left of the VisualCAD window you will find the Quick Access Toolbar. It contains many of the most commonly used commands such as Cut, Copy, Paste, Save, Undo, Redo, etc. The cool thing is that you can customize this toolpath to add commands that YOU use regularly.

1. Pick the drop-down menu to the right of the Quick Access Toolbar.



2. Select More Commands... from the menu to display the Customize dialog. You can use this dialog to add or remove commands to the Quick Access Toolbar.



7.1.7 Viewing the Command Prompts

Viewing the Command Prompts

The Command Input bar 

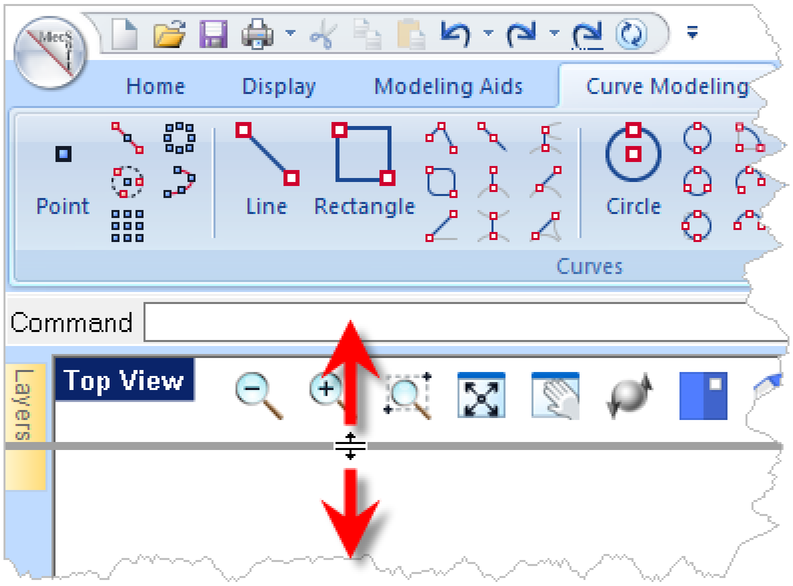
is located just below the ribbon bar. You can drag the bar down to see the current and previous command prompts. This is helpful while learning how to use VisualCAD.

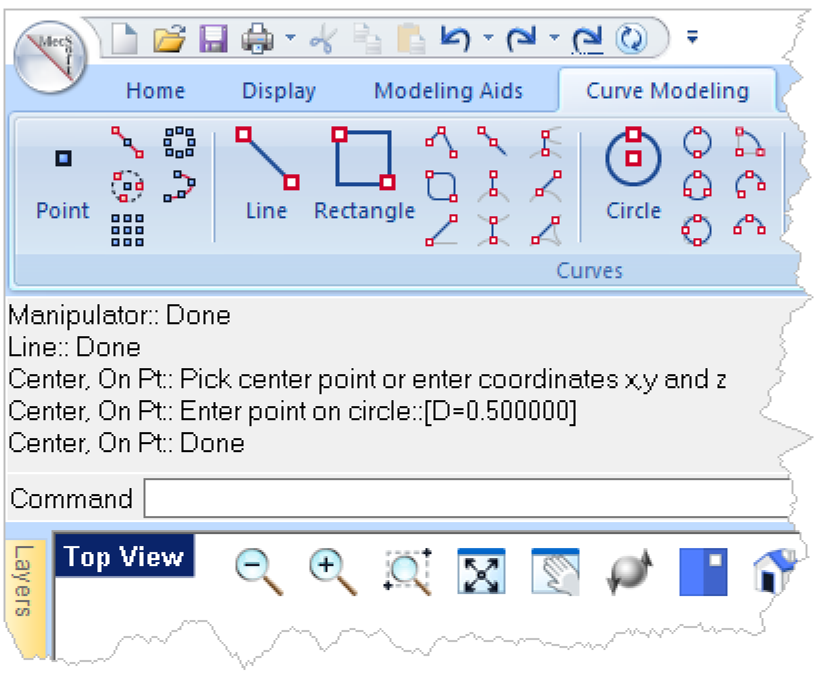
1. Move the cursor onto the bottom of the Command Input bar 

until it changes.

2. Then left-click and drag the Command Input bar 

down to reveal to review the history of commands.

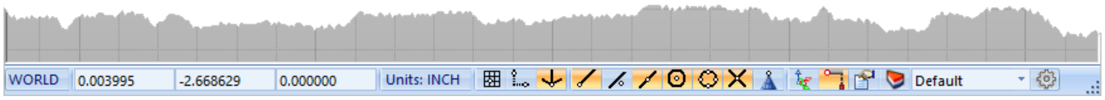




7.1.8 The Status Bar

The Status Bar

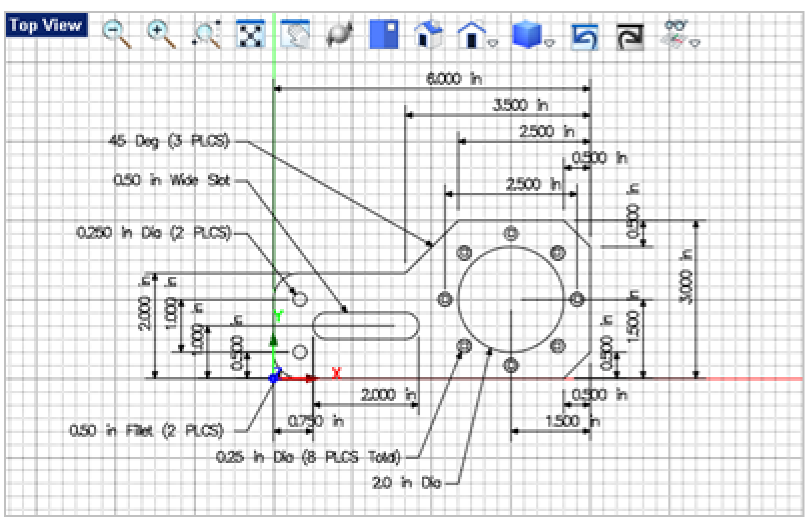
Located across the very bottom of the VisualCAD display you will find the Status Bar. It contains information and quick access to some very helpful commands. From left to right on the Status Bar you will find This is where you access the Layer Manager, Properties dialogs, Graphic Manipulator and more. Here is the list:



1. World Coordinates tracker.
2. Units.
3. Grid, Ortho and Object Snaps.
4. Graphic Manipulator.
5. Construction Visual Aids.
6. Properties Manager.
7. Layer Manager.
8. Active Layer Selector.
9. System Options.

7.2 #2: 2D Drawing & Dimensioning

In this exercise you will learn about the Layer Manager and how to create and work with layers. You will also learn how to draw quickly in VisualCAD using the menus and command line shortcuts. You will also learn how to take advantage of the Visual Aids feature to speed up your drawing creating tasks including lines, arcs curves, fillets, chamfers, trimming, dimensioning and more.



2D Drawing & Dimensioning

What you will learn:

In this exercise you will perform the following VisualCAD tasks:

1. Create Layers.

2. Curve Drawing.
3. Using Visual Aids.
4. Dimensioning.

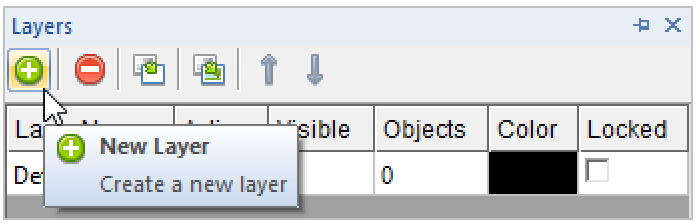
7.2.1 Create Layers

Create new Layers

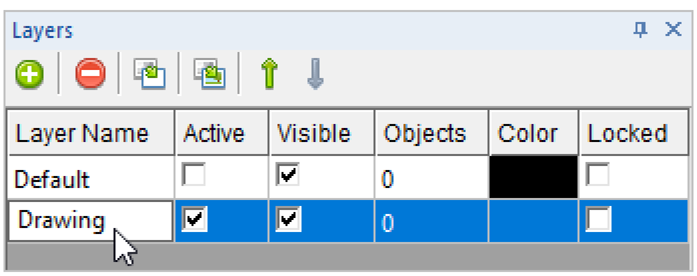
In this step you will use the Layer Manager to create some new layers and then set the active layer.

1. Display the Layer Manager. You can pick the Layer Manager icon located on the Status Bar.

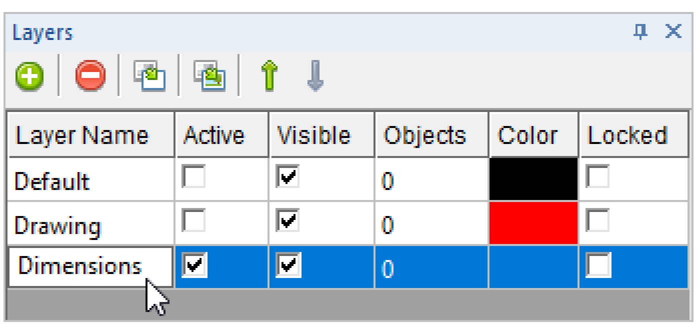
2. From the Layer Manager select the Add Layer icon to add a new layer.



3. Now lets rename the new layer to Drawing. First double-left-click in the Name field to activate it, and enter Drawing as the new layer name.



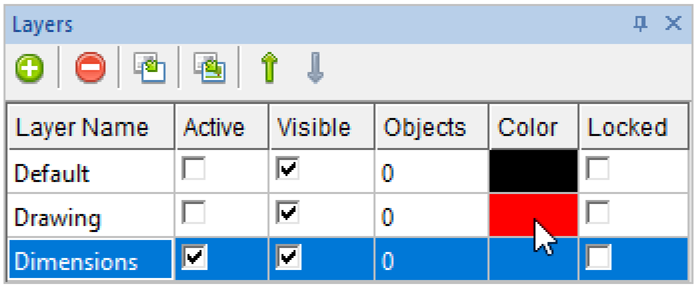
4. Repeat the procedure to create a new layer, this time named Dimensions.



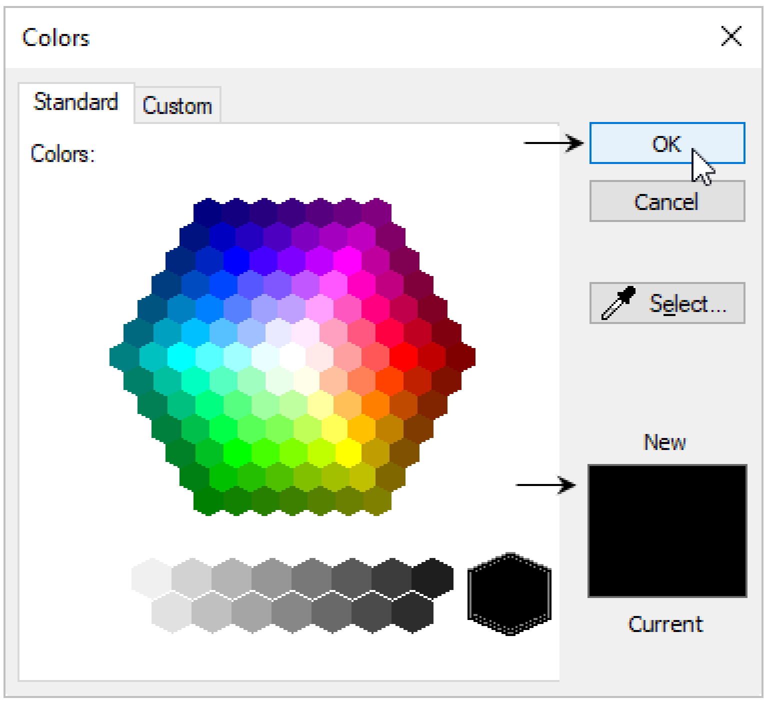
5. If the Layer Manager keeps Auto-Hiding to the left side of the display, you can select the Auto Hide pin icon at the top of the Layer Manager. to toggle this feature.

The pin down icon means that the dialog will stay displayed. The pin left icon means that the dialog will auto-hide.

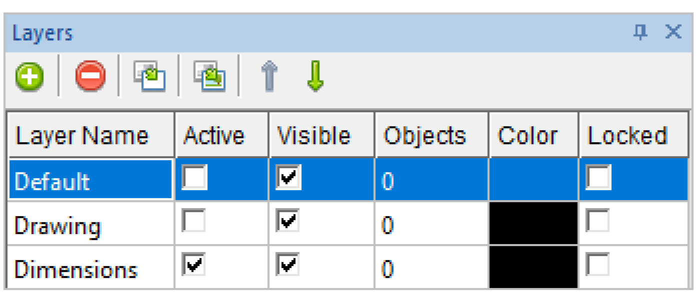
6. Now let's change the color of these two layers. In the Color field left-click on the color box for the layer named Drawing. Again, this will display the Colors dialog.



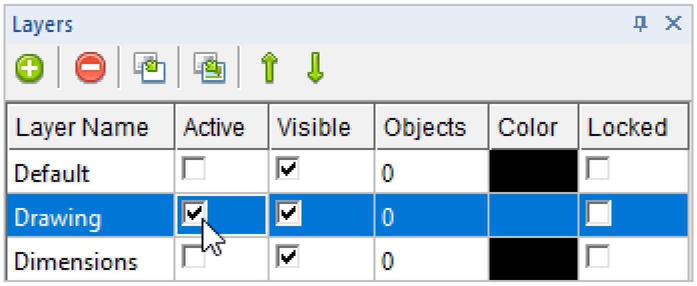
7. From the Colors dialog that displays, accept the default color black by pick OK to close the dialog.



8. Repeat the procedure for the layer named Dimensions.



9. Now from the Active column, check the box to make the layer named Drawing as the Active Layer.



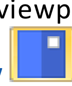
7.2.2 Curve Drawing

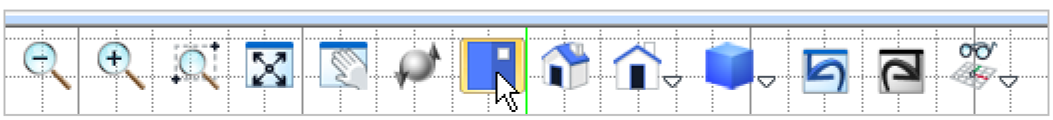
Curve Drawing

In this step you will learn how to use the keyboard to quickly drawing a polyline and then create some fillets and chamfers.

1. You only need the Top View  displayed for this exercise so if you have multiple Viewports displayed, double-left-click on the Top View tab to change the display to that Viewport.

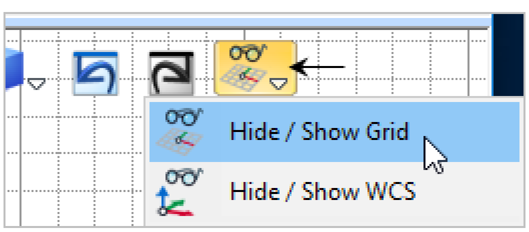
2. Alternately you can right-click on the Top View tab and select Single Viewport.

3. If you do not see the Top View  tab on any of your Viewports, simply double-left-click on any viewport tab to set the display to that single Viewport and then select the Top View  icon from the View Toolbar.

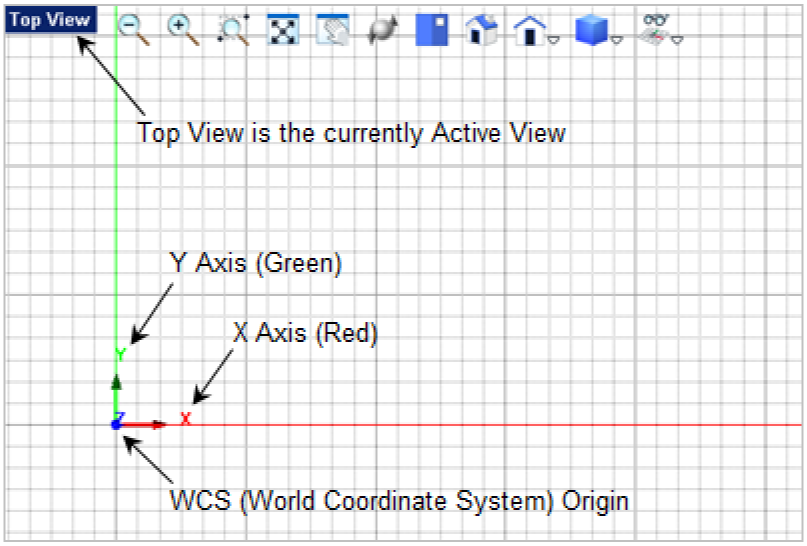


4. For visual purposes, we are displaying the construction Grid and the Grid options are set to those in the Preferences Exercise.

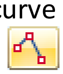
5. If the Grid is not displayed you can toggle it on by selecting Hide / Show Grid from the View Toolbar.

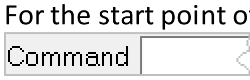


6. Scroll the middle mouse button to zoom in/out so that you see the World Origin WCS triad toward the bottom left of the screen. If you do not see the X (Red) and Y (Green) axes displayed, make sure you are in the Top View.



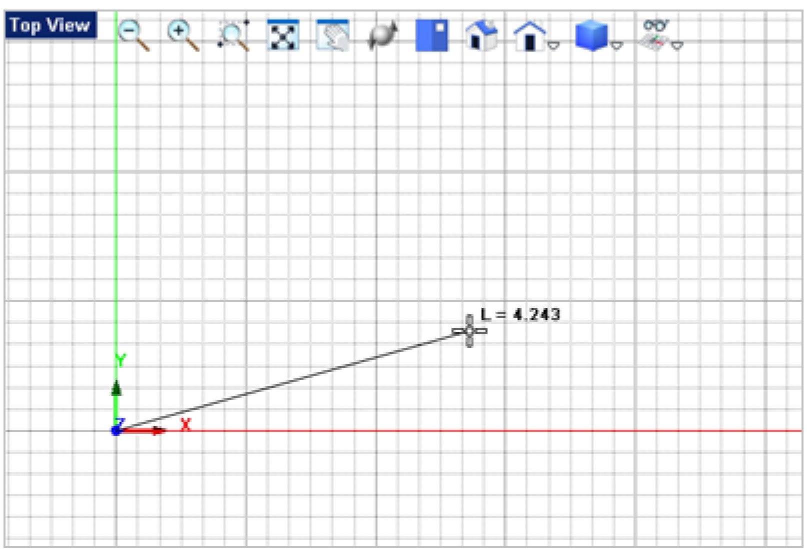
7. Now we can start curve drawing. From the Curve Modeling 

tab, select the Polyline  command.

8. The command prompt says:
Polyline:: Pick point or enter coordinates x,y and z
For the start point of the polyline, enter 0,0 in the command window 

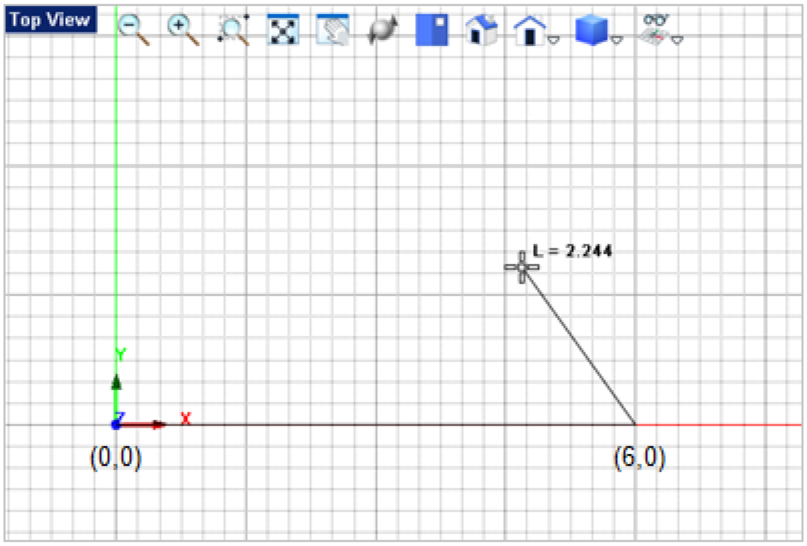
and press <Enter>.

You will see the polyline is starting at the XY origin on the left.



9. For the next point in the Polyline, enter 6,0 

and press <Enter>. This is in Absolute World Coordinates, 6 units in X and 0 units in Y measured from the World Origin.



10. Now if you are drawing and you do not know the exact absolute coordinate values you can enter Relative Coordinate values in the command window.

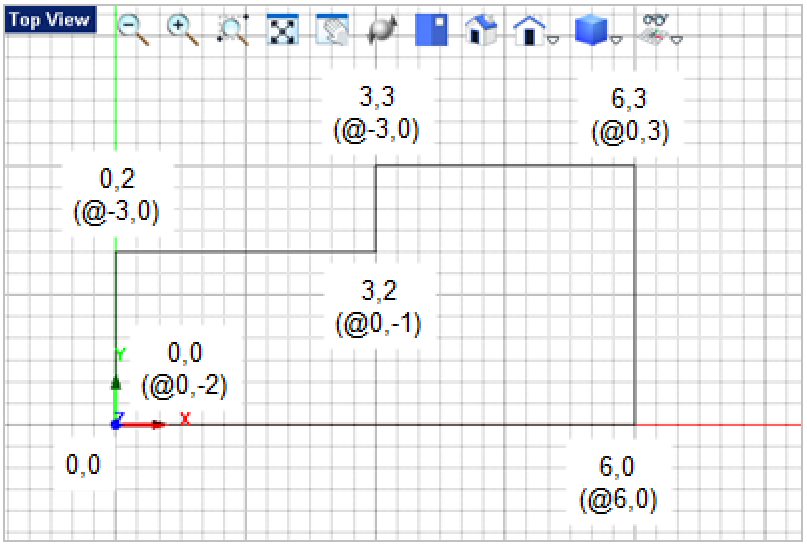
For the next coordinate enter @0,3 in the command window 

and press <Enter>. This will place the next point in the polyline 3" from the last point in the positive Y Axis direction.

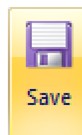
Listed below are the absolute and relative coordinate values for the entire polyline. Use them to complete the polyline, pressing <Enter> after each coordinate entered. If you make a mistake, just Undo and start again.

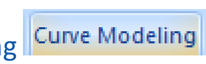
Polyline Absolute (Relative) coordinate values
0,0
6,0 (@6,0)
6,3 (@0,3)
3,3 (@-3,0)
3,2 (@0,-1)
0,2 (@-3,0)
0,0 (@0,-2)

Your polyline should like this:



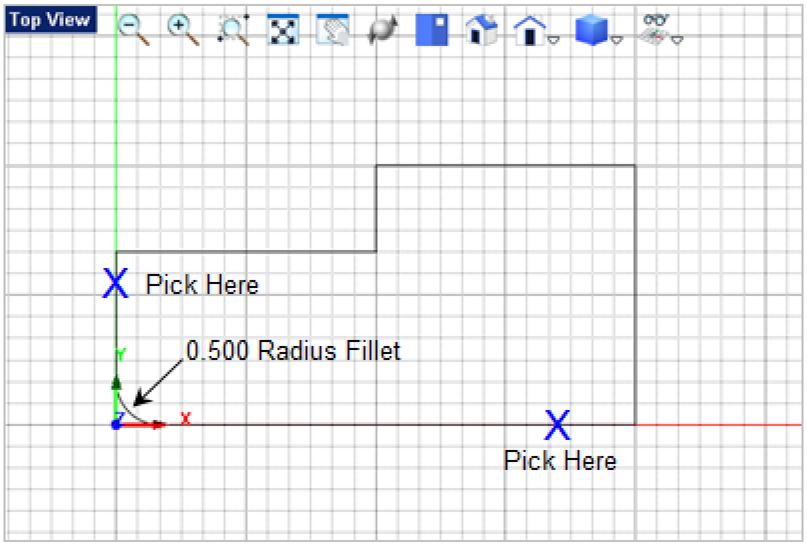
Note: In the future you can also specify a point at an angle and length from the previous point using the following format: @6<45 where the length of the line segment is 6 and the direction is 45 degrees from the previous point.

11. Always Save your work! Select the Save  or Save As  command from the Home tab or the VisualCAD Main Menu or press <Ctrl+S>.

12. Now let's add some fillets. From the Curve Modeling  tab select the Fillet  command.

13. For the fillet radius, enter 0.5 in the command window and press <Enter>.

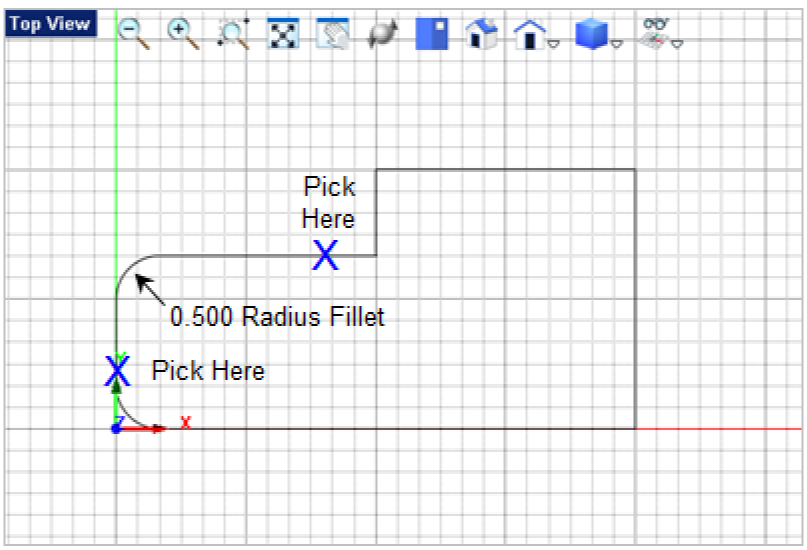
14. For the two lines to fillet, select the lines marked in the illustration below. For filleting it is best to select the line on the end furthest from the end you want to fillet.

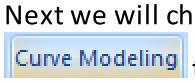


15. Press <Enter> to repeat the Fillet 

command.

16. The default fillet radius is now 0.5 so proceed directly to picking the two lines to fillet and press <Enter>.



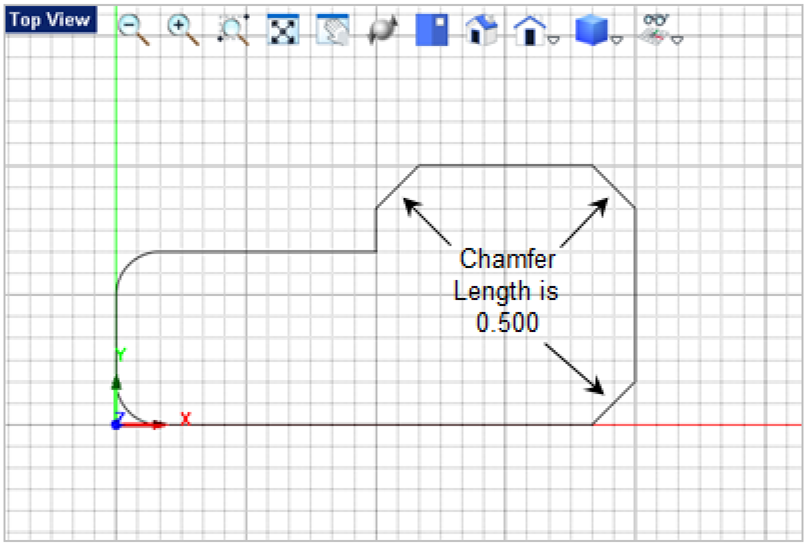
17. Next we will chamfer the three remaining outer corners. From the Curve Modeling 

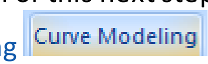
tab select the Chamfer  command.

18. For the chamfer length, enter 0.5 in the command window 

and press <Enter>.

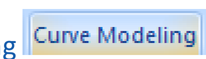
19. For the lines to chamfer, select lines marked in the illustration below. For chamfers it is best to select the line on the end furthest from the end you want to chamfer.

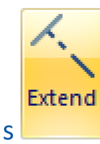


20. Currently our curves are still are one polyline. For this next step we need to explode the polyline. From the Curve Modeling 

tab select the Explode curves  command.

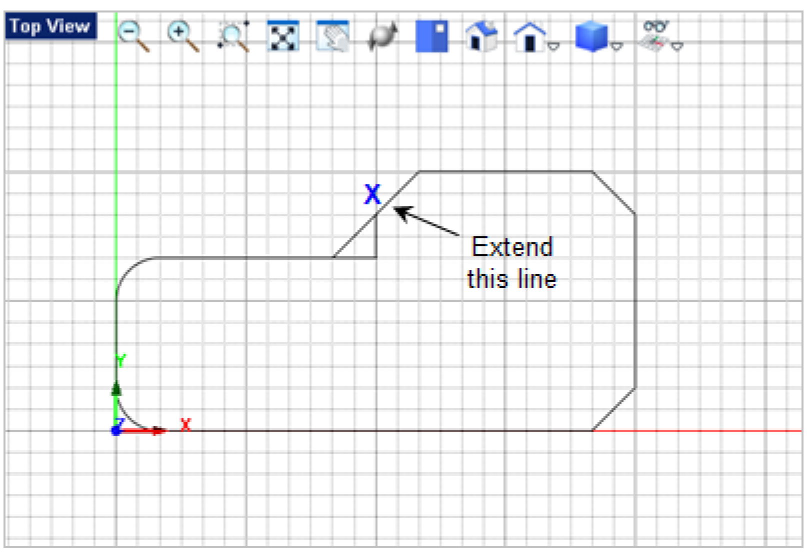
21. Select the polyline and the right-click the mouse or press <Enter>. The polyline is now individual curves.

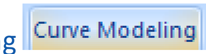
22. Now from the Curve Modeling 

tab select the Extend Curves 

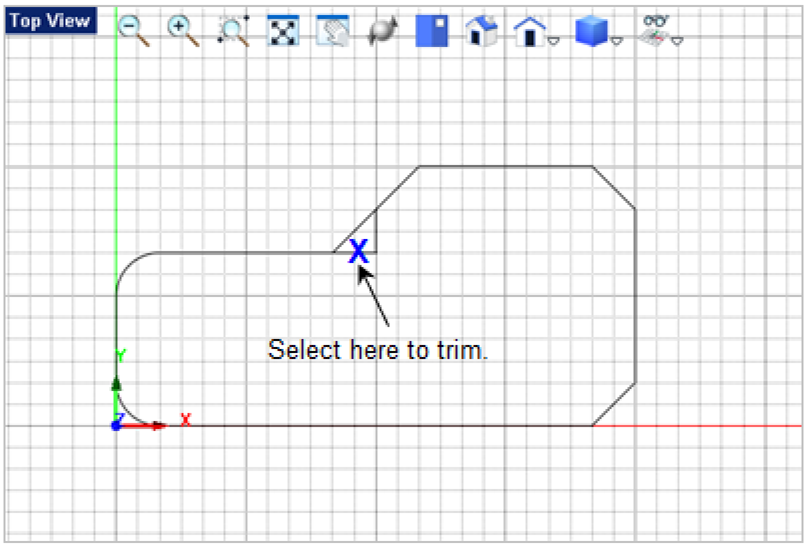
command.

23. Select the change line marked below to extend it to the horizontal line.

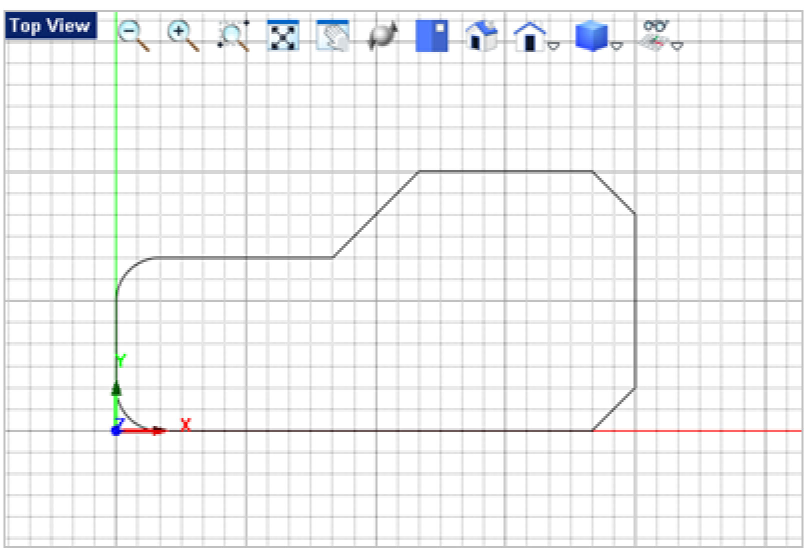


24. Now from the Curve Modeling  tab select the Trim Curves  command.

25. Pick the line segment to trim. When using the Trim command, you do not need to select the trimming curve. Just select what curve you want trimmed.



26. Next, delete the unneeded line. Just select it and then press the <Del> key or select the Cut  command icon from the Quick Access Toolbar.



7.2.3 Using Visual Aids

Using Visual Aids

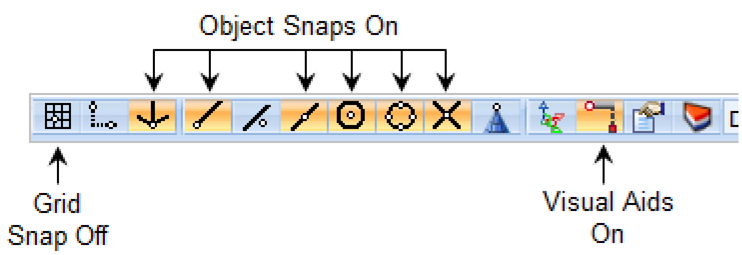
In this step you will learn how to draw with the use of the Visual Aids mode. With Visual Aids enabled, construction lines automatically appear as references geometry. This allows for easier visual placement of new geometry. Let's see how it works.

1. Curve creation using Visual Aids can be enabled by selecting the Visual Aids icon on the Status Toolbar.

   While Visual Aids is toggled On, the currently active Object Snaps are referenced visually depending on where the cursor is located on the drawing.

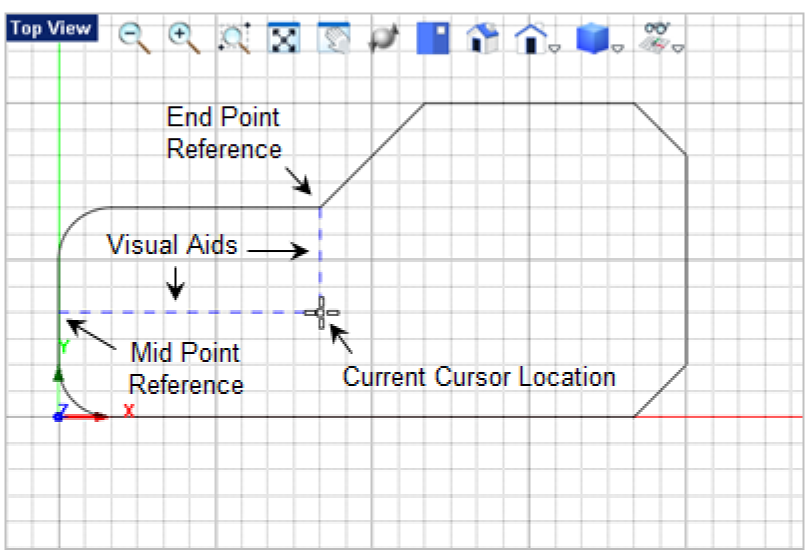
2. For now, enable the Origin Point Snap, End Point Snap, Mid Point Snap, Center Point Snap and Quad Point Snap.

3. Also disable the Grid Snap. Your Status Bar should look like this:

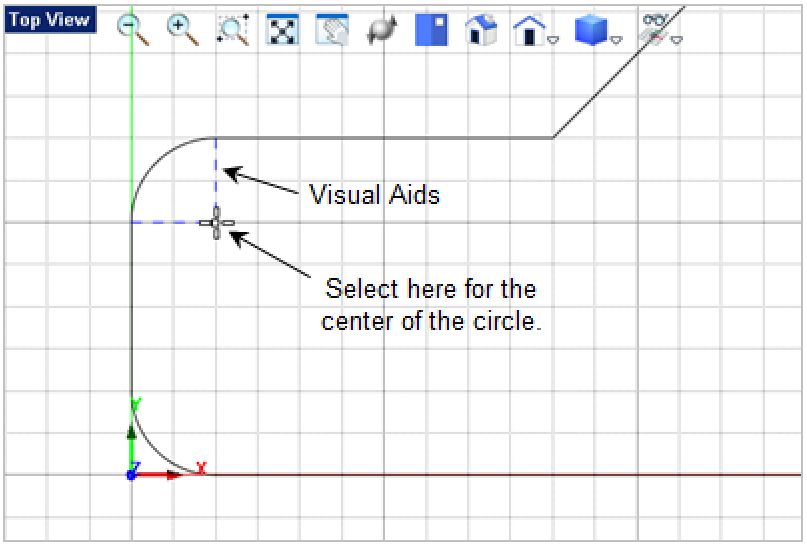


4. Now from the Curve Modeling tab select the Circle on Point command.

5. For the circle center point move the cursor near geometry in your drawing. You will see Visual Aids appear (i.e., reference lines).



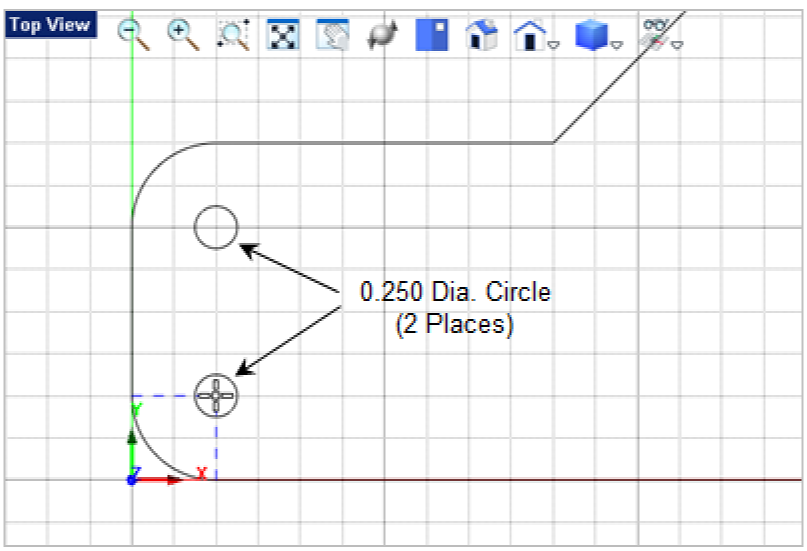
6. We want to locate the center of circle at the center of the fillet located in the upper left corner of the drawing. Move the cursor to this area and you will see two Visual Aids appear when the end points of the fillet are recognized.



7. For the circle diameter enter 0.25 

and press <Enter>.

8. Press <Enter> to repeat the command and draw a circle at the center of the lower left fillet.

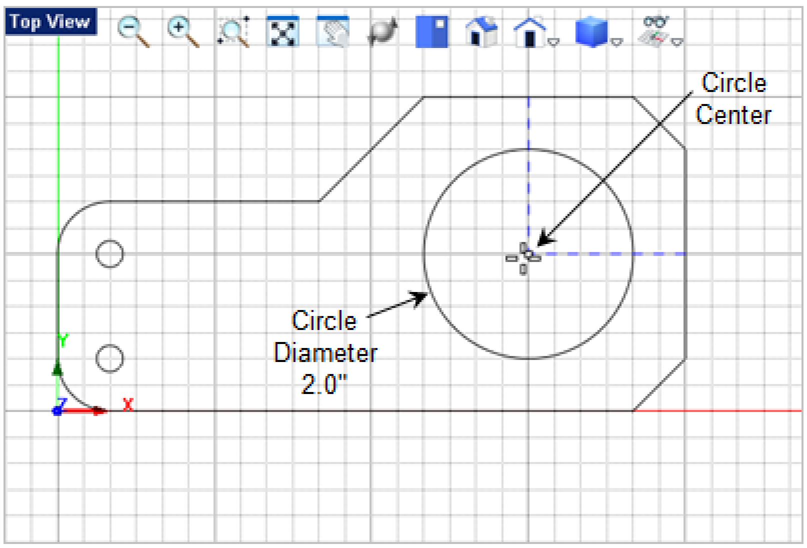


7.2.4 More Drawing Tools

More Drawing Tools

In this step you will learn a few additional curve creation and curve editing commands.

1. Lets draw another circle. From the Curve Modeling Curve Modeling tab select the Circle on Point Circle command.
2. For the center point of the circle use the Visual Aids to locate and select the Mid Point intersection of the upper and right side lines.
3. For the circle diameter enter 2.0 in the command window Command and press <Enter>.

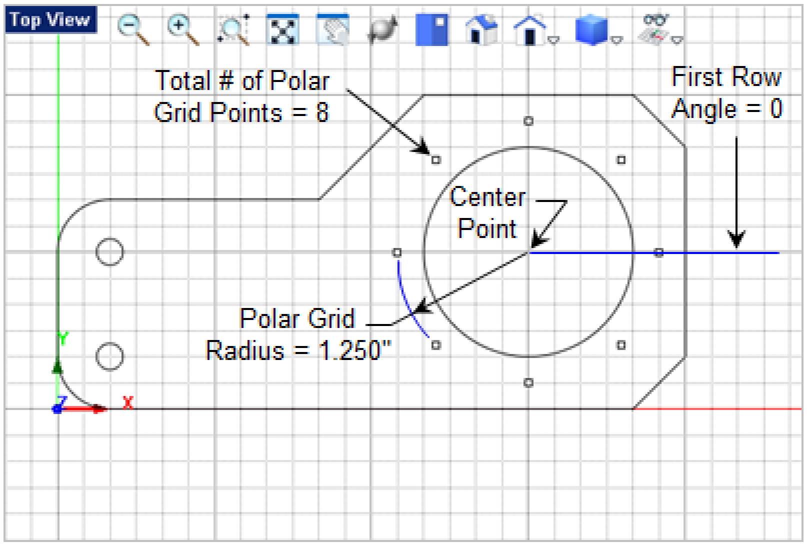


4. Next we will draw a polar array of points around the circle we just drew.

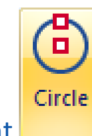
> **Why am I drawing Points?** Because when machining Hole Making operations in CAM such as Drill, Tap, Bore, etc., you can select points as Control Geometry! The hole geometry DOES NOT have to be present in the 2D drawing or 3D model in order to drill at that location.

5. From the Curve Modeling Curve Modeling tab select the Polar Grid of Points command.
6. For the total number of points on the circumference, enter 8.
7. For the angle of the first row about the X Axis enter 0.
8. Pick the center point for the polar grid or enter the coordinate values. We want the points centered around the 2" circle we just drew, so set you Object Snap to Center Point and move the cursor over the circle until the center point is highlighted and the select it.

9. For the polar grid radius, enter 1.250 and then press <Enter>. The polar grid of points will be created around the circle as shown below.



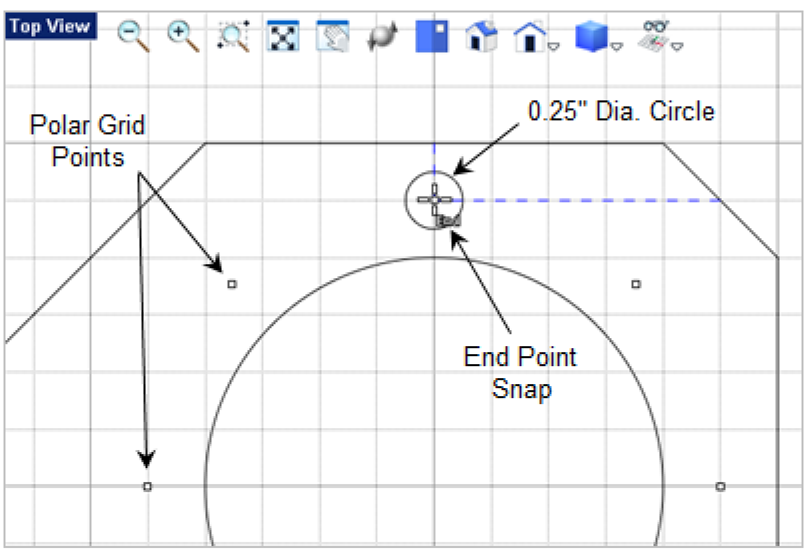
10. For this exercise we will also add the polar array of circles. From the Curve Modeling 

tab select the Circle on Point 

command.

11. For the center point, make sure the End Point Snap  is on and then select the end point at the 12 o-clock position.

12. For the circle radius enter 0.25 and then press <Enter>.



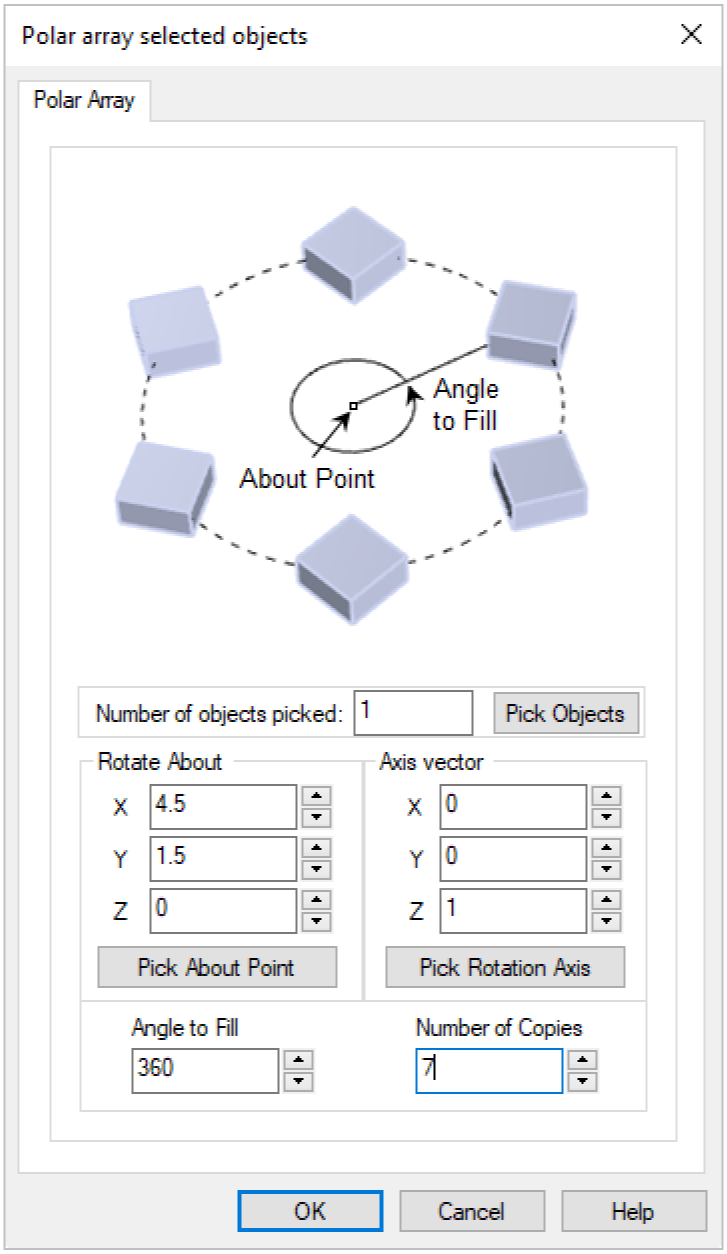
13. Now we will duplicate the circle by creating a polar array. First select the circle you just created.

14. Then from the Modify / Transform 

tab.

15. Select the Polar Array 

command. The Polar array selected objects dialog will display.



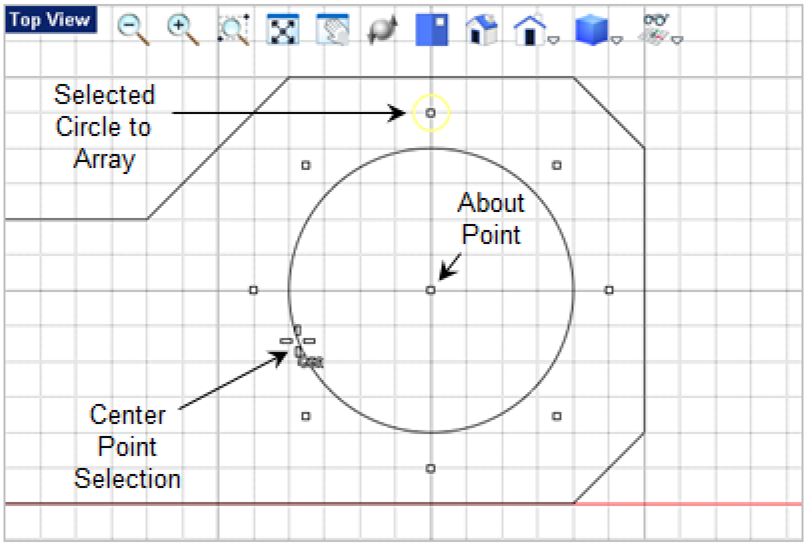
16. The Number of Objects field should be 1. This represents the circle that you have selected.

17. Under the Axis Vector, the values should be X0, Y0, Z1.

18. Under the Rotate About section, select the Pick About Point button. The dialog will minimize and wait for you to select a point.

19. From the Status Bar activate the Center Point Object Snap .

20. Now move the cursor over the 2" diameter circle until the center point is displayed and then selected it.

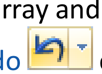


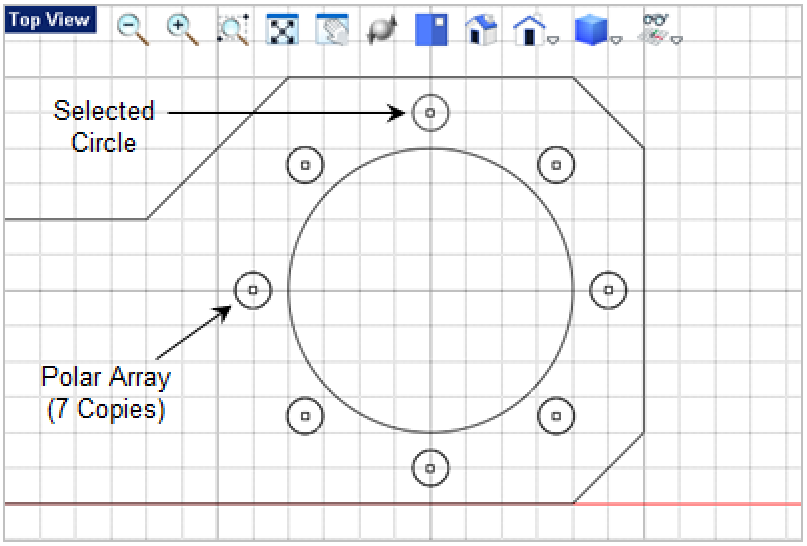
21. The dialog will re-appear with the XYZ coordinated entered for the Rotate About section. The value should be X4.5, Y1.5 and Z0.

22. For the Number of Copies field enter 7.

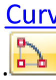
23. The Angle to Fill field should be 360.

24. Now pick OK to close the dialog and the array should be drawn and selected.

25. Press <Enter> to accept the array and your drawing should look like the illustration below. If not, select the Undo  command and go back and repeat the previous steps.



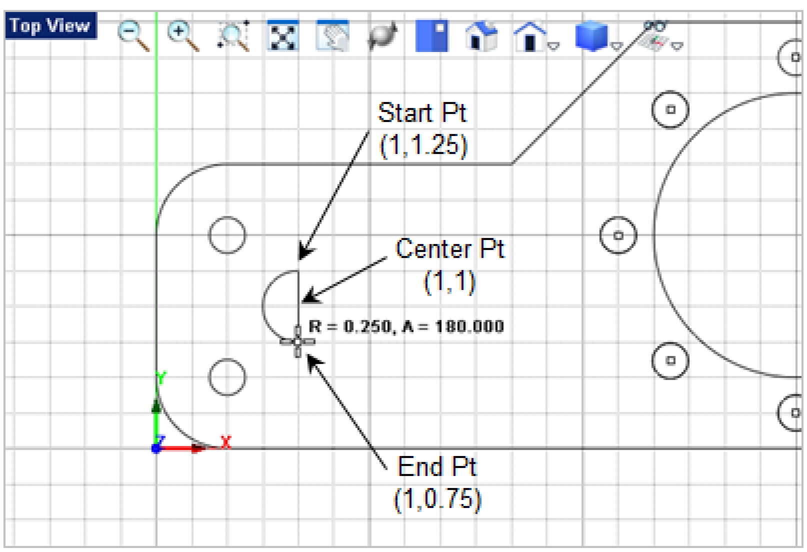
26. Now we will drawing a slot. From the Curve Curve Modeling tab 

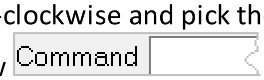
select the Arc Center, Start, Angle command. 

27. In the Status Bar make sure the Grid Snap is On .

28. For the center point select the grid point shown below or enter 1,1 in the command window and press <Enter>.

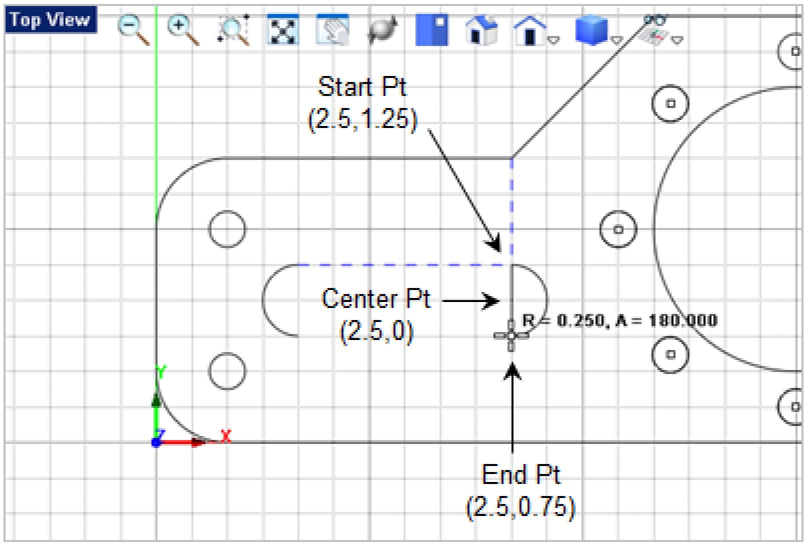
29. Following the command prompts. It says to select the start point of the arc. Locate and select the start point shown below or enter 1,0.75 in the command window and press <Enter>.



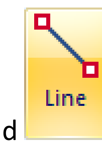
30. For the arc end point move the cursor counter-clockwise and pick the end point shown or enter 1,0.75 in the command window 

and press <Enter>.

31. Now we want to draw another arc, press <Enter> to repeat the command and select or enter the center, start and end points shown below.



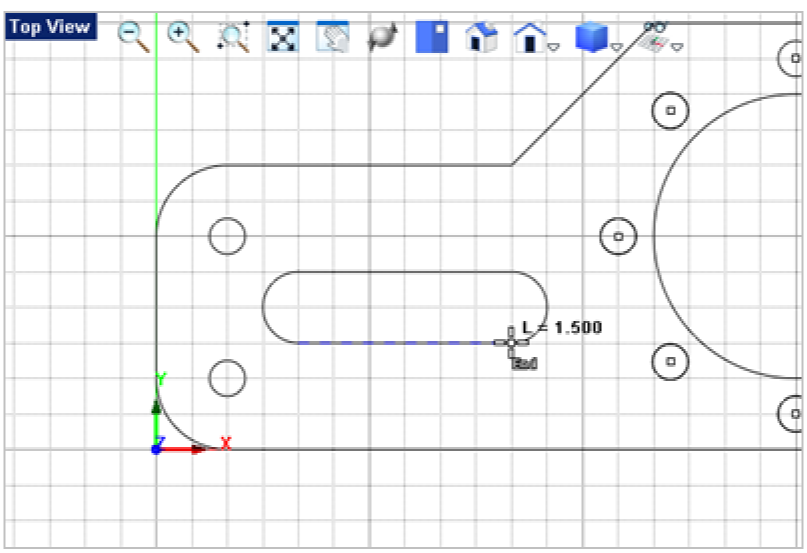
32. Now we want to draw a line to connect the two arcs across the top. From the Curve Modeling tab 

select the Line command 

.

33. For the start point for the line, set the Object Snap to End Point  from the Status Bar and select the start and end of the top line of the slot.

34. Press <Enter> to repeat the Line command and select the start and end of the bottom line of the slot. Your drawing should look like this:



7.2.5 Dimensioning

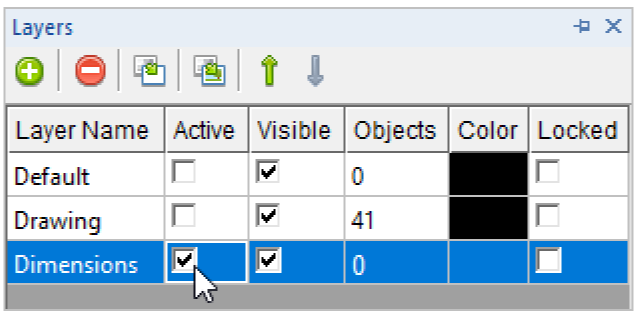
Dimensioning

In this step you will learn how to place dimensions on the drawing.

1. Start by using the middle mouse wheel to zoom out to leave yourself plenty of room to place your dimensions. If your mouse does not have a middle wheel, select the Zoom Out icon from the View Toolbar. You can also use the Pan icon to pan the drawing within the display window.

2. Now change the active layer. From the Status Toolbar select the Layer Manager icon.

3. From the Layer Manager go to the Active column and check the box next to Dimensions to make it the Active Layer.



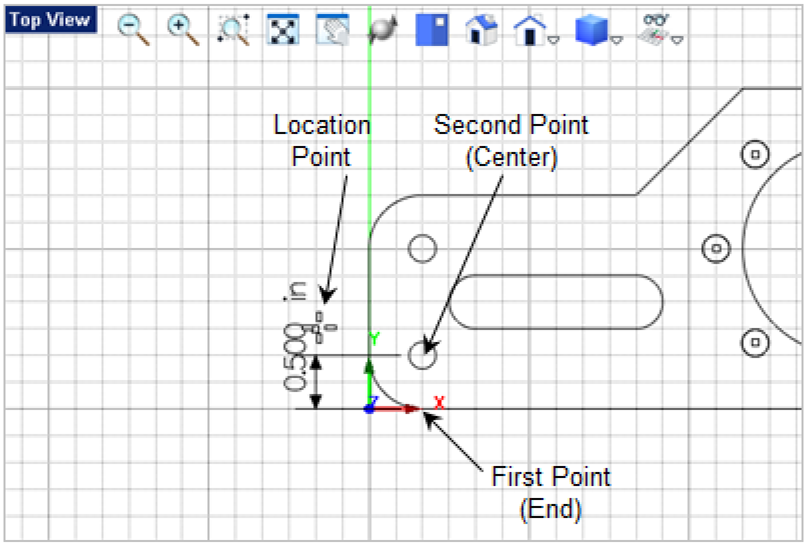
Layer Name	Active	Visible	Objects	Color	Locked
Default	☐	☑	0		☐
Drawing	☐	☑	41		☐
Dimensions	☑	☑	0		☐

4. From the Dimension Dimensions tab, select the Vertical Dimension Vertical command.

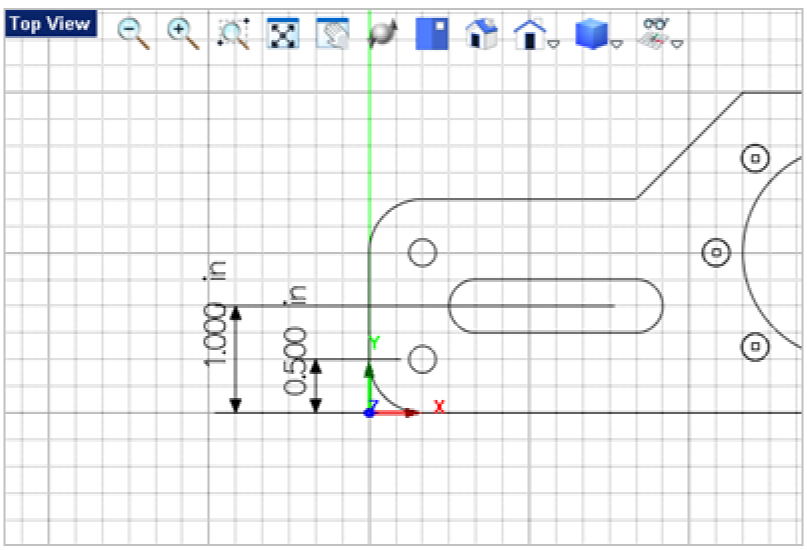
5. Activate the Object Snaps required (End, Center and Grid)

6. Select the first and second points that you wish to dimension.

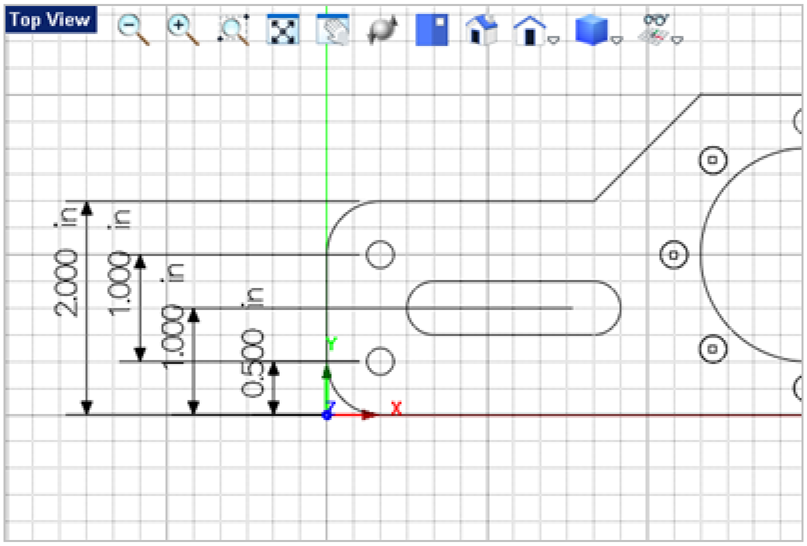
7. Then select a point to locate the dimension. refer to the illustration below:



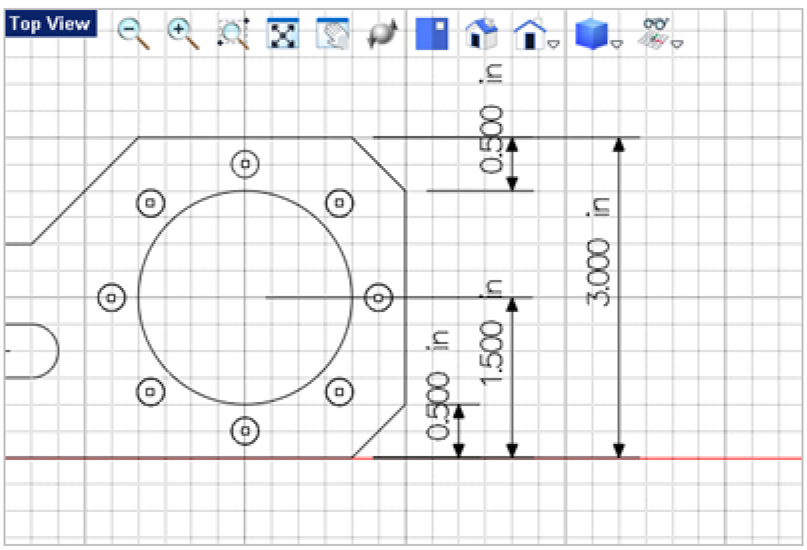
8. Press <Enter> to repeat the command.
9. Select the first, second and location point for the dimension as shown below:



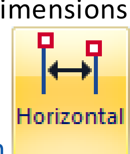
10. Repeat the command to create three additional vertical dimensions on the left as illustrated below:



11. Repeat the command to create three additional vertical dimensions on the right as illustrated below:

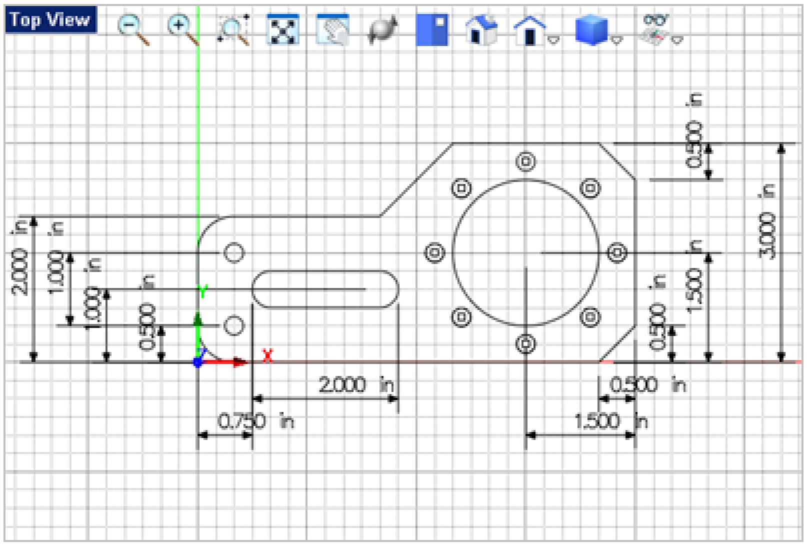


12. Now we will create the horizontal dimensions. From the Dimension 

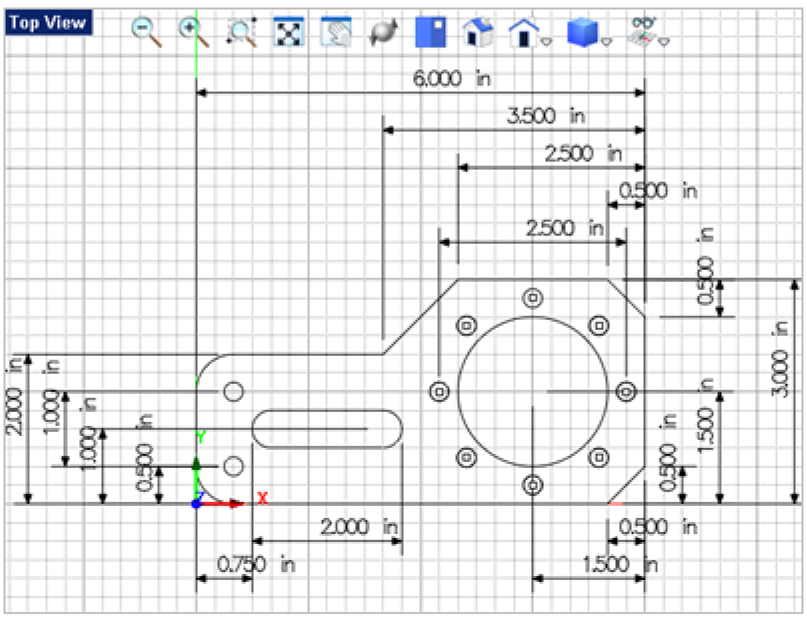
tab select the Horizontal Dimension 

command.

13. Select the first second and location points to create the lower horizontal dimensions shown below:



14. Repeat the command to create the upper horizontal dimensions shown below:

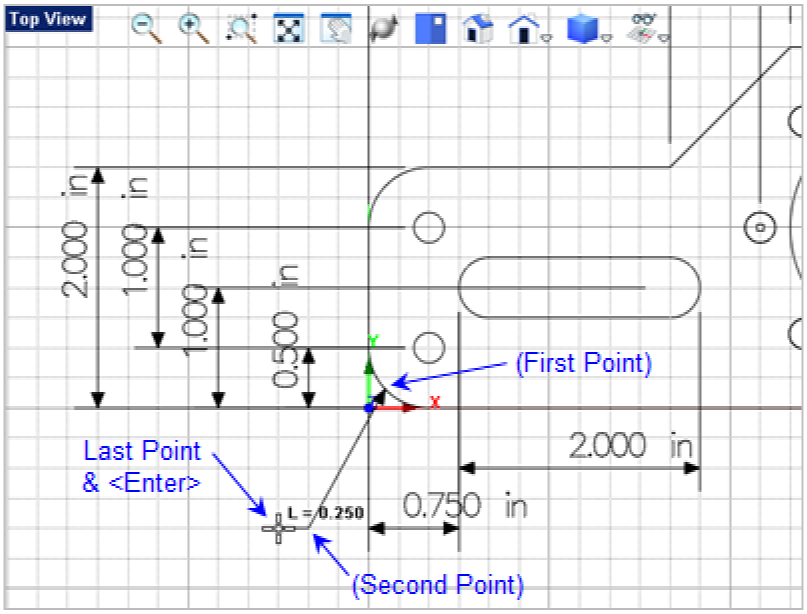


15. Now we will create some leader lines. From the Dimension [Dimensions] tab select the Leader Line [Leader Line] command.

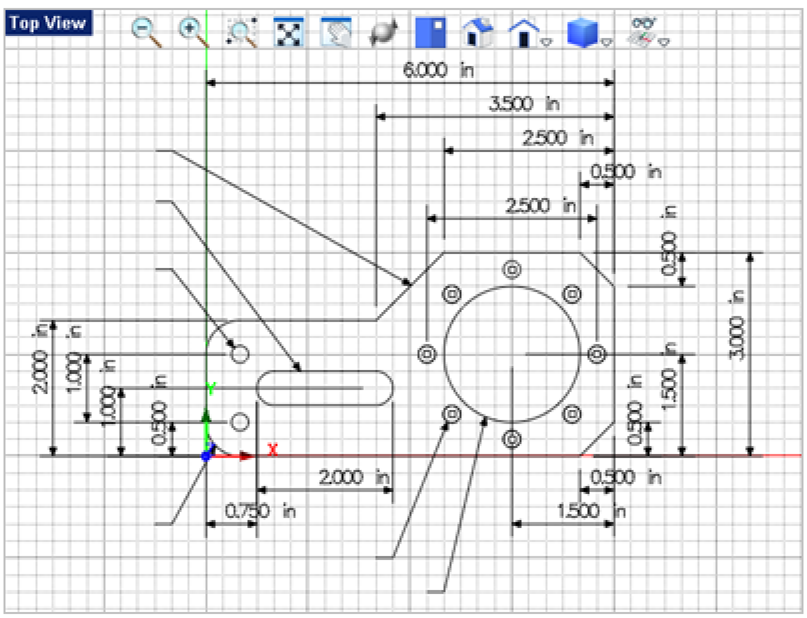
16. The first command prompts says: "Enter the text string." We will add the annotation separately so you can press <Enter> to continue.

17. The next prompt says: " Pick point or enter coordinates x,y, and z" Change the Object Snap to Near and select a point on the lower fillet for the first point.

18. Then pick additional points to define the leader line and the press <Enter> or right-click to end the command. The points to pick are shown in the illustration below:



19. Now repeat the Leader Line command and add the leader lines shown below:

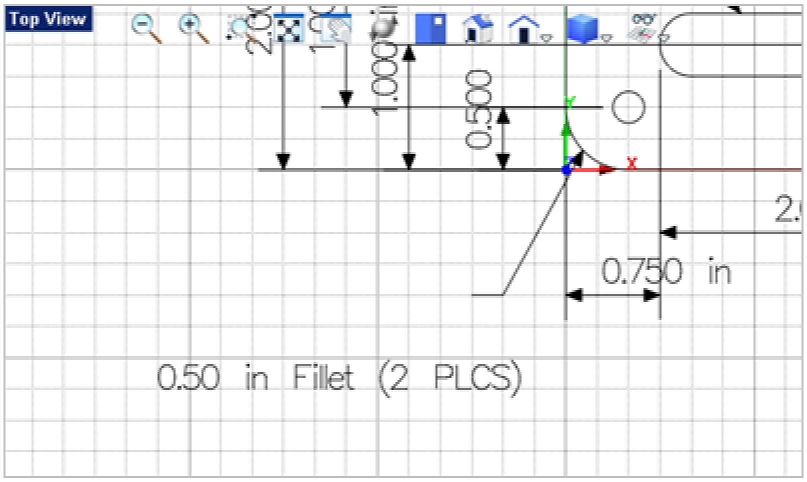


20. Now we will add the annotation text. From the Dimension 

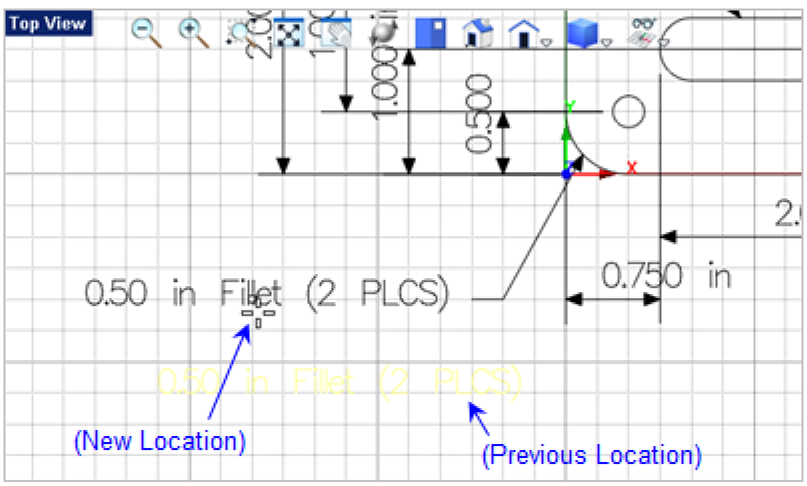
tab select the Annotation 

command.

21. For the text string, at the command prompt  enter 0.50 in Fillet (2 PLCS) and press <Enter>

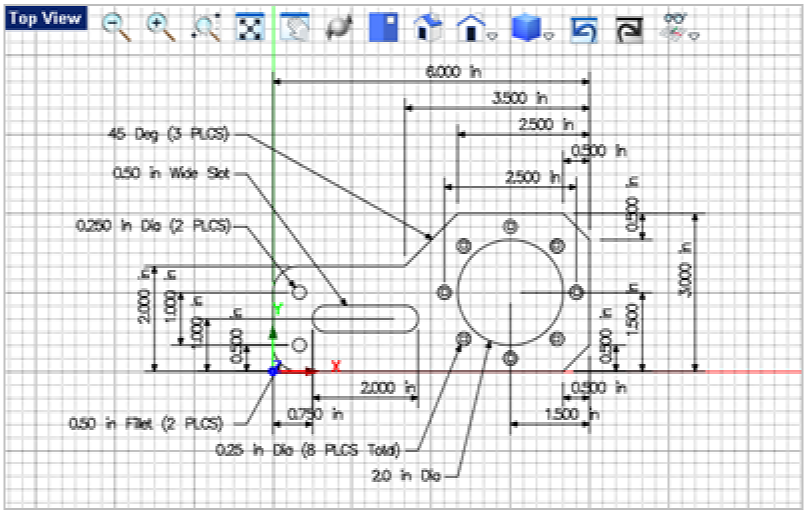


22. Now select the text you just created and left-click-hold to drag it into position to the left of the leader line as shown below. If you toggle the Grid Snap  off you can position the text more precisely.



23. Repeat the Annotation command to create the additional leader text shown below:

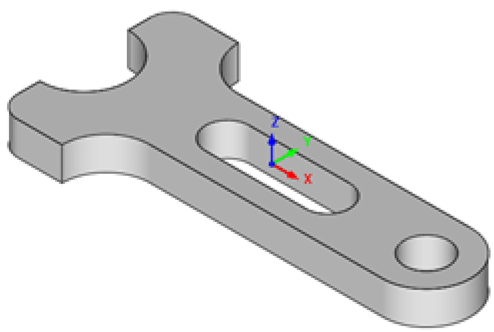
24. Save the file as Drawing_and_Dimensioning_Completed.vcp.



Congratulations you have completed this exercise!

7.3 #3: Model a Spanner Plate

In this exercise you will model the Spanner Plate shown below using Curve Modeling and Solid Modeling tools. The exercise includes the creation of Points, Lines, Fillets, Mirroring, Merging and Extruding curves.



Model a Spanner Plate

What you will learn:

In this exercise you will perform the following VisualCAD tasks:

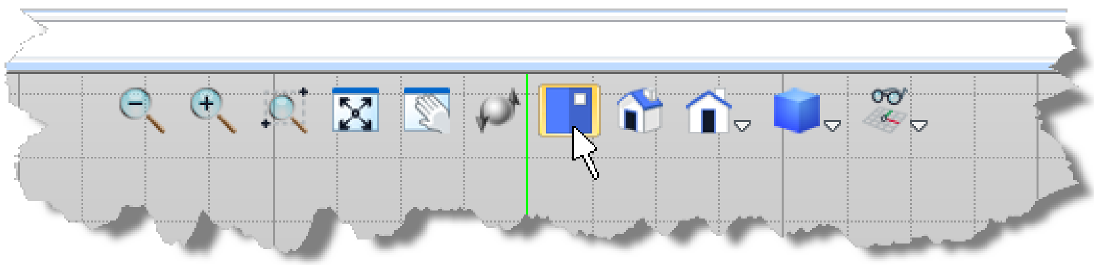
1. Set to the Top View.
2. Create Reference Points.
3. Create Inner Cutouts.
4. Trim Curves.
5. Offset, Extend & Trim Curves.

6. Fillet Curves.
7. Mirror Curves.
8. Merge Curves.
9. Extrude Curves.

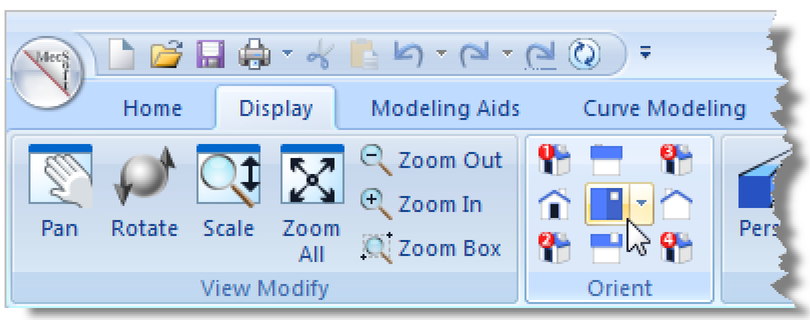
7.3.1 Set to the Top View

Set to the Top View

Set the view to Top View. This can be done by selecting the Top View icon from the View Toolbar or the Orient pane of the Display Ribbon Bar.



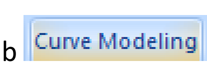
View Toolbar



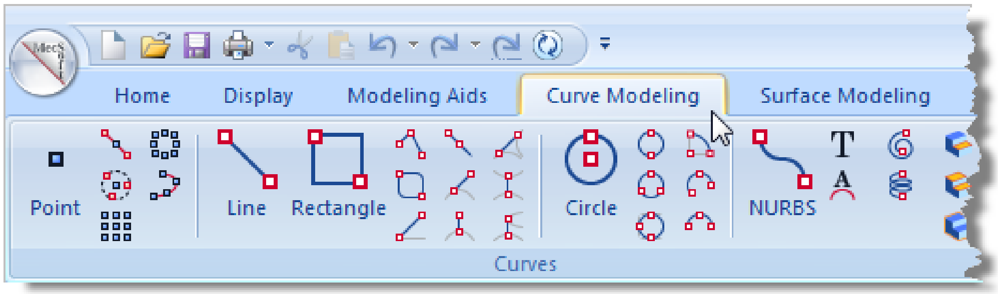
Display Ribbon Bar - Set to the Top View

7.3.2 Create Reference Points

Create Reference Points

1. Select the Curve Modeling tab 

from the top Ribbon Bar.
2. Select the Curve Modeling Ribbon Bar:



Select the Curve Modeling Ribbon Bar

3. Select Point 

.

4. Place the point at the Origin by moving the mouse cursor over to origin and with a Left Mouse Button Click or by entering 0,0 in the command bar 

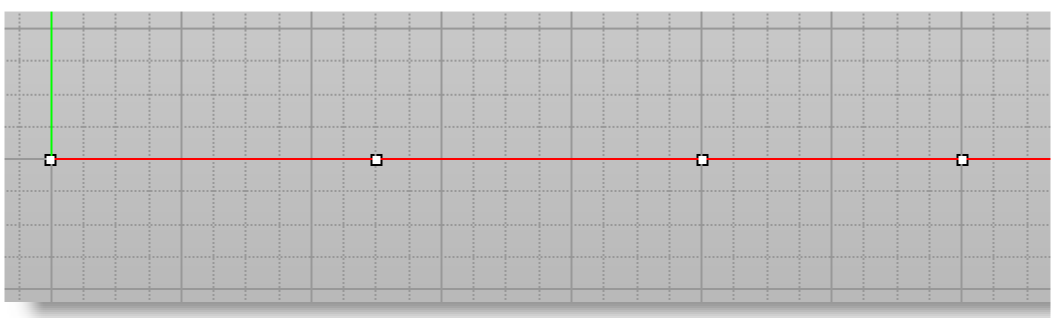
.

5. As you move the mouse on the viewport, the co-ordinate value at the mouse location is displayed under the status bar.

   Create additional points by repeating the Create Point command at the following coordinate locations.

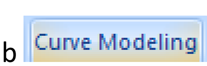
   2.5,0
   5.0,0
   7.0,0

   The created points are as shown below.



7.3.3 Create Inner Cutouts

Create Inner Cutouts

1. Select the Curve Modeling tab 

from the top Ribbon Bar.

2. Keep the Curve Modeling Ribbon Bar displayed.

3. Select the Circle Center, Center, On Pt icon.

The command input bar  would now prompt you to specify the center for the circle.

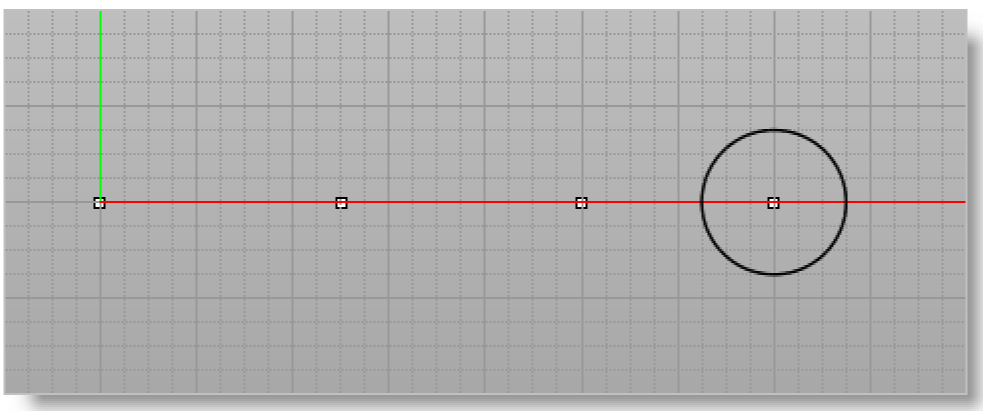
Command Prompt:
Center, On Pt:: Pick center point or enter coordinates x,y and z

4. For the center point, pick last created point or type 7,0

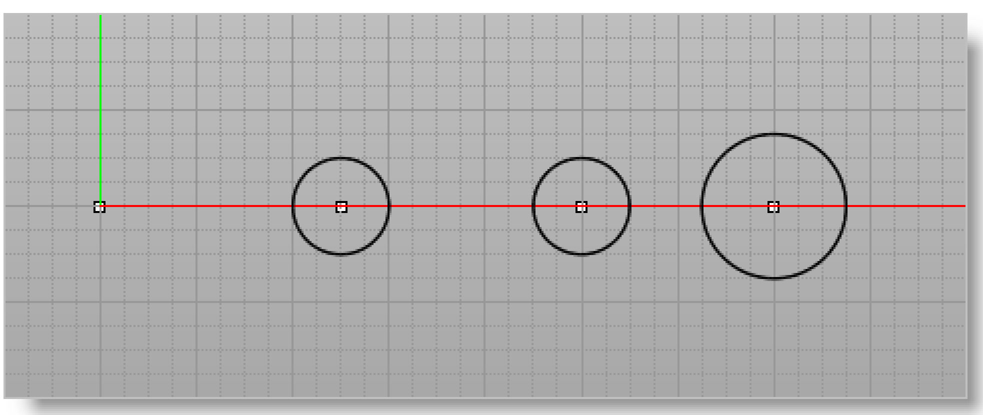
Command Prompt:
Center, On Pt:: Enter point on circle::[D=0.500000]

5. Specify 1.50 for Diameter in the command bar  and press Enter.

The created circle is as shown below.

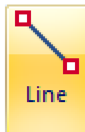


6. Repeat the Circle command to create 2 more circles at the center points (2.5,0 and 5,0) enter a Diameter of 1.00 for each circle.

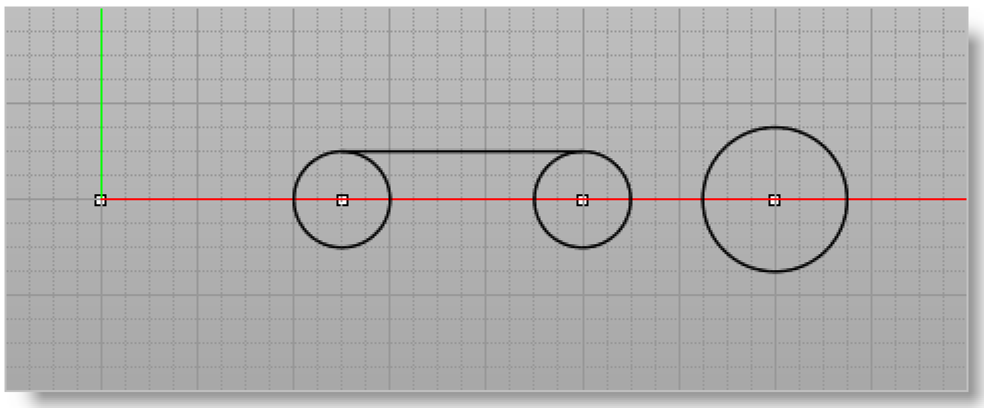


7. From the Status Bar, toggle Grid Snap off and then toggle Quad Point Snap on.

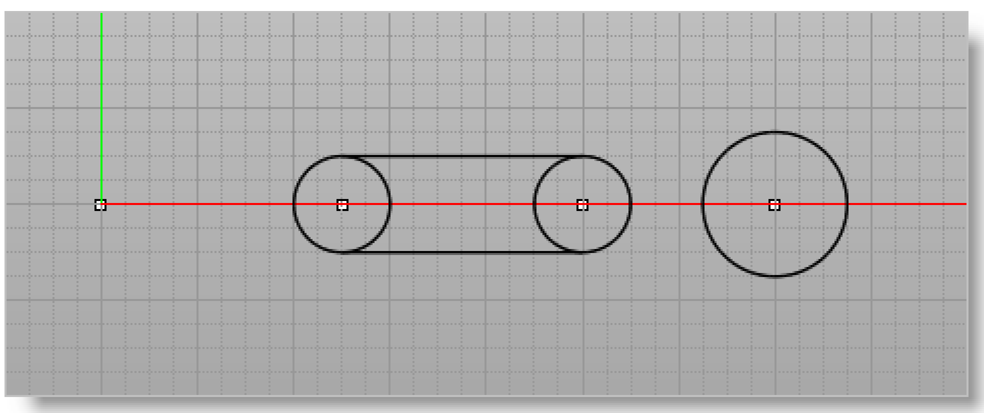
8. Select the Create Single Line icon.



9. Create a line between the top quadrants points of the smaller circles.



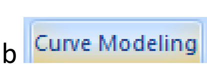
10. Repeat the line for lower quadrant points.



7.3.4 Trim Curves

Trim Curves

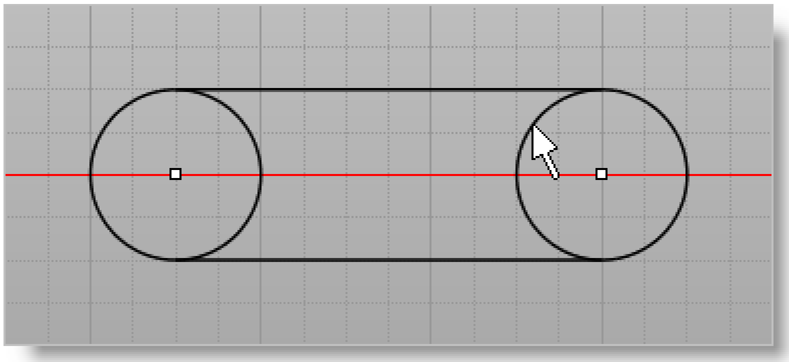
The circles must be trimmed.

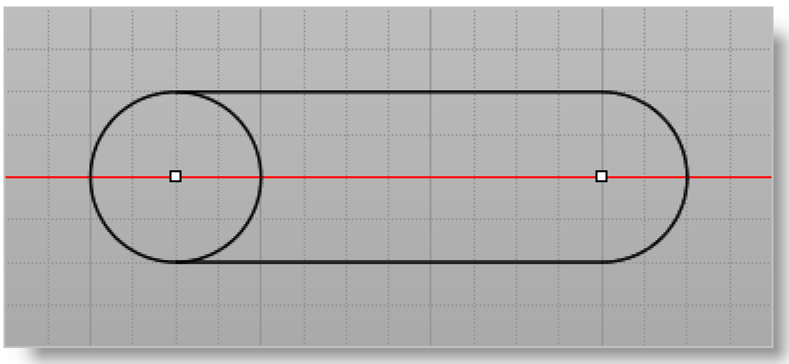
1. Select the Curve Modeling tab 

from the top Ribbon Bar.

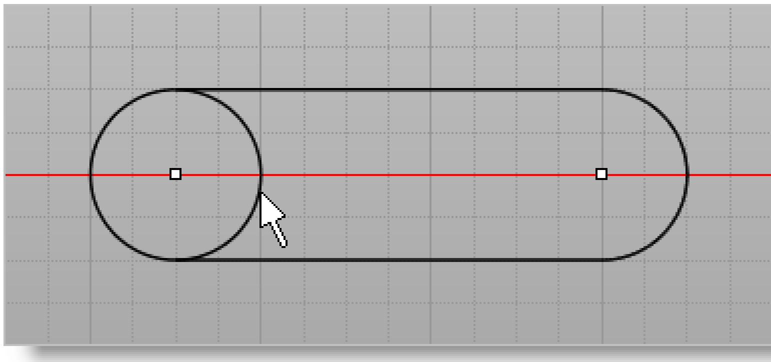
2. Select Trim 

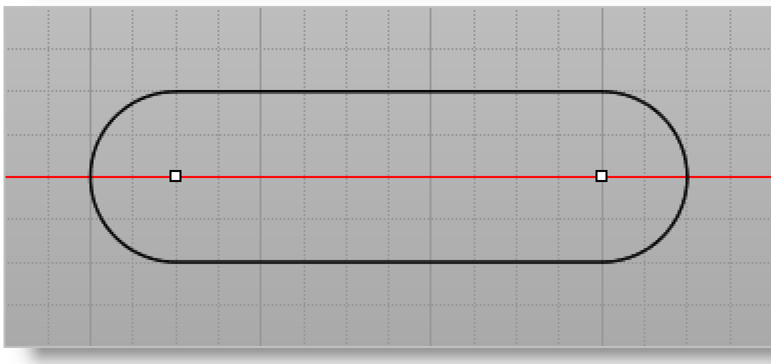
3. Click on the portion of the circle you want to delete.





4. Repeat for the circle portion on the other side.





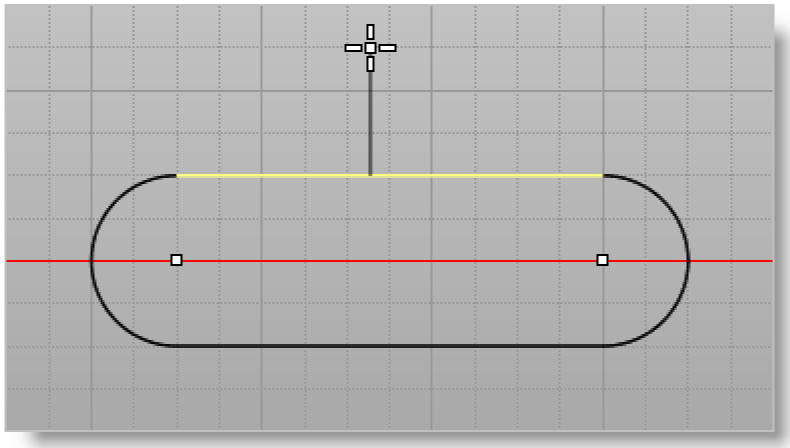
7.3.5 Offset, Extend & Trim Curves

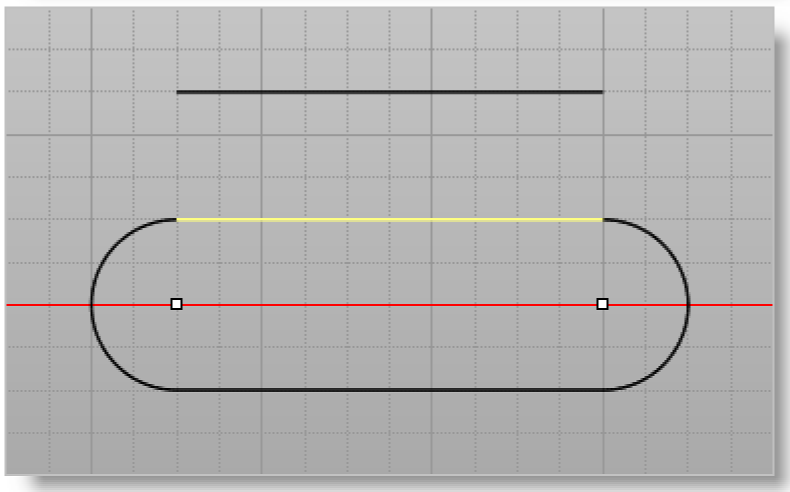
Offset, Extend & Trim Curves

1. Select the Curve Modeling tab Curve Modeling from the top Ribbon Bar.
2. Keep the Curve Modeling Ribbon Bar displayed and then select the top horizontal line.
3. Pick the Offset Curve 

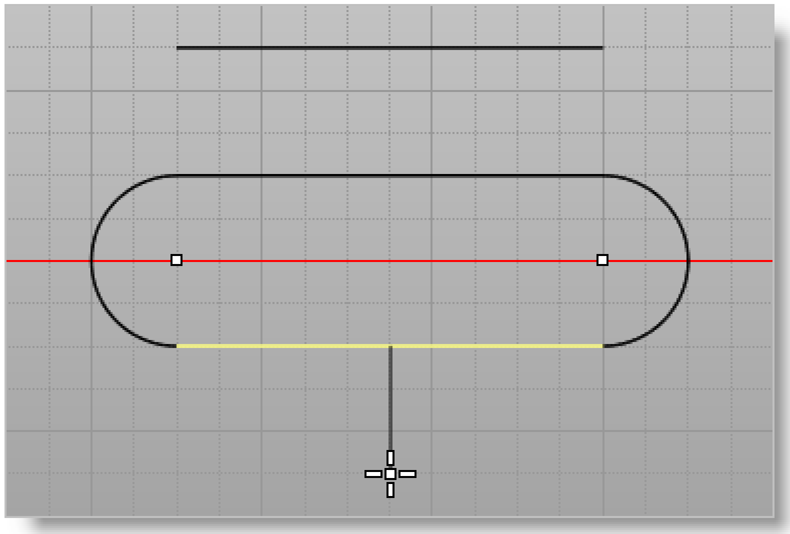
icon from the Edit Curves pane.
A preview line will appear, indicating the direction of offset. Point the preview line to offset the curve outward and then type 0.75 from the keyboard and press Enter.

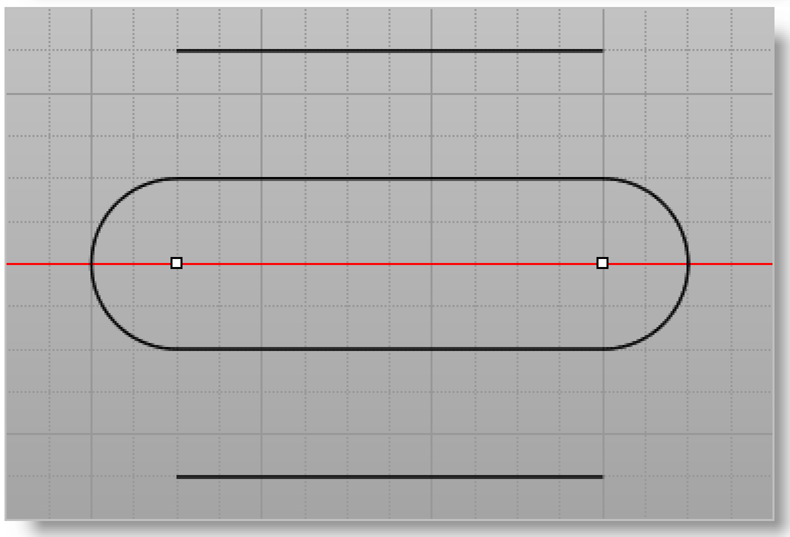
> For Offset Curve, your cursor points to the offset direction, so do not move the cursor to the Command Prompt Command field. Simply type 0.75 and press Enter to input the value.





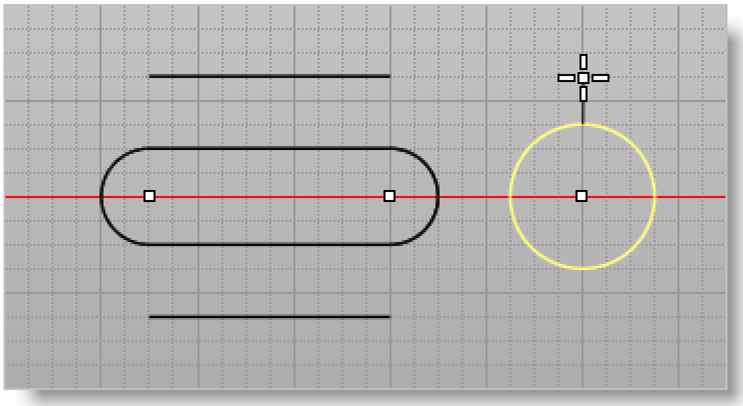
4. Repeat the offset for the horizontal line at the bottom, making sure to keep the cursor and directional indicator line pointing downward.

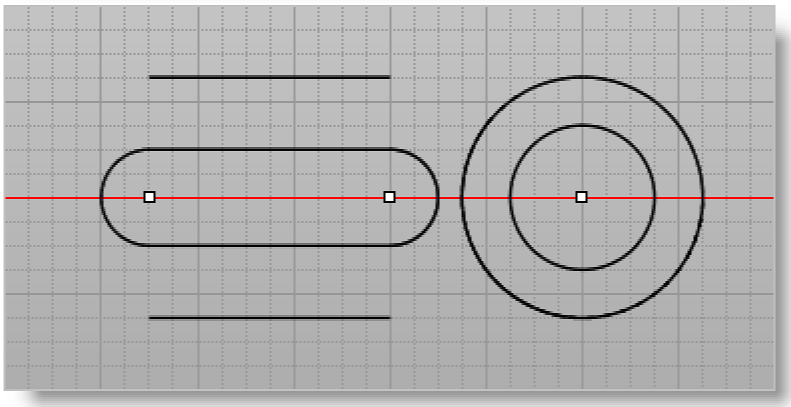




5. Select the larger circle on the right and then pick the Offset Curve icon again.

6. With the cursor, point the offset direction outward from the circle, type 0.50 and press Enter.

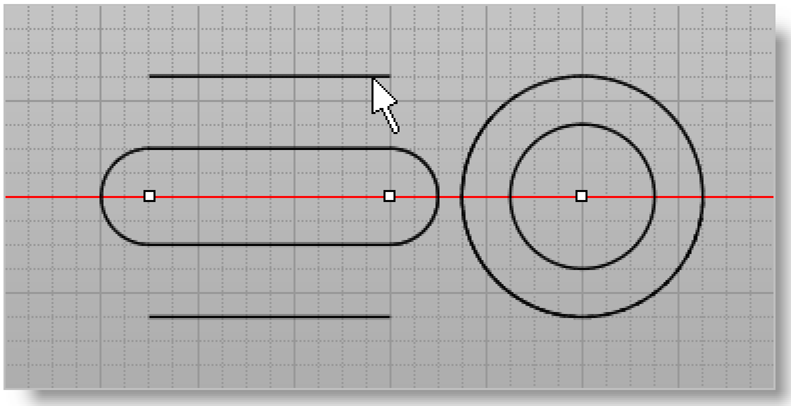


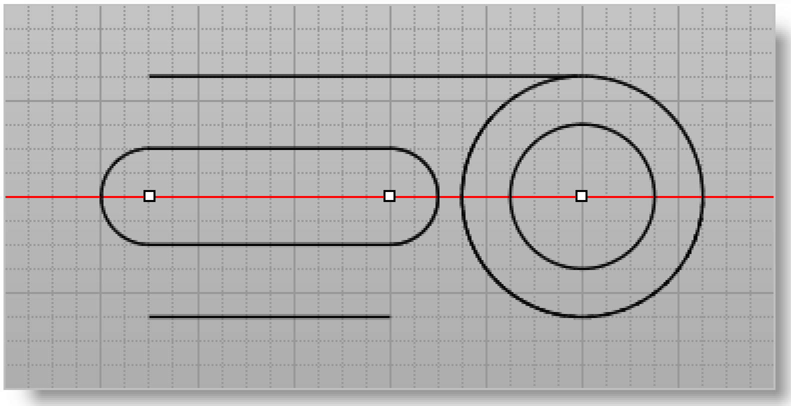


7. To modify the offset lines so that they meet the larger circle, pick the Extend Curve 

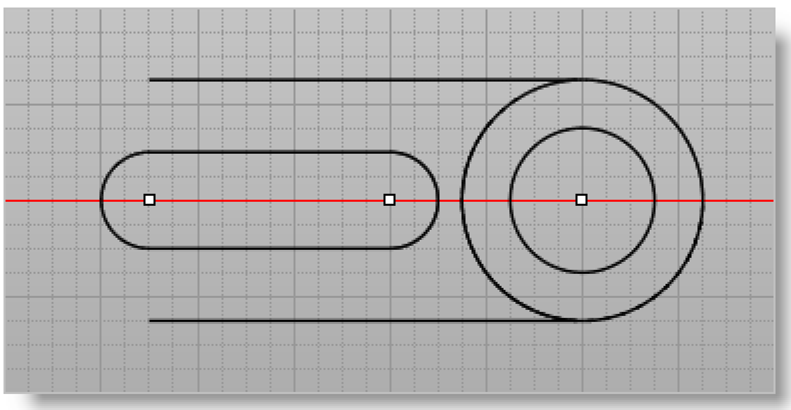
icon from Edit Curves pane. It is located just to the right of the Offset Curve icon.

8. Click one of the offset lines towards the right. The line extends to the point where it meets the circle.

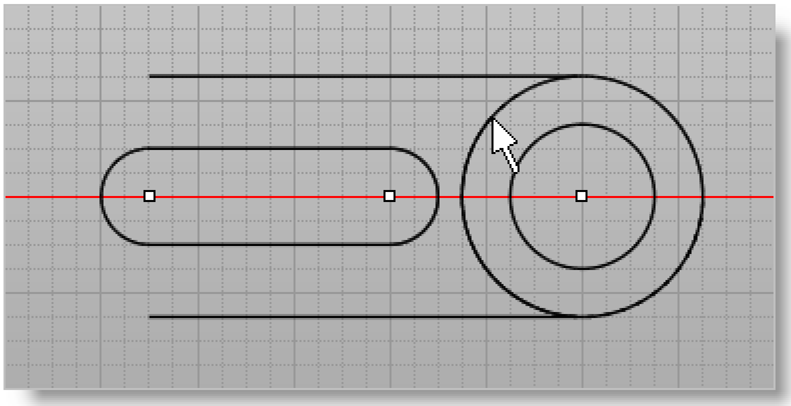


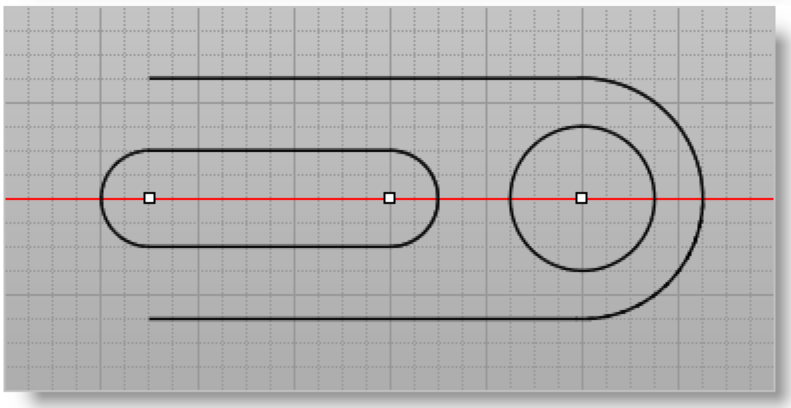


9. Repeat the extend curve for the line at the bottom.



10. Select Trim  and trim the large circle where it meets the two extended lines.

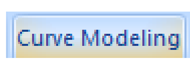




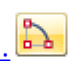
7.3.6 Fillet Curves

Fillet Curves

We will now create the left section of the spanner using the filleting and mirroring tools. We will start by creating the arc at the end of the spanner.

1. Select the Curve Modeling tab 

from the top Ribbon Bar.

2. From the Curve Modeling Ribbon Bar, select Center, Start, Angle Pts. 

   Command Prompt:
   Center,Start,Angle Pts:: Pick center point or enter coordinates x,y and z

3. For the Center, type 0,0 in the Command Prompt 

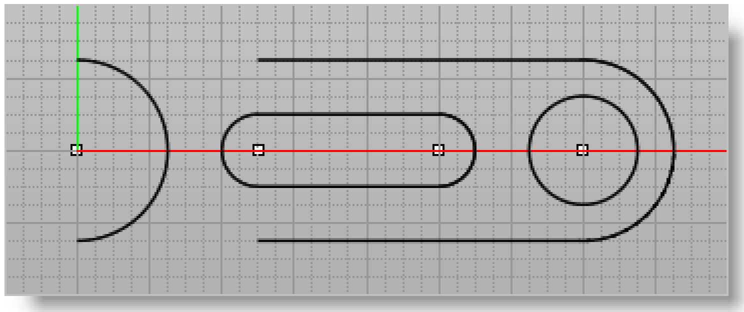
and then press Enter.

4. For Start, type 0,1.25 in the Command Prompt 

and then press Enter.

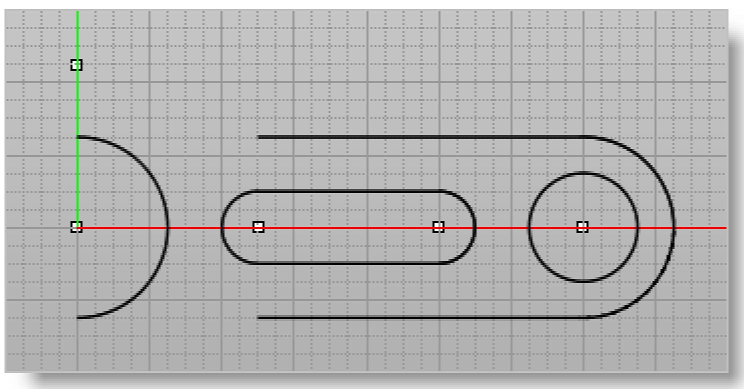
5. For End, type -180 in the Command Prompt 

and then press Enter.
Your drawing should look like the following:

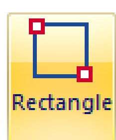


6. Now create a reference point located at 0,2.25 using the Point 

command. The icon is located on the left side of the Curve Pane of the Curve Modeling Ribbon Bar.

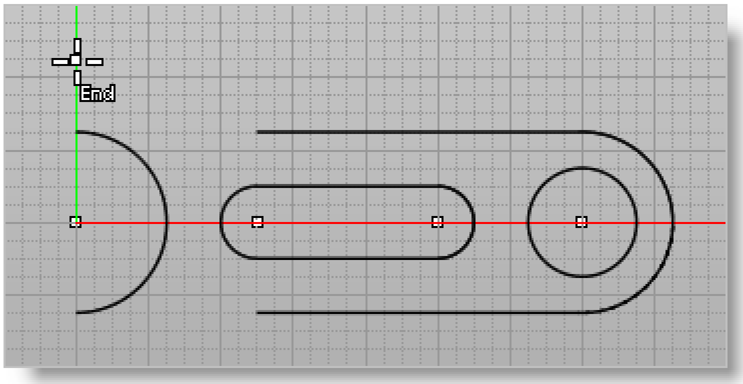


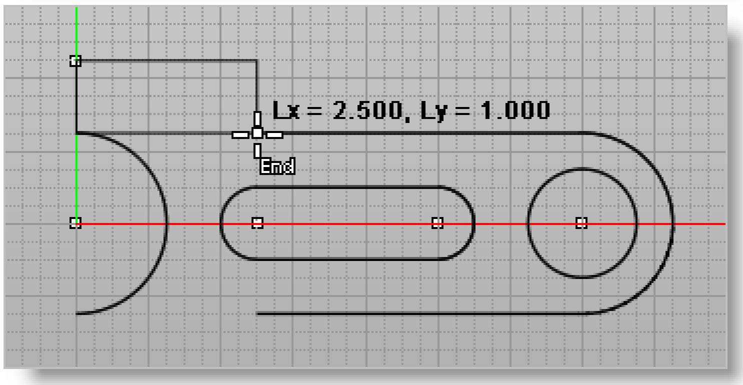
7. From the Status Bar, toggle End Point Snap  On.

8. Now, select Rectangle 

.

9. For the 1st corner pick the point you just created and for the other corner snap to the End Point of the horizontal line as shown below.





10. This rectangle is one object – one multi-segmented curve. In order to be able to edit or modify this rectangle, it must be broken into individual lines.

     To do this, select the Explode icon. It is located on the far right side of the Edit Curves pane of the Curve Modeling Ribbon Bar.

11. Select the rectangle and then right-click or press Enter.
    Now each line of the rectangle is a separate object.

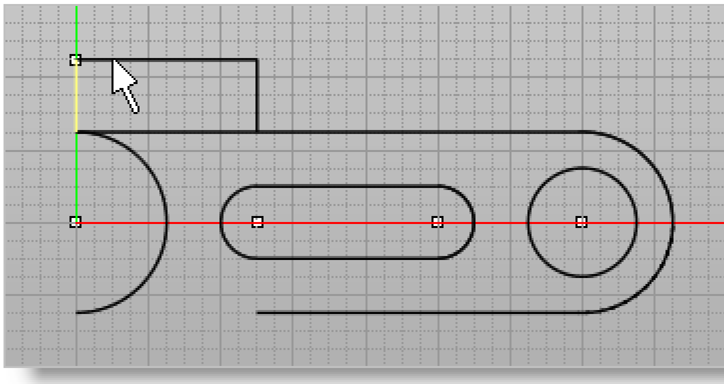
12. Select Fillet 

.

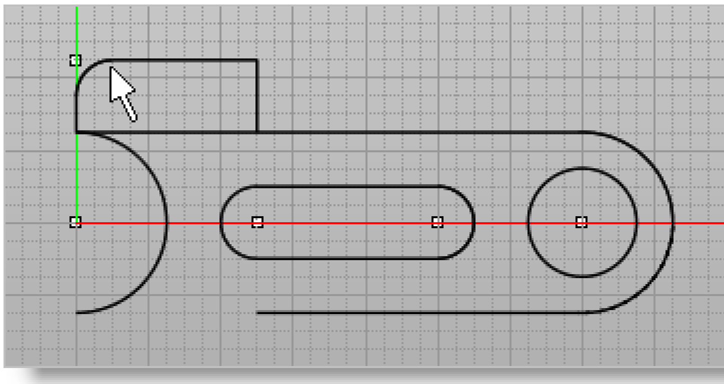
    Command Prompt:
    Fillet:: Enter fillet radius [R=0.500000]

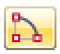
13. For Radius enter 0.5 and press Enter.

14. Select the 2 curves as shown below to fillet the curves.



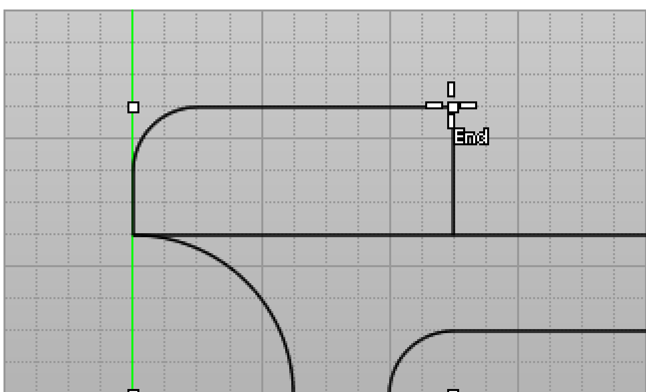
The filleted curve is as shown below.



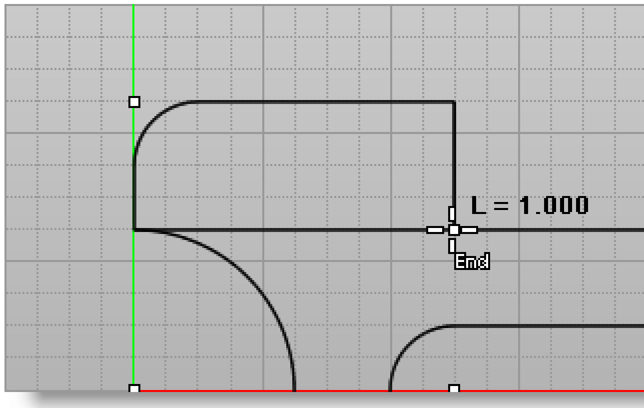
15. Let's now create another arc. Again, from the Curve Modeling Ribbon Bar, select Center, Start, Angle Pts. 

    Command Prompt:
    Center,Start,Angle Pts:: Pick center point or enter coordinates x,y and z

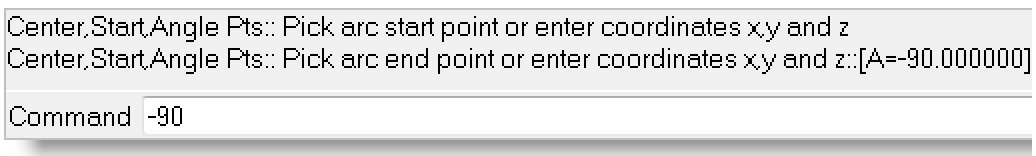
16. For Center pick the end point on the upper right corner of the rectangle.



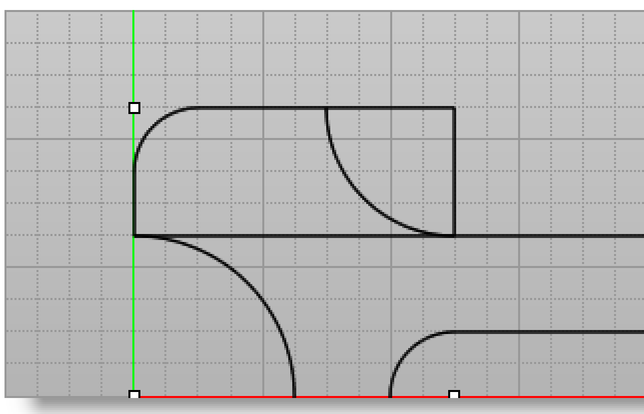
17. For Start pick the lower right corner of rectangle as shown below.



18. For Angle, type in -90 and then press Enter to create an arc in the top right corner of the rectangle.



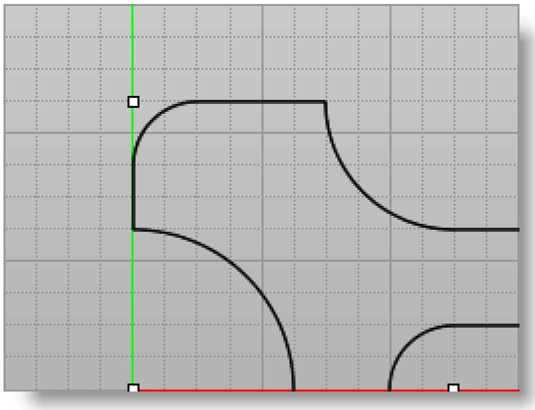
The created Arc is shown below:



19. To clean up the corners we select Trim 

again.

20. Select trimming curves to obtain the shape as shown below:

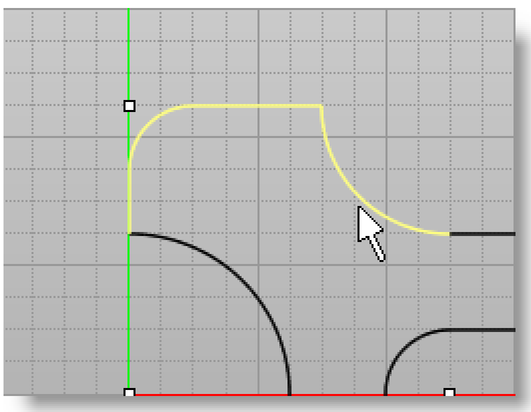


7.3.7 Mirror Curves

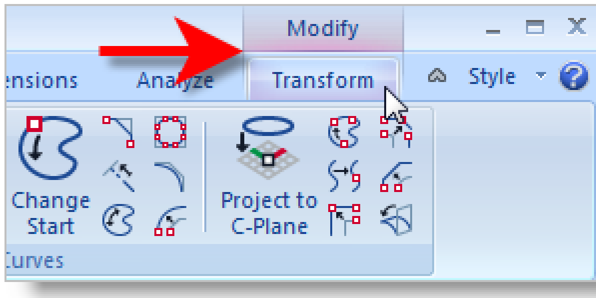
Mirror Curves

We will now select the 4 curves that form the head of the spanner and mirror them.

1. Select the 4 curves as shown below. Hold down the Ctrl key to select multiple objects.

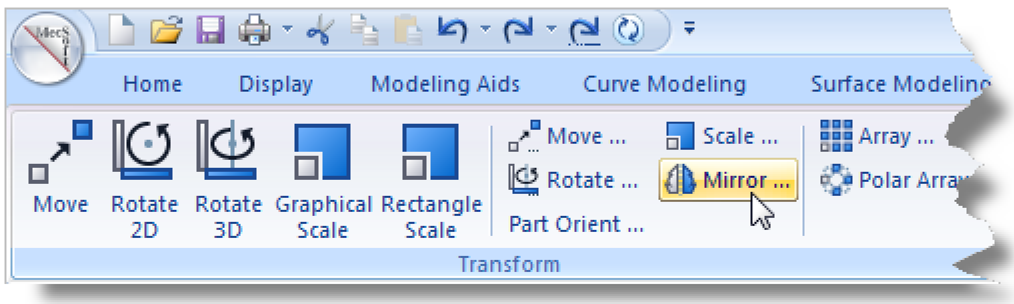


2. With the 4 curves selected, you will notice that VisualCAD's Modify mode automatically becomes active and that the Transform Ribbon Bar becomes available. Refer to the menu image below:



The Transform Ribbon Bar is Activated

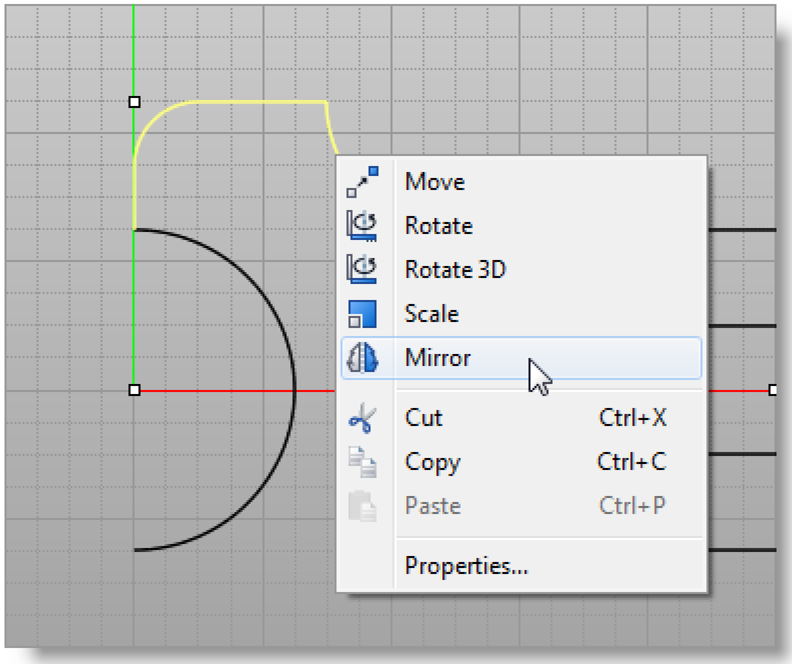
3. Now there are two methods to activate the Mirror command. Select the method that suits you best.
You can select the Transform Ribbon Bar tab and then select the Mirror icon as shown:



The Mirror icon on the Transform Ribbon Bar

- or -

While the 4 curves are selected, you can simply Right-click and select Mirror from the pop-up menu as shown below:

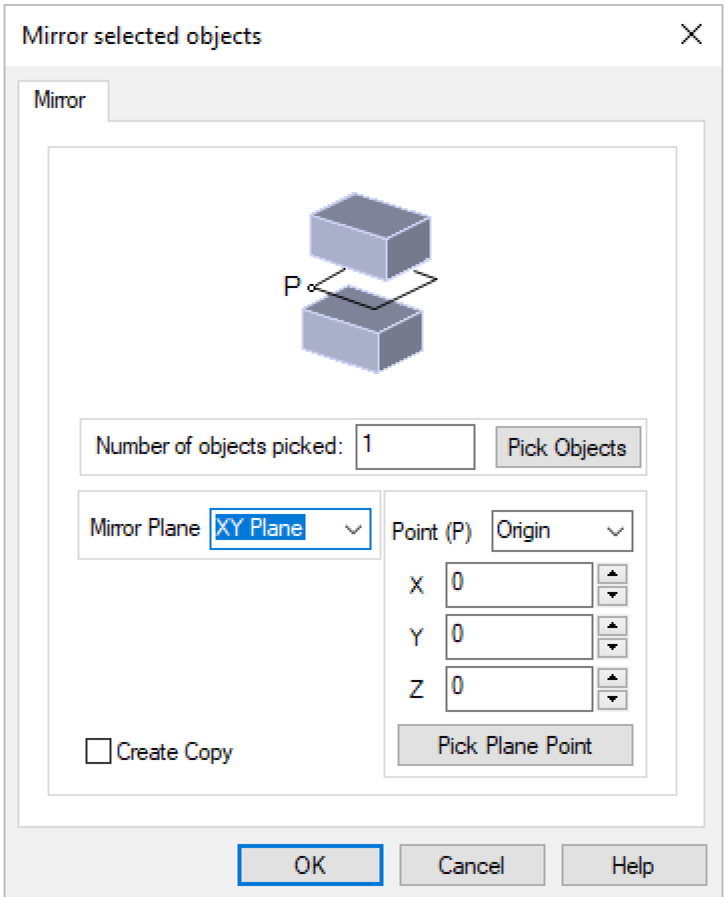


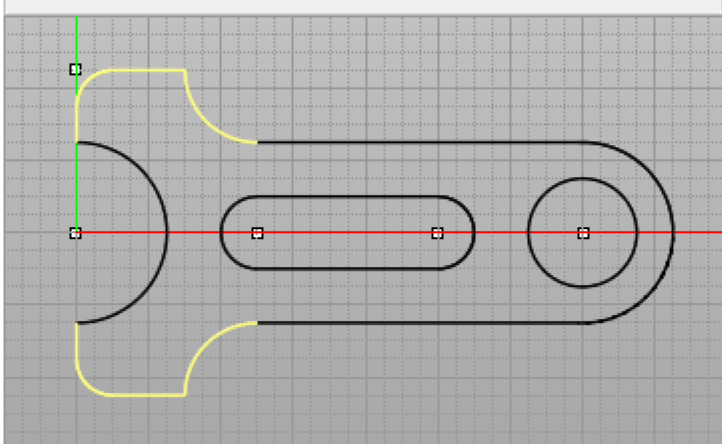
Right-clcik Menu for Editing Objects

4. From the Mirror selected objects dialog box:

   Set the Mirror Plane to XZ Plane
   Leave Point (P) as Origin since the origin lies on the center line of the part.
   Check Create Copy
   Pick OK.

   The curves are mirrored as shown below.



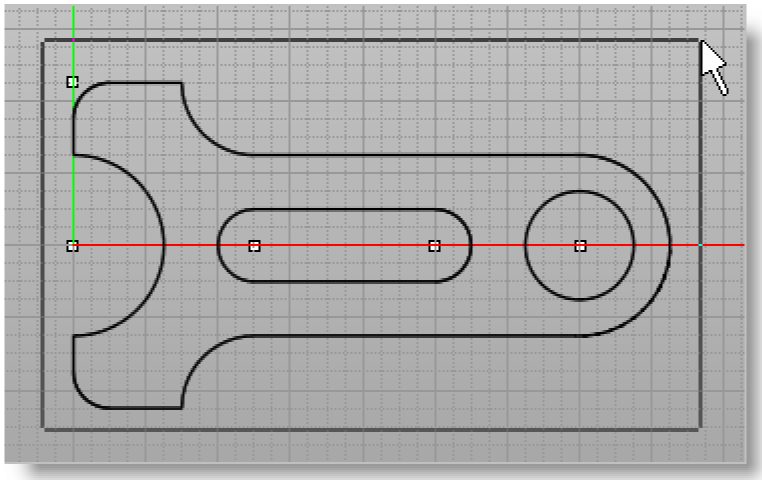


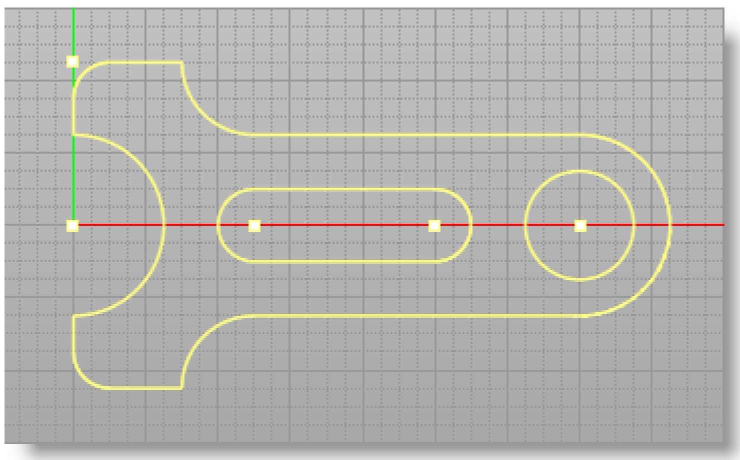
7.3.8 Merge Curves

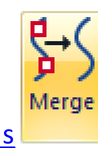
Merge Curves

Now we will merge our curves into three separate closed curves.

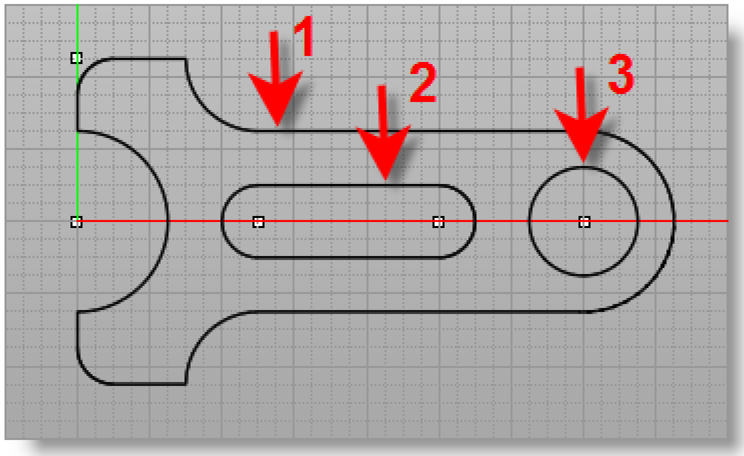
1. Select all the curves by using a crossing window selection:





2. Now, go back to the Curve Modeling Ribbon Bar select the Merge Curves 

icon and then right-click or press Enter. The curves are merged to 3 closed curves as shown below.

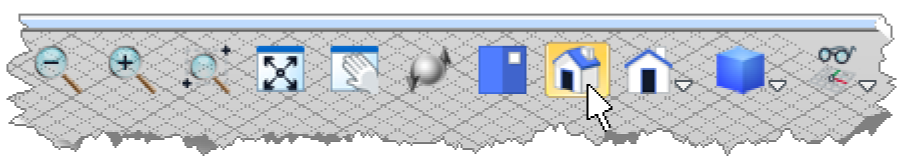


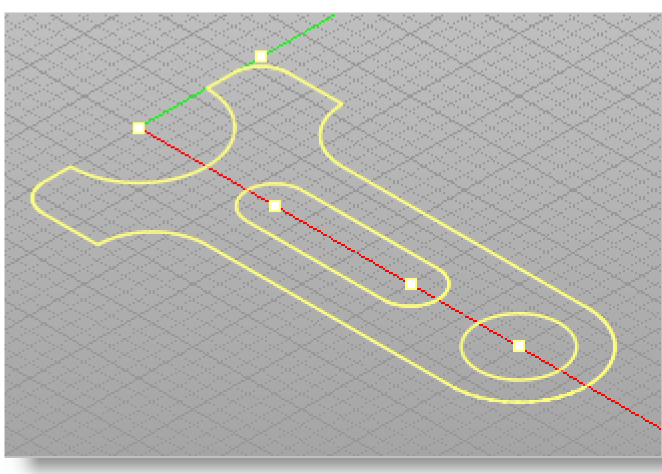
7.3.9 Extrude Curves

Extrude Curves

Now to complete the exercise we will extrude our 3 curves to create the final Spanner 3D model.

1. Select all the curves by using a crossing window selection:
2. Select the 3 closed curves.
3. Select Iso from the View Toolbar 

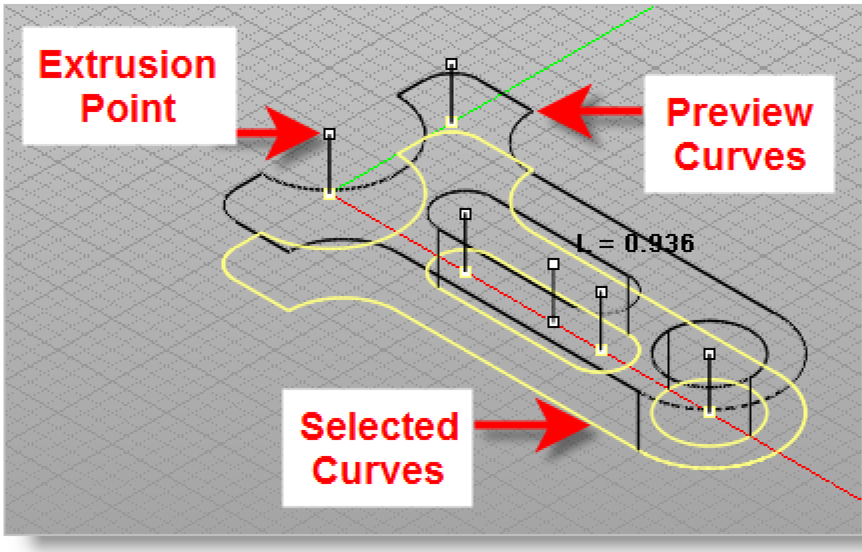




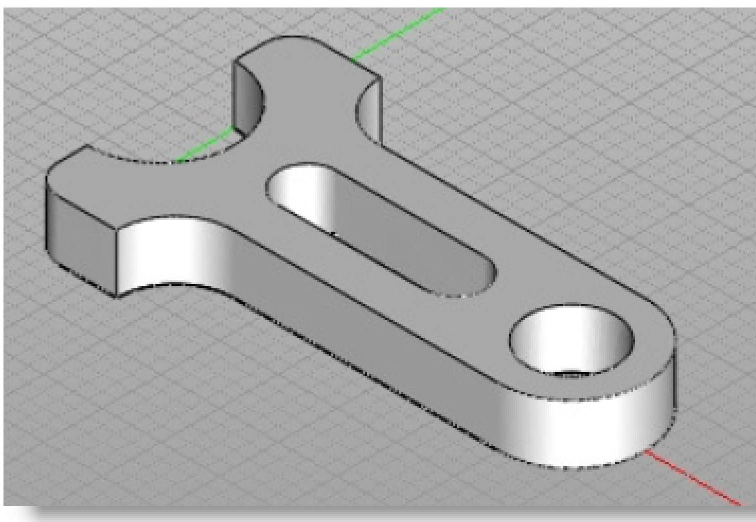
4. From the Solid Modeling Ribbon Bar, select Extrude.

5. Preview curves will appear to determine the direction of extrusion.
Set the extrusion to point in the positive Z direction by moving the cursor above the geometry.
Then type 1.0 in the command input bar and press the <Enter> key.

Command Prompt:
Extrude:: Enter extrusion distance::[L=1.000000]



The curves are now extruded and displayed as shown below.

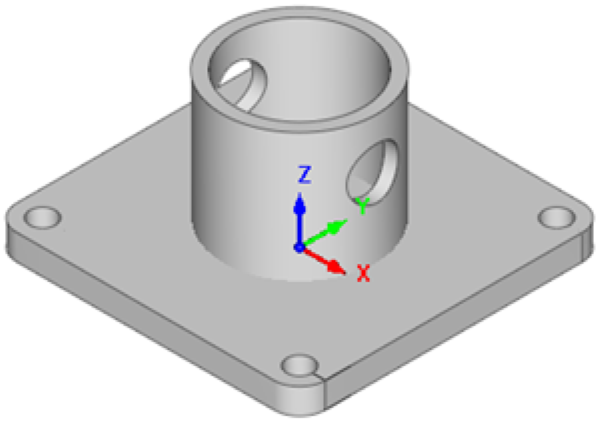


6. Save the file as Spanner_Completed.vcp.

This completes this exercise.

7.4 #4: Model a Base Plate

In this exercise you will model something a bit more complicated, the base plate shown below. You will learn how to draw and model basic shapes to create this 3D model.



Model a Base Plate

What you will learn:

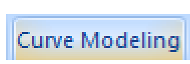
In this exercise you will perform the following VisualCAD tasks:

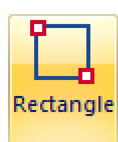
1. Draw the Base Plate Profile.
2. Extrude the Base Plate.
3. Model the Tube.
4. Create a New Layer.
5. Edit Geometry Properties.

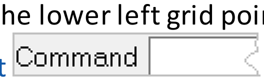
7.4.1 Draw the Base Plate Profile

Draw the Base Plate Profile

In this exercise you will learn the following VisualCAD concepts:

1. Select the Curve Modeling tab 

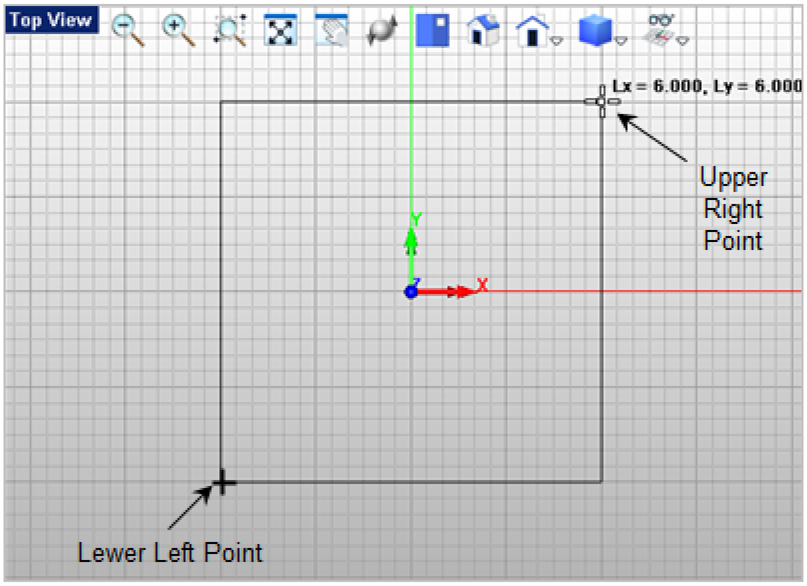
from the top Ribbon Bar.
2. Now select the Rectangle command. 

3. For the first corner of the rectangle, select the lower left grid point location shown below or type -3,-3 in the Command Prompt 

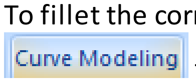
and press <Enter>.

4. For the second corner of the rectangle, select the upper right grid point shown below or type 3,3 in the Command Prompt 

and press <Enter>. This will create a rectangle 6" square.



5. To fillet the corners, select the Fillet 

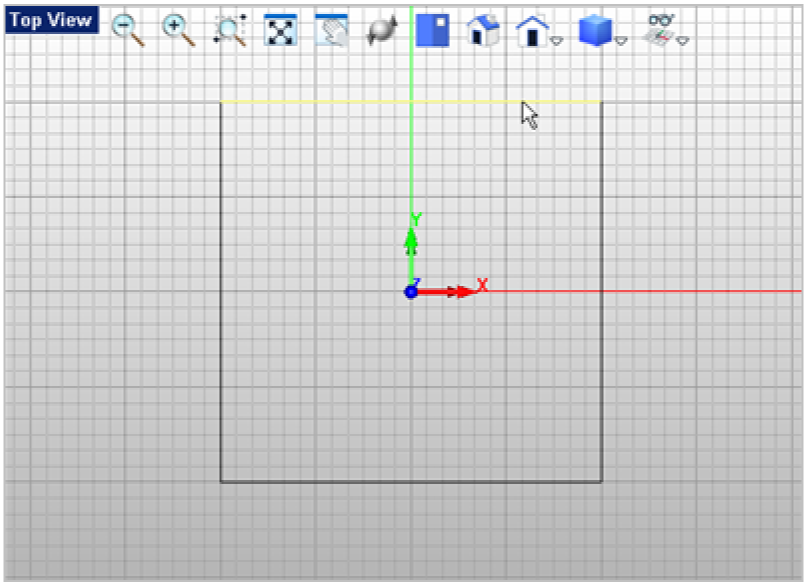
command from the Curve Modeling 

tab.

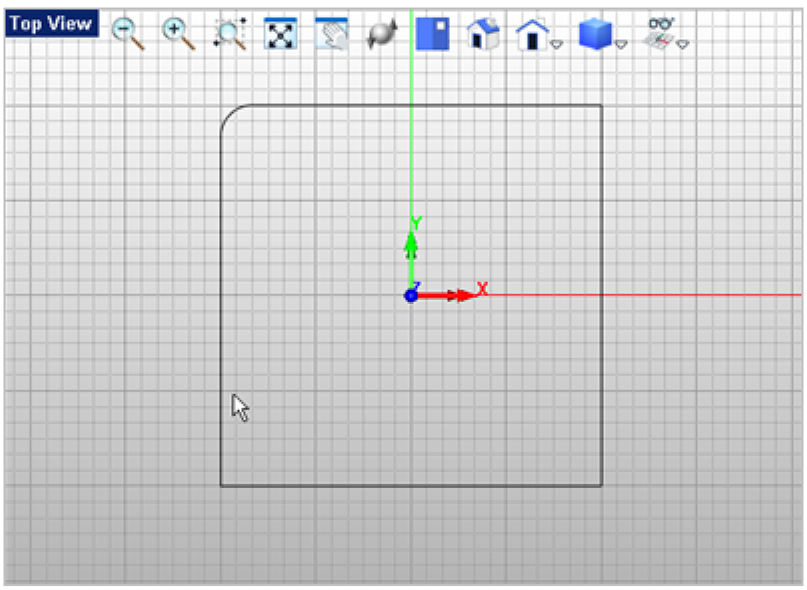
6. For the fillet Radius, enter 0.5 in the command prompt 

and press <Enter>.

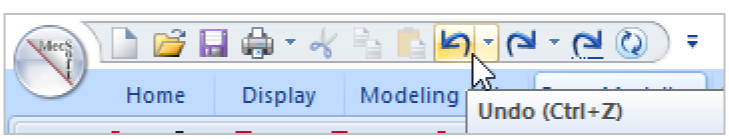
7. Now select the first edge to fillet as shown below. **Note**: When selecting curves to fillet, it is best to select furthest away from the corner that you wish to be filleted.

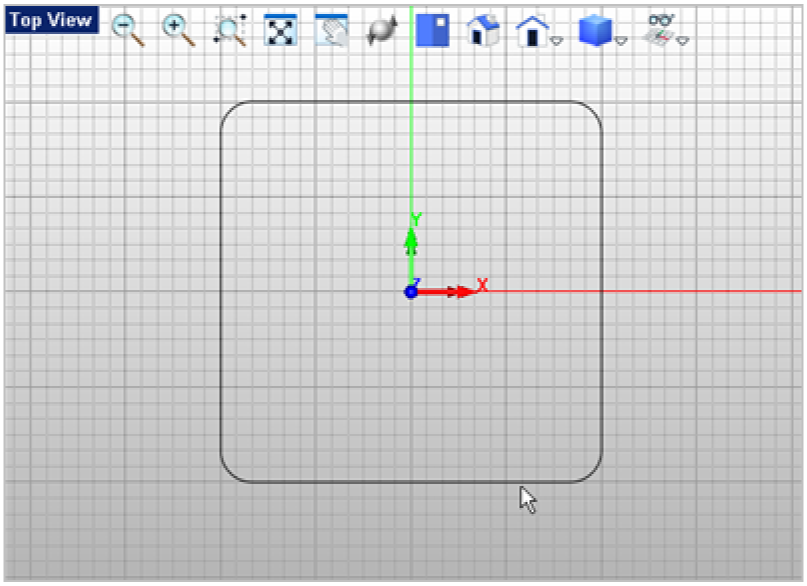


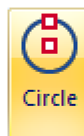
8. Now select the second edge to fillet and a 0.5 radius fillet is added to the rectangle as shown below.



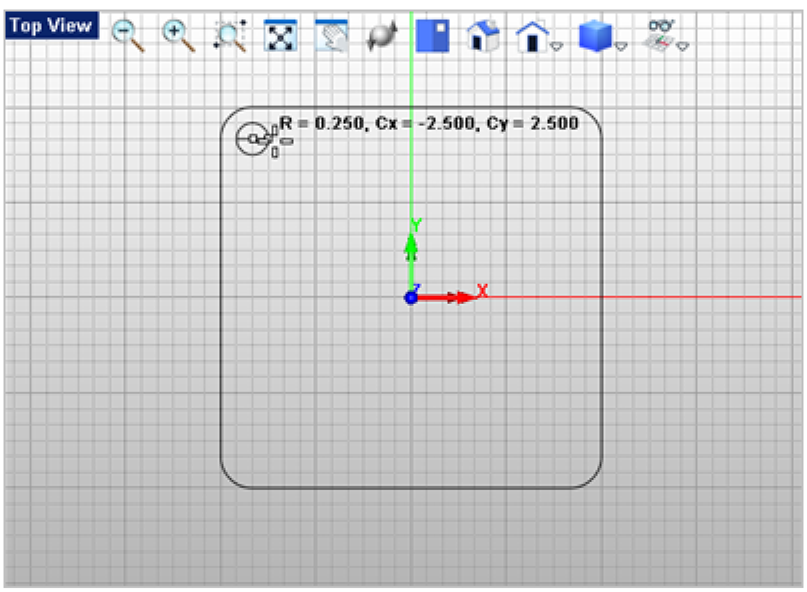
9. Now press <Enter> to repeat the [Fillet]() command and fillet the remaining 3 corners of the rectangle as shown below. If you make a mistake, you can select the [Undo]() command located on the [Quick Access toolbar]() and try again.





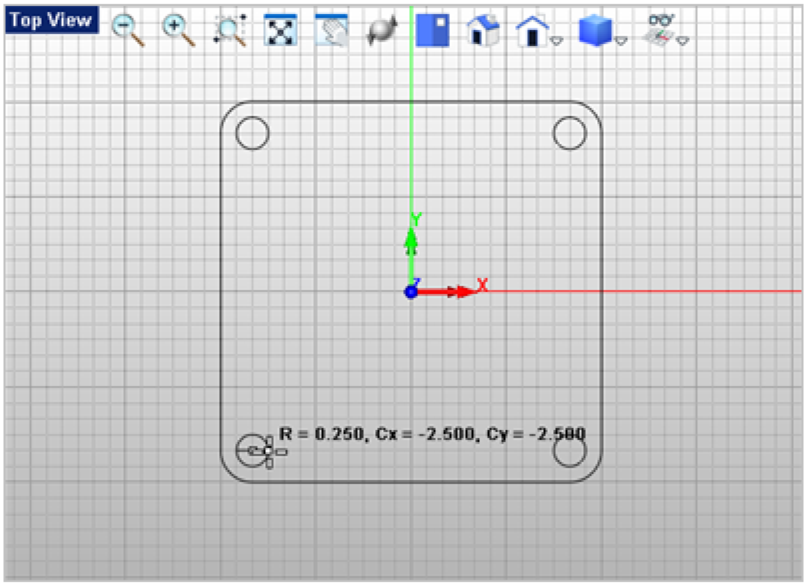
10. Now lets add the mounting holes. From the Curve Modeling Curve Modeling tab, select the Circle on Pt command. 

11. For the center point of the first hole select the grid point shown below or enter -2.5,2.5 in the command prompt Command and press <Enter>.



12. Now for the circle radius you can select the next grid point to the right of the center point you just selected or enter 0.5 for the diameter and press <Enter>.

13. Repeat the Circle on Pt command and draw the other three circles in the locations shown below.

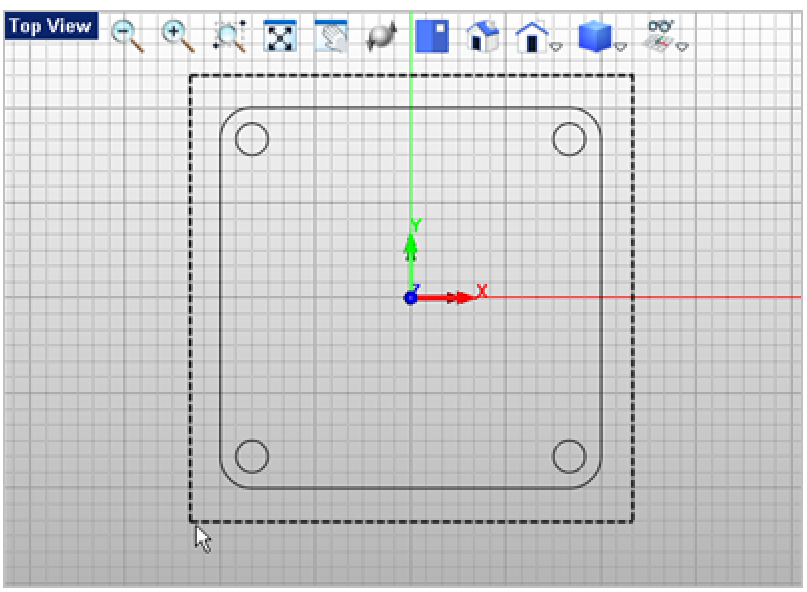


7.4.2 Extrude the Base Plate

Extrude the Base Plate

In this step we will extrude our drawing to model the base plate in 3D.

1. First, window select the drawing by picking two points. Click and hold the left mouse button to drag a window around the drawing as shown below. If you drag from left to right everything completely the window is selected. If you drag from right to left everything that cross over the window will be selected.

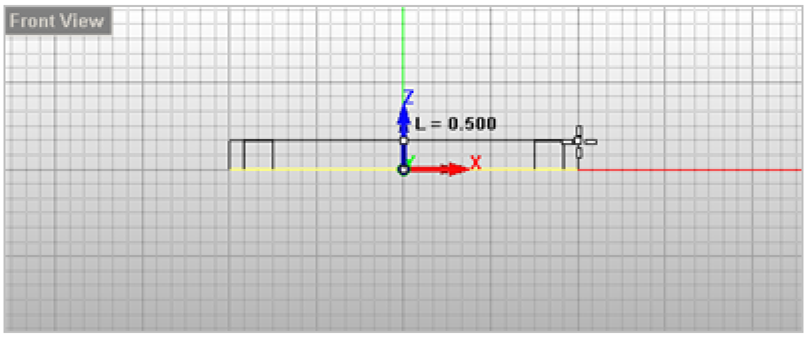


2. After your drawing is selected, go to the Mesh Modeling tab 

from the Ribbon Bar and select the Extrude command. 

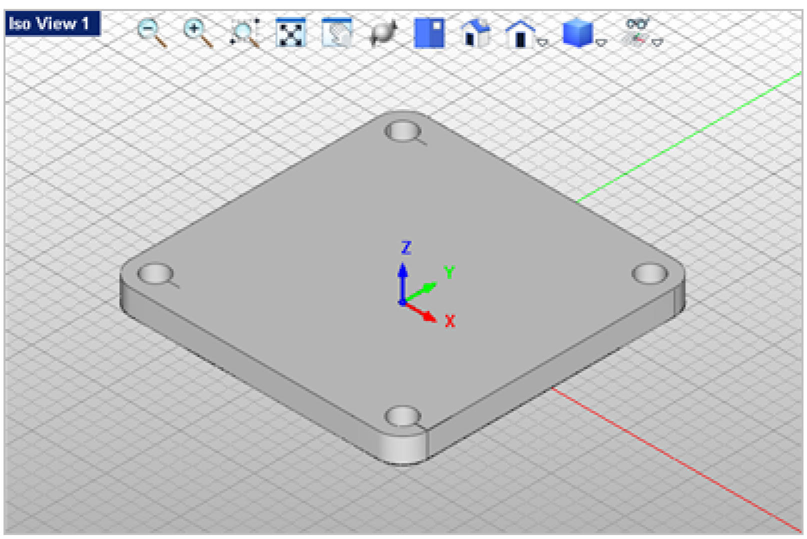
3. Now move the cursor down to the Front View 

and you will see that you can drag the height of the extrusion. Select the second grid point in the positive Z direction or type 0.5 and press <Enter>.

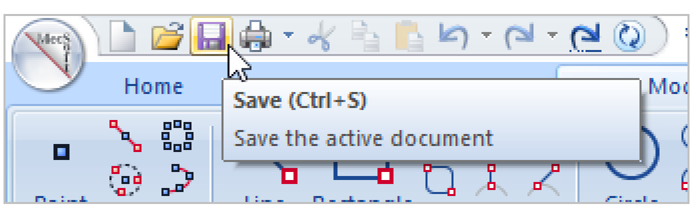


You will see the extrusion display in all four viewports. The Iso View 1 

view is shown below.



4. Save your work by selecting the Save icon from the Quick Access toolbar.



7.4.3 Model the Tube

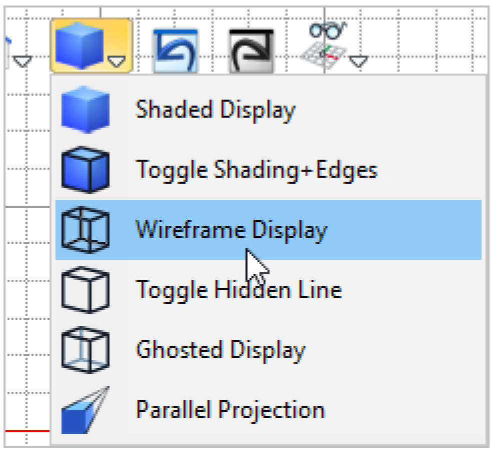
Model the Tube

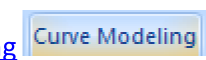
In this step we will draw two concentric circles and then extrude them to model the circular tube that sits at the base plate.

1. Select the Top View 

to activate it.

2. From the View Toolbar drop down the Display menu  and select Wireframe Display. This will allow us to better see our next drawing.



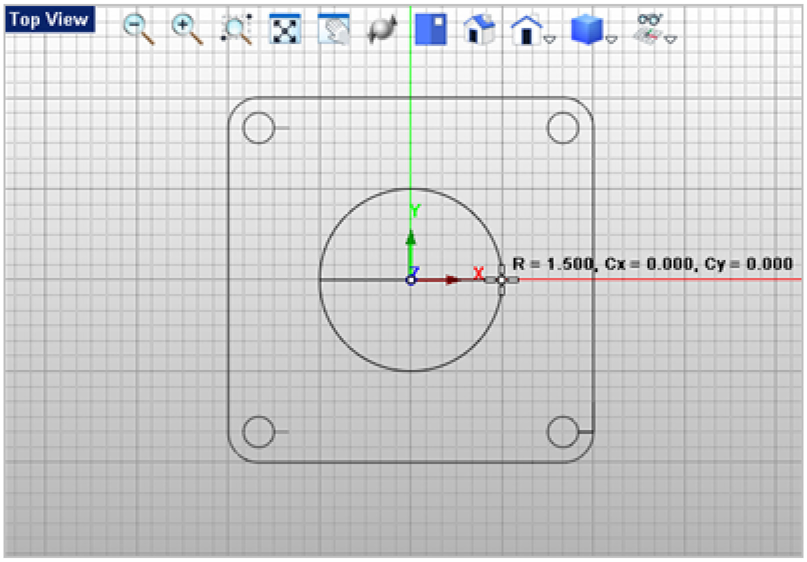
3. Now from the Curve Modeling 

tab select the Circle on Point 

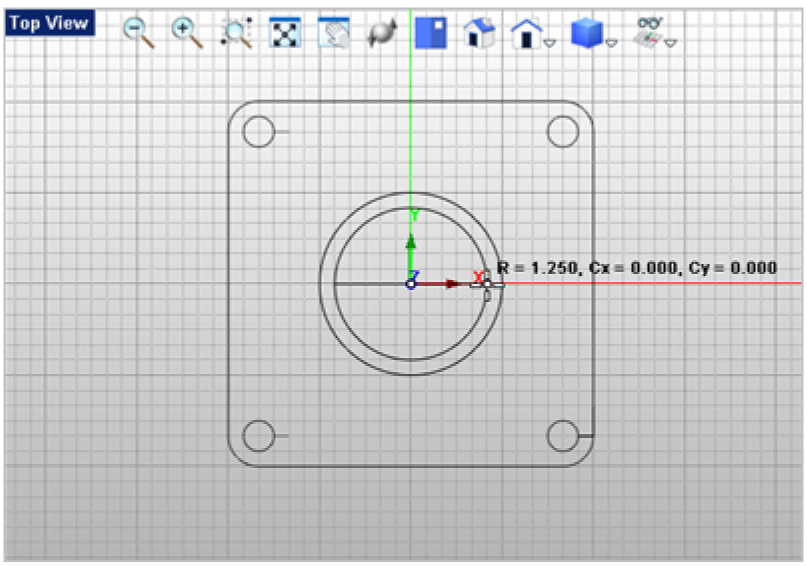
command again.

4. For the center point select the grid point at 0,0 center of the drawing.

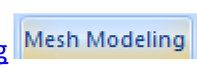
5. Now for the circle diameter you can select the grid point shown below or type in 3 and press <Enter>. A circle 3" in diameter is drawn.



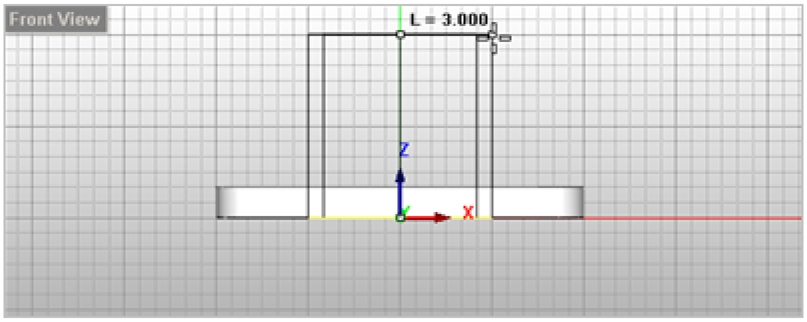
6. Now repeat the command, this time drawing a circle with a diameter of 2.5" in diameter.



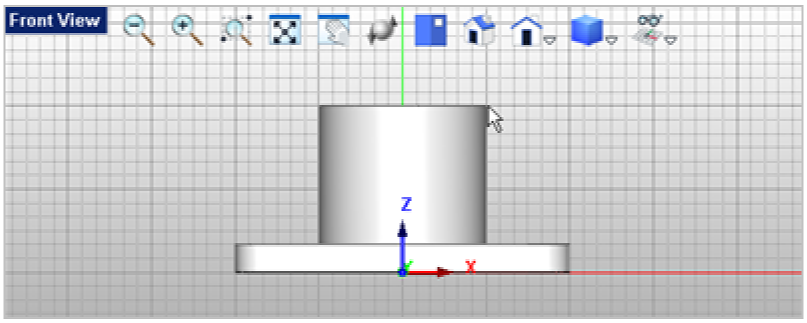
7. Once the two concentric circles are drawn, select them both. You can window select them or while holding down the <Ctrl> key, just pick both of them.

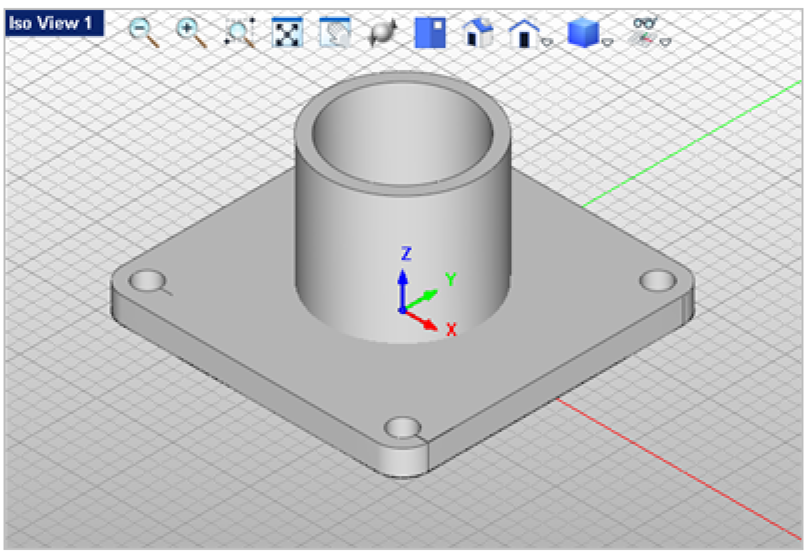
8. With the two circles selected, go to the Mesh Modeling  tab and select the Extrude  command again.

9. Now move the cursor to the Front View  and select the grid point located at 3" in the positive Z direction as shown below.

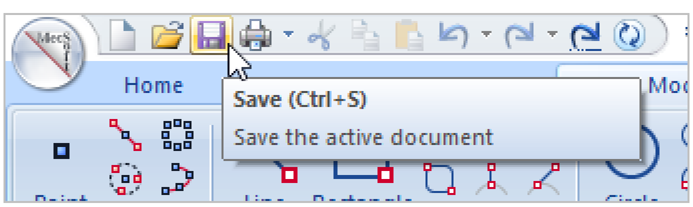


10. The center tube is now created and shown in each window.





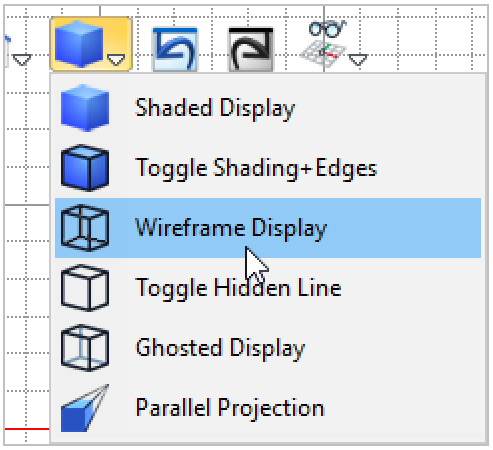
11. [Save](#) your work by selecting the [Save](#) icon from the [Quick Access Toolbar](#).



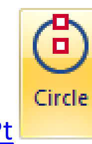
12. Activate the Right View 

by selecting anywhere on the grid.

13. From the View Toolbar, drop down the Display  menu and select Wireframe Display. Again, this will allow us to better see our next drawing.

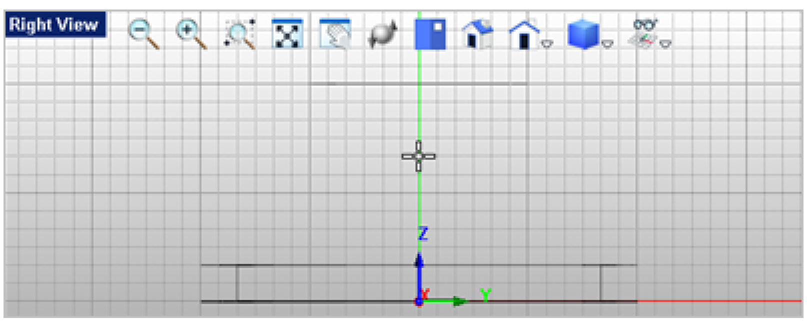


14. From the Curve Modeling 

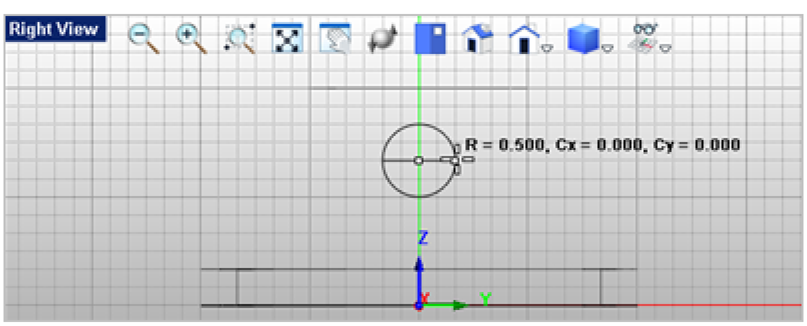
tab, select the Circle on Pt 

command again.

15. For the center point of the circle, select the grid point shown below.



16. Now for the circle diameter you can select the grid point shown below or enter 1.0 for the diameter.



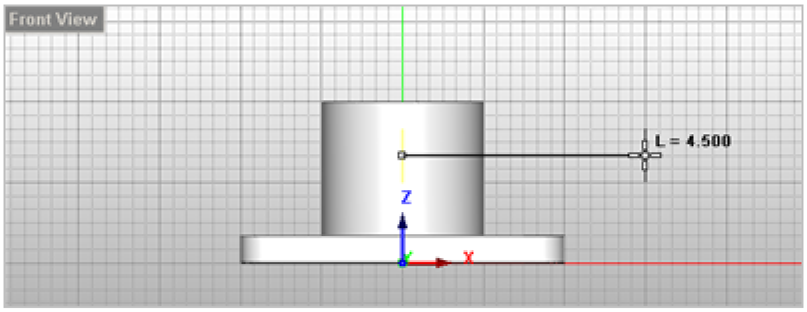
17. Now select the circle you just created, then go to the Mesh Modeling 

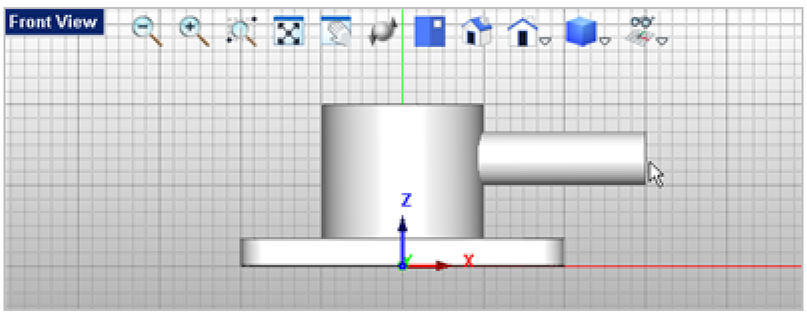
tab and select the Extrude 

command again.

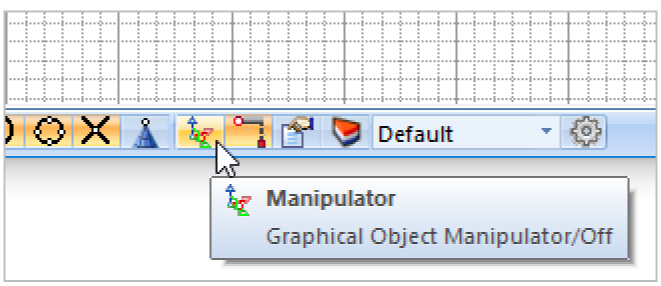
18. For the length of the cylinder move the cursor over to the Front View 

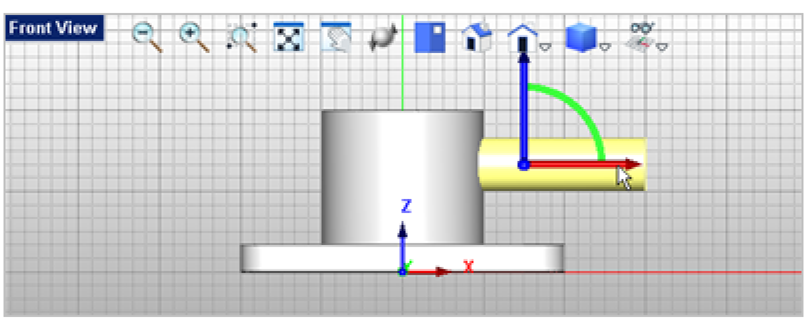
and select the grid point shown below.





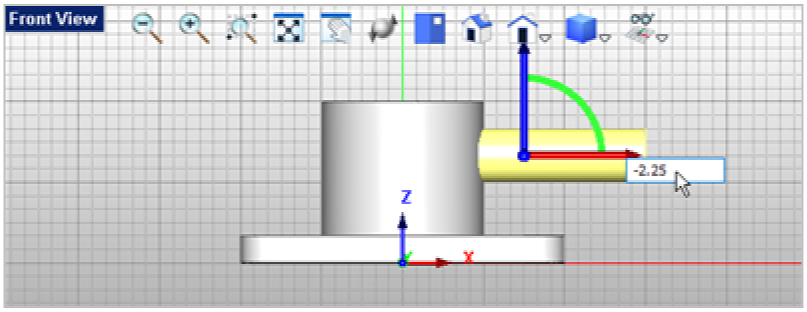
19. Now select the cylinder and toggle the display of the Graphic Manipulator  on. This toggle icon is located on the Status Bar at bottom of the display.



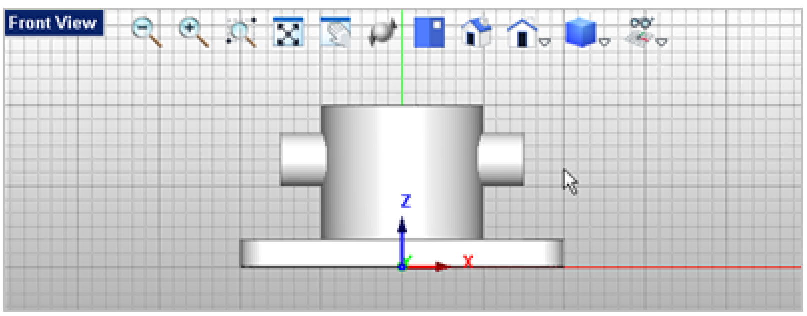


20. With the cylinder selected pick on the red X axis of the Graphic Manipulator to display the input box and enter -2.25 

and press <Enter> as shown below.

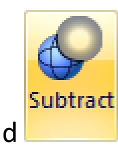


The cylinder is translated to the left negative X direction by 2.25 inches so that it passes complete through the tube on both sides.



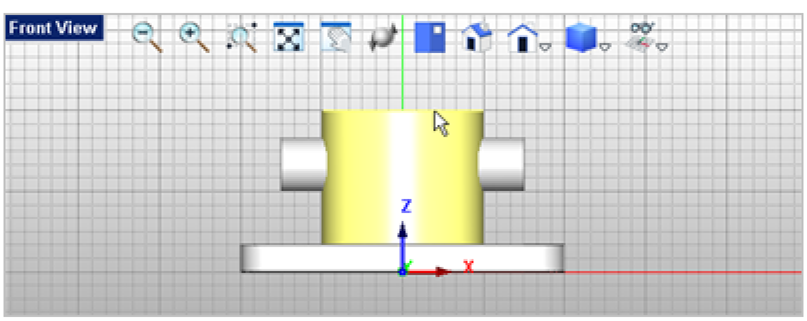
21. You can toggle the Graphic Manipulator  off.

22. From the Mesh Modeling 

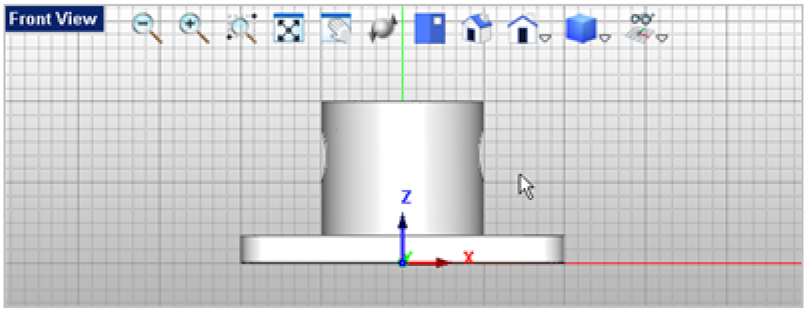
tab, select the Subtract command 

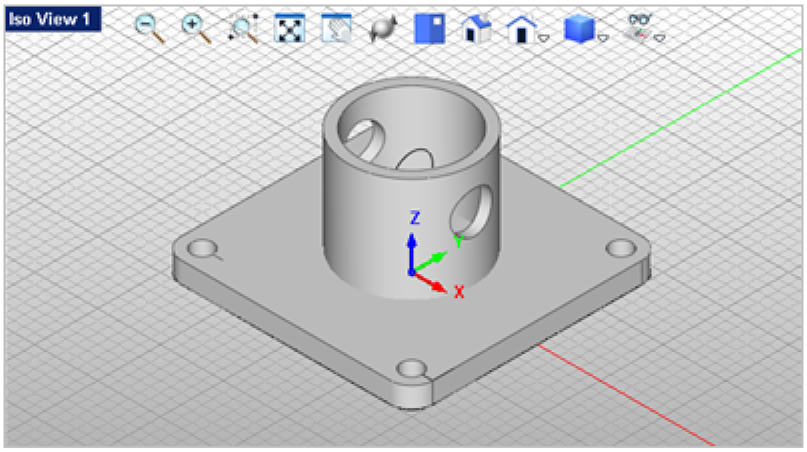
. It's located in the Edit Meshes pane. We are going to Subtract the cylinder from the tube.

23. For the first mesh select the tube.



24. For the second mesh select the cylinder. The hole is created in both sides of the tube.



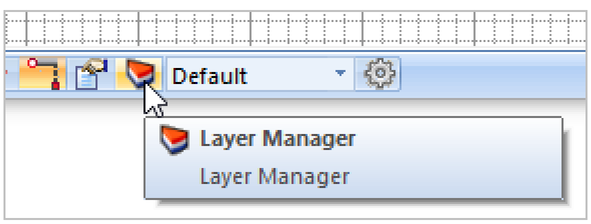


7.4.4 Create a New Layer

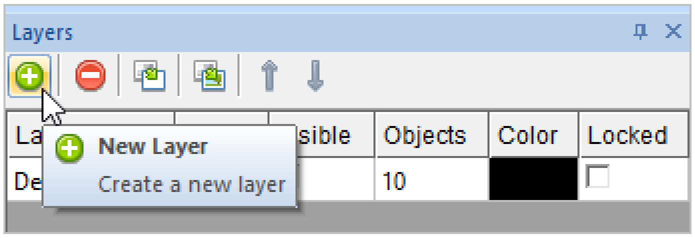
Create a New Layer

In this step we will create a new layer, turn it off and the move our curves to it to hide them from the display.

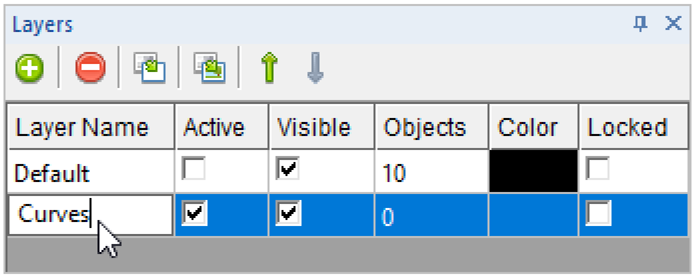
1. From the Status Toolbar, select the icon to display the Layer Manager .



2. At the top of the Layer Manager select the New Layer icon and a new layer will be added to the list.

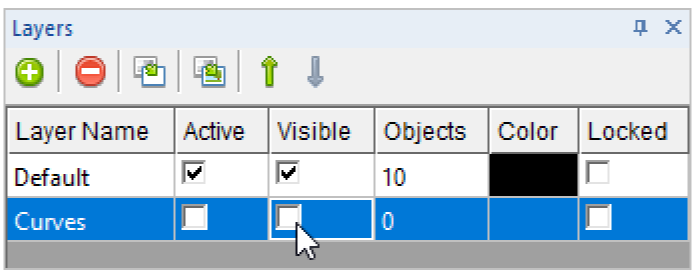


3. Now double-left click on the New Layer name and type in the names "Curves" and press <Enter>.

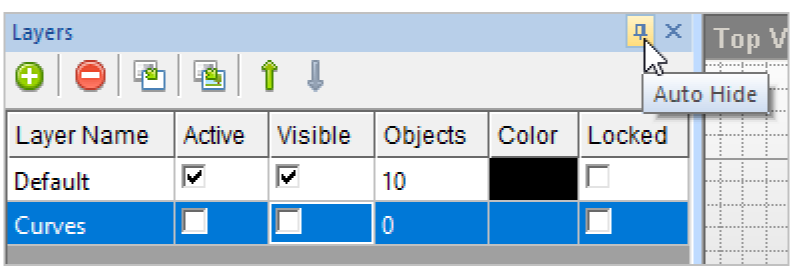


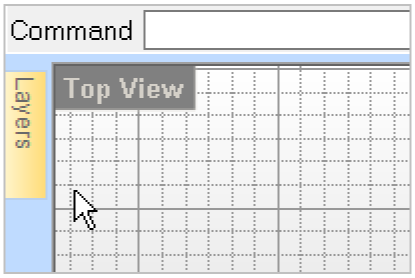
4. Now to hide the Curves layer first check the box to make the Default layer the Active layer.

5. Then uncheck the box to make the Curves layer hidden.



6. Now let's Auto Hide the Layer Manager. Select the Auto Hide pin icon located at the top right of the Layer Manager and will see that the dialog disappears to the left side of the screen.



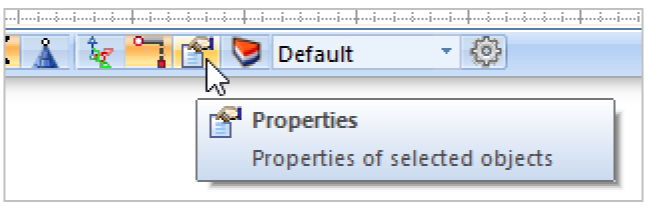


7.4.5 Edit Geometry Properties

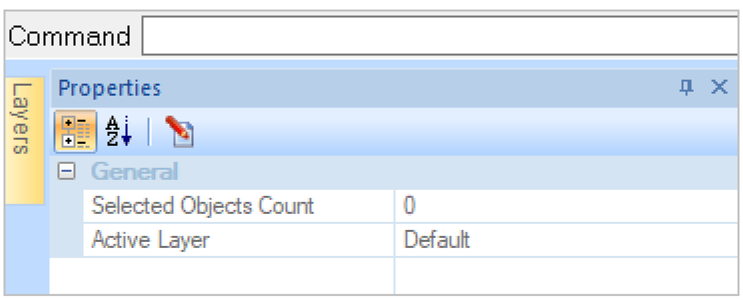
Edit Properties of Geometry

In this final step we will select all of our reference curves and edit their properties so that they are moved to a layer whose visibility is currently off.

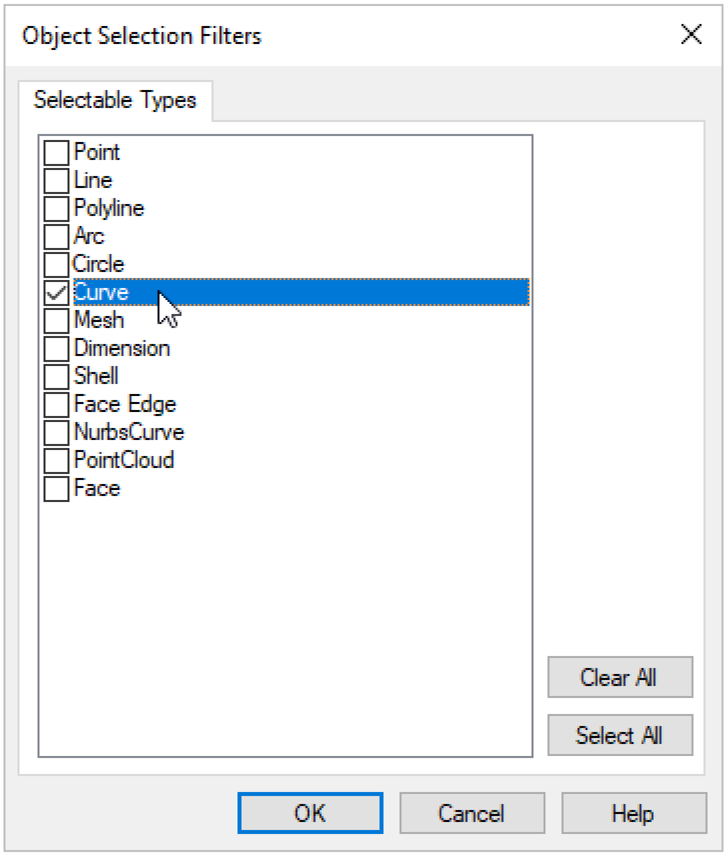
1. Let's start by selecting the Properties icon located on the Status Toolbar at the bottom of the display (just to the left of the Layer Manager icon).



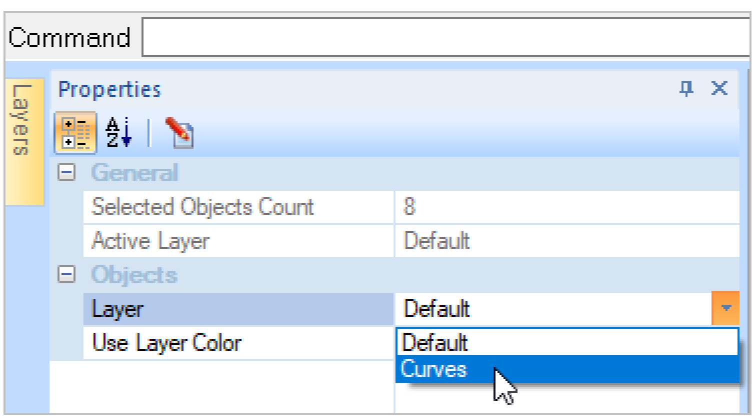
2. You will see the Properties Manager dialog display om the left side of the display.



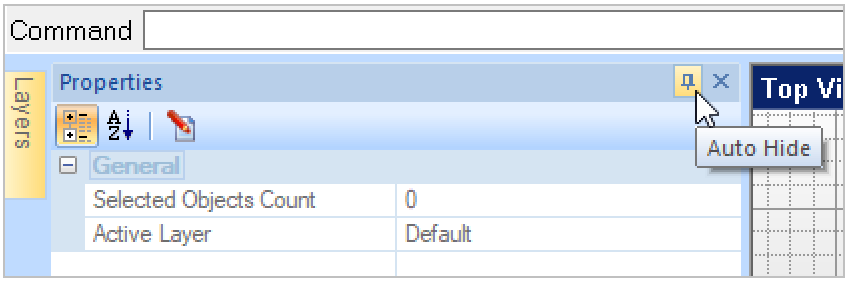
3. Now we will automatically select all of our curve geometry. Go to the Modeling Aids tab and select pick the Select by Type icon. This will display the Object Selection Filters dialog.

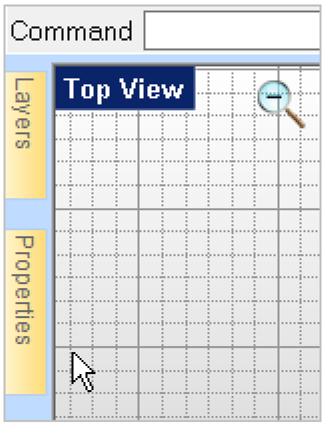


4. From the dialog check the box for Curves and uncheck all other boxes and then pick OK to close the dialog. You will see that all of the curves are highlighted for selection.
   Now go to the Properties Manager and under the Objects section drop down the menu for Layer and select Curves.

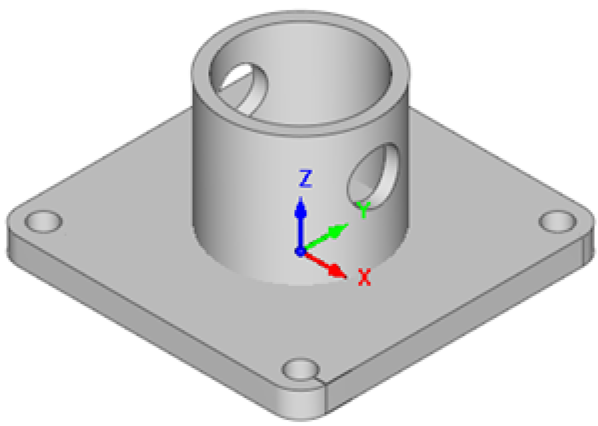


5. Now close the Properties Manager by selecting the Auto Hide pin located at the top right of the Properties Manager.



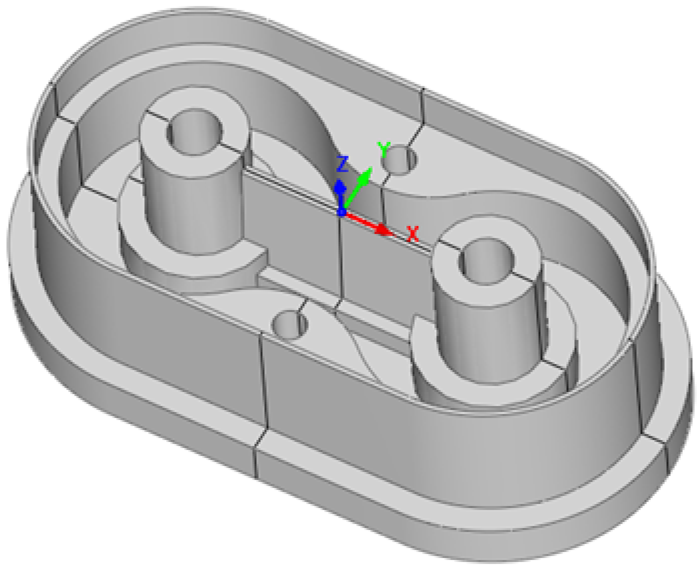


6. This exercise is complete and your part now looks like this:
7. Save the file as Base_Plate_Completed.vcp.



7.5 #5: Model a Mold Insert

In this exercise you will model something a bit more complicated, the mold insert shown below. It builds upon the tasks you have learned in the previous exercises. You will draw curve profiles and then turn them into part features. The Graphic Manipulator is used extensively in this exercise as well as modeling from multiple viewports. Just go slowly and save your work often and Undo if needed.



Model a Mold Insert

What you will learn:

In this exercise you will perform the following VisualCAD tasks:

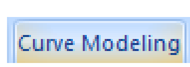
1. Extrude the Body.
2. Extrude the Flange.
3. Extrude the Upper Pocket.
4. Extrude the Lower Pocket.
5. Revolve the Center Bosses.
6. Extrude the Connection Bar.
7. Extrude the Connection Wall.
8. Extrude Ejector Pin Holes.
9. Change Geometry Layer.
10. Create Section Curves.

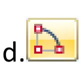
7.5.1 Extrude the Body

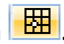
Extrude the Body

In this step we will model the body by creating an extrusion.

1. Activate the Top View 

tab by selecting anywhere in the viewport.
2. Select the Curve Modeling tab 

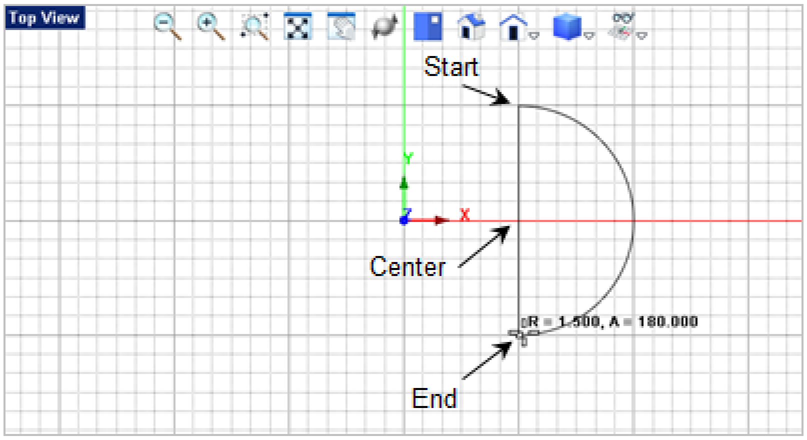
from the top Ribbon Bar.
3. Now select the Arc Center, Start, Angle command. 

4. In the [Status Bar] make sure the [Grid Snap] is On .

5. For the center point select the grid point shown below. The status bar will track where your cursor is located in World Coordinates. Move the cursor until the grid point at 1.5,0,0 is located and pick it.

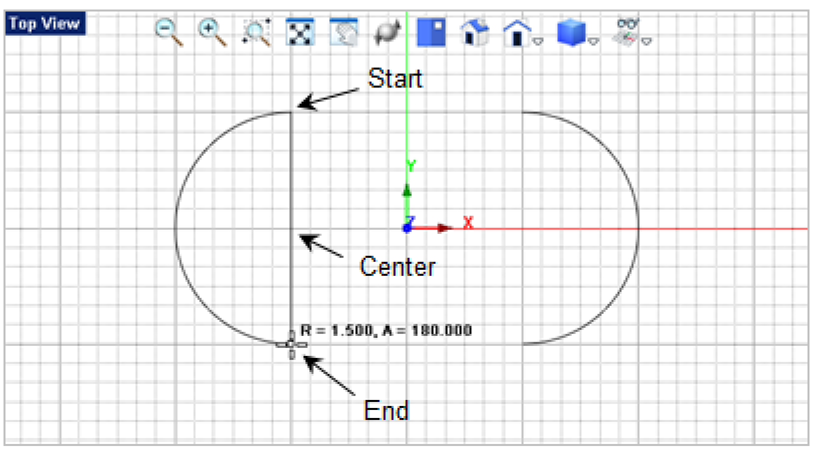
6. Following the command prompts 

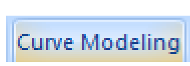
. It says to select the start point of the arc. Locate and select the start point shown below.



7. For the arc end point move the cursor clockwise and pick the end point shown. You will see that the arc is previewed as you move the cursor.

8. Now we want to draw another arc, press <Enter> to repeat the command and select the center, start and end points shown below.



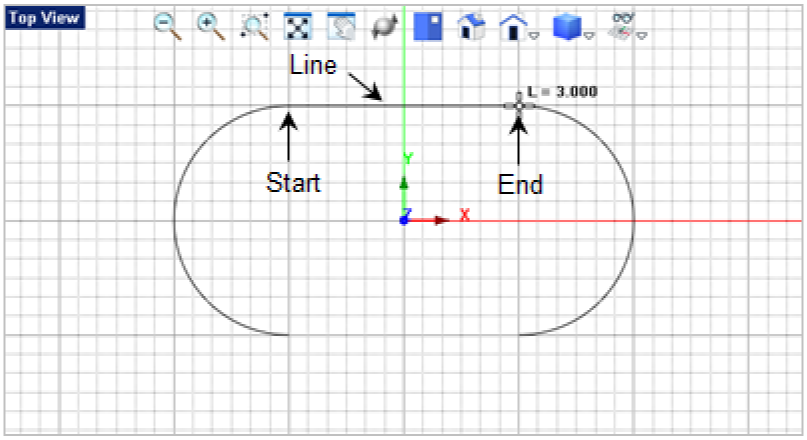
9. Now we want to draw a line to connect the two arcs across the top. From the Curve Modeling tab 

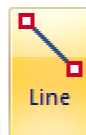
select the Line command 

.

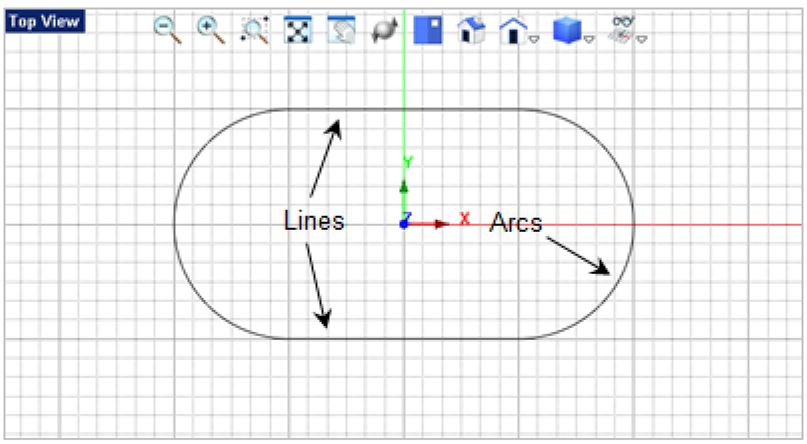
10. For the start point for the line select the grid point located at the top end of the arc of the left and then for the end point of the line select the top end of the arc on the

right. Optionally, you can set the Object Snap to End Point  from the Status Bar and select the end points of the arcs to draw the line.

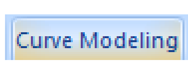


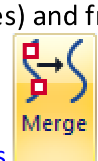
11. Press <Enter> to repeat the Line command 

and draw the second line connection the two arcs across the bottom. Your drawing should look like the image below.



12. For modeling its best that you merge your profile geometry into a single curve entity.

13. Window select the geometry (both arcs and both lines) and from the Curve Modeling tab 

select the Merge Curves 

command and then right-click the mouse to merge them. If you pick anywhere on the geometry you will see that one curve is now defined.

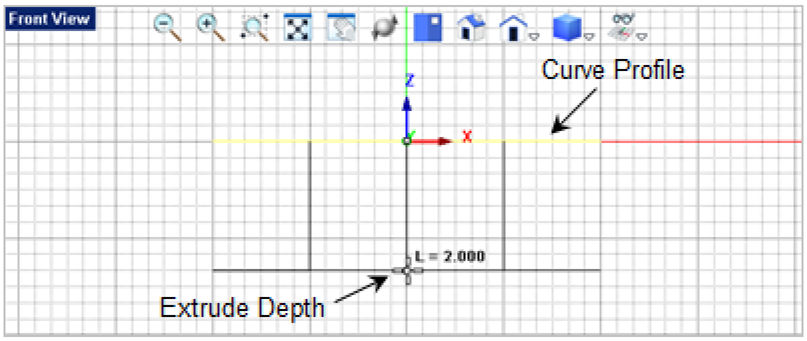
14. Now lets create the body by extruding our profile curve. With the curve selected, change to the [Mesh Modeling](#) tab 

and select the [Extrude Mesh](#) 

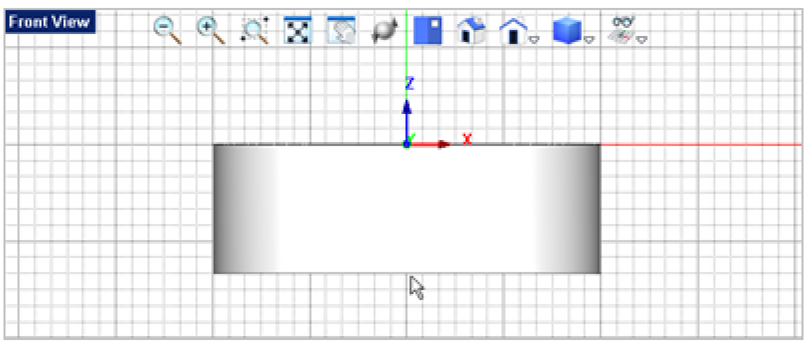
command.

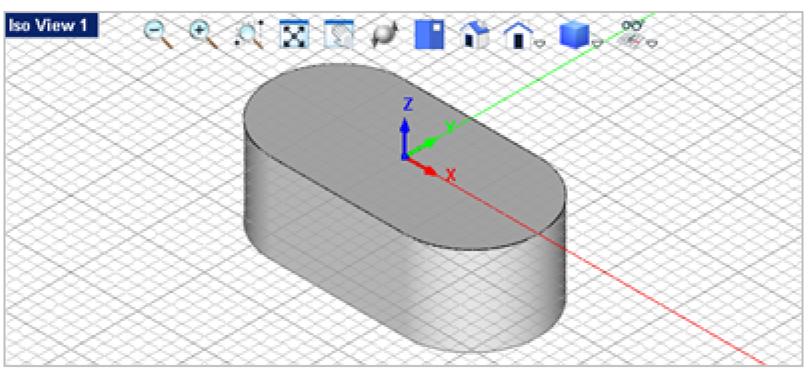
15. For the extrusion distance, drag the cursor down to the Front View 

and select the grid point shown below. Its located at 2.0" below the selected curve.



The Mesh Solid is created and shown below.





7.5.2 Extrude the Flange

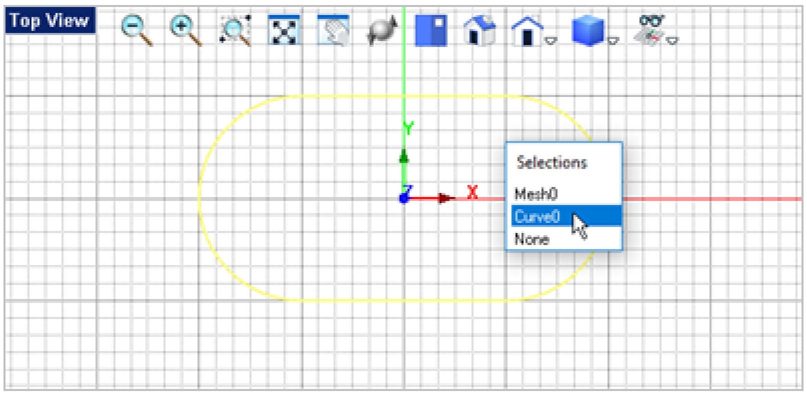
Extrude the Flange

Now we will extrude the flange.

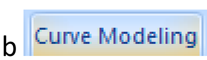
1. Activate the Top View 

by selection the profile curve you just created. You will notice that a selection list pops up because there is now more than one entity in the same located.

   Note: If the Selections menu shown below does not display, go to the Home tab in VisualCAD and select Options. Then under System clear the check box next to Use Preselection Highlight.



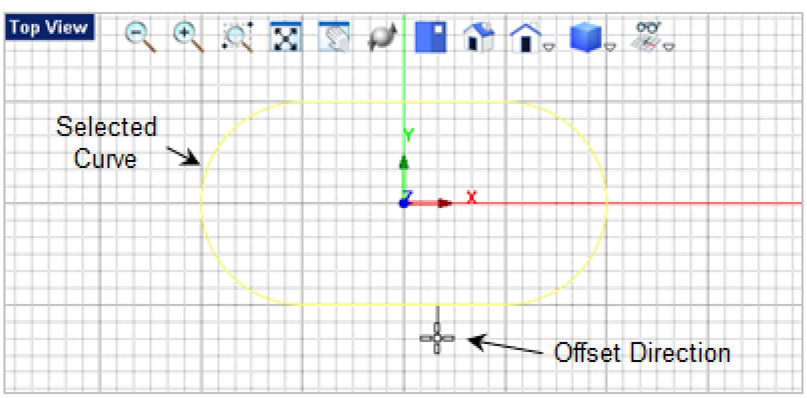
2. From the selection list pick Curve. The number value to the right of the curve may change depending on when the entity is created.

3. With the curve selected, from the Curve Modeling tab 

select the Offset Curve 

command.

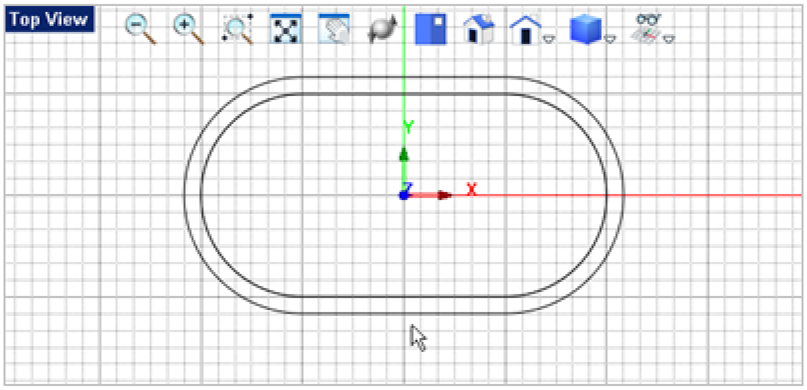
4. If you move the cursor you see that an offset direction is indicated. Move the cursor below and on the outside of the curve as shown below.

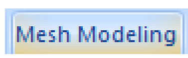


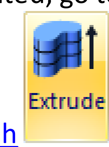
5. Most of the time in this exercise we are selecting grid points. The distance between each grid point is 0.25". For the offset distance you can select the next grid point below the curve.

Optionally, while the offset direction is displayed you can enter 0.25 

and press <Enter> to create the offset curve shown below.



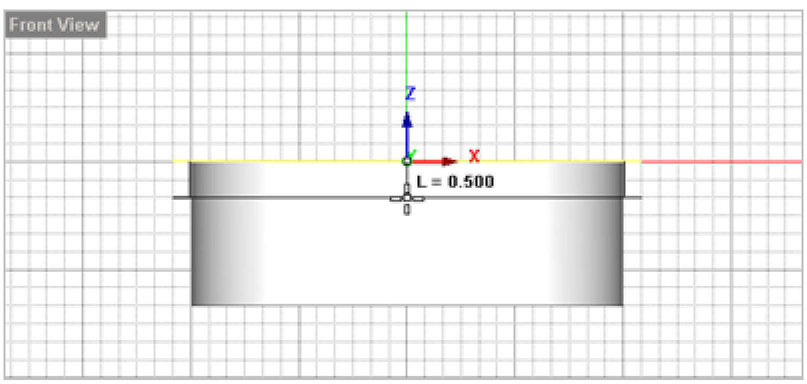
6. Now select the offset curve you just created, go to the Mesh Modeling tab 

and pick the Extrude Mesh 

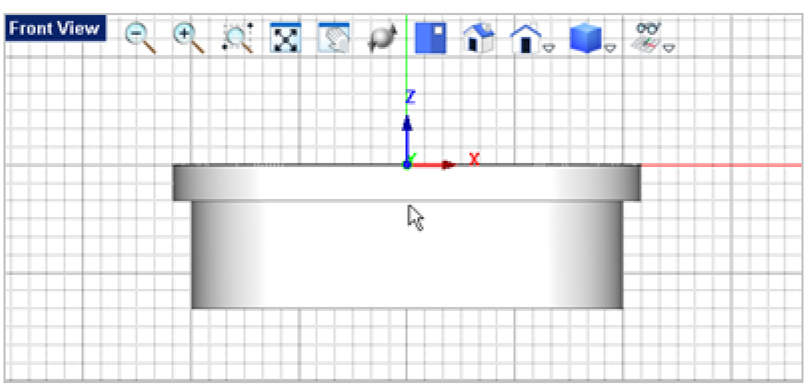
command again.

7. Drag the cursor to the Front View 

and pick the second grid point below the selected curve. The cursor indicates that the extrusion length is 0.500".



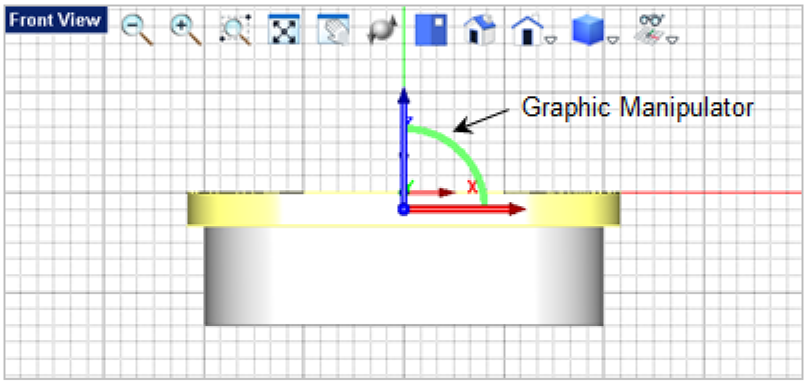
The flange is created and displayed.



8. Now lets move the flange to the bottom of the body. To do this we will use the Graphic Manipulator. To toggle the Graphic Manipulator on, go to the Status Bar and select the icon .

   The Graphic Manipulator allows you to easily translate or rotate a selected entity in or about the X, Y or Z of the World Axis.

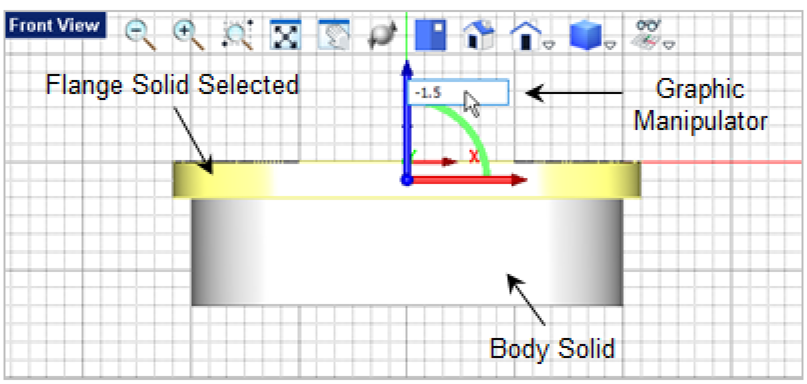
9. Now select the flange and you will see that the Graphic Manipulator is displayed on top of the selected entity.



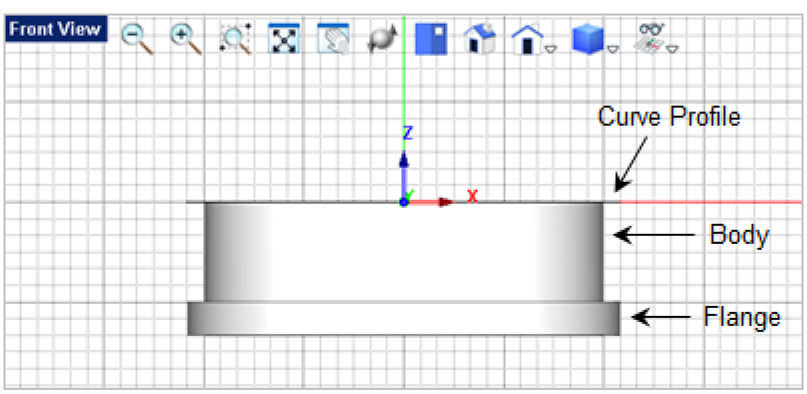
10. Pick on the blue Z Axis arrow to display the input window.

11. Enter -1.5 

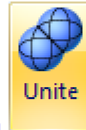
and press <Enter>.



The flange is moved 1.5" in the negative Z direction.

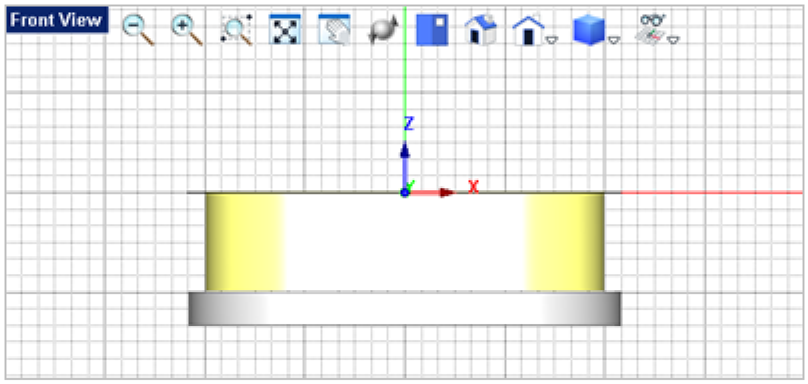


12. Now lets unite the flange with the body. From the Mesh Modeling tab, 

select the Unite Mesh 

command.

13. For the first mesh solid select the body.



14. For the second mesh solid select the flange. The two will be joined into one mesh solid.

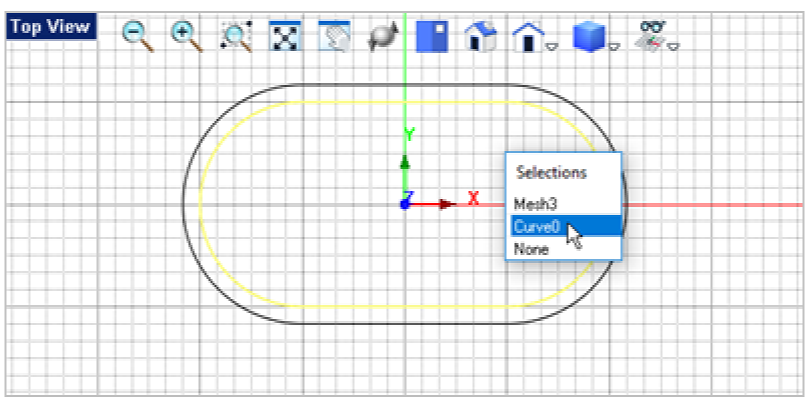
7.5.3 Extrude the Upper Pocket

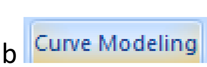
Extrude the Upper Pocket

In this step we will create the upper pocket and subtract it from the body.

1. From the Top View 

select the profile curve of the body by clicking on it and selecting Curve from the selection list.



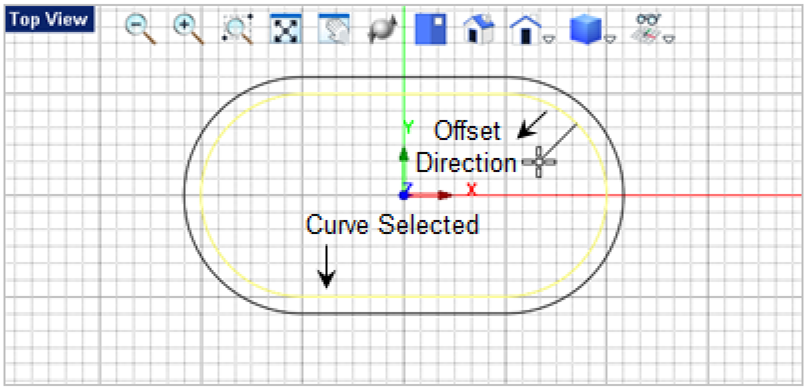
2. With the curve select, go to the Curve Modeling tab 

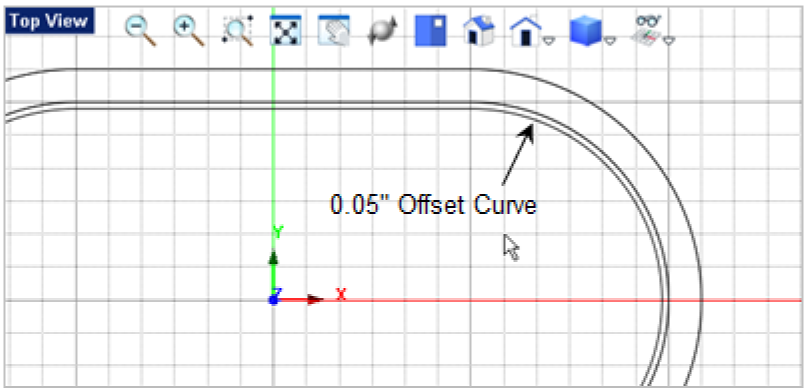
and select the Offset Curve 

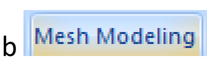
command.

3. Drag the offset direction to the inner side of the curve and then enter 0.05 

and press <Enter>.





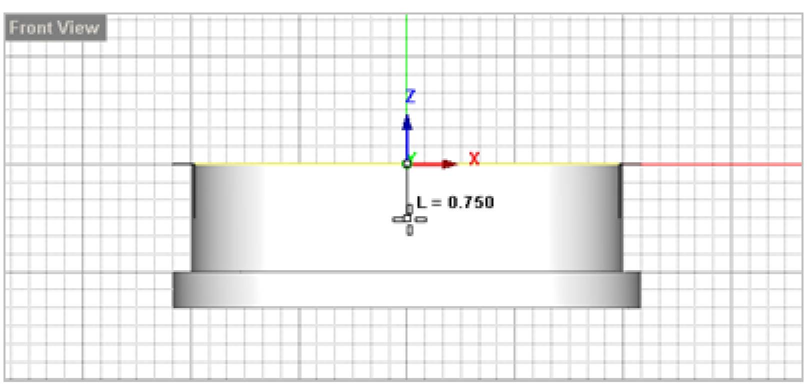
4. Now select the offset curve, go to the Mesh Modeling tab 

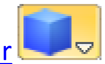
and select the Extrude Mesh 

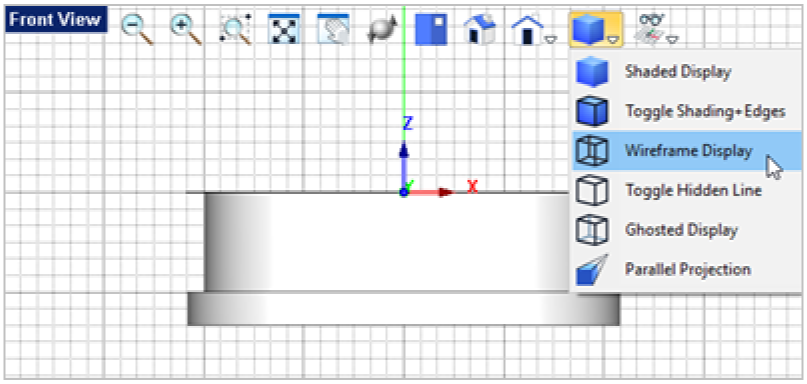
command.

5. Drag the cursor down to the Front View 

and select the grid point at 0.750" as shown below.



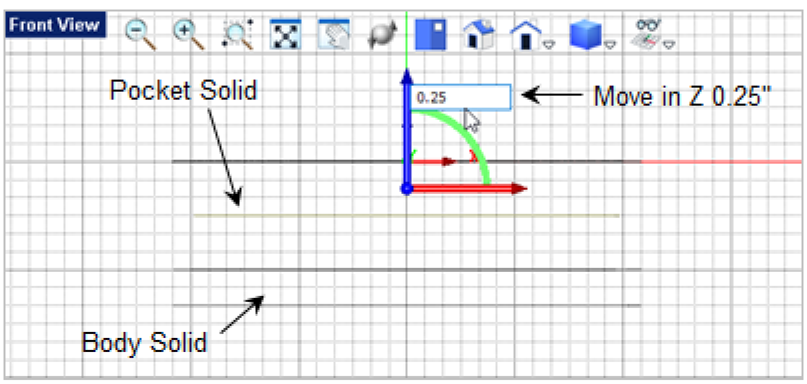
6. Now to see the pocket mesh solid, from the View Toolbar  select the Wireframe Display item from the display menu.



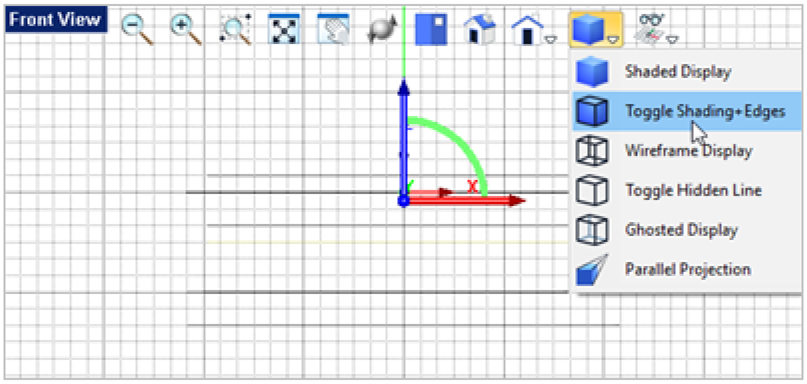
7. We want to move the pocket mesh solid so make sure the Graphic Manipulator icon is still toggled on  (it's located on the Status Bar remember?).

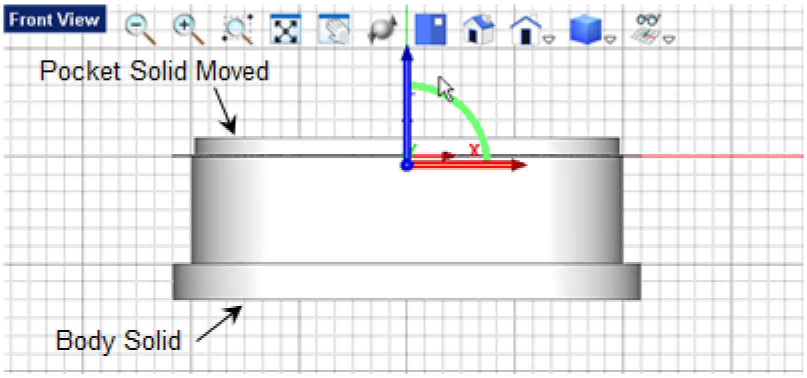
8. With the Front View  set to Wireframe display you can see to select the pocket mesh solid. Select it now and the Graphic Manipulator will display on it.

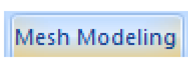
9. Select the blue Z Axis arrow and enter 0.25  and press <Enter>. The pocket mesh solid will move upward 0.25".

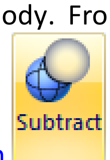


10. From the View Toolbar  drop down the display menu and select Toggle Shading + Edges.





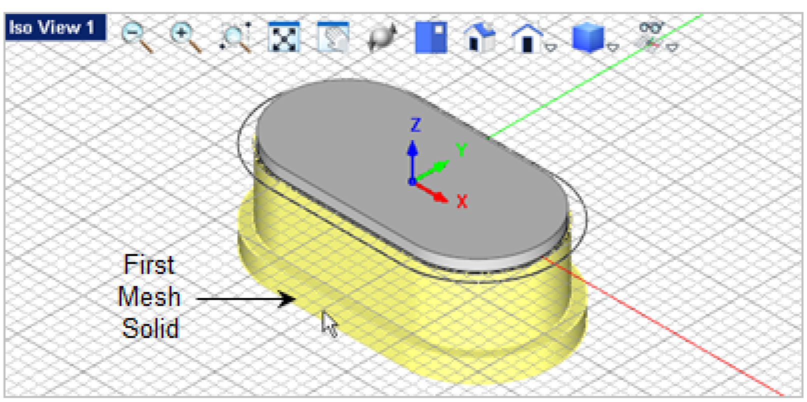
11. Now lets subtract the pocket from the body. From the [Mesh Modeling](#) tab 

select the [Subtract Mesh](#) 

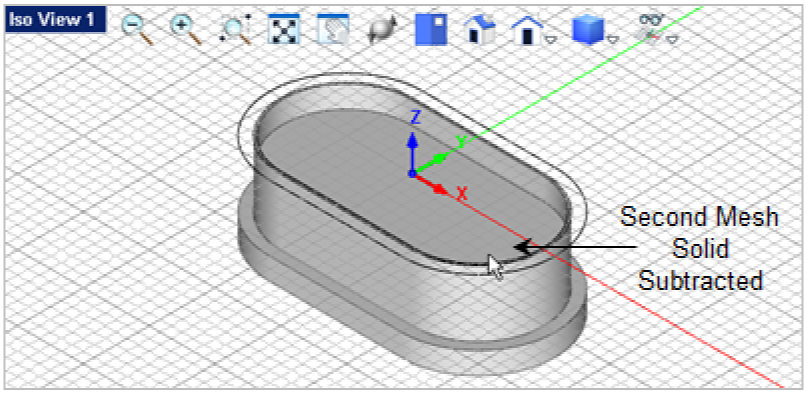
command.

12. For the first mesh solid select the body. This is the solid that will be subtracted from. You can do this from any Viewport. We will select it from Iso View 1 

.



13. Now for the second mesh solid, select the pocket solid and it will be subtracted from the body.

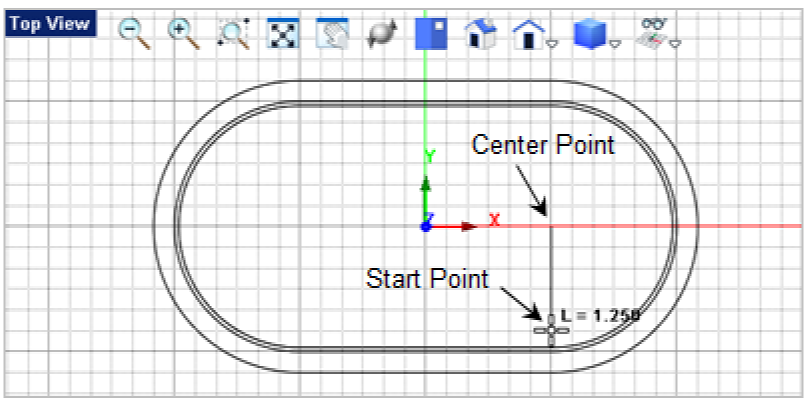


7.5.4 Extrude the Lower Pocket

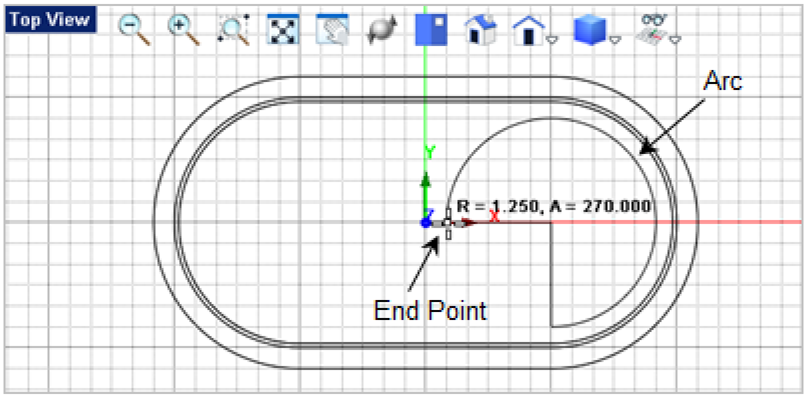
Extrude the Lower Pocket

In this step we will create the lower pocket mesh solid and subtract it from the body.

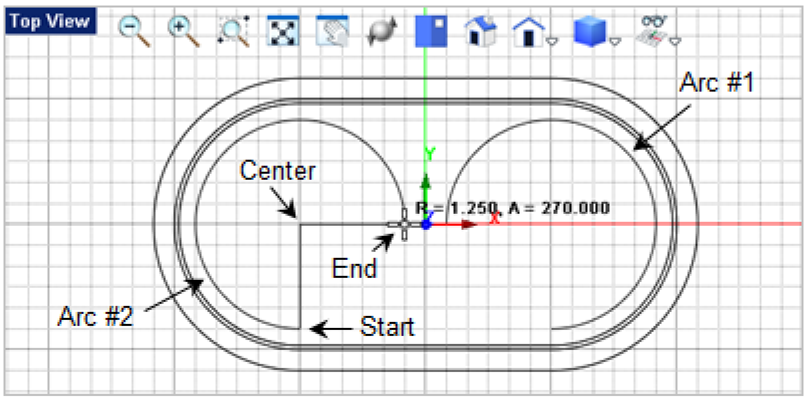
1. We will start in the Top View  so select anywhere in that view to activate it.
2. Now we will create two partial arcs, each 270 degrees. From the Curve Modeling tab , select select the Arc Center, Start, Angle command  again.
3. Select the center point first and then select the start point. Both are shown below.



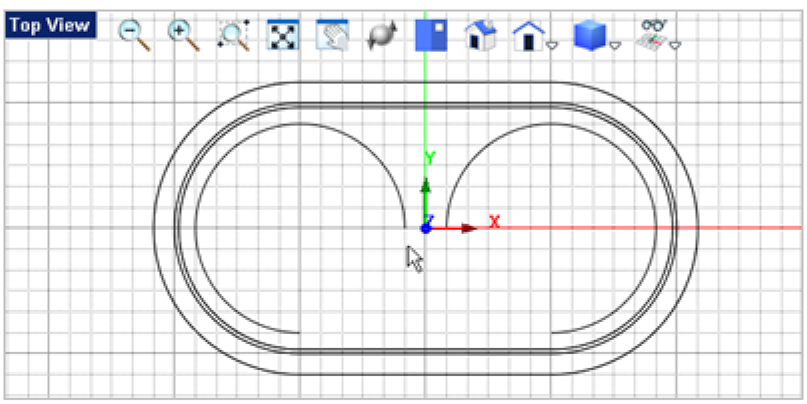
4. Now for the end point of the arc, drag the cursor counter clockwise until the arc preview is at 270 degrees as shown below.



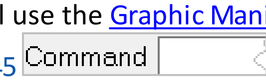
5. Now press the <Enter> key to repeat the command. This time select the center, start and end points shown below.

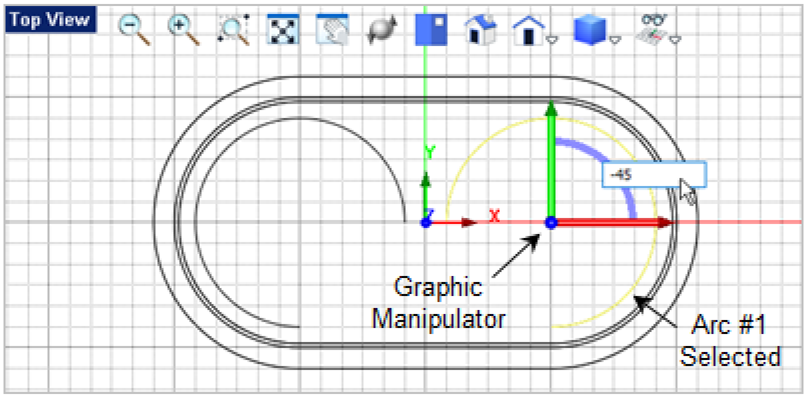


You drawing should now look like this:

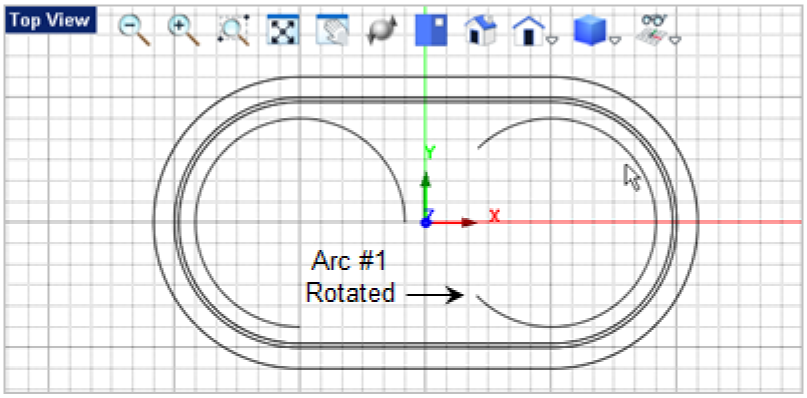


6. Now we will rotate each arc 45 degrees to position them properly. Toggle the Graphic Manipulator on , select the first arc.

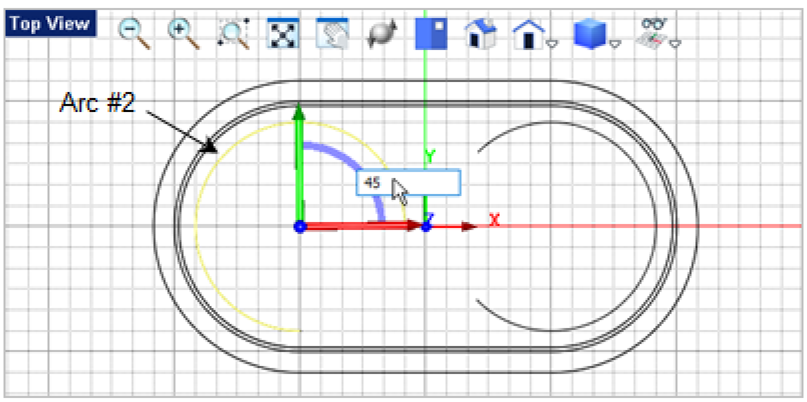
7. This time we will use the Graphic Manipulator's rotate feature. Select the blue Z Axis arc, enter -45  and then press <Enter>.



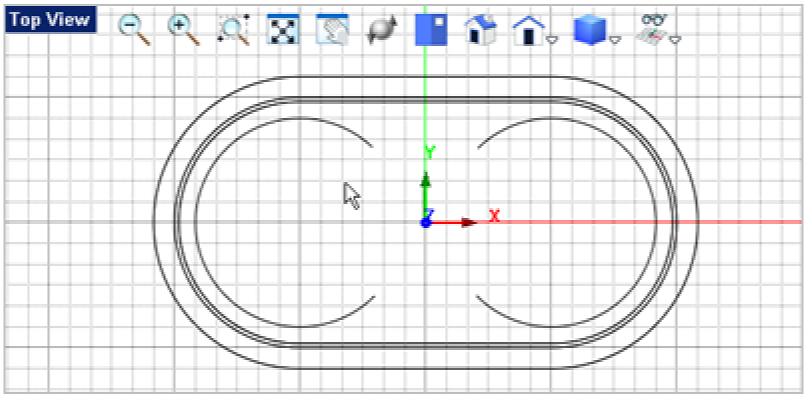
The arc is rotated negative 45 degrees about the Z axis as shown.



8. Now select the second arc, pick the Z axis arc, enter 45 and press <Enter>.



The second arc is rotate 45 degrees about the Z axis as shown.



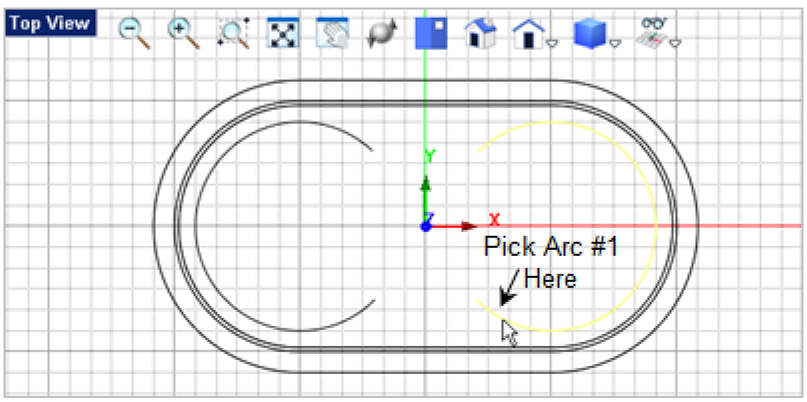
9. To complete the second pocket profile we will fillet the two arcs. From the Curve Modeling tab 

select the Fillet 

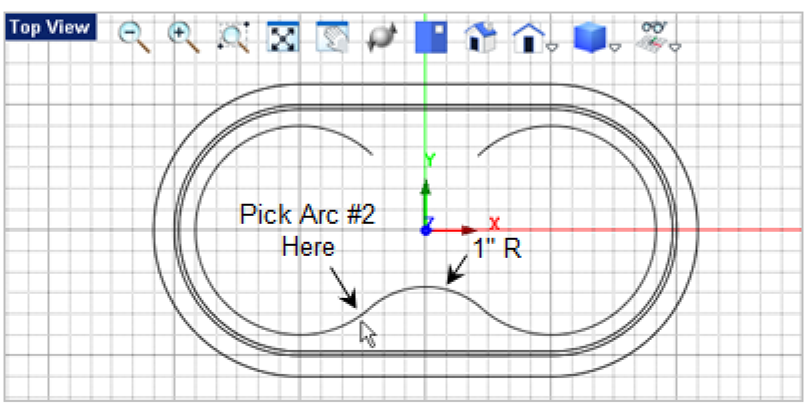
command.

10. Enter 1.0 for the fillet radius and press <Enter>.

11. Now pick the first arc near the end shown.



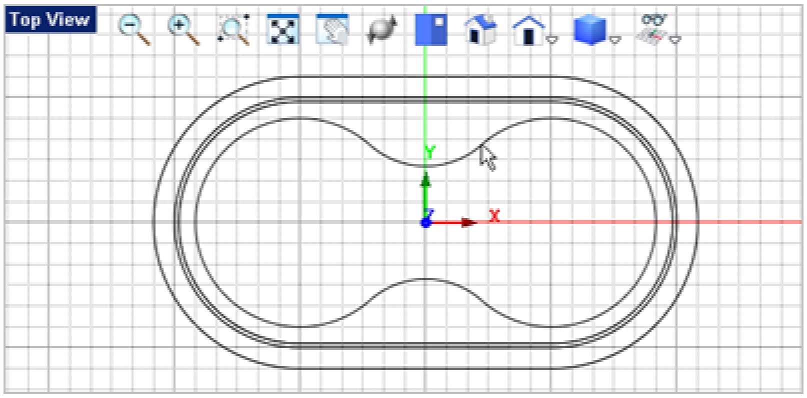
And then pick the second arc near the end shown to create the 1" fillet between the two arcs.



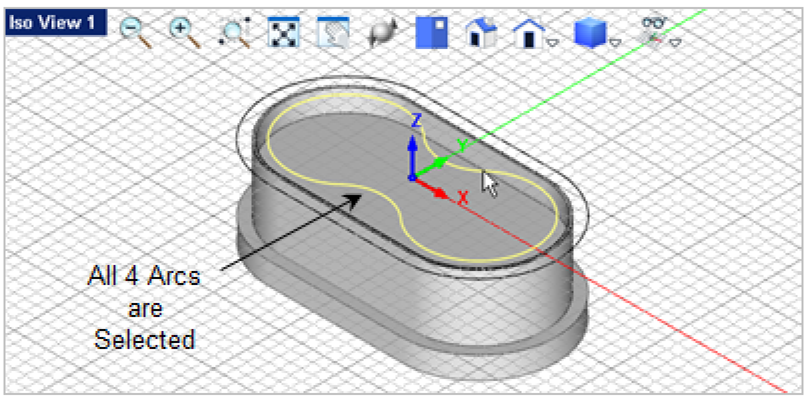
12. Press <Enter> to repeat the Fillet 

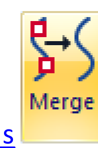
command.

13. Select the two arcs near their ends to create the second fillet as shown.



14. Now select the four arcs. You can press the <Ctrl> key to select multiple entities.



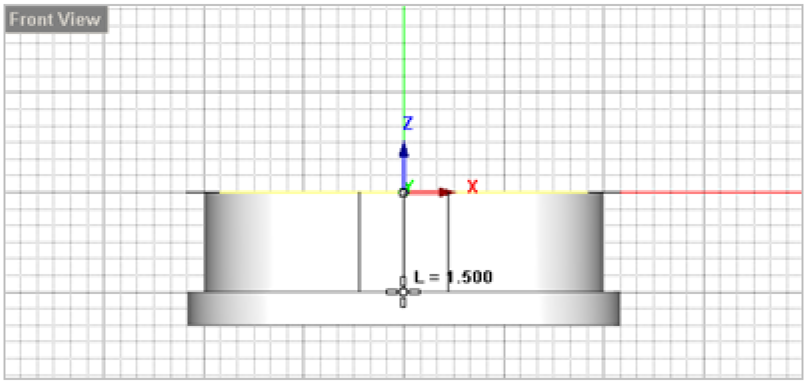
15. Now from the Curve Modeling tab Curve Modeling select the Merge Curves 

command and then right-click or press <Enter> to merge them into one curve.

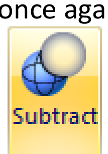
16. With the curve still selected go the Mesh Modeling tab Mesh Modeling and select the Extrude Mesh 

command one again.

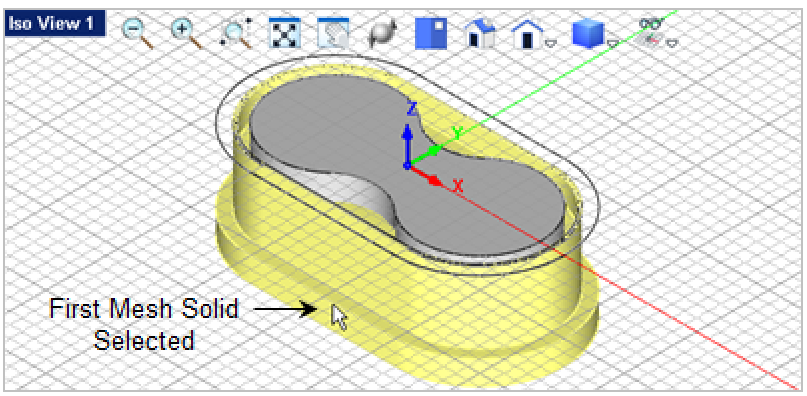
17. Drag the cursor down to the Front View **Front View** and select the grid point that is 1.50" below the selected curve to create the second pocket mesh solid.



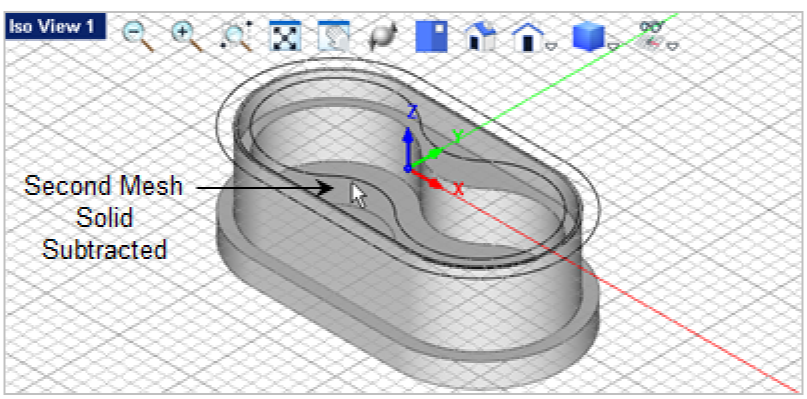
18. Now once again from the Mesh Modeling tab 

select the Subtract Mesh 

command and for the first solid, pick the body.



19. Now for the second solid pick the second pocket mesh solid and it will be subtracted from the body as shown.

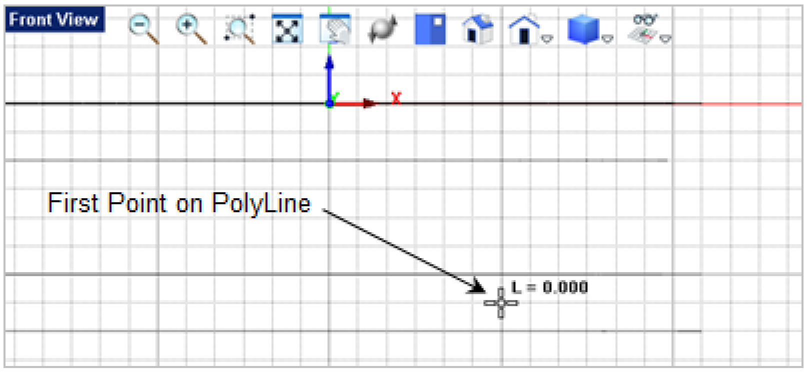


7.5.5 Revolve the Center Bosses

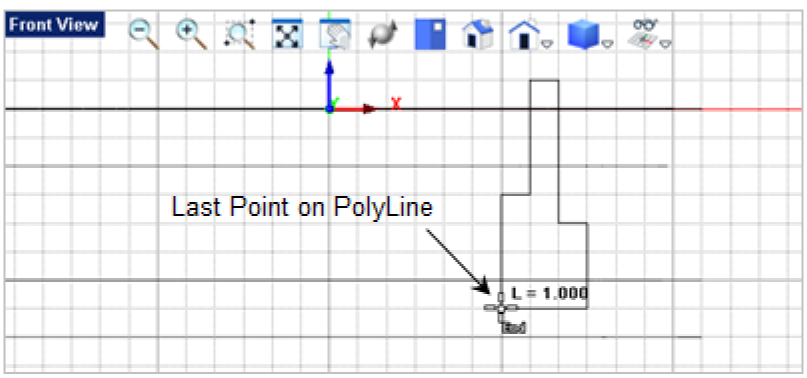
Revolve the Center Bosses

In this step we will create the two center bosses by revolving a profile to create a mesh solid, mirror it to create the second boss and then add both bosses to the body.

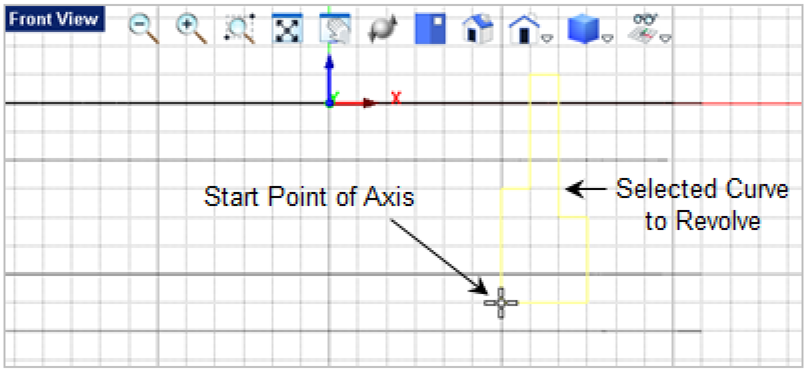
1. From the Front View, select Wireframe Display from the Display menu on the View Toolbar.
2. From the Curve Modeling tab, select the Polyline command.
3. Pick the grid point shown below for the first point on in the polyline.



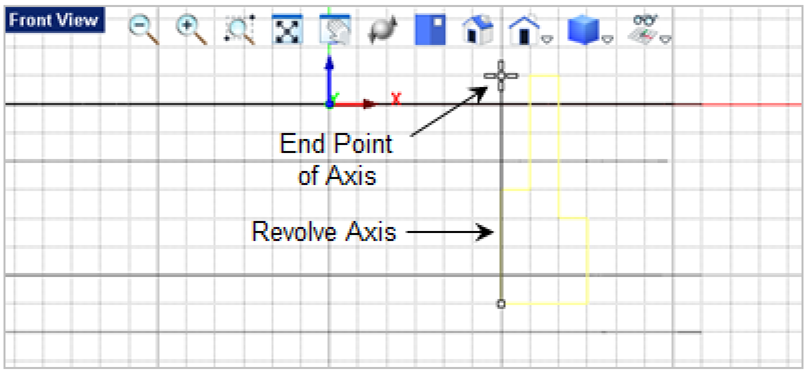
4. Then pick the remaining 9 grid points to complete the closed polyline. The last grid point is the same as the first grid point. This forms a closed polyline.



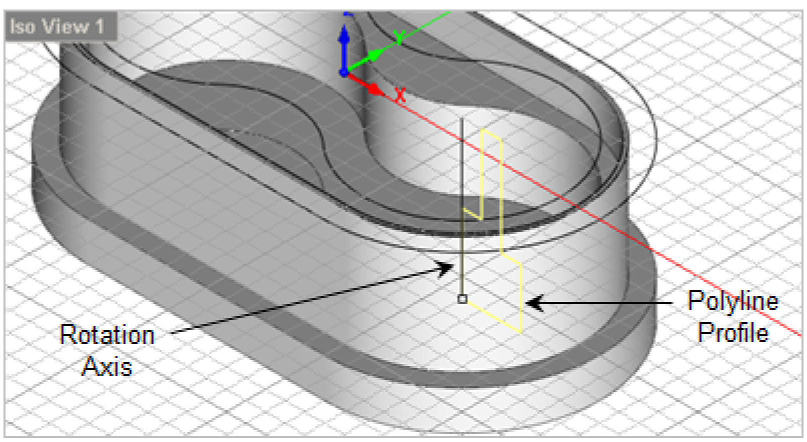
5. Now select the polyline, and then go to the Mesh Modeling tab and select the Revolve Mesh command. You can toggle off the Graphic Manipulator as we do not need it here.
6. The command prompt says to pick the start point of the axis of rotation.

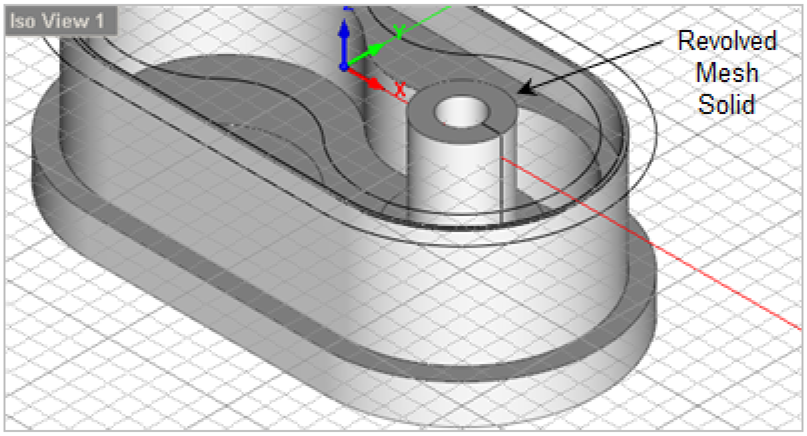


7. Now pick the second point on the axis of rotation as shown.



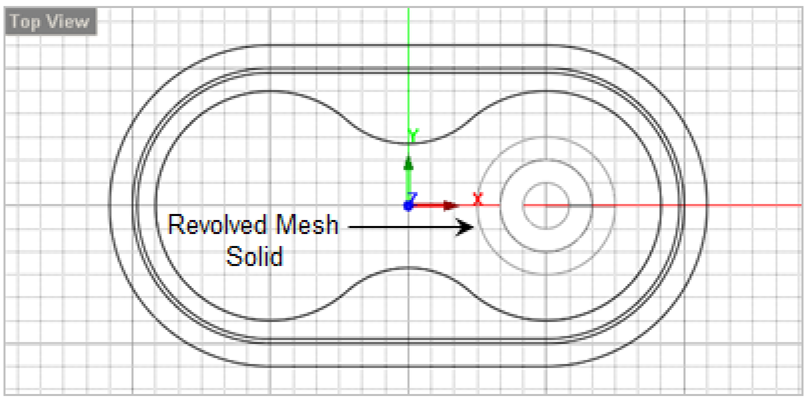
8. The revolved mesh solid is created and can be best viewed from Iso View 1 .





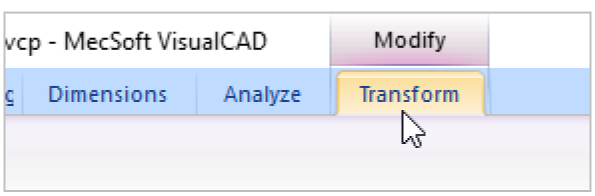
9. Now go to the Front View 

and select the boss mesh solid we just created.



10. With the boss mesh solid selected you will see the Modify / Transform tab 

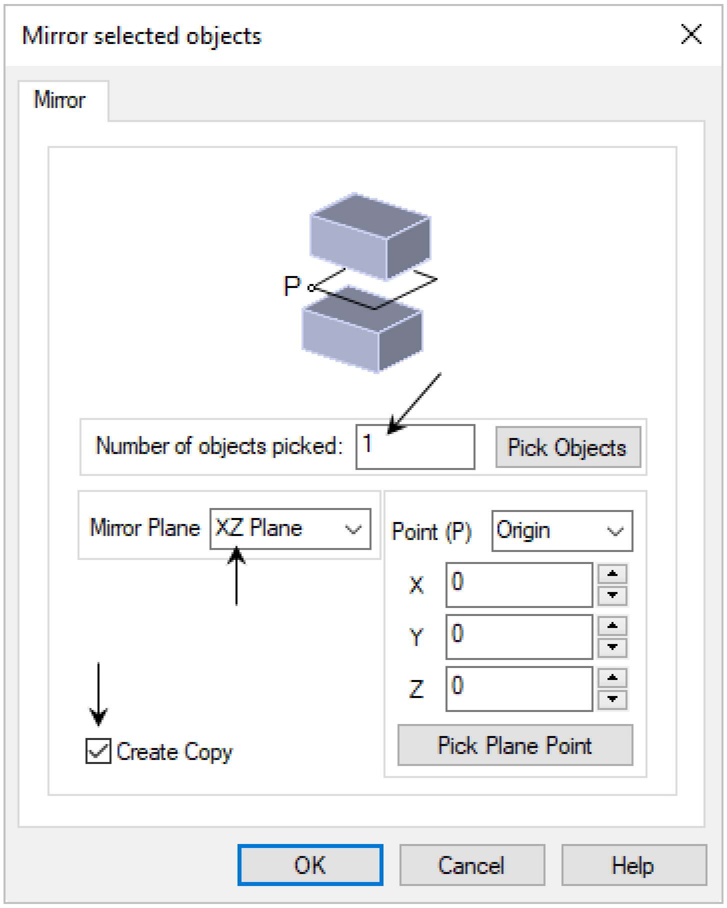
appear on the right end of the Ribbon Bar. It ONLY appears when an entity is selected.



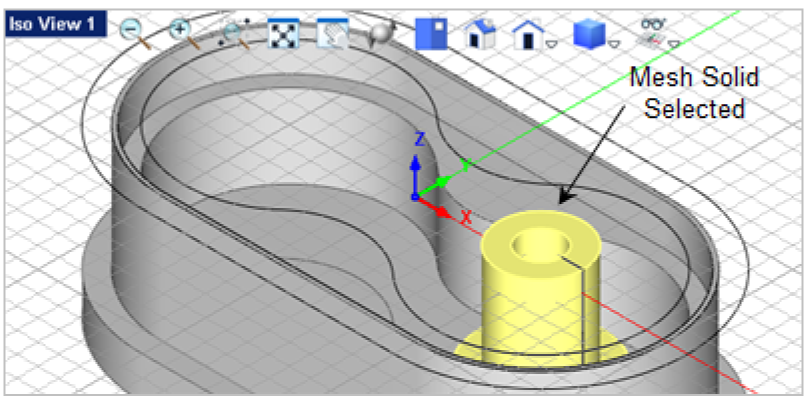
11. From the Transform tab, select the Mirror 

command to display the Mirror selected objects dialog.

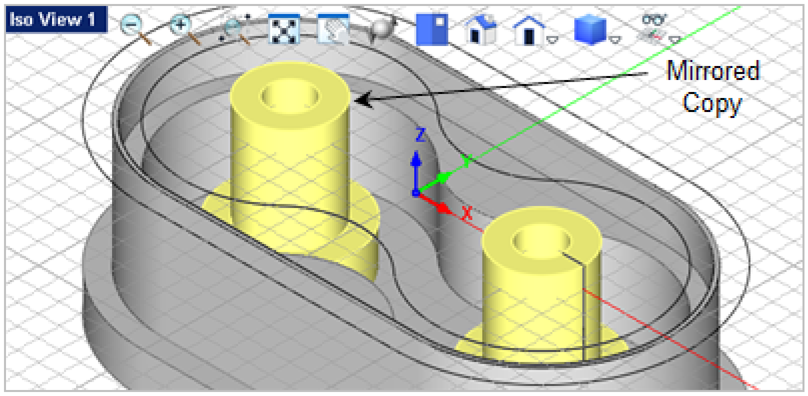
12. In this dialog, set the Mirror Plane to YZ Plane, check the box next to Create Copy, verify that the Number of objects picked value is 1 and then pick OK.



The mesh solid you selected:



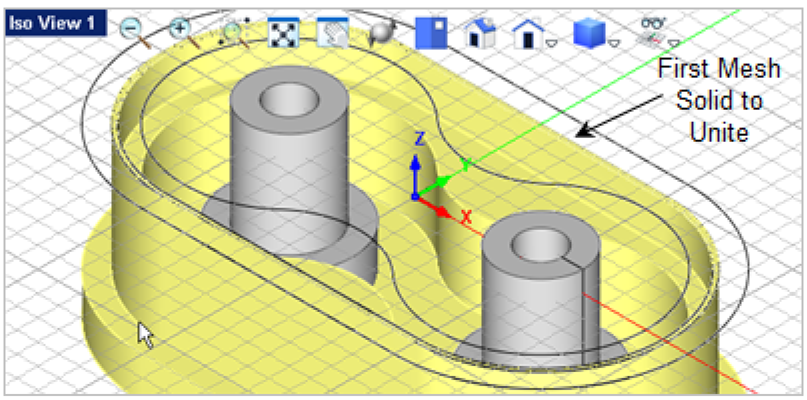
Has now been mirrored about the YZ plane creating a second copy as shown.



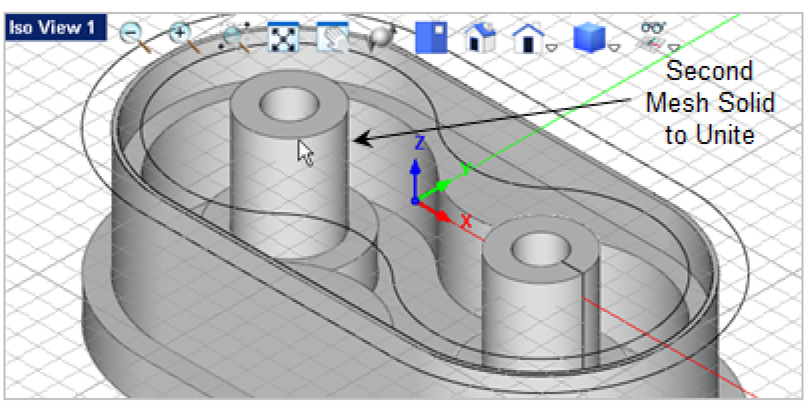
13. Now lets unite the two bosses with the body. From the [Mesh Modeling](#) tab, 

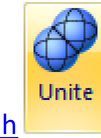
select the [Unite Mesh](#) 

command and then select the body.

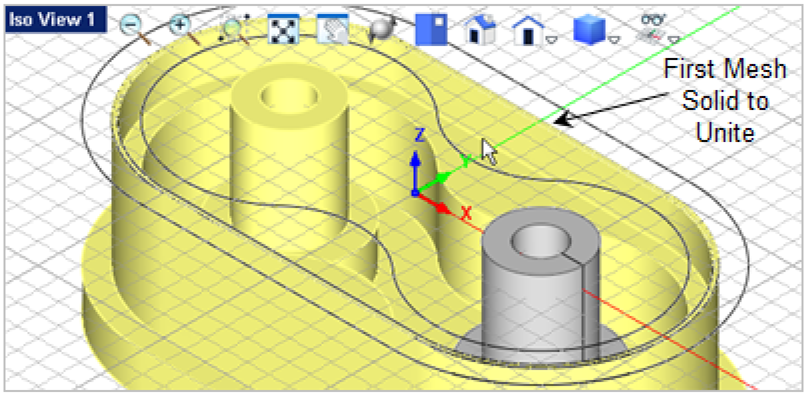


14. Now select the second solid to unite (i.,e. the one of the two bosses we just created.)

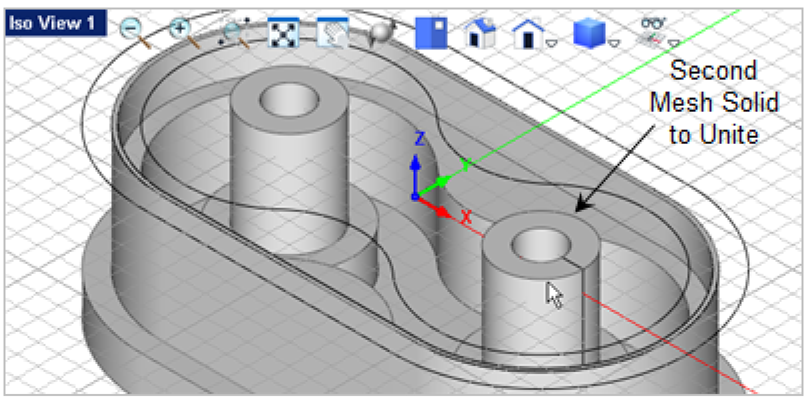


15. Now press <Enter> to repeat the Unite Mesh 

command and select the body solid one again.



And then select the other boss solid to unite them.



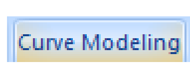
7.5.6 Extrude the Connection Bar

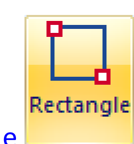
Extrude the Connection Bar

In this step we will create the square bar that connects the two bosses at the base of the second pocket.

1. First activate the Right View 

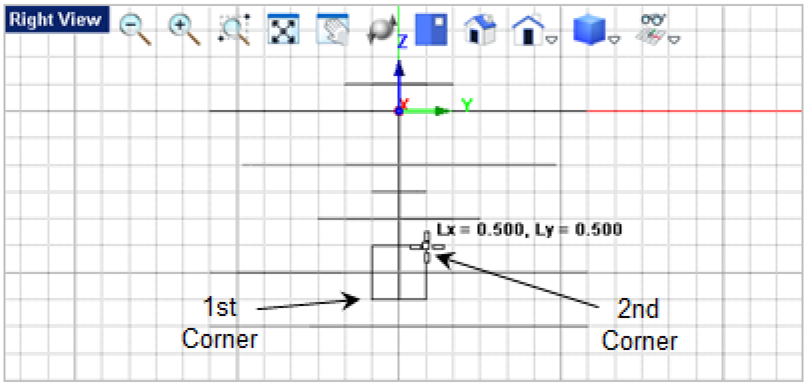
and then from the Display menu  on the View Toolbar, select Wireframe Display.

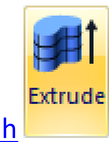
2. From the Curve Modeling tab, 

select the Rectangle 

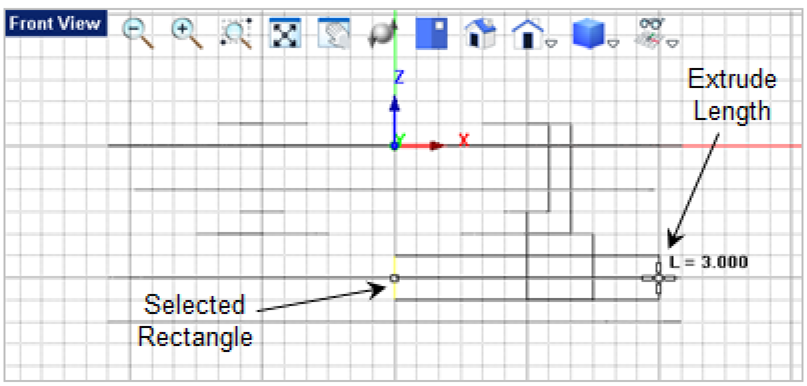
command.

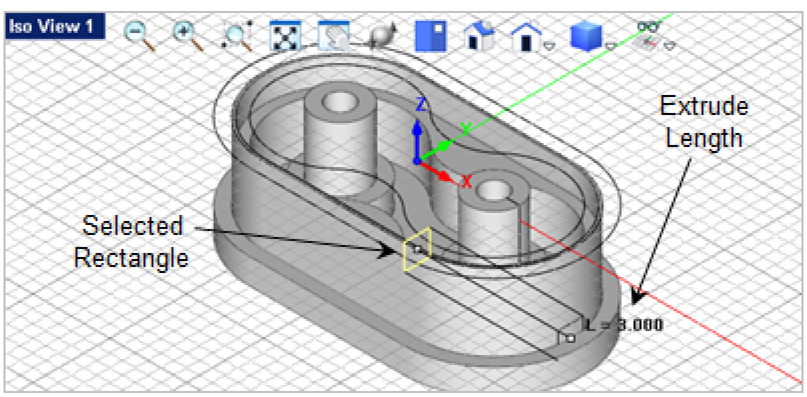
3. Select the grid points for the 1st and 2nd corners of the rectangle as shown.

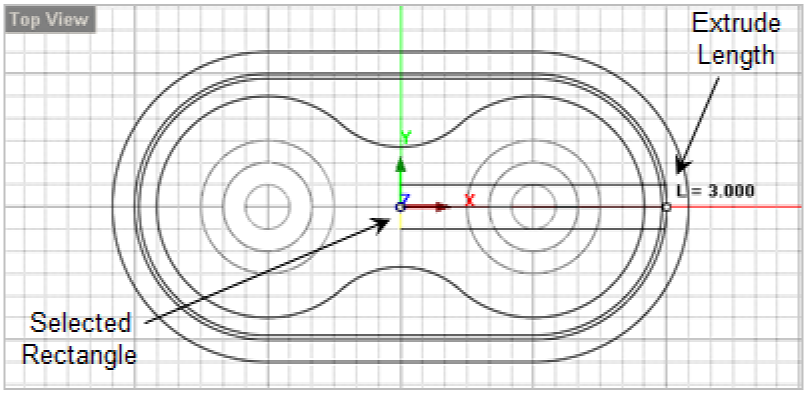


4. Now from the Mesh Modeling tab, Mesh Modeling select the Extrude Mesh  command.

5. Pick the rectangle we just created and drag the cursor over to the Front View **Front View** and pick the Grid point located 3.00" to the right as shown below.







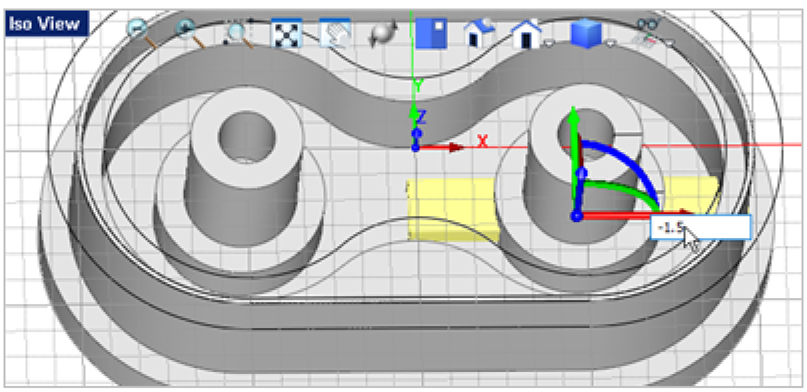
6. Now we want to use the [Graphic Manipulator] again so go to the [Status Bar] and make sure it is toggled On .

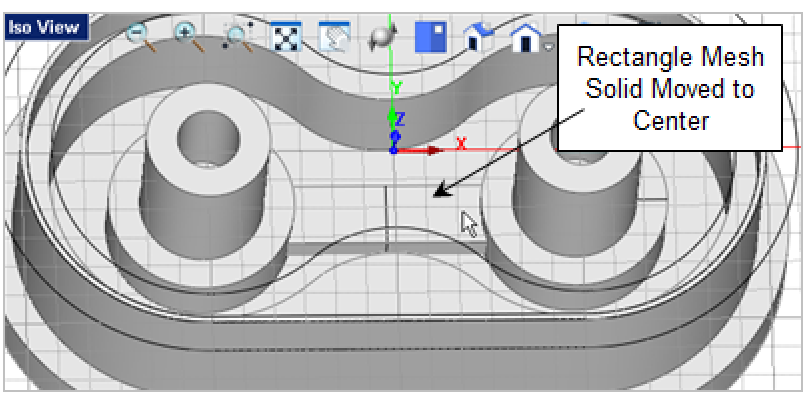
7. Select Iso View 1 

and right-click the mouse and drag rotate the part until you get a good view of the Connection Bar and select it.

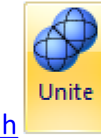
8. From the [Graphic Manipulator] pick the red X axis arrow, enter -1.5 

and then press <Enter> to see the Connector Bar move to the left to line up on center between the two bosses as shown.

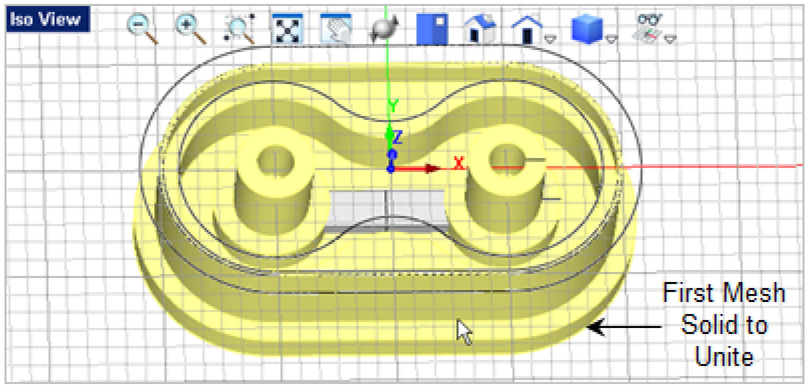


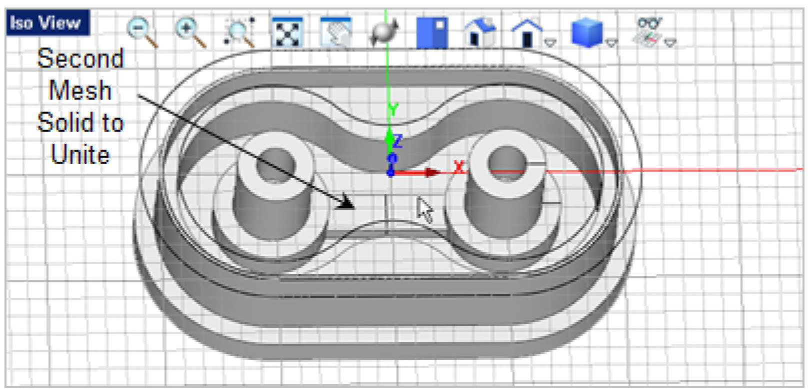


9. Now again we will unite the Connector Bar with the Body. From the Mesh Modeling tab, 

select the Unite Mesh 

command and select the body for the first mesh solid and the then select the Connector Bar to unite them.





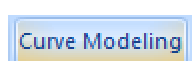
7.5.7 Extrude the Connection Wall

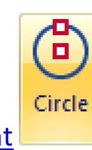
Extrude the Connection Wall

In this step we will create the wall that pass between the two center bosses.

1. Again, activate the Right View 

and then from the Display  menu on the View Toolbar, select Wireframe Display.

2. From the Curve Modeling tab, 

select the Circle on Point 

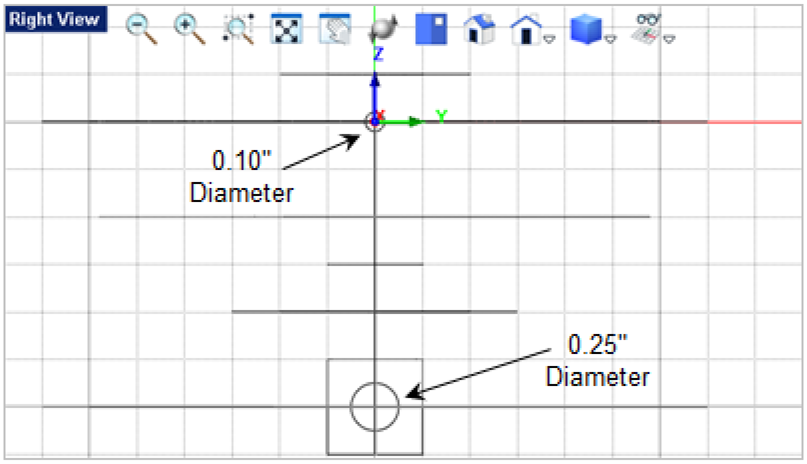
command.

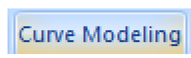
3. For the center point select the grid point shown below.

4. Enter 0.25 

for the diameter and press <Enter>.

5. Press <Enter> again to repeat the command. For the second circle select the grid point shown below and then enter 0.10  for the diameter and pres <Enter>. We will use these two circles to locate the corners of a polyline that defines the cross-section of the connection wall.



6. Now from the Curve Modeling tab,  select the Polyline  command.

7. From the Status Bar toggle on the Quad Point Snap  and toggle off all of the other object snaps.

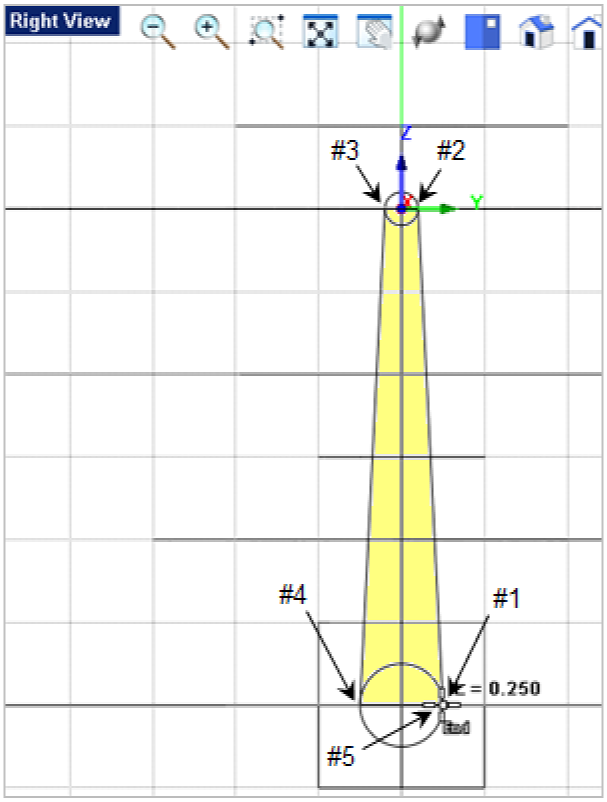
8. We will be drawing a 4-sided closed polyline that encompasses the yellow area shown in the image below. For the start point of the polyline select the quad point on the right side of the lower 0.25" diameter circle at the #1 location shown.

9. Then continue, by selecting the Quad point located at the right side of the upper 0.10" diameter circle.

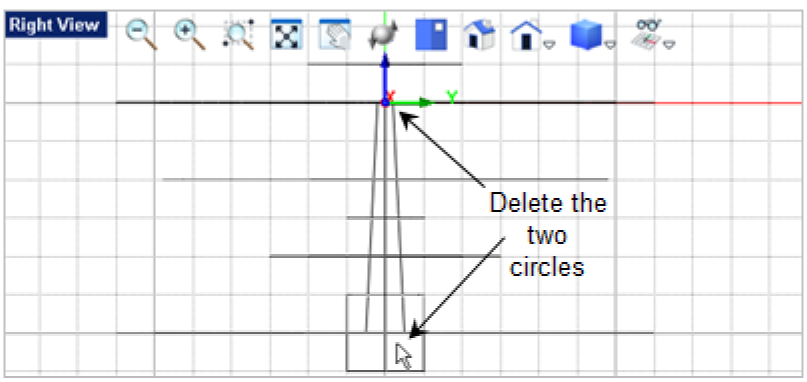
10. For the next point in the polyline select the Quad point on the left side of the upper 0.10" diameter circle at the #3 location.

11. Then continue by selecting the Quad point at the #4 location.

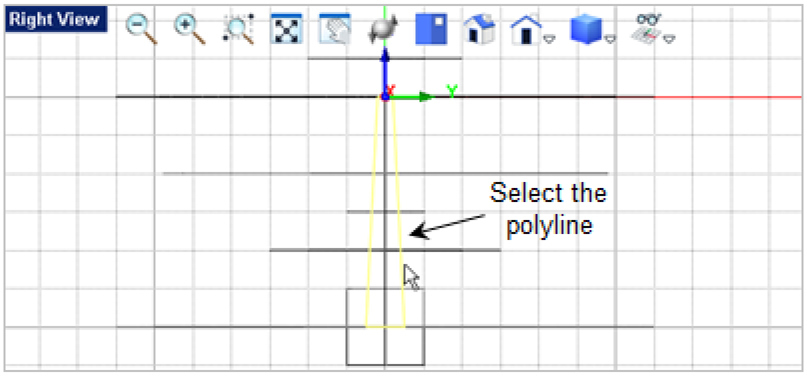
12. To complete the polyline select the Quad point at the #1 location where the polyline started. The Polyline  command will end automatically when the end point coincides with the start point.

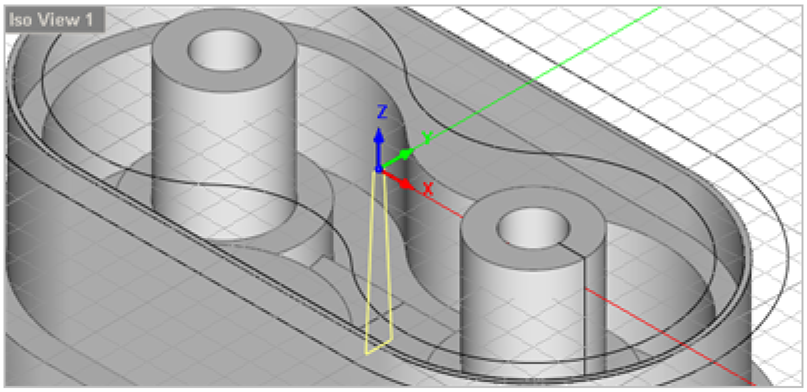


13. Now you can delete the two circles. Just select them and press the <Del> key.



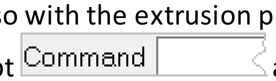
14. Now we will extrude the 4-sided polyline to create the Connecting Wall. First select the polyline we just created.



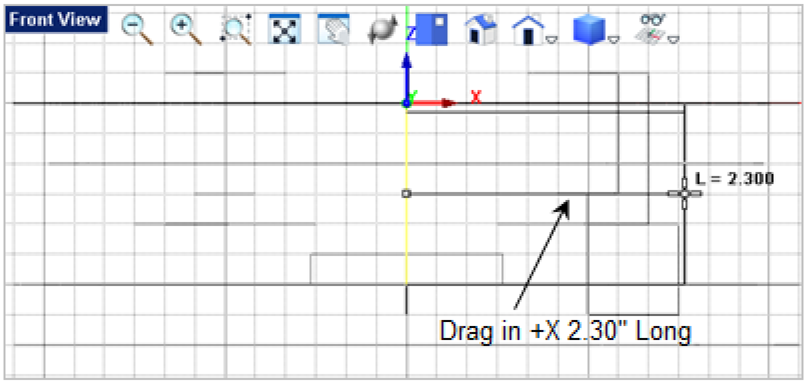


15. Now go to the Mesh Modeling tab  and select the Extrude Mesh  command again.

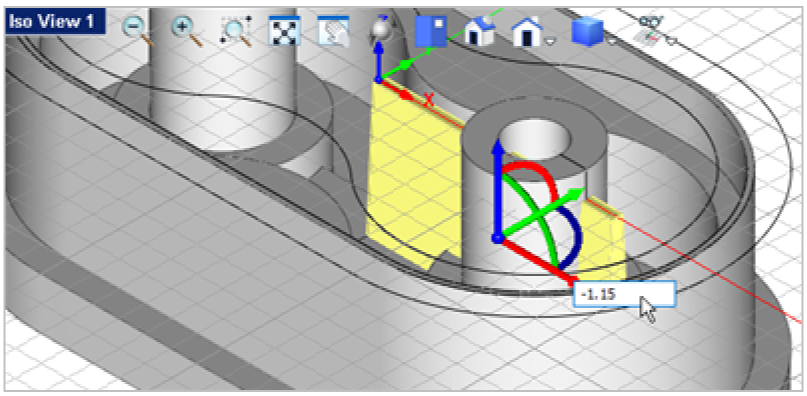
16. Drag the cursor to the Front View  and drag the length of the extrusion to the right of the center line where the selected polyline resides. Refer to the image below.

17. The point we want is not on a grid point, so with the extrusion previewed (do not select a point), go to the command prompt  and type in the distance 2.300 and press <Enter>.

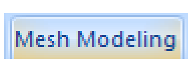
The Connection Wall is created.



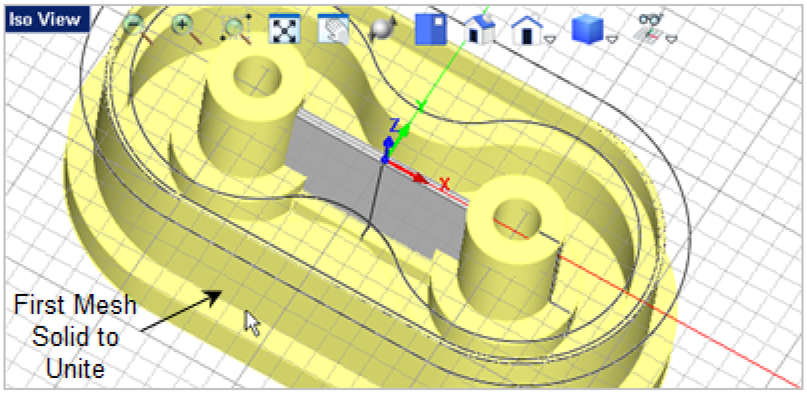
18. Now from Iso View 1  select the Connection Wall to display the Graphic Manipulator. If you do not see it, make sure it is toggled on from the Status Bar  .



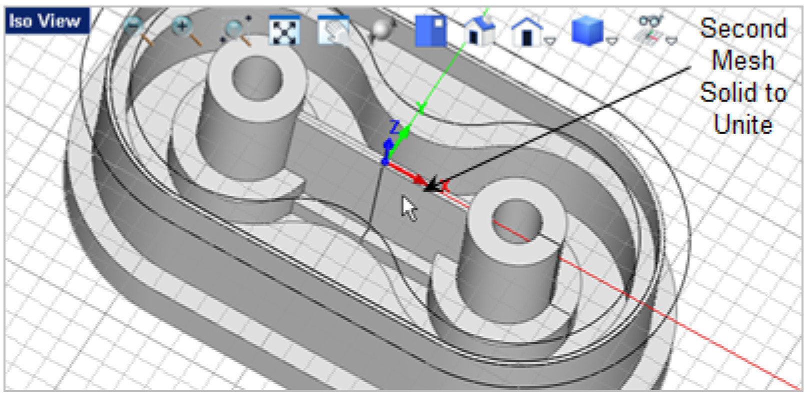
19. Pick the red X Axis arrow and type in -1.15  and press <Enter>. The Connection Wall should move into place at the center spanning between the two bosses as shown in the image below.

20. Now lets unite the Connection Wall with the body. Go to the Mesh Modeling tab  and select the Unite Mesh  command.

21. For the first mesh solid, select the body.



Then select the Connection Wall to unite it with the body.

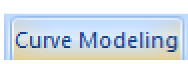


7.5.8 Extrude Ejector Pin Holes

Extrude the Ejector Pin Holes

In mold design, ejector pins are used to eject the molded part from the cavity side of the mold. As the mold press opens, the ejector pins slide and protrude through the cavity forcing the plastic part out of the mold where it falls into a catch bin. The ejector pin holes pass complete through the insert.

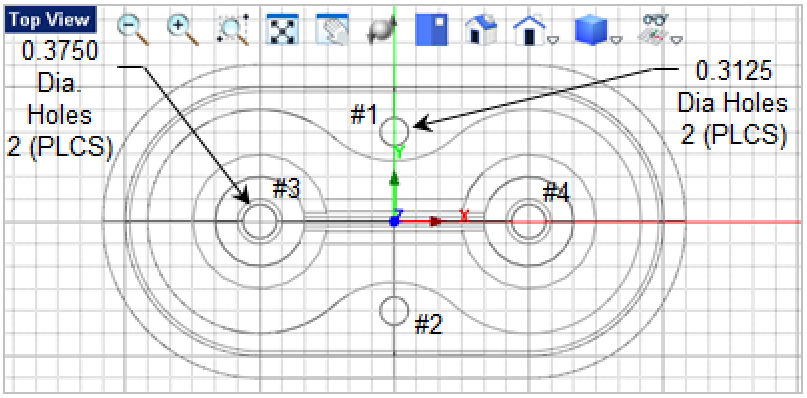
1. Activate the Top View 

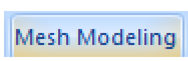
and then go to the Curve Modeling tab 

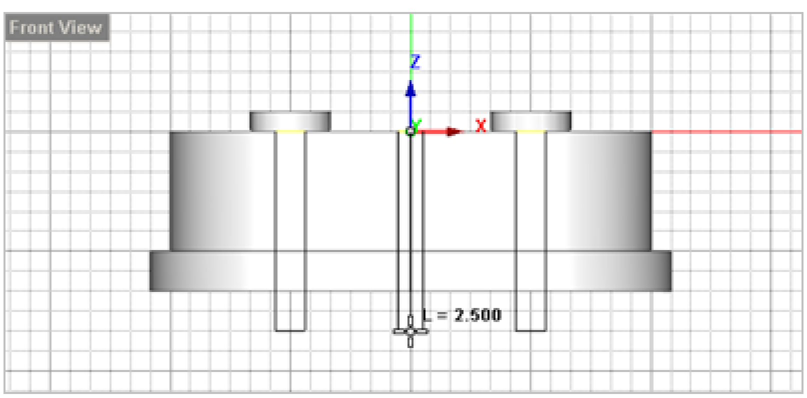
and select the Circle on Point 

command.

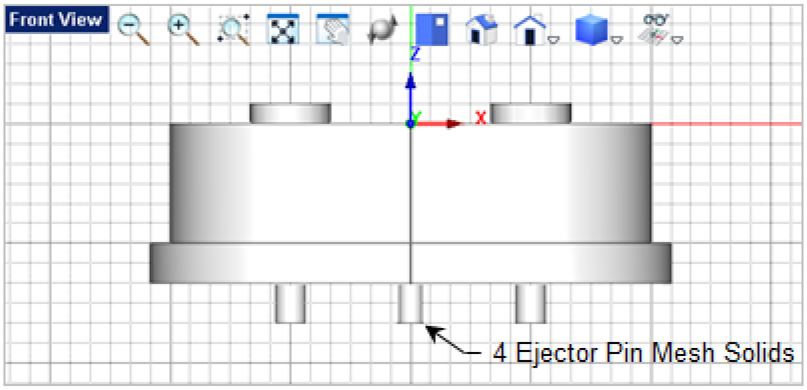
2. We will draw a total of 4 holes. Two will be 0.3125" in diameter (located at #1 and #2) and two will be 0.3750" diameter (located at #3 and #4).



3. Select the grid point at the #1 location and then enter 0.3125 in the command prompt  and then press <Enter>.
4. Press <Enter> to repeat the command.
5. Select the grid point at the #2 location and press <Enter> again to accept the new default diameter.
6. Now press <Enter> to repeat the command, select the grid point at the #4 location and then enter 0.375 in the command prompt  and then <Enter>.
7. Now press <Enter> again, select the grid point at the #4 location and press <Enter> to accept the new default diameter. The 4 circles are drawn.
8. Now with the <Ctrl> key pressed, select the 4 holes, go to the Mesh Modeling tab  and select the Extrude Mesh  command.
9. Now move the cursor down to the Front View  and drag the extrude length down past the bottom of the body and pick the grid pint located at 2.500" below the part.



The 4 mesh solids are created:

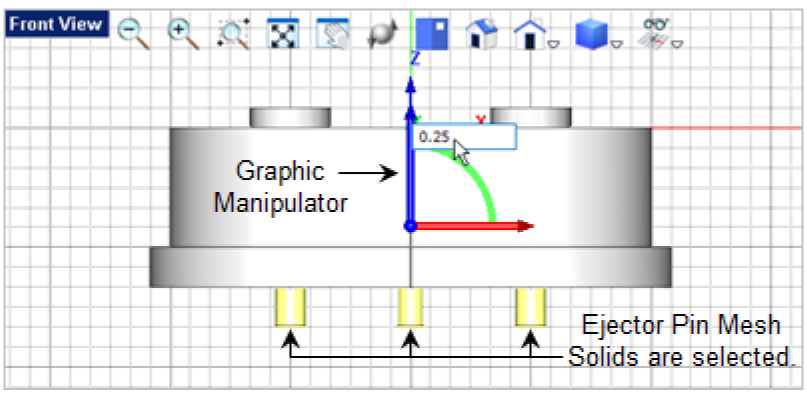


10. Now let's move the 4 ejector pin holes upward to pass through the body completely.
    Window select right to left so that the cross-hatched window passes through all 4 ejector pin mesh solids.

11. If the [Graphic Manipulator] does not display, pick the icon on the [Status Bar] to toggle it on .

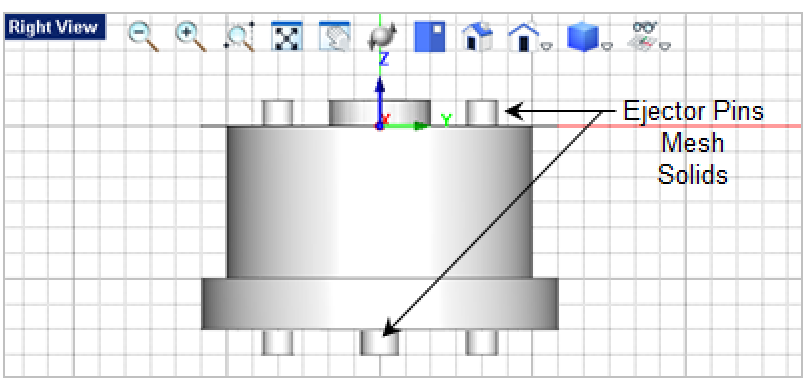
12. Select the Blue Z arrow and in the input window enter 0.25 

and press the <Enter> key twice. The first accepts the value and the second to move the mesh solids.

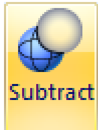


Activate the Right View 

and we see the ejector pin mesh solids.

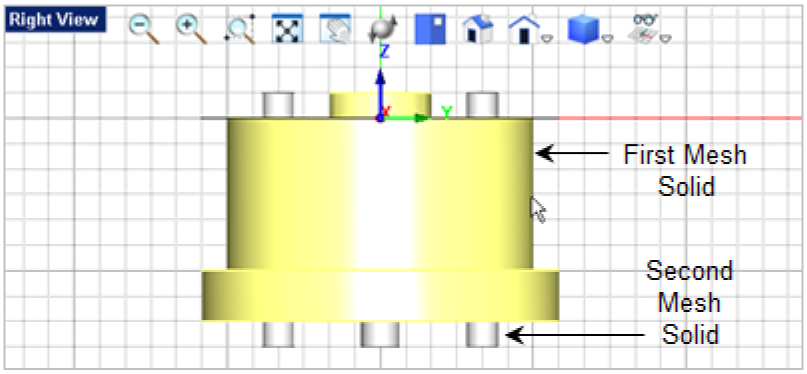


13. Now to subtract the ejector pin mesh solids from the body to create the ejector pin holes go to the Mesh Modeling tab 

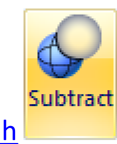
and select the Subtract Mesh 

command.

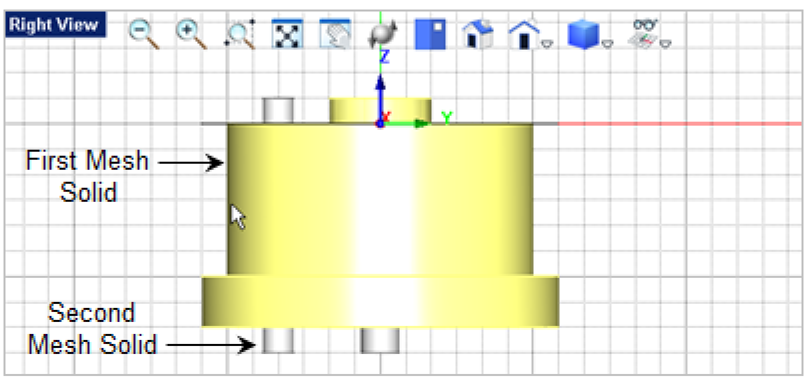
14. Select the body as the first mesh solid.



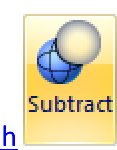
15. Then select the 0.3125" ejector pin mesh solid on the right to subtract it from the body.

16. Press the <Enter> key to repeat the Subtract Mesh 

command and select the body again as the first mesh solid.



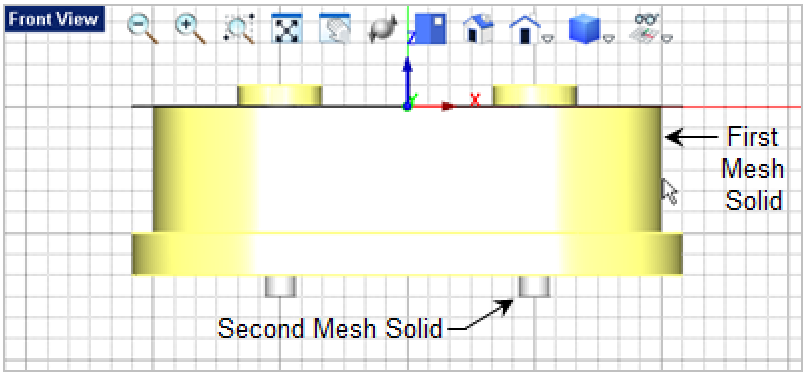
17. Then select the 0.3125" ejector pin mesh solid on the left to subtract it from the body.

18. Press the <Enter> key to repeat the Subtract Mesh 

command and this time to subtract the 0.3750" ejector pins move to the Front View 

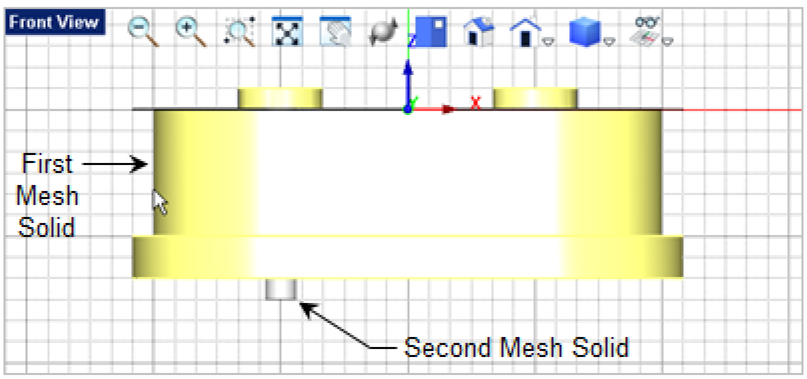
.

19. Select the body again as the first mesh solid.



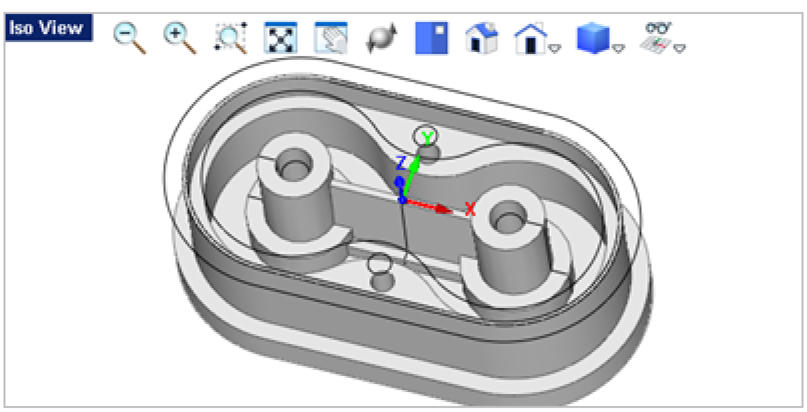
20. This time select the 0.3750" ejector pin mesh solid on the right to subtract it from the body.

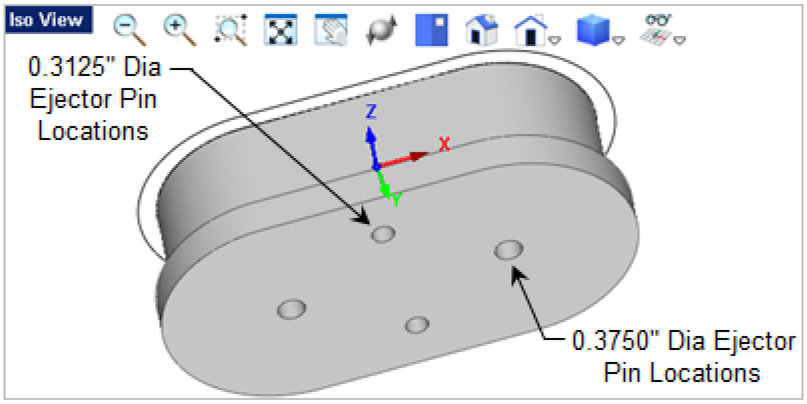
21. Press the <Enter> key again to repeat the command and then select the body mesh solid.



22. Now select the remaining 0.3750" ejector pin mesh solid to subtract it from the body.

    You will now see that the 4 ejector pin hole locations pass completely through the mold insert as shown in the images below:





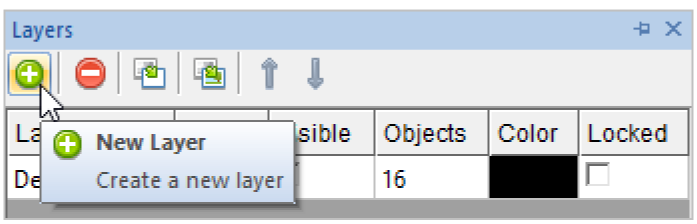
7.5.9 Change Geometry Layer

Change Geometry Layer

We have a lot of curves displaying on the part. Let's clean them up.

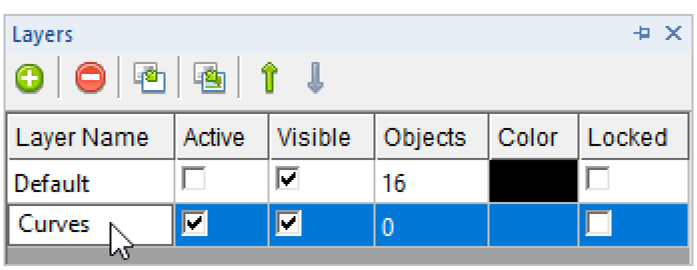
1. Display the Layer Manager. You can pick the Layer Manager icon located on the Status Bar or select the Layers flay-out tab located on the left side of the drawing window. If you recall we unpinned the Layer Manager allowing it to collapse to the side of the screen.

2. From the Layer Manager select the Add Layer icon to add a new layer.

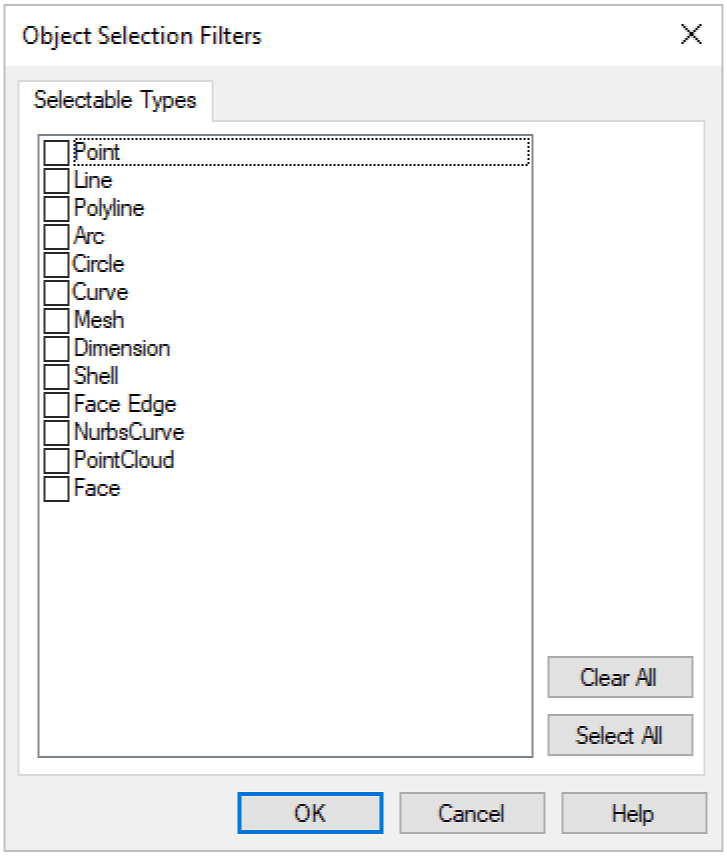


3. Now lets rename the new layer to Curves. First double-left-click in the Name field to activate it, and enter Curves as the new layer name.

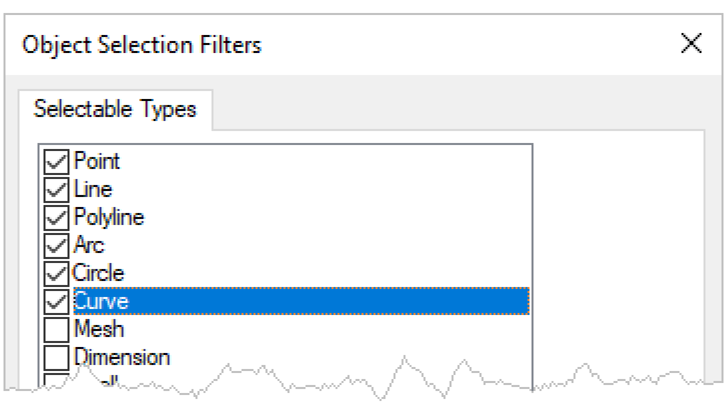
4. Next lets make the Default layer the Active Layer by checking the box in the Active column for the layer Default. The Default layer is now the active layer. Let's also uncheck the box in the visible column for the Curves layer. Now any geometry placed on the Curves layer will be hidden from view. Your Layer Manager should look like the one shown below.



5. Now VisualCAD has a quick way of selecting entities by type. Go to the Modeling Aids  tab and select the By Type  icon. This command displays the Object Selection Filters dialog that allows you to check mark the entities that you want to select. That dialog is shown below:



6. Place a check mark in the box for each of the following entity types: Point, Line,Polyline, Arc, Circle, and Curve.

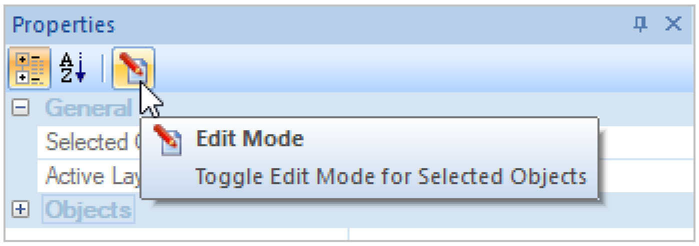


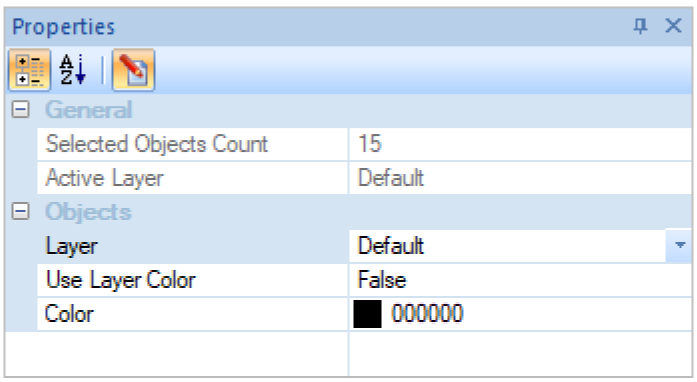
7. Now pick the OK button. You will see that all of the reference curves we created in this exercise are highlighted for selection.

8. With the entitles selected, select the Properties icon from the Status Bar. Its located just to the left of the Layer Manager icon.

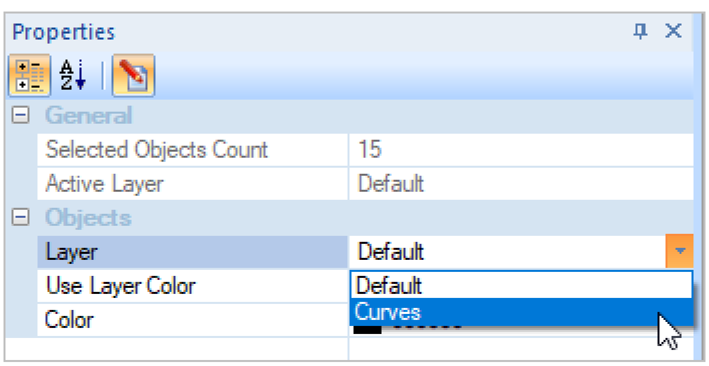
9. The Properties Manager will display on the left side of the screen similar to the Layer Manager.

10. At the top of the Properties Manager select the Edit icon. This will display additional fields that you can edit.

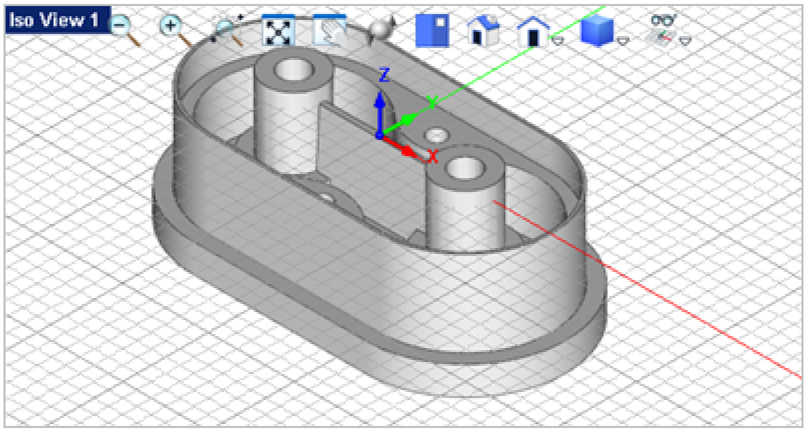




11. We want to change the layer of the curves so drop down the layer menu by selecting the menu indicator on the right side in the Layer row and select the new layer named Curves that we recently created and hidden.



12. Now when you click anywhere in the drawing window you will see that all of the curves have been moved to the Curves layer and are not hidden from view.



7.5.10 Create Section Curves

Create Section Curves

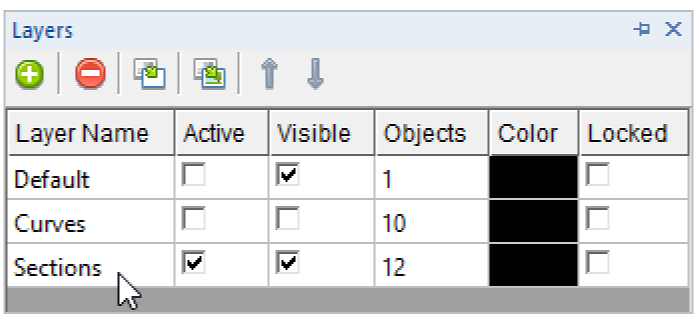
Congratulations, our part is now nearly complete! However, we do want to create cross-section curves that will aid you when it comes time to create a detail drawing of the part.

Section curves are drawn parallel to the active Viewport. So the first step is to select the view that you want the section to be parallel to. We want to create three sections, one through the center parallel to the Front View and another two through each boss parallel to the Right View.

1. First lets again create a new layer and make it the active layer so that the section curves will be assigned to that layer automatically when they are created.

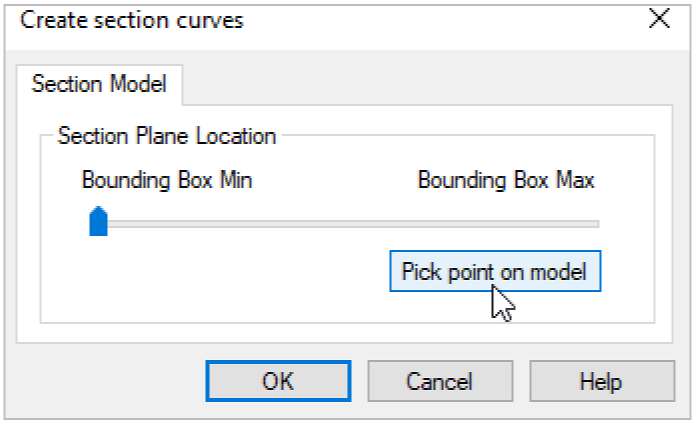
2. Display the Layer Manager one again.

3. Create a new layer and name it Sections and also make it the active layer. The Layer Manager should look like this:

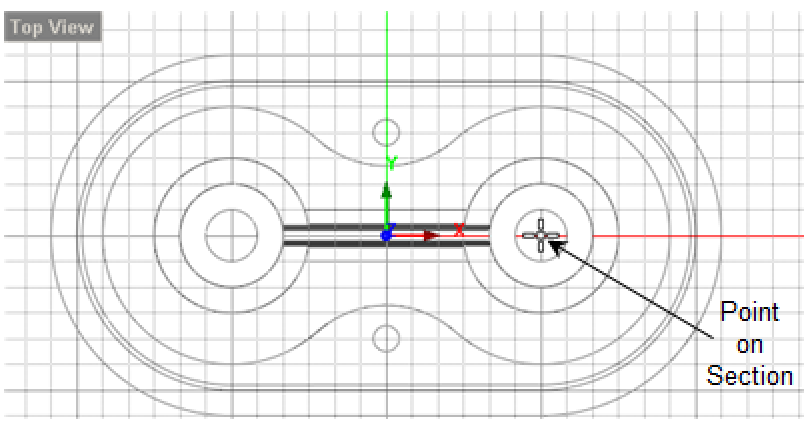

Layers

Layer Name	Active	Visible	Objects	Color	Locked
Default	☐	☑	1		☐
Curves	☐	☐	10		☐
Sections	☑	☑	12		☐

4. Now go to the Curve Modeling tab Curve Modeling and select the Section Curve command. The Create section curves dialog will display.

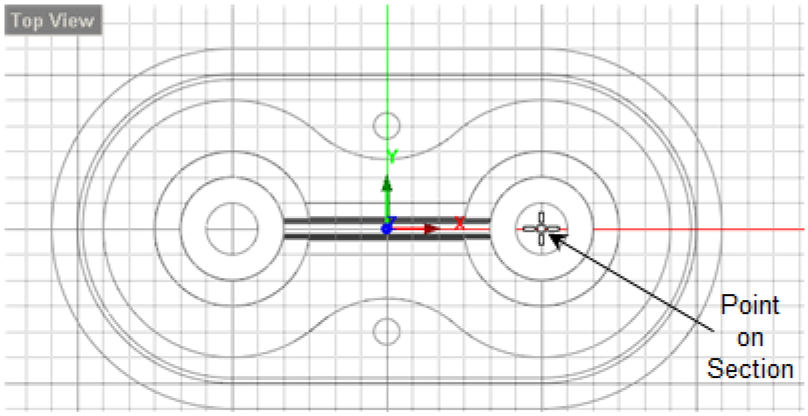


5. First select anywhere in the Front View Front View to activate it.
6. Now if you move the slider in the dialog you will see the section curves move along the part.
7. We want the location of the section to be precise so lets pick the Button that says "Pick point on model".
8. Now from the Top View Top View pick the grid point location at the center of one of the two bosses as shown below. Notice that the Top View IS NOT ACTIVE. We want the Front View Front View to stay active while the section point is picked.

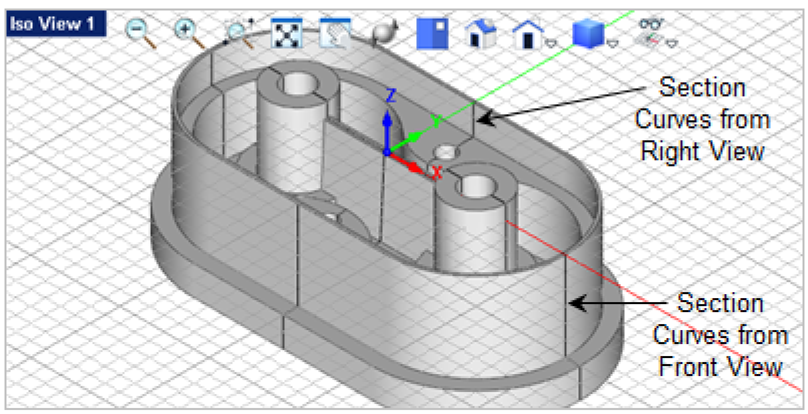


9. When you pick the point at the center of the boss, you will see the section curves display on the part. They pass through the center parallel to the Front View just like we want.
10. Now press the <Enter> key to repeat the section command and then activate the Right View Right View.
11. From the Create section curves dialog select the button again that says "Pick point on model".

12. Now from the Top View pick the grid point location at the center of the boss on the right side. Notice that the Top View IS NOT ACTIVE. We want the Right View to stay active while the section point is picked.



13. When you pick the point at the center of the boss, you will see the section curves display on the part. They pass through the center of the boss parallel to the Right View just like we want.

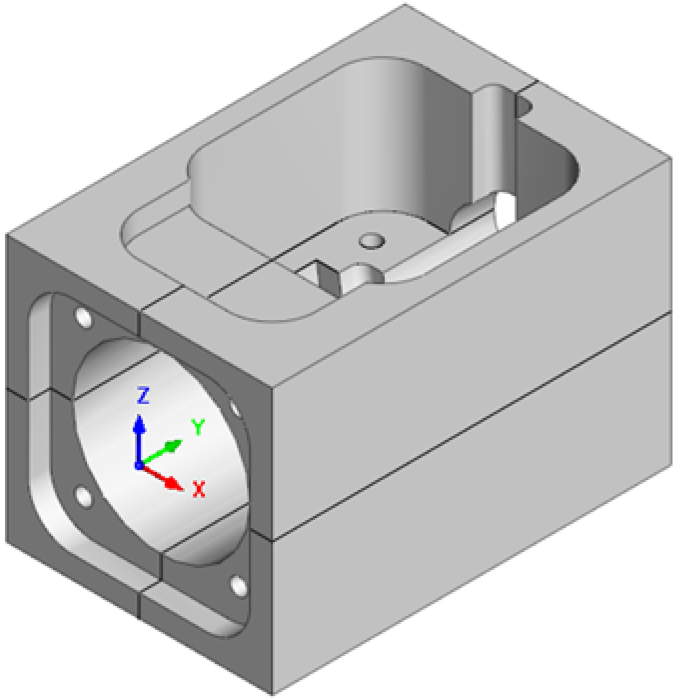


Congratulations!! You modeled your Very first mold insert using VisualCAD!

14. Save the file as Insert_Completed.vcp.

7.6 #6: Model a Connector Block

In this exercise you will model the Connector Block shown below. It too builds upon the tasks you have learned in the previous exercises. You will draw curve profiles and then turn them into part features. The Graphic Manipulator is again used extensively in this exercise as well as modeling from multiple viewports. Again, just go slowly and save your work often and Undo if needed.



Model a Connector Block

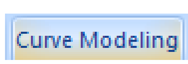
What you will learn:

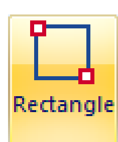
In this exercise you will perform the following VisualCAD tasks:

1. Model the Main Body.
2. Model the Front Access.
3. Model the Top Access.
4. Model the Mounting Holes.
5. Modify the Top Access.
6. Create Section Curves.

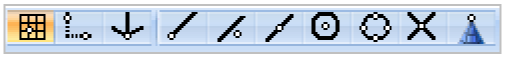
7.6.1 Model the Body

Model the Main Body

1. Select the Curve Modeling tab  from the top Ribbon Bar.

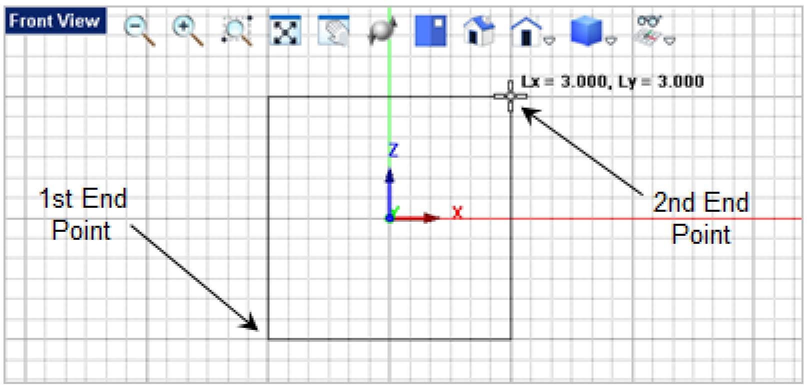
2. Now select the Rectangle command. 

3. Turn on Grid snap and turn off all other object Snaps. Object Snap toggles are located on the Status Toolbar at the bottom of the display.



4. For the first corner of the rectangle, select the lower left grid point location shown below or type -1.5, 0, -1.5 in the Command Prompt Command and press <Enter>.

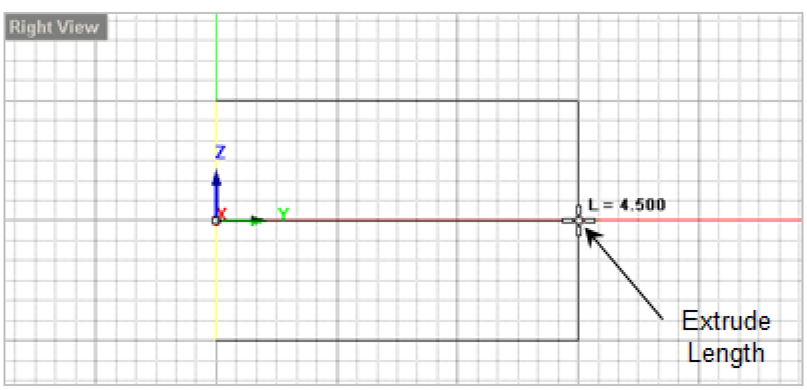
5. For the second corner of the rectangle, select the upper right grid point shown above or type 3,3 in the Command Prompt Command and press <Enter>. This will create a rectangle 3" square.



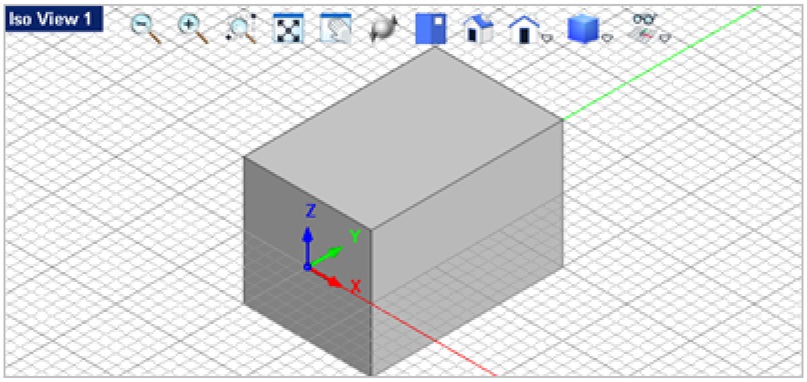
6. Now select the rectangle you just drew and then from the Mesh Modeling tab select the Extrude Mesh

command.

7. Now drag the cursor over to the Right View and select the grid point at 4.5 or just enter 4.5 in the command prompt Command and press <Enter>.



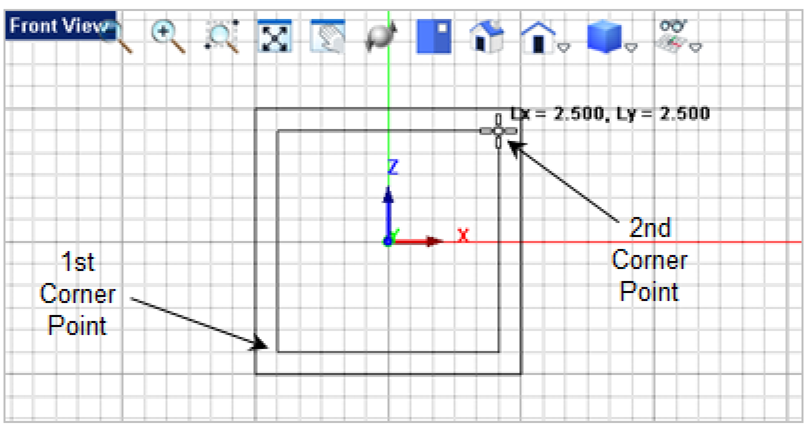
Your part should look like this in Iso View 1:



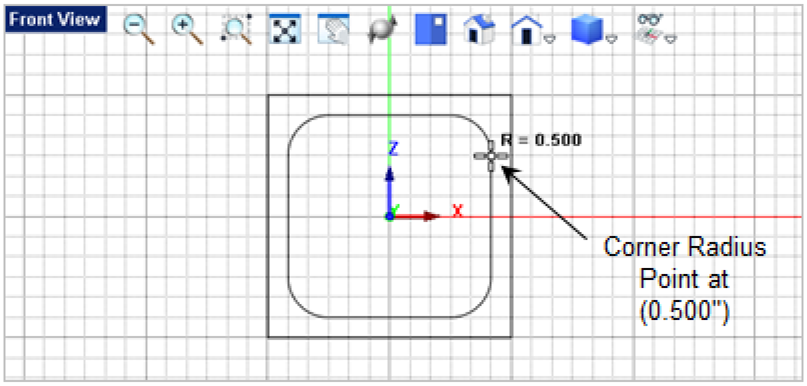
7.6.2 Model the Front Access

Model the Front Access

1. Activate the Front View.
2. From the tab Curve Modeling select the Rounded Rectangle command.
3. For the first corner of the rectangle, select the lower left grid point location shown below or type 1.25,1.25,0 in the Command Prompt Command and press <Enter>.

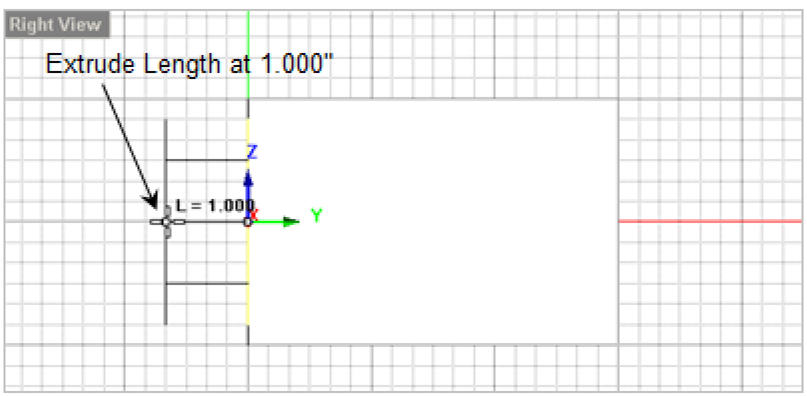


4. For the second corner of the rectangle, select the lower left grid point location shown below or type 1.25,1.25,0 in the Command Prompt Command and press <Enter>.
5. Now for the corner radius select the Grid point shown below or enter 0.5 in the Command Window Command and press <Enter>.

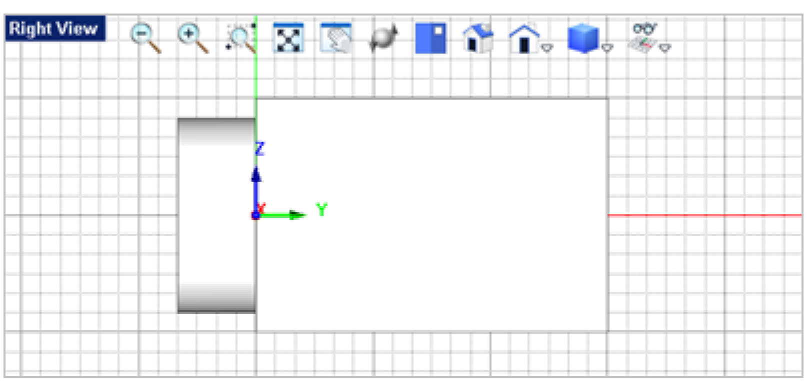


6. Now select the rounded rectangle you just created and from the  tab, select the Extrude Mesh  command.

7. Move the cursor over to the  and select the Extrude Length at the 1.000 Grid point or enter 1 in the command window  and press <Enter>.



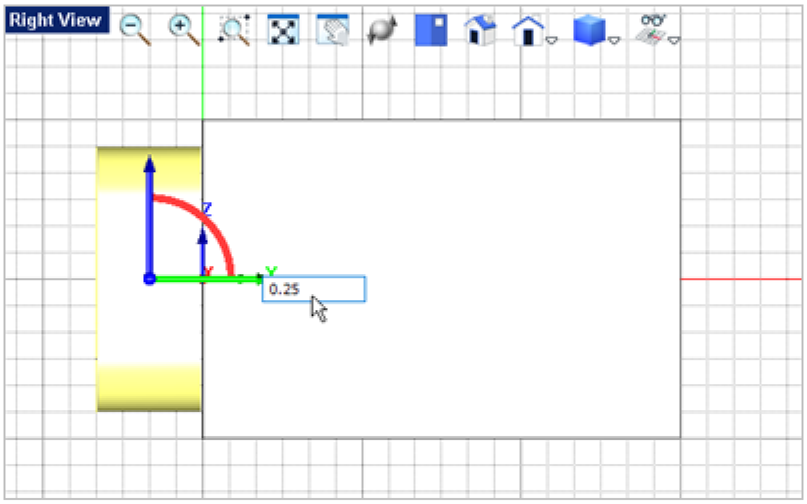
The extruded profile is shown below.



8. Now we will move the extrusion so that it intersects the body. Activate the Graphic Manipulator from the Status Bar.

9. Select the extrusion mesh solid to display the Graphic Manipulator.

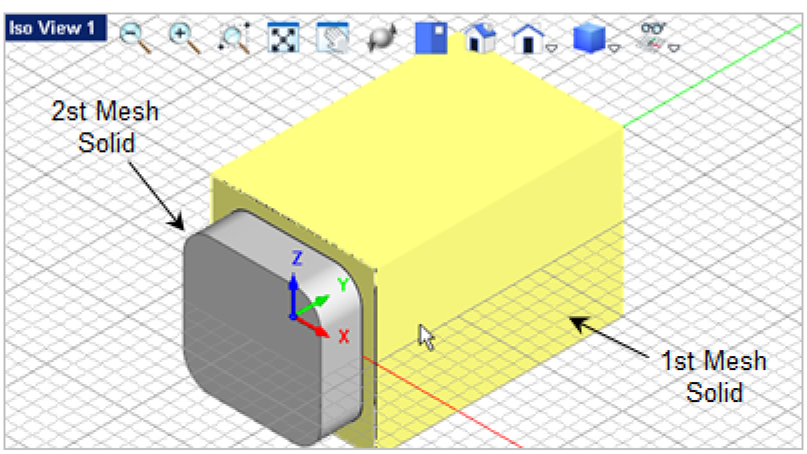
10. Pick the Green Y axis of the Graphic Manipulator and enter 0.25 in the input field and press <Enter>. The extrusion will move in the Y direction the amount specified.



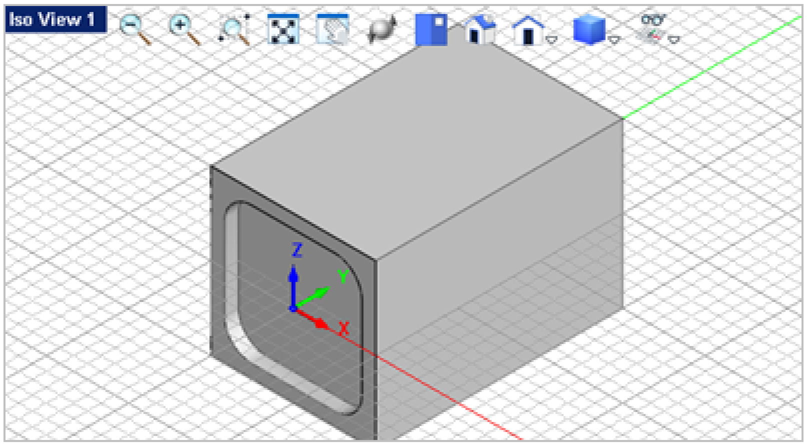
11. Press <Enter> again to accept the move.

12. Now activate the Iso View 1 window and from the Mesh Modeling tab select the Subtract Mesh Subtract command.

13. For the first mesh solid select the Body.

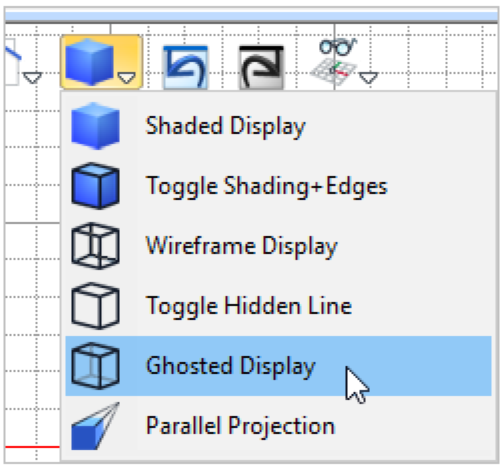


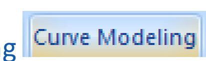
14. For the second mesh solid select the extrusion. The extrusion will be subtracted from the Body as shown below.



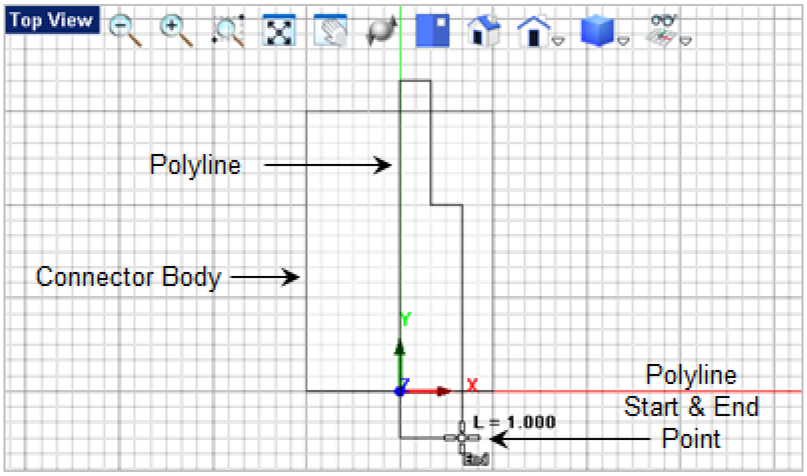
15. Now activate the Top View.

16. Select the View  menu from the View Toolbar and then select Ghosted Display. Note: The View Toolbar will only display in the active viewport.

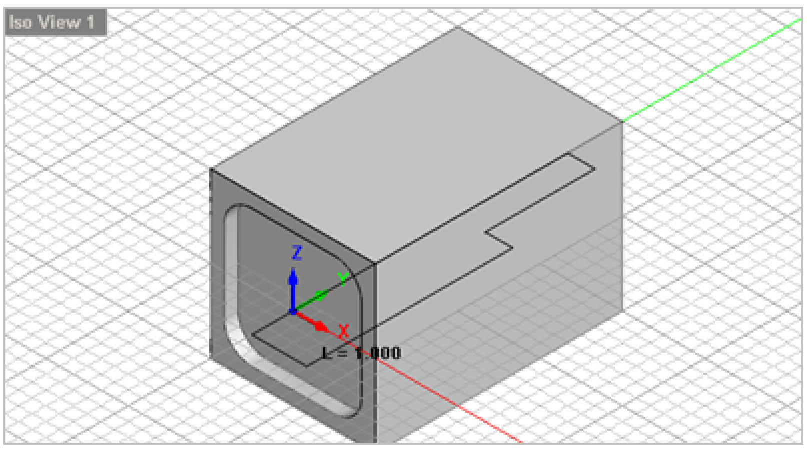


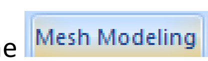
17. Now from the Curve Modeling  tab select the Polyline  command.

18. You will select the grid points shown below to define a closed polyline. The first grid point is the same as the last grid point.



In Iso View 1 the closed curve polyline looks like this:



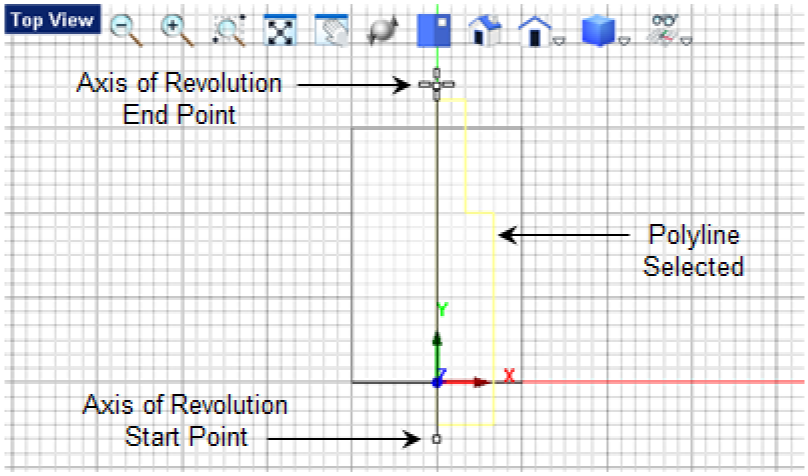
19. Now select the closed curve polyline you just created and from the 

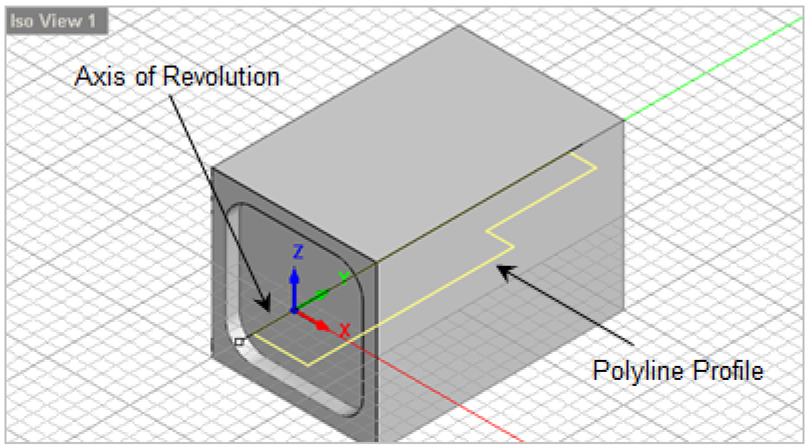
tab select the Revolve Mesh 

command.

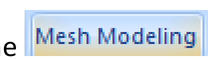
20. The command line 

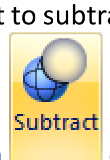
prompts you to "Pick start point if axis of revolution or enter coordinates x,y and z". Pick the grid points shown below for the start and end points of the axis of revolution.



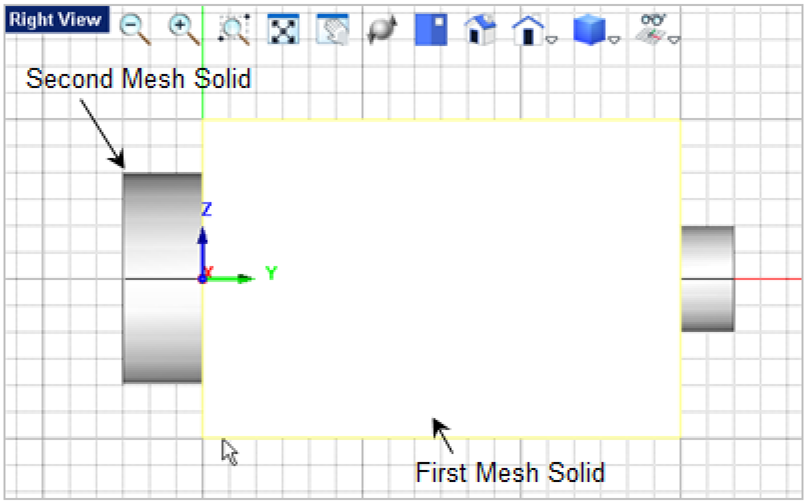


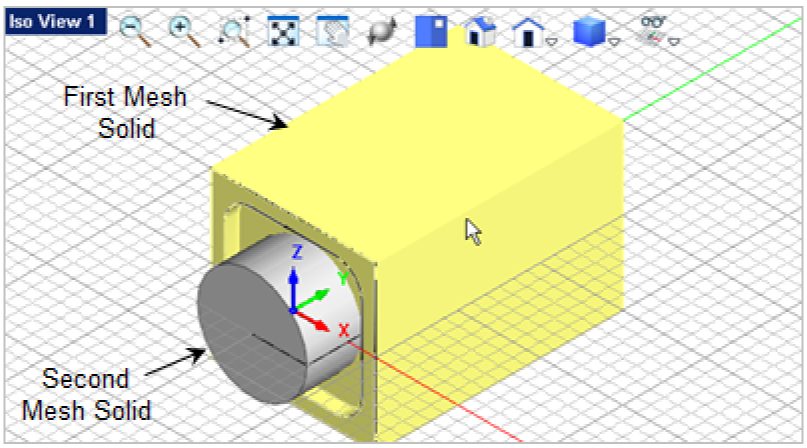
The Revolve mesh solid is created.

21. Now you want to subtract it from the Body. From the 

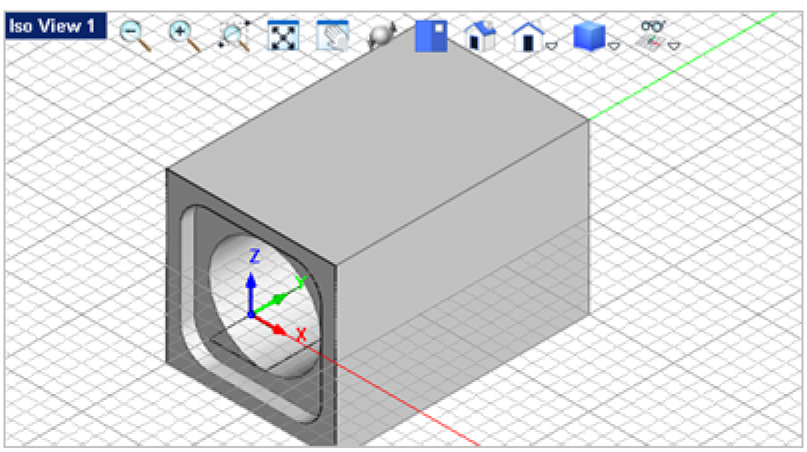
tab select the Subtract Mesh 

command.
22. For the first mesh solid select the Body.
23. For the second mesh solid select the Revolve.





The Body is updated and should look like the image below.



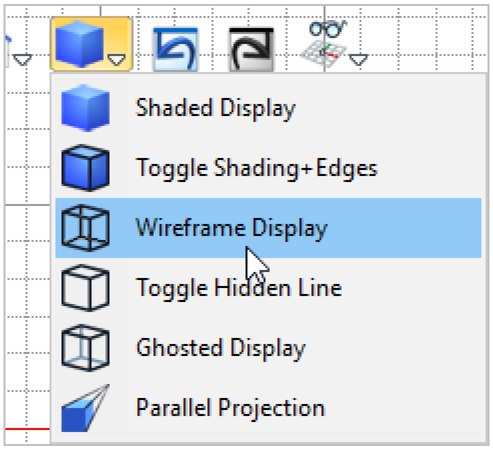
7.6.3 Model the Top Access

Model the Top Access

1. Activate the 

.

2. Select the View  menu from the View Toolbar and then select Wireframe Display. Note: The View Toolbar will only display in the active viewport.

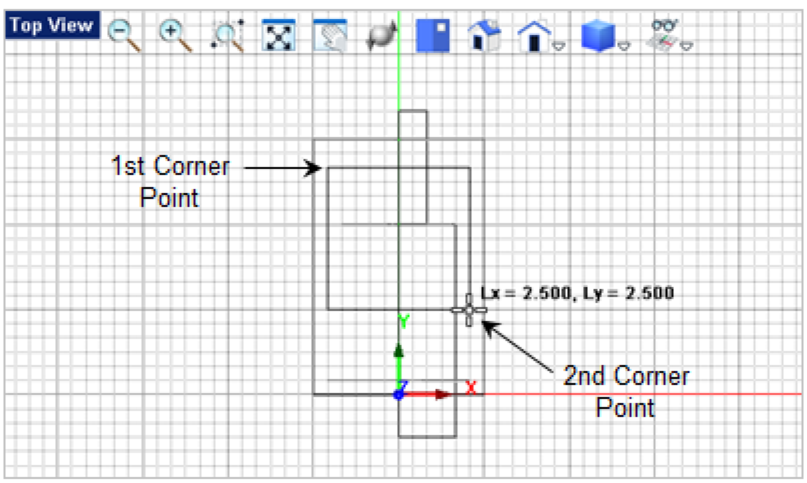


From the 

tab select the Rounder Rectangle  command.

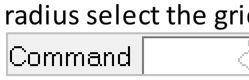
3. For the first corner of the rectangle, select the upper right grid point location shown below or type 1.25,1.25,0 in the Command Prompt 

and press <Enter>.

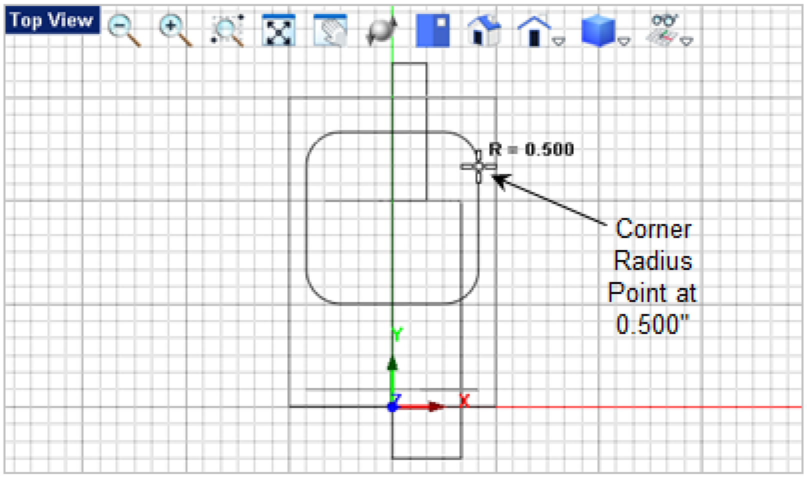


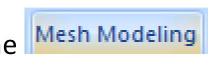
4. For the second corner of the rectangle, select the lower left grid point location shown below or type 1.25,1.5,0 in the Command Prompt 

and press <Enter>.

5. Now for the corner radius select the grid point shown below or enter 0.5 in the Command Window 

and press <Enter>.



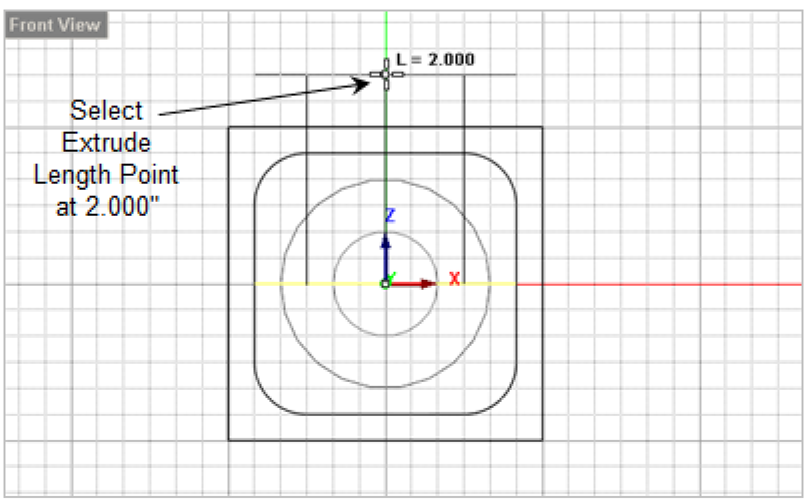
6. Now select the rounded rectangle you just created and from the 

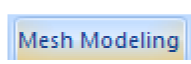
tab, select the Extrude Mesh 

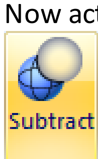
command.

7. Move the cursor down to the Front View and select the Extrude Length at the 2.000 Grid point or enter 0,0,2 in the command window 

and press <Enter>.

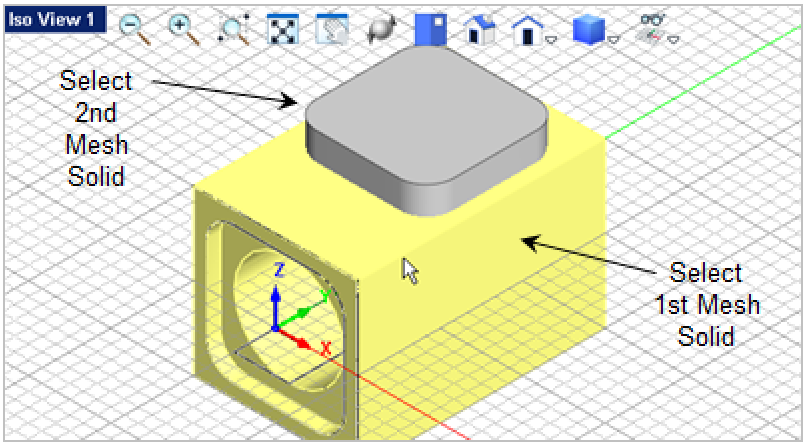


8. Now activate Iso View 1 and from the 

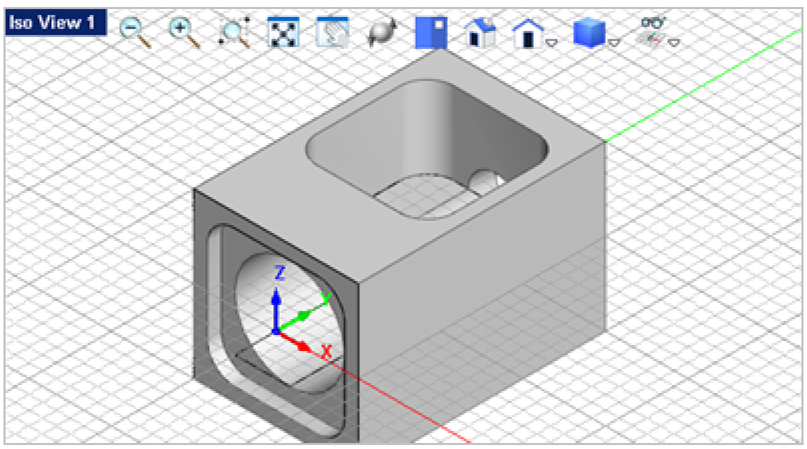
tab select the Subtract Mesh 

command.

9. For the first mesh solid select the Body.



10. For the second mesh solid select the extrusion. The extrusion will be subtracted from the Body as shown below.

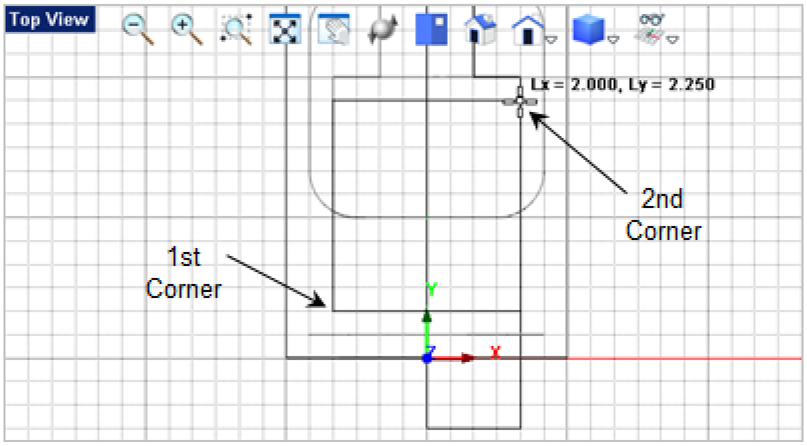


11. The top access has an additional pocket near the front of the body that needs to be modeled.

12. Activate the Top View again.

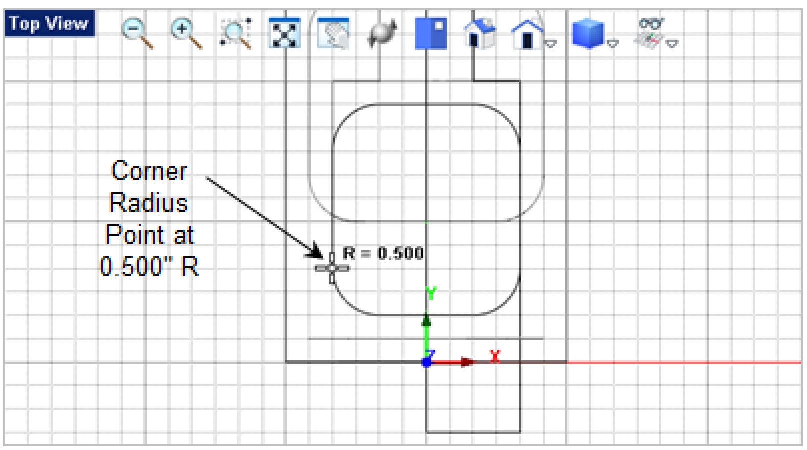
13. From the Curve Modeling tab select the Rounded Rectangle command.

14. For the first corner of the rectangle, select the lower left grid point location shown below or type -1,0.5,0 in the Command Prompt Command and press <Enter>.



15. For the second corner of the rectangle, select the upper right grid point location shown above or type 1,2.75,0 in the Command Prompt Command and press <Enter>.

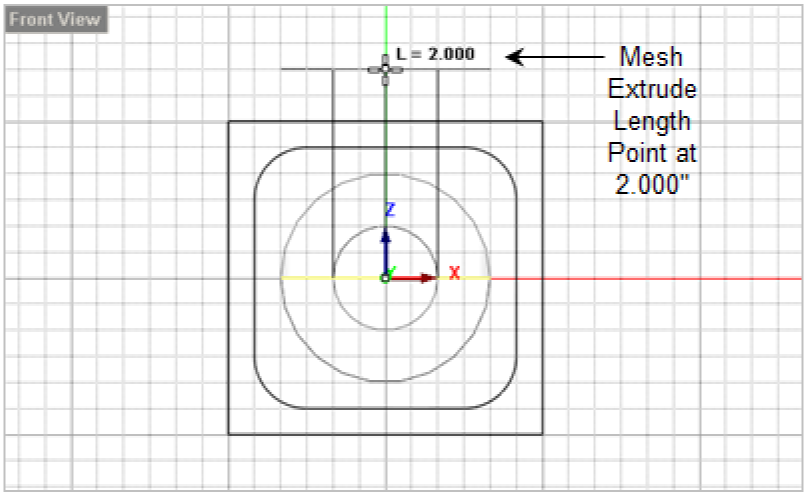
16. Now for the corner radius select the grid point shown below or enter 0.5 in the Command Window Command and press <Enter>.

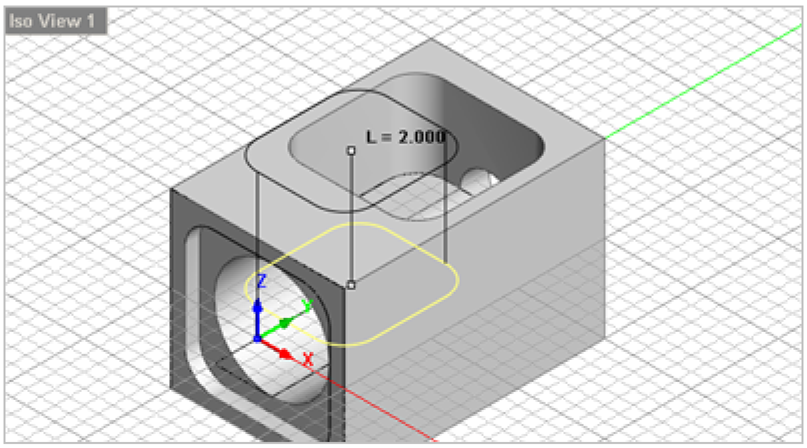


17. Now select the rounded rectangle you just created and from the Mesh Modeling tab, select the Extrude Mesh 

command.

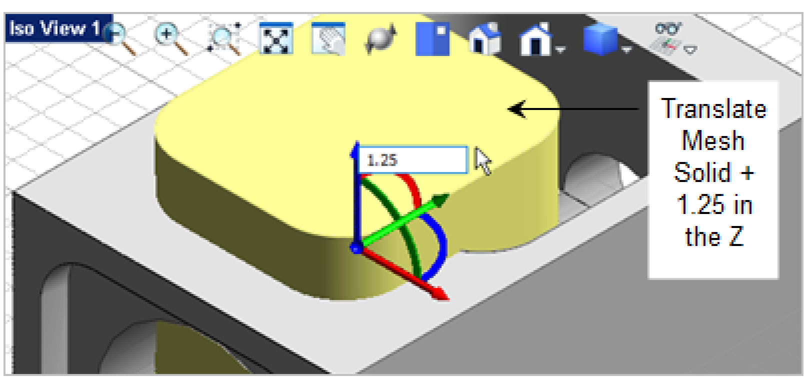
18. Move the cursor down to the Front View and select the Extrude Length at the 2.000 Grid point or enter 0,0,2 in the command window Command and press <Enter>.

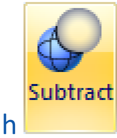


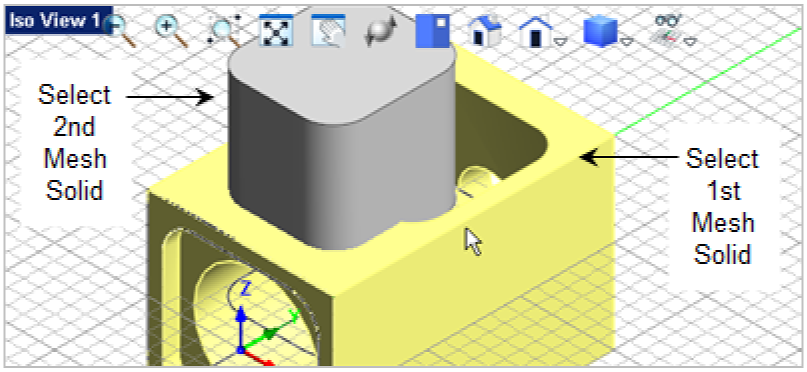


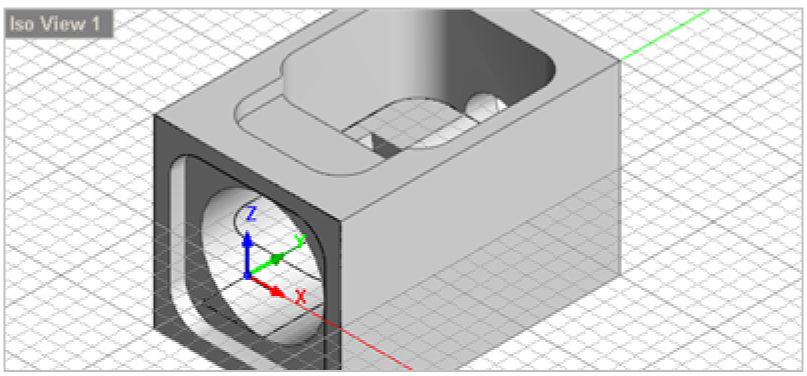
19. Now from Iso View 1, select the extrusion you just created to display the Graphic Manipulator .

20. Click on the Blue Z Axis arrow and enter 1.25 Command to translate the extrusion upward as shown in the illustration below.



21. From the  tab select the Subtract Mesh  command.
22. For the first mesh solid select the Body.
23. For the second mesh solid select the extrusion you just created.

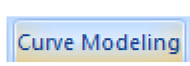
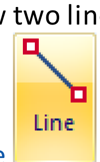




7.6.4 Model the Mounting Holes

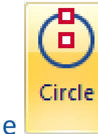
Model the Mounting Holes

In this step we will model the tap drill mounting holes in the front and top access. Note: For machining purposes only a Point is required to perform a hole making operation. We are only modeling the holes here for learning purposes.

1. Activate the .
2. To locate the center of the first hole you will draw two lines on the grid. From the Curve Modeling  tab select the Line  command.

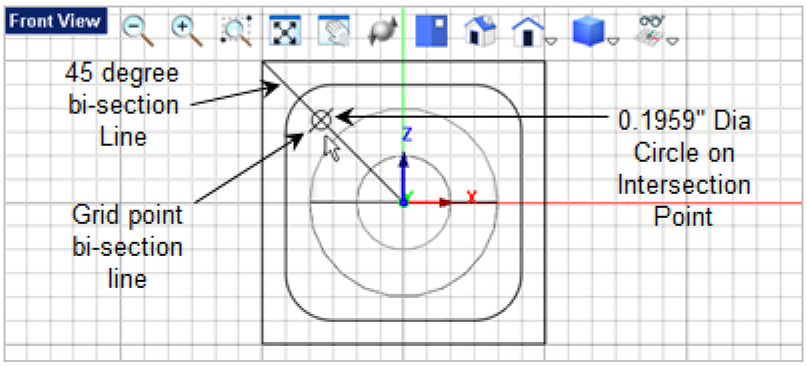
3. Draw the two lines shown below. The first starts at the center and ends at the top left corner. The second passes diagonally across two grid points bisection the first line at 90 degrees.

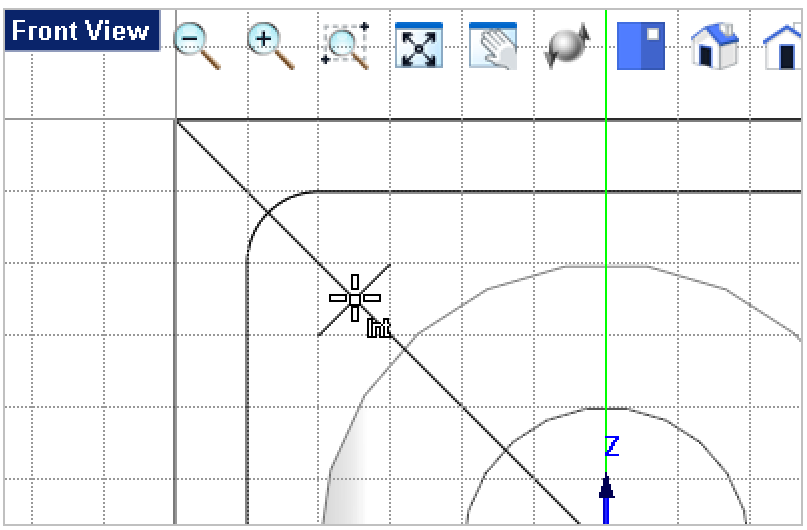
4. Now from the Curve Modeling 

tab select the Circle 

command.

5. For the circle center, activate the Intersect Object Snap  from the Status Bar and then move the cursor over the intersection of the two lines until you see Int display next to the cursor and then select the intersection point.





6. For the circle diameter enter 0.1959 in the Command Window 

and press <Enter>. This is the #7 drill diameter for a 1/4-20 tap.

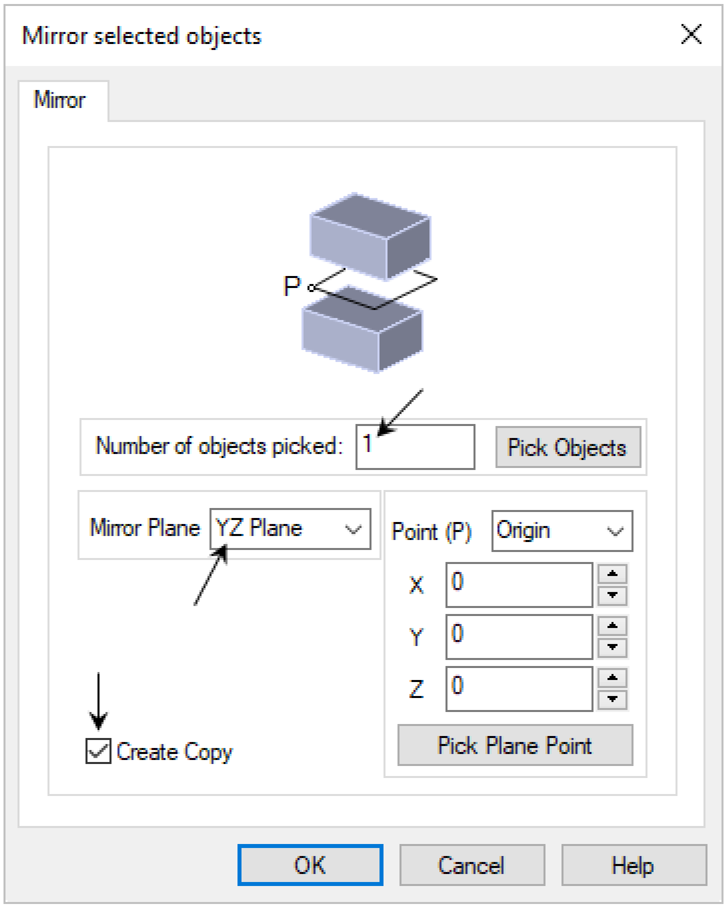
7. Now you need 4 of these holes. Let's use the mirror command to create the other three. First select the circle you just created, then go to the Modify / Transform 

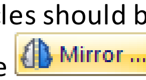
tab and select the 

command.

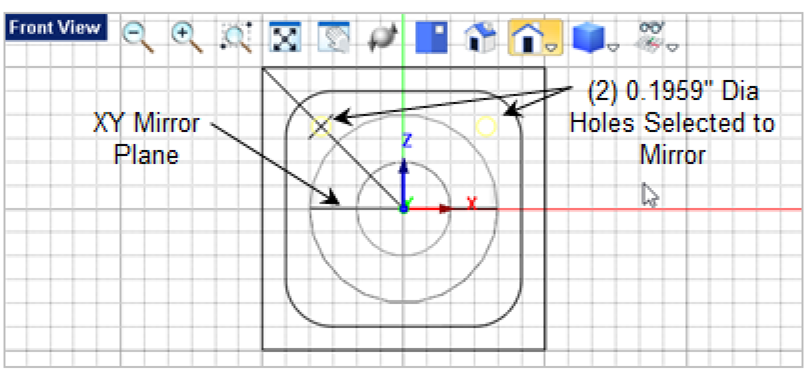
Note that the Modify / Transfer tab ONLY appears when an entity is selected!

8. From the Mirror selected objects dialog, the Number of selected objects should be 1, The Mirror Plane should be YZ Plane and box next to Create Copy should be checked.



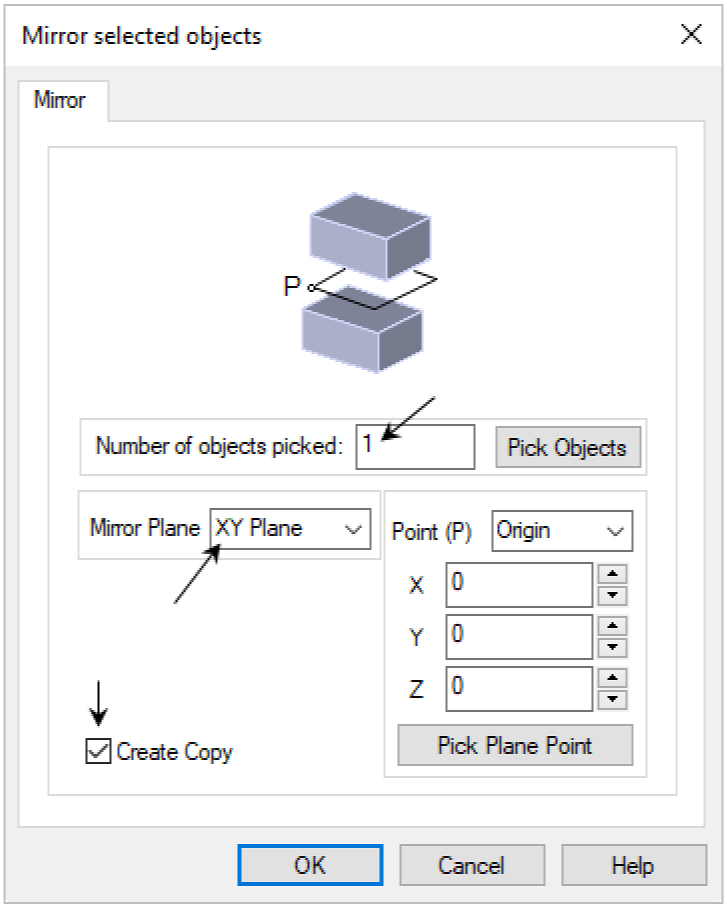
9. Pick OK from the dialog and the circle will be mirrored and both circles should be selected automatically. If not, select both of the circles and pick the 

command again.

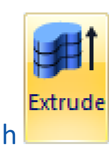


10. This time from the Mirror selected objects dialog, the Number of selected objects should be 2, The Mirror Plane should be XY Plane and box next to Create Copy should be checked.

    Note: You do not have to select the geometry first, as you can see this dialog has a Pick Objects button.

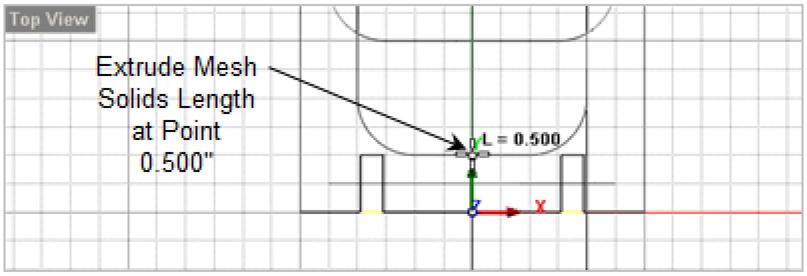


11. Now with all 4 circles drawn, all 4 should be selected automatically. If not select them now.

12. Now go to the Mesh Modeling tab and select the Extrude Mesh 

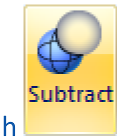
command.

13. For the extrude length move the cursor to the Top View and select the grid point at 0.500" in the positive Z direction or simple enter 0.5 in the command window Command and press <Enter>.



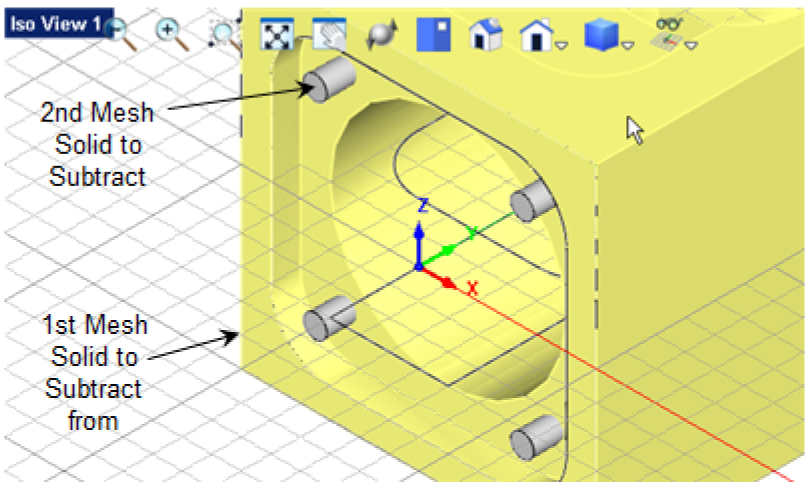
14. Now let's subtract the extrusions from the body to create the holes. Go to 

and from the 

tab select the Subtract Mesh 

command.

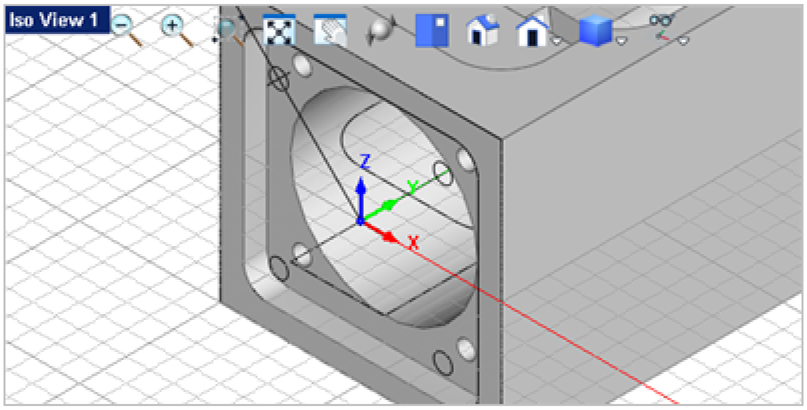
15. For the first mesh solid select the body.



16. For the second mesh solid select one of the extrusions to subtract it.

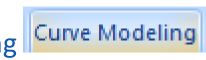
17. Press the <Enter> key to repeat the command. This time select the body and then the second extrusion.

18. Repeat the command two more time until all 4 extrusions are subtracted from the body as shown below.

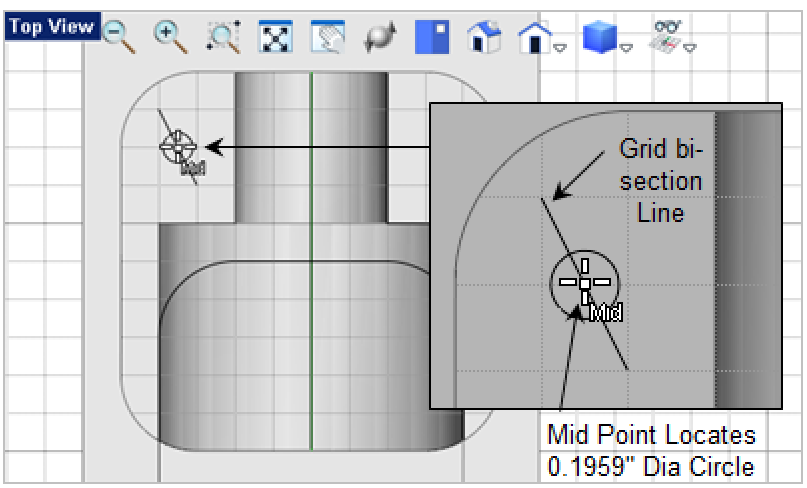


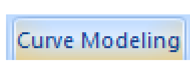
19. Now we have 2 additional holes of the same diameter located in the top access. Activate the .

20. Let's change the display mode for the Top View. Select the Display menu from the View Toolbar in the Top View and select Toggle Shaded + Edges.

21. To locate the first hole, go to the Curve Modeling  tab and select the Line  command.

22. Pick the two grid points shown below for the start and end points of the line. The line should bi-sect the two grid boxes from left to right as shown below.



23. Now from the Curve Modeling  tab select the Circle  command.

24. From the Stats Bar activate the Mid Point Object Snap  and deactivate all of the others.

25. For the center point of the circle select the mid point of the line you just created shown in the illustration above.

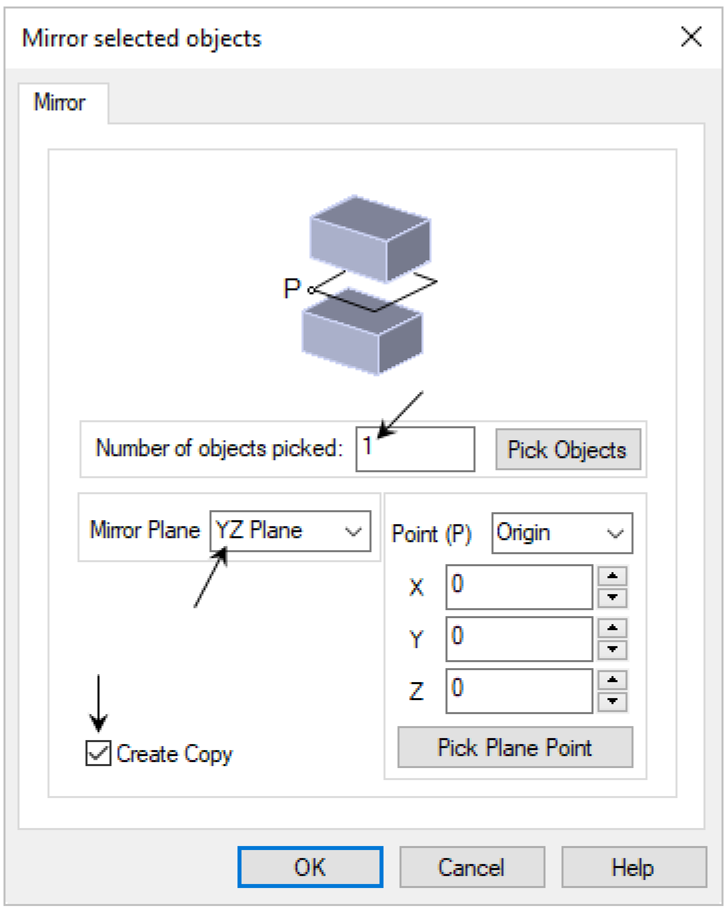
26. For the circle diameter enter 0.1959 in the Command Window 

and press <Enter>.

27. Now select the circle you just created and go to the 

tab and select the Mirror 

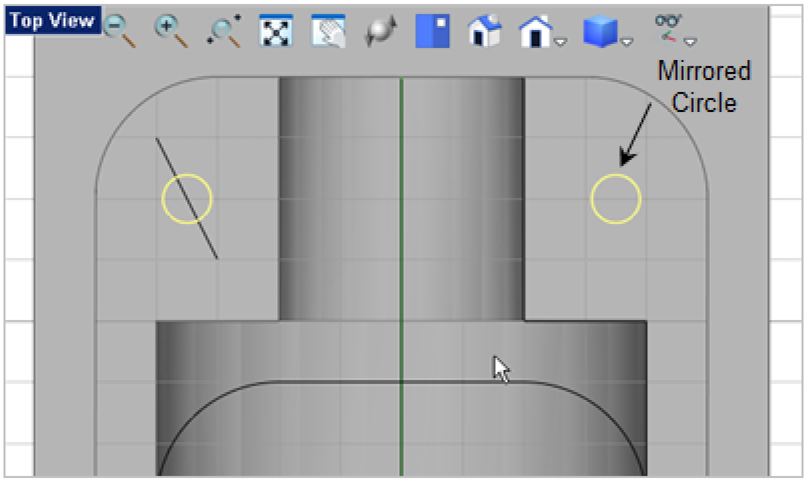
command.



28. From the Mirror select objects dialog the Number of selected objects should be 1, the Mirror Plane should be YZ Plane and the Create Copy box should be checked.

29. Now pick OK to mirror the circle.

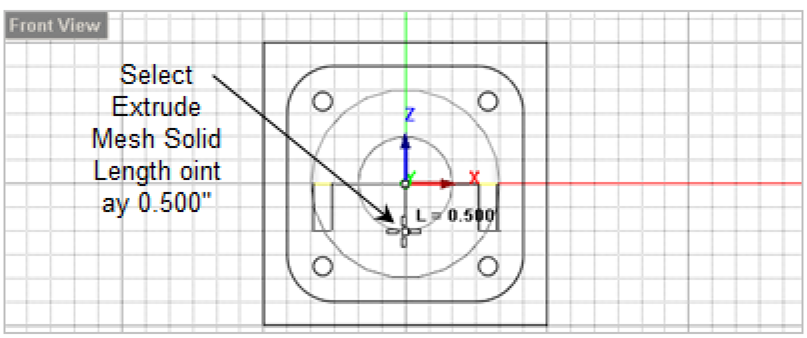
30. Both circles should now be selected. If not select them both now.



31. With the circles select go to the Mesh Modeling tab and select the Extrude Mesh Extrude command.

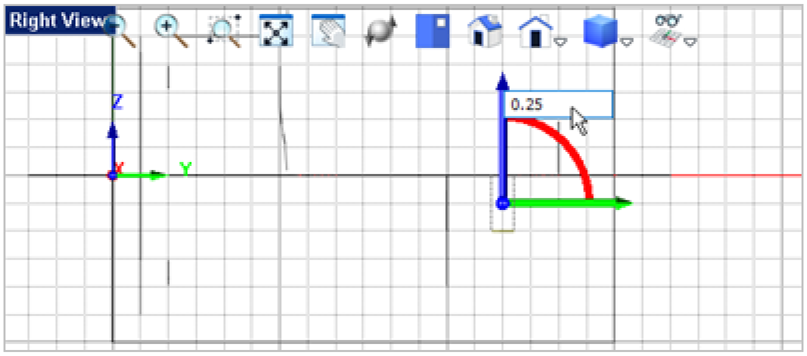
32. From the Status Bar activate the Grid Object Snap and deactivate the others.

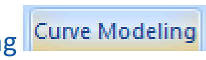
33. Move the cursor down to the Front View and select the grid point to set the extrusion length to 0.500" in the negative Z direction or simply enter 0.5 in the command window Command and press <Enter>.

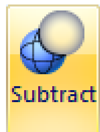


34. Now select the two extrusions and go to the Right View. Make sure the Graphic Manipulator is toggled On.

35. From the Graphic Manipulator, pick the Blue Z Axis arrow and then enter 0.25 in the input window and press <Enter>. The extrusions will translate up in the Z axis 0.25".



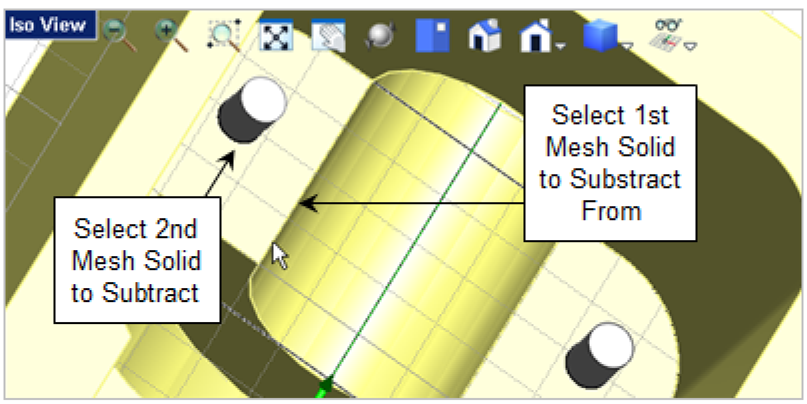
36. Now subtract them from the body. From the Curve Modeling 

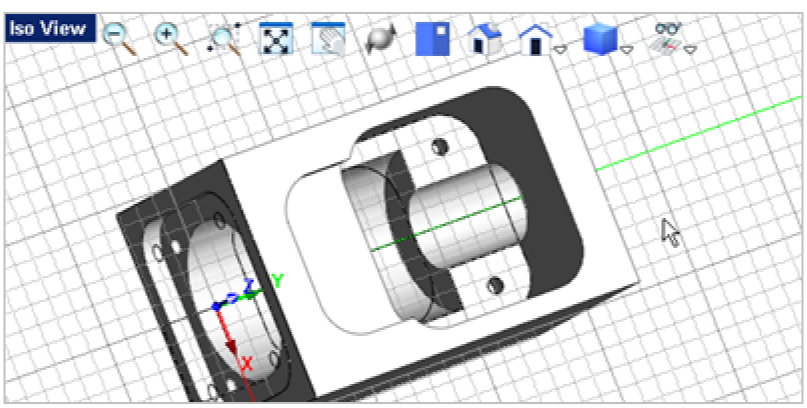
tab select the Subtract Mesh 

command.

37. For the first mesh solid select the body. For the second mesh solid select one of the extrusions.

38. Press <Enter> to repeat the command and subtract the second extrusion from the body.



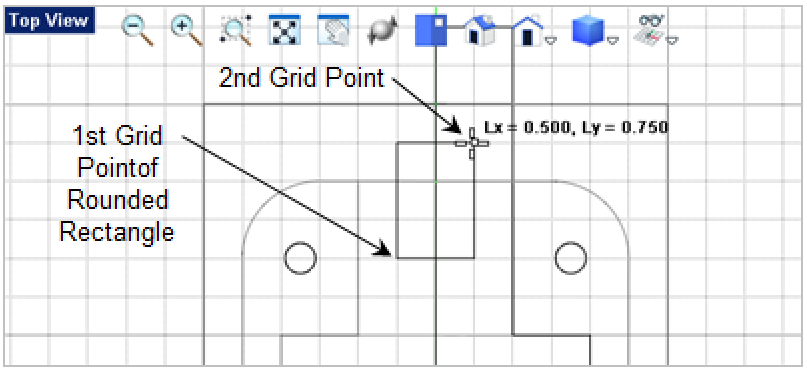


7.6.5 Modify the Top Access

Modify the Top Access

In this step we need to make a few modifications to the top access. The first is an extruded cut at the back of the access. The second is a revolved cut.

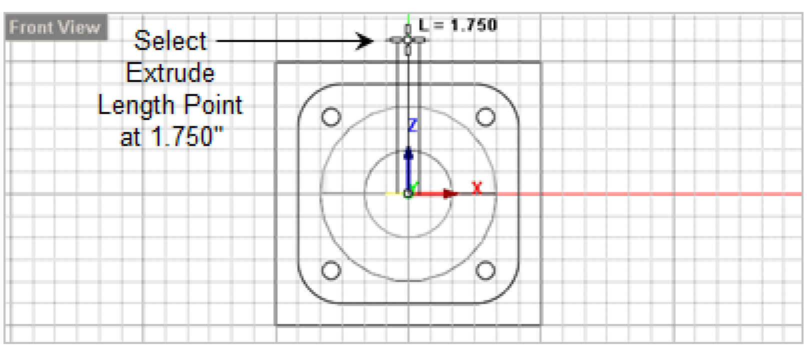
1. Activate the Top View.
2. From the Curve Modeling tab select the Rounded Rectangle command.
3. For the first and second corners of the rectangle select the grid points shown below.

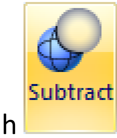


4. For the corner radius, enter 0.125 in the command window Command and press <Enter>.
5. Now select the rectangle and from the Mesh Modeling tab select the Extrude Mesh

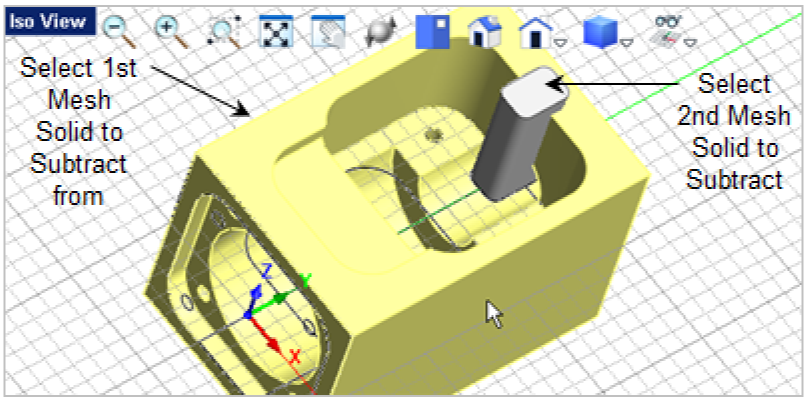
command.

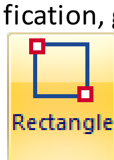
6. Move to the Front View and drag the extrude length to 1.750".



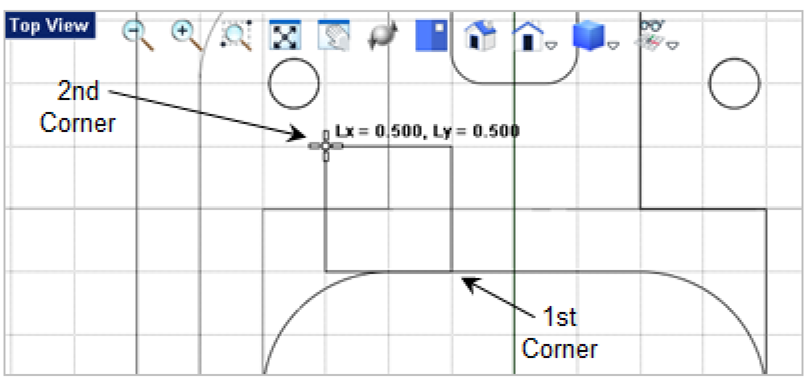
7. Now go to the  tab and select the Subtract Mesh  command again.

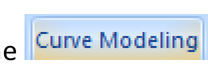
8. For the first mesh solid select the body. For the second mesh solid select the extrusion.



9. For the second modification, got to the  and from the  tab select the Rectangle  command.

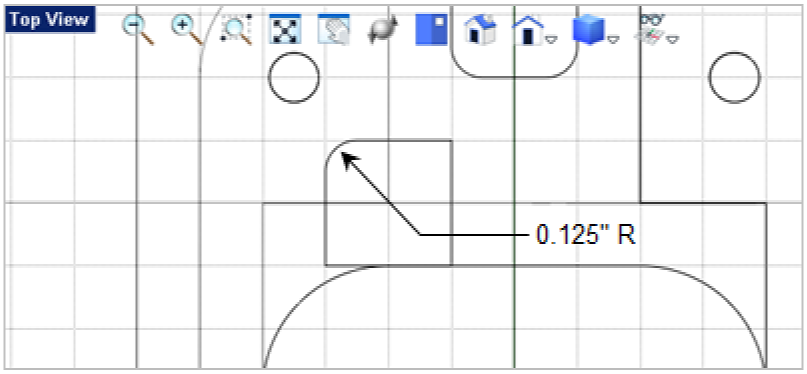
10. Select the two corner grid points shown below.

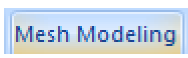


11. Now we need to fillet one of the corners. From the  tab select the Fillet Curve  command.

12. In the command prompt  enter 0.125 for the fillet radius and press <Enter>.

13. Now select the top and left side of the rectangle to fillet the upper left corner as shown below.

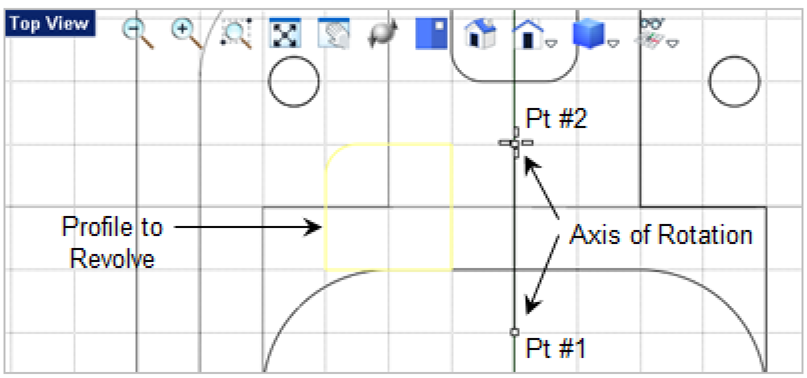


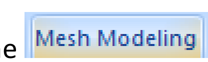
14. We need to revolve this, so select the closed rectangular profile, go to the 

tab and select the Revolve Mesh 

command.

15. For the axis of revolution, select the two grid points shown below.

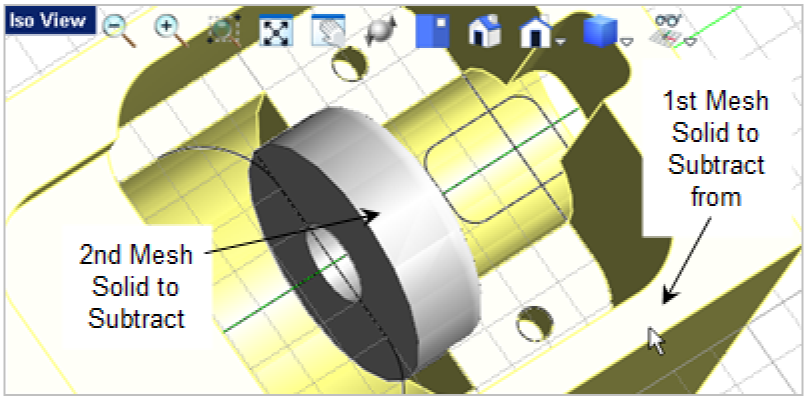


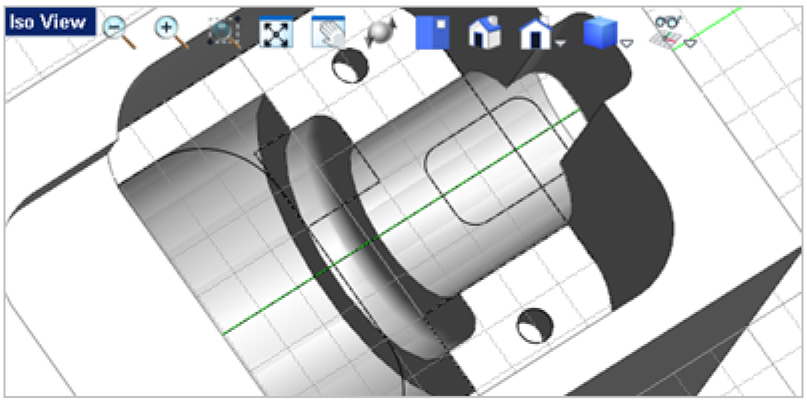
16. Now subtract the revolve from the body. From the 

tab select the Subtract Mesh 

command.

17. For the first mesh solid select the body and for the second select the revolve.



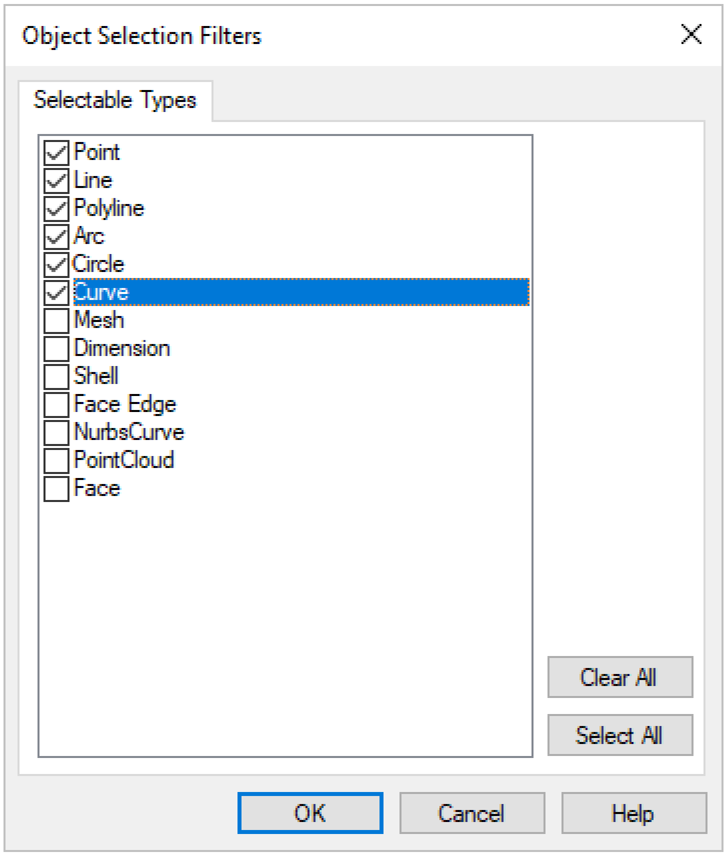


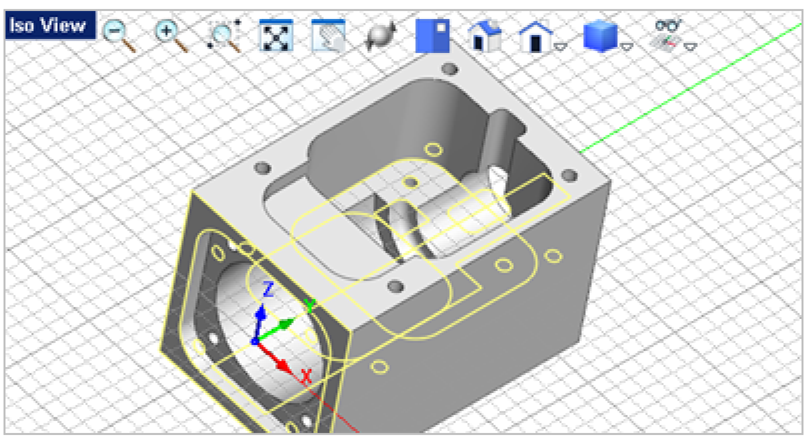
7.6.6 Create Section Curves

Create Section Curves

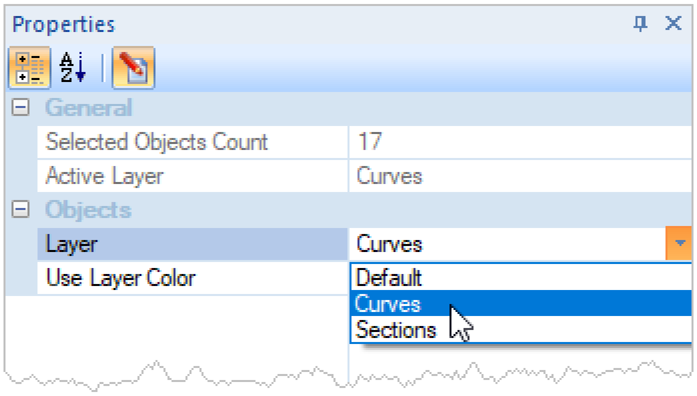
Your almost done!! In this step we want to create some section curves in the Top and Right side views. The section will appear on the XY Plane (for the Top view) and on the ZY Plane for the Right View.

1. First lets move all of our curves onto a separate layer and hide them. From the Modeling Aids tab, select the By Type ... command. This icon is located on the Selection pane of the Modeling Aids tab.
2. From the Object selection filters dialog, check the boxes next to Point, Line, Polyline, Arc, Circle and Curve and then pick OK. All of the curve related entities will now be selected.



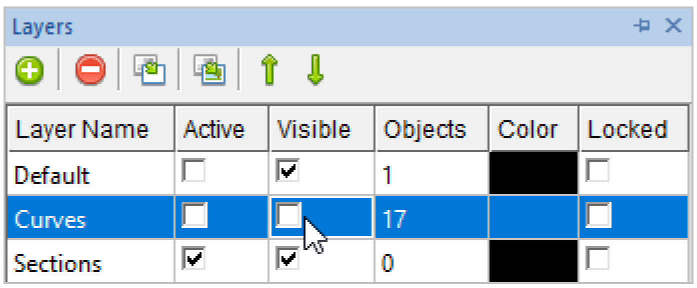


3. From the Status Bar select the Properties icon  to display the Properties Manager.



4. This dialogs reports that a total of 17 entitles are selected. In the Objects section drop down the menu for Layer and select Curves from the list of layers.

5. Now from the Status Bar select the Layer Manager icon .

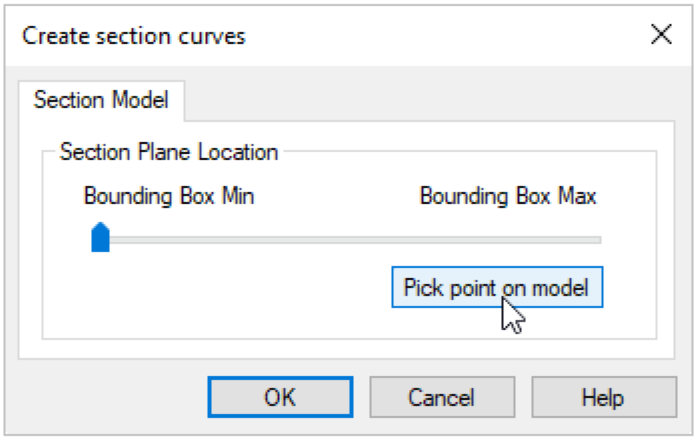


6. From the Layer Manager, under the Active column, check the box to make the layer named Sections the active layer.

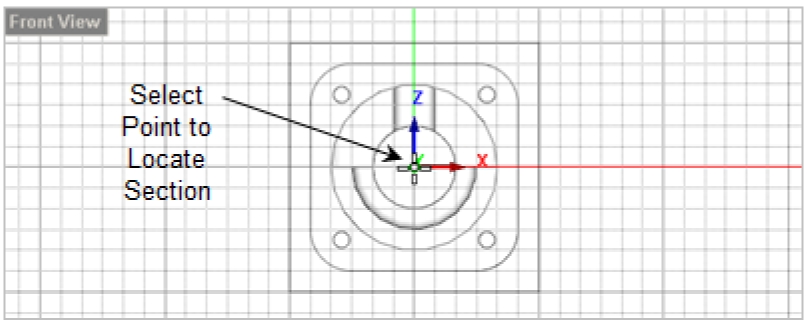
7. You can close the Properties and Layer managers by allowing them to auto-hide or pick the "X" icon to close them.

8. Now we're ready to create the section curves. First activate the Top View. Sections are generated for the currently active view.

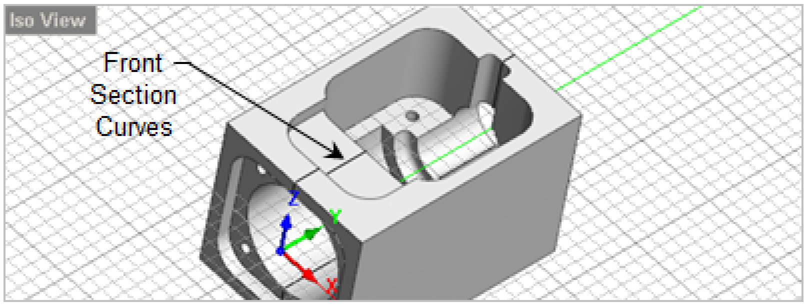
9. Go to the Curve Modeling tab and select the Section Curves command. This will display the Create section curves dialog.

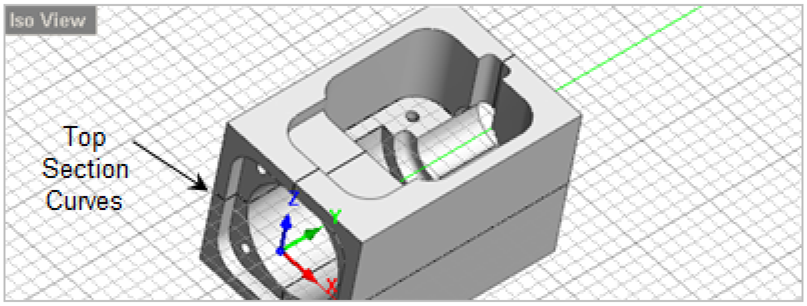


10. If you move the slider you will see a highlight of the section display in all views.
11. Now from the dialog select the Pick point on model button. The dialog minimizes allowing you to select a point. From the Front View, select the XY origin grid point to locate the section.

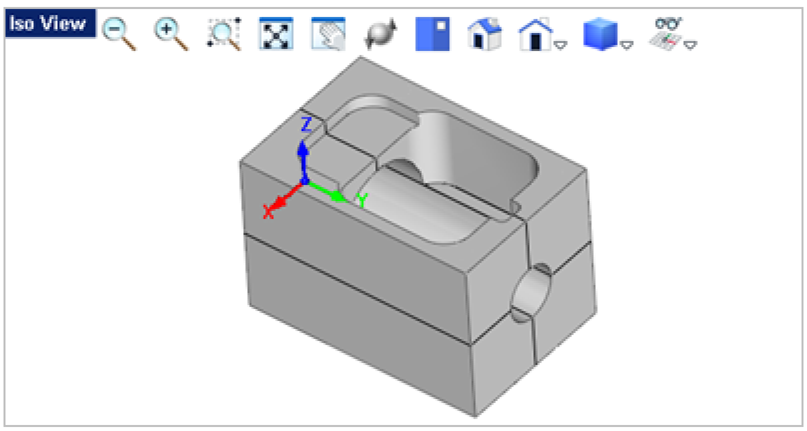


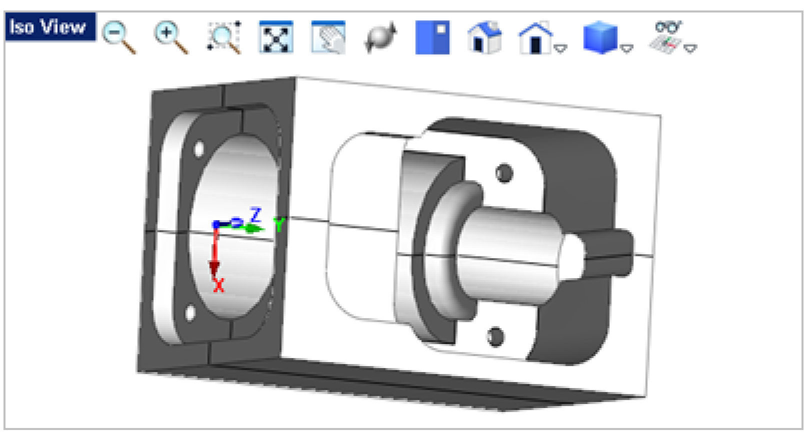
12. The dialog reappears and you can pick OK to close it.
13. Now activate the **Right View**.
14. Go to the Curve Modeling tab and select the  command again.
15. From the Create section curves dialog, select the Pick button and then select the same grid point, 0,0,0 origin in the **Front View**.
16. You will now notice that section curves were created in both the XY Plane and the YZ Plane.





Congratulations!!! You completed the exercise!!

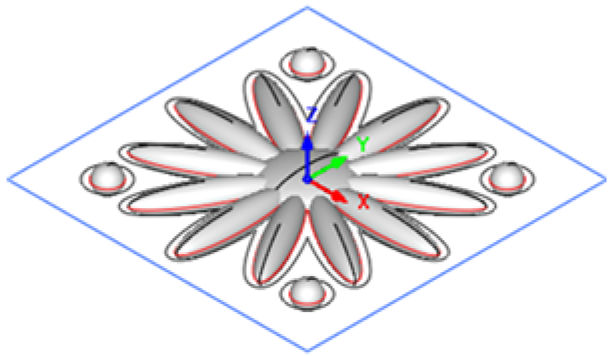
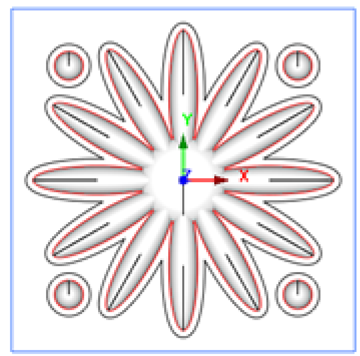




17. Save the file as Connector_Block_Completed.vcp.

7.7 #7: Model a Daisy Decor

In this fun exercise you will use solid (or mesh) spheres, non-uniform scaling, polar arrays, sectioning and offsetting to produce this simplistic decorative design. The offset curves and rectangle are used during machining as containment and stock definition.

Model a Daisy Decor

What you will learn:

In this exercise you will perform the following VisualCAD tasks:

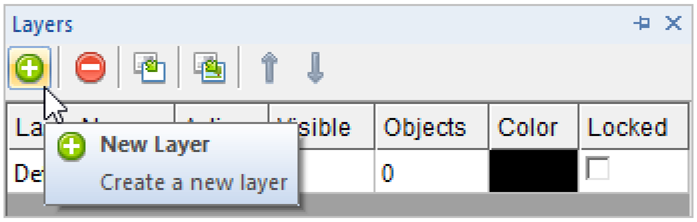
1. Create new Layers.
2. Model & Scale the Body.
3. Model & Scale the Pedal.
4. Ghosted Display Mode.
5. Polar Array the Pedal.
6. Additional Spheres Accents.
7. Create & Trim Section Curves.
8. Create Offset Curves.
9. Create the Stock Boundary.

7.7.1 Create new Layers

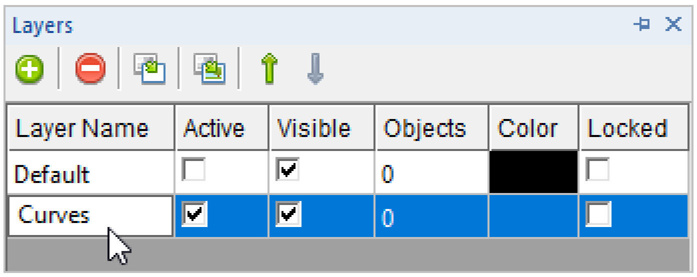
Create new Layers

In this step you will use the Layer Manager to create some new layers and then set the active layer.

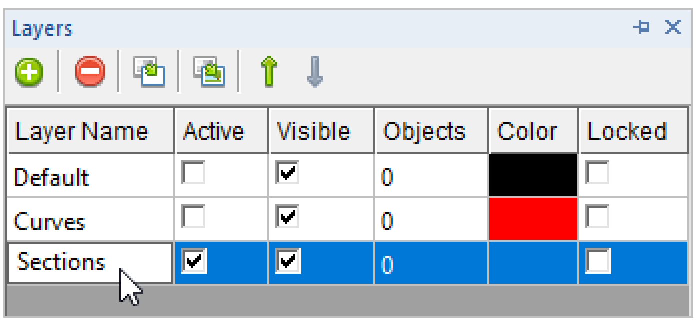
1. Display the Layer Manager. You can pick the Layer Manager icon located on the Status Bar.
2. From the Layer Manager select the Add Layer icon to add a new layer.



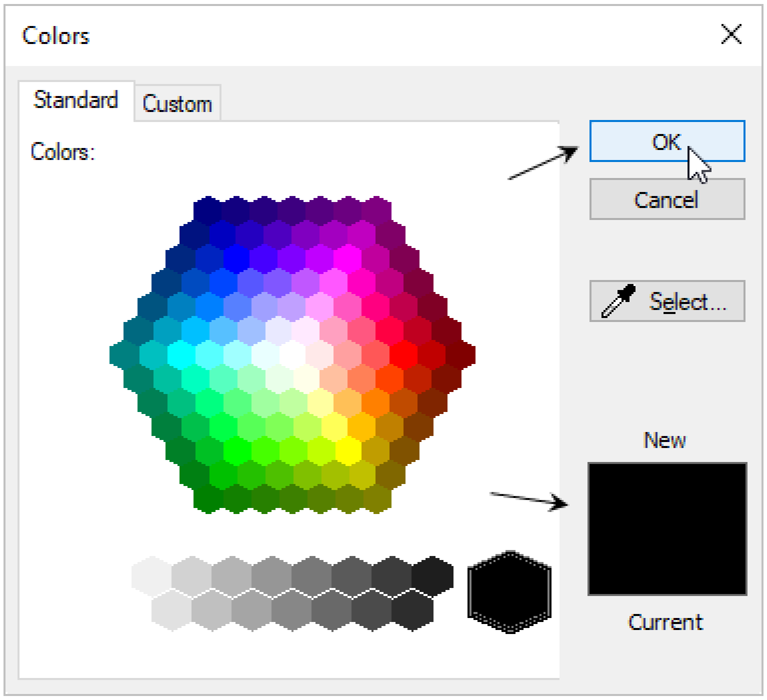
3. Now lets rename the new layer to Curves. First double-left-click in the Name field to activate it, and enter Curves as the new layer name.



4. Repeat the procedure to create a new layer, this time named Sections.

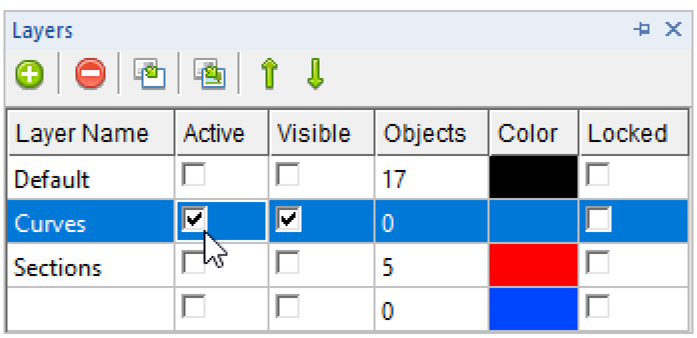


5. Now repeat the procedure again, this time named Stock.
6. Now let's change the color of a couple of the new layers. In the Color field left-click on the color box for the layer named Curves. This will display the Colors dialog.
7. Accept the default color Black and then pick OK to close the dialog.

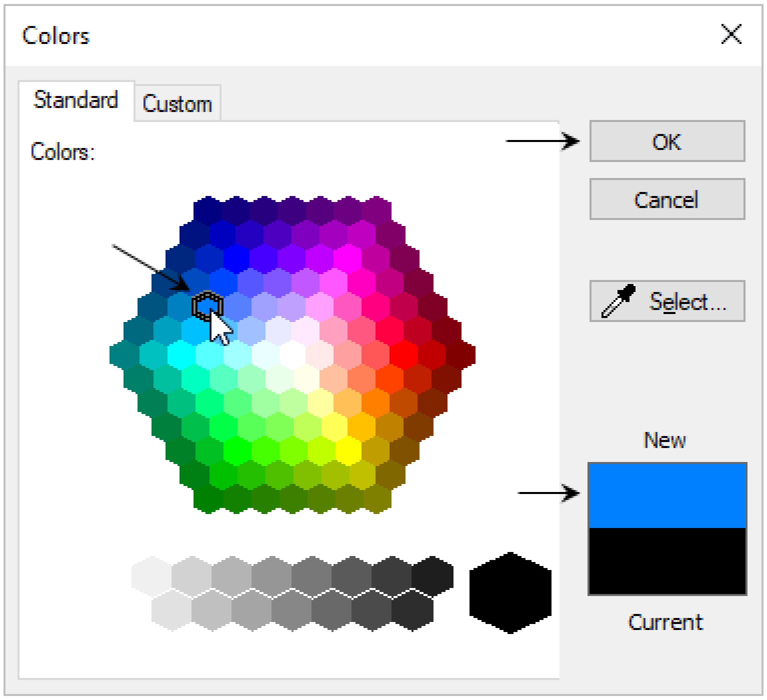


8. If the Layer Manager keeps Auto-Hiding to the left side of the display, select the Auto Hide pin icon at the top of the Layer Manager. You can also see that the layer named Curves was changed to color black.

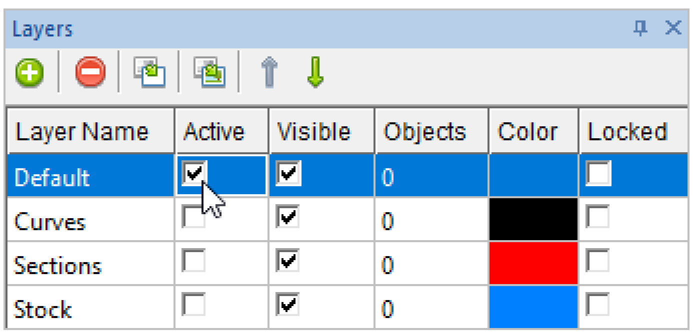
9. Now let's change the color of another layer. In the Color field left-click on the color box for the layer named Stock. Again, this will display the Colors dialog.



10. This time select the Blue color and then pick OK to close the dialog. You can use different color if you wish. If you have Auto Hide pinned, the Layer Manager will stay displayed.



11. This time let's set the Active Layer. In the Active column, check the box for the layer named Default. and then unpin the Layer Manager to close it.



Layer Name	Active	Visible	Objects	Color	Locked
Default	☑	☑	0		☐
Curves	☐	☑	0		☐
Sections	☐	☑	0		☐
Stock	☐	☑	0		☐

7.7.2 Model & Scale the Body

Model & Scale the Body

In this step you will model the center body using a solid sphere and non-uniform scaling. Note: You can replace the Solid commands with Mesh commands if desired.

1. Activate the Top View  tab by selecting anywhere in the viewport.

2. Select the Solid Modeling  tab from the top Ribbon Bar.

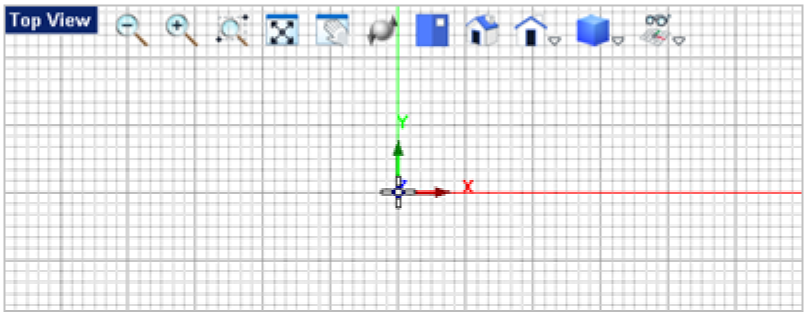
3. Now select the Sphere 

command.

4. In the Status Bar make sure the Grid Snap is On .

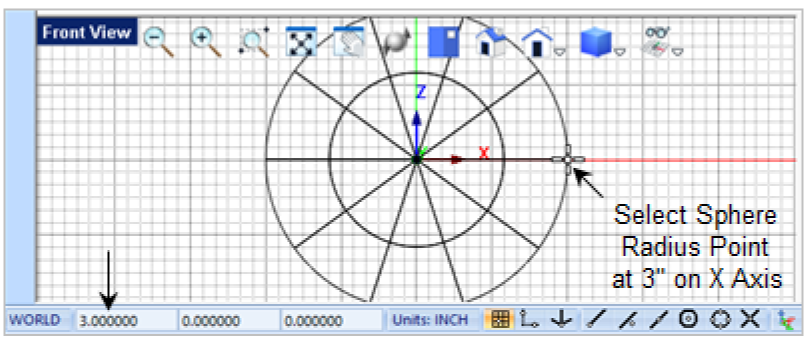
5. For the center point select the grid point at (0,0,0) or just enter the coordinates 0,0,0 in command input bar 

.

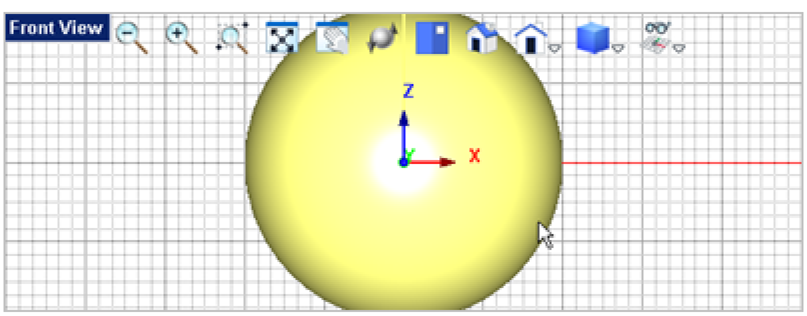


6. For the sphere radius, enter 3,0,0 

or select the corresponding grid point and press <Enter>.

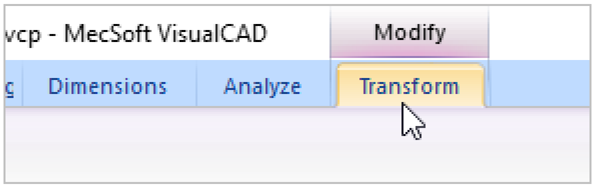


7. Select the solid sphere you just created.



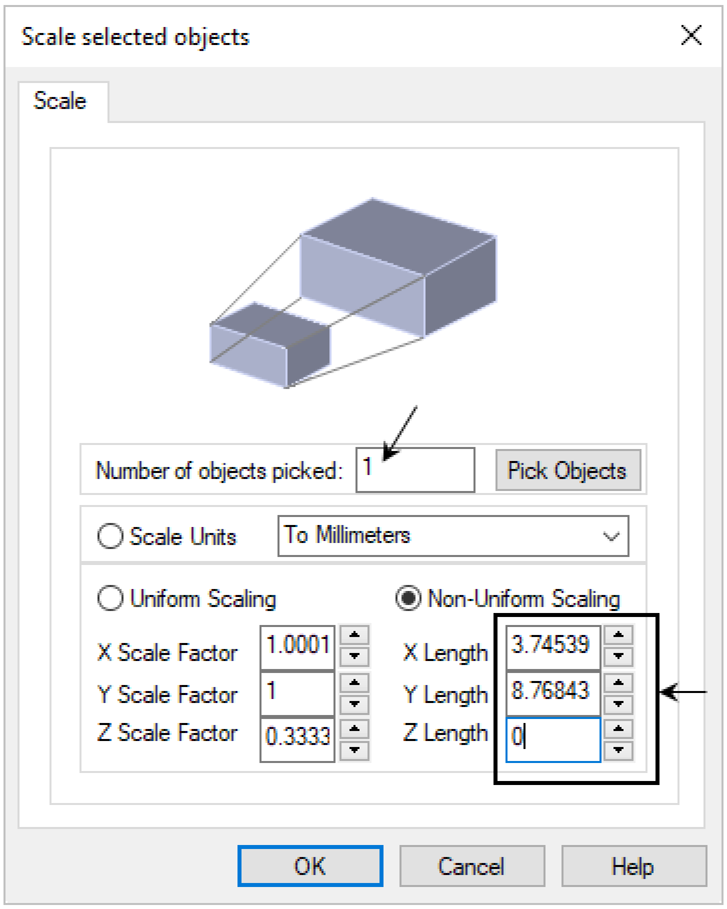
8. You see the Modify / Transform tab 

appear on the right end of the Ribbon Bar. It ONLY appears when an entity is selected. Select the Transform tab.

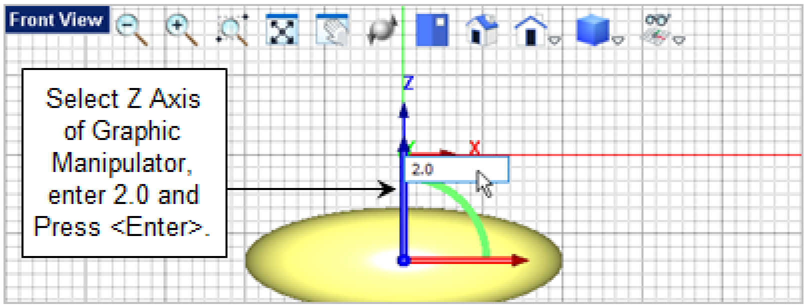


9. Select the Scale  command.

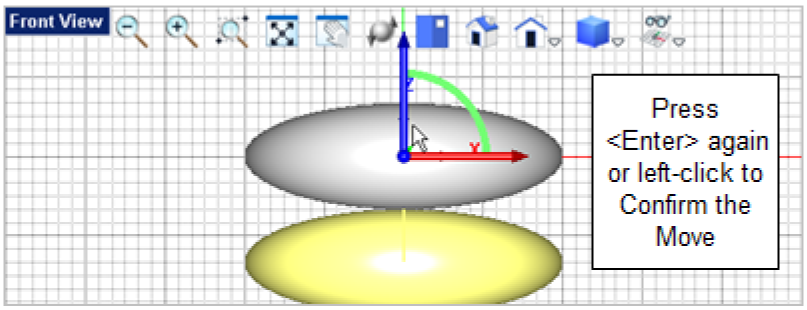
10. From the Scale Selected Objects dialog, select the Non-Uniform Scaling option.



11. Set the X Length to 6, Y Length to 6 and the Z Length to 2.
12. Pick OK and the sphere is scaled as shown below.
13. Now press <Enter> to accept the modification.
14. Let's move the scaled sphere up to the origin. First make sure the Graphic Manipulator is On. The icon is located on the Status Bar .
15. Now select the sphere.
16. Select the Blue Z arrow to display the input field, enter 2.0 and press <Enter>. The sphere will move up in the positive Z Axis to the 0,0,0 origin as shown below.



17. Press <Enter> again to accept the move.

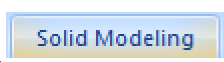


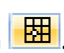
7.7.3 Model & Scale the Pedal

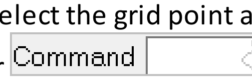
Model & Scale the outer Pedal

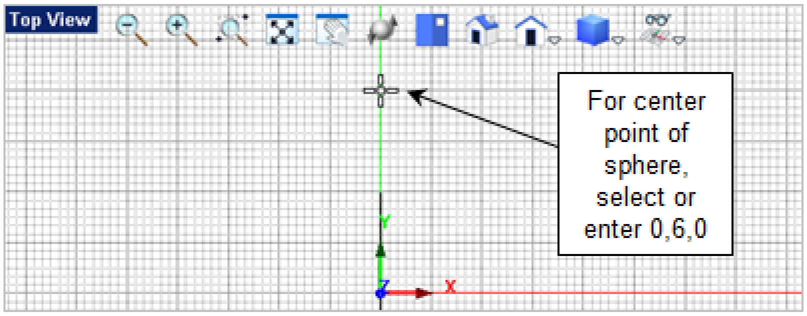
In this step you will model and scale one of the outer pedals, again using a solid sphere and non-uniform scaling. Note: You can replace the Solid commands with Mesh commands if desired.

1. Activate the Top View  tab by selecting anywhere in the viewport.

2. Form the Solid Modeling  tab select the Sphere  command.

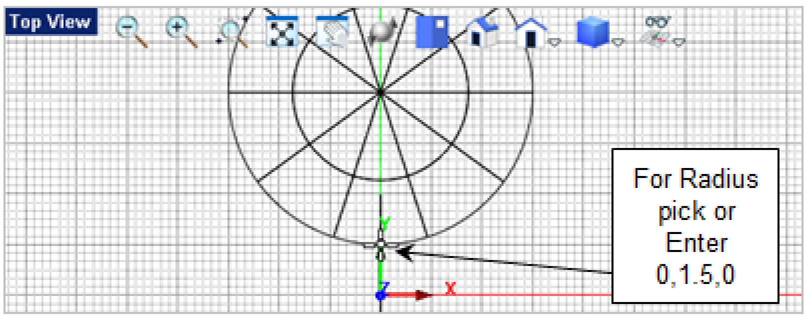
3. In the Status Bar make sure the Grid Snap is On .

4. For the center point select the grid point at (0,6,0) or just enter the coordinates 0,6,0 in command input bar  and press <Enter>.

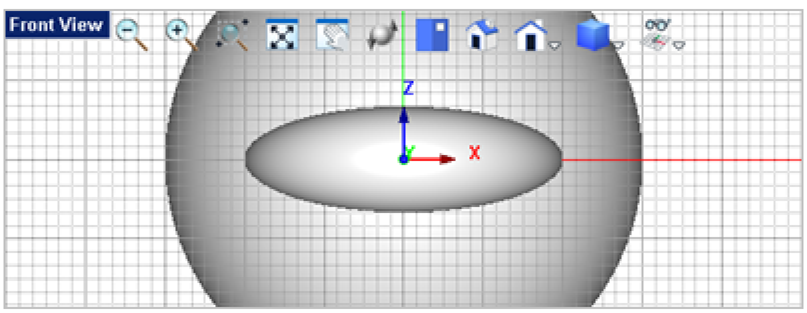


5. For the sphere radius, enter 4.5 in the command input window 

and press <Enter> or select the grid point located at 0,1.5,0.

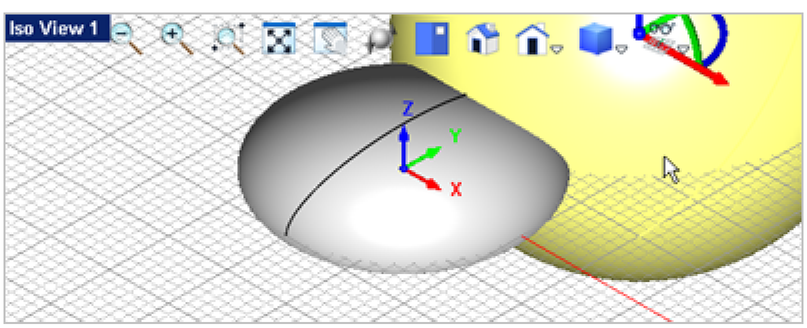


Here is what you part should look like from the Front View:

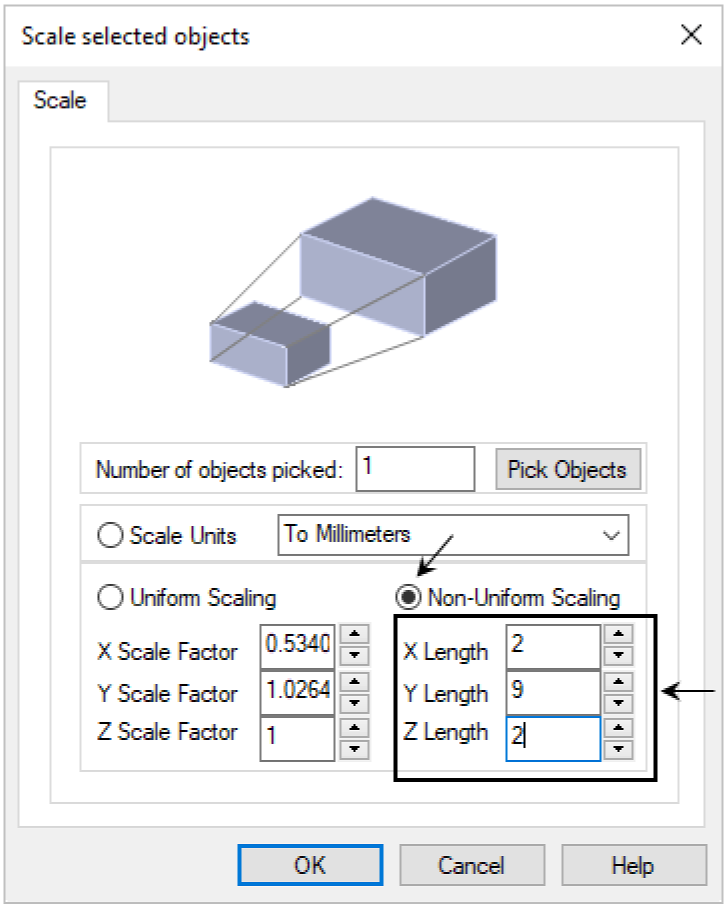


6. From Iso View 1, 

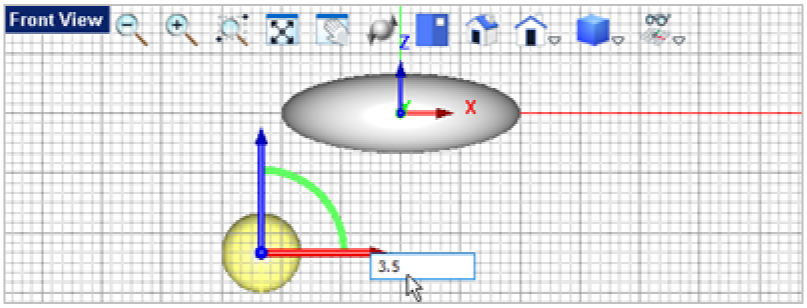
select the solid sphere you just created.



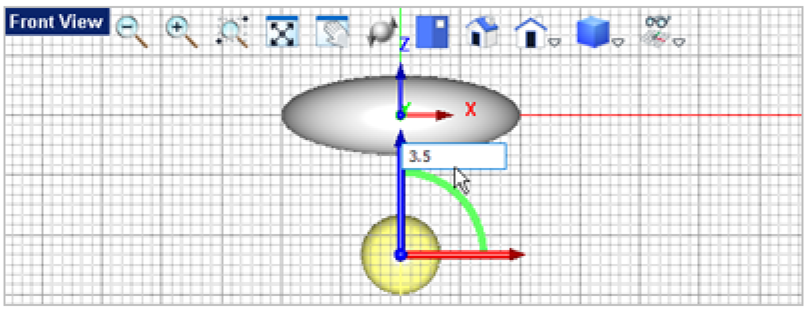
7. Select the Transform Modify Transform tab from the ribbon bar.
8. Select the Scale Scale ... command.
9. From the Scale Selected Objects dialog, select the Non-Uniform Scaling option.



10. Set the X Length to 2, Y Length to 9 and the Z Length to 2.
11. Pick OK and the sphere is scaled as shown below.
12. Now press the <Enter> key to accept the modification.
13. Let's move the scaled sphere into position along the Y Axis. First make sure the Graphic Manipulator is On. The icon is located on the Status Bar.
14. From the Front View Front View, select the Red X arrow to display the input field, enter 3.5 and press <Enter>.



15. Press <Enter> again to confirm the move and the sphere will move up to the Z 0 location as shown below.



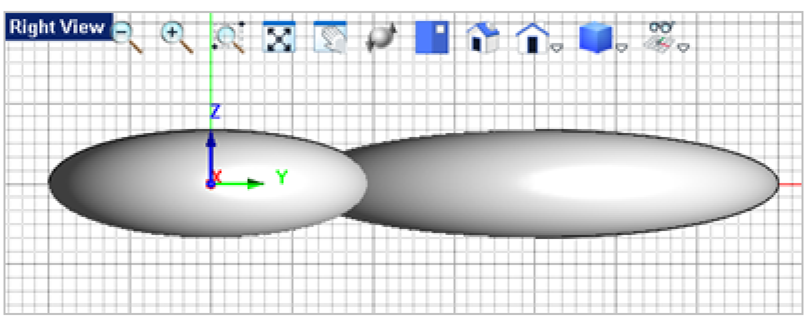
16. Now select the sphere again.

17. Select the Blue Z arrow to display the input field, enter 3.5 

and press <Enter>.

18. Press <Enter> again to confirm the move and the sphere will move to the X 0 location. If you activate the Right View 

your part should look like the image below.



7.7.4 Ghosted Display Mode

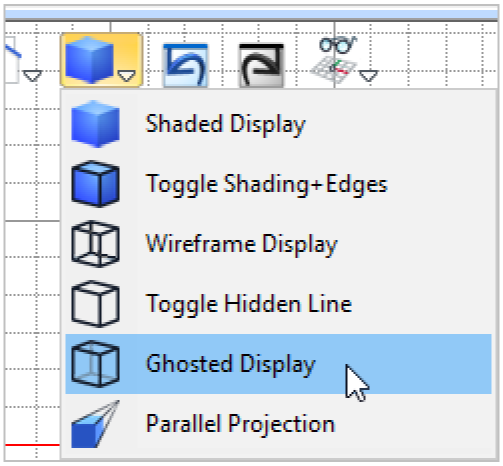
Change the Top View to Ghosted Display

For spheres drawn from the top view, it is best viewed using the ghosted display.

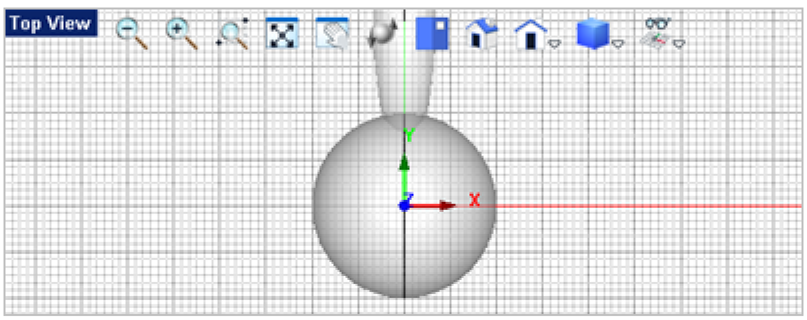
1. Activate the Top View 

tab by selecting anywhere in the viewport.

2. Select the View  menu from the View Toolbar and then select Ghosted Display. Note: The View Toolbar will only display in the active viewport.



The display should now look like this:

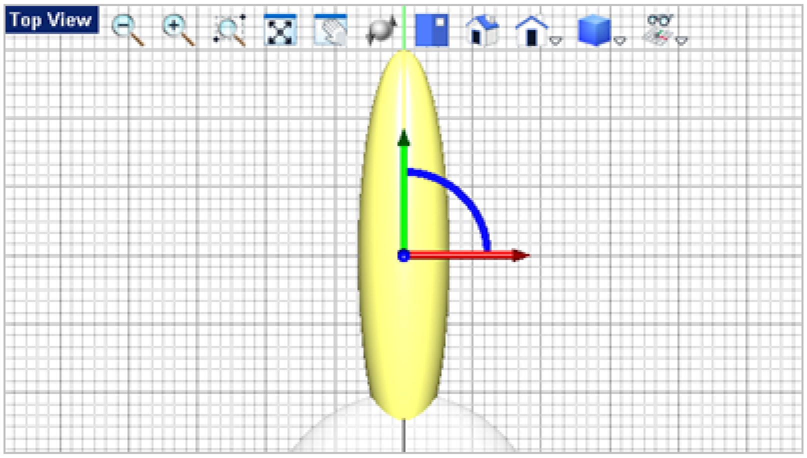


7.7.5 Polar Array the Pedal

Create the Polar Array

In this step you will create a polar array of the pedal sphere we just created.

1. Activate the Top View  tab by selecting anywhere in the viewport.
2. Select the solid sphere pedal you just created in the previous step.

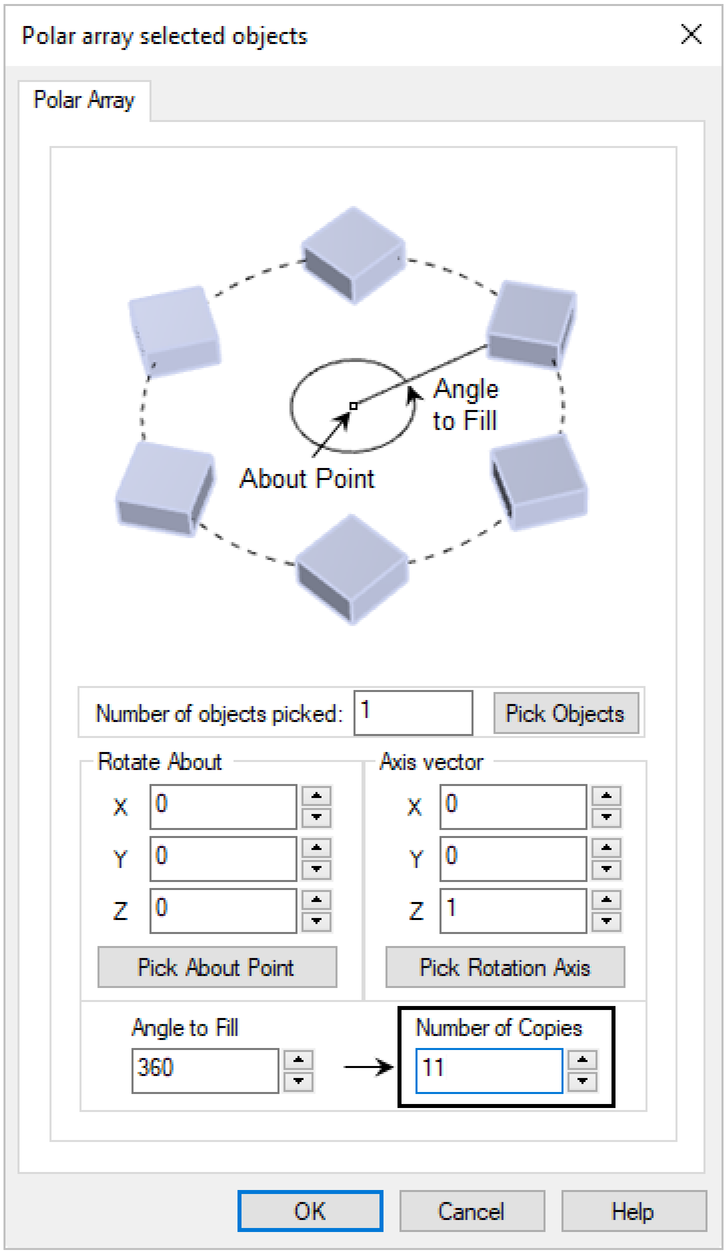


3. Select the Transform 

tab from the ribbon bar.

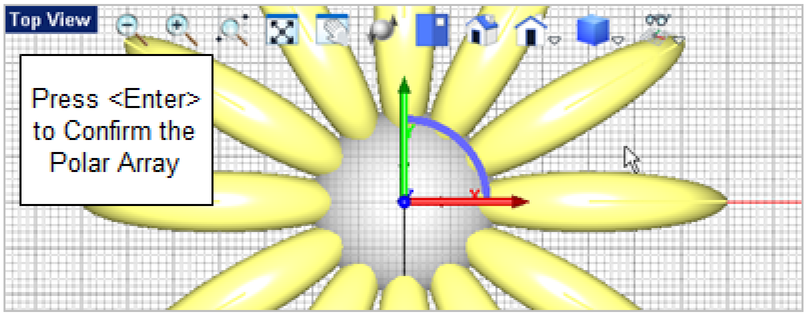
4. Select the Polar Array 

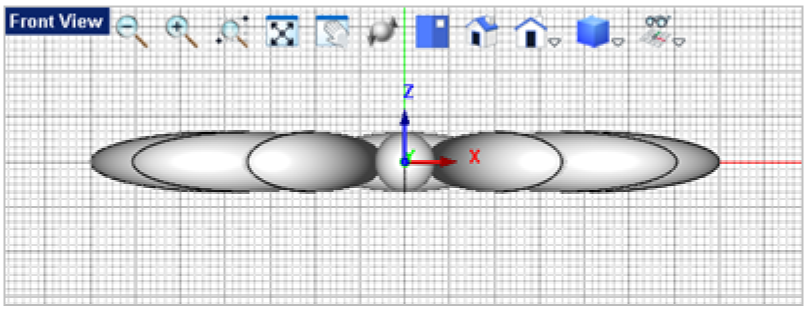
command. The Polar array selected objects dialog will display.

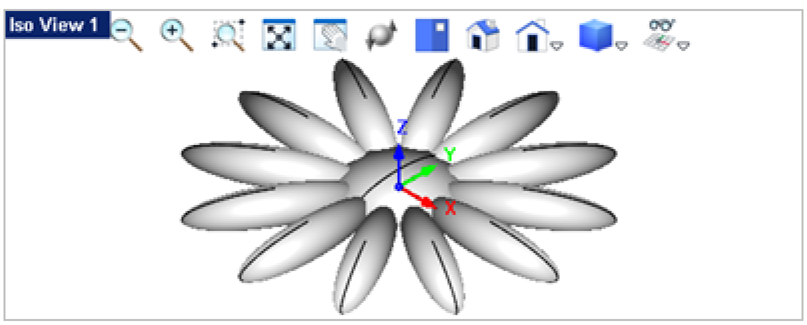


5. In the Number of Objects field, enter 11.
6. The remaining options in the dialog should use the default values shown in the dialog above.
7. Pick OK from the dialog to create and preview the polar array.

8. Press <Enter> to accept the polar array. Your drawing should look like the images below. If not, select the Undo  command and go back and repeat the previous steps.





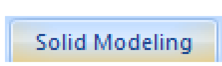


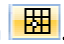
7.7.6 Additional Spheres

Create the 4 additional Spheres

In this step you will create the four spheres located in the outer corners of the design. Note: You can replace the Solid commands with Mesh commands if desired.

1. Activate the Top View  tab by selecting anywhere in the viewport.

2. Form the Solid Modeling  tab select the Sphere  command.

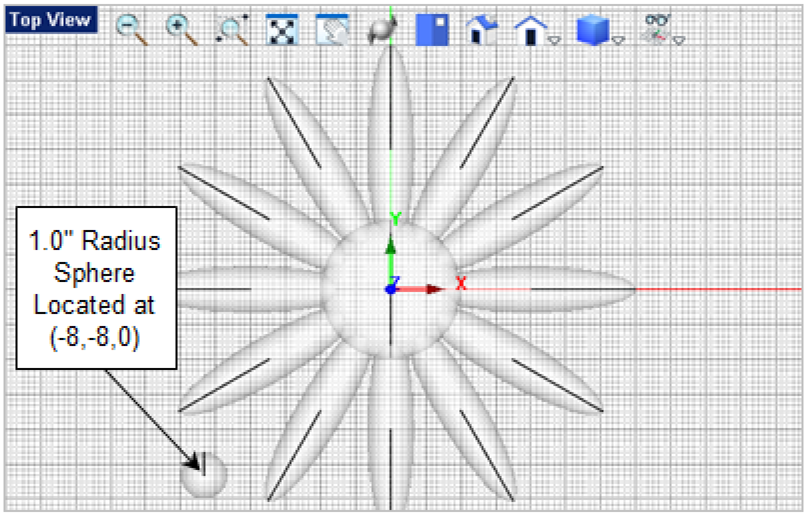
3. In the Status Bar make sure the Grid Snap is On .

4. For the center point select the grid point at (-8,-8,0) or just enter the coordinates -8,-8,0 in command input bar 

and press <Enter>.

5. For the sphere radius, enter 1.0 in the command input bar 

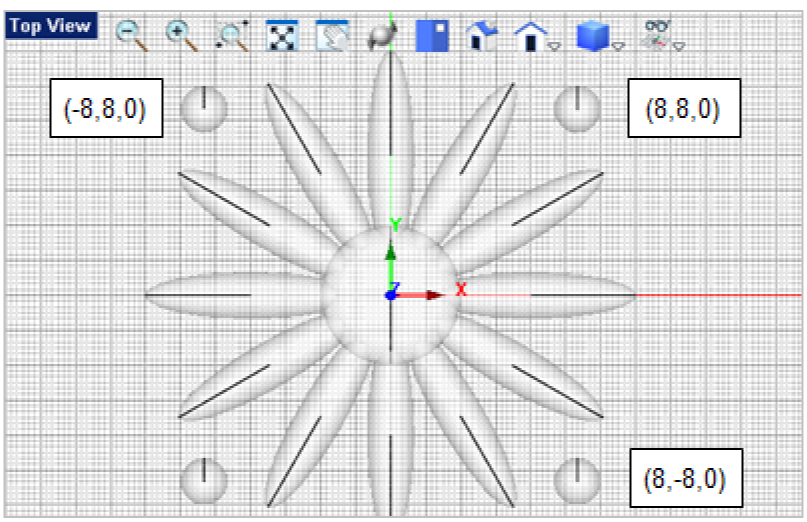
and press <Enter> or select the grid point located at -8,-8,0 shown below.



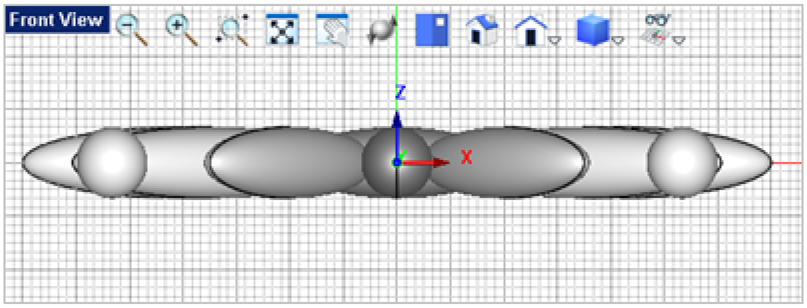
6. Press <Enter> to repeat the Solid Sphere 

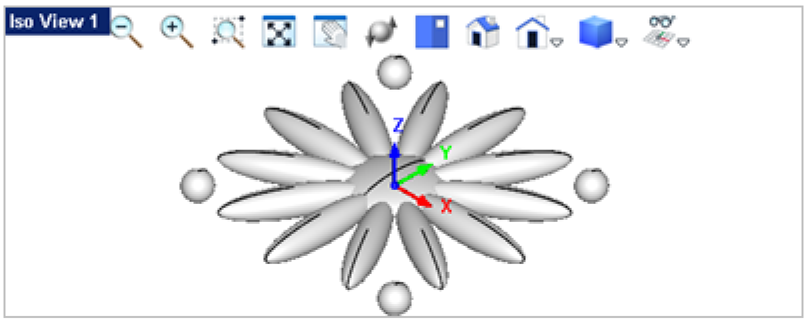
command.

7. Create the three additional spheres located at the coordinates shown in the illustration below.



Your drawing should look like this:





7.7.7 Create & Trim Section Curves

Create Section Curves

In this step we will create section curves that lie on the XY plane. Section curves are useful for creating offset boundaries which in turn can be used for toolpath containment.

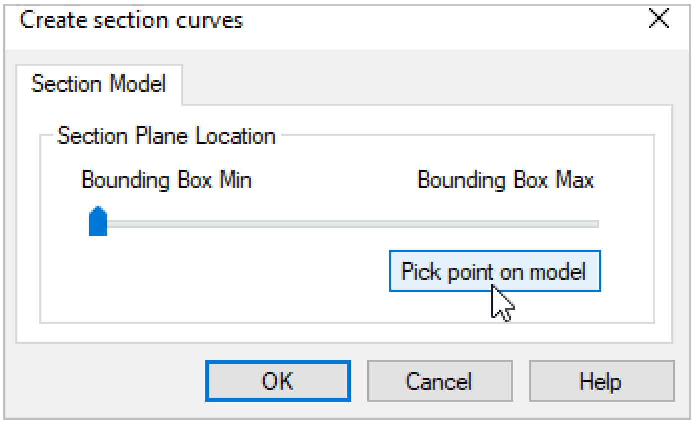
1. Open the Layer Manager and set the Active Layer to Sections. Refer to the previous steps for displaying and using the Layer Manager.

Layers

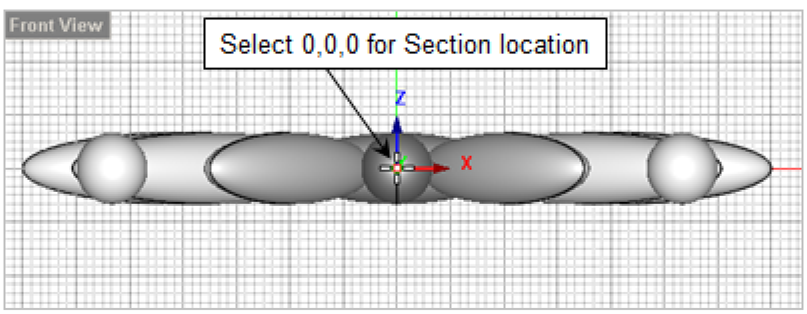
Layer Name	Active	Visible	Objects	Color	Locked
Default	☐	☑	0		☐
Curves	☐	☑	0		☐
Sections	☑	☑	0		☐
Stock	☐	☑	0		☐

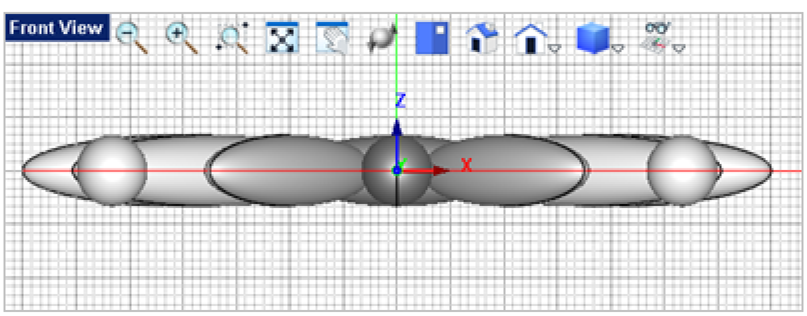
2. Section curve are created in the plane of the active layer. We want the section curves to be generated on the XY plane so activate the Top View **Top View** tab by selecting anywhere in the viewport.

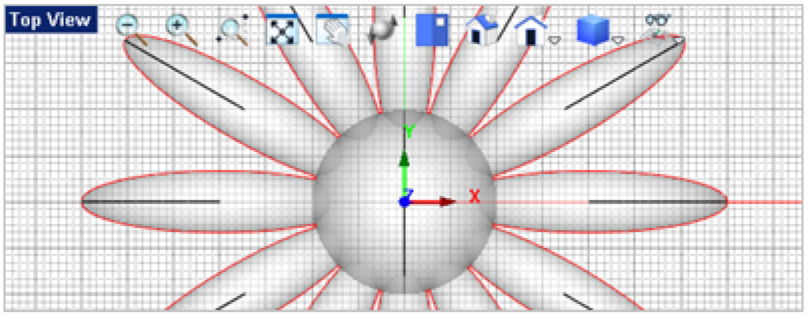
3. Form the Curve Modeling **Curve Modeling** tab select the Section Curve command. The Create section curves dialog will display.

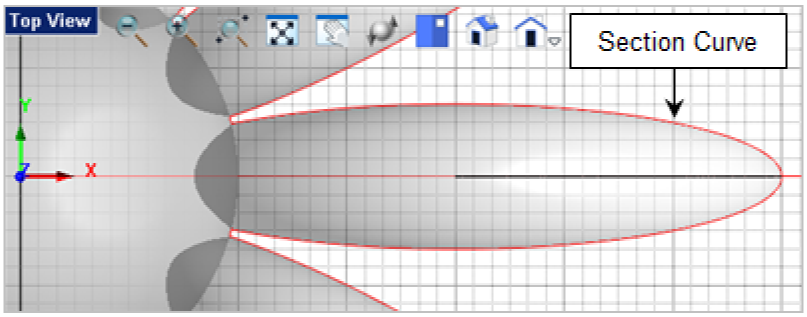


4. If you move the slider you will see dynamically where the section curves lie. In this case, with the Top View active, moving the slider will display section curves dynamically in the XY plane and in the Z Axis.
5. We want to precisely locate the section plane so select the Pick point on model button from the dialog.
6. Then in the Front View pick the Z 0 grid point or simply enter 0,0,0 in the command input bar and press <Enter>. The section curves will be created and displayed in all four viewports.

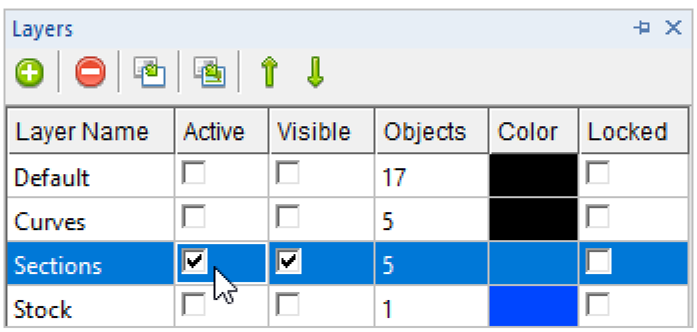






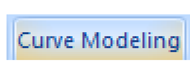


7. Open the Layer Manager and set the Active Layer to Sections. Refer to the previous steps for displaying and using the Layer Manager .

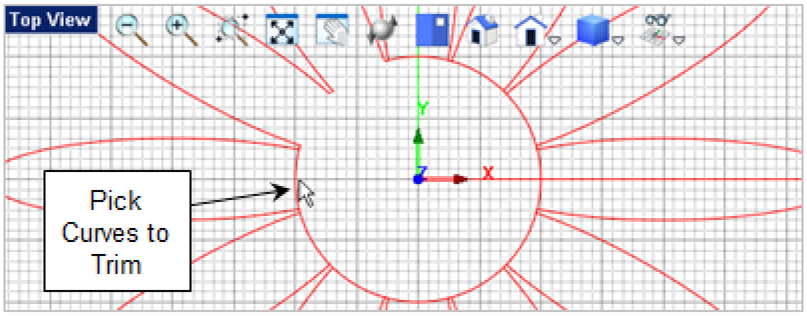


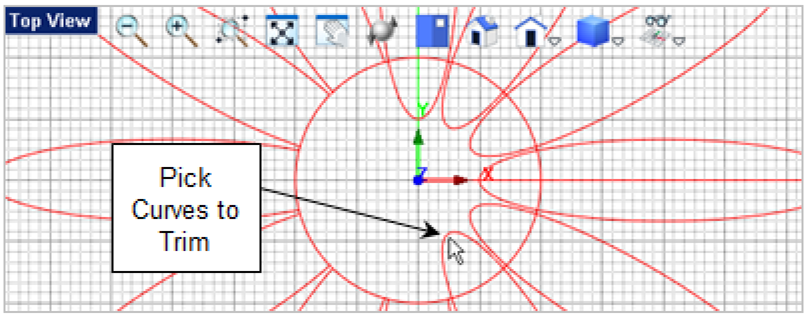
In the following steps we will trim our section curves and merge them.

8. From the Top View 
, scroll to zoom in on the center where the section curves from the body and the pedals overlap each other.

9. From the Curve Modeling 
 tab, select the Trim 
 command.

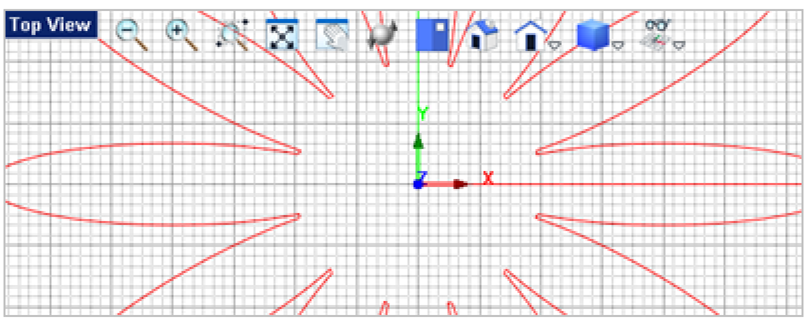
10. Pick on the curve where you want it to be trimmed away.



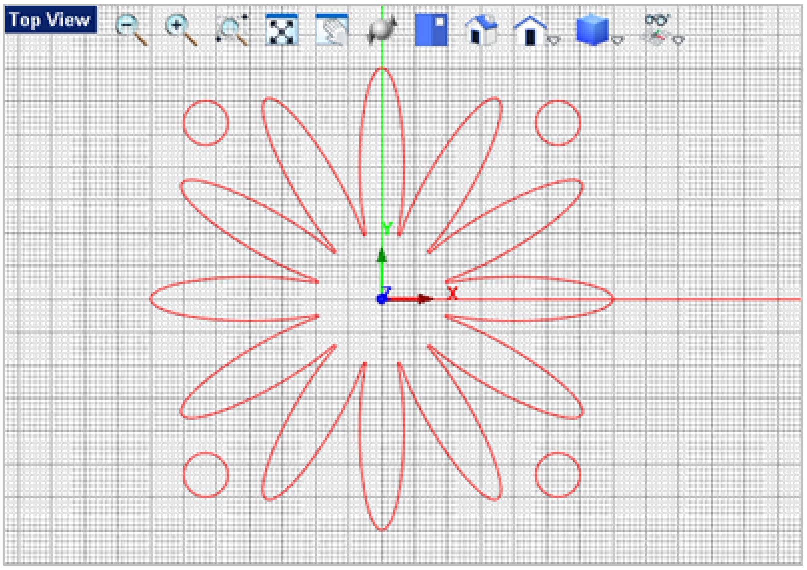


11. Press the <Enter> to repeat the Trim  command and then pick another location to trim.

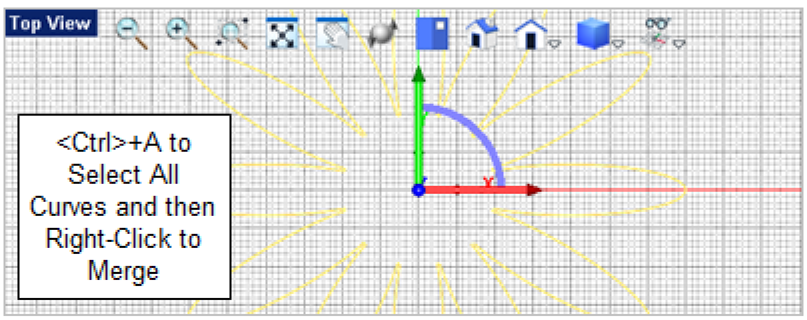
12. Repeat this procedure until all of the curves are trimmed as shown below.

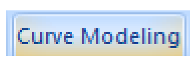
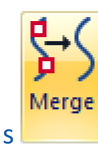


13. Now scroll to zoom out to see all of the curves.



14. Press <Ctrl+A> to select all of the curves.



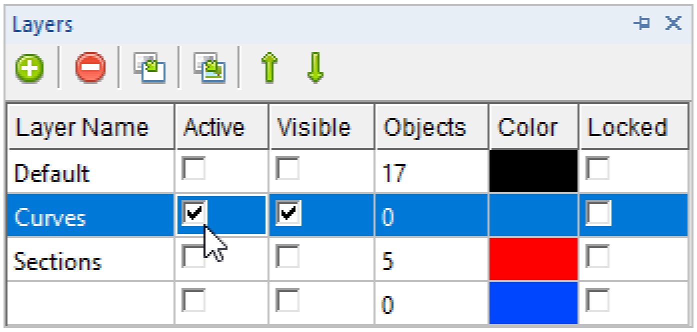
15. Now from the Curve Modeling  tab select the Merge Curves  command. This will merge all of the curves into one curve. This makes it easier to work with. Only curves that are joined end to end will be merged. Separate curves such as those created from the section of the outer spheres will remain separate curves.

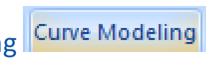
7.7.8 Create Offset Curves

Create Offset Curves

1. In the following steps you will create Offset curves from your Section Curves . Creating offset curves will also aid you during machining. For example, if you do not want to rough the entire stock, you can use section, offset or any other curve boundaries to limit/contain your toolpaths.

2. Open the Layer Manager  and set the Active Layer to Curves and then make all other layers hidden. Refer to the previous steps for displaying and using the Layer Manager.



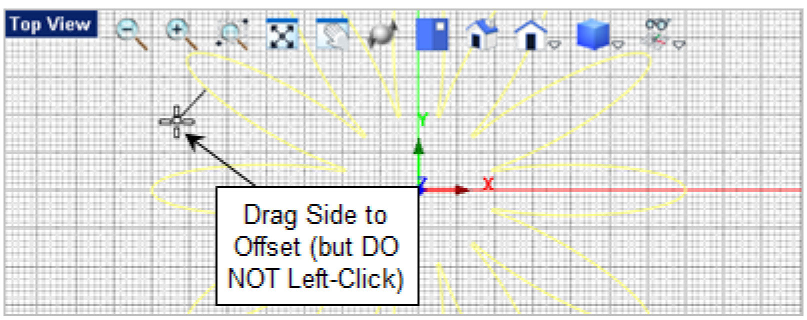
3. Now, select the main section curve, then go to the Curve Modeling 

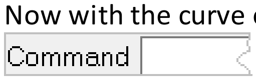
tab and select the Offset 

command.

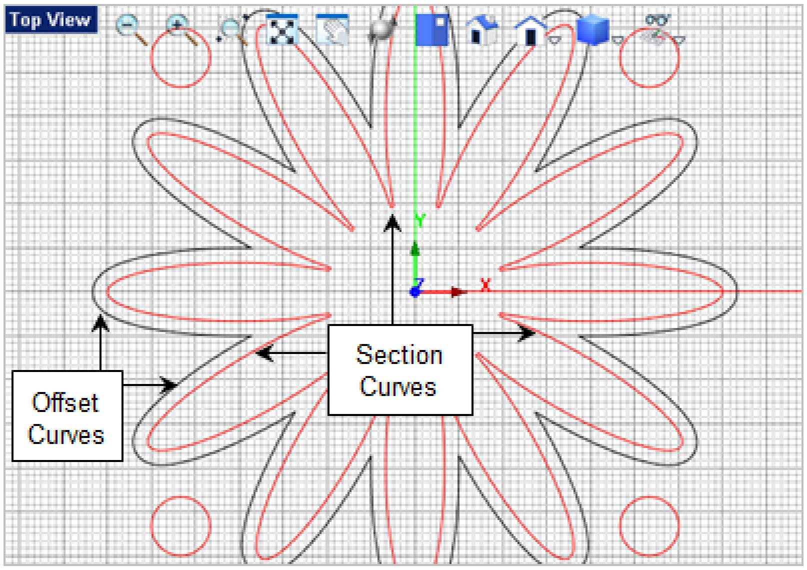
4. From the Top View 

move the cursor towards the outside of the curve so that you see the offset direction indicator point toward the outside. DO NOT pick a point!



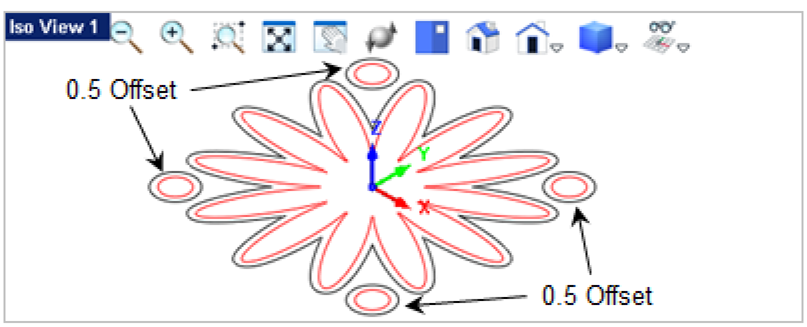
5. Now with the curve offset direction displayed enter 0.5 in the command input bar 

and press the <Enter> key. The curve will be offset toward the outside by 0.5 inches.



6. Press the <Enter> key to repeat the Offset  command. This time select the section curves from one of the outer spheres, drag the cursor to indicate the offset direction and press <Enter> again. The offset value will default to the last value entered.

7. Repeat the  command to create offset curves for the other 3 remaining section curves.

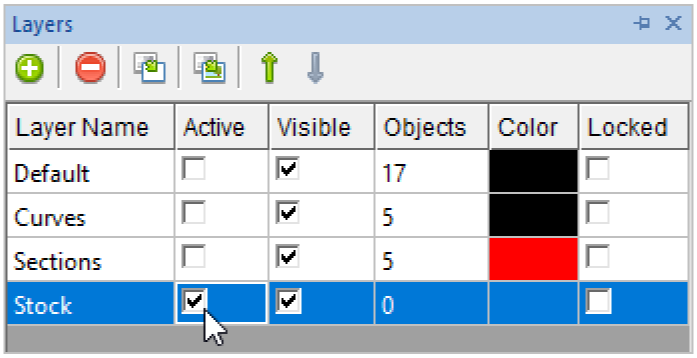


7.7.9 Create the Stock Boundary

Create the Stock Boundary Rectangle

In this step you will create a 24" x 24" rectangle around the part. This can be used later during machining to help define the stock dimensions.

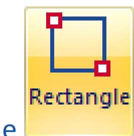
1. Open the Layer Manager  and set the Active Layer to Stock. Refer to the previous steps for displaying and using the Layer Manager.



2. Activate the Top View 

tab by selecting anywhere in the viewport.

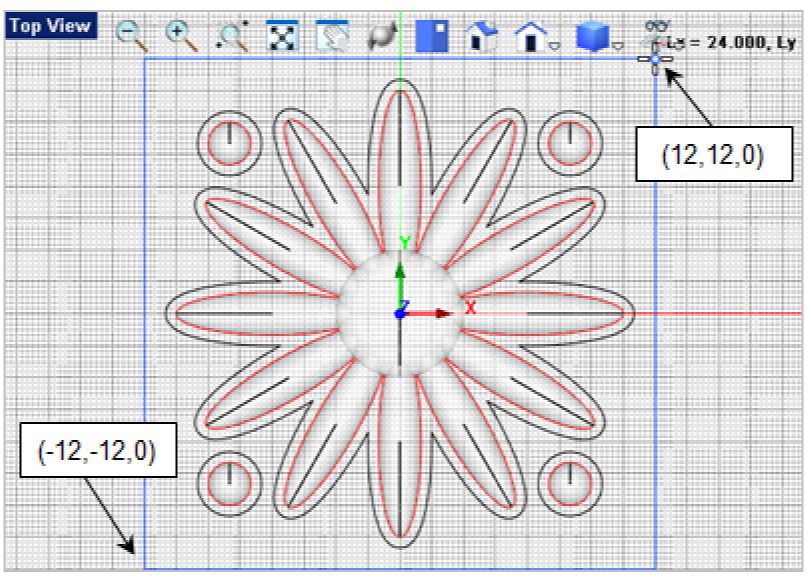
3. Form the Curve Modeling 

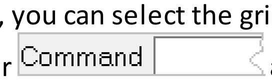
tab select the Rectangle 

command.

4. For the first corner of the rectangle, you can select the grid point shown below or enter -12,-12,0 in the command input bar 

and press <Enter>.



5. For the second corner of the rectangle, you can select the grid point shown below or enter 12,12,0 in the command input bar 

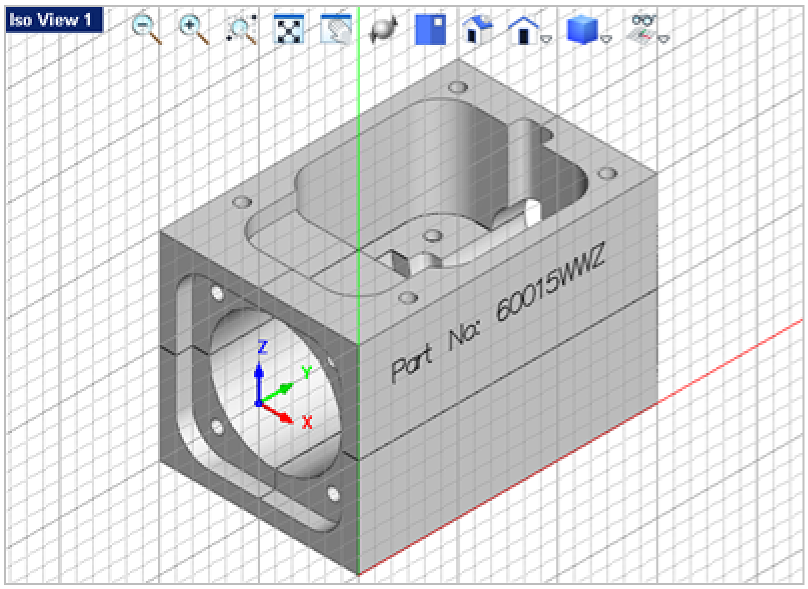
and press <Enter>.

6. Save the file as Daisy_Decor_Completed.vcp.

Congratulations, you have completed this exercise!

7.8 #8: Using Construction Planes

In this exercise you will learn how to use Construction Planes (C-Planes) to navigate and add features and text to your 3D models. C-Planes are also critical in learning how to orient imported parts for machining. Thus learning how to orient C-Planes and how to orient parts with the use of C-planes is critical knowledge for all VisualCAD users.



Using Construction Planes

What you will learn:

In this exercise you will perform the following VisualCAD tasks:

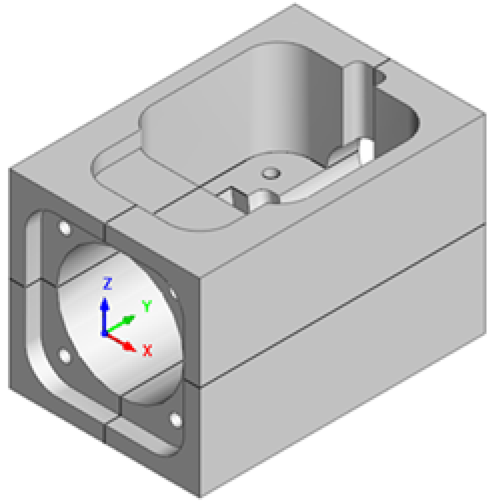
1. Orient the C-Plane.
2. Draw on a Face.
3. Other C-Plane Options.
4. Orient the Part.

7.8.1 Orient the C-Plane

Orient the C-Plane

In this step we'll show you how to orient the Construction Plane (C-Plane).

1. First if you have completed and save the part model from Exercise #5 Connector Block, open that part now. It is shown below:



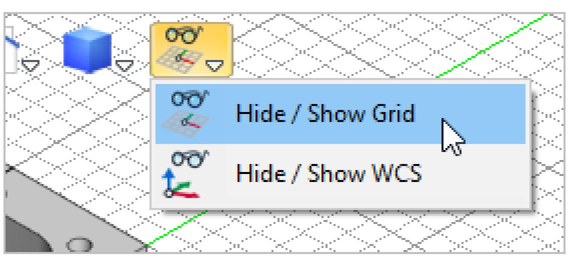
Connector Block

2. Now select the Modeling Aids tab. On the very right end of the ribbon bar you will find the C-Plane pane.

3. From the Modeling Aids  tab select the C-Plane to 3D Face  command.

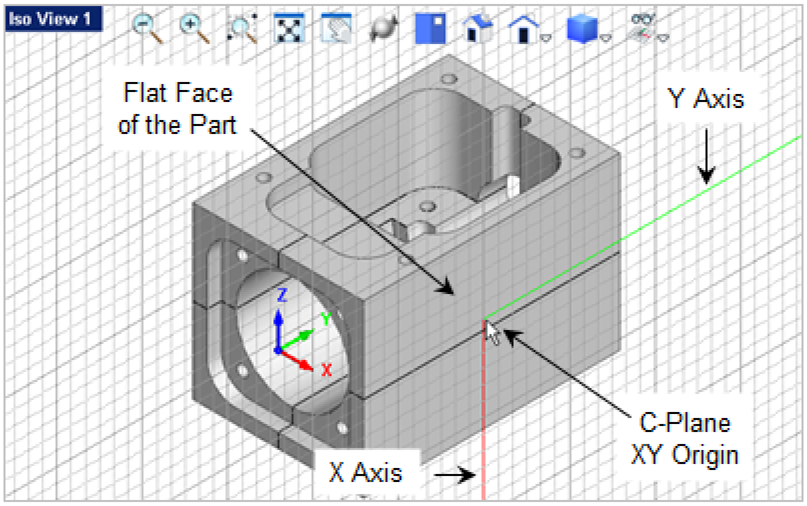
4. The command prompt says: "Pick a flat area". Select the right side face of the part and you will see that the C-Plane snaps to that face.

   If the Grid is not visible, select Hid / Show Grid from the View Toolbar.

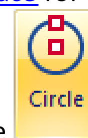


> It is best to have the Construction Plane Grid visible. This allows you to better keep track of where the C-Plane is located and oriented.

5. You can now see that the C-Plane and the Grid is alined with the right side flat face of the part. You will notice that the C-Plane origin is position at the point on the flat face you selected.

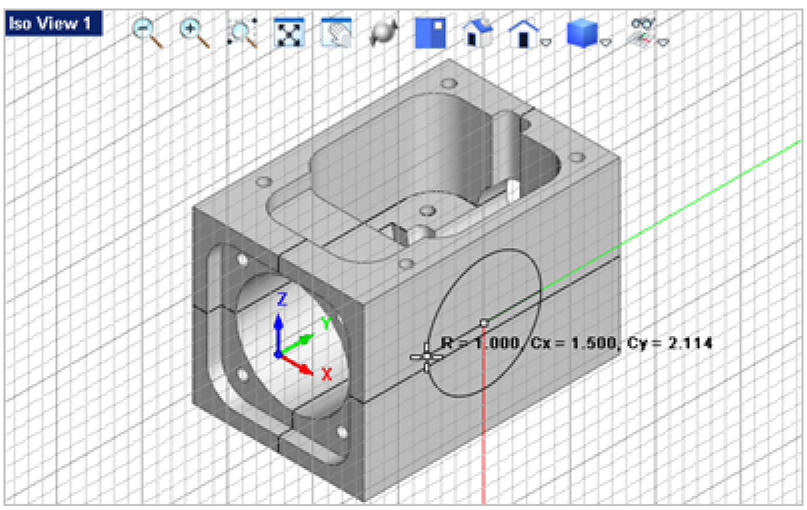


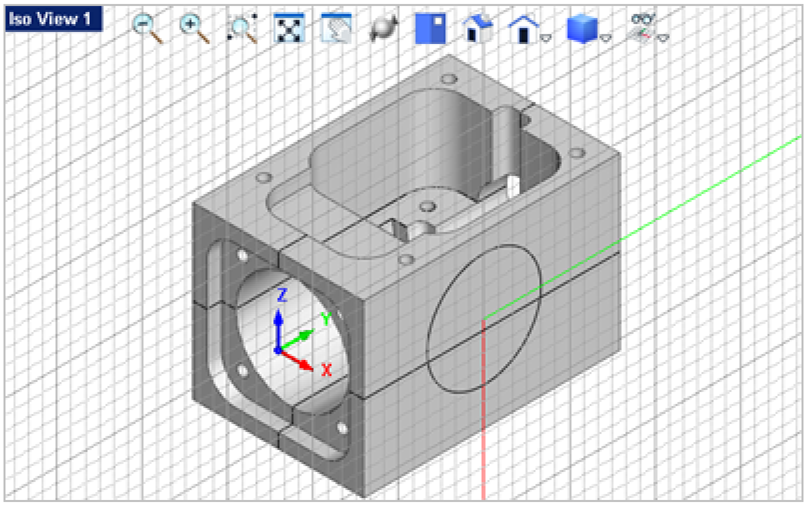
6. You can now draw on the C-Plane (see Text on a Part Face for more information).

From the Curve Modeling  tab select the  Circle on Point command.

7. For the center point, toggle on the Grid Object Snap  and select the origin point or simply enter 0,0 in the command window and press <Enter>.

8. Now select a point on the diameter of the circle.





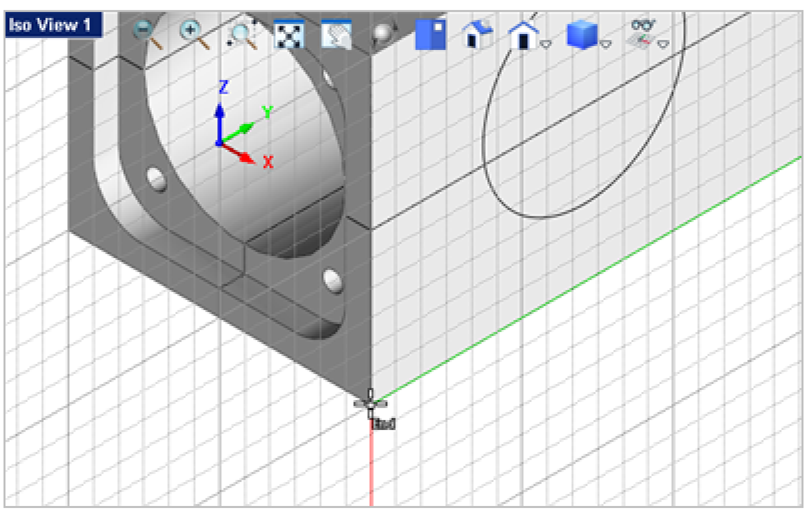
9. You can also move the C-Plane origin. From the Modeling Aids 

tab select the C-Plane Origin 

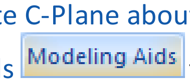
command.

10. The command prompt says: "Pick origin point or enter coordinates x,y and Z" Enable the End Point Snap  from the Status Toolbar and then select the bottom right end point of the face.

    You will see that the C-Plane origin and the Grid is moved to this point.

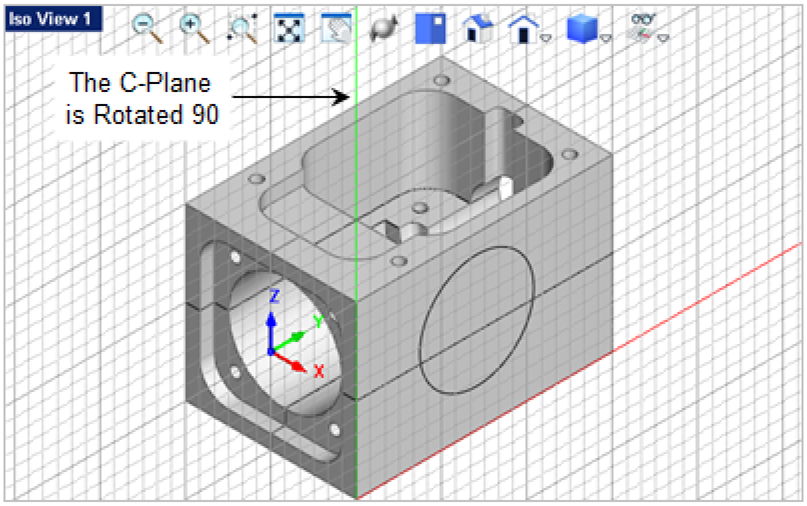


11. If you need the X Axis of the C-Plane aligned horizontally across the bottom of the part you can select the Rotate C-Plane about Axis 

command. It is also located on the Modeling Aids 

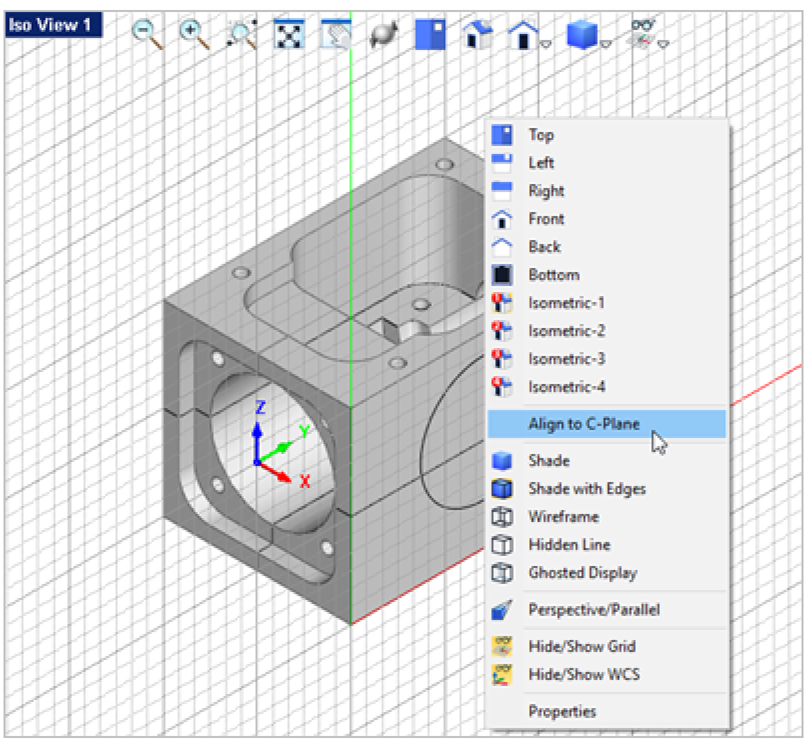
tab.

12. The command prompt says: "Enter axis to rotate about and angle to rotate by (eg: X,45)" We want to rotate the Z Axis 90 degrees so enter Z,90 in the command window Command and press <Enter>.

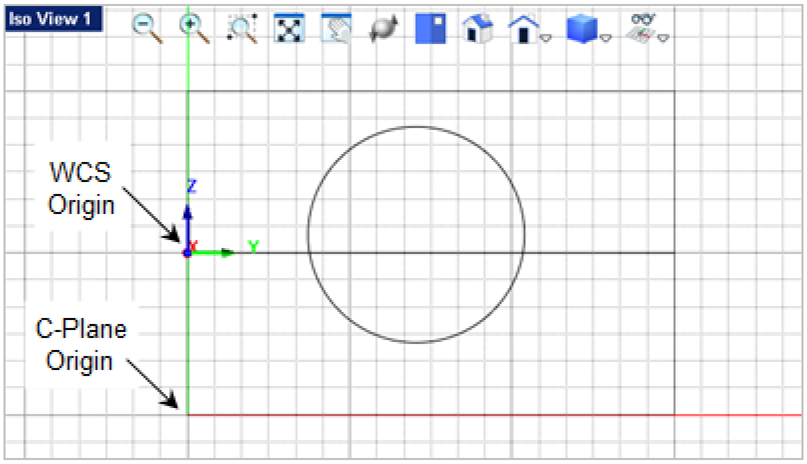


13. Now what if you want the view orientation aligned with the C-Plane?

14. Just right-click anywhere in the view to display the View pop-up menu and select the Align to C-Plane command. This menu contains many Viewport related commands.

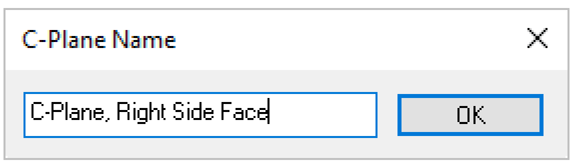


15. With the C-Plane aligned with the active Viewport pay close attention to where the origin of the C-Plane is:



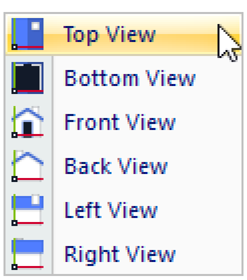
16. Now you may want to come back to the C-Plane later so let's save it. From the Modeling Aids Modeling Aids tab select the Save C-Plane command.

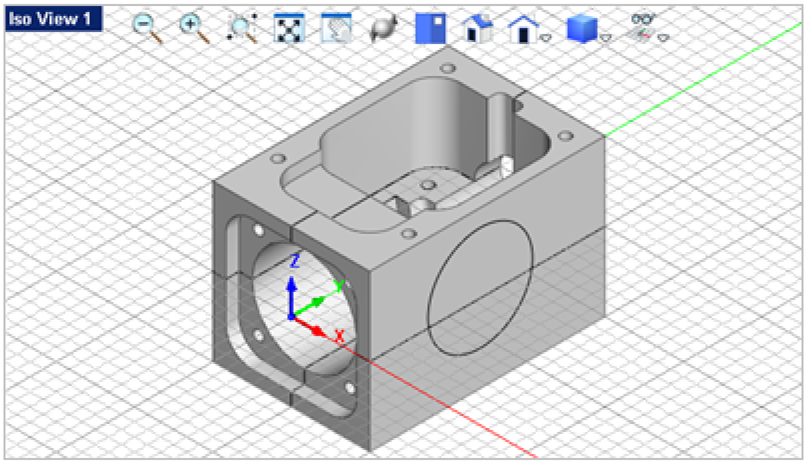
17. In the C-Plane Name dialog enter a unique name for the active C-Plane and then pick OK.

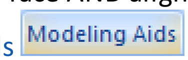


18. The active C-Plane is now saved with the part file.

19. Now let's change the C-Plane back to the default top view. From the Modeling Aids Modeling Aids tab select the View View menu and select Top View.





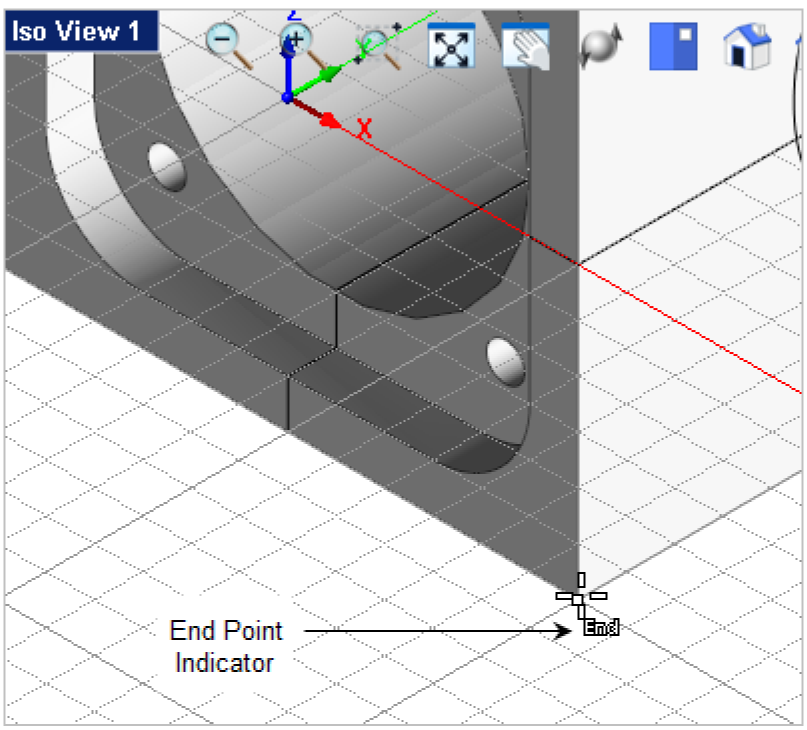
20. Here is an easier way to align the C-Plane to the face AND align the XY Axis of the C-Plane at the same time. From the Modeling Aids 

tab select the C-Plane by 3 Points 

command.

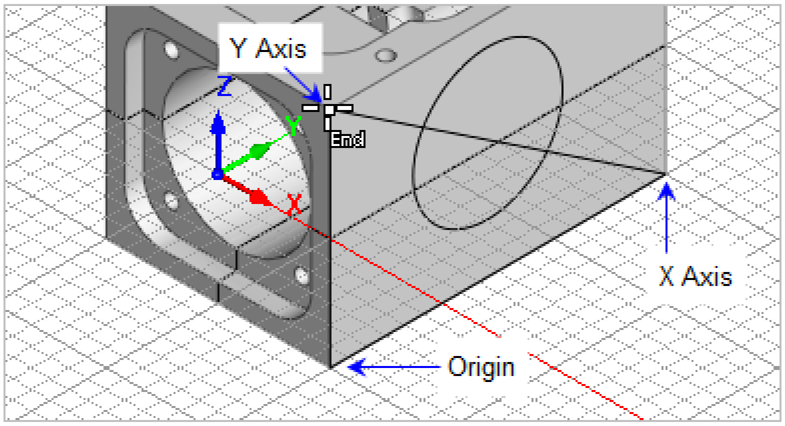
21. The first point selected will be the origin, so make sure the End Point Snap  is toggled on.

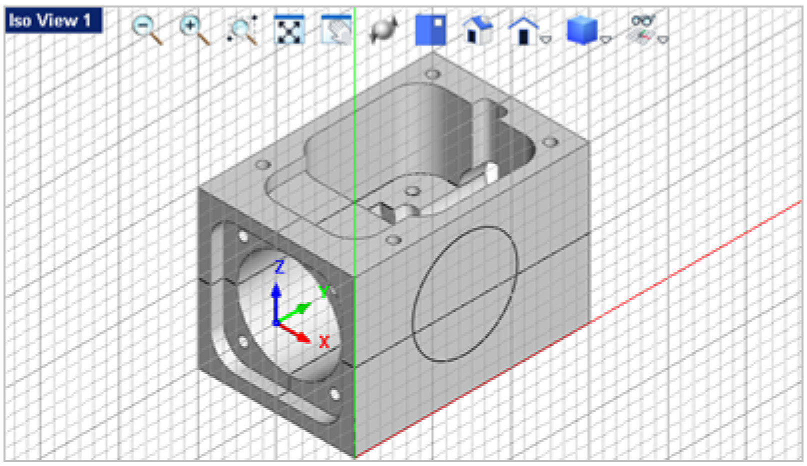
22. Then select the bottom left corner end point of the right side face of the part. **Note**: Do Not pick until you see the "End" indicated at the cursor location.



23. For the X Axis point select the end point at the back end of the part on the right side face.

24. For the Y Axis point select the end point at the top of the part on the right side face.



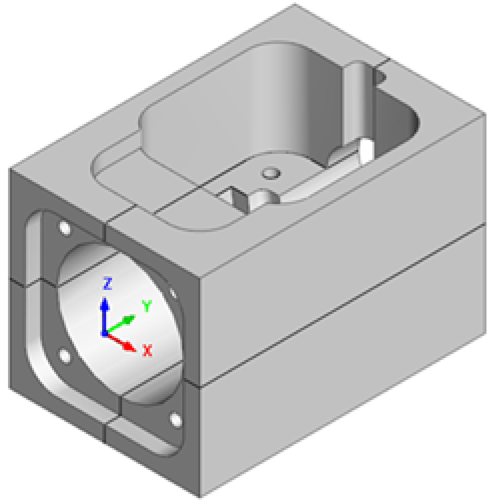


7.8.2 Text on a Part Face

Drawing on a Face

In the last step (see Orient the C-Plane) we discussed different ways to orient the C-Plane. In this step we'll show you how to draw on a selected part face. This will come in handy if you want to identify your 3D parts or for modifying them.

1. First if you have completed and save the part model from Exercise #5 Connector Block, open that part now. It is shown below:



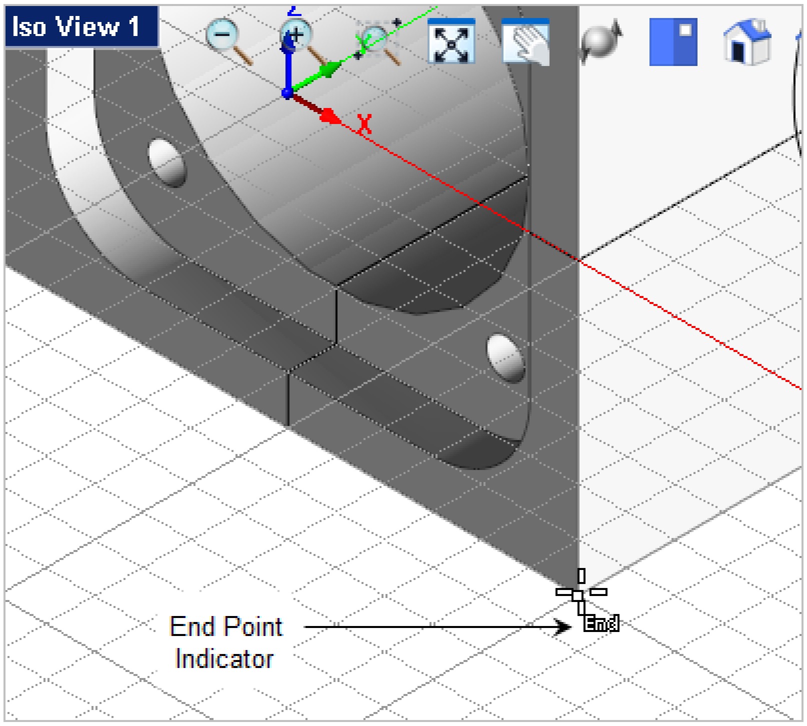
Connector Block

2. Now select the Modeling Aids  tab. On the very right end of the ribbon bar you will find the C-Plane pane.

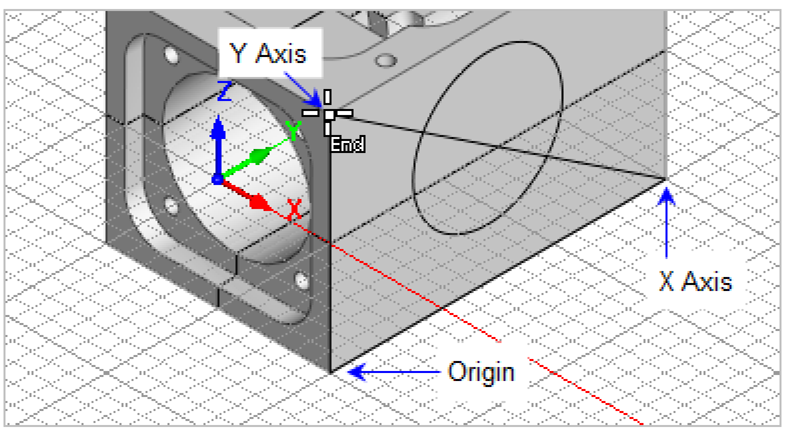
3. From the Modeling Aids  tab select the C-Plane by 3 Points  command.

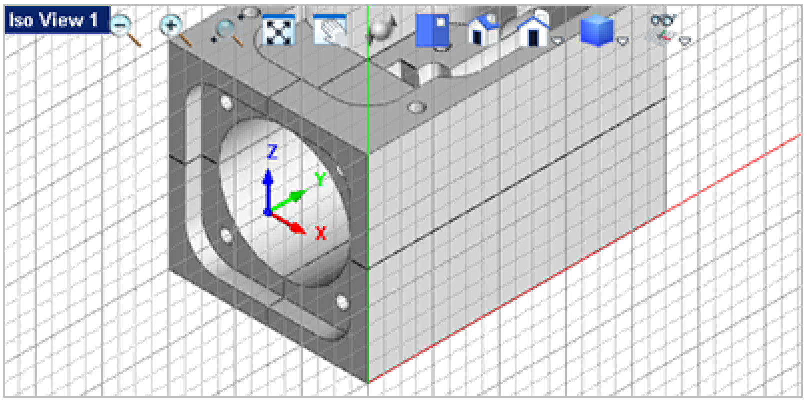
4. The first point selected will be the origin, so make sure the End Point Snap  is toggled on.

5. Then select the bottom left corner end point of the right side face of the part. **Note**: Do Not pick until you see the "End" indicated at the cursor location.

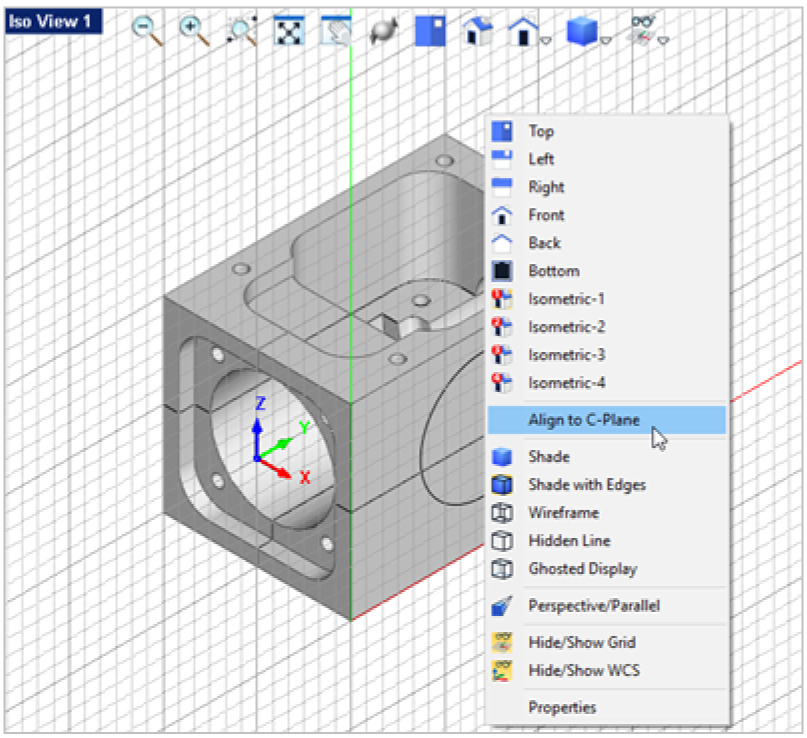


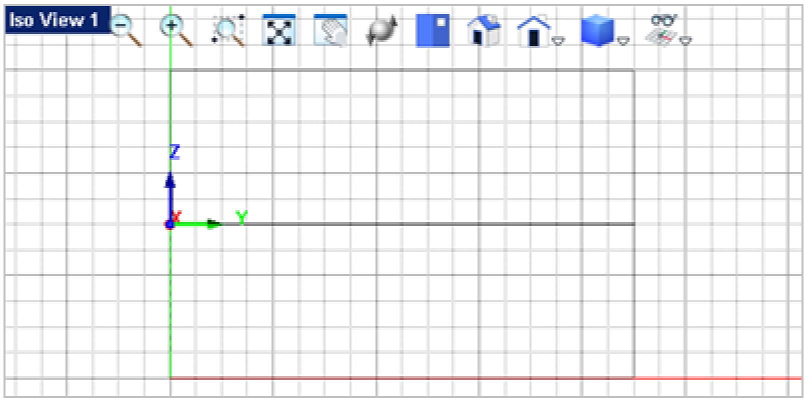
6. For the X Axis point select the end point at the back end of the part on the right side face.
7. For the Y Axis point select the end point at the top of the part on the right side face.





8. Now right-click anywhere in the view to display the Viewpoint pop-up menu and select the Align to C-Plane command. This menu contains the Viewport related commands.

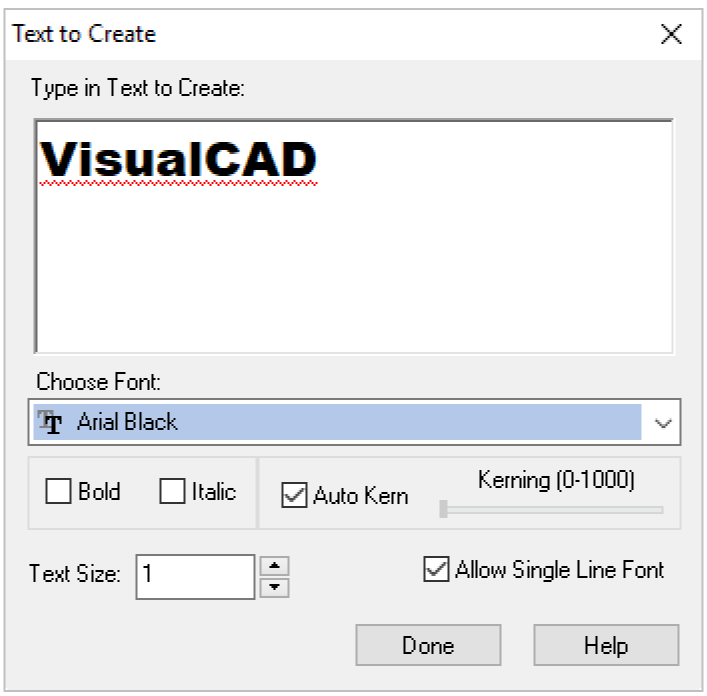




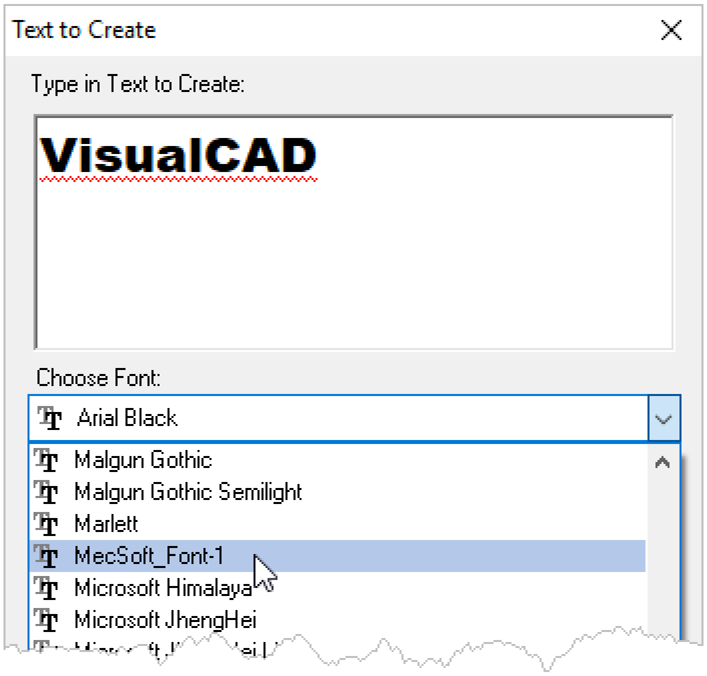
9. If you want to simply label your part you can add dimension text that will lie on the C-Plane. See Drawing & Dimensioning for that exercise.

10. In this exercise we will create text curves that you can engrave with the 2-1/2 Engraving toolpath strategy, go to the Curve Modeling tab and select the Create Text command.

11. This will display the Text to Create dialog.

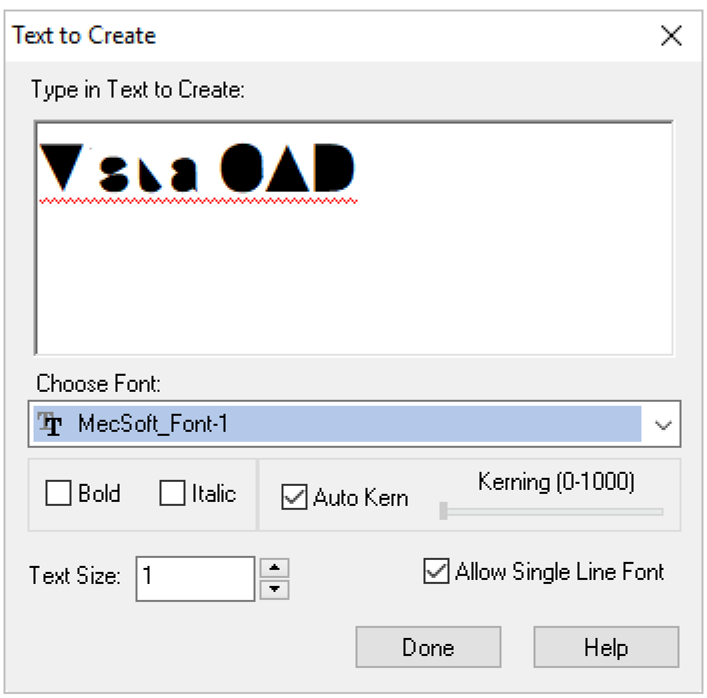


12. For engraving text you may want to use a single stroke font. From the Font drop-down menu select the text font named MecSoft_Font-1.



💡 If you do not see the MecSoft Font, close VisualCAD and [click here to Download the single stroke font](). Unzip and copy the file MecSoft_Font_1.ttf to the C:\Windows\Fonts folder.

13. The text preview will now look like this.



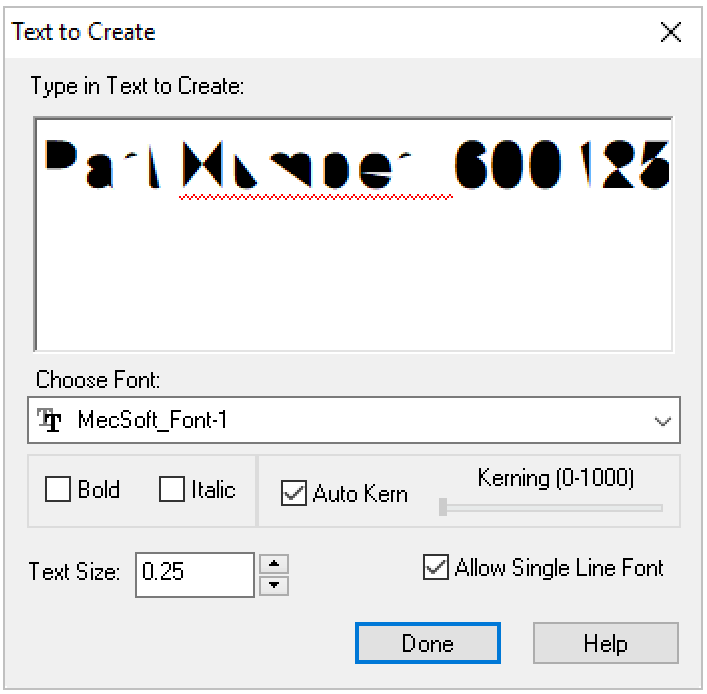
14. Make the following adjustments to the dialog:

    Allow Single Line Font: Checked
    Text Size: 0.25

15. Now place in the Type in Text to Create window, replace the text VisualCAD with the following text:

    Part Number: 6001257

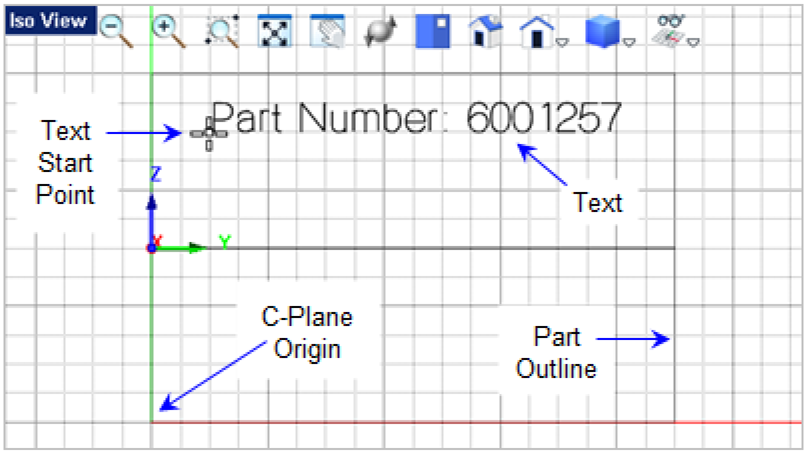
16. The dialog should look like this:



17. Now pick the Done button. The dialog will close and command prompt says: Enter start point of text.

18. You will see the text as you move the cursor around the Viewport.

19. Enable the Grid Snap and then select the grid point to position the text, or 0.5,2.5 in the command window Command and press <Enter>. The text should appear like this:

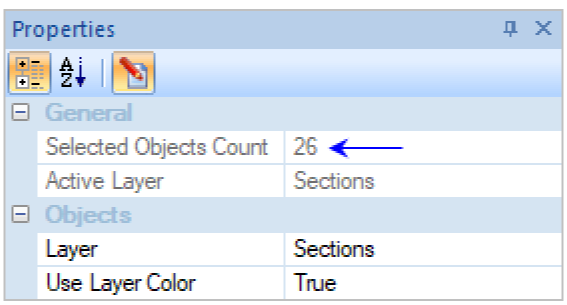


The text is actually individual curves that you can select as Control Geometry in your machining operations.

20. To select the curves, first make sure that the option Use Preselection Highlight is unchecked. It is located on the System section of the Options dialog (Home tab > Options > System). This was covered in the Set System Options section of Exercise #1.

21. Now select one of the curves in the text or window select all of the curves in the text and then pick the Properties icon from the Status Bar to display the Properties dialog.

22. You will see that the text is actually 26 individual objects.



7.8.3 Other C-Plane Commands

Other C-Plane Commands

Be sure review the other C-Plane commands listed below.

1. Select the Modeling Aids Modeling Aids tab.

2. The C-Plane command pane is located on the right side of the ribbon bar:

Icon	Command
View	Immediately sets the orientation of the C-Plane to be parallel to the plane of the currently active view. The origin remains the same.
Origin	Moves the origin of the C-Plane to a selected point or entered coordinates. The current configuration of the construction plane is saved whenever the model is saved.
Elevation	Moves the elevation of the C-Plane along its normal (perpendicular to the plane). Input is a single distance and may be positive or negative. A positive number moves the plane towards the positive end of the coordinate axes even if the viewing location is along the negative axis. All input values are treated as an incremental distance relative to the current location of the plane.
Rotate	Rotates the C-Plane about one of the principle axes of the current C-Plane, X, Y, or Z. The center of rotation is the center of the plane and not necessarily the origin. Input is 'axis,degrees'.
By 3 Pts	Three points are selected (origin, x-axis and y-axis) or entered to re-orient the C-Plane. The y-axis point defines the C-Plane and the general direction of the positive y-axis.
To Face	The C-Plane will be oriented to a selected flat plane of a geometry object. The pick point will become the C-Plane origin.
By X Axis	Two points are entered to re-orient the x-axis of the C-Plane:
To WCS	Resets the C-Plane back to its default orientation relative to the WCS and the currently active viewport. The origin will be the WCS origin.
	This option saves the C-Plane orientation from the currently active view. A name must be provided for later reference when loading.
	This option allows the retrieval of a saved C-Plane into the currently active view. A list of previously saved C-Planes is

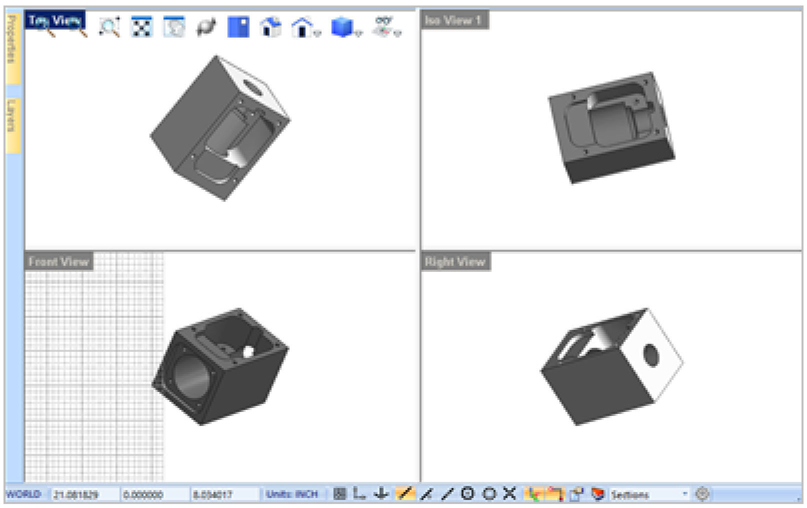
	presented. These saved C-Planes may also be deleted in this option or renamed.

7.8.4 Orient the Part

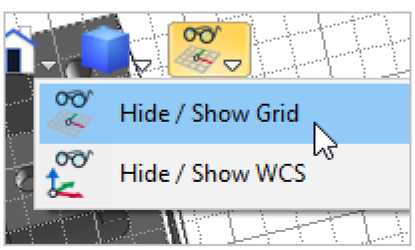
Orient the Part

There may be times when you need to re-orient the part. This can happen at times when a part that you import into VisualCAD has it's WCS position skewed. In order to work with the part it has be oriented.

We have deliberately skewed the part from Exercise #5 Connector Block so that it no longer lies parallel to any view. Here is what it looks like:



1. The first step toward to orienting this part is to define the C-Plane. For more about this see the topics Orient the C-Plane and Text on a Part Face in this exercise.
2. First make sure the Grid is visible. You can toggle the Grid on/off by selecting Hide / Show Grid from the View Toolbar.



3. Now activate the view that allows you to see the planar face that you want to use to orient the part by. In this example it is Iso View 1 

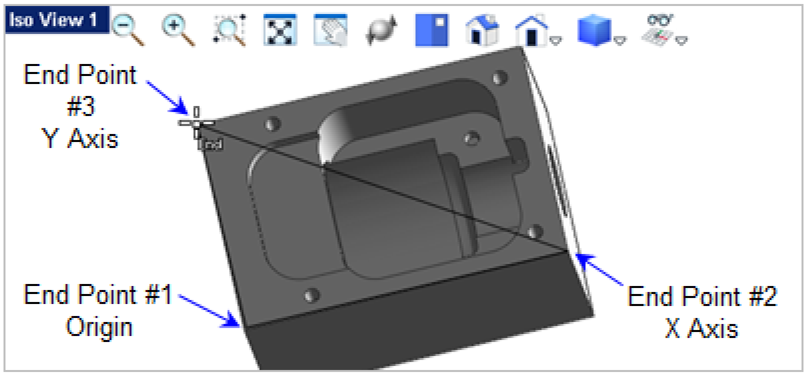
.

4. Go to the Modeling Aids 

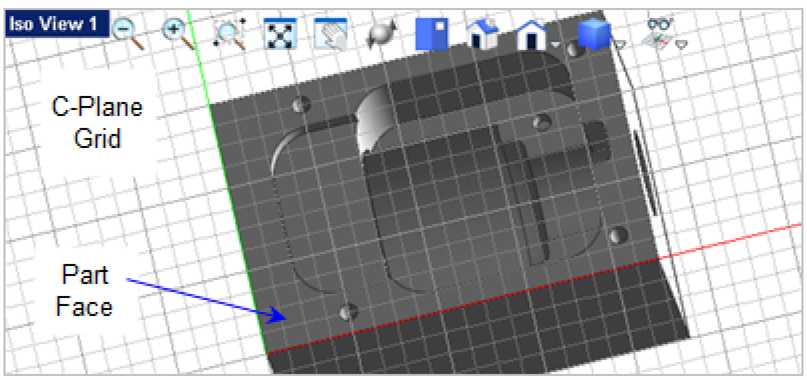
tab and select the C-Plane by 3 Points 

command.

5. Activate the End Point Snap  from the Status Toolbar and select the end points in the order shown below:

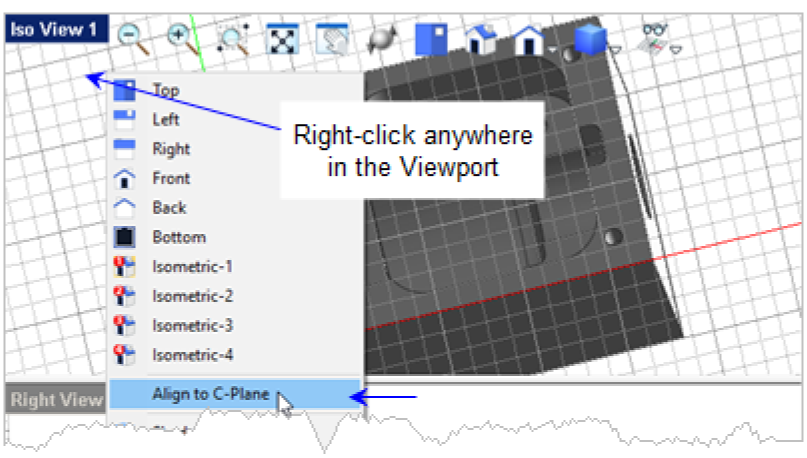


The C-Plane will orient to the 3 points selected.

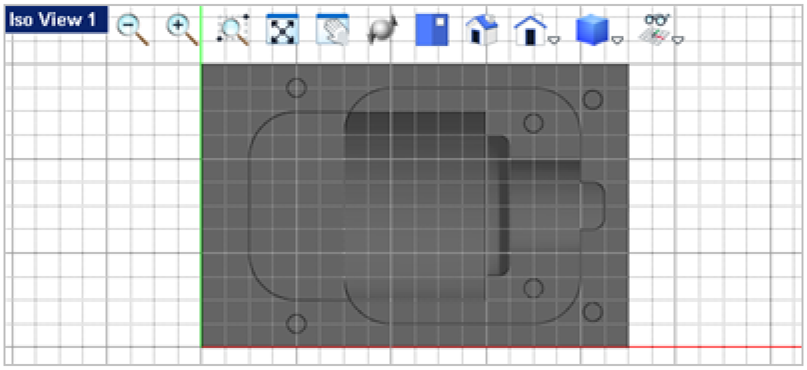


6. Now right-click anywhere in the Viewport (do not select any geometry) and select Align to C-Plane. This will align Iso View 1 

parallel to the C-Plane.



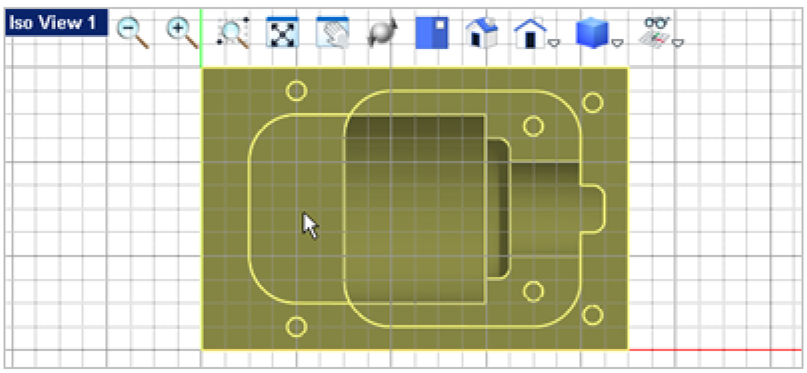
The C-Plane will align to the Viewport.



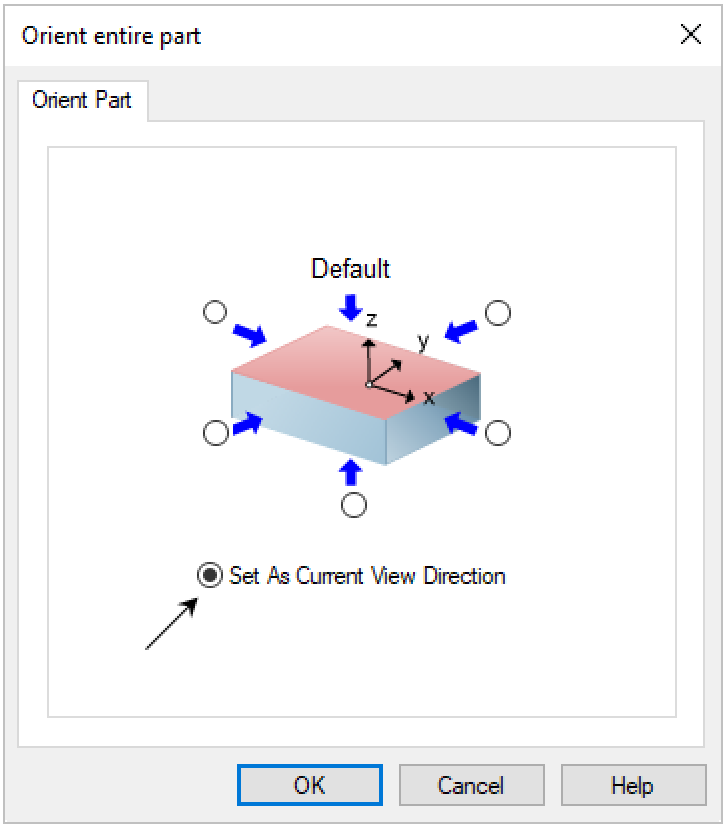
7. Now we can orient the part. Select the part and go to the Modify / Transform 

tab and select the Part Orient ... 

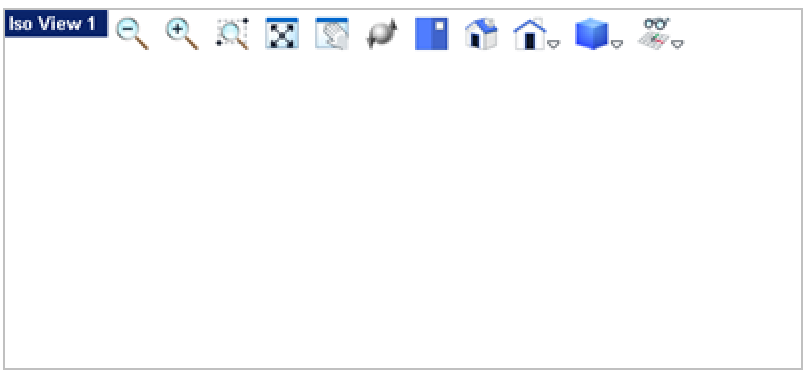
command.



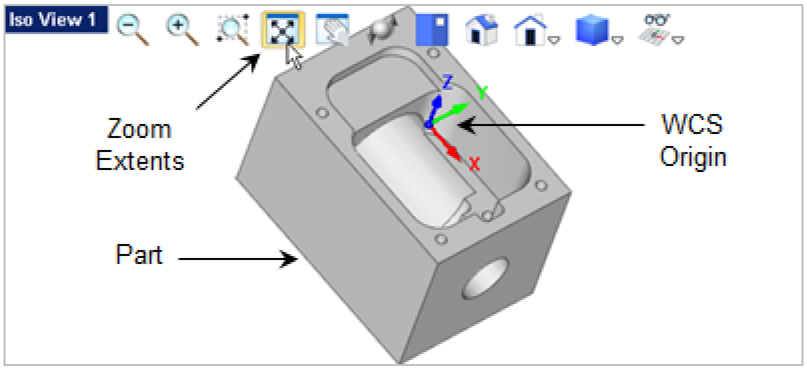
8. This will display the Orient entire part dialog. Pick the option to Set as current view direction.



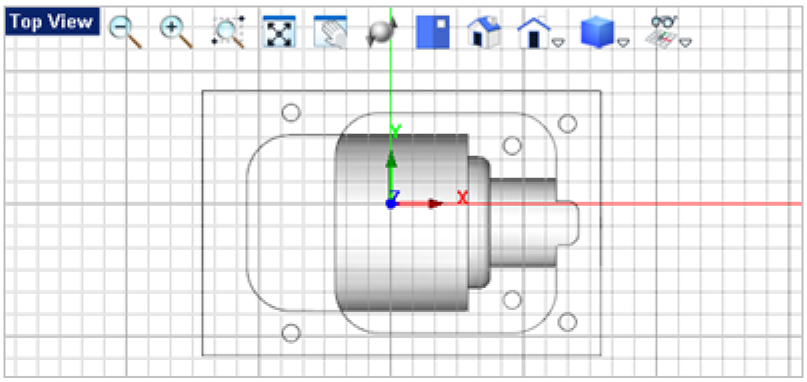
9. Pick OK and the part appears to disappear from the Viewport.



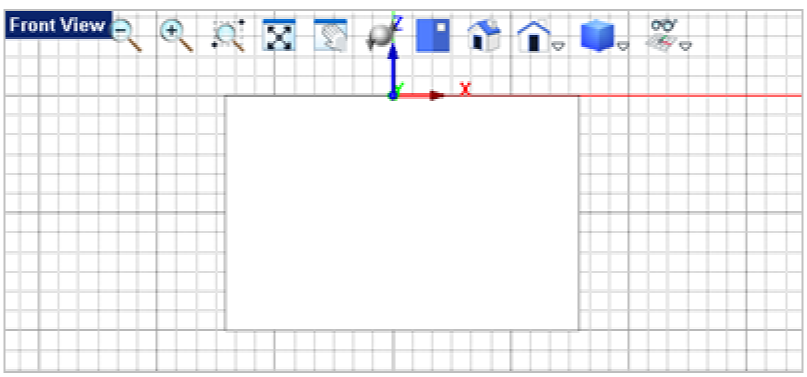
10. Now pick the Fit View  icon from the View Toolbar. The part will appear with the WCS oriented in/on the center of the face that we used to align the C-Plane

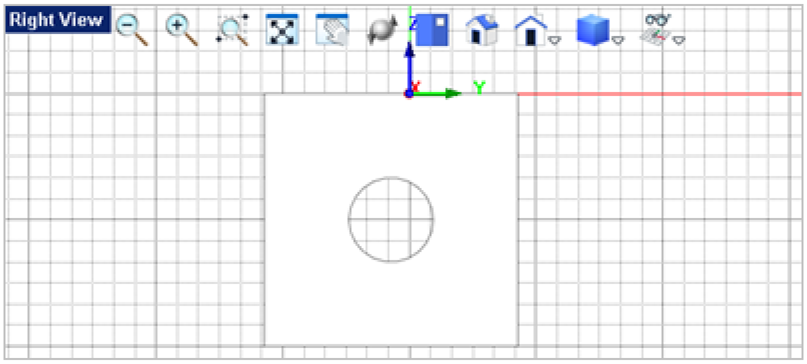


11. Now activate the Top View  and pick the Fit View  icon from the View Toolbar.

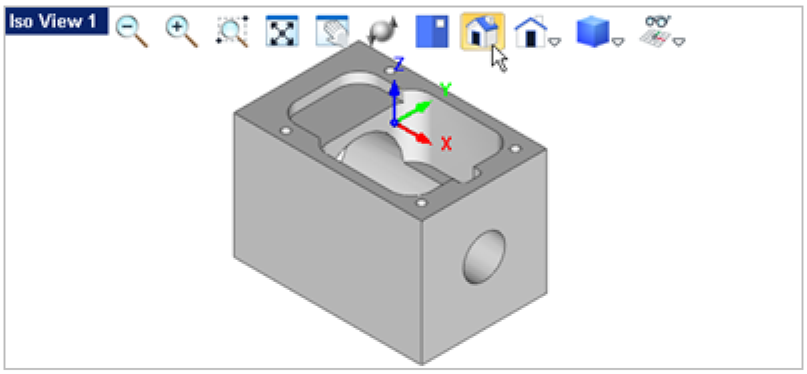


12. Repeat this for the Front View  and the Right View . Remember that you have to activate the Viewport before the View Toolbar will display. Left-click anywhere within the Viewport to activate it.



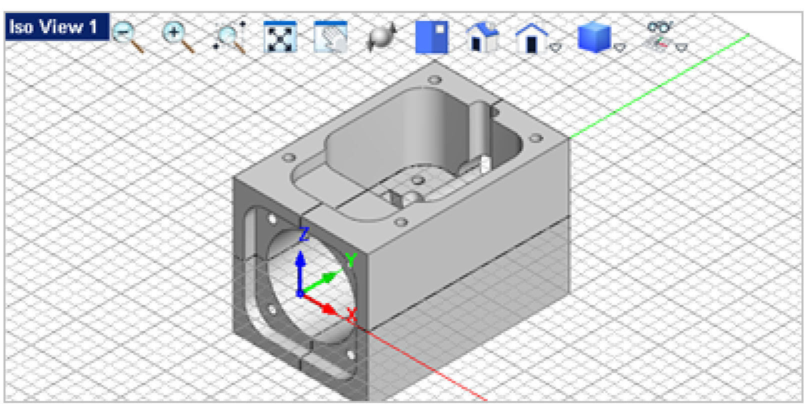


13. Activate Iso View 1  again and this time select the Iso View  icon from the View Toolbar to orient this Viewport.



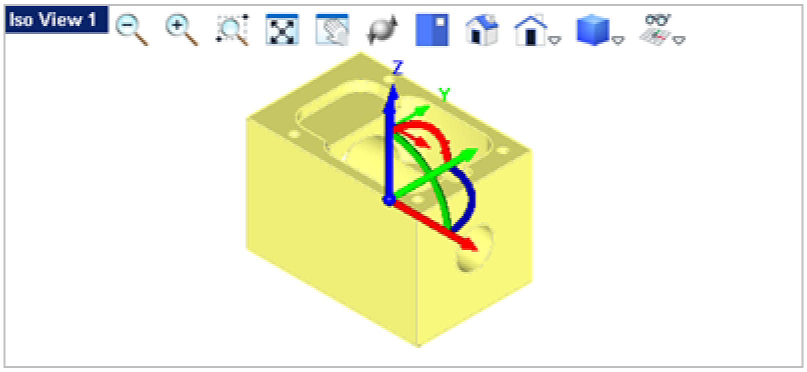
Great! You have the part oriented orthographically (i.e., Top, Front, Right).

14. However, the part is still not oriented like the original part as shown below. You see that the length of the part was oriented along the Y Axis and the WCS was positioned at the center of the front view. Let's show you some additional commands to fix that.

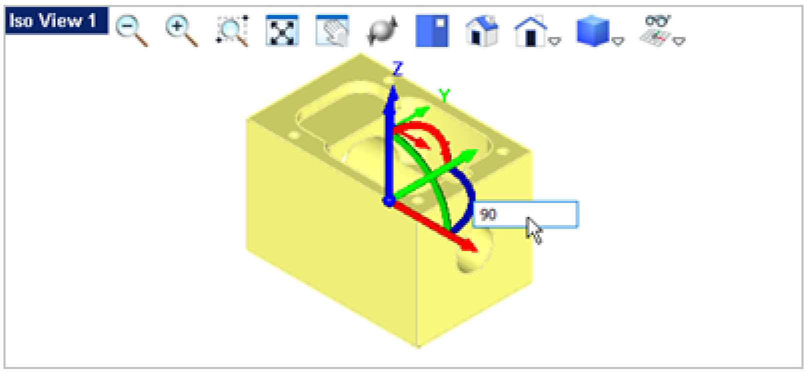


15. Activate Iso View 1  and then from the Status Toolbar, toggle the Graphic Manipulator  On.

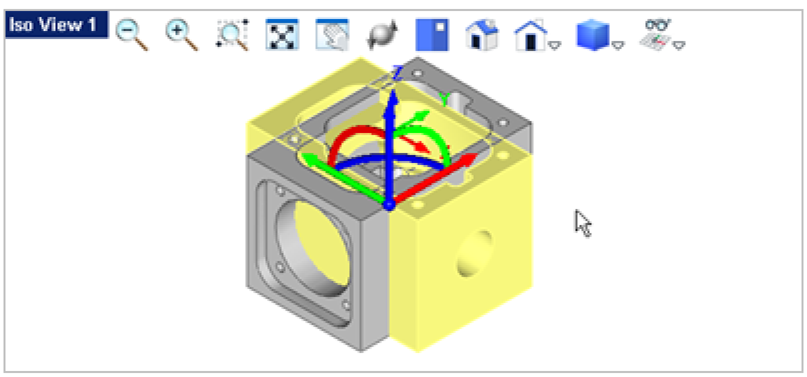
16. Now select the part to see the Graphic Manipulator displayed.

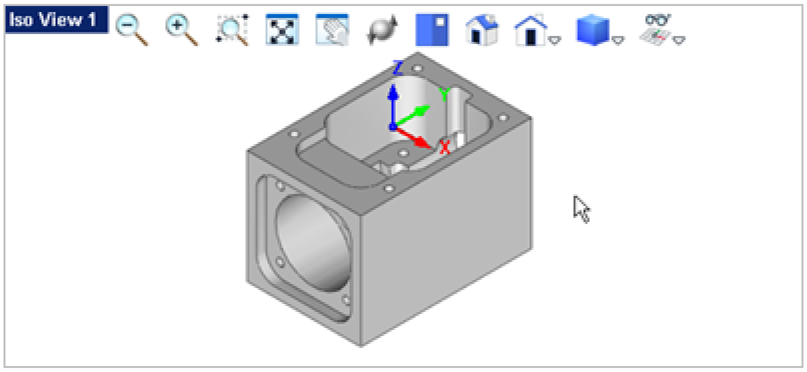


17. Pick the Z Rotation arc of the Graphic Manipulator  to display the input window, enter 90 and then press <Enter>.



18. Now you see the part is rotated so press <Enter> to accept the rotation.

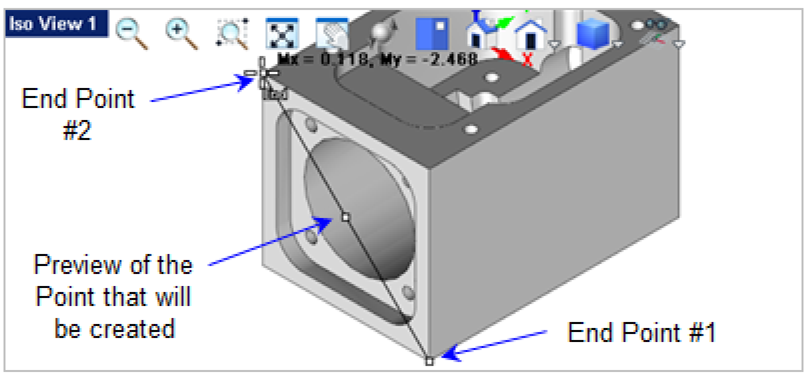




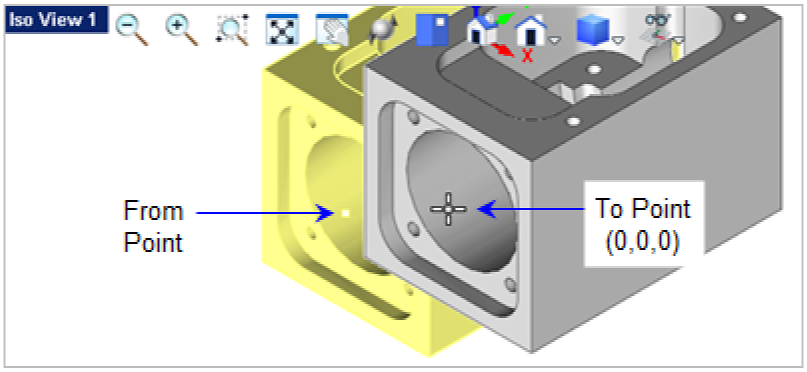
Great where almost there!

To move the WCS to the original location which was centered on the front face of the part we need to create a point there to move from.

19. Zoom in on the front face of the part.
20. From the Curve Modeling Curve Modeling tab select the command Create mid point between two points.
21. Activate the End Point Snap from the Status toolbar.
22. Select the two diagonal end points on the front face of the part as shown.

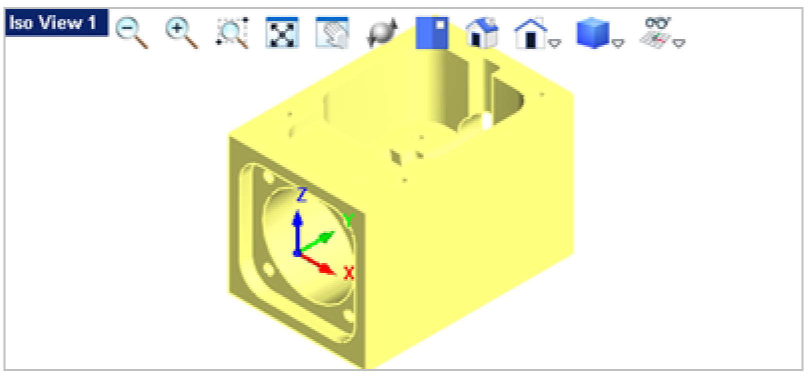


23. You can now toggle the Graphic Manipulator off.
24. With the point created, window select the entire part (and point) or simply press <Ctrl+A>.
25. From the Modify / Transform Modify Transform tab select the Move by mouse Move command.
26. The command prompt says: Pick from point or enter coordinates x,y and z. Make sure the End Point Snap is active from the Status Toolbar and select the point you just created.



27. Now the command prompt says: Pick to point or enter coordinates x,y and z. In the command window enter 0,0,0 Command and then press <Enter>.

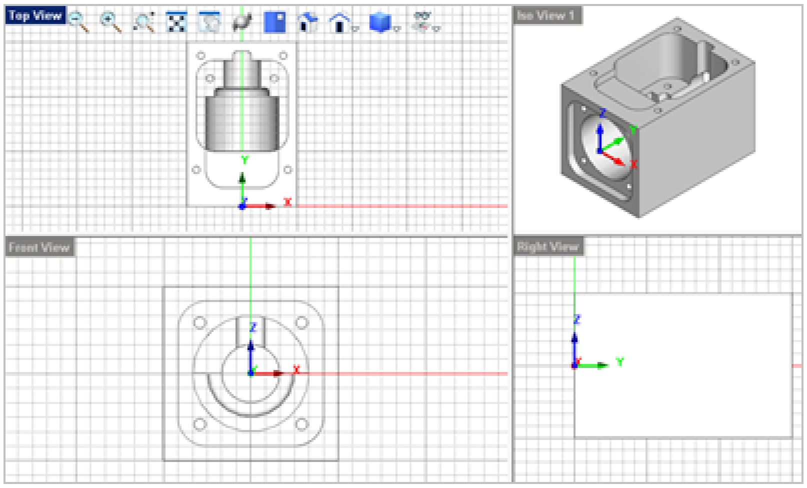
    You will see that the part moved to the WCS origin. Now press <Enter> again to accept the move.



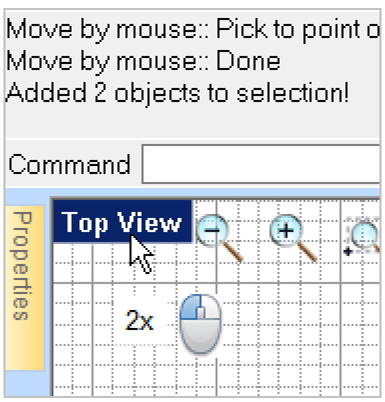
28. Now the part is aligned correctly accept for the C-Plane. Previously the C-Plane was aligned to the XY plane of WCS.

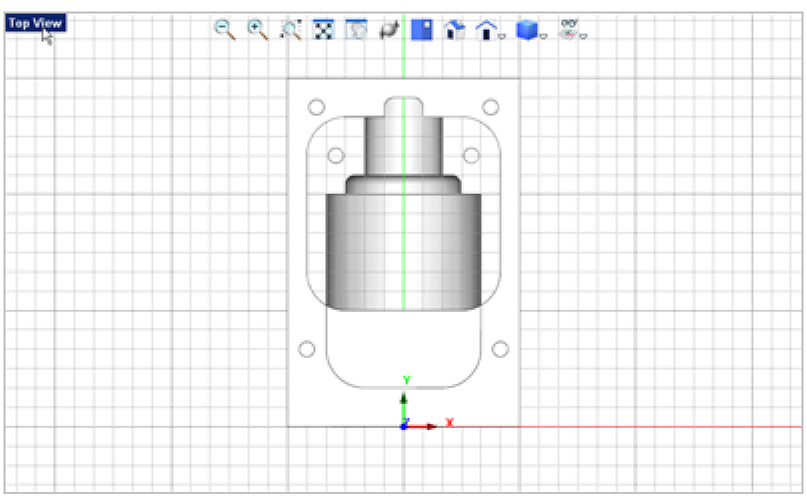
    Here is a neat trick to fix that:

29. Activate the Top View Top View.

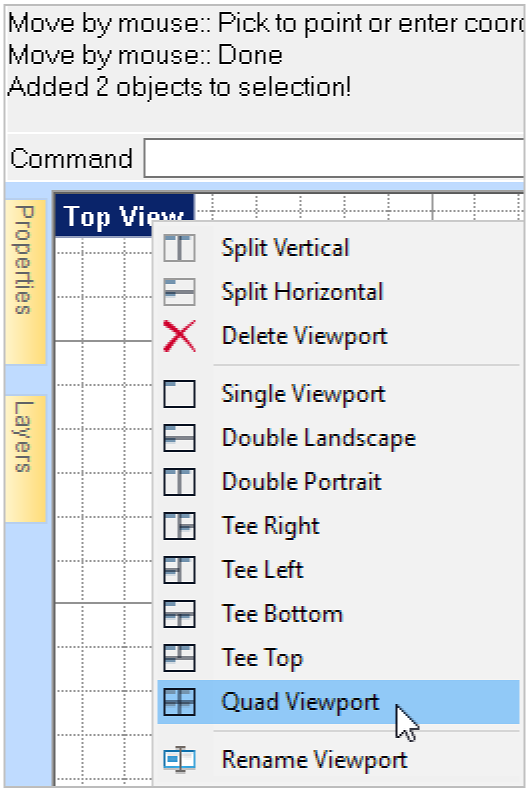


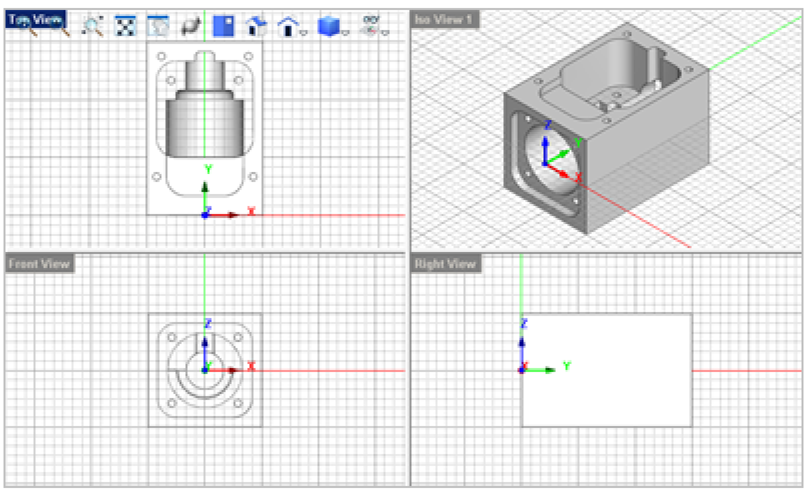
30. Now double-left-click on the Top View tab. This will change the display to one Viewport set to the Top View.





31. Now right-click on the Top View tab and select Quad Viewport.





You see that Iso View 1 

inherited the C-Plane of the Top View because that was the active View when you selected Quad Viewport.

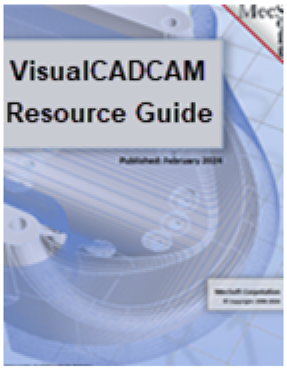
32. Save the file as Construction_Planes_Completed.vcp.

Congratulations on completing this exercise!

Find More Resources

Download this PDF Guide for a list of the available VisualCAD Resources.

2025 VisualCAD Resource Guide



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