

VisualCAD Exercise Guide 2026

Thursday, March 12, 2026

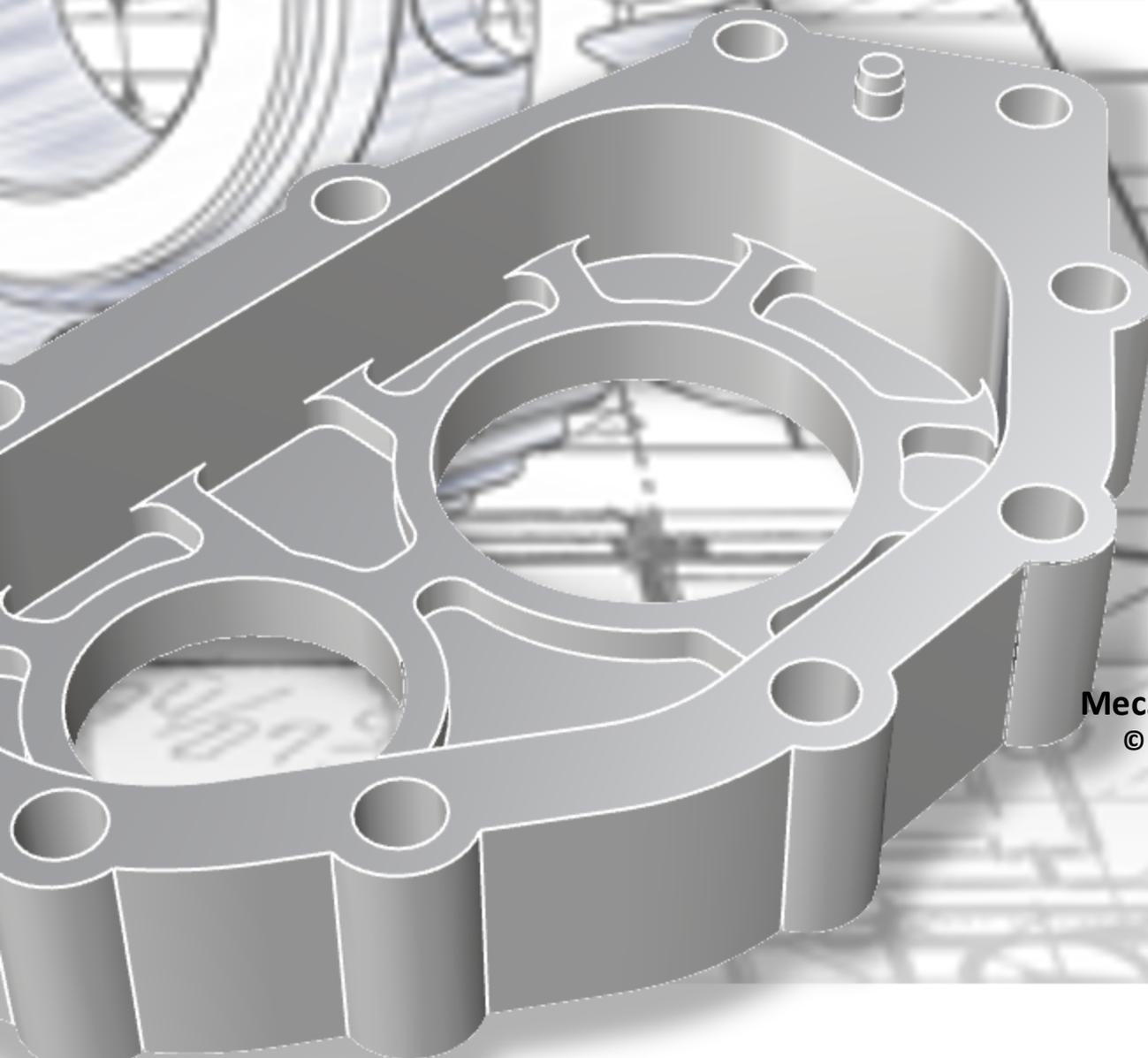


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Welcome to VisualCAD!



VisualCAD 2026

[Prefer Printed Documentation? Click Here!](#)

[What's New](#) | [Quick Start Play List](#)

VisualCAD/CAM 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/CAM Screen Layout](#)

[VisualCAD/CAM Features Overview](#)

[Default Shortcut Keys](#)

[Customize VisualCAD/CAM Display](#)

1.1 Features Overview

Here is an overview of the features you will find in [VisualCAD/CAM](#):



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/CAM](#) 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/CAM](#)

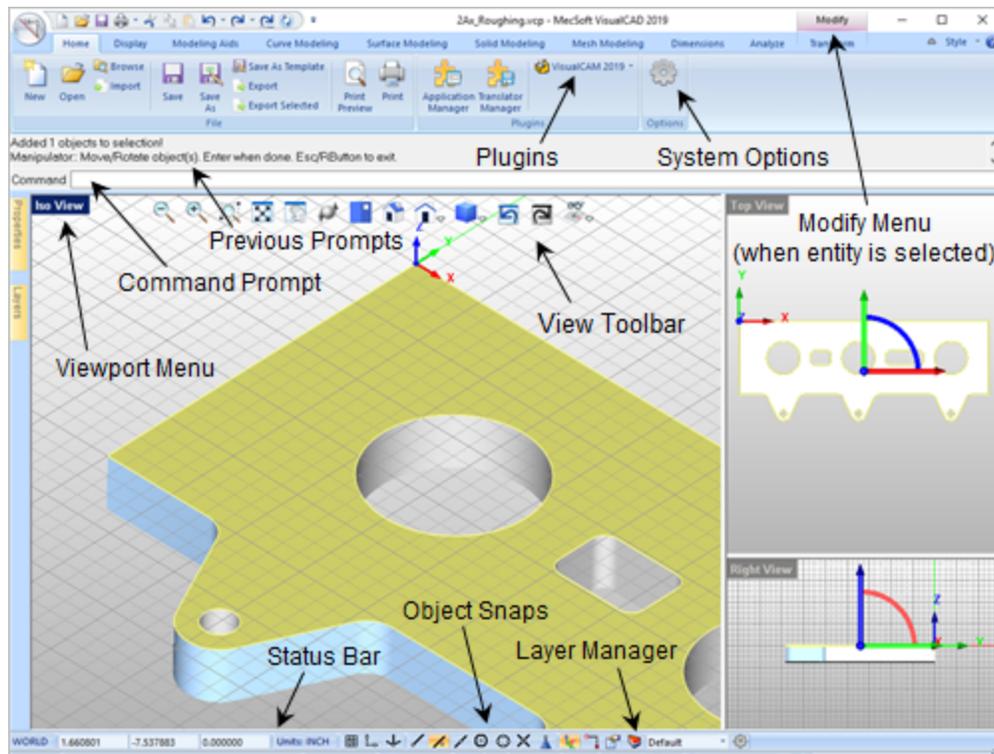
[The VisualCAD/CAM Screen Layout](#)

[Default Shortcut Keys](#)

[Customize VisualCAD/CAM 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/CAM](#)

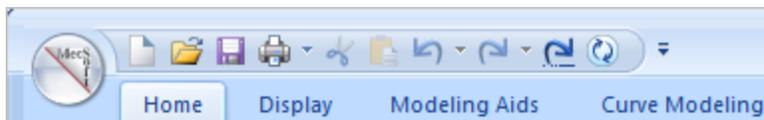
[VisualCAD/CAM Features Overview](#)

[Default Shortcut Keys](#)

[Customize VisualCAD/CAM Display](#)

1.3 Quick Access Toolbar

The [Quick Access Toolbar](#) is located at the top of the [VisualCAD/CAM](#) 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/CAM Features Overview](#)
- [Default Shortcut Keys](#)
- [Preset <Alt> Accelerator Keys](#)
- [Customize VisualCAD/CAM Display](#)

1.4 Default Shortcut Keys

VisualCAD/CAM installs with the following keyboard shortcuts preset. You can customize your keyboard to quickly execute many other VisualCAD/CAM commands using the [Customize Quick Access Toolbar](#) and [Customize Keyboard](#) dialogs.



Default Keyboard Shortcuts



Related Topics

[Customize Quick Access Toolbar](#)

[Customize Keyboard](#)

[Customize the Display](#)

[Preset <Alt> Accelerator Keys](#)

[The VisualCAD/CAM Screen Layout](#)

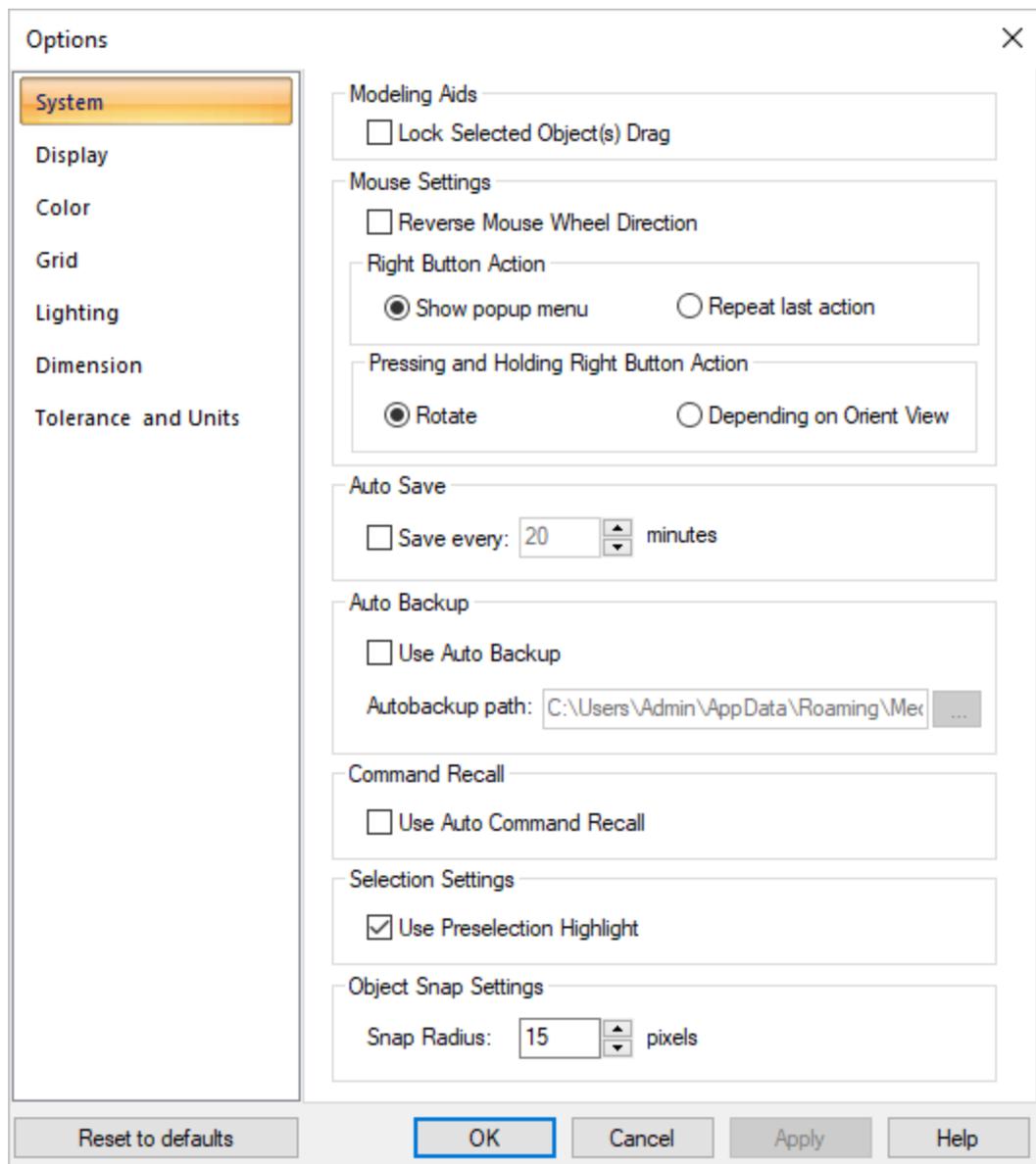
1.5 Options



 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

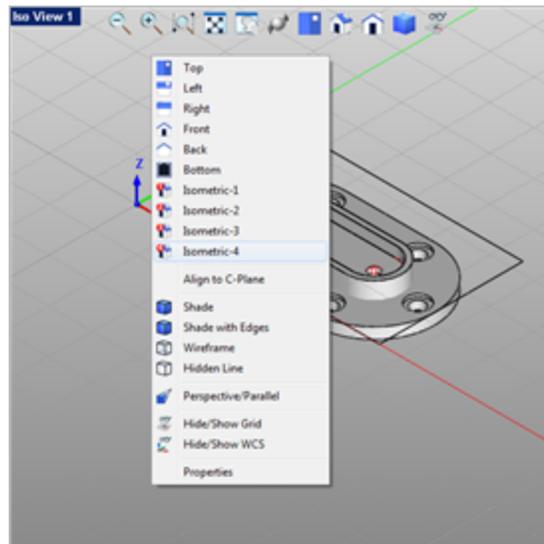
The following Mouse options are supported:

Reverse Mouse Wheel Direction

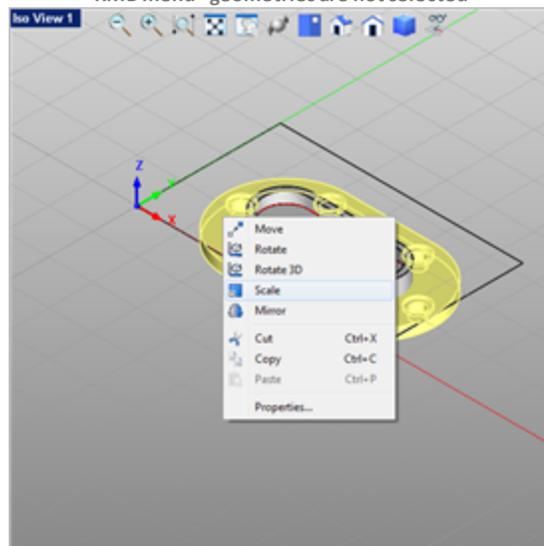
Reverses the direction of view zooming when using the mouse wheel.

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/CAM** 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!

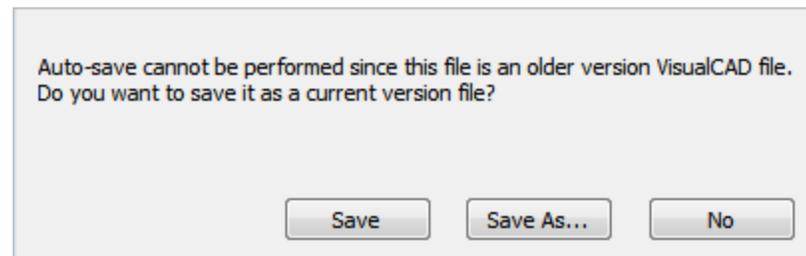
Under the **AutoSave** function, check the box next to '**Save every xx minutes**' to activate the **AutoSave** function. The time interval can be entered in minutes.

! **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:

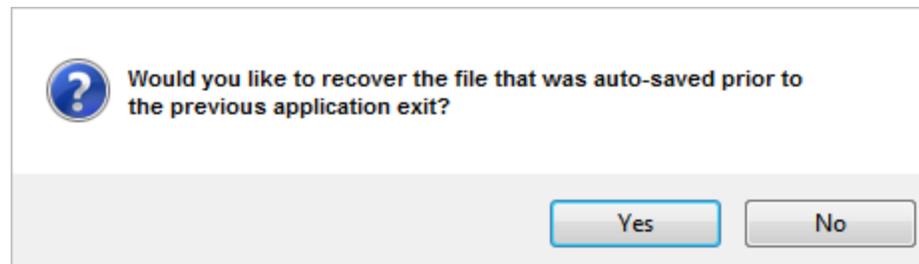
1. When the application closes normally.
2. When you close the file either explicitly or by opening another file.
3. When you select not to recover an **AutoSave** file that was detected on startup



File Recovery

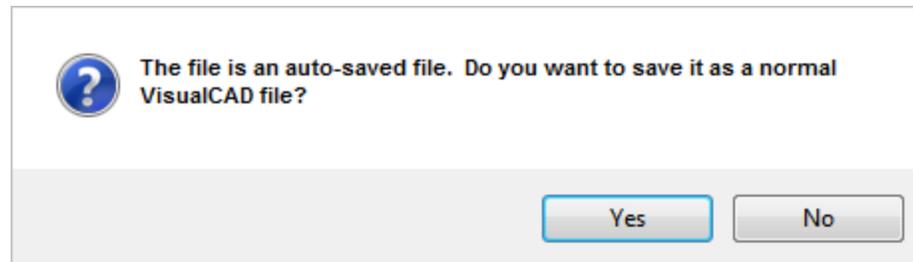
In the event of a computer crash or if **VisualCAD/CAM** is terminated abruptly by ending the process from task manager, **VisualCAD/CAM** will recover your file if **AutoSave** was active and your part file was automatically saved at least once.

If an **AutoSave** file has been created prior to the crash or termination, **VisualCAD/CAM** 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/CAM** 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/CAM** 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/CAM** 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

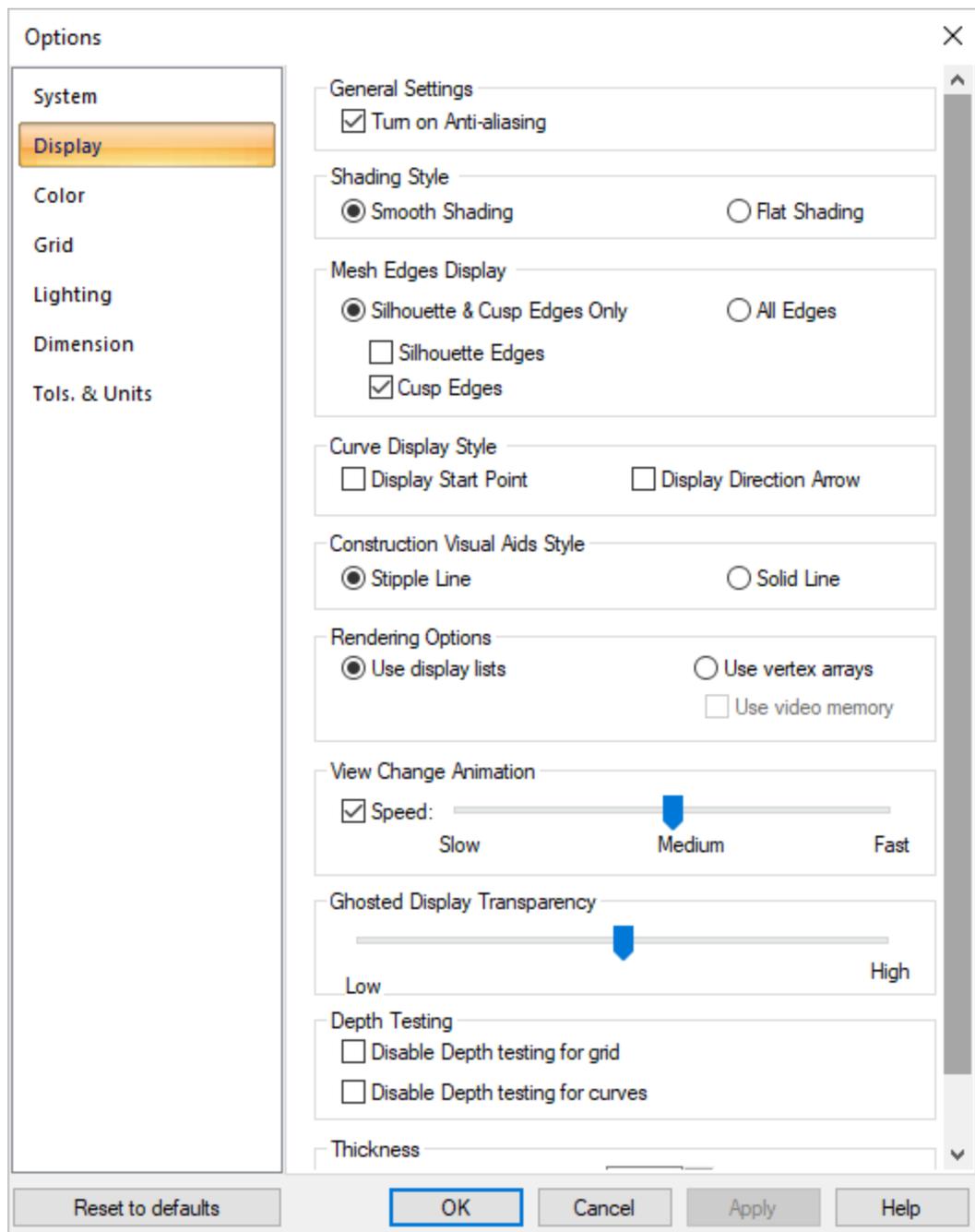
[Options](#)

1.5.2 Display

Controls how geometry is displayed. Select from the following:



Dialog Box: Options > Display



Dialog Box: Options > Display

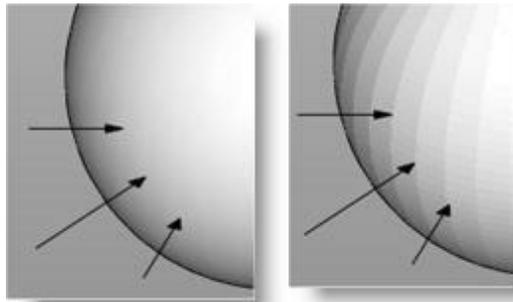
General Settings

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

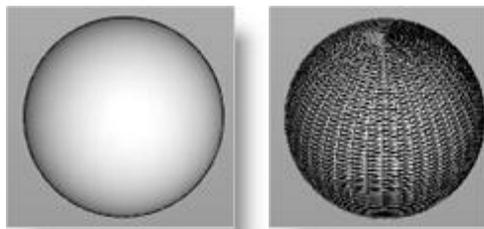
Flat Shading

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.

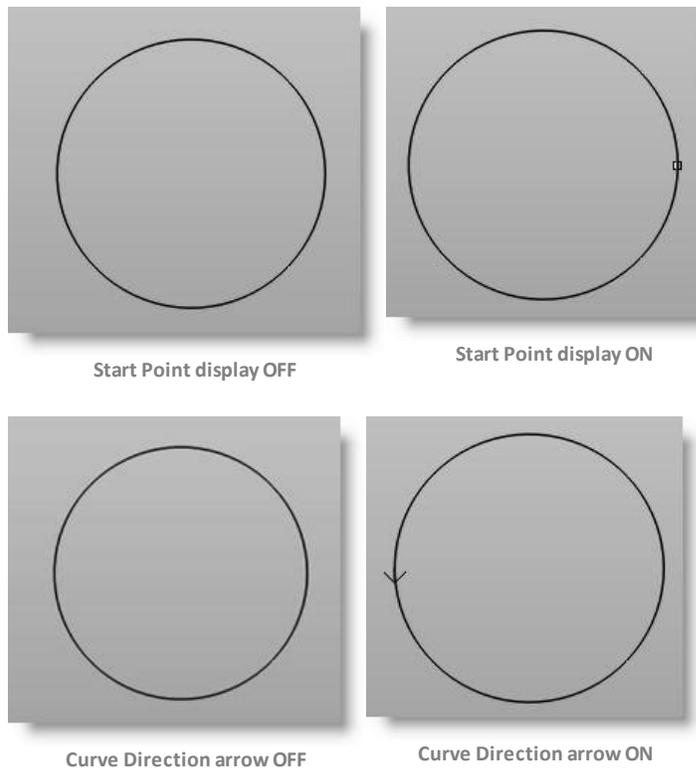


The pictures demonstrate the difference between silhouette & cusp edges only and all edge display. The left picture displays only sharp edges (there are none on a sphere); meanwhile, the right picture displays all the edges. The default is [Silhouette & Cusp Edges Only](#). When selected, this allows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



Curve Display Style

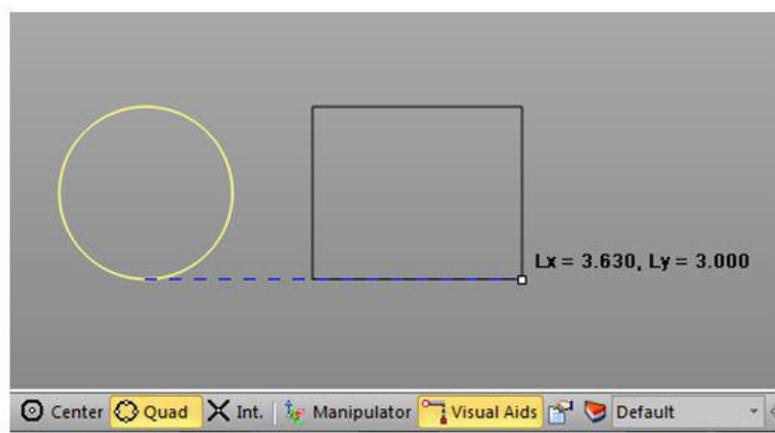
This provides an option to turn on or off display of curve start point and direction arrow.



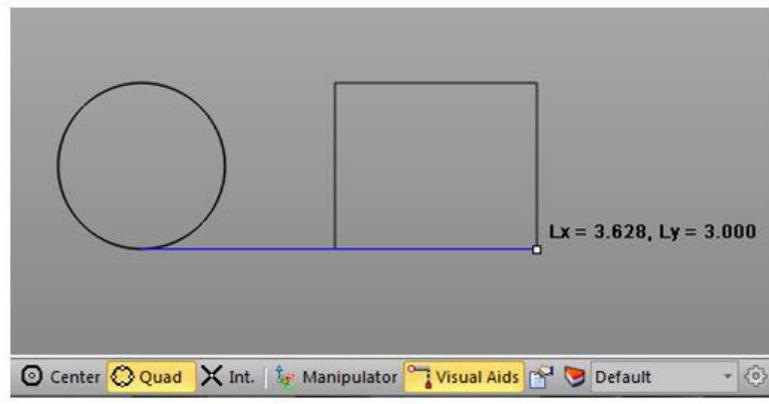
Selecting [Start Point](#) and [Curve Direction](#) arrow display under options displays both start point and curve direction arrow.

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.

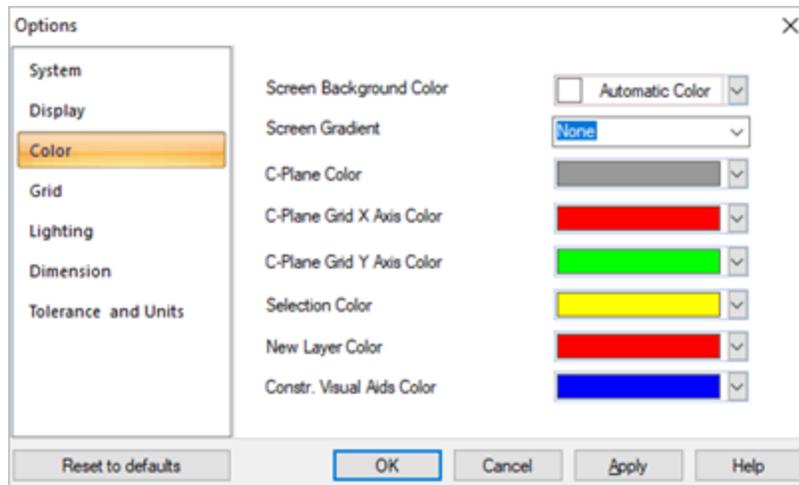
Related Topics

[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

The screenshot shows the 'Options' dialog box with the 'Grid' tab selected. The 'Position of Grid Origin' section has four radio buttons: 'Center' (selected), 'Lower Left', 'Lower Right', 'Upper Left', and 'Upper Right'. A 'Pick Point' button is located below these options. The 'Grid Extents in X Direction' is set to 800, 'Grid Extents in Y Direction' is set to 800, 'Distance between each Minor Grid Line' is set to 5, and '# of Divisions between Each Major Line' is set to 6. The dialog has buttons for 'Reset to defaults', 'OK', 'Cancel', 'Apply', and 'Help'.

Dialog Box: Options > Grid



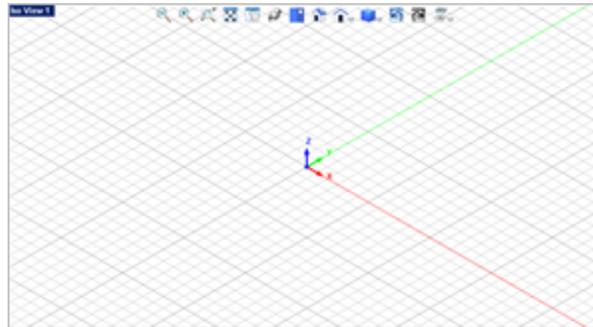
Position of Grid Origin

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

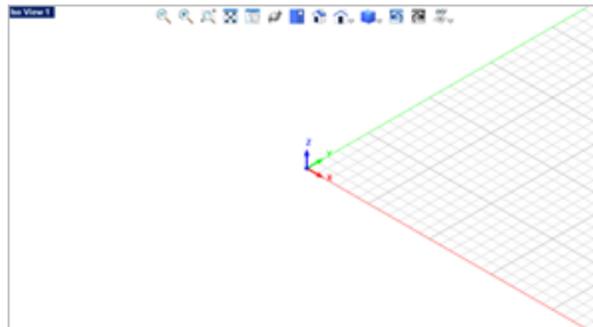
Move the center of the grid to the [WCS](#).



Move the center of the grid to the WCS.

Lower Left

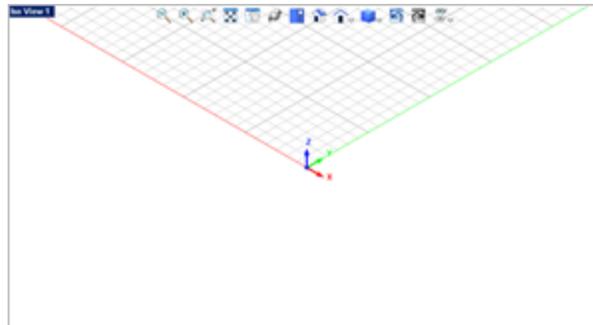
Move the lower left corner of the grid to the [WCS](#).



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

Lower Right

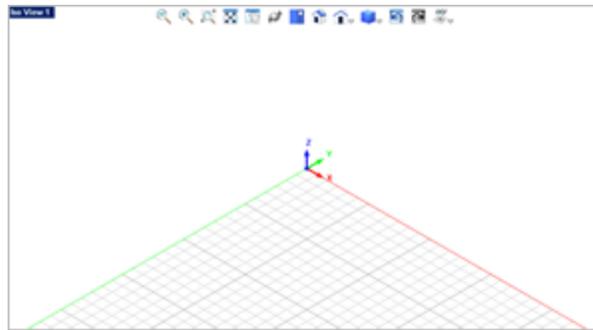
Move the lower right corner of the grid to the [WCS](#).



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

Upper Left

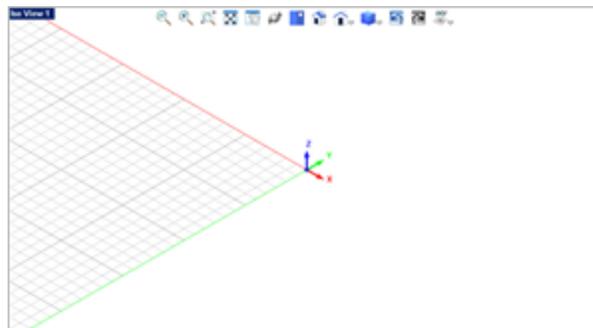
Move the upper left corner of the grid to the [WCS](#).



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

Upper Right

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



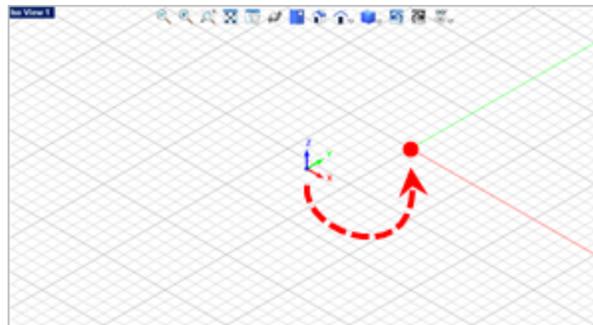
Move the upper right corner of the grid to the WCS.

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

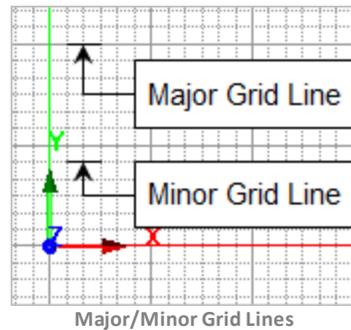
This controls how many major divisions are displayed along the X axis direction.

Grid Extents in Y Direction

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

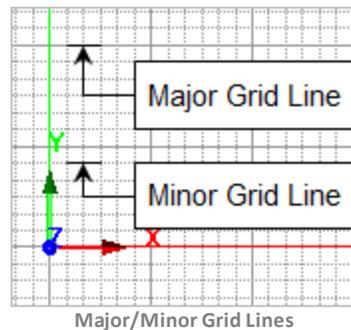
Distance between each Minor Grid Line

This controls the distance between each minor grid line.



of Divisions between Each Major Line

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



Related Topics

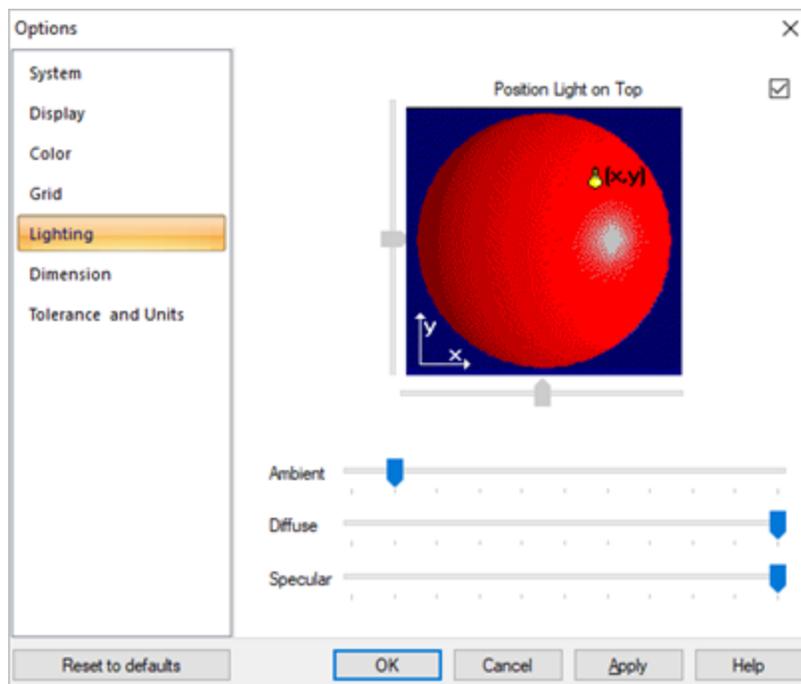
[Options](#)

1.5.5 Lighting

This controls the characteristics of lighting for the model.

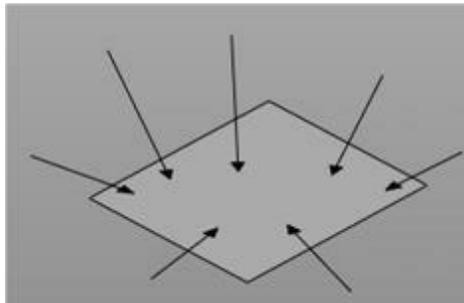


[Dialog Box: Options > Lighting](#)



Dialog Box: Options > Lighting

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

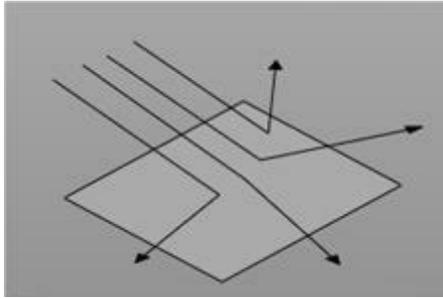


Position Light on Top

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.

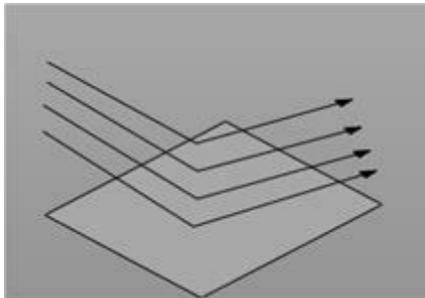
Ambient

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.



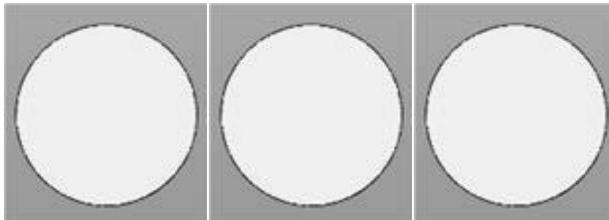
Diffuse

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

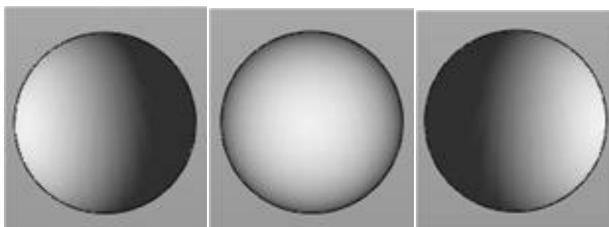


Specular

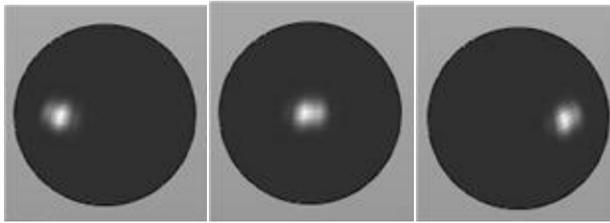
Specular light comes from a single source and is scattered in a single direction. It is reflective light. Shiny metal objects demonstrate good specular reflection. Shading is not a significant consideration with specular light.



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.

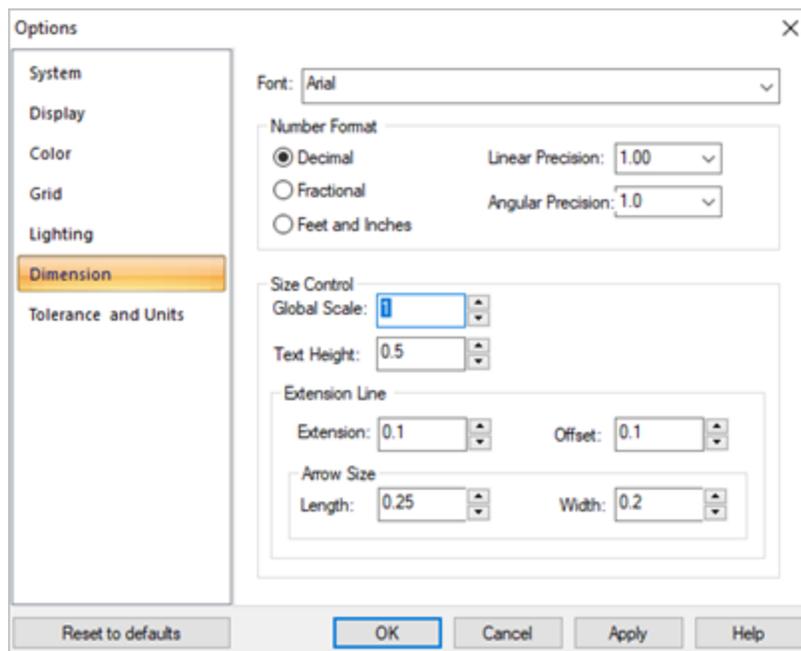
Related Topics

[Options](#)

1.5.6 Dimension

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

Dialog Box: Options > Dimension



Dialog Box: Options > Dimension

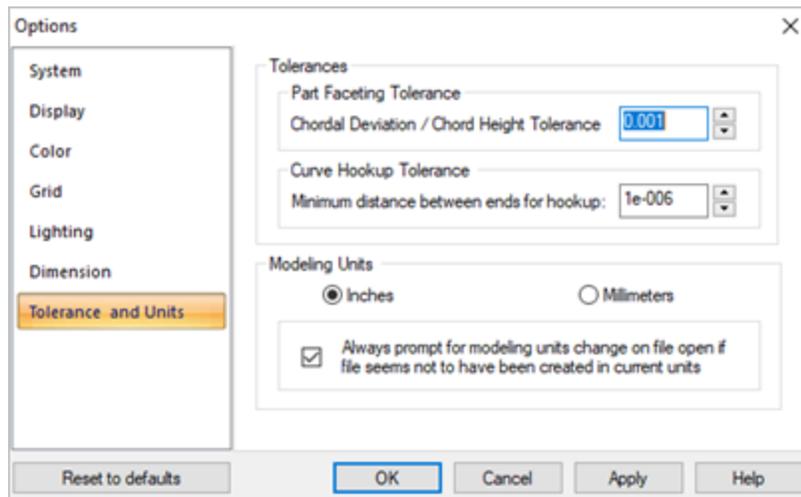
Related Topics

[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

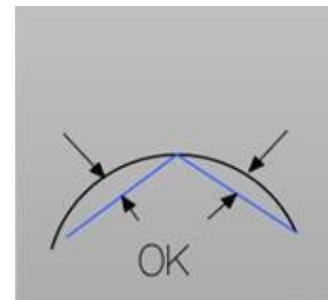
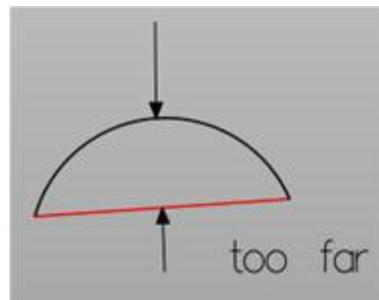
This controls the accuracy of the models. Smaller tolerance numbers often result in significantly big files and slower processing time but are more accurate.

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.

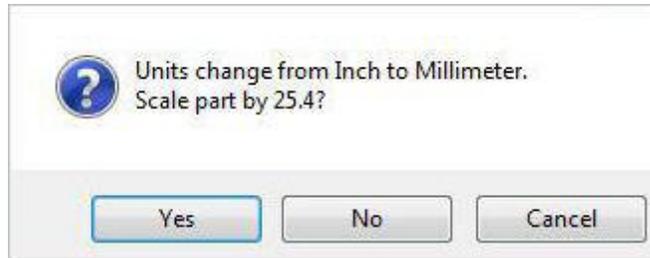


Curve Hookup Tolerance

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

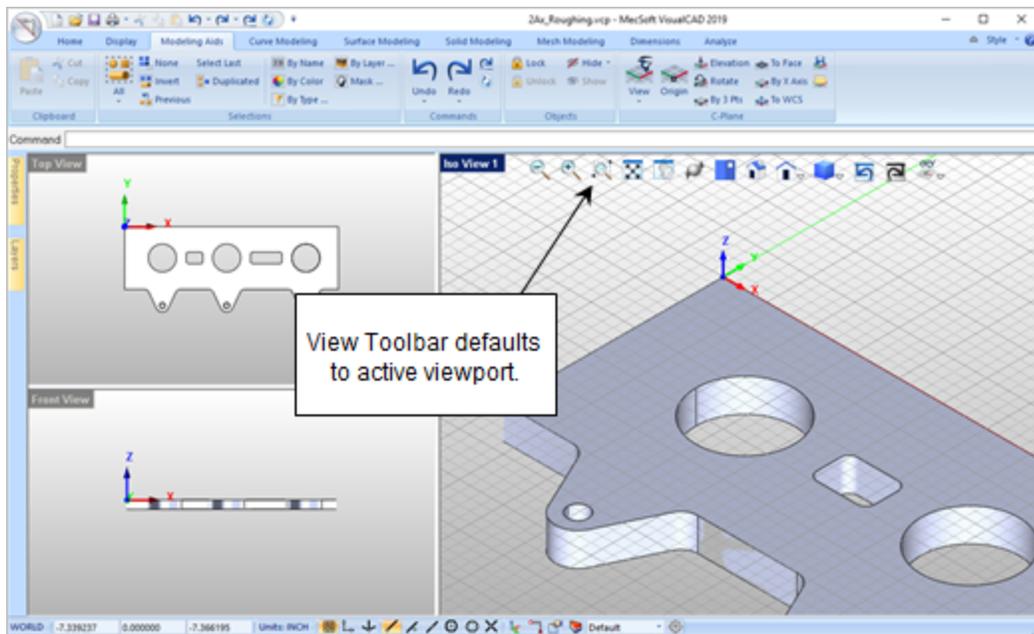
The [View Toolbar](#) contains easily accessible commands that control the appearance of the currently active view. This includes motion of the camera as well as how objects are displayed. When a view change is invoked from the [View Toolbar](#) toolbar, the existing view magnification factor is maintained.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

The View Toolbar



The View Toolbar

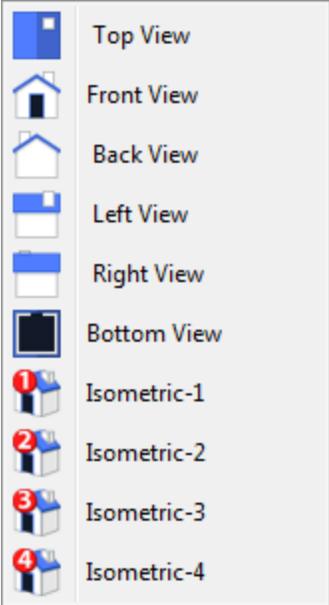
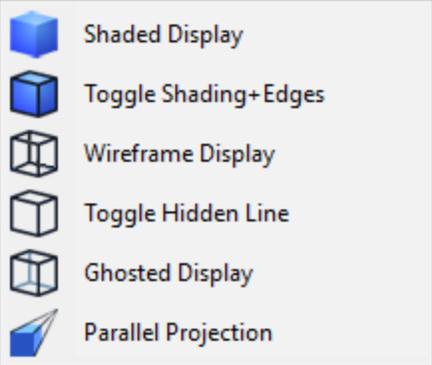
The View Toolbar Displayed in the Active Viewport



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	

	 <p style="text-align: center;">View Menu</p>	<p>Top View</p> <hr/> <p>Front View</p> <hr/> <p>Back View</p> <hr/> <p>Left View</p> <hr/> <p>Right View</p> <hr/> <p>Bottom View</p> <hr/> <p>Isometric-1,2,3,4</p>
	 <p style="text-align: center;">Display Menu</p>	<p>Shaded Display</p> <hr/> <p>Toggle Shaded+Edges</p> <hr/> <p>Wireframe Display</p> <hr/> <p>Toggle Hidden Lines</p> <hr/> <p>Ghosted Display</p> <hr/> <p>Toggle Parallel/Perspective Projection</p>
	<p>Previous View</p>	<p>These are Undo/Redo View commands.</p>
	<p>Next View</p>	
	 <p style="text-align: center;">Show/Hide Menu</p>	<p>Hide/Show Grid</p> <hr/> <p>Hide/Show WCS</p>

 [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/CAM 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.

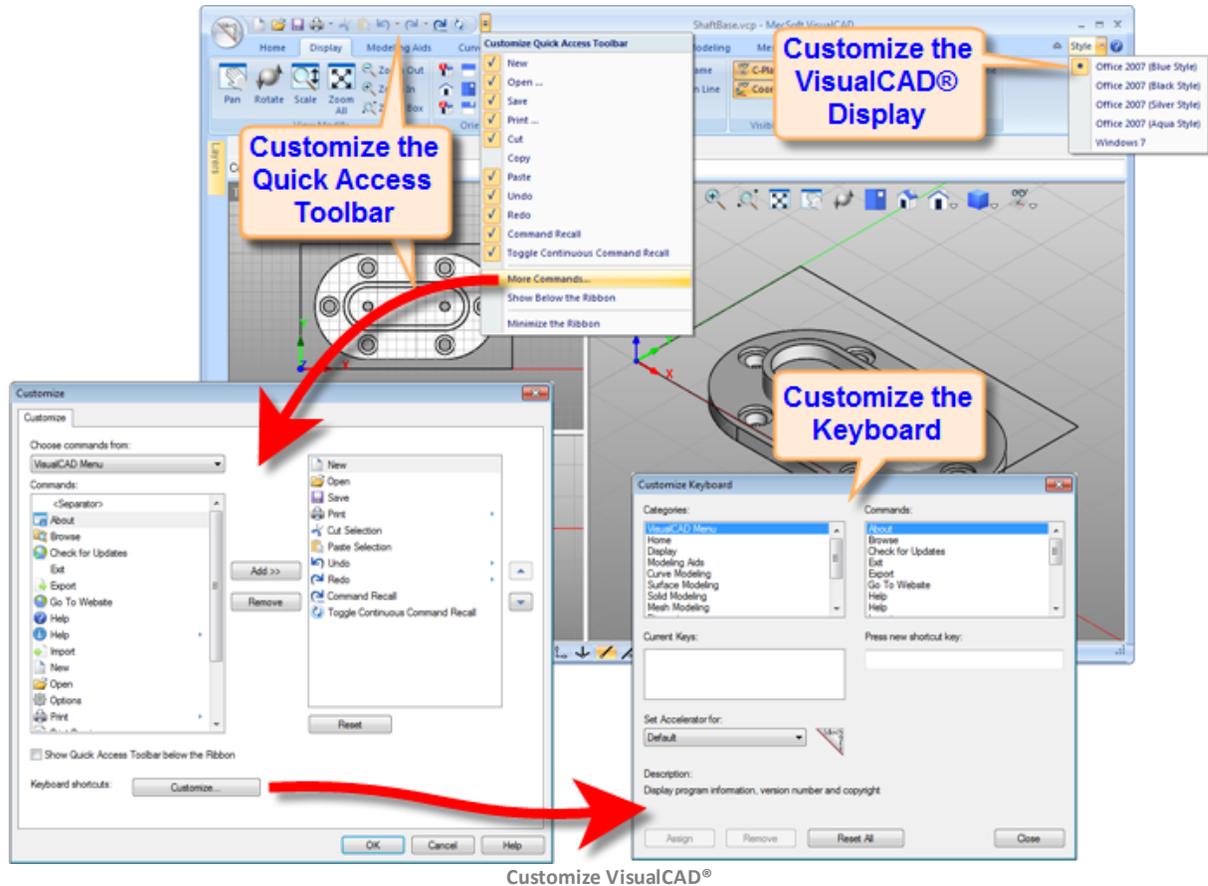


Related Topics

[View Tool Bar](#)

1.7 Customize VisualCAD

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



Related Topics

[Customize Quick Access Toolbar](#)

[Minimize Ribbon Bar](#)

[Preset <Alt> Accelerator Keys](#)

[Customize Dialog](#)

[Customize the Keyboard](#)

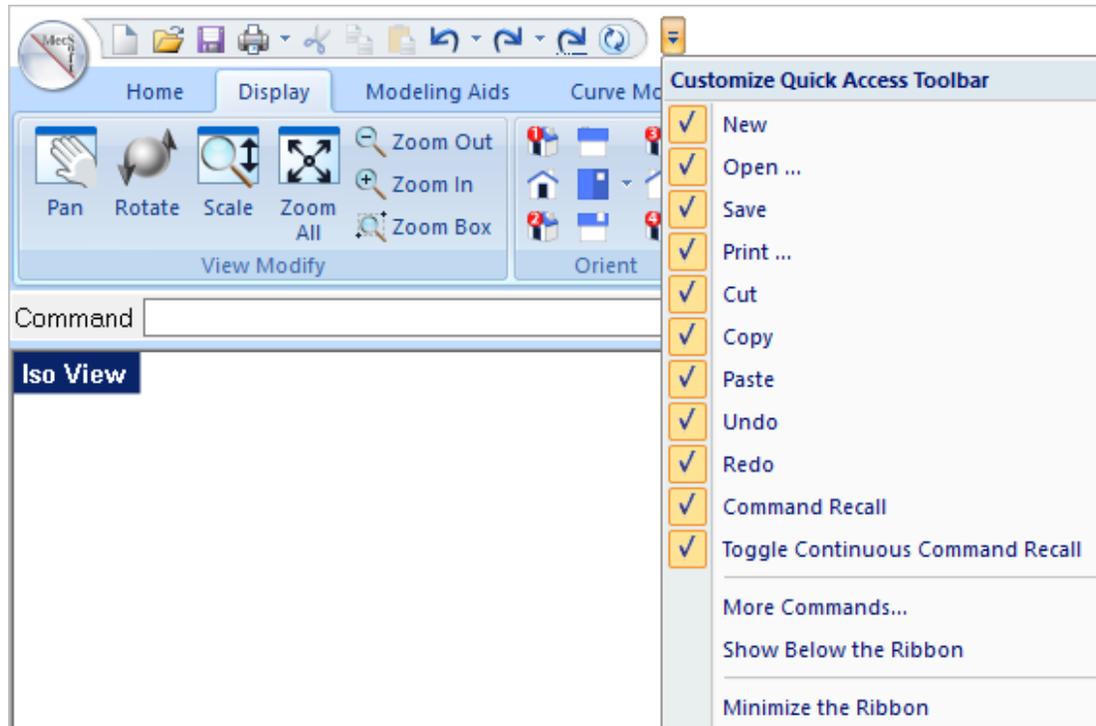
[Styles Menu](#)

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

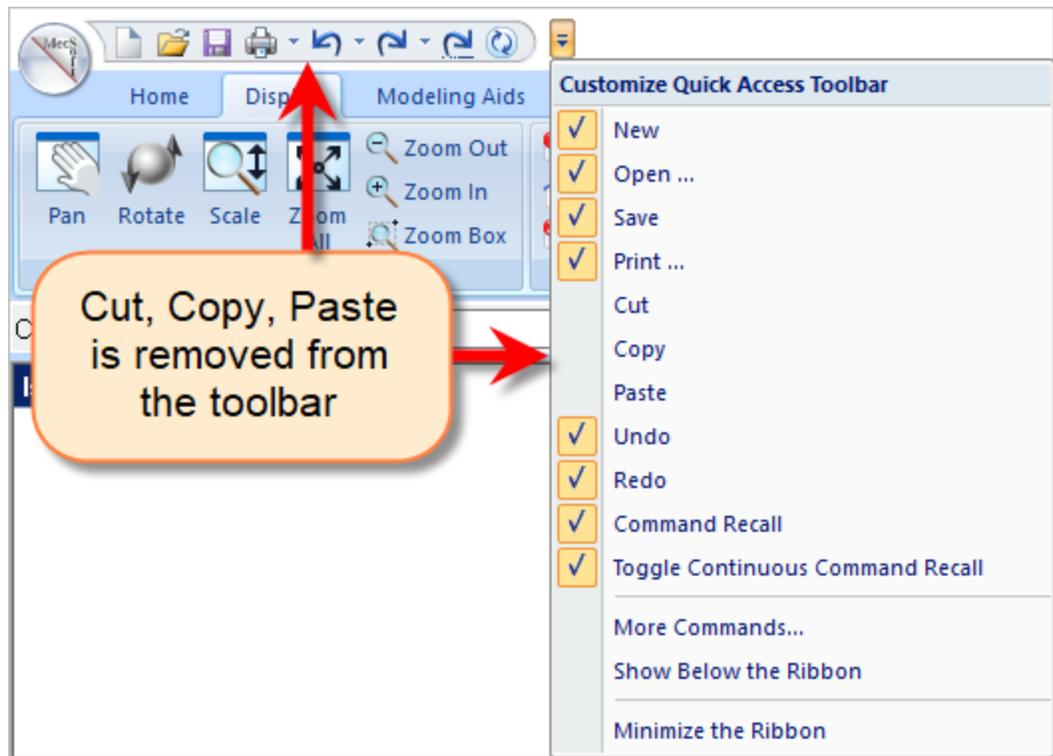
Select the indicator  to the right of the [Quick Access Toolbar](#) to display the menu:



Displaying the Quick Access Toolbar Menu

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.



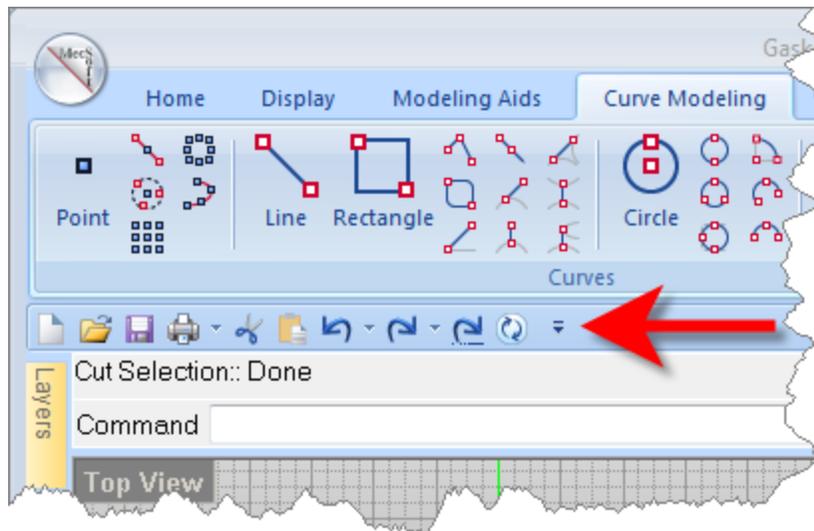
Quickly Turn Commands On/Off from the Quick Access Toolbar

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

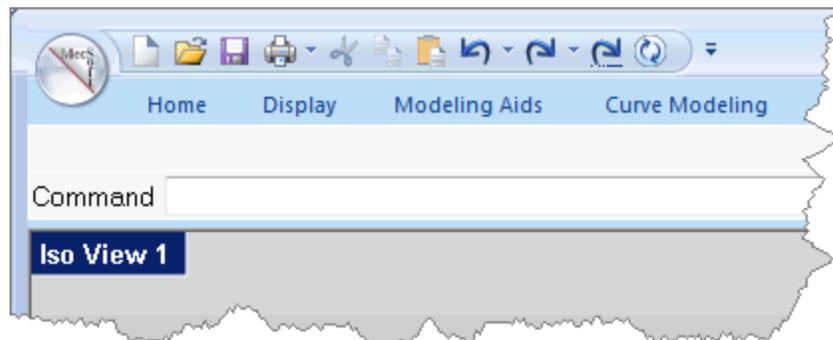
You can move the [Quick Access Toolbar](#) to reside below the [Ribbon](#) bar by selecting [Show Below Ribbon](#) from the [Customize Quick Access Toolbar](#) menu. You can also do this by selecting appropriate check box in the [Customize](#) dialog.



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/CAM](#)

[Customize Quick Access Toolbar](#)

[Preset <Alt> Accelerator Keys](#)

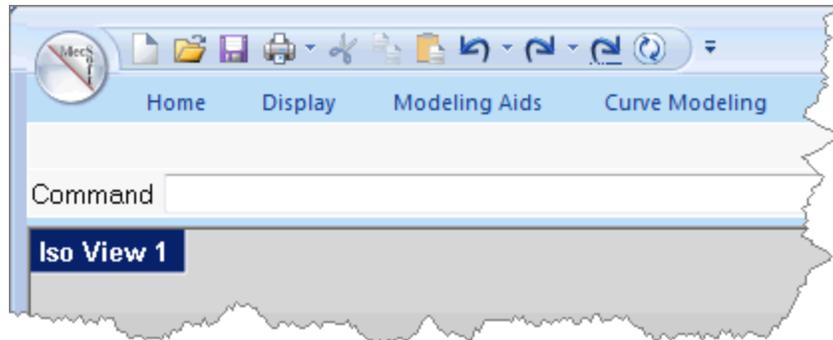
[Customize Dialog](#)

[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/CAM](#)

[Customize Quick Access Toolbar](#)

[Preset <Alt> Accelerator Keys](#)

[Customize Dialog](#)

[Customize the Keyboard](#)

[Styles Menu](#)

1.7.3 Preset <Alt> Accelerator Keys

VisualCAD/CAM 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/CAM will display (and activate) **<Alt>** key assignments for the **VisualCAD/CAM 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.

- 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: Menu

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

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



Preset <Alt> Keys: Menu

Preset <Alt> Keys: Home Ribbon Bar

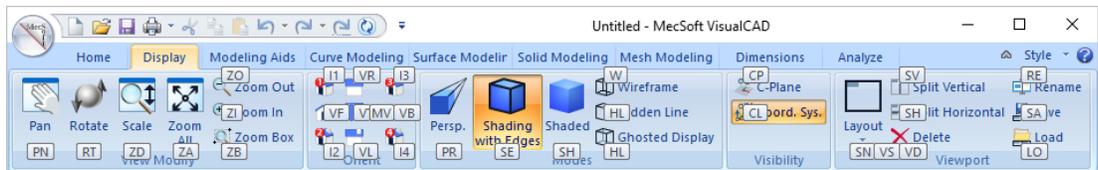
The following preset **<Alt>** Accelerator key combinations are assigned to commands on the **Home Ribbon Bar**.



Preset <Alt> Keys: Home Ribbon Bar

Preset <Alt> Keys: Display Ribbon Bar

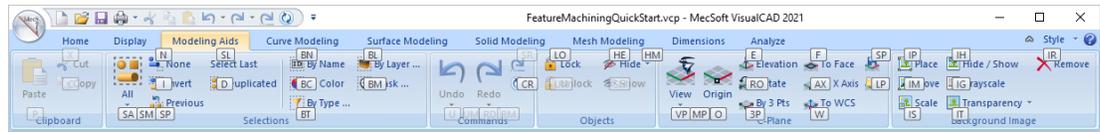
The following preset **<Alt>** Accelerator key combinations are assigned to commands on the **Display Ribbon Bar**.



Preset <Alt> Keys: Display Ribbon Bar

Preset <Alt> Keys: Modeling Aids Ribbon Bar

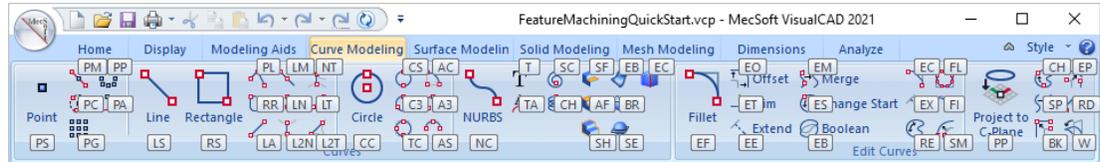
The following preset **<Alt>** Accelerator key combinations are assigned to commands on the **Modeling Aids Ribbon Bar**.



Preset <Alt> Keys: Modeling Aids Ribbon Bar

Preset <Alt> Keys: Curve Modeling Ribbon Bar

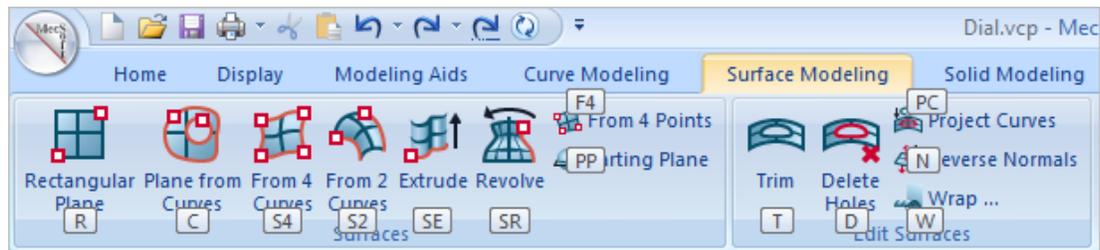
The following preset <Alt> Accelerator key combinations are assigned to commands on the Curve Modeling Ribbon Bar.



Preset <Alt> Keys: Curve Modeling Ribbon Bar

Preset <Alt> Keys: Surface Modeling Ribbon Bar

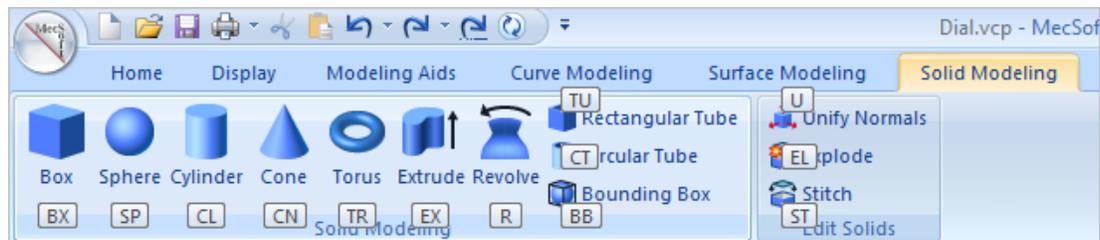
The following preset <Alt> Accelerator key combinations are assigned to commands on the Surface Modeling Ribbon Bar.



Preset <Alt> Keys: Surface Modeling Ribbon Bar

Preset <Alt> Keys: Solid Modeling Ribbon Bar

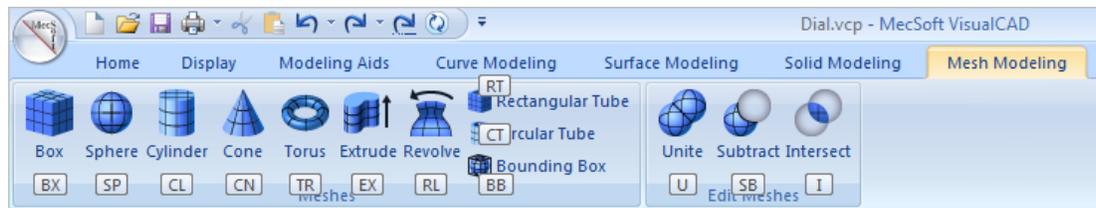
The following preset <Alt> Accelerator key combinations are assigned to commands on the Solid Modeling Ribbon Bar.



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.

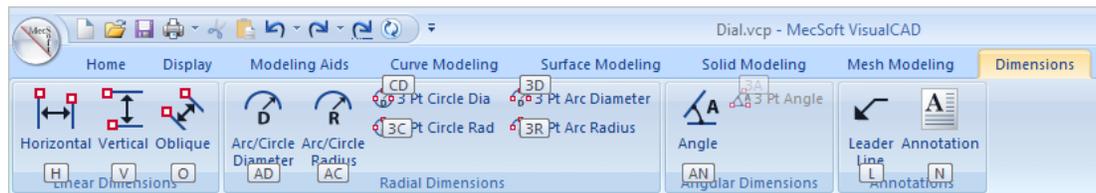


Preset <Alt> Keys: Mesh Modeling Ribbon Bar



Preset <Alt> Keys: Dimensions Ribbon Bar

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

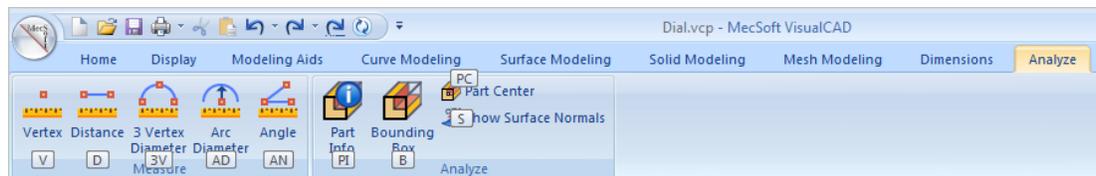


Preset <Alt> Keys: Dimensions Modeling Ribbon Bar



Preset <Alt> Keys: Analyze Ribbon Bar

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

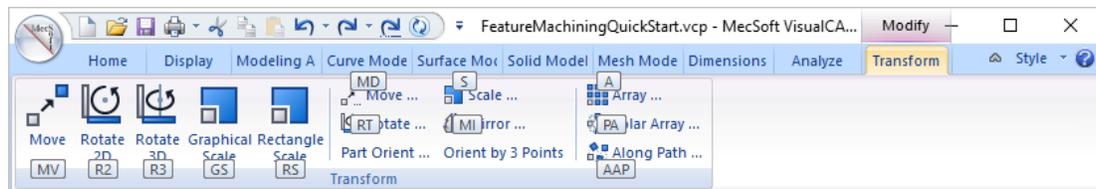


Preset <Alt> Keys: Analyze Ribbon Bar



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/CAM](#)

[Customize Quick Access Toolbar](#)

[Minimize Ribbon Bar](#)

[Customize Dialog](#)

[Customize the Keyboard](#)

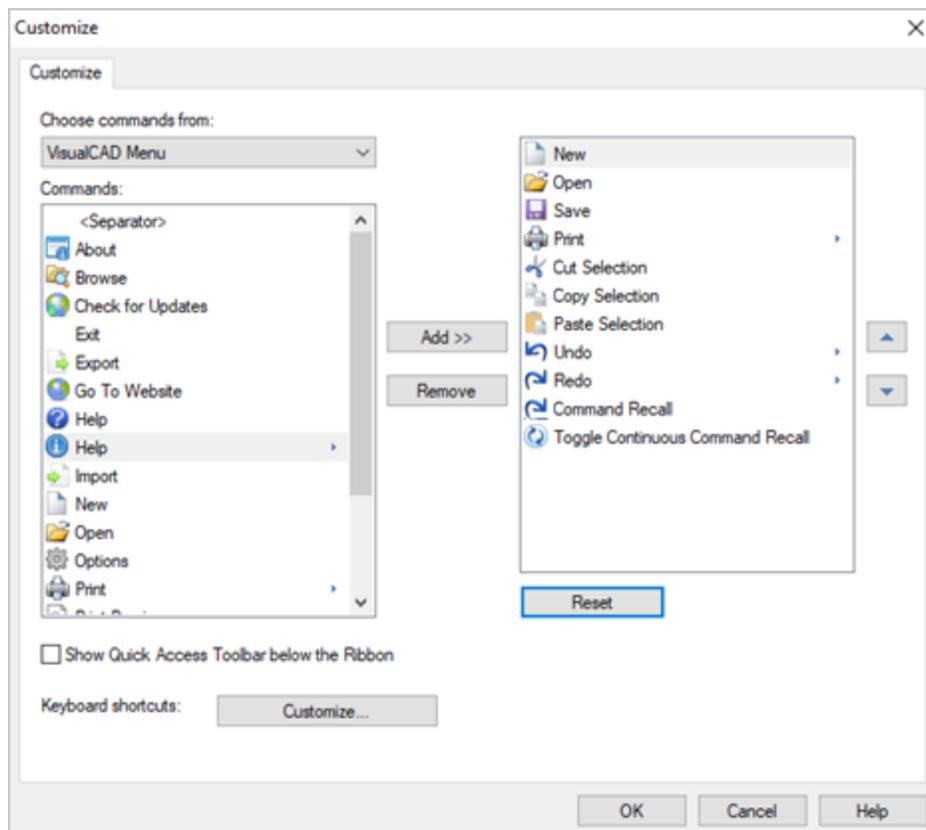
[Styles Menu](#)

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/CAM 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/CAM 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:

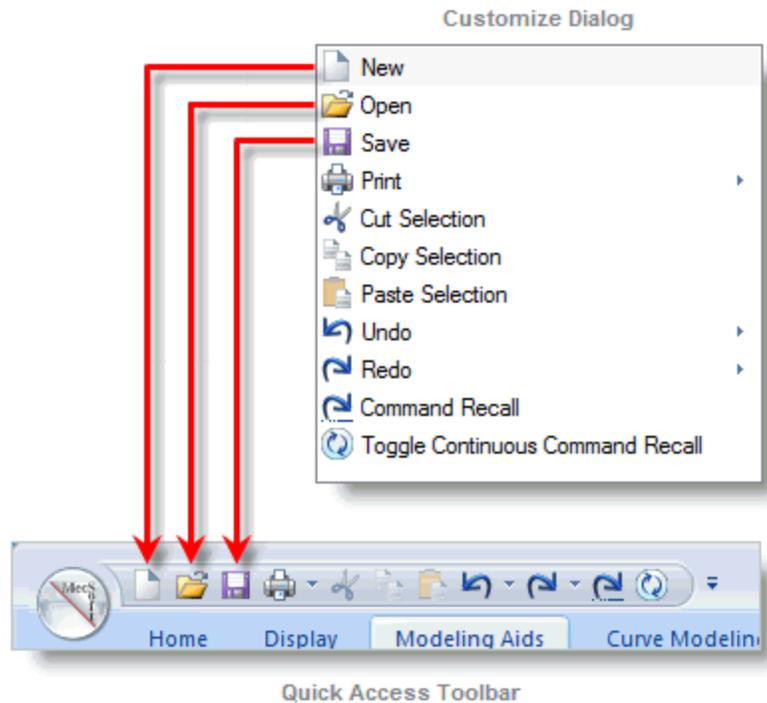
This window lists all of the commands on the Ribbon bar selected from the [Choose commands from:](#) list. Select a command from the list and then pick the [Add >>](#) button to add it to the list on the right.

Add >> / Remove

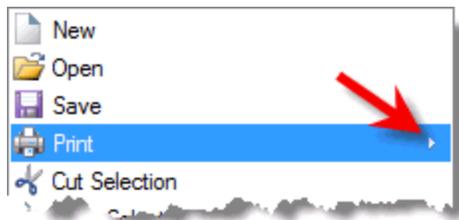
The [Add >>](#) button adds a selected command from the [Commands:](#) list on the left to the list on the right. The [Remove](#) button removes a selected command from the list on the right.

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](#).



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

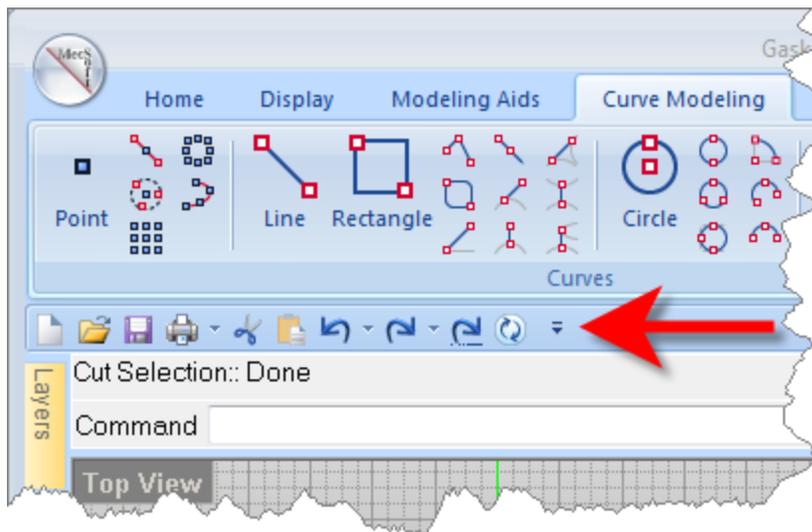
  Pick the **Up** or **Down** buttons to move a selected command higher (to the left) or lower (to the right) on the [Quick Access Toolbar](#).

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/CAM](#)

[Customize Quick Access Toolbar](#)

[Minimize Ribbon Bar](#)

[Preset <Alt> Accelerator Keys](#)

[Customize Dialog](#)

[Styles Menu](#)

1.7.5 Customize the Keyboard

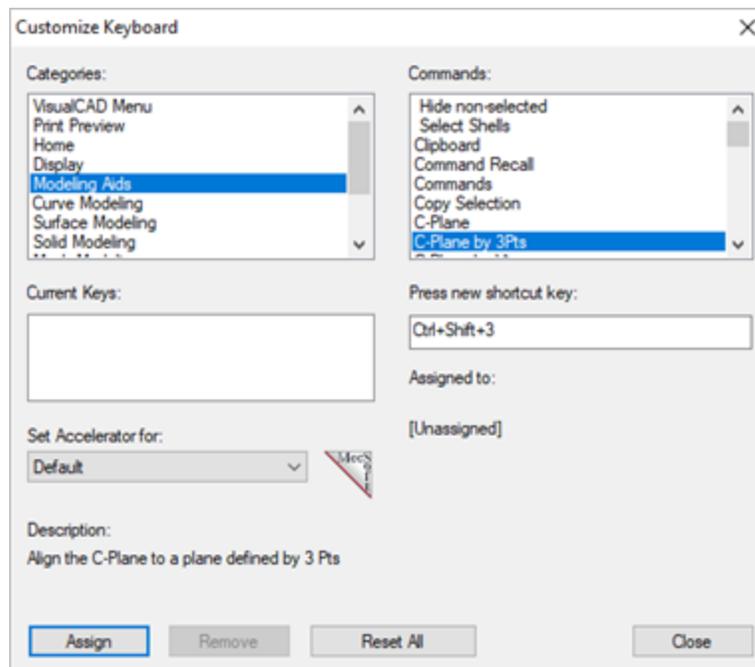
Invoked by the [Customize dialog](#), the [Customize Keyboard](#) dialog shown below allows you to redefine the [Default Shortcut Keys](#) assigned to your keyboard. Keyboard shortcuts are combinations of keys that can be pressed to execute a [VisualCAD/CAM](#) command. Pressing the shortcut keys does 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/CAM](#).



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/CAM](#) will then execute the

command when those shortcut keys are pressed.



Related Topics

[Customize VisualCAD/CAM](#)

[Customize Quick Access Toolbar](#)

[Minimize Ribbon Bar](#)

[Preset <Alt> Accelerator Keys](#)

[Customize Dialog](#)

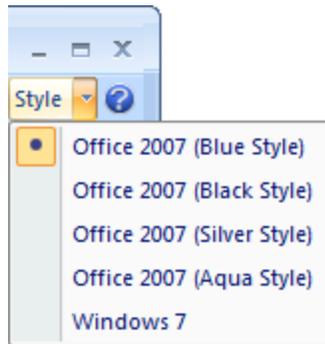
[Styles Menu](#)

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.



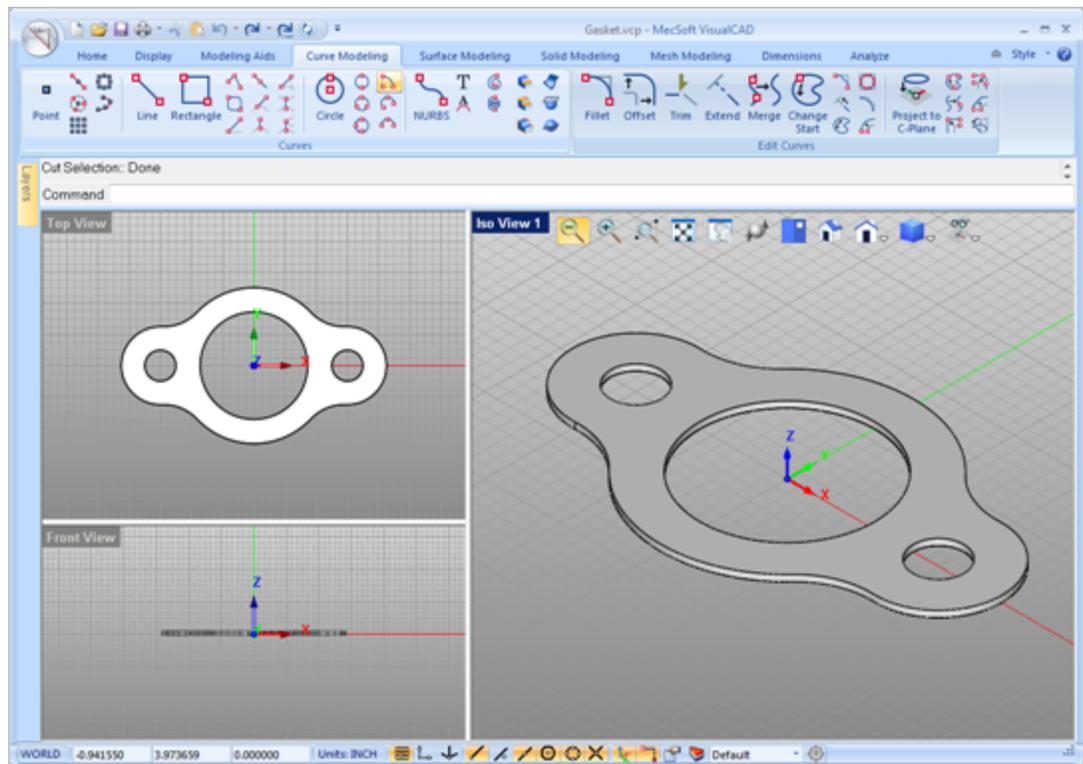
The Styles Menu



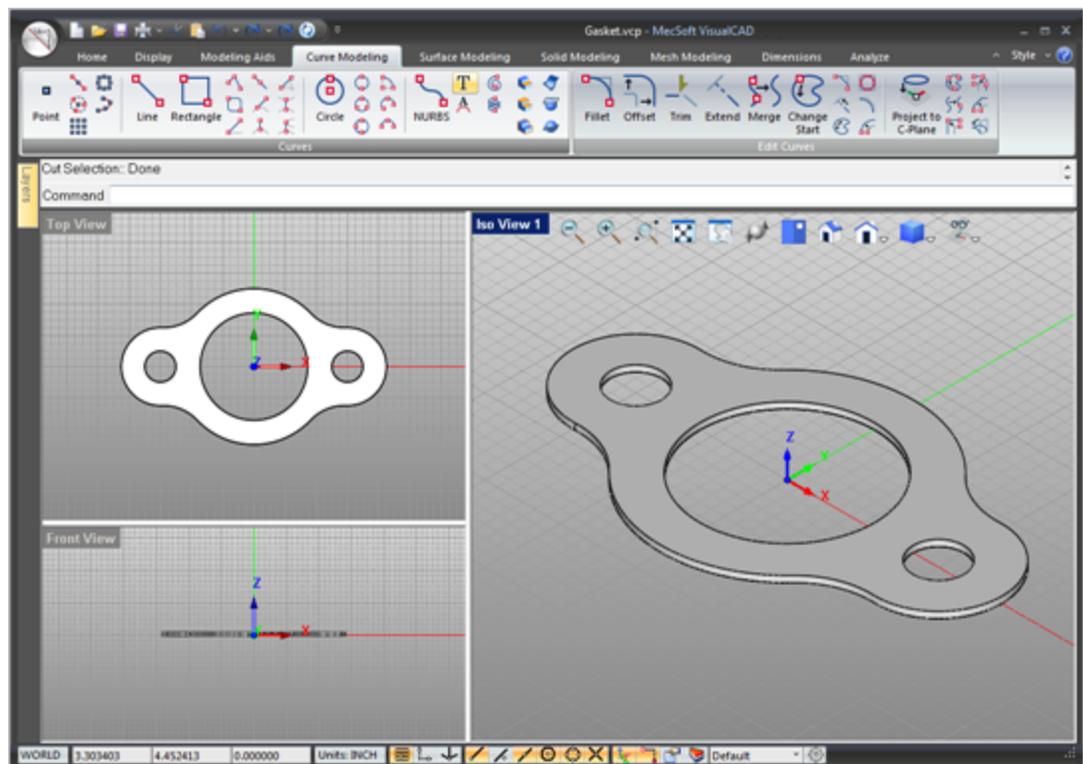
Office 2007 Blue



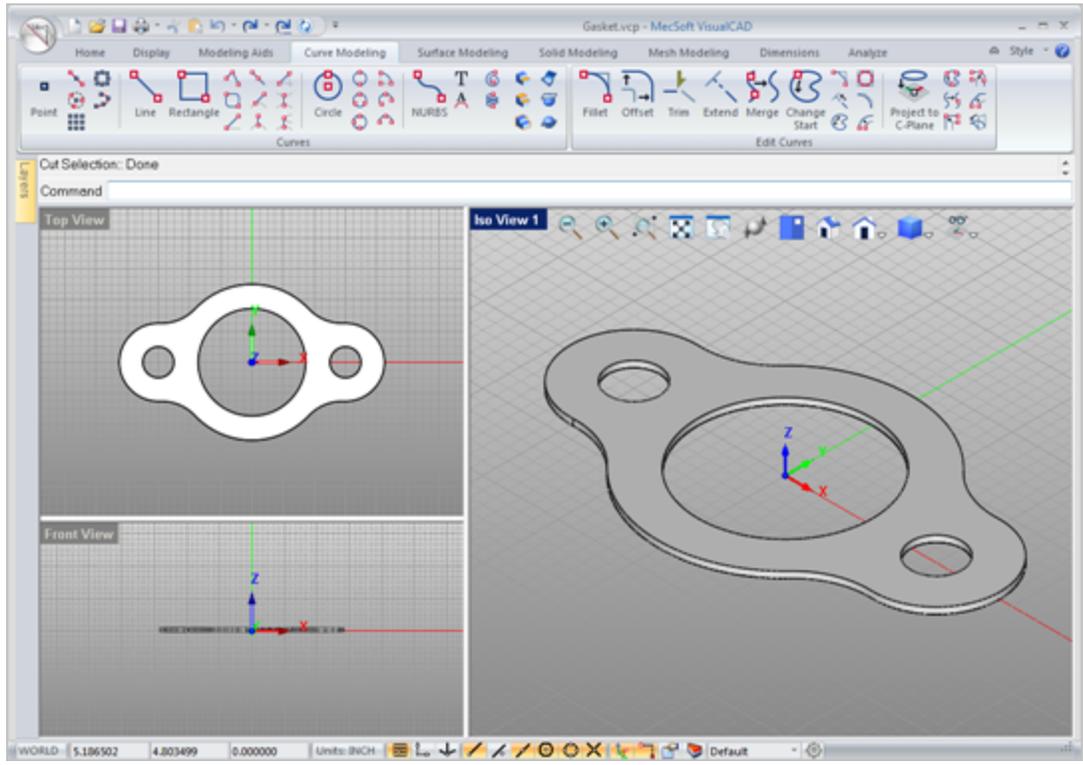
Office 2007 (Blue Style)



 Office 2007 (Black Style)

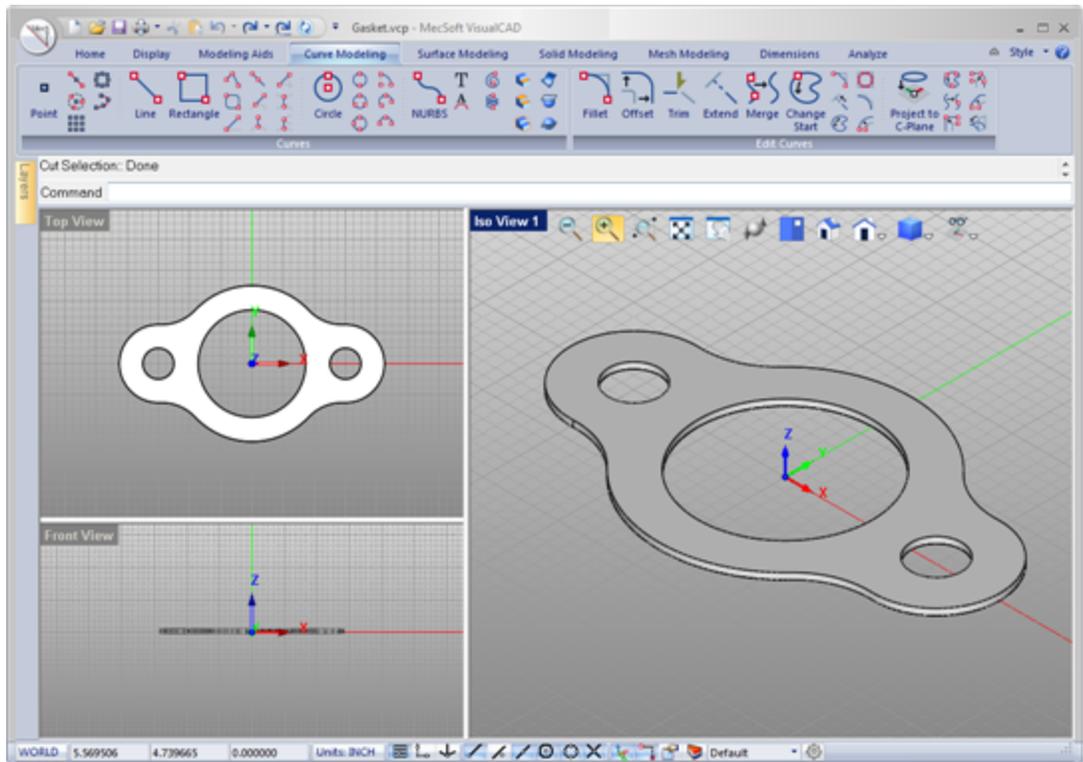


 Office 2007 (Silver Style)



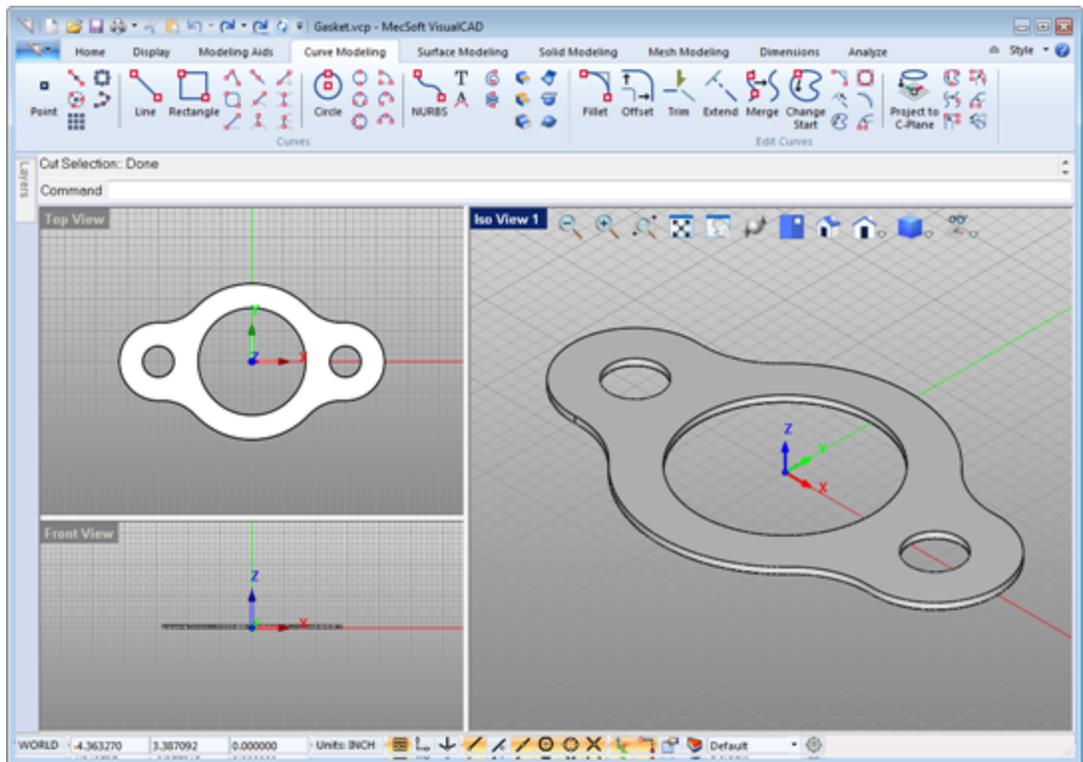
Office 2007 (Silver Style)

 Office 2007 (Aqua Style)



Office 2007 (Aqua Style)

Windows 7



Windows 7



Related Topics

[Customize VisualCAD/CAM](#)

[Customize Quick Access Toolbar](#)

[Minimize Ribbon Bar](#)

[Preset <Alt> Accelerator Keys](#)

[Customize Dialog](#)

[Customize the Keyboard](#)

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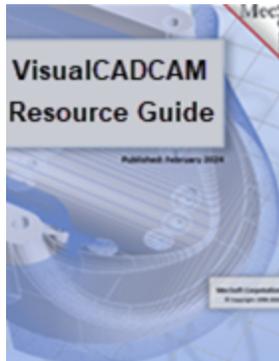
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About this Guide

MecSoft is very excited to provide this FREE 200-page guide to using VisualCAD, MecSoft's free CAD program! VisualCAD can be download as part of VisualCAD/CAM. This guide includes 8 detailed exercises with over 300 detailed graphical illustrations that will get you up and running with VisualCAD in no time at all!

Here's what's inside:

- [Download the completed source files here!](#)
- **VisualCAD Preferences**
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.
- **2D Drawing & Dimensioning**
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.
- **3D Modeling**
3D modeling exercises take you deeper into modeling in 3D with VisualCAD. You will learn how to draw complex profiles and then extrude and revolve them to build complex 3D features from multiple viewports. How to use VisualCAD's Graphic Manipulator to build and position components and features. You will also learn how to create cross sections at any point in the 3D model.
- **Construction Planes**
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.



Exercises:

Here is a list of the exercises in this guide:

1. [VisualCAD Preferences](#)
2. [2D Drawing & Dimensioning](#)
3. [Model a Spanner](#)
4. [Model a Base Plate](#)
5. [Model a Mold Insert](#)
6. [Model a Connector Block](#)

7. [Model a Daisy Decor](#)
8. [Using Construction Planes](#)

#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:

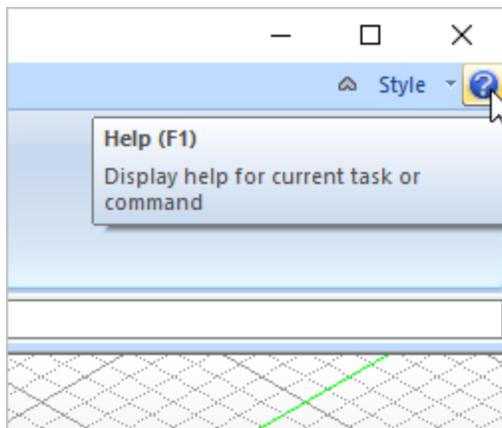
1. [Access the online help.](#)
2. [Set the Display Style.](#)
3. [Set the Units to Inches.](#)
4. [Set System Options.](#)
5. [Set to Quad Viewports.](#)
6. [Viewing the Command Prompts.](#)
7. [The Status Bar.](#)

5.1 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.

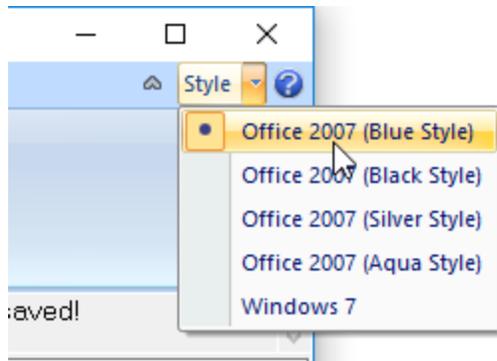


5.2 Set the Display Style



Set the Display Style

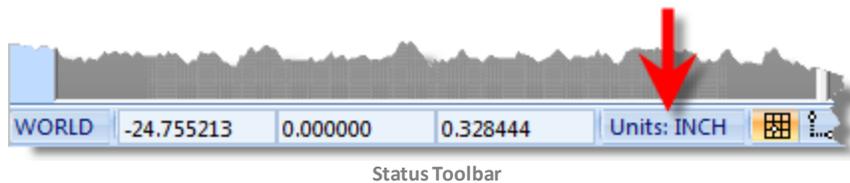
VisualCAD allows you to change the [Style](#) of the display. At the very top right of the VisualCAD display window, select the [Style](#) drop-down menu and select [Office 2007 \(Blue Style\)](#) ([Blue Style](#)).



5.3 Set the Units to Inches

Set the Units to Inches

Open [VisualCAD](#) and check to see if the part unit is set to Inch.



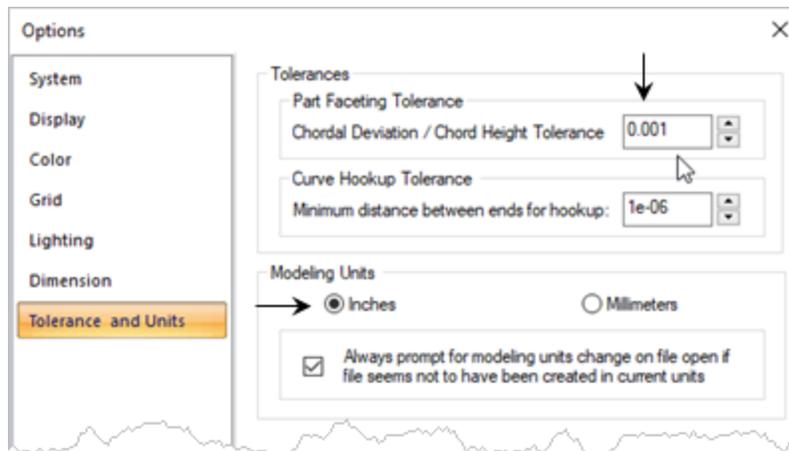
If [Units](#) is set to [MM](#), double-click on it to change to [INCH](#).

5.4 Set Systems Options

Set Systems Options

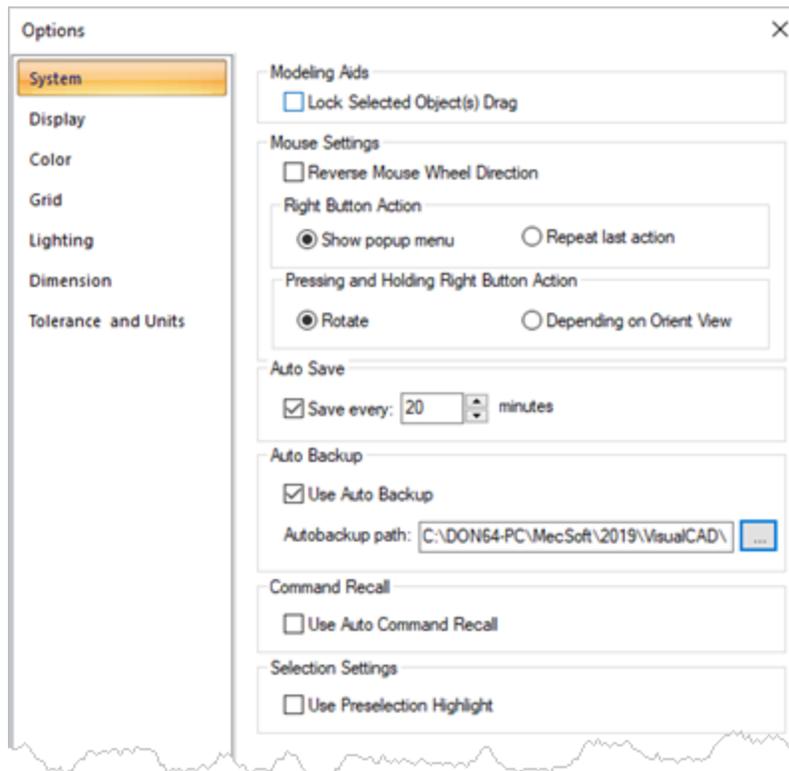
In this exercise you will learn how to set some of the systems options in [VisualCAD](#).

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](#).

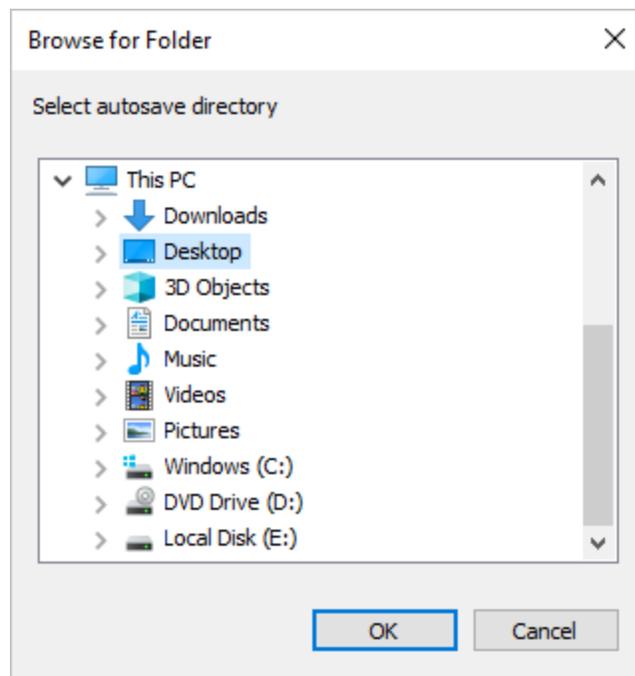


System Options Dialog

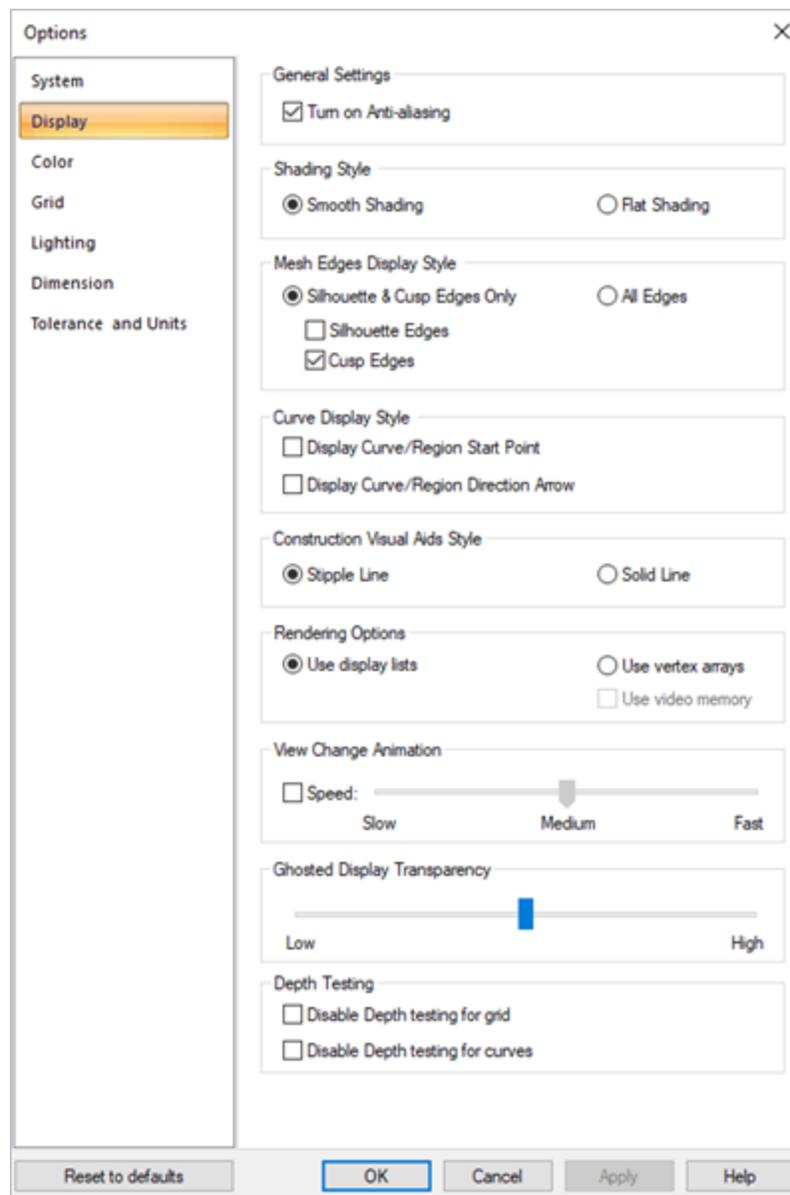
6. Select **System** from the left side of the dialog.



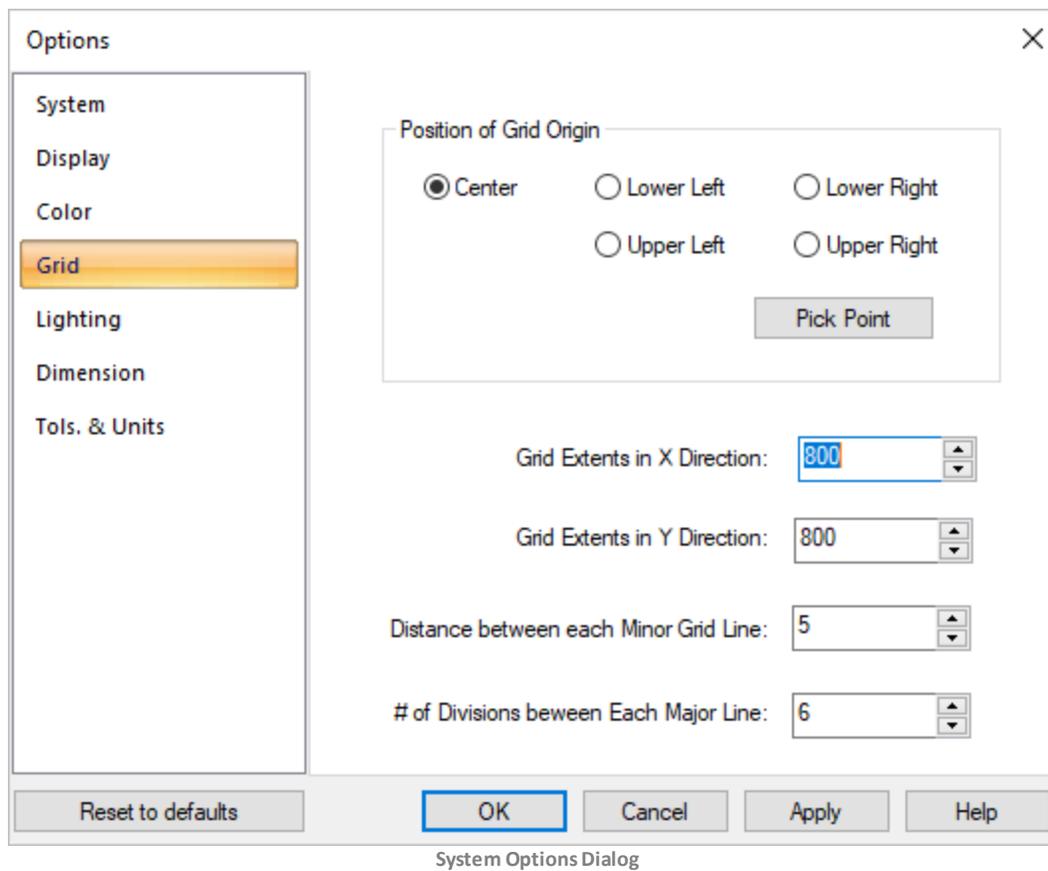
7. Under **Auto Save**, check the box and then set the value to save every **10** minutes.
8. Check the box to use **Auto Backup** and then select the **...** box to display the **Browse for Folder** dialog.
9. Select the folder where you want backup files to be saved and then pick **OK**.



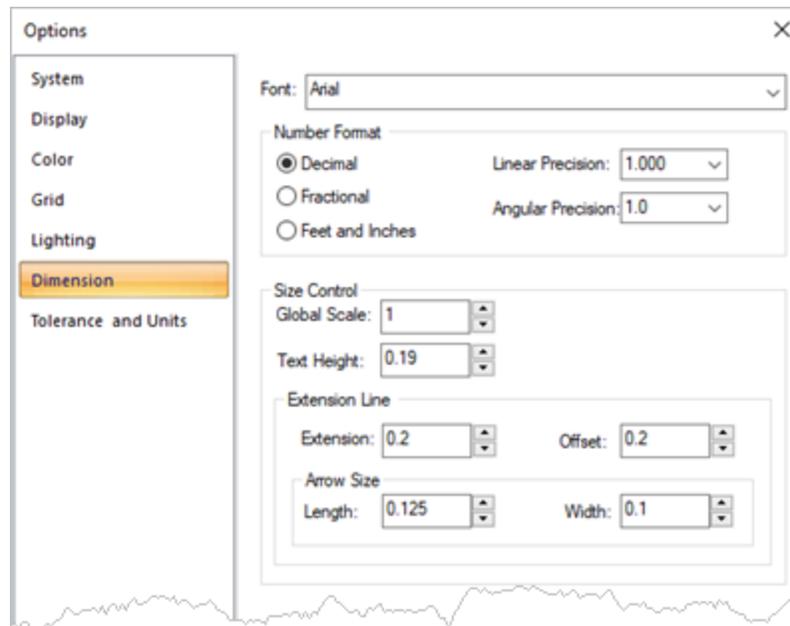
10. Set the remaining [System Options](#) as shown in the dialog above.
11. Now select [Display](#) from the left side of the dialog.
12. Under [View Change Animation](#) uncheck the box next to [Speed](#).



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.



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.



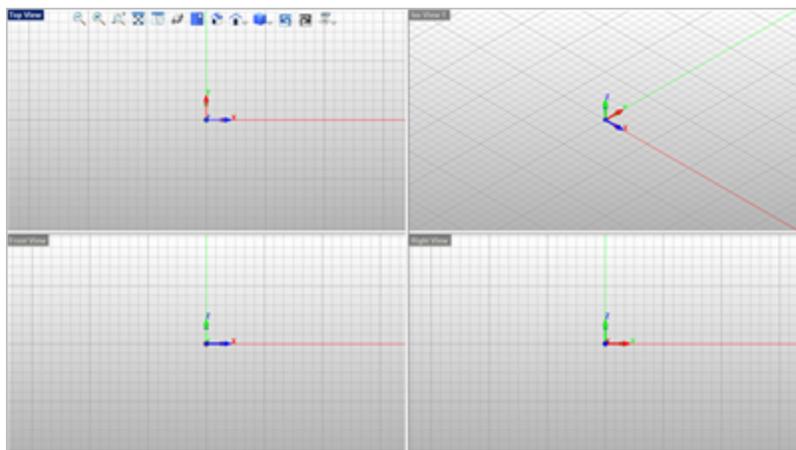
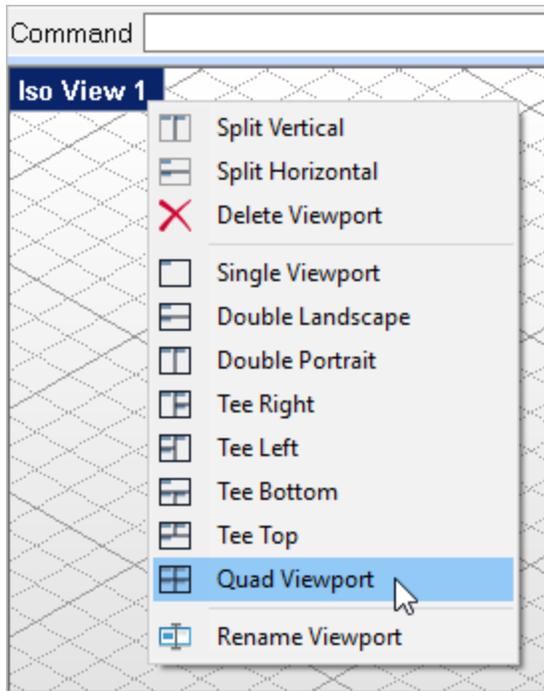
5.5 Set to Quad Viewports

Set to Quad Viewports

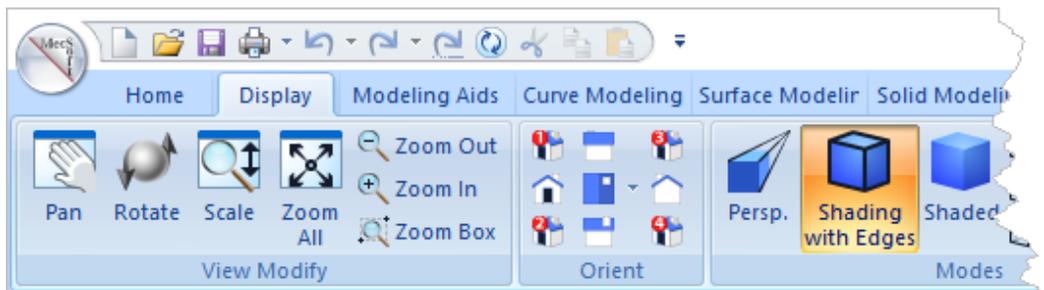
It is much faster to draw and model in [VisualCAD](#) if you have multiple [Viewports](#) displayed.

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.

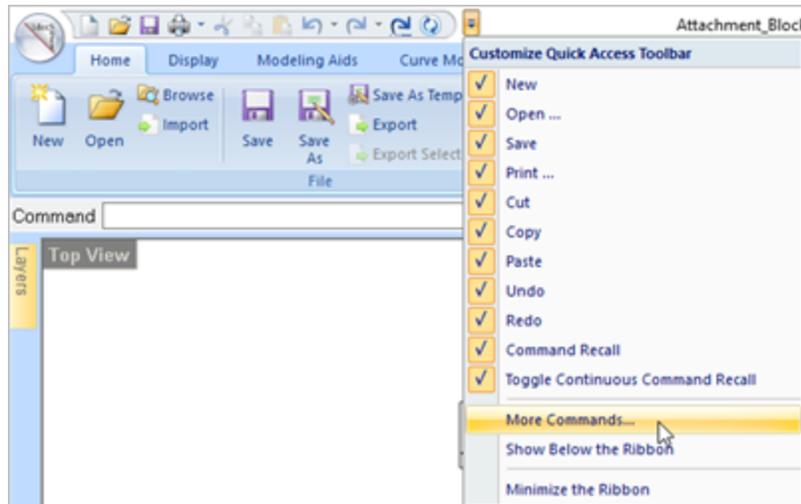


5.6 The Quick Access Toolbar

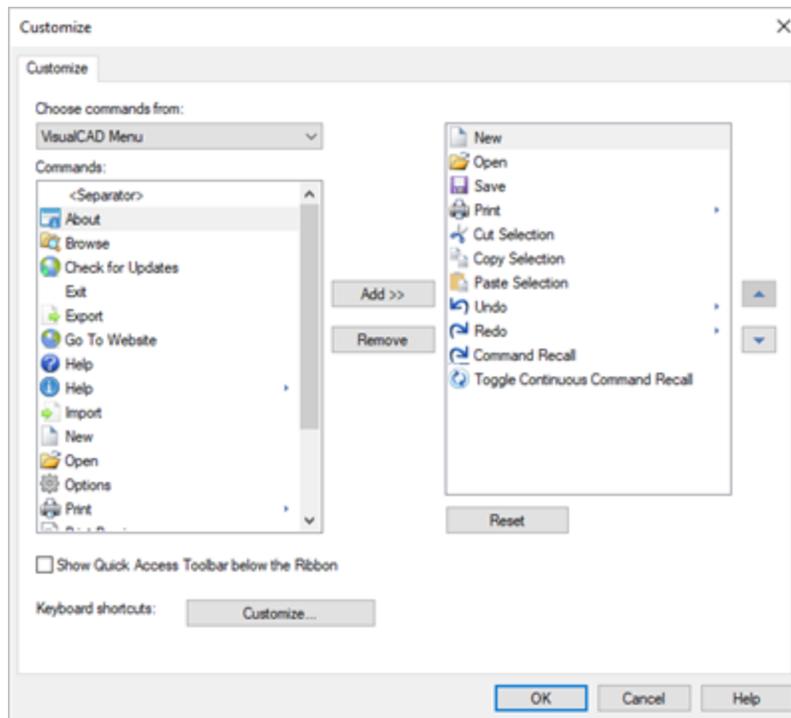
The Quick Access Toolbar

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](#).

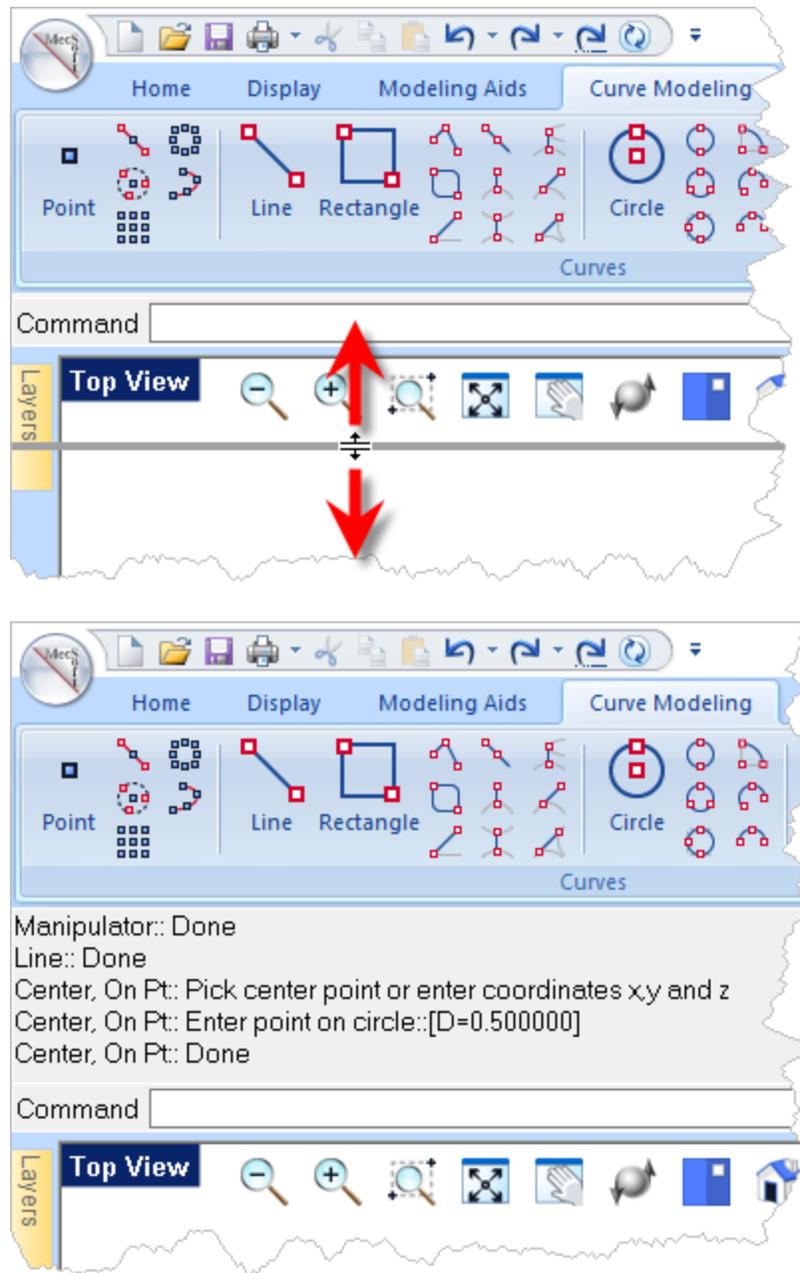


5.7 Viewing the Command Prompts

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.



5.8 The Status Bar

The Status Bar

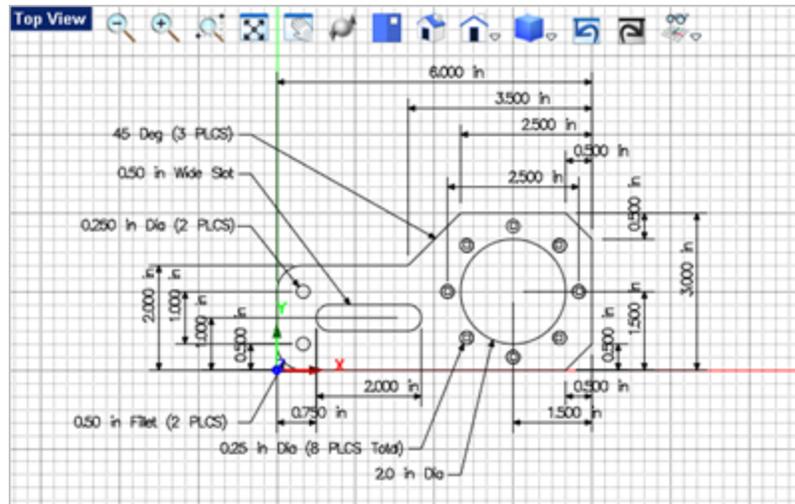
Located across the very bottom of the VisualCAD display you will find the Status Bar. It contains information and quick access to some very helpful commands. From left to right on the Status Bar you will find This is where you access the Layer Manager, Properties dialogs, Graphic Manipulator and more. 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.](#)

#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](#).

6.1 Create Layers



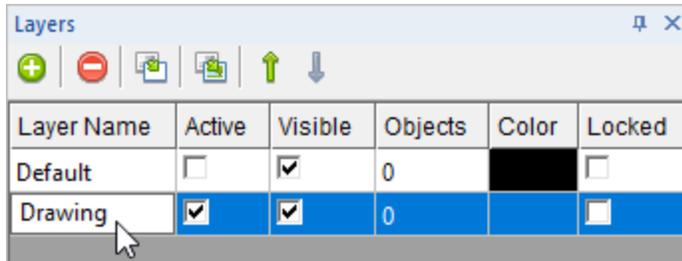
Create new Layers

In this step you will use the [Layer Manager](#) to create some new layers and then set the active layer.

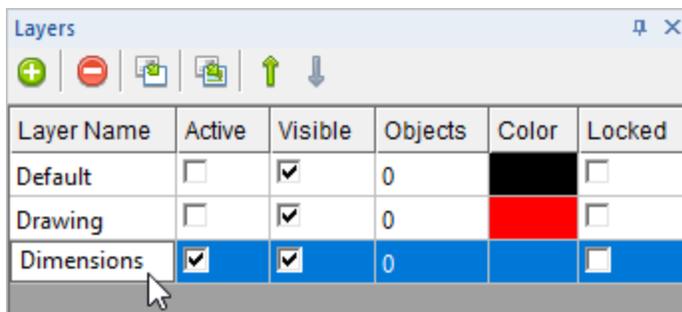
1. Display the [Layer Manager](#). You can pick the [Layer Manager](#) icon  located on the [Status Bar](#).
2. From the [Layer Manager](#) select the [Add Layer](#) icon  to add a new layer.



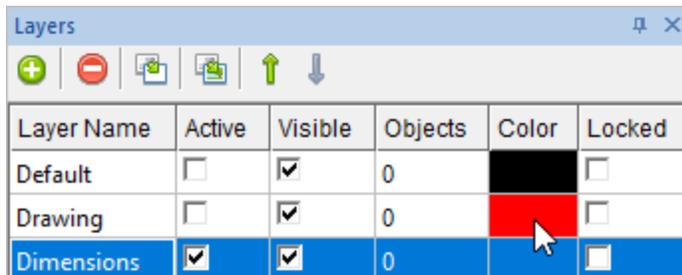
- Now let's rename the new layer to **Drawing**. First double-left-click in the **Name** field to activate it, and enter **Drawing** as the new layer name.



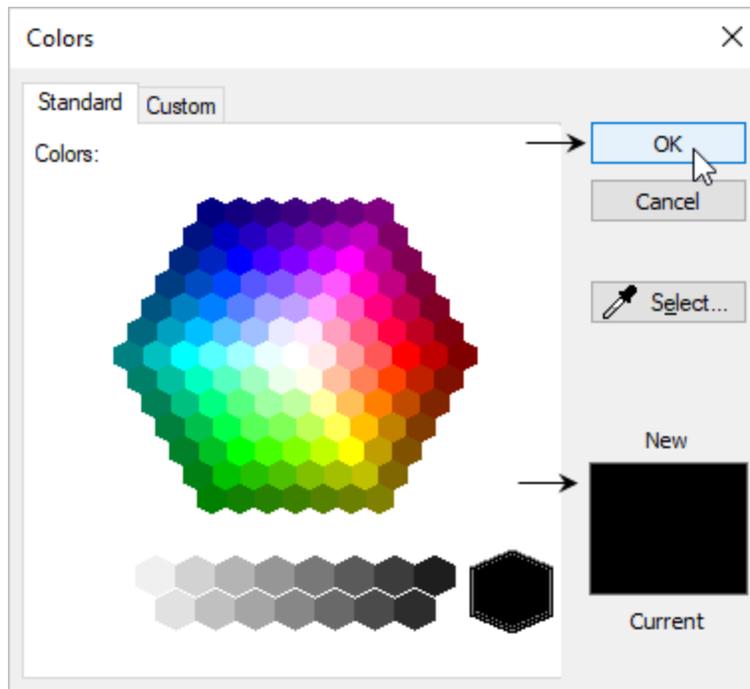
- Repeat the procedure to create a new layer, this time named **Dimensions**.



- 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.
- 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.



- 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**.

Layer Name	Active	Visible	Objects	Color	Locked
Default	<input type="checkbox"/>	<input checked="" type="checkbox"/>	0		<input type="checkbox"/>
Drawing	<input type="checkbox"/>	<input checked="" type="checkbox"/>	0	Black	<input type="checkbox"/>
Dimensions	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0	Black	<input type="checkbox"/>

9. Now from the **Active** column, check the box to make the layer named **Drawing** as the **Active Layer**.

Layer Name	Active	Visible	Objects	Color	Locked
Default	<input type="checkbox"/>	<input checked="" type="checkbox"/>	0	Black	<input type="checkbox"/>
Drawing	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0		<input type="checkbox"/>
Dimensions	<input type="checkbox"/>	<input checked="" type="checkbox"/>	0	Black	<input type="checkbox"/>

6.2 Curve Drawing

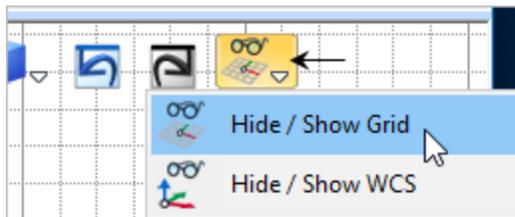
Curve Drawing

In this step you will learn how to use the keyboard to quickly drawing a polyline and then create some fillets and chamfers.

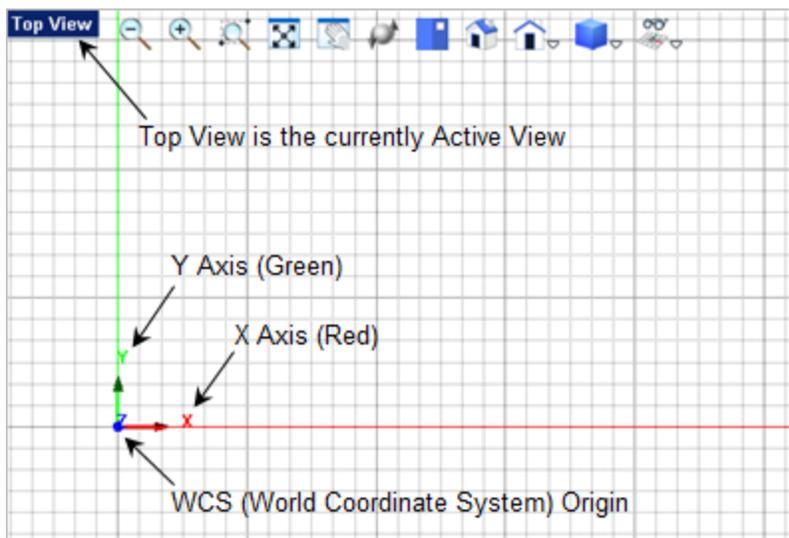
1. You only need the **Top View** **Top View** displayed for this exercise so if you have multiple **Viewports** displayed, double-left-click on the **Top View** tab to change the display to that **Viewport**.
2. Alternately you can right-click on the **Top View** tab and select **Single Viewport**.
3. If you do not see the **Top View** **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**.

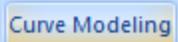


4. For visual purposes, we are displaying the construction **Grid** and the **Grid** options are set to those in the [Preferences Exercise](#).
5. If the **Grid** is not displayed you can toggle it on by selecting **Hide / Show Grid** from the **View Toolbar**.

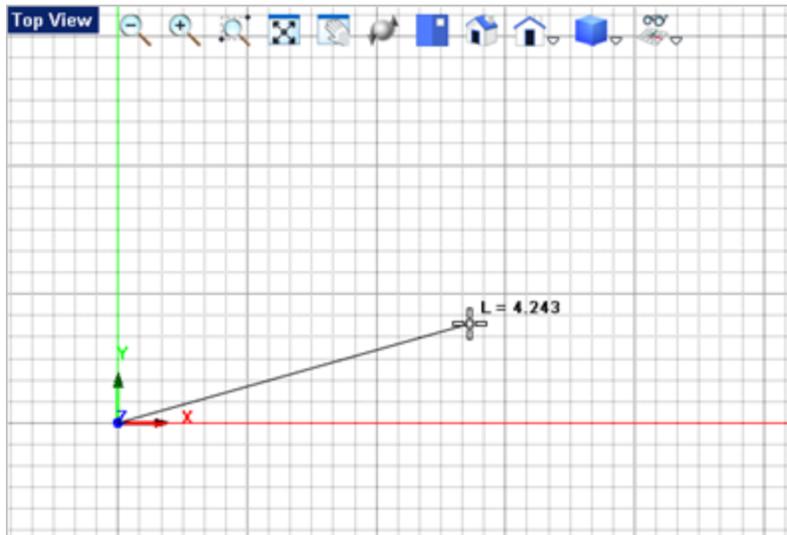


6. Scroll the middle mouse button to zoom in/out so that you see the **World Origin** **WCS** triad toward the bottom left of the screen. If you do not see the **X (Red)** and **Y (Green)** axes displayed, make sure you are in the **Top View**.

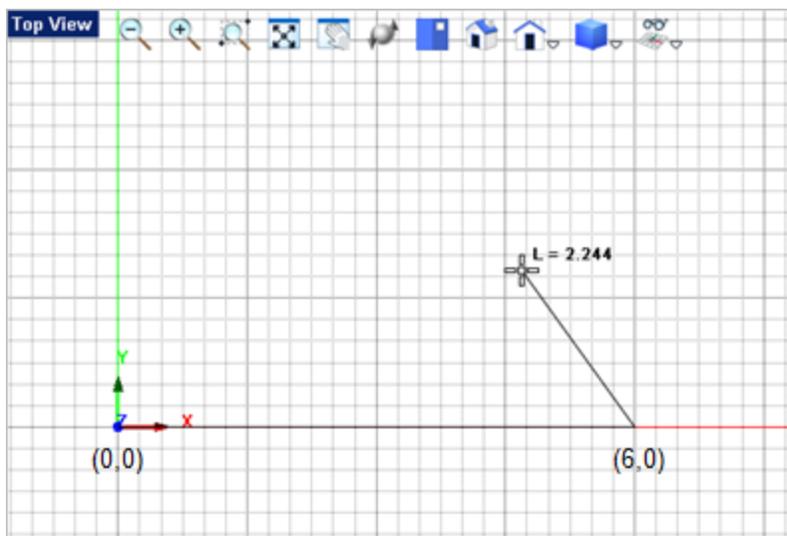


7. Now we can start curve drawing. From the **Curve Modeling**  tab, select the **Polyline**  command.
8. The command prompt says:
Polyline:: Pick point or enter coordinates x,y and z
For the start point of the polyline, enter **0,0** in the command window
Command and press **<Enter>**.

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**.



- 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.

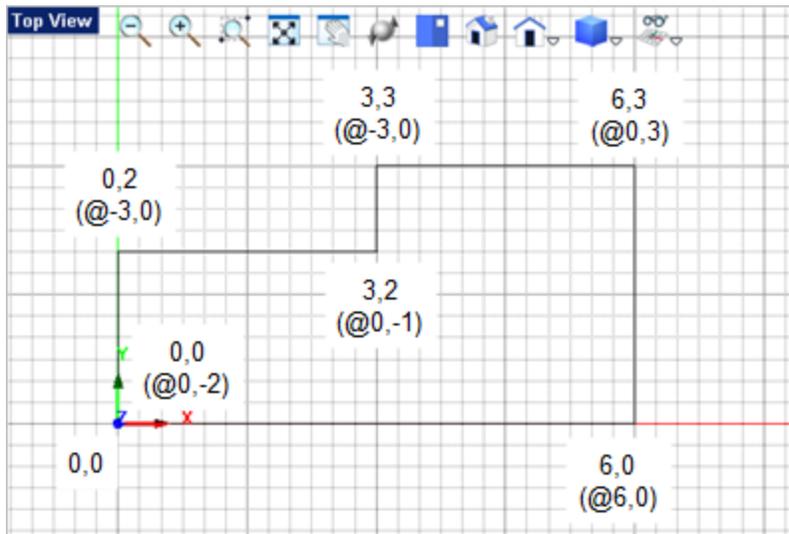
For the next coordinate enter **@0,3** in the command window and press **<Enter>**. This will place the next point in the polyline 3" from the last point in the positive **Y Axis** direction.

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)
- 0,0 (@0,-2)

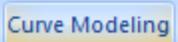
Your polyline should like this:



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.

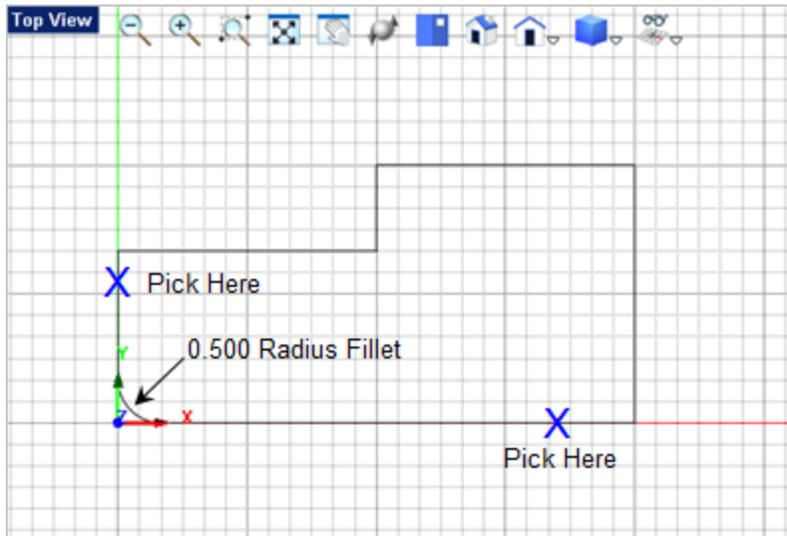


- Always **Save** your work! Select the **Save** or **Save As** command from the **Home** tab or the **VisualCAD Main Menu** or press **<Ctrl+S>**.

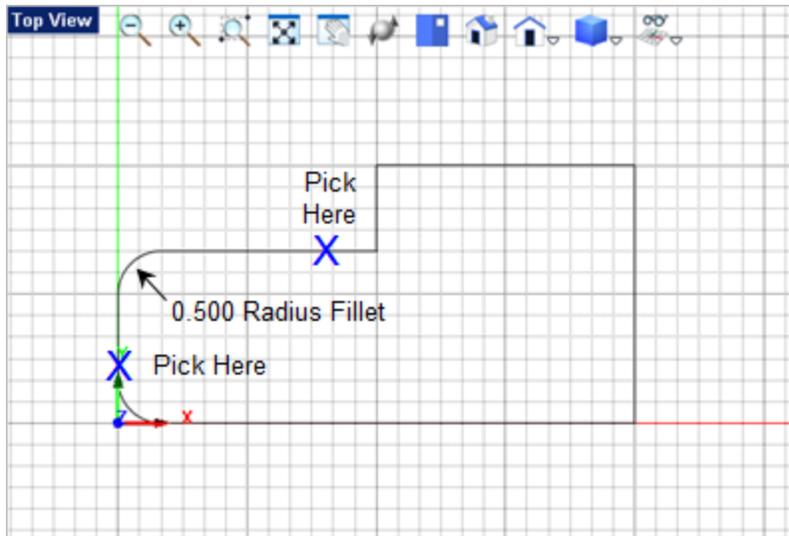
12. Now let's add some fillets. From the **Curve Modeling**  tab select the

 **Fillet** command.

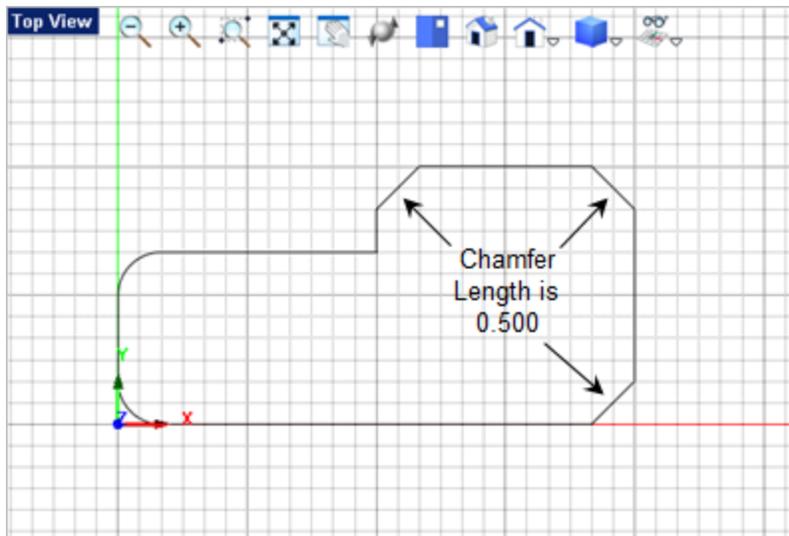
13. For the fillet radius, enter **0.5** in the command window and press **<Enter>**.
14. For the two lines to fillet, select the lines marked in the illustration below. For filleting it is best to select the line on the end furthest from the end you want to fillet.



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>**.



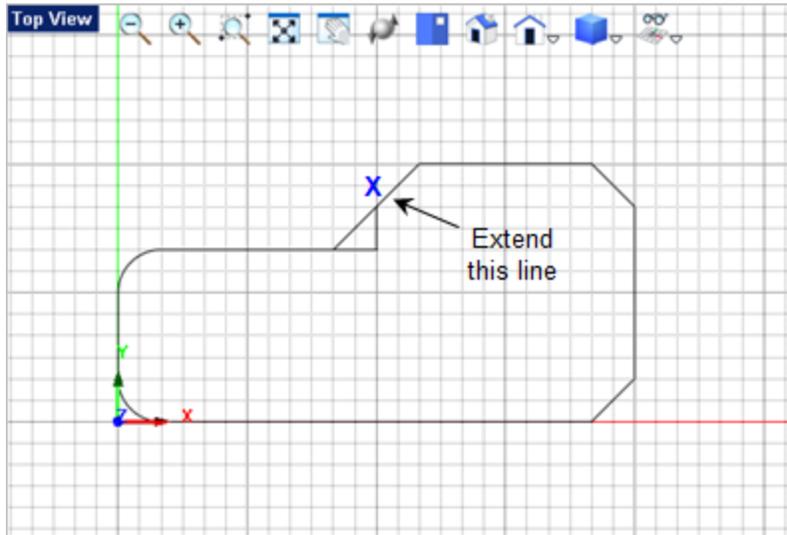
17. Next we will chamfer the three remaining outer corners. From the [Curve Modeling](#) [Curve Modeling](#) tab select the [Chamfer](#)  command.
18. For the chamfer length, enter 0.5 in the command window and press <Enter>.
19. For the lines to chamfer, select lines marked in the illustration below. For chamfers it is best to select the line on the end furthest from the end you want to chamfer.



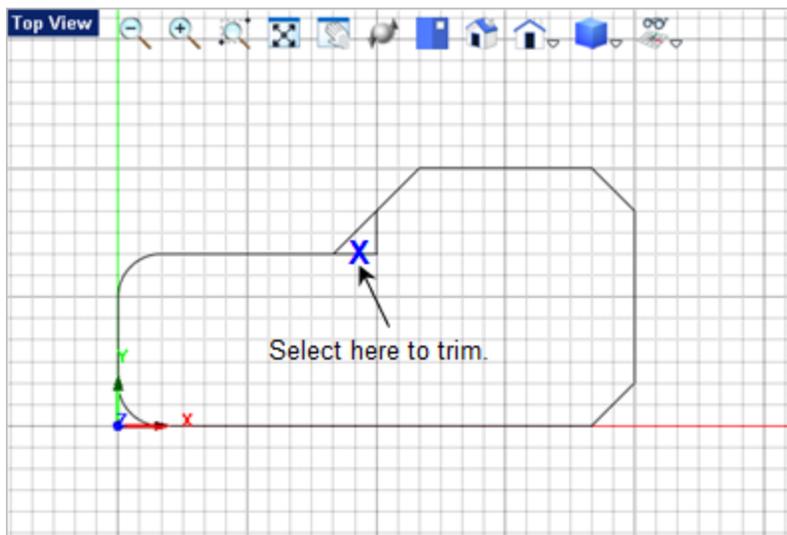
20. Currently our curves are still are one polyline. For this next step we need to explode the polyline. From the [Curve Modeling](#) [Curve Modeling](#) tab select the [Explode curves](#)  command.
21. Select the polyline and the right-click the mouse or press <Enter>. The polyline is now individual curves.



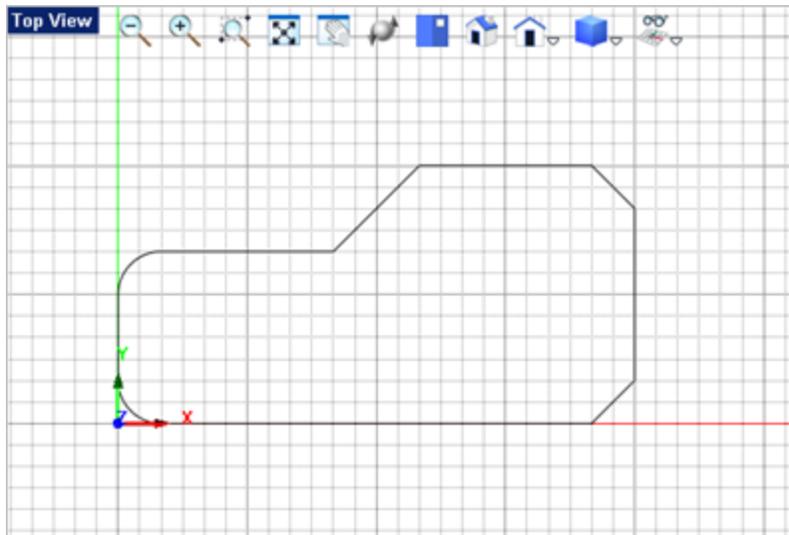
22. Now from the **Curve Modeling** **Curve Modeling** tab select the **Extend Curves** command.
23. Select the change line marked below to extend it to the horizontal line.



24. Now from the **Curve Modeling** **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.



- Next, delete the unneeded line. Just select it and then press the key or select the **Cut**  command icon from the [Quick Access Toolbar](#).



6.3 Using Visual Aids

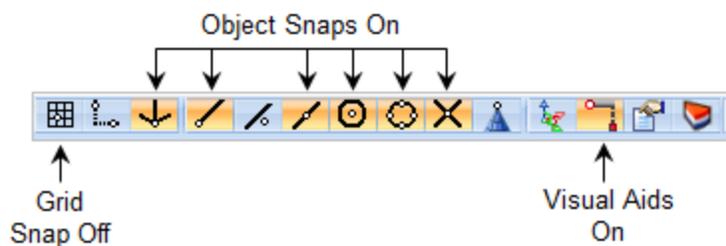
Using Visual Aids

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.

- Curve creation using [Visual Aids](#) can be enabled by selecting the [Visual Aids](#)  icon on the [Status Toolbar](#).

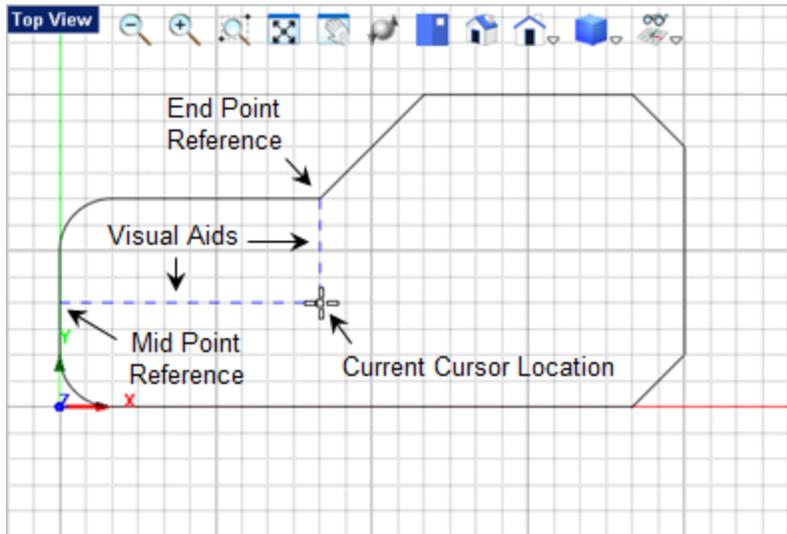
While [Visual Aids](#) is toggled **On**, the currently active [Object Snaps](#) are referenced visually depending on where the cursor is located on the drawing.

- For now, enable the [Origin Point Snap](#) , [End Point Snap](#) , [Mid Point Snap](#) , [Center Point Snap](#)  and [Quad Point Snap](#) .
- Also disable the [Grid Snap](#) . Your [Status Bar](#) should look like this:

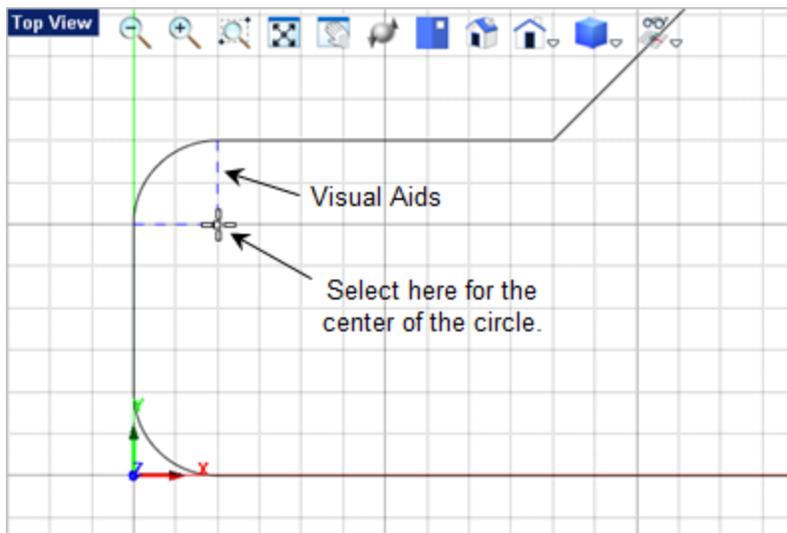




4. Now from the **Curve Modeling** Curve Modeling tab select the **Circle on Point** command.
5. For the circle center point move the cursor near geometry in your drawing. You will see **Visual Aids** appear (i.e., reference lines).

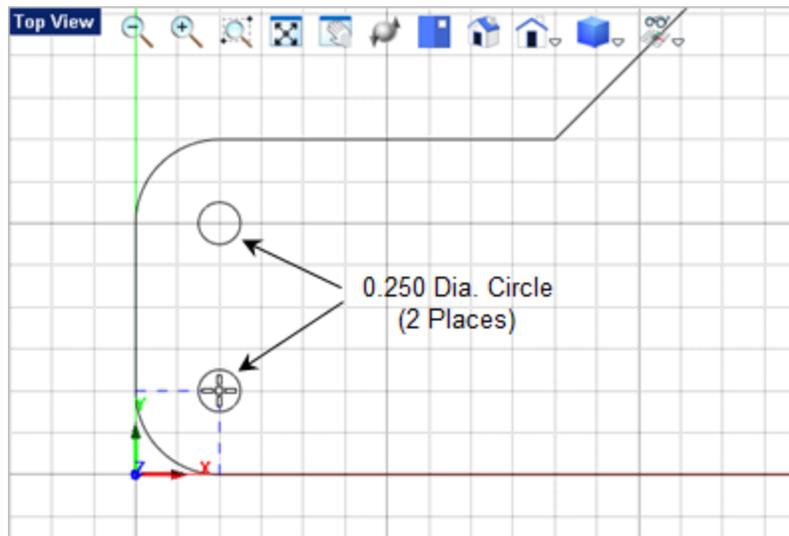


6. We want to locate the center of circle at the center of the fillet located in the upper left corner of the drawing. Move the cursor to this area and you will see two **Visual Aids** appear when the end points of the fillet are recognized.



7. For the circle diameter enter **0.25** Command and press **<Enter>**.

8. Press <Enter> to repeat the command and draw a circle at the center of the lower left fillet.



6.4 More Drawing Tools

More Drawing Tools

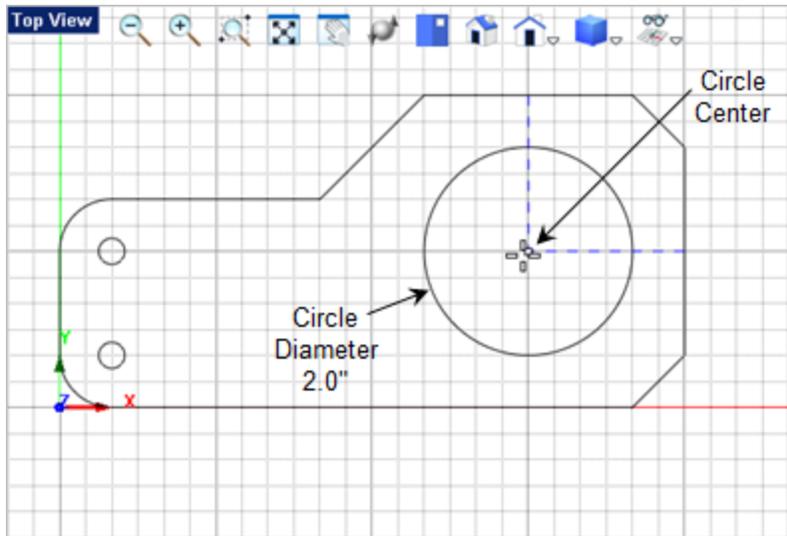
In this step you will learn a few additional curve creation and curve editing commands.

1. Lets draw another circle. From the **Curve Modeling**  tab select the



Circle on Point command.

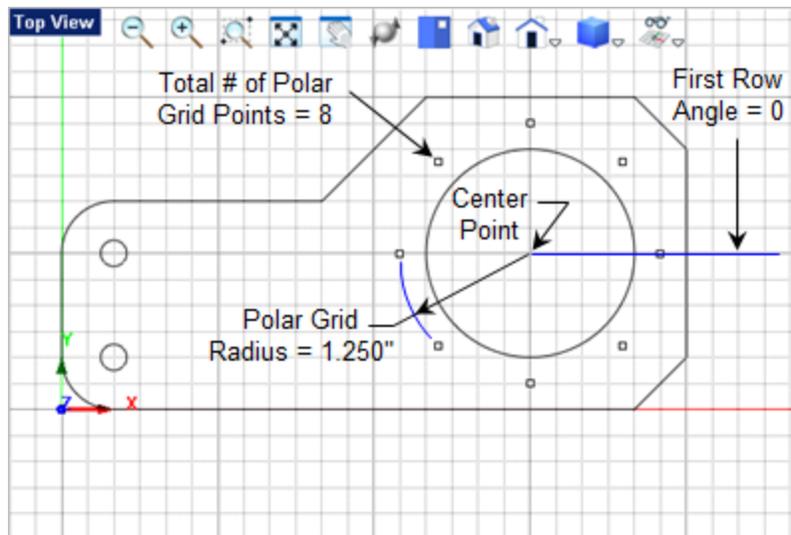
2. For the center point of the circle use the **Visual Aids**  to locate and select the **Mid Point**  intersection of the upper and right side lines.
3. For the circle diameter enter **2.0** in the command window and press <Enter>.



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.

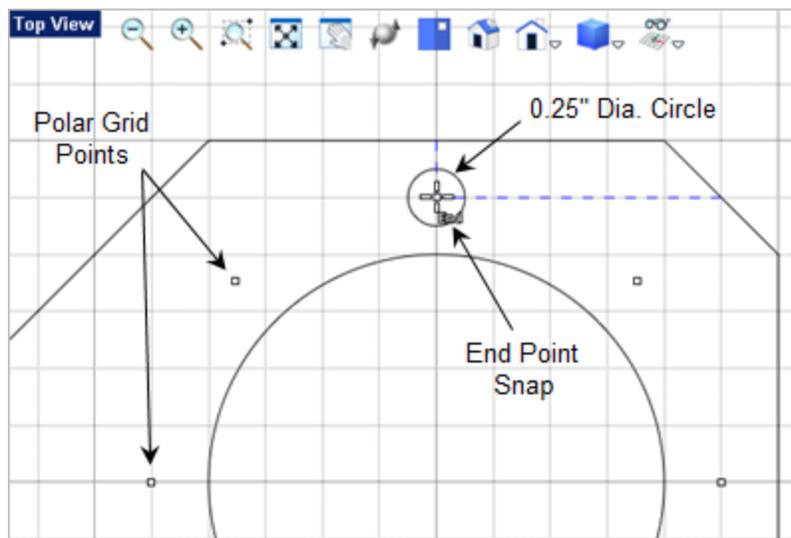


- For this exercise we will also add the polar array of circles. From the [Curve](#)

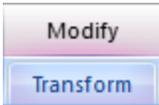


Modeling [Curve Modeling](#) tab select the [Circle on Point](#) command.

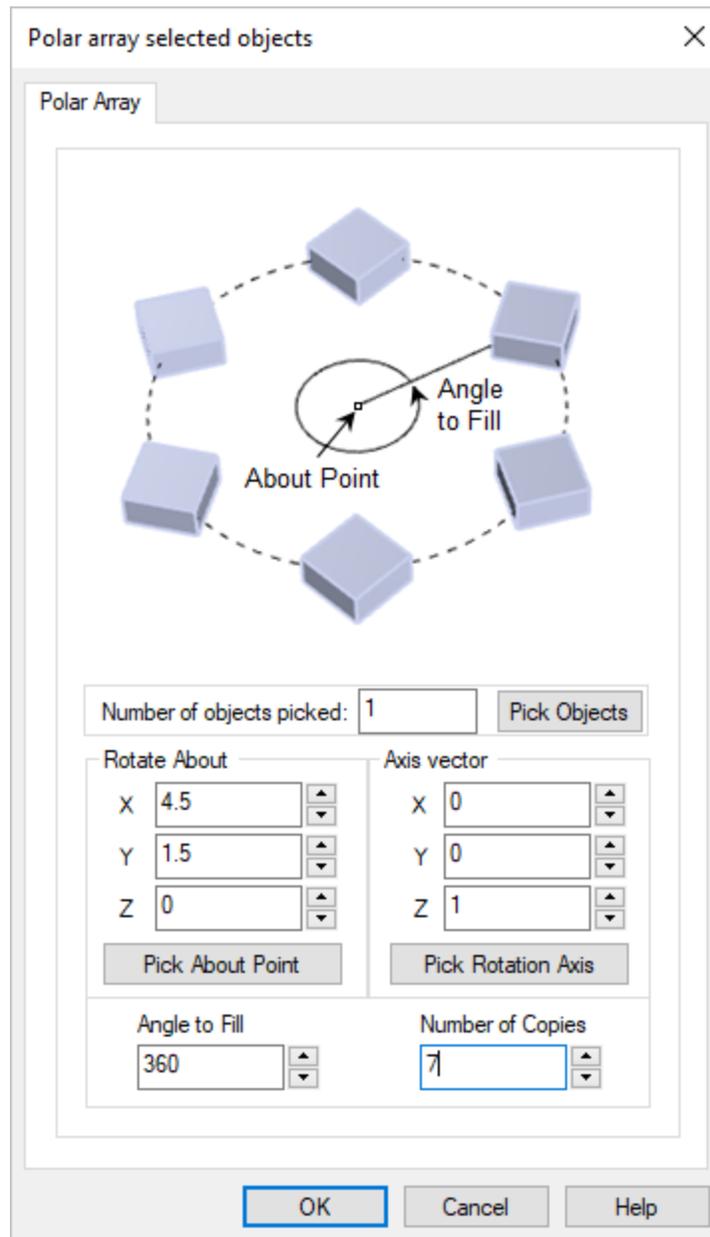
- For the center point, make sure the [End Point Snap](#)  is on and then select the end point at the 12 o'clock position.
- For the circle radius enter [0.25](#) and then press [<Enter>](#).



- 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.

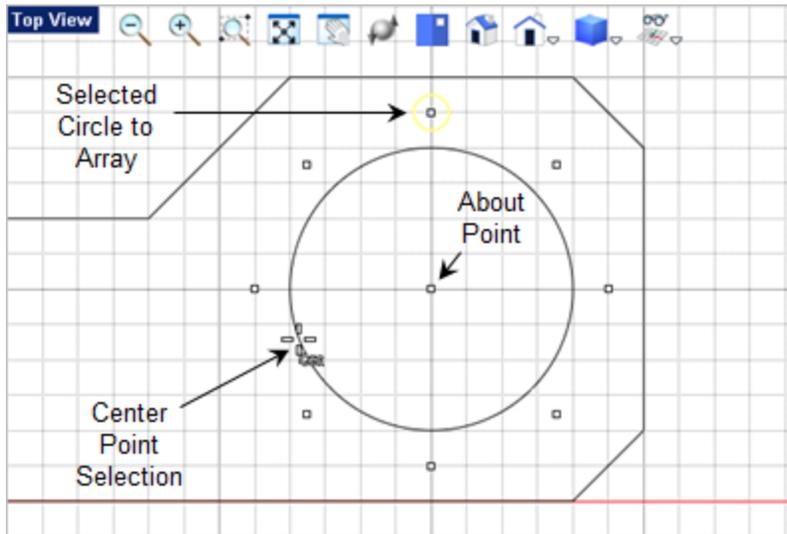
15. Select the **Polar Array**  command. The **Polar array selected objects** dialog will display.



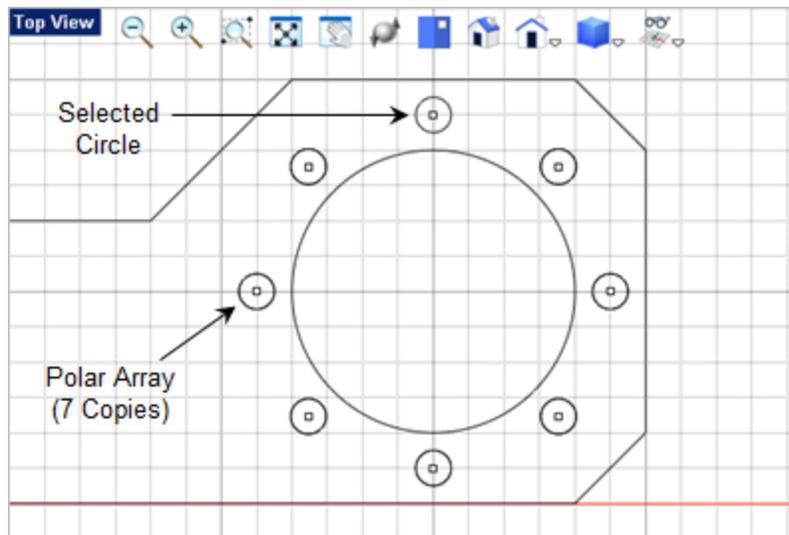
16. The **Number of Objects** field should be **1**. This represents the circle that you have selected.

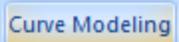
17. Under the **Axis Vector**, the values should be **X0, Y0, Z1**.

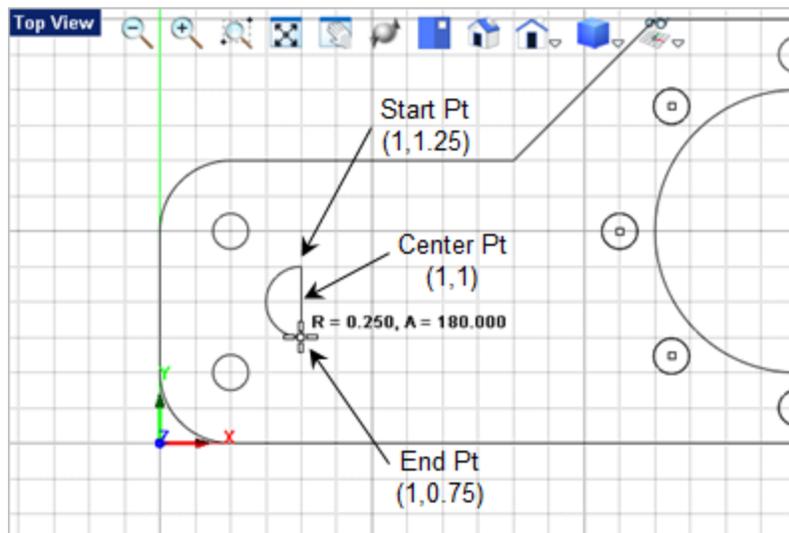
18. Under the **Rotate About** section, select the **Pick About Point** button. The dialog will minimize and wait for you to select a point.
19. From the **Status Bar** activate the **Center Point Object Snap** .
20. Now move the cursor over the 2" diameter circle until the center point is displayed 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.

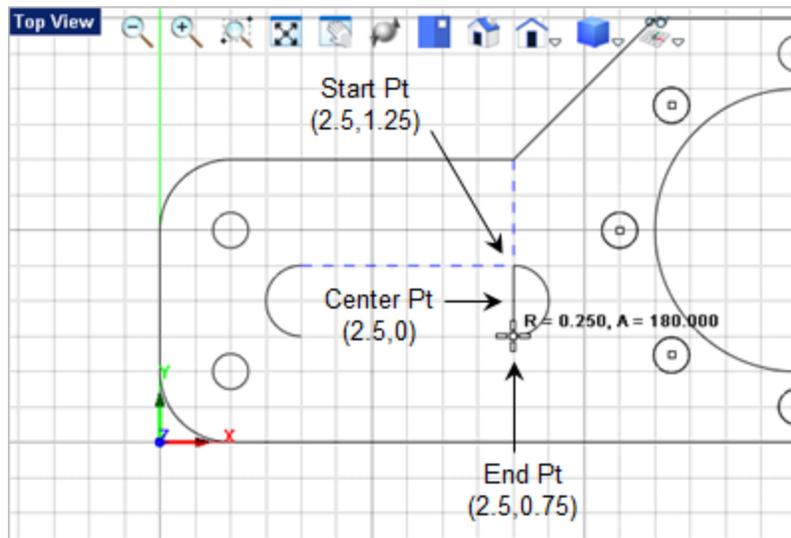


26. Now we will drawing a slot. From the Curve [Curve Modeling](#) tab  select the [Arc Center, Start, Angle](#) command. 
27. In the [Status Bar](#) make sure the [Grid Snap](#) is On .
28. For the center point select the grid point shown below or enter 1,1 in the command window and press <Enter>.
29. 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.

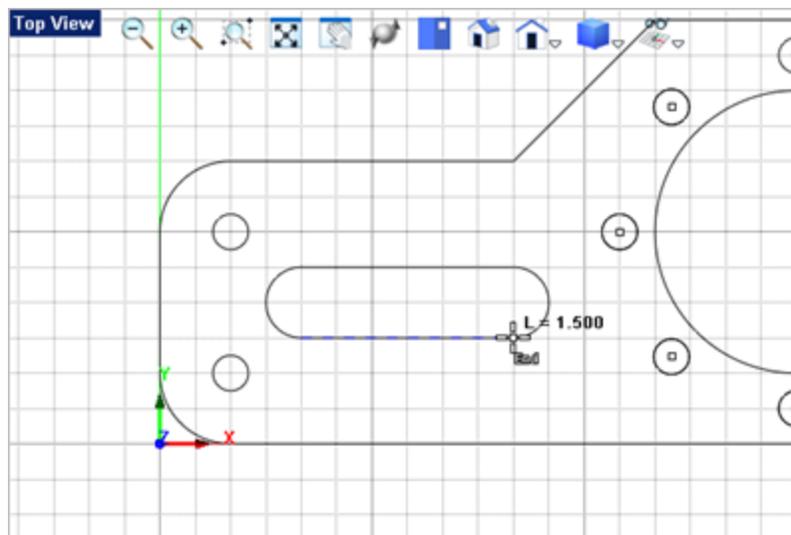


32. Now we want to draw a line to connect the two arcs across the top. From the [Curve](#)



Modeling tab [Curve Modeling](#) 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:

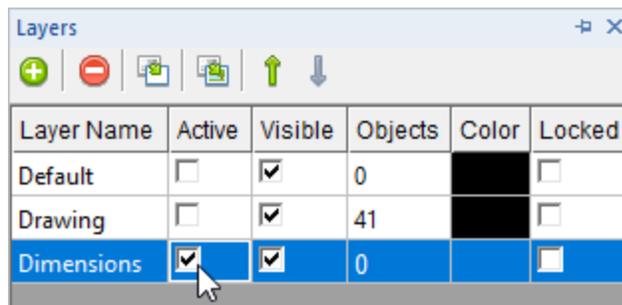


6.5 Dimensioning

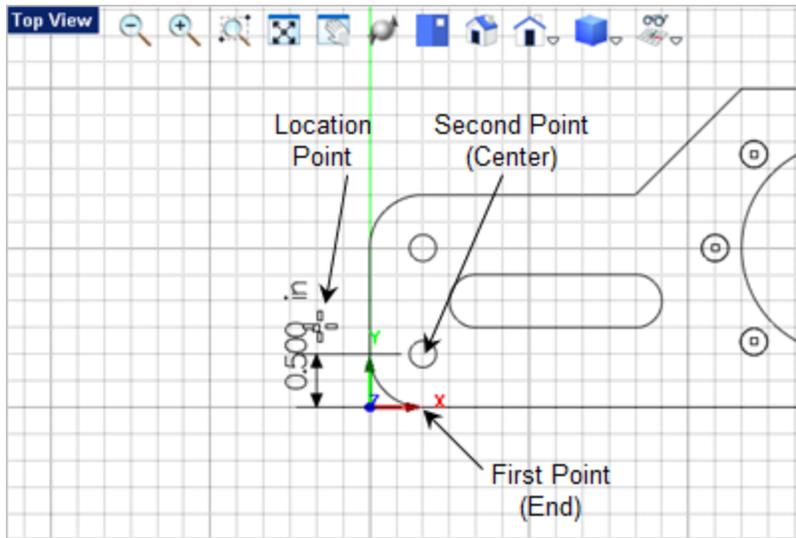
Dimensioning

In this step you will learn how to place dimensions on the drawing.

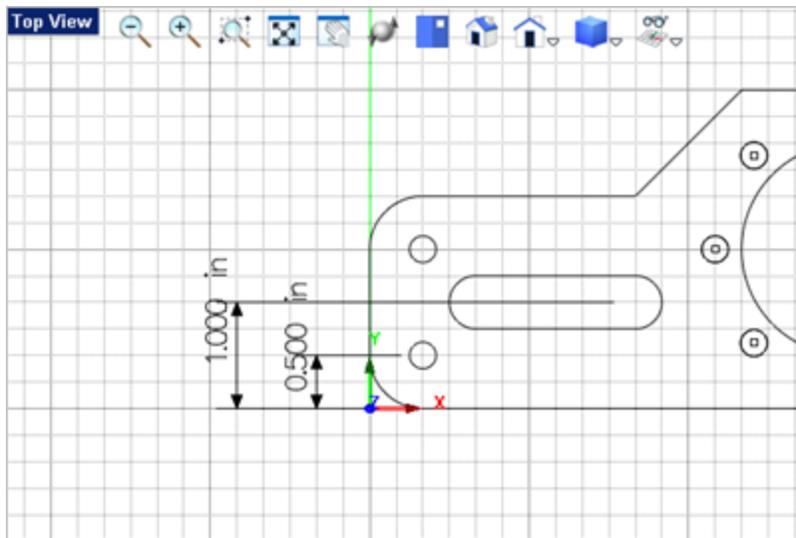
1. Start by using the middle mouse wheel to zoom out to leave yourself plenty of room to place your dimensions. If your mouse does not have a middle wheel, select the **Zoom Out**  icon from the **View Toolbar**. You can also use the **Pan**  icon to pan the drawing within the display window.
2. Now 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**.



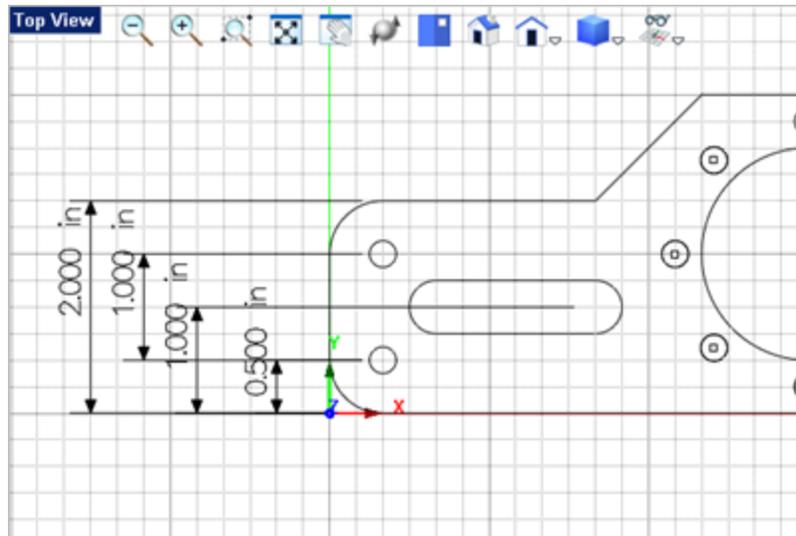
4. From the **Dimension**  tab, select the **Vertical Dimension**  command.
5. Activate the **Object Snaps** required (**End** , **Center**  and **Grid** )
6. Select the first and second points that you wish to dimension.
7. Then select a point to locate the dimension. refer to the illustration below:



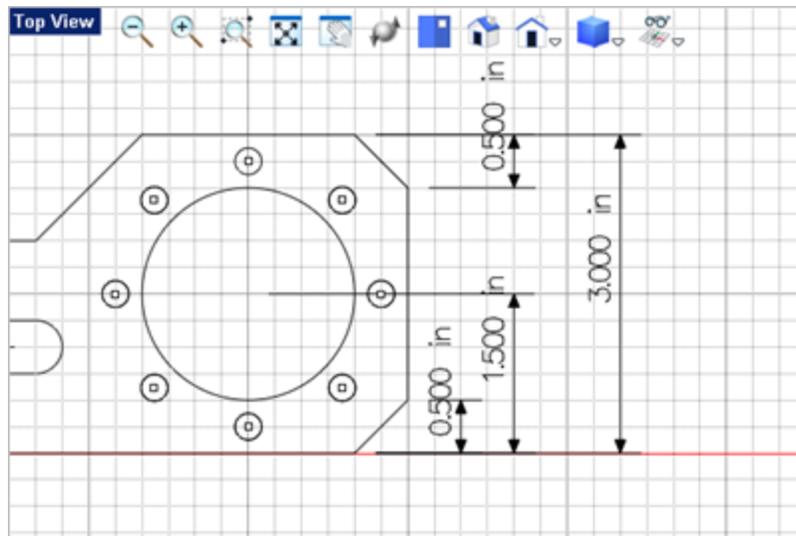
8. Press <Enter> to repeat the command.
9. Select the first, second and location point for the dimension as shown below:



10. Repeat the command to create three additional vertical dimensions on the left as illustrated below:



- Repeat the command to create three additional vertical dimensions on the right as illustrated below:

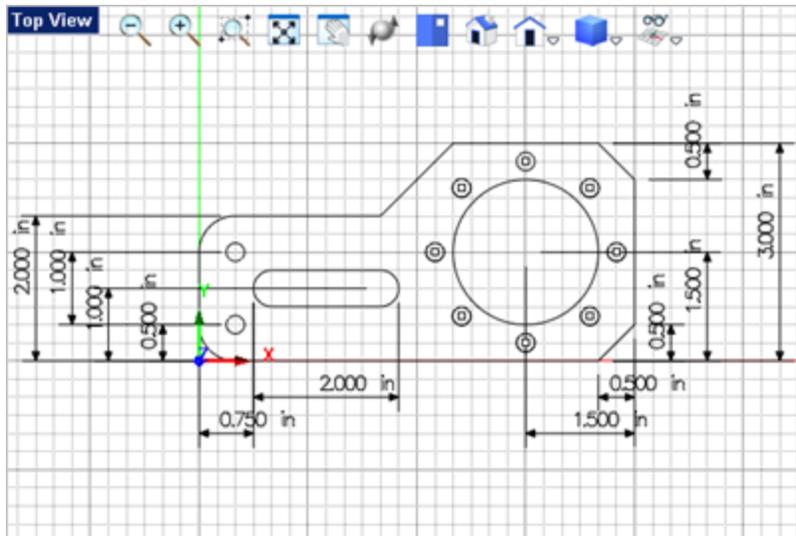


- Now we will create the horizontal dimensions. From the [Dimension](#) [Dimensions](#)

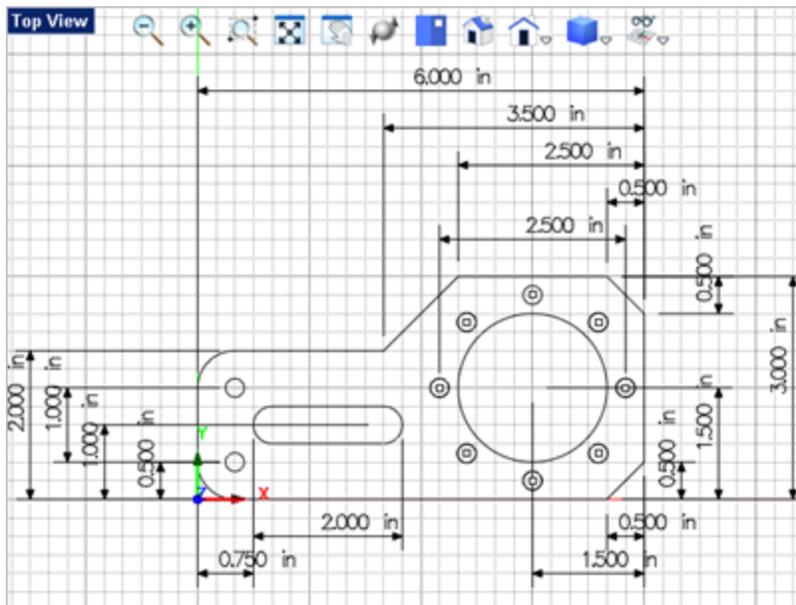


tab select the [Horizontal Dimension](#) command.

- 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:



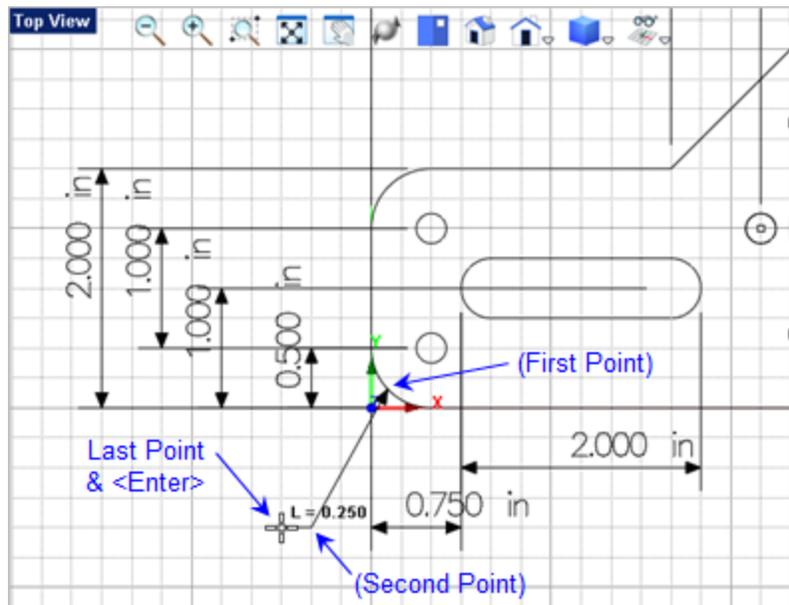
15. Now we will create some leader lines. From the **Dimension** Dimensions tab select



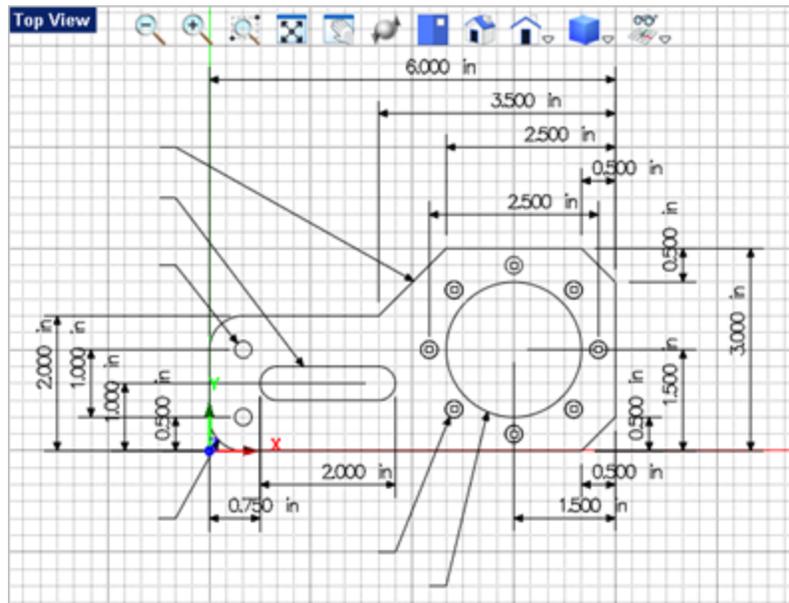
the **Leader Line** command.

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.

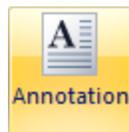
18. Then pick additional points to define the leader line and the press **<Enter>** or right-click to end the command. The points to pick are shown in the illustration below:



19. Now repeat the **Leader Line** command and add the leader lines shown below:

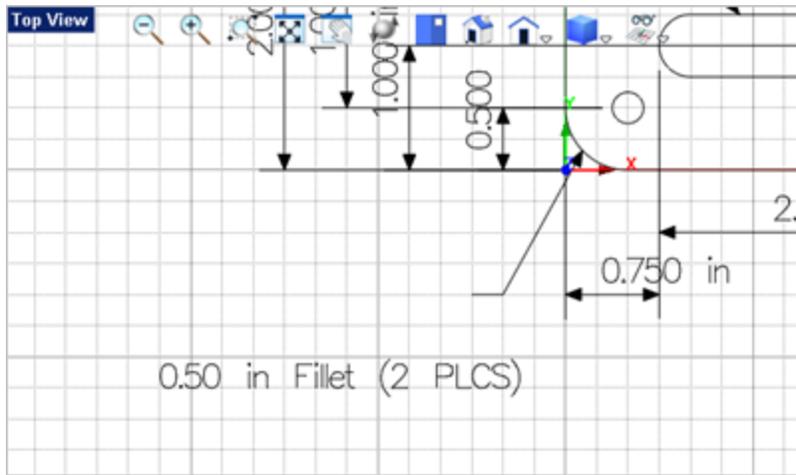


20. Now we will add the annotation text. From the **Dimension** **Dimensions** tab select

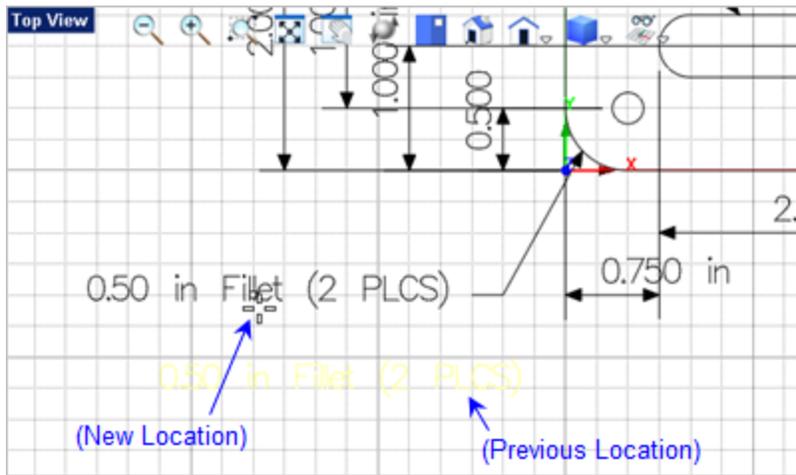


the **Annotation** command.

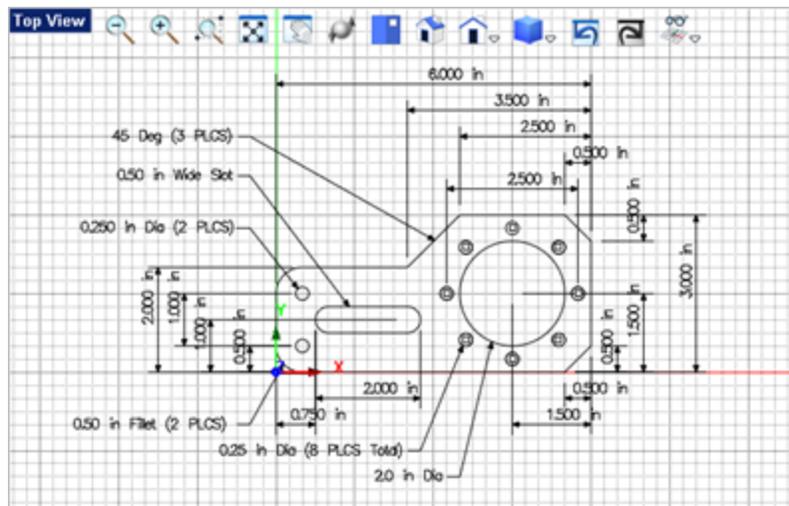
- 21. For the text string, at the command prompt enter **0.50 in Fillet (2 PLCS)** and press <Enter>



- 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.



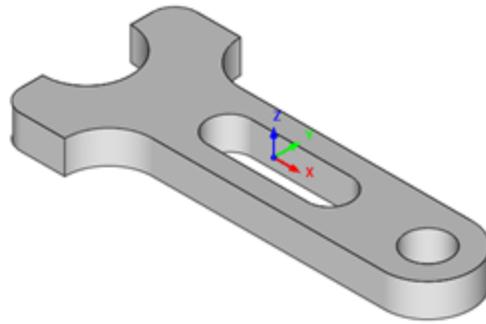
- 23. Repeat the **Annotation** command to create the additional leader text shown below:
- 24. Save the file as **Drawing_and_Dimensioning_Completed.vcp**.



Congratulations you have completed this exercise!

#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](#), [Fillet](#)s, [Mirroring](#), [Merging](#) and [Extruding](#) curves.



Model a Spanner Plate



What you will learn:

In this exercise you will perform the following [VisualCAD](#) tasks:

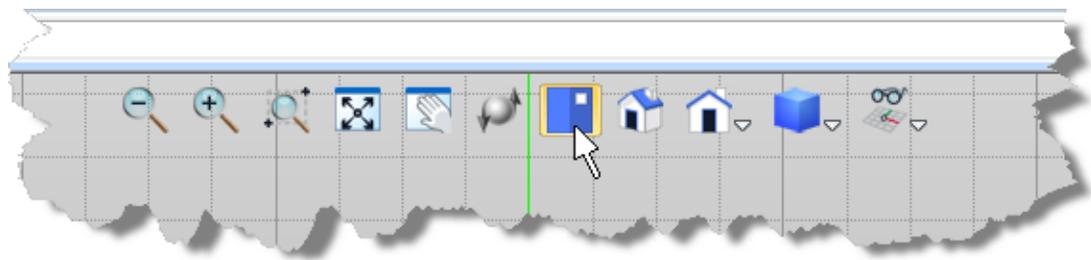
1. [Set to the Top View.](#)
2. [Create Reference Points.](#)
3. [Create Inner Cutouts.](#)
4. [Trim Curves.](#)
5. [Offset, Extend & Trim Curves.](#)
6. [Fillet Curves.](#)
7. [Mirror Curves.](#)
8. [Merge Curves.](#)
9. [Extrude Curves.](#)

7.1 Set to the Top View

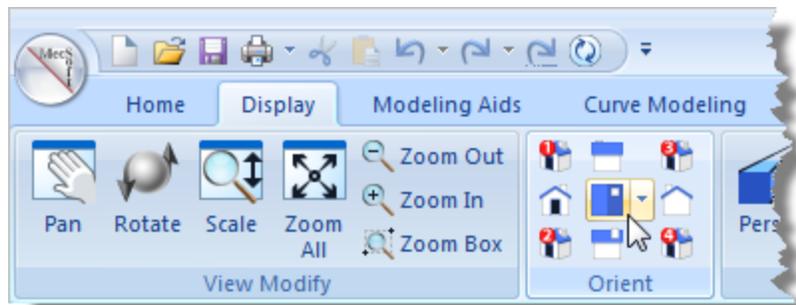


Set to the Top View

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

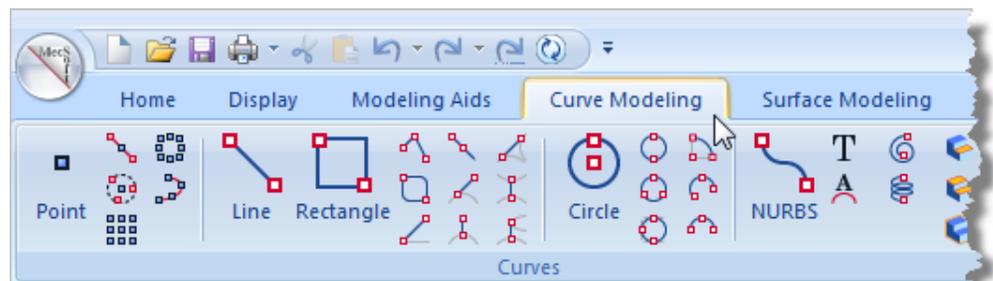


Display Ribbon Bar - Set to the Top View

7.2 Create Reference Points

Create Reference Points

1. Select the Curve Modeling tab **Curve Modeling** 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.

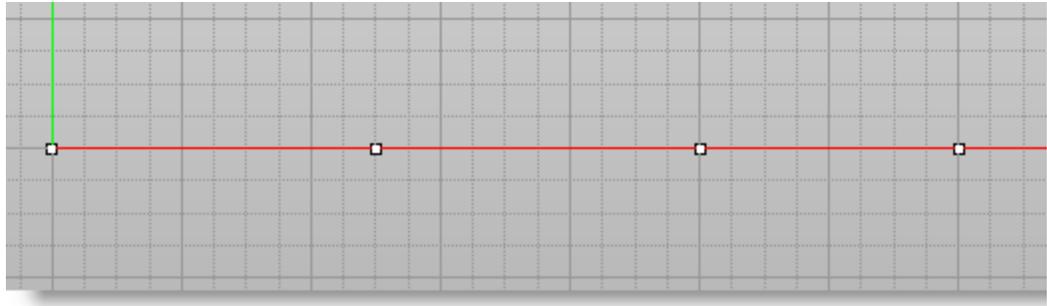
Create additional points by repeating the Create Point command at the following coordinate locations.

2.5,0

5.0,0

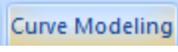
7.0,0

The created points are as shown below.



7.3 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

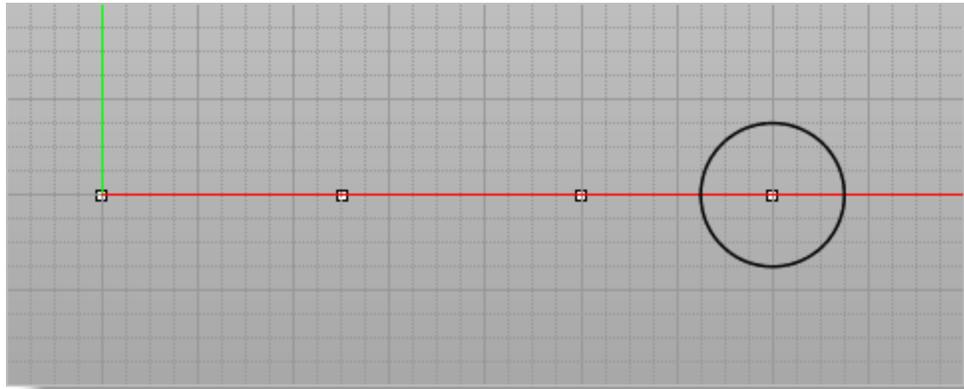
4. For the center point, pick last created point or type **7,0**

Command Prompt:

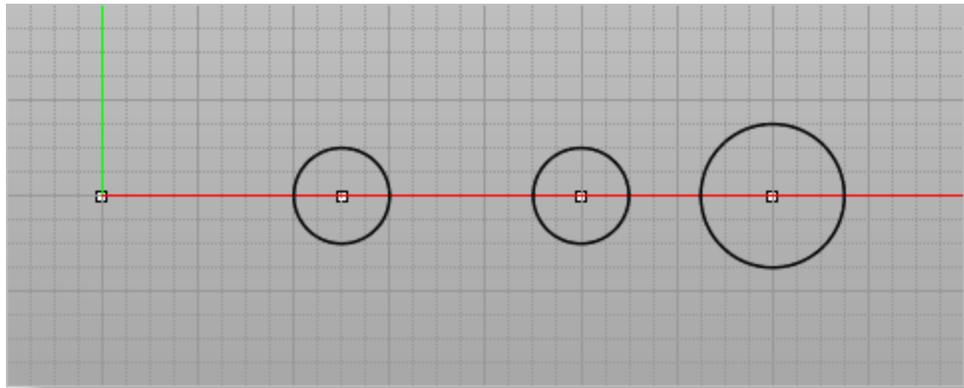
Center, On Pt:: Enter point on circle::[D=0.500000]

5. Specify **1.50** for **Diameter** in the command bar  and press **Enter**.

The created circle is as shown below.



- 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.



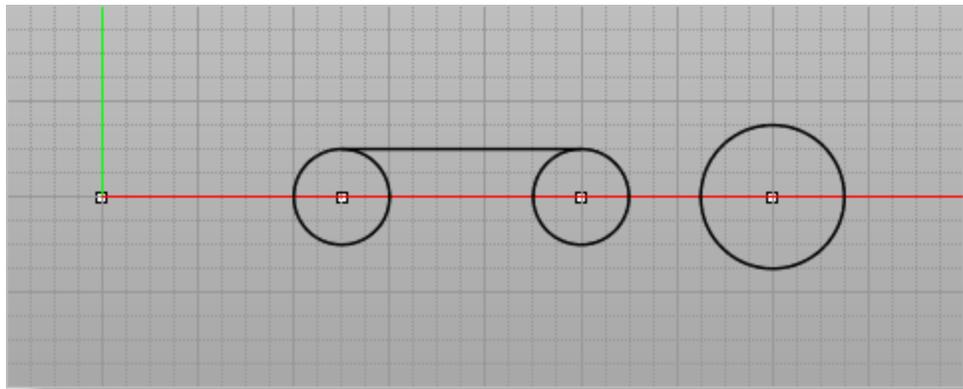
- From the **Status Bar**, toggle **Grid Snap** off and then toggle **Quad Point Snap** on.



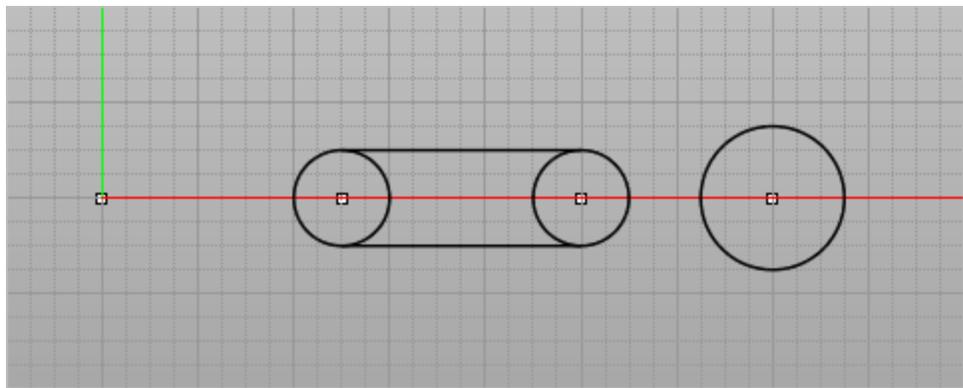
- Select the **Create Single Line** icon.



- Create a line between the top quadrants points of the smaller circles.



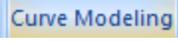
10. Repeat the line for lower quadrant points.



7.4 Trim Curves

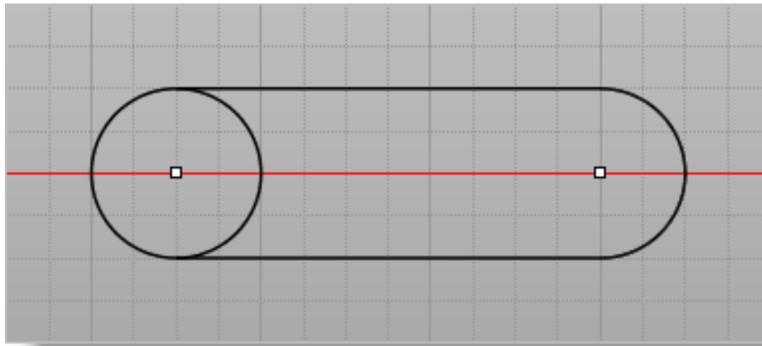
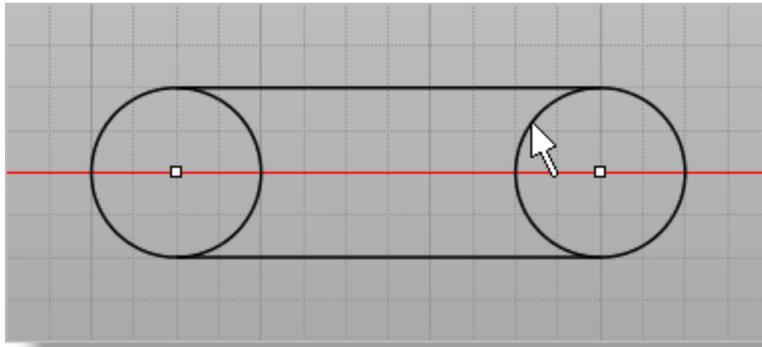
Trim Curves

The circles must be trimmed.

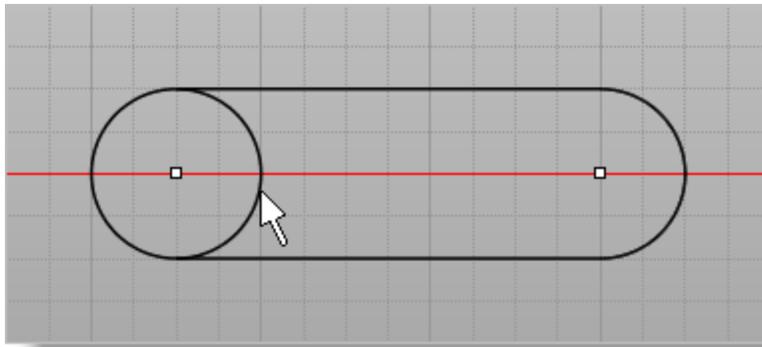
1. Select the **Curve Modeling** tab  from the top **Ribbon Bar**.



2. Select **Trim**
3. Click on the portion of the circle you want to delete.

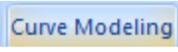


4. Repeat for the circle portion on the other side.



7.5 Offset, Extend & Trim Curves

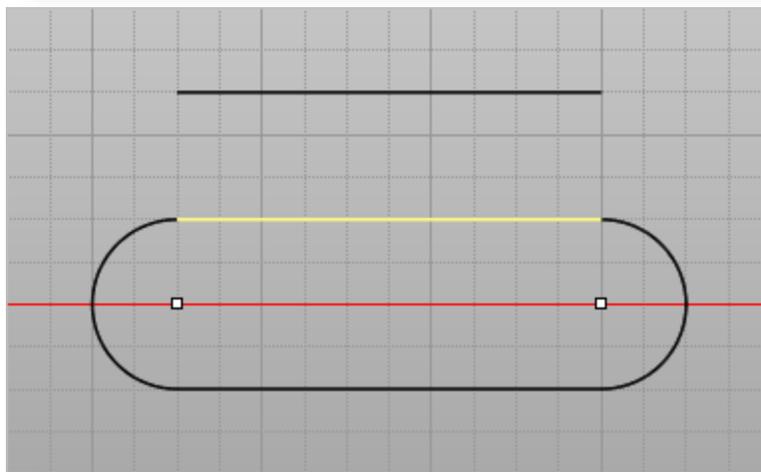
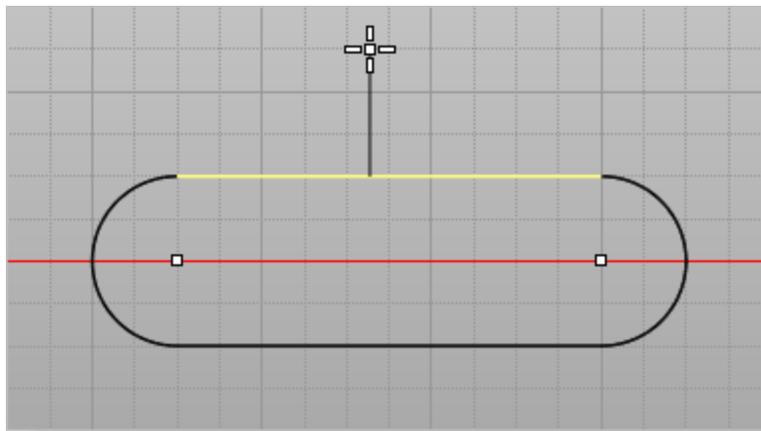
Offset, Extend & Trim Curves

1. Select the **Curve Modeling** tab  from the top **Ribbon Bar**.
2. Keep the **Curve Modeling Ribbon Bar** displayed and then select the top horizontal line.

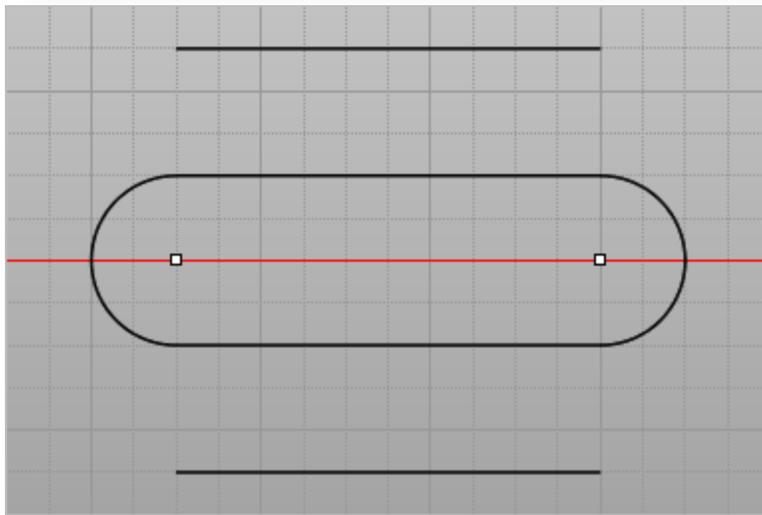
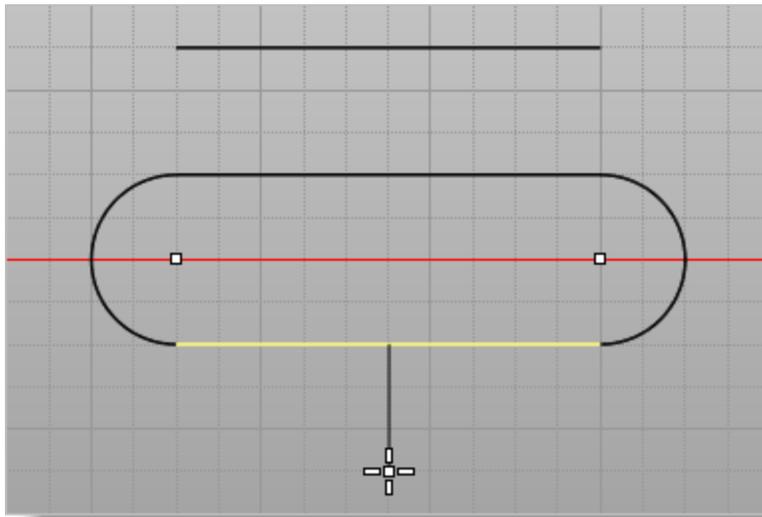


3. Pick the **Offset Curve** icon from the **Edit Curves** pane. A preview line will appear, indicating the direction of offset. Point the preview line to offset the curve outward and then type **0.75** from the keyboard and press **Enter**.

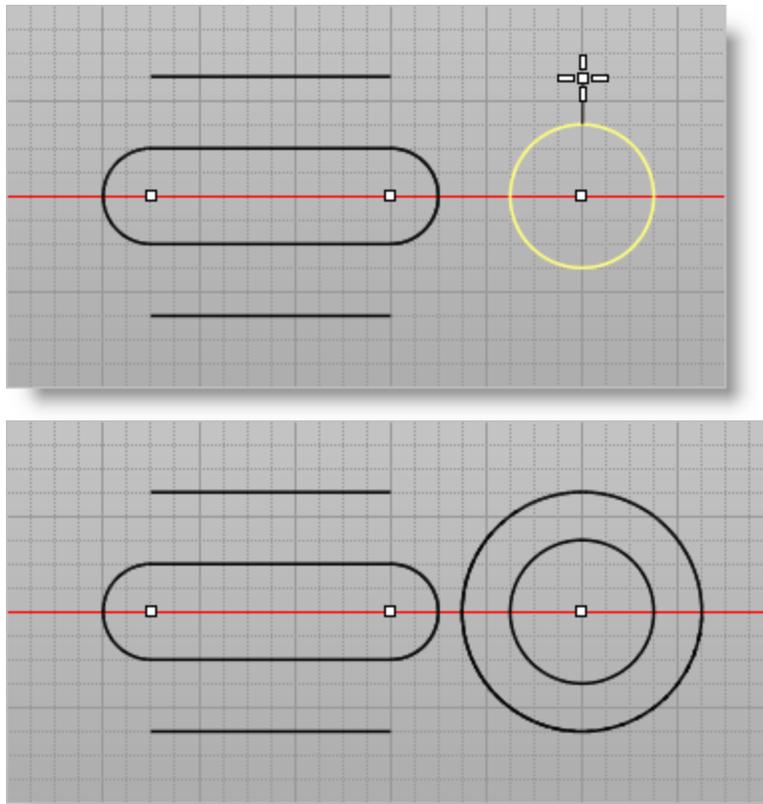
 For **Offset Curve**, your cursor points to the offset direction, so do not move the cursor to the **Command Prompt** field. Simply type **0.75** and press **Enter** to input the value.



- Repeat the offset for the horizontal line at the bottom, making sure to keep the cursor and directional indicator line pointing downward.



- Select the larger circle on the right and then pick the [Offset Curve](#) icon again.
- With the cursor, point the offset direction outward from the circle, type [0.50](#) and press [Enter](#).

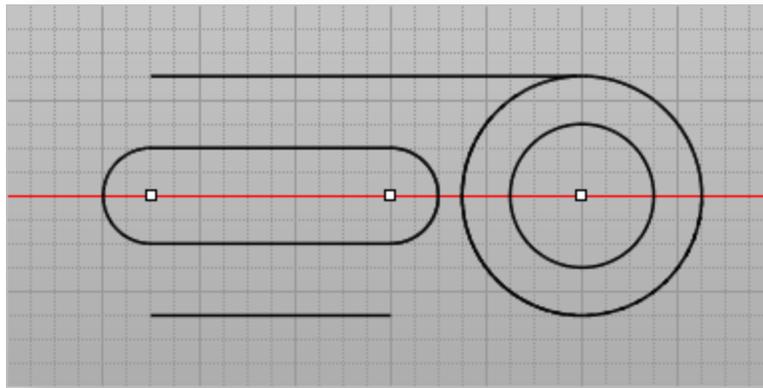
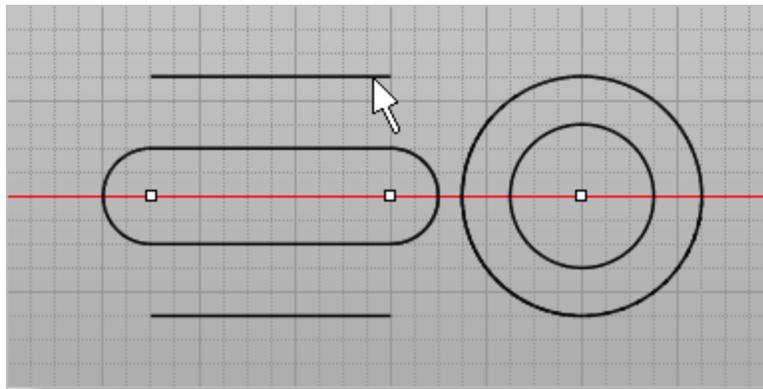


7. To modify the offset lines so that they meet the larger circle, pick the [Extend Curve](#)

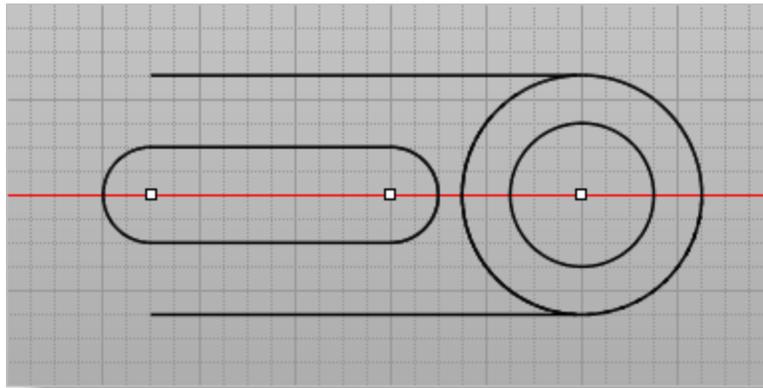


icon from [Edit Curves](#) pane. It is located just to the right of the [Offset Curve](#) icon.

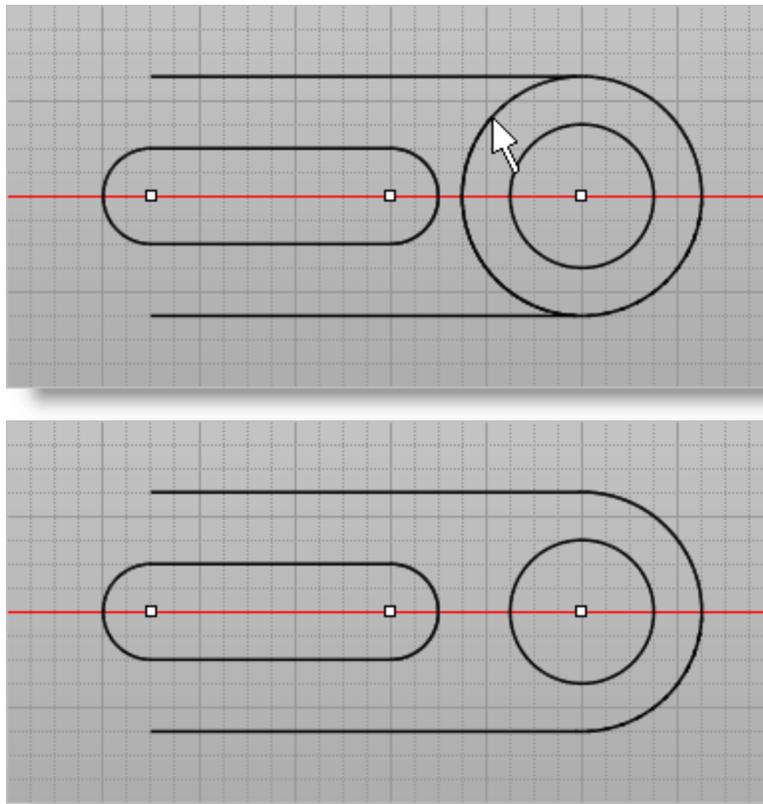
8. Click one of the offset lines towards the right. The line extends to the point where it meets the circle.



9. Repeat the extend curve for the line at the bottom.



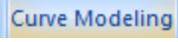
10. Select **Trim** and trim the large circle where it meets the two extended lines.



7.6 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.** .

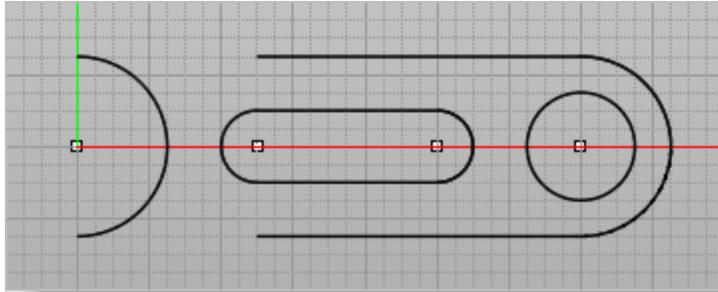
Command Prompt:

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

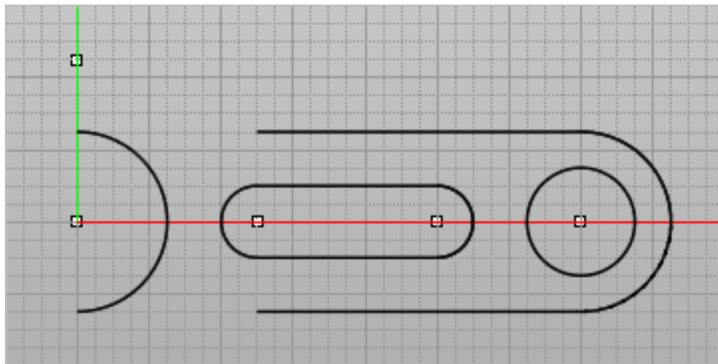
3. For the **Center**, type **0,0** in the **Command Prompt** and then press **Enter**.
4. For **Start**, type **0,1.25** in the **Command Prompt** and then press **Enter**.

5. For **End**, type **-180** in the **Command Prompt**  and then press **Enter**.

Your drawing should look like the following:



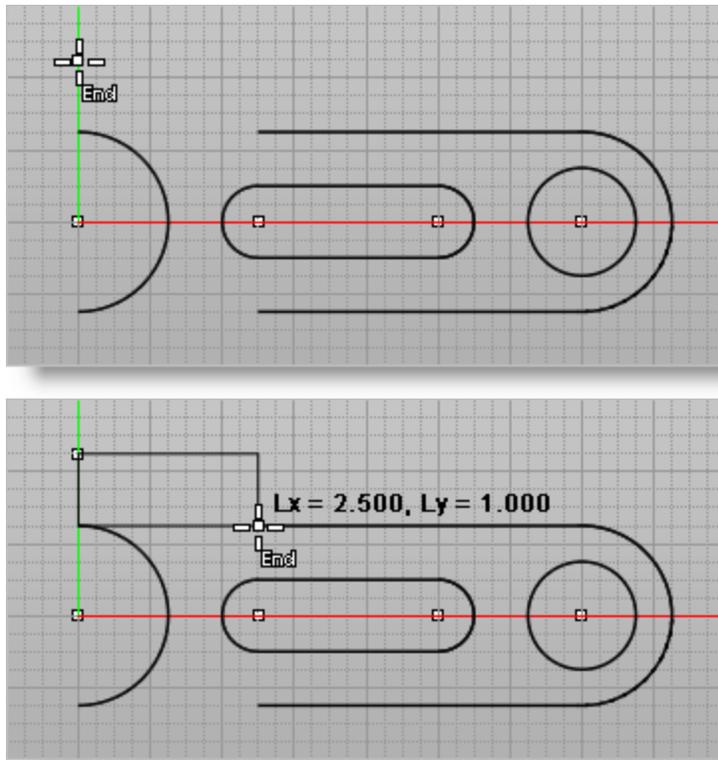
6. Now create a reference point located at **0,2.25** using the **Point**  command. The icon is located on the left side of the **Curve Pane** of the **Curve Modeling Ribbon Bar**.



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.

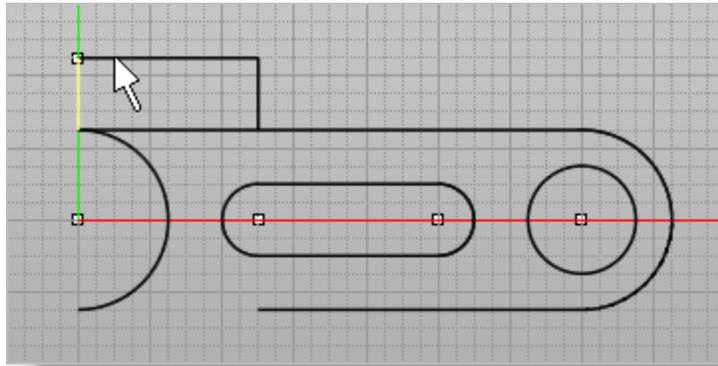


12. Select [Fillet](#).

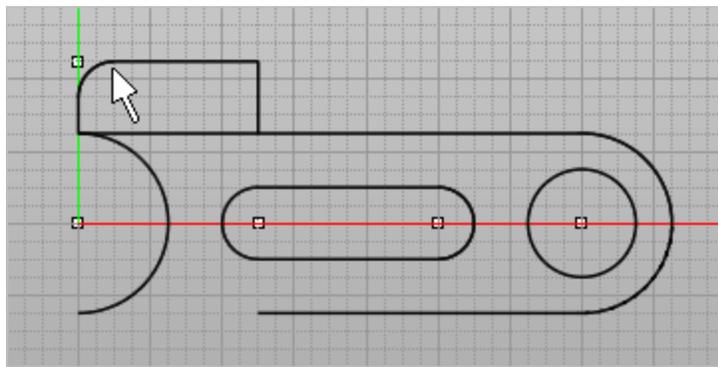
Command Prompt:

[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.

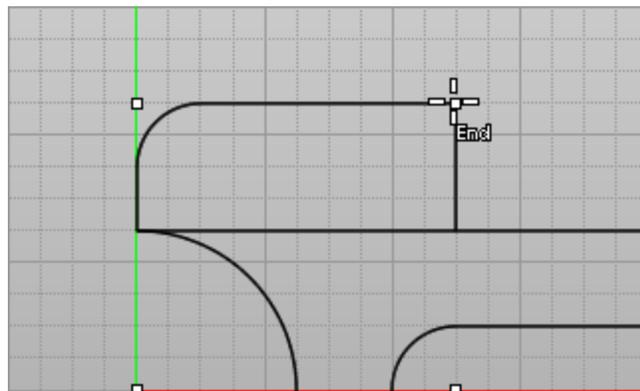


15. Let's now create another arc. Again, from the [Curve Modeling Ribbon Bar](#), select [Center, Start, Angle Pts.](#) 

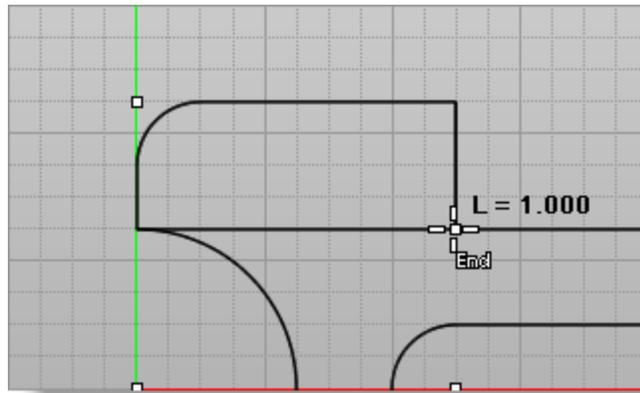
Command Prompt:

[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.



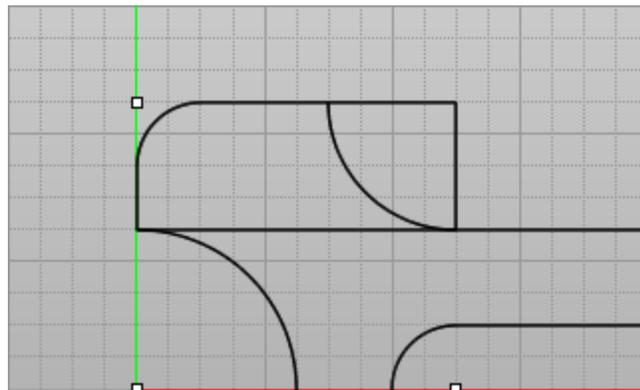
17. For [Start](#) pick the lower right corner of rectangle as shown below.



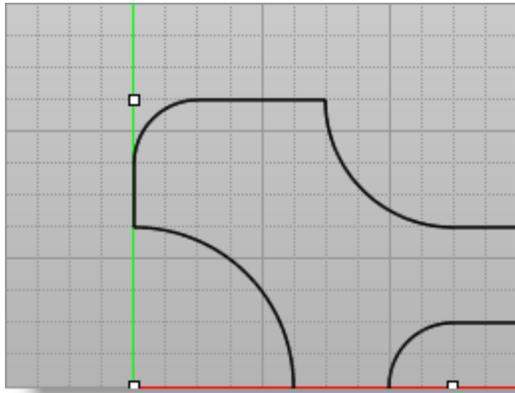
- 18. For **Angle**, type in **-90** and then press **Enter** to create an arc in the top right corner of the rectangle.

```
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=-90.000000]  
Command -90
```

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:

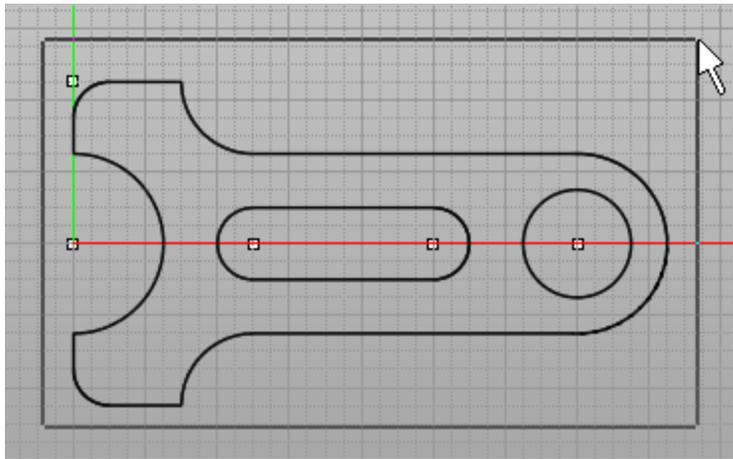


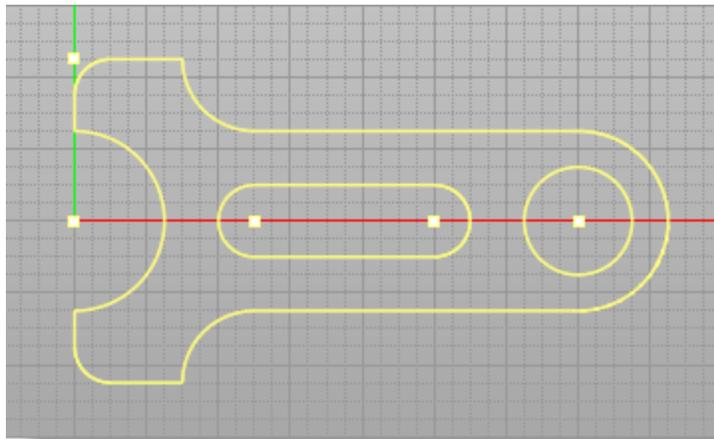
7.7 Merge Curves

Merge Curves

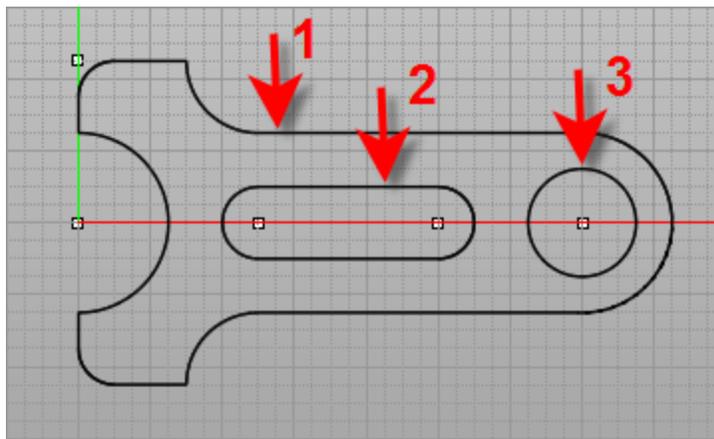
Now we will merge our curves into three separate closed curves.

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.

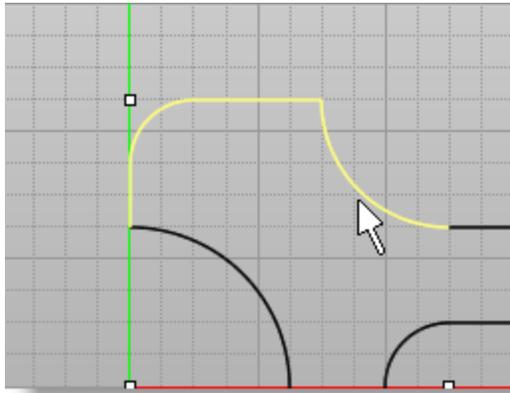


7.7.1 Mirror Curves

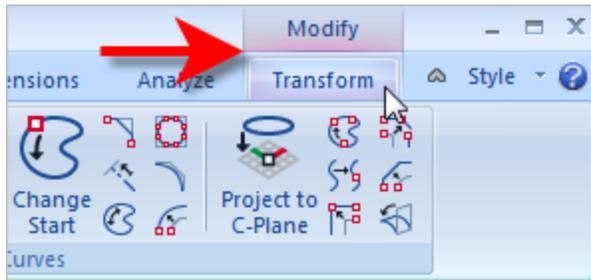
Mirror Curves

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.

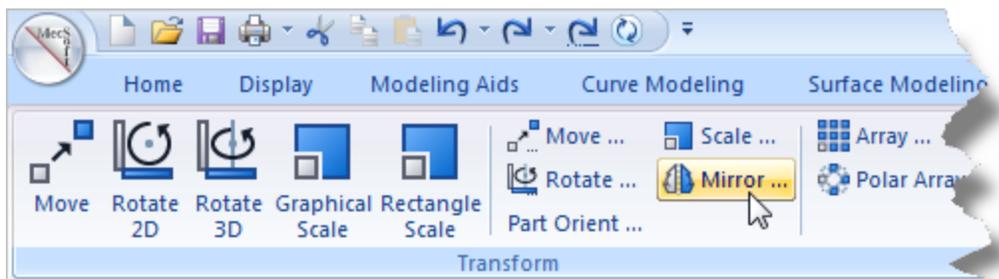


- With the 4 curves selected, you will notice that VisualCAD's Modify mode automatically becomes active and that the Transform Ribbon Bar becomes available. Refer to the menu image below:



The Transform Ribbon Bar is Activated

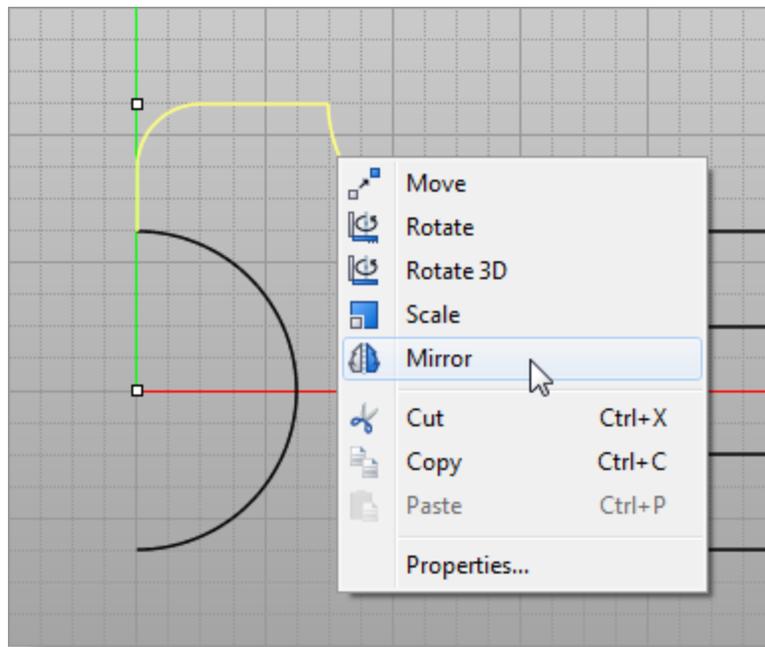
- Now there are two methods to activate the Mirror command. Select the method that suits you best. You can select the Transform Ribbon Bar tab and then select the Mirror icon as shown:



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-click Menu for Editing Objects

4. From the [Mirror selected objects](#) dialog box:

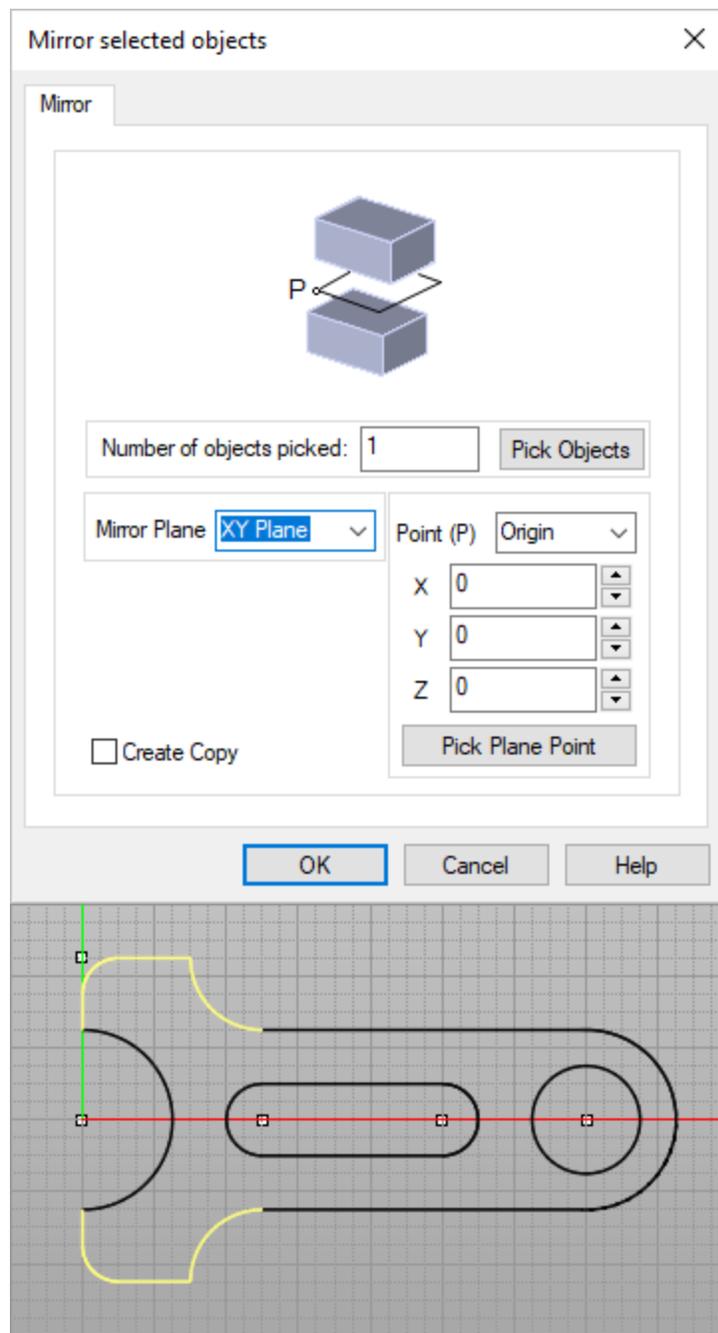
Set the [Mirror Plane](#) to [XZ Plane](#)

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.

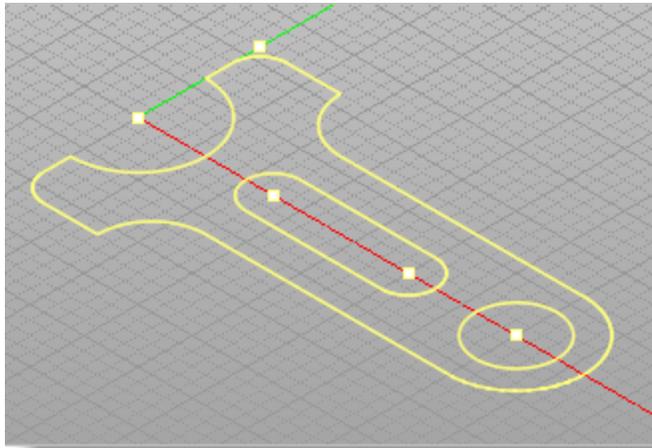
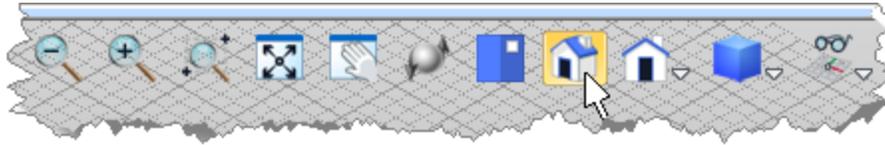


7.8 Extrude Curves

Extrude Curves

Now to complete the exercise we will extrude our 3 curves to create the final [Spanner 3D](#) model.

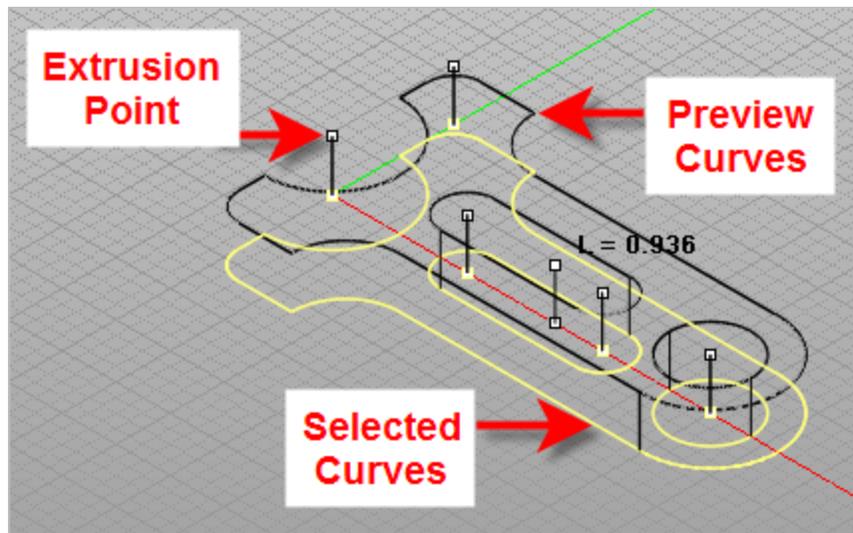
1. Select all the curves by using a crossing window selection:
2. Select the 3 closed curves.
3. Select **Iso** from the [View Toolbar](#) 



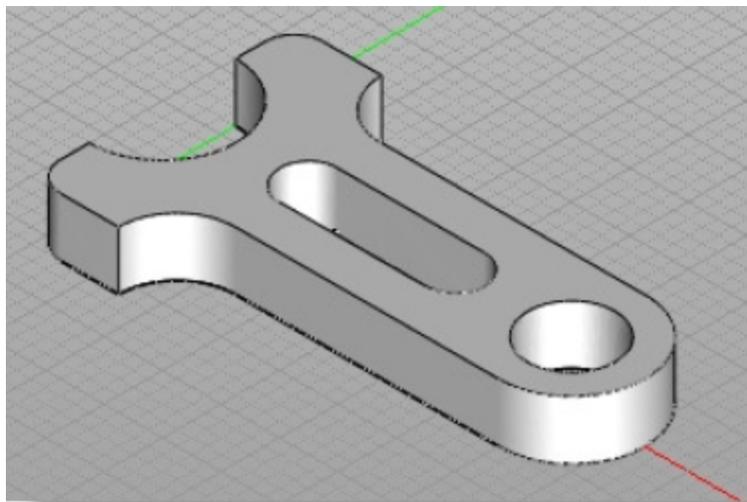
4. From the [Solid Modeling Ribbon Bar](#), select **Extrude**.
5. Preview curves will appear to determine the direction of extrusion. Set the extrusion to point in the positive Z direction by moving the cursor above the geometry. Then type **1.0** in the command input bar and press the **<Enter>** key.

Command Prompt:

Extrude:: Enter extrusion distance::[L=1.000000]



The curves are now extruded and displayed as shown below.

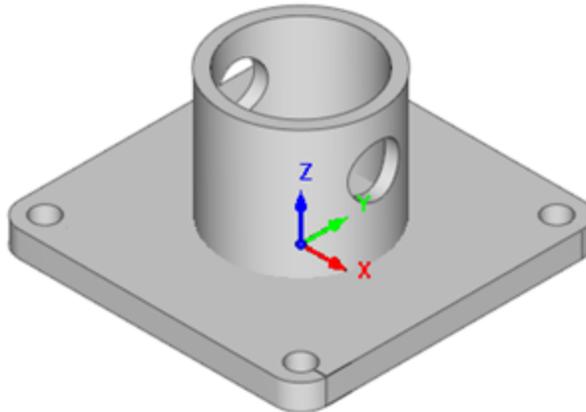


6. Save the file as [Spanner_Completed.vcp](#).

This completes this exercise.

#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:

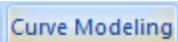
1. [Draw the Base Plate Profile.](#)
2. [Extrude the Base Plate.](#)
3. [Model the Tube.](#)
4. [Create a New Layer.](#)
5. [Edit Geometry Properties.](#)

8.1 Draw the Base Plate Profile



Draw the Base Plate Profile

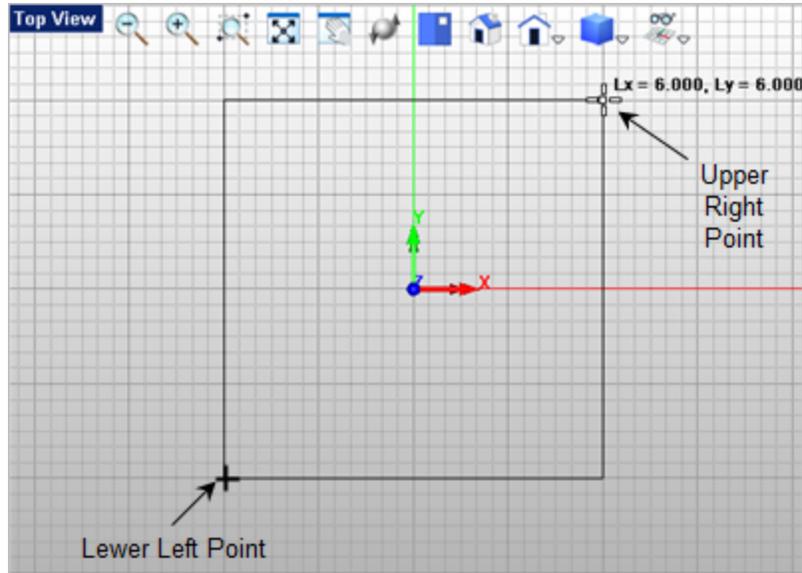
In this exercise you will learn the following VisualCAD concepts:

1. Select the **Curve Modeling** tab  from the top **Ribbon Bar**.

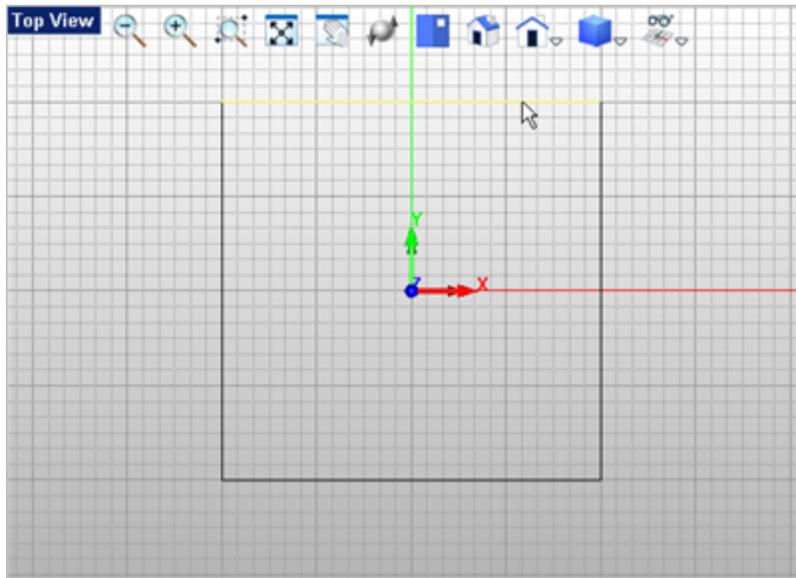


2. Now select the **Rectangle** command.
3. For the first corner of the rectangle, select the lower left grid point location shown below or type **-3,-3** in the **Command Prompt**  and press **<Enter>**.

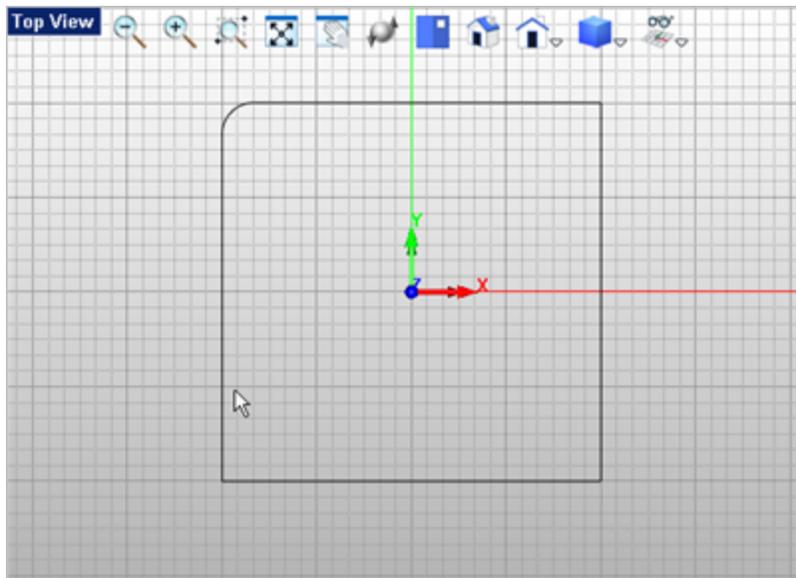
- For the second corner of the rectangle, select the upper right grid point shown below or type 3,3 in the **Command Prompt** and press <Enter>. This will create a rectangle 6" square.



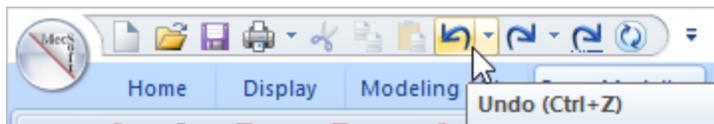
- To fillet the corners, select the **Fillet**  command from the **Curve Modeling**  tab.
- For the fillet **Radius**, enter 0.5 in the command prompt and press <Enter>.
- Now select the first edge to fillet as shown below. **Note:** When selecting curves to fillet, it is best to select furthest away from the corner that you wish to be filleted.

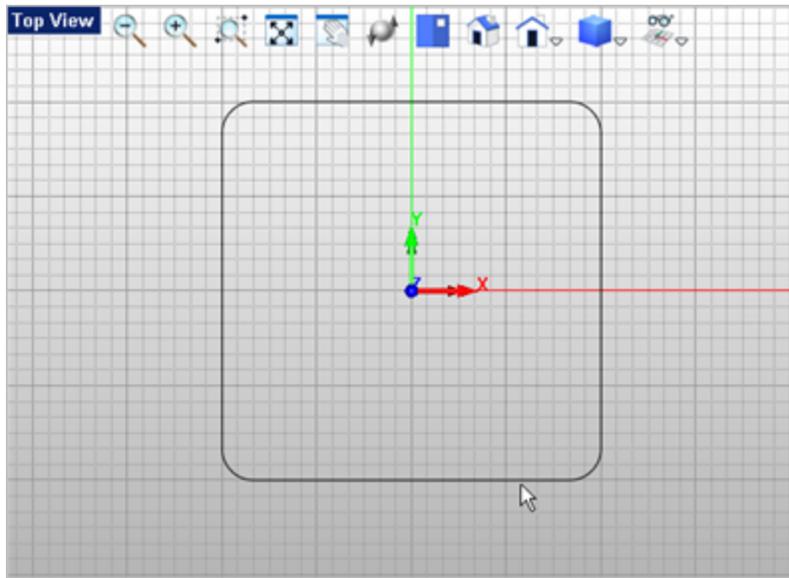


- 8. Now select the second edge to fillet and a 0.5 radius fillet is added to the rectangle as shown below.



- 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.



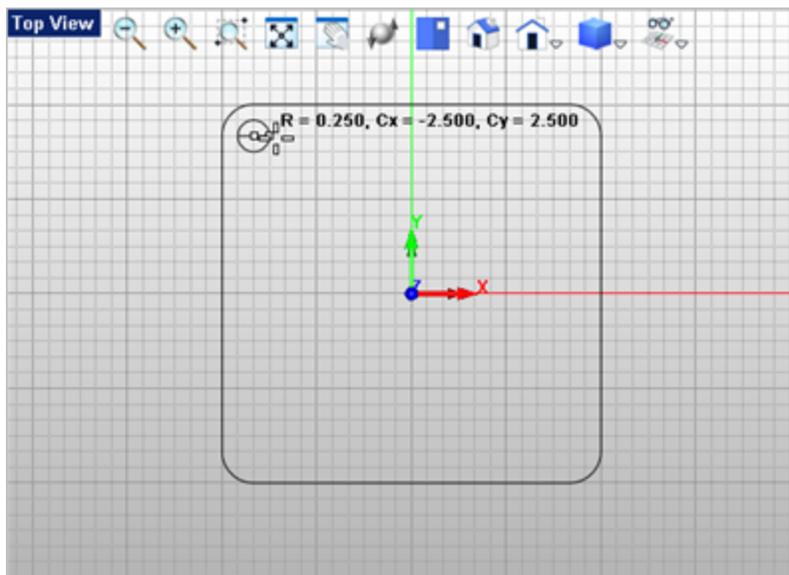


10. Now lets add the mounting holes. From the **Curve Modeling** [Curve Modeling](#) tab,



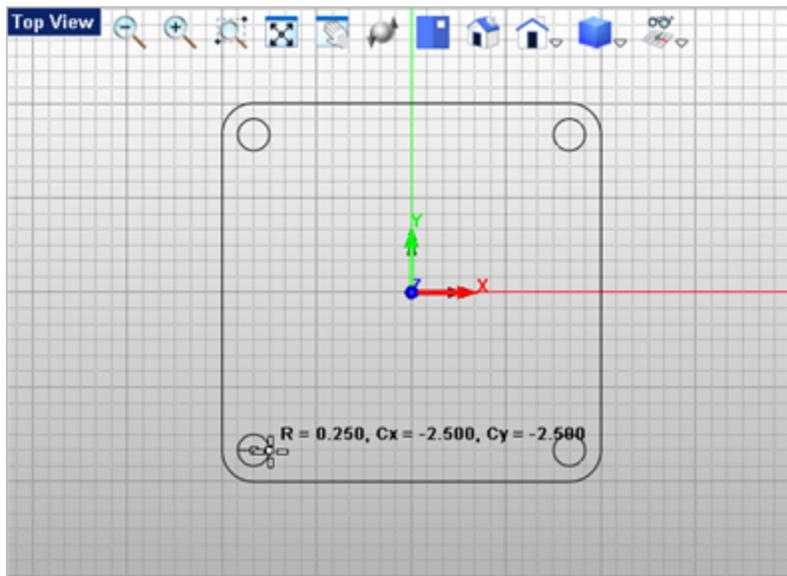
select the **Circle on Pt** command.

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 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>.

- Repeat the **Circle on Pt** command and draw the other three circles in the locations shown below.

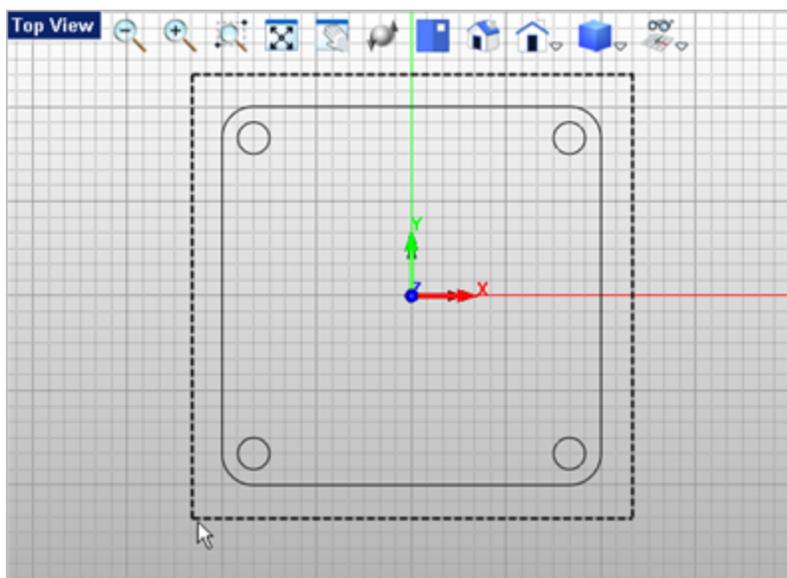


8.2 Extrude the Base Plate

Extrude the Base Plate

In this step we will extrude our drawing to model the base plate in 3D.

- 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 inside the window is selected. If you drag from right to left everything that crosses over the window will be selected.

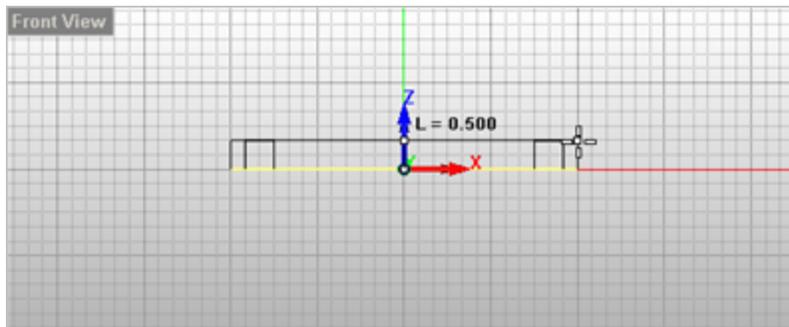


2. After your drawing is selected, go to the [Mesh Modeling](#) tab **Mesh Modeling** from

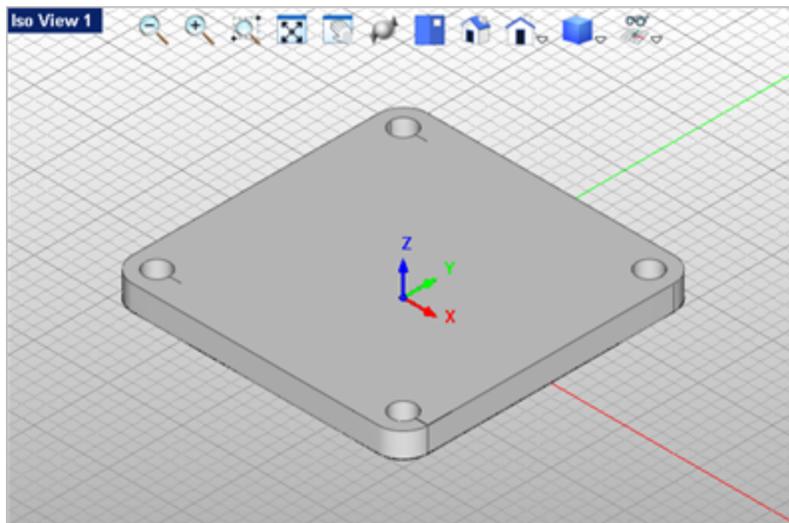


the [Ribbon Bar](#) and select the [Extrude](#) command.

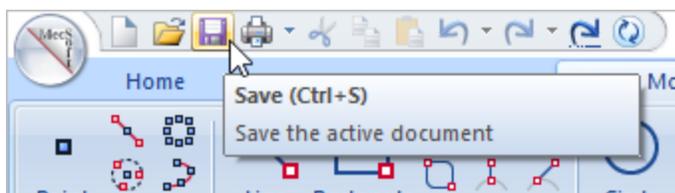
3. Now move the cursor down to the [Front View](#) **Front View** and you will see that you can drag the height of the extrusion. Select the second grid point in the positive Z direction or type [0.5](#) and press [<Enter>](#).



You will see the extrusion display in all four viewports. The [Iso View 1](#) **Iso View 1** view is shown below.



4. [Save](#) your work by selecting the [Save](#) icon from the [Quick Access toolbar](#).

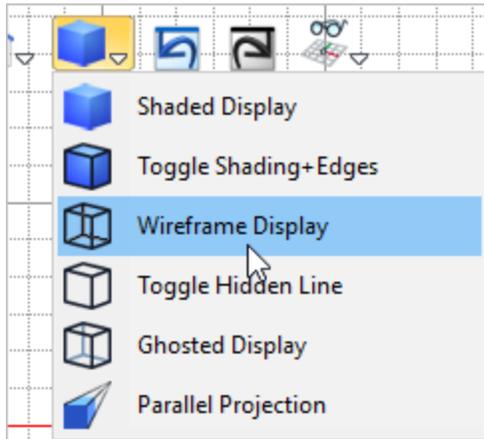


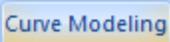
8.3 Model the Tube

 **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.

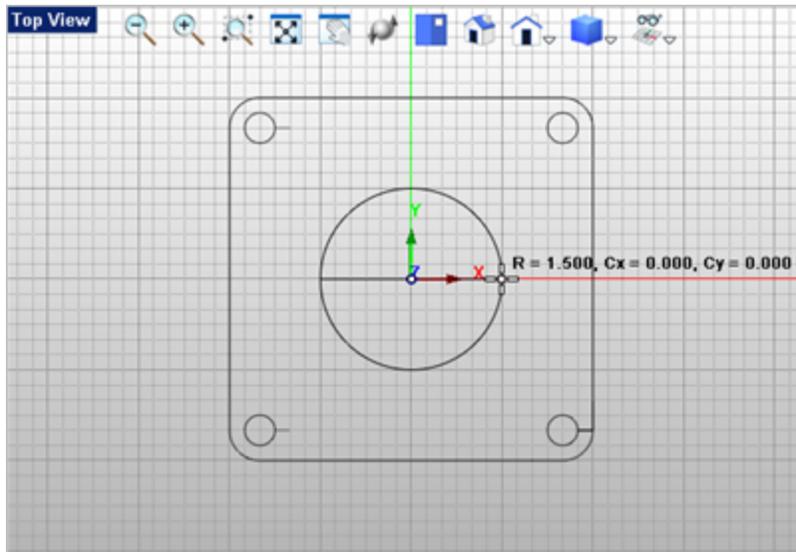


3. Now from the **Curve Modeling**  tab select the **Circle on Point**

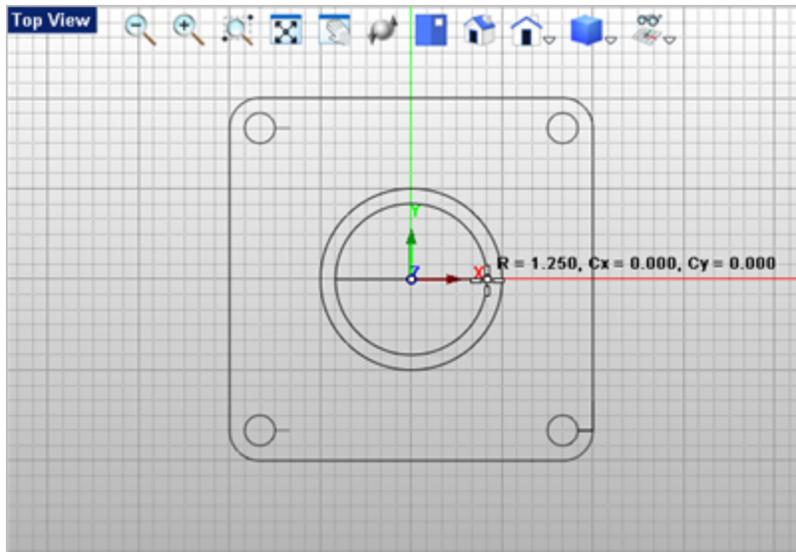


command again.

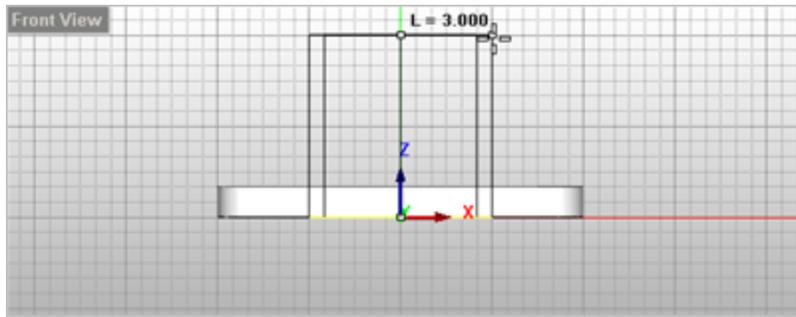
4. For the center point select the grid point at **0,0** center of the drawing.
5. Now for the circle diameter you can select the grid point shown below or type in **3** and press **<Enter>**. A circle 3" in diameter is drawn.



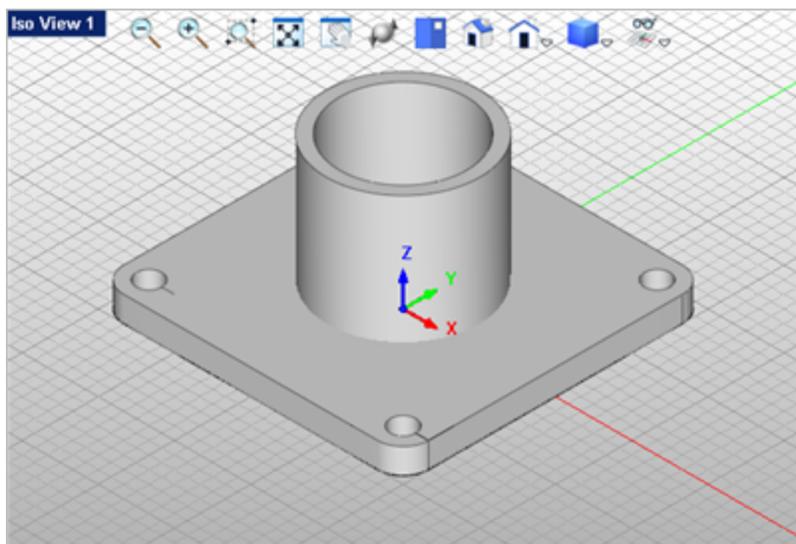
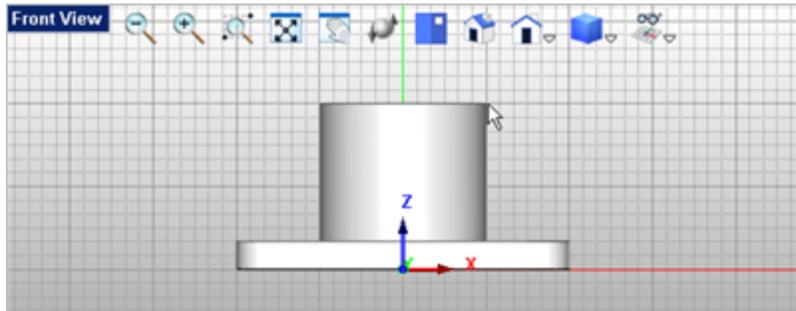
6. Now repeat the command, this time drawing a circle with a diameter of 2.5" in diameter.



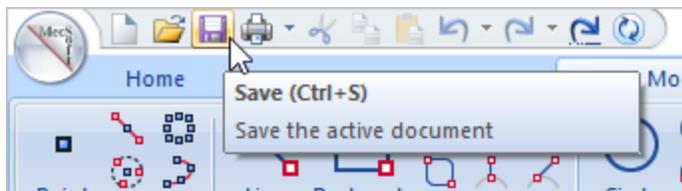
7. Once the two concentric circles are drawn, select them both. You can window select them or while holding down the <Ctrl> key, just pick both of them.
8. With the two circles selected, go to the **Mesh Modeling** [Mesh Modeling](#) tab and  select the **Extrude**  command again.
9. Now move the cursor to the **Front View** [Front View](#) and select the grid point located at 3" in the positive Z direction as shown below.



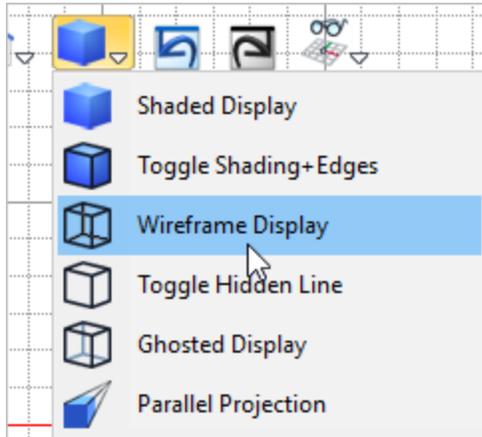
10. The center tube is now created and shown in each window.

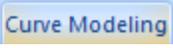


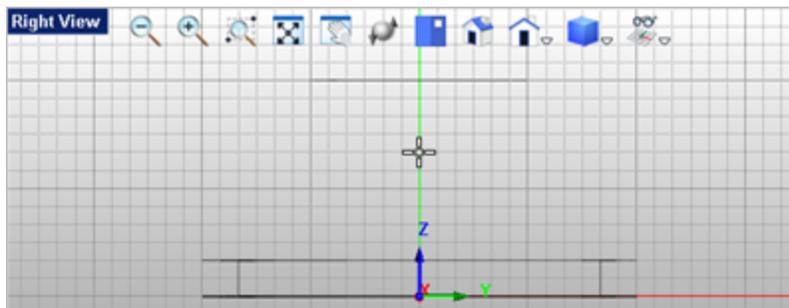
11. Save your work by selecting the Save icon from the [Quick Access Toolbar](#).



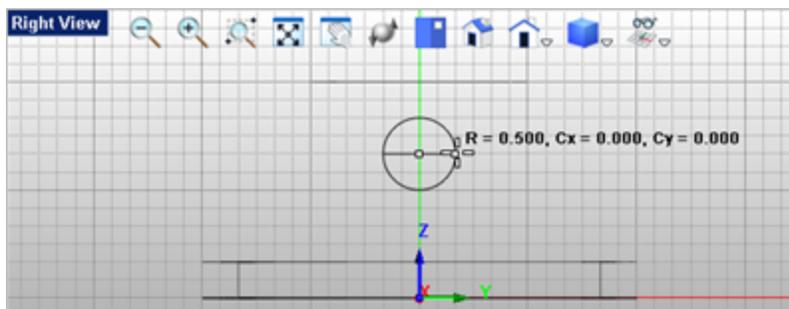
12. Activate the **Right View**  by selecting anywhere on the grid.
13. From the **View Toolbar**, drop down the **Display**  menu and select **Wireframe Display**. Again, this will allow us to better see our next drawing.



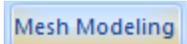
14. From the **Curve Modeling**  tab, select the **Circle on Pt** command again.
15. For the center point of the circle, select the grid point shown below.



16. Now for the circle diameter you can select the grid point shown below or enter **1.0** for the diameter.

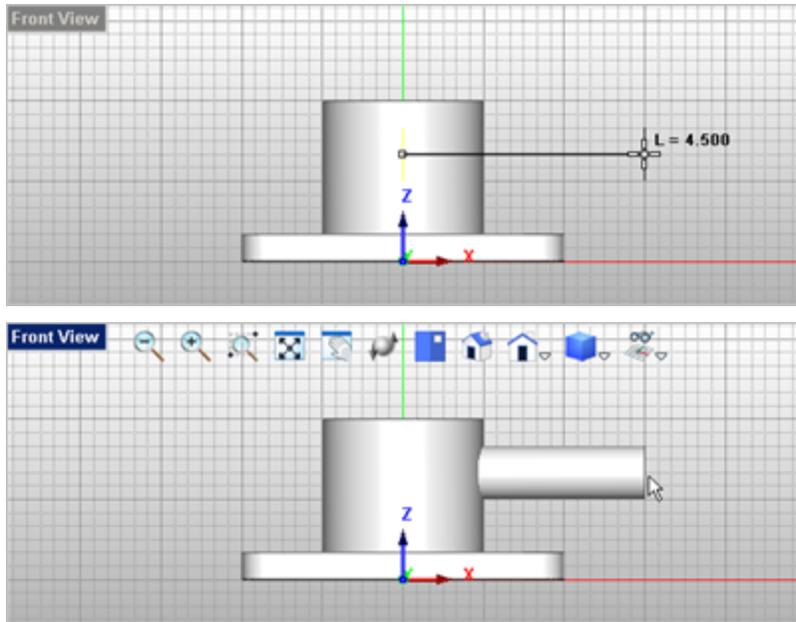


17. Now select the circle you just created, then go to the Mesh Modeling

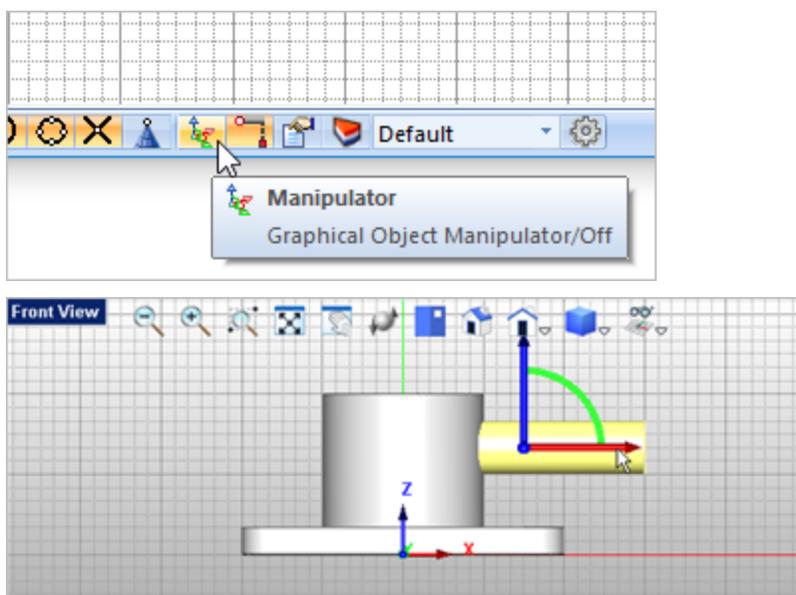


tab and select the Extrude command again.

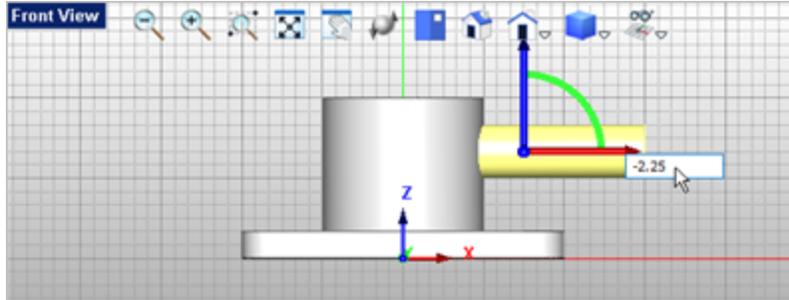
18. For the length of the cylinder move the cursor over to the Front View and select the grid point shown below.



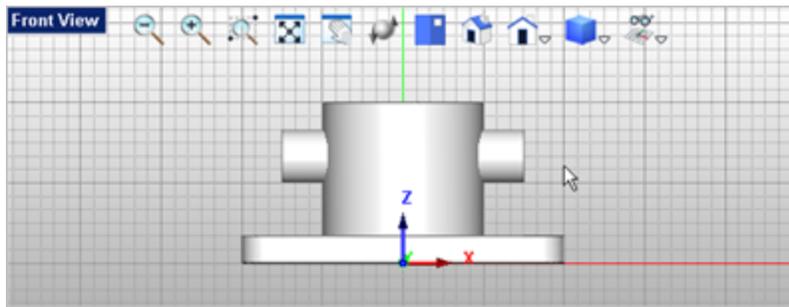
19. Now select the cylinder and toggle the display of the Graphic Manipulator on. This toggle icon is located on the Status Bar at bottom of the display.



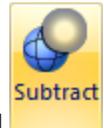
20. With the cylinder selected pick on the red X axis of the [Graphic Manipulator](#) to display the input box and enter -2.25 and press <Enter> as shown below.



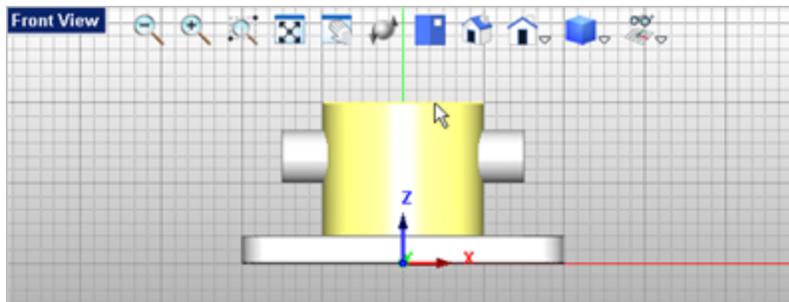
The cylinder is translated to the left negative X direction by 2.25 inches so that it passes complete through the tube on both sides.



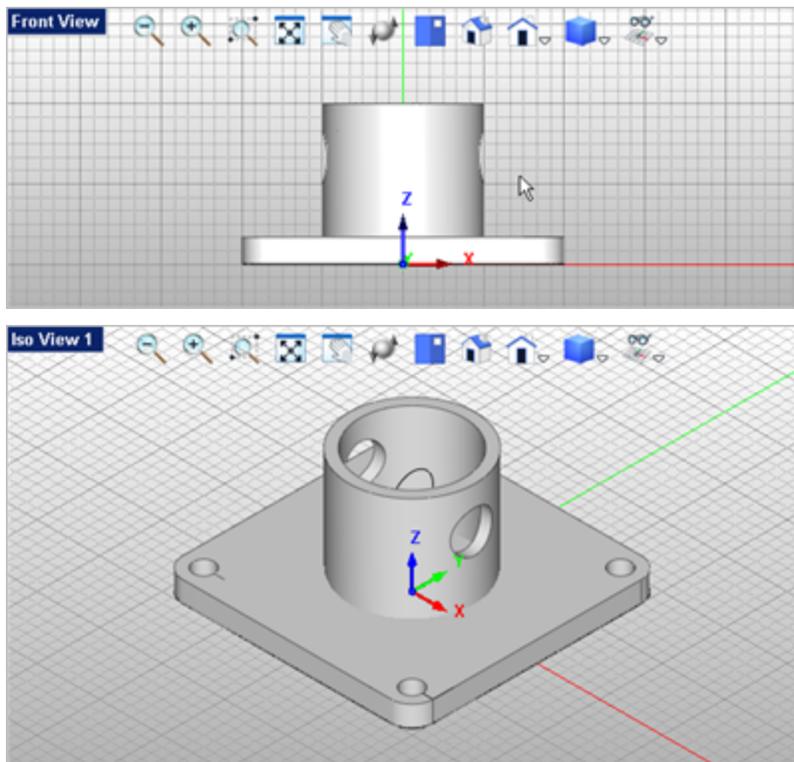
21. You can toggle the [Graphic Manipulator](#)  off.



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.

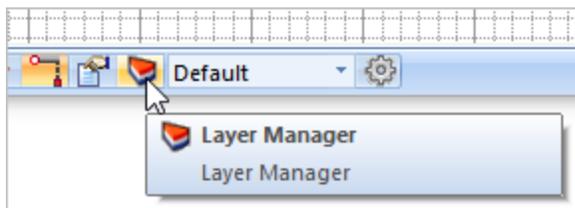


8.4 Create a New Layer

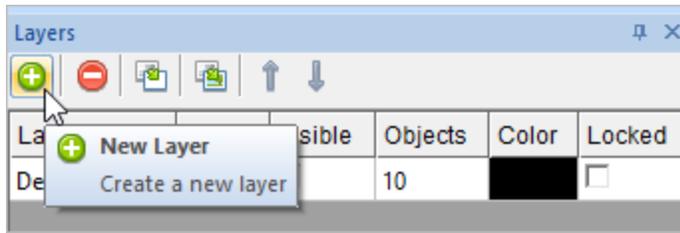
Create a New Layer

In this step we will create a new layer, turn it off and then move our curves to it to hide them from the display.

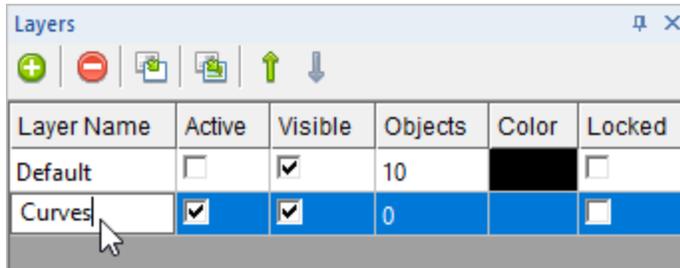
1. From the **Status Toolbar**, select the icon to display the **Layer Manager** .



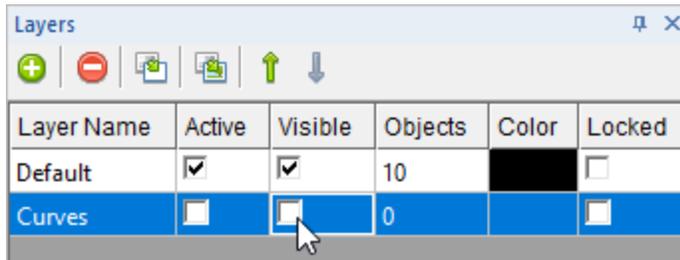
2. At the top of the **Layer Manager** select the **New Layer** icon and a new layer will be added to the list.



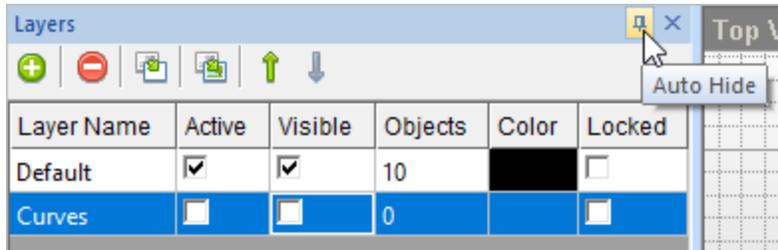
- Now double-left click on the **New Layer** name and type in the names "Curves" and press <Enter>.

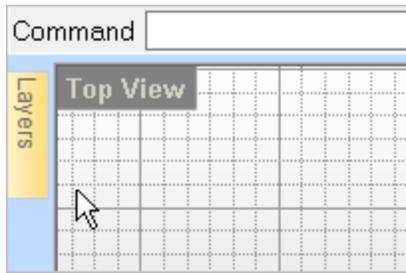


- Now to hide the **Curves** layer first check the box to make the **Default** layer the **Active** layer.
- Then uncheck the box to make the **Curves** layer hidden.



- 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.



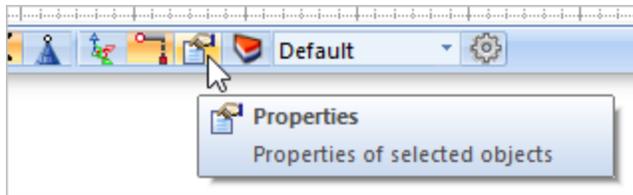


8.5 Edit Geometry Properties

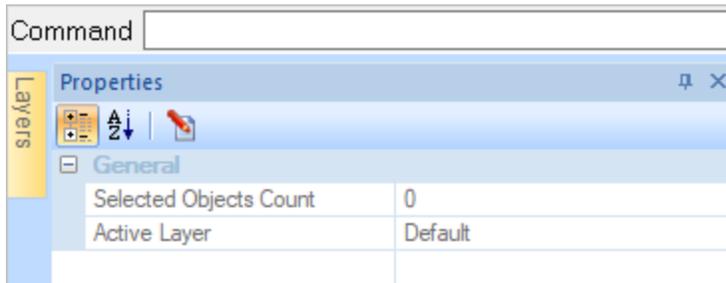
Edit Properties of Geometry

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.

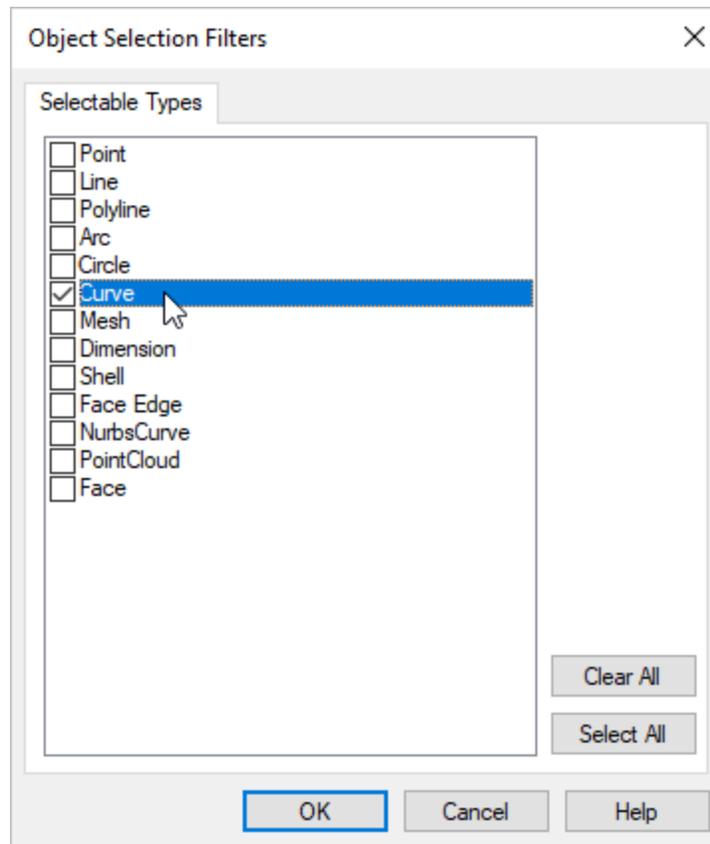
1. Let's start by selecting the **Properties** icon  located on the **Status Toolbar** at the bottom of the display (just to the left of the **Layer Manager**  icon).



2. You will see the **Properties Manager** dialog display on the left side of the display.

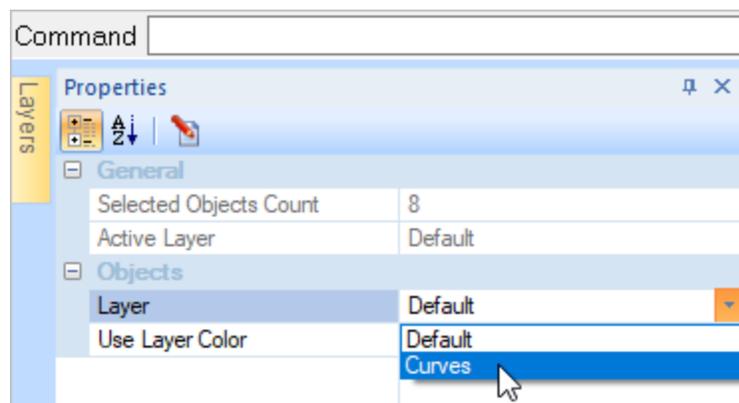


3. Now we will automatically select all of our curve geometry. Go to the **Modeling Aids** tab and select pick the **Select by Type**  icon. This will display the **Object Selection Filters** dialog.

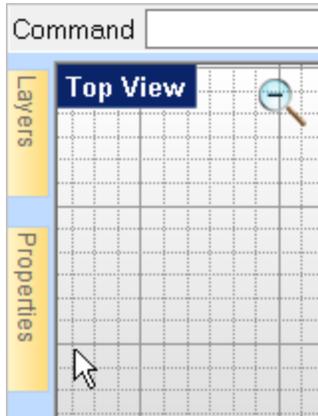
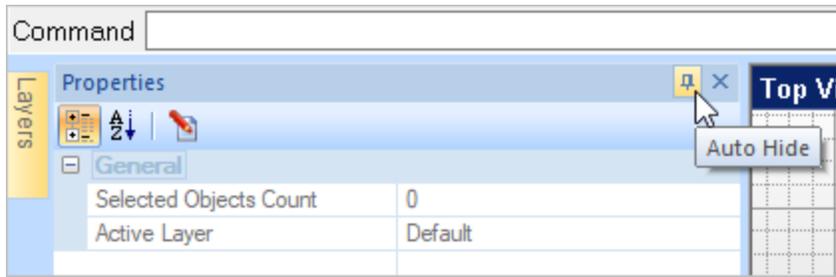


4. From the dialog check the box for **Curves** and uncheck all other boxes and then pick **OK** to close the dialog. You will see that all of the curves are highlighted for selection.

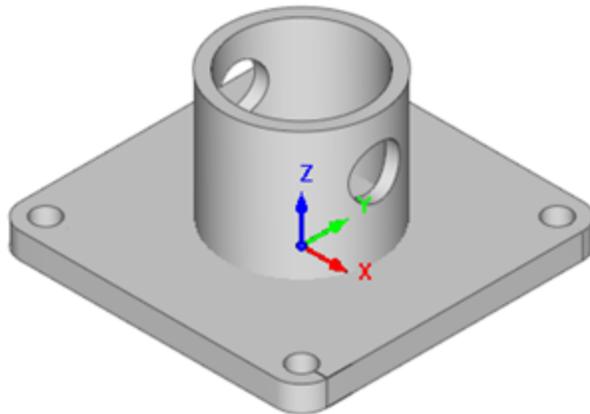
Now go to the **Properties Manager** and under the **Objects** section drop down the menu for **Layer** and select **Curves**.



5. Now close the **Properties Manager** by selecting the **Auto Hide** pin located at the top right of the **Properties Manager**.

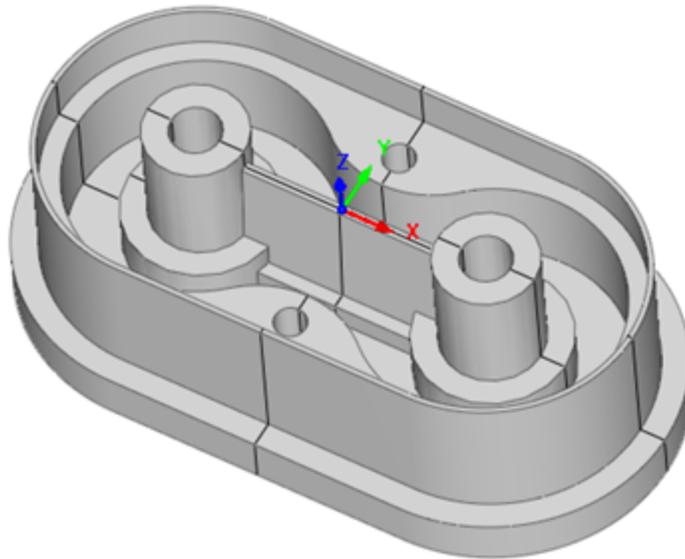


- 6. This exercise is complete and your part now looks like this:
- 7. Save the file as [Base_Plate_Completed.vcp](#).



#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:

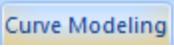
In this exercise you will perform the following [VisualCAD](#) tasks:

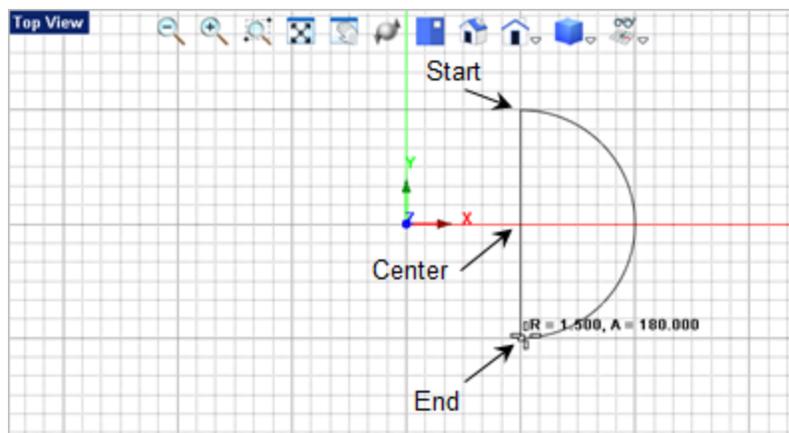
1. [Extrude the Body.](#)
2. [Extrude the Flange.](#)
3. [Extrude the Upper Pocket.](#)
4. [Extrude the Lower Pocket.](#)
5. [Revolve the Center Bosses.](#)
6. [Extrude the Connection Bar.](#)
7. [Extrude the Connection Wall.](#)
8. [Extrude Ejector Pin Holes.](#)
9. [Change Geometry Layer.](#)
10. [Create Section Curves.](#)

9.1 Extrude the Body

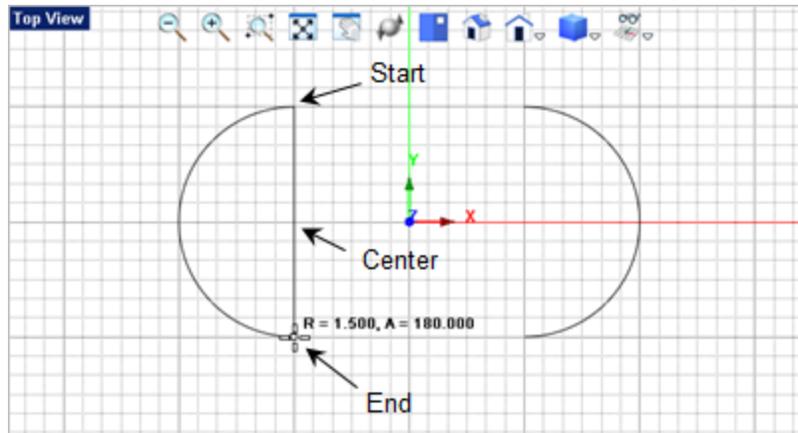
Extrude the Body

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 **Command** . 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.

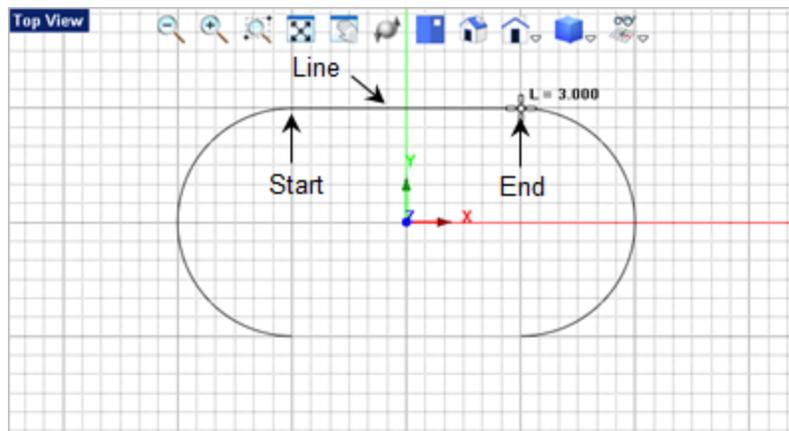


9. Now we want to draw a line to connect the two arcs across the top. From the [Curve](#)

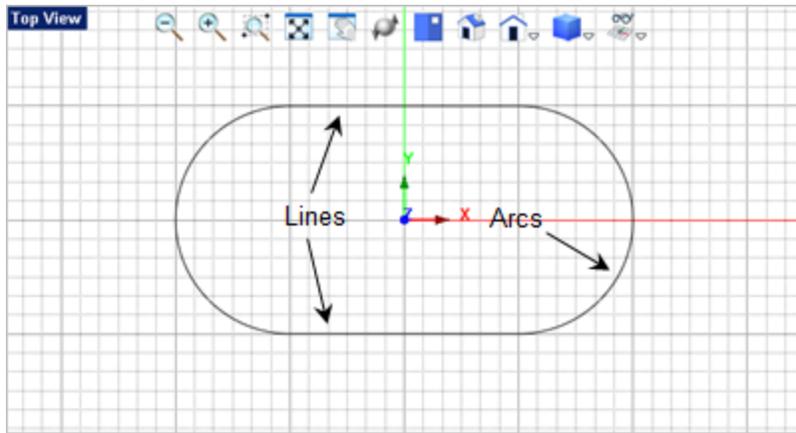


Modeling tab [Curve Modeling](#) select the [Line](#) command

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](#)



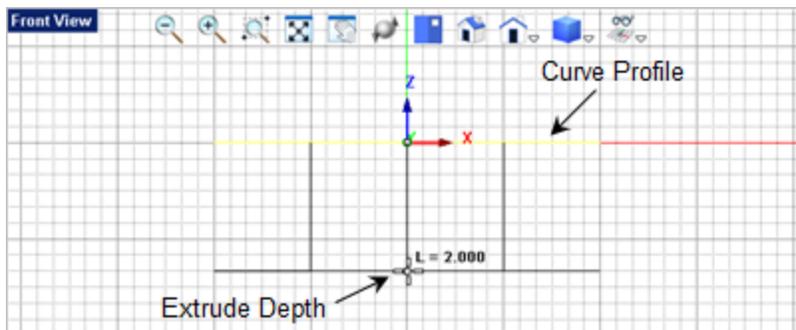
[Modeling](#) tab [Curve Modeling](#) select the [Merge Curves](#) command and then right-click the mouse to merge them. If you pick anywhere on the geometry you will see that one curve is now defined.

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](#)

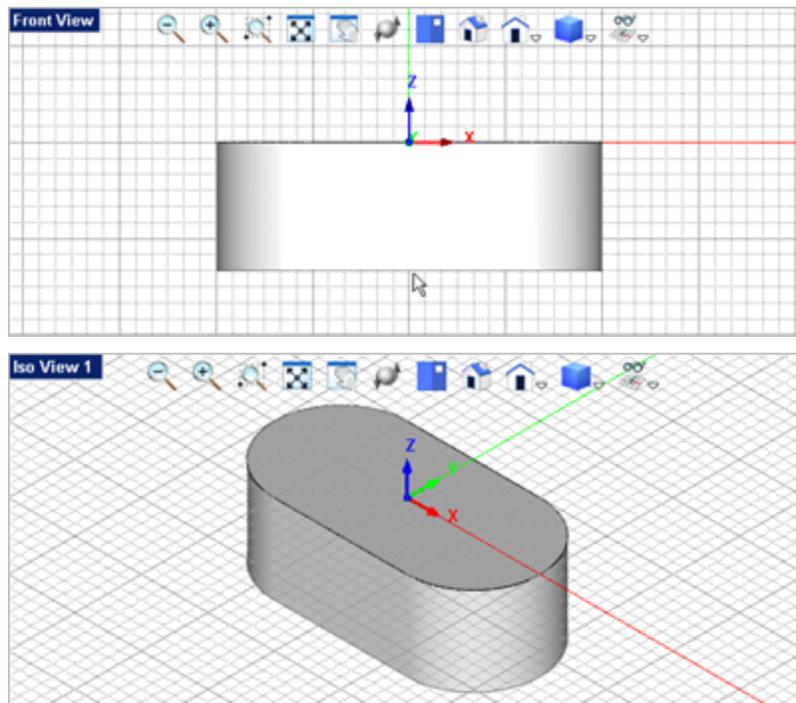


command.

15. For the extrusion distance, drag the cursor down to the [Front View](#) [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.



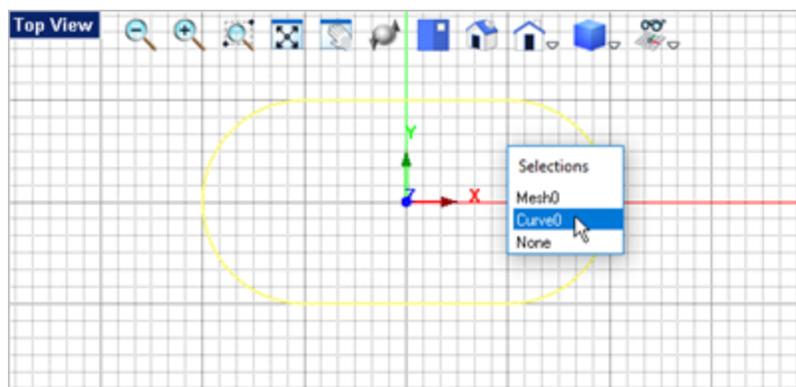
9.2 Extrude the Flange

Extrude the Flange

Now we will extrude the flange.

1. Activate the **Top View** **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**.



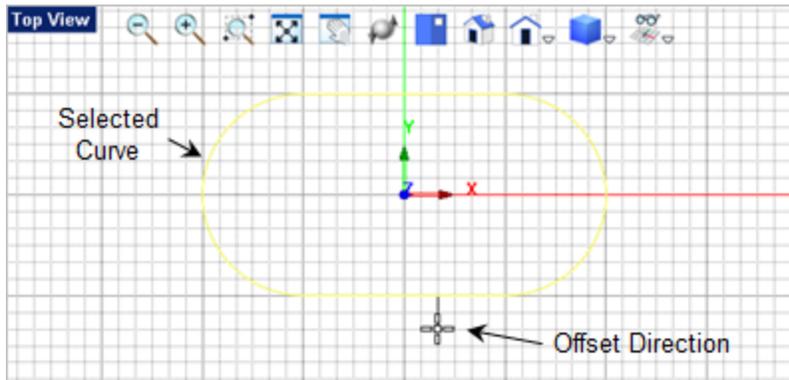
2. From the selection list pick **Curve**. The number value to the right of the curve may change depending on when the entity is created.

3. With the curve selected, from the **Curve Modeling** tab **Curve Modeling** select the



Offset Curve command.

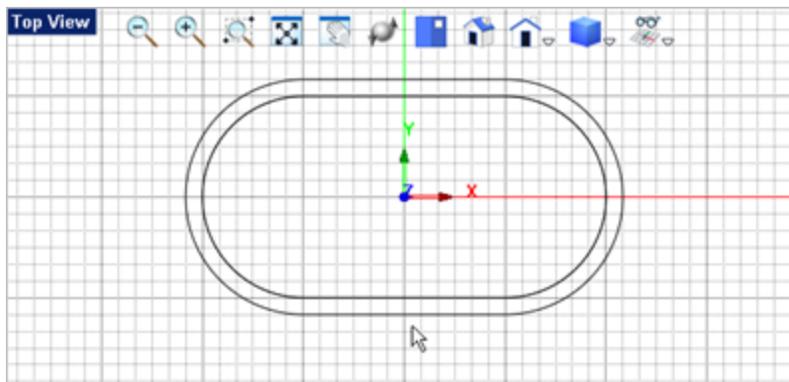
4. If you move the cursor you see that an offset direction is indicated. Move the cursor below and on the outside of the curve as shown below.



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**

Command and press **<Enter>** to create the offset curve shown below.

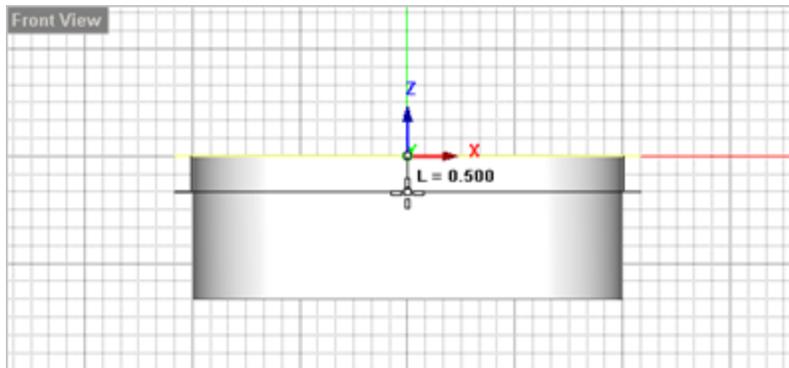


6. Now select the offset curve you just created, go to the **Mesh Modeling** tab

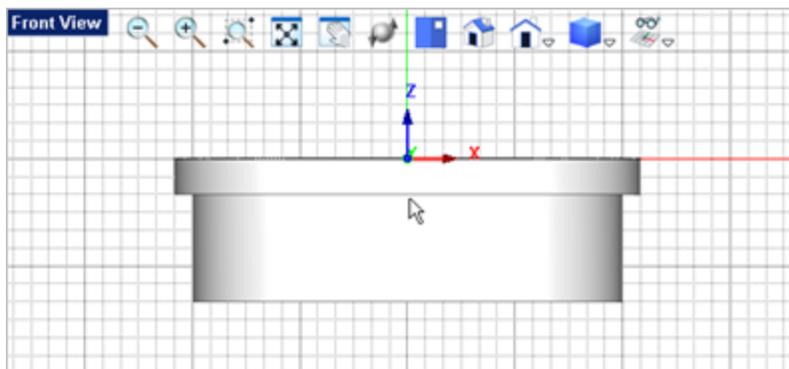


Mesh Modeling and pick the **Extrude Mesh** command again.

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"**.



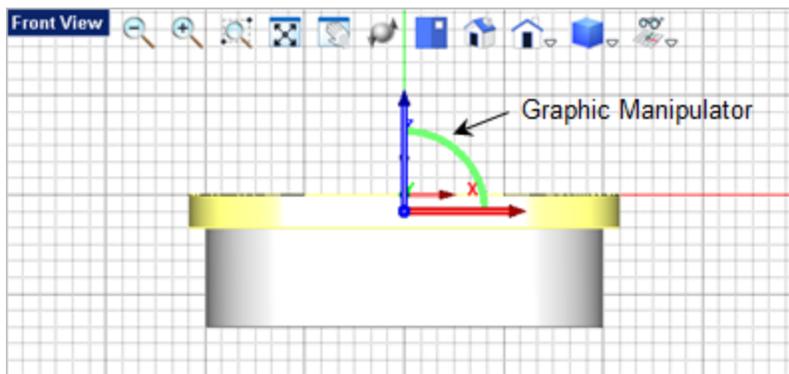
The flange is created and displayed.



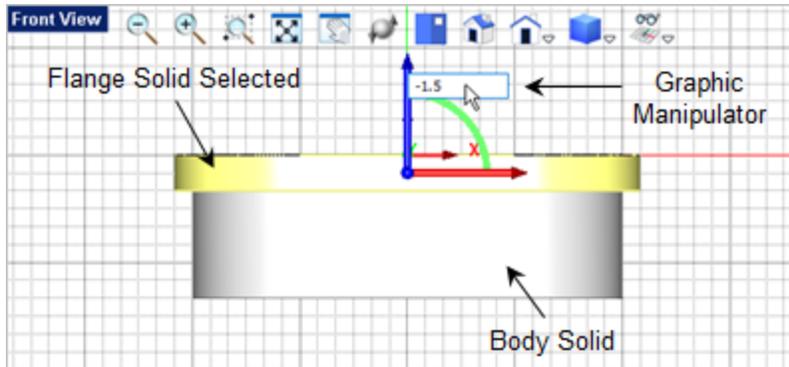
8. Now lets move the flange to the bottom of the body. To do this we will use the **Graphic Manipulator**. To toggle the **Graphic Manipulator** on, go to the **Status Bar** and select the icon .

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**.

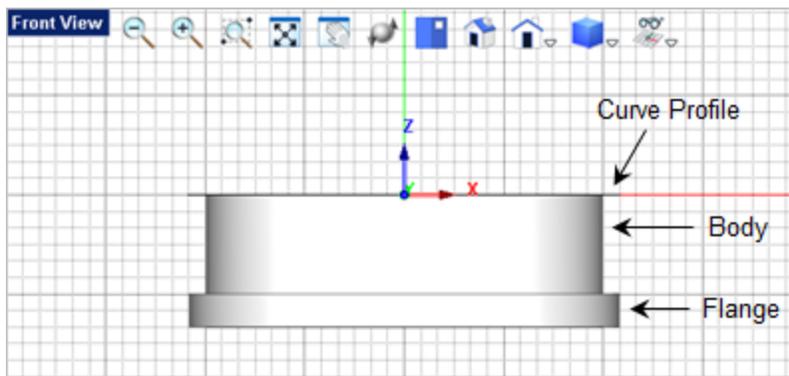
9. Now select the flange and you will see that the **Graphic Manipulator** is displayed on top of the selected entity.



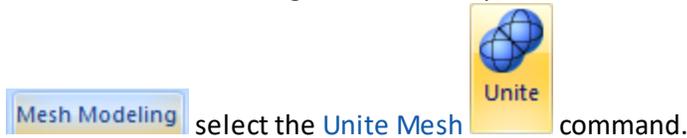
- 10. Pick on the blue Z Axis arrow to display the input window.
- 11. Enter -1.5 Command and press <Enter>.



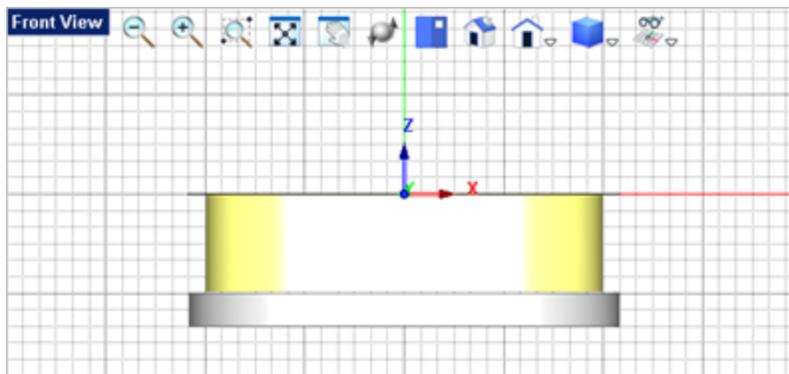
The flange is moved 1.5" in the negative Z direction.



- 12. Now lets unite the flange with the body. From the Mesh Modeling tab,



- 13. For the first mesh solid select the body.



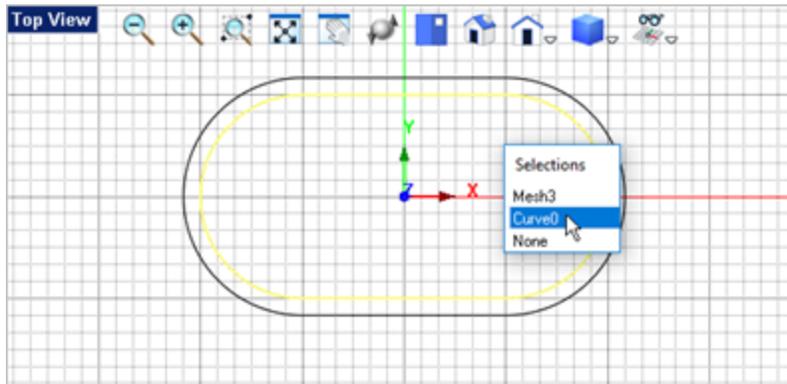
- For the second mesh solid select the flange. The two will be joined into one mesh solid.

9.3 Extrude the Upper Pocket

Extrude the Upper Pocket

In this step we will create the upper pocket and subtract it from the body.

- From the **Top View** **Top View** select the profile curve of the body by clicking on it and selecting **Curve** from the selection list.

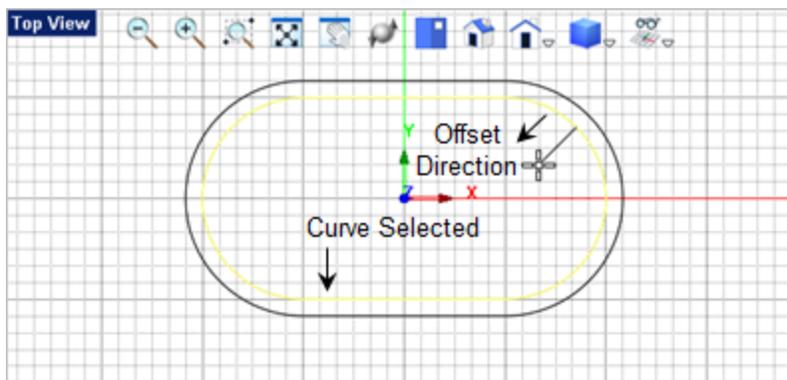


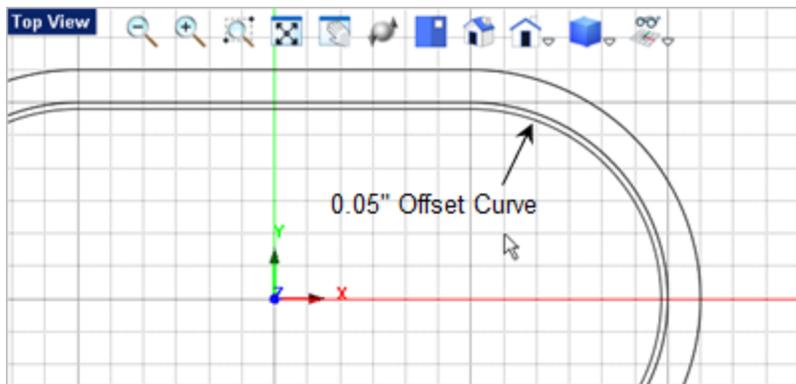
- With the curve select, go to the **Curve Modeling** tab **Curve Modeling** and select the



Offset Curve command.

- Drag the offset direction to the inner side of the curve and then enter **0.05**
 and press <Enter>.



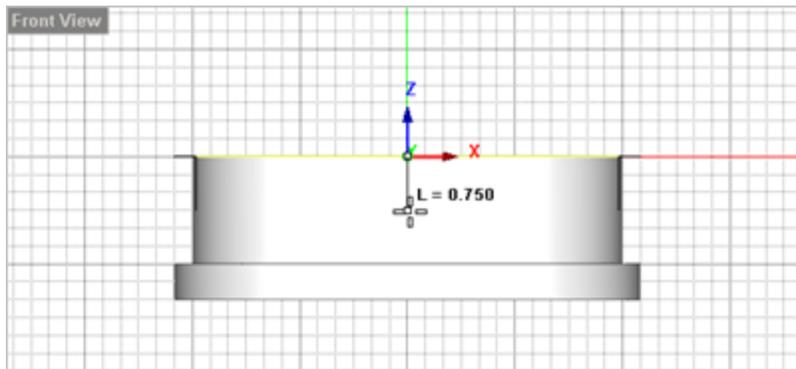


4. Now select the offset curve, go to the **Mesh Modeling** tab and select

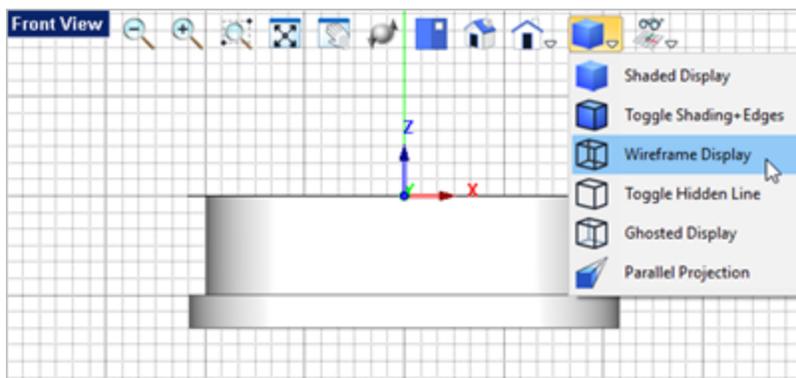


the **Extrude Mesh** command.

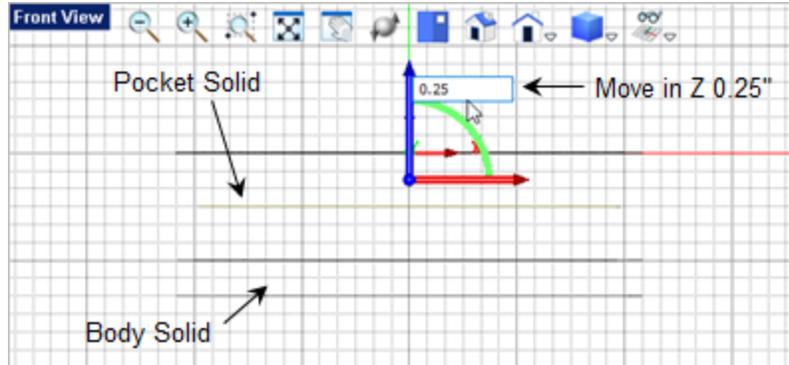
5. Drag the cursor down to the **Front View** and select the grid point at **0.750"** as shown below.



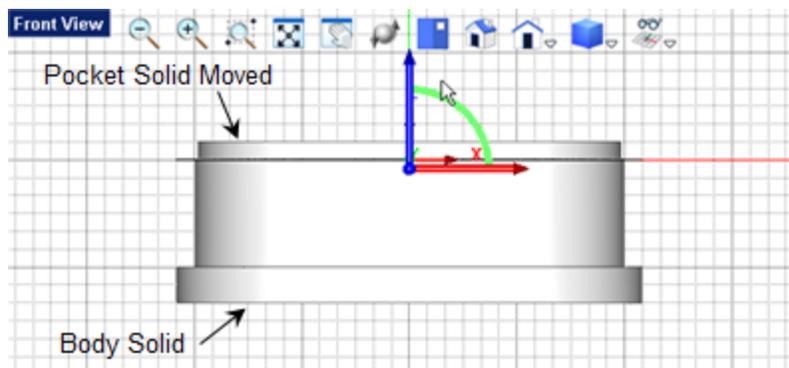
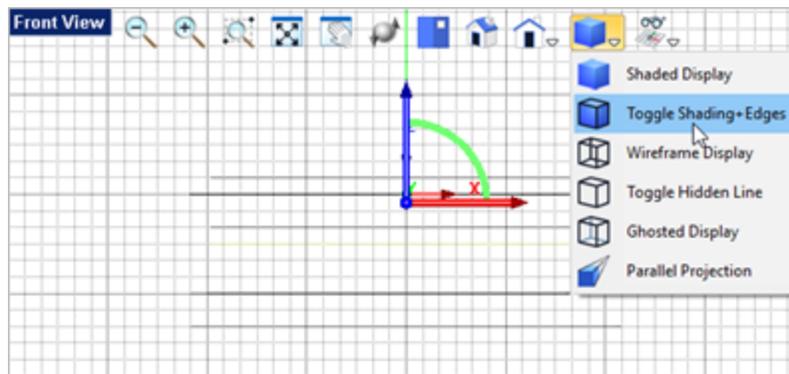
6. Now to see the pocket mesh solid, from the **View Toolbar** select the **Wireframe Display** item from the display menu.



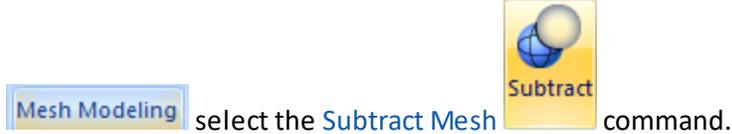
7. We want to move the pocket mesh solid so make sure the [Graphic Manipulator](#) icon is still toggled on  (it's located on the [Status Bar](#) remember?).
8. With the [Front View](#) **Front View** set to [Wireframe display](#) you can see to select the pocket mesh solid. Select it now and the [Graphic 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".



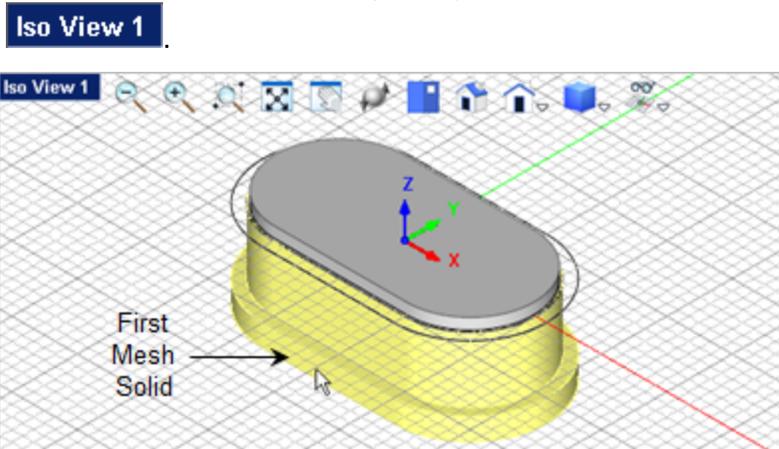
10. From the [View Toolbar](#)  drop down the display menu and select [Toggle Shading + Edges](#).



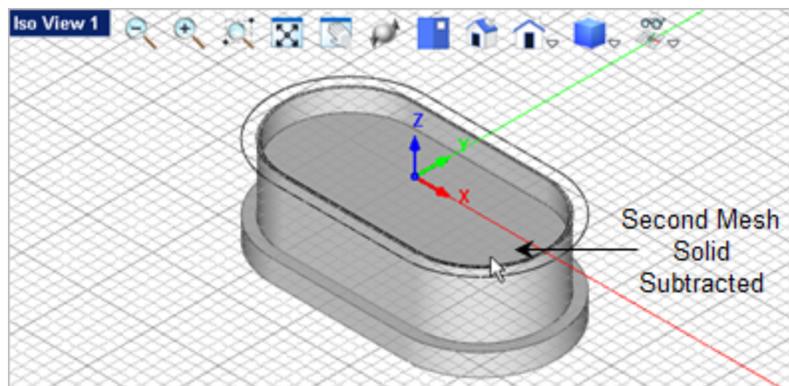
11. Now lets subtract the pocket from the body. From the **Mesh Modeling** tab



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.



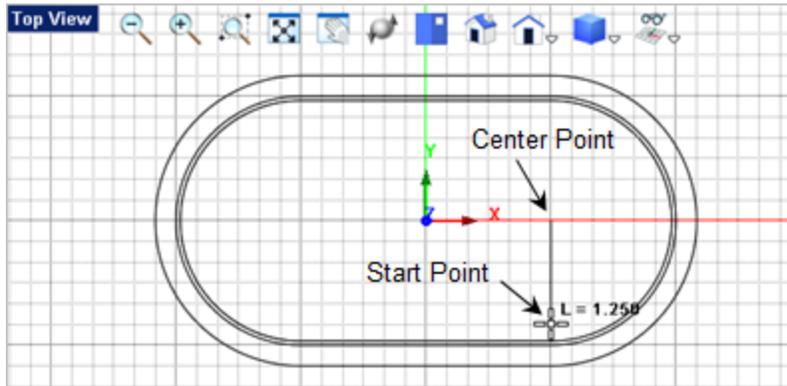
9.4 Extrude the Lower Pocket

Extrude the Lower Pocket

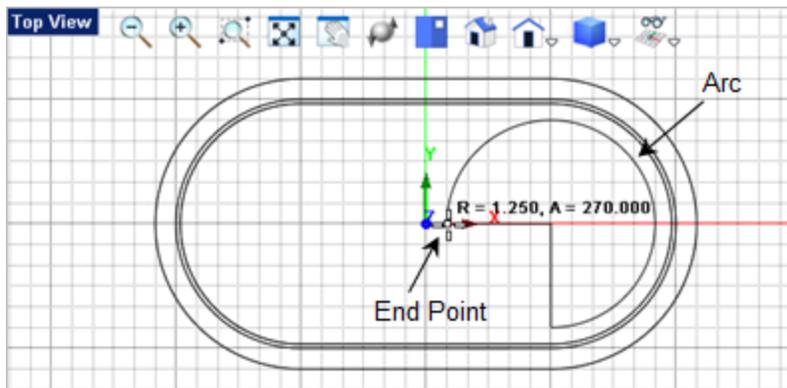
In this step we will create the lower pocket mesh solid and subtract it from the body.

1. We will start in the **Top View** **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 **Curve Modeling**, select select the **Arc Center, Start, Angle** command  again.

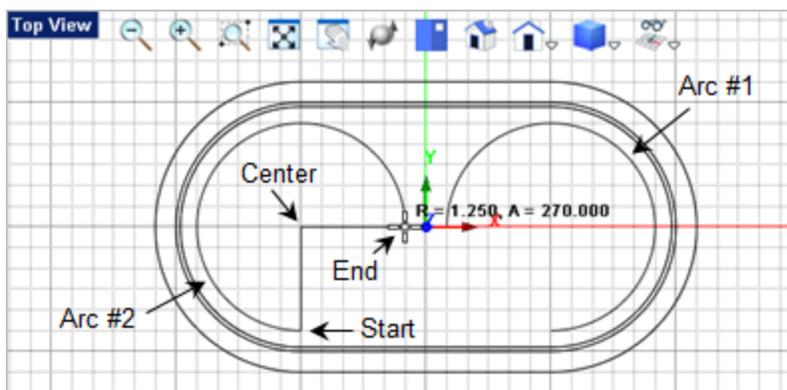
3. Select the center point first and then select the start point. Both are shown below.



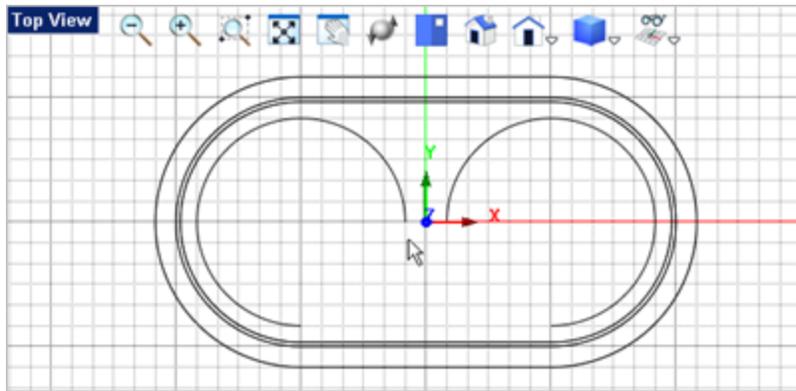
4. Now for the end point of the arc, drag the cursor counter clockwise until the arc preview is at 270 degrees as shown below.



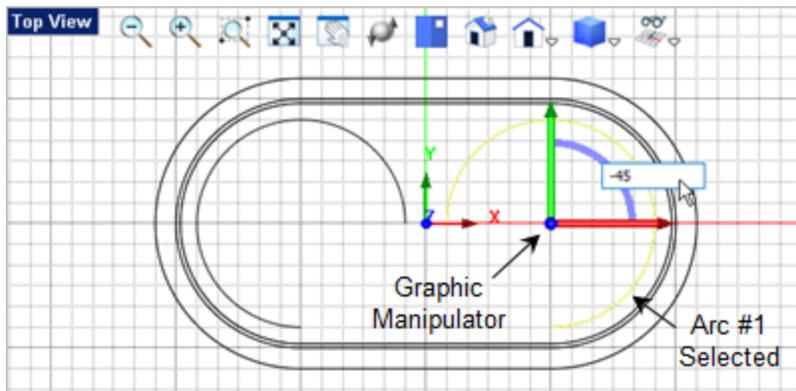
5. Now press the <Enter> key to repeat the command. This time select the center, start and end points shown below.



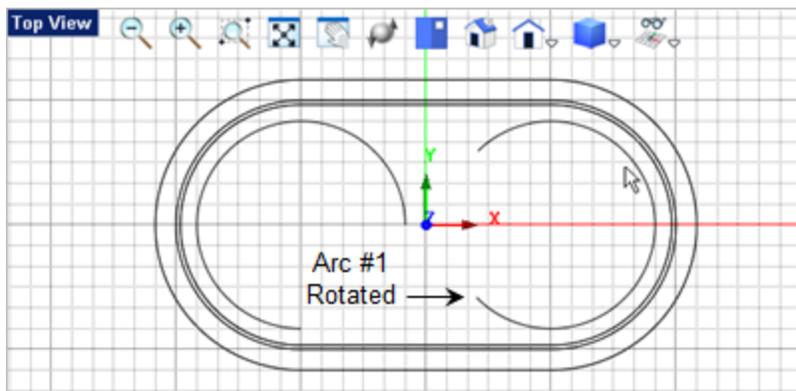
You drawing should now look like this:



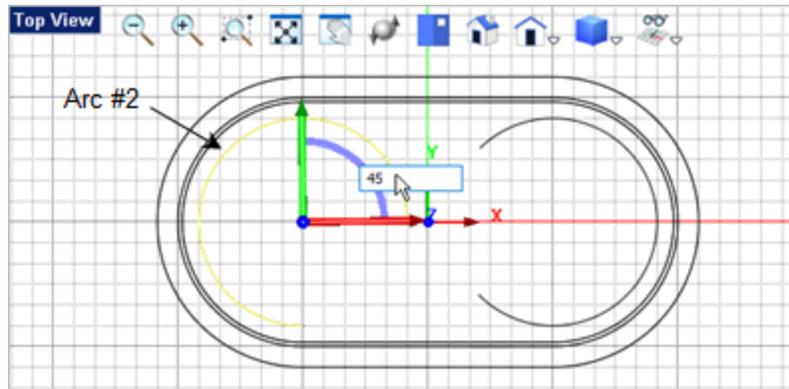
- 6. Now we will rotate each arc 45 degrees to position them properly. Toggle the **Graphic Manipulator** on , select the first arc.
- 7. This time we will use the **Graphic Manipulator's** rotate feature. Select the blue Z Axis arc, enter -45 and then press <Enter>.



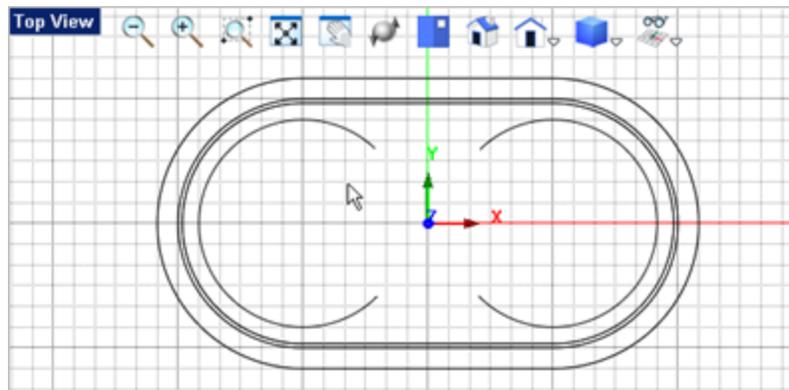
The arc is rotated negative 45 degrees about the Z axis as shown.



- 8. Now select the second arc, pick the Z axis arc, enter 45 and press <Enter>.



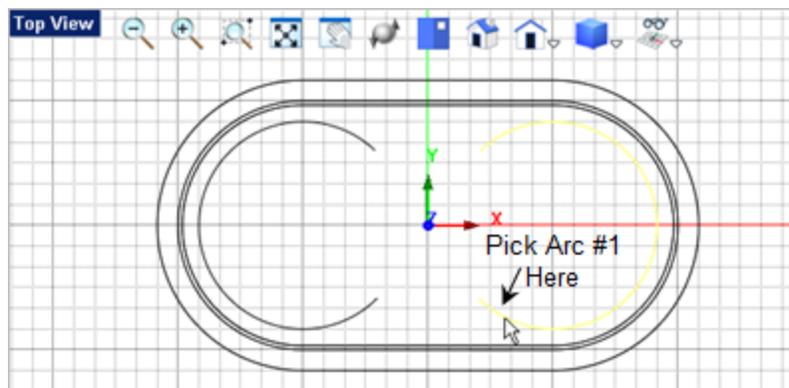
The second arc is rotate 45 degrees about the Z axis as shown.



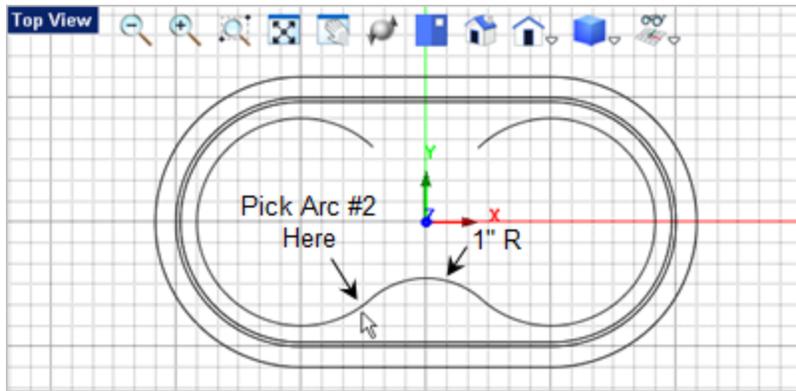
9. To complete the second pocket profile we will fillet the two arcs. From the [Curve](#)

Modeling tab [Curve Modeling](#) select the [Fillet](#)  command.

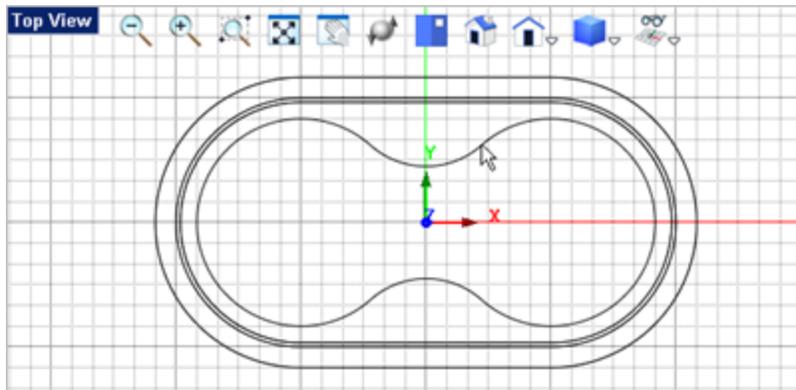
10. Enter 1.0 for the fillet radius and press <Enter>.
 11. Now pick the first arc near the end shown.



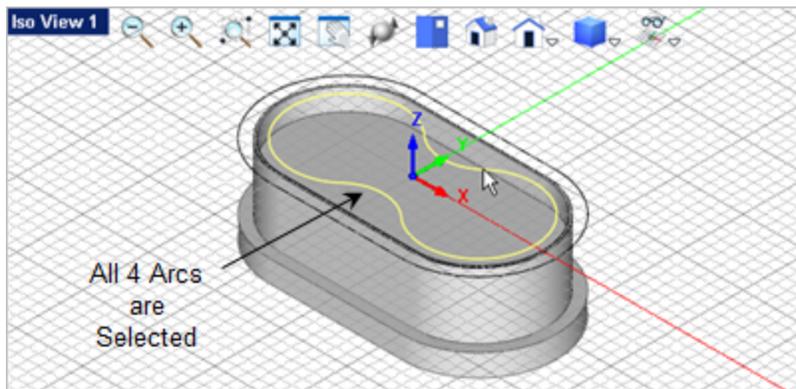
And then pick the second arc near the end shown to create the 1" fillet between the two arcs.



- 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.



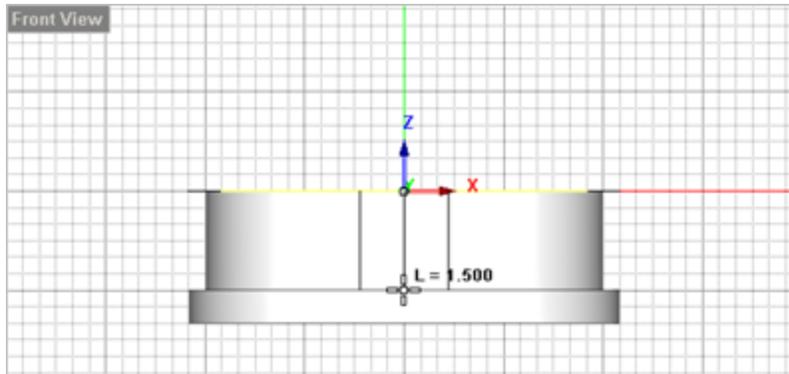


15. Now from the **Curve Modeling** tab **Curve Modeling** select the **Merge Curves** command and then right-click or press <Enter> to merge them into one curve.
16. With the curve still selected go the **Mesh Modeling** tab **Mesh Modeling** and select

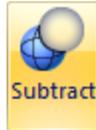


the **Extrude Mesh** command one again.

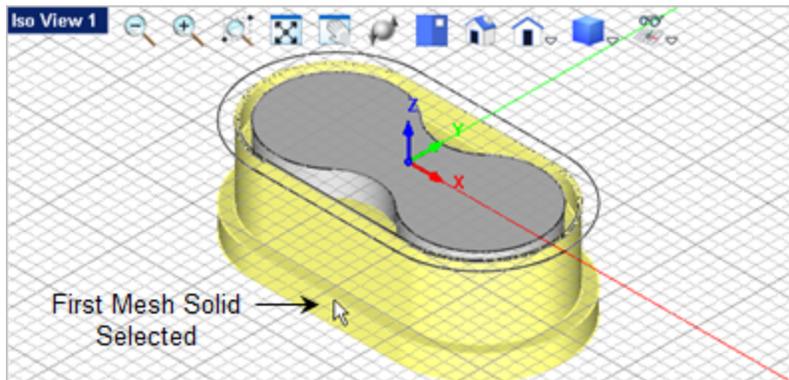
17. Drag the cursor down to the **Front View** **Front View** and select the grid point that is 1.50" below the selected curve to create the second pocket mesh solid.



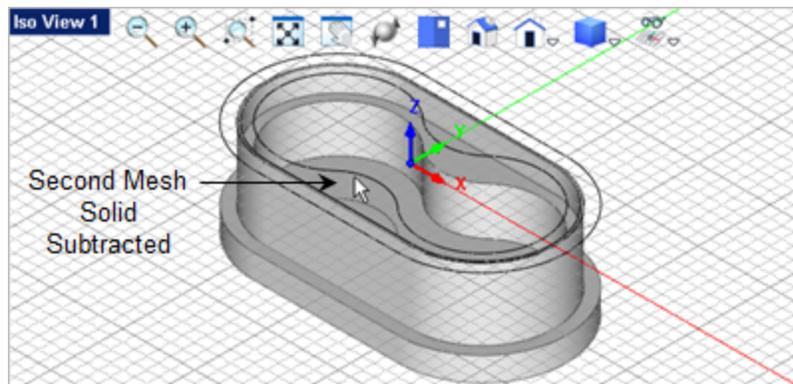
18. Now once again from the **Mesh Modeling** tab **Mesh Modeling** select the **Subtract**



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.

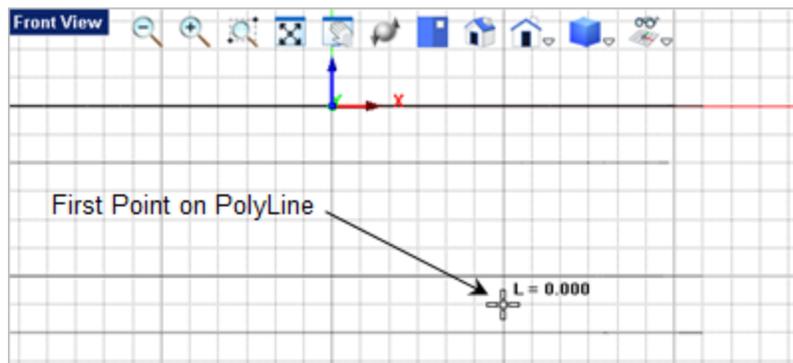


9.5 Revolve the Center Bosses

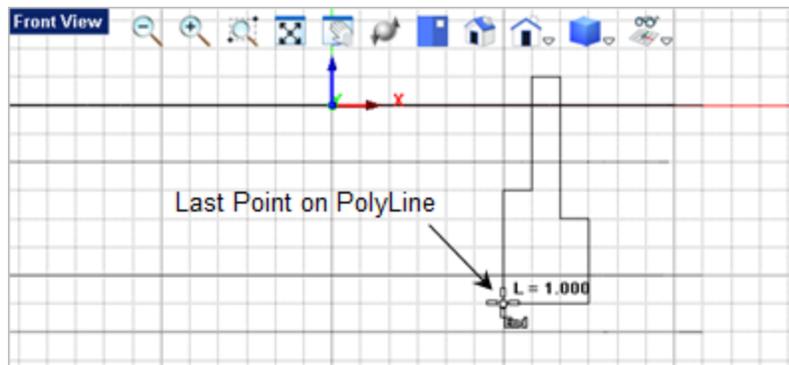
Revolve the Center Bosses

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**, **Front View** select **Wireframe Display** from the **Display**  menu on the **View Toolbar**.
2. From the **Curve Modeling** tab, **Curve Modeling** 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.

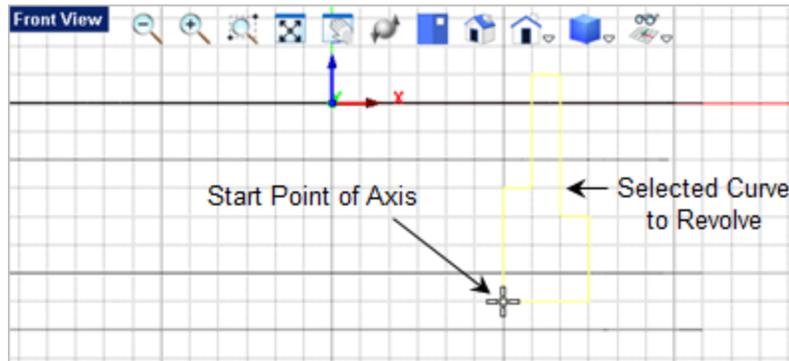


5. Now select the polyline, and then go to the **Mesh Modeling** tab **Mesh Modeling** and

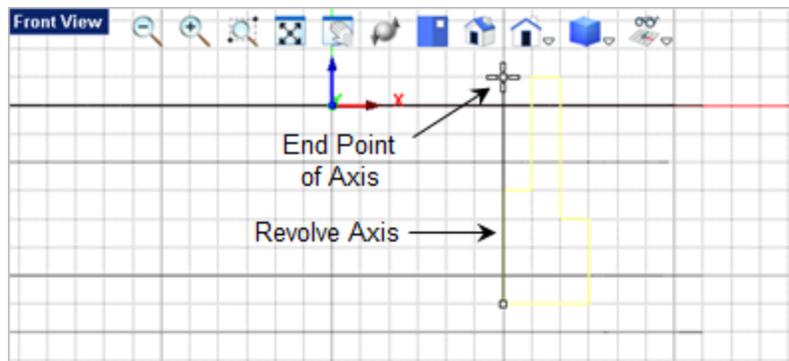


select the **Revolve Mesh** command. You can toggle off the **Graphic Manipulator** as we do not need it here.

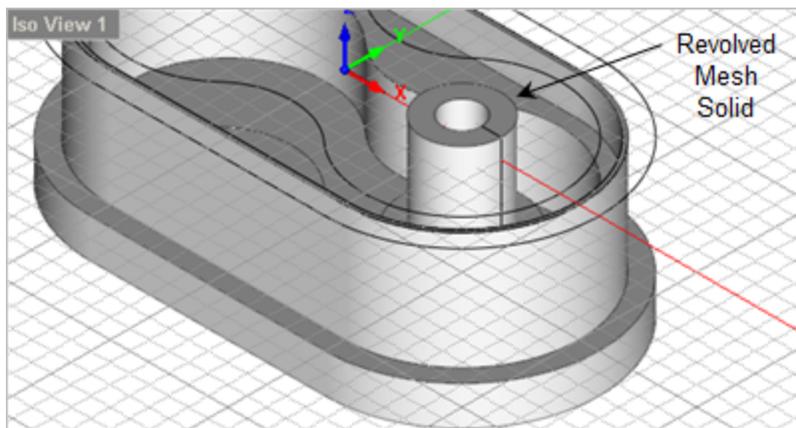
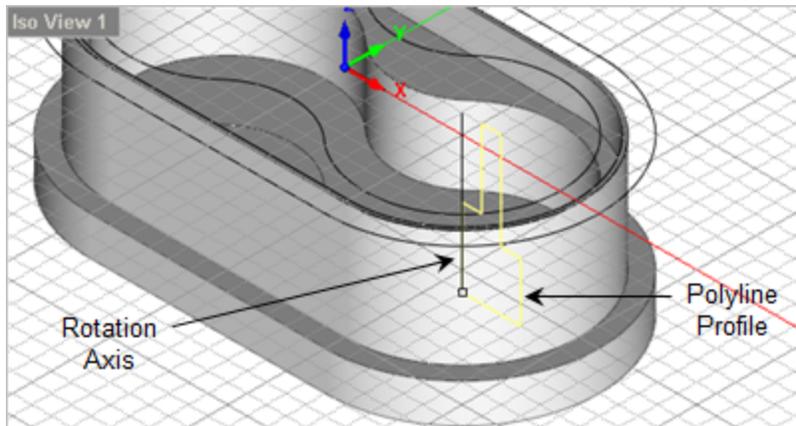
6. The command prompt **Command** says to pick the start point of the axis of rotation.



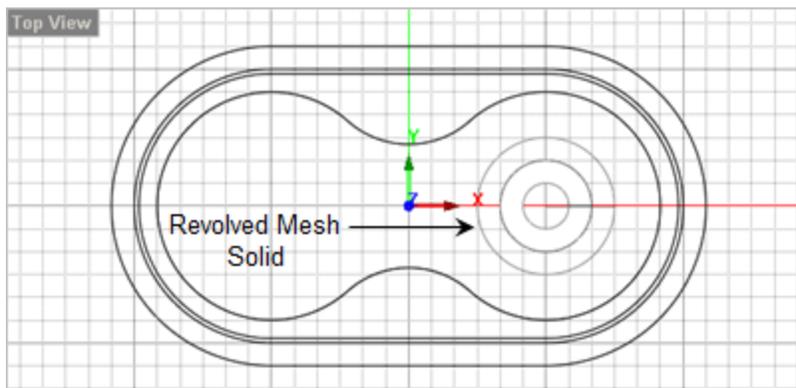
7. Now pick the second point on the axis of rotation as shown.



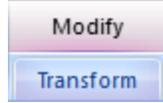
- 8. The revolved mesh solid is created and can be best viewed from [Iso View 1](#)
Iso View 1



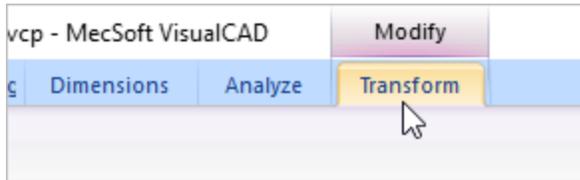
- 9. Now go to the [Front View](#) **Front View** and select the boss mesh solid we just created.



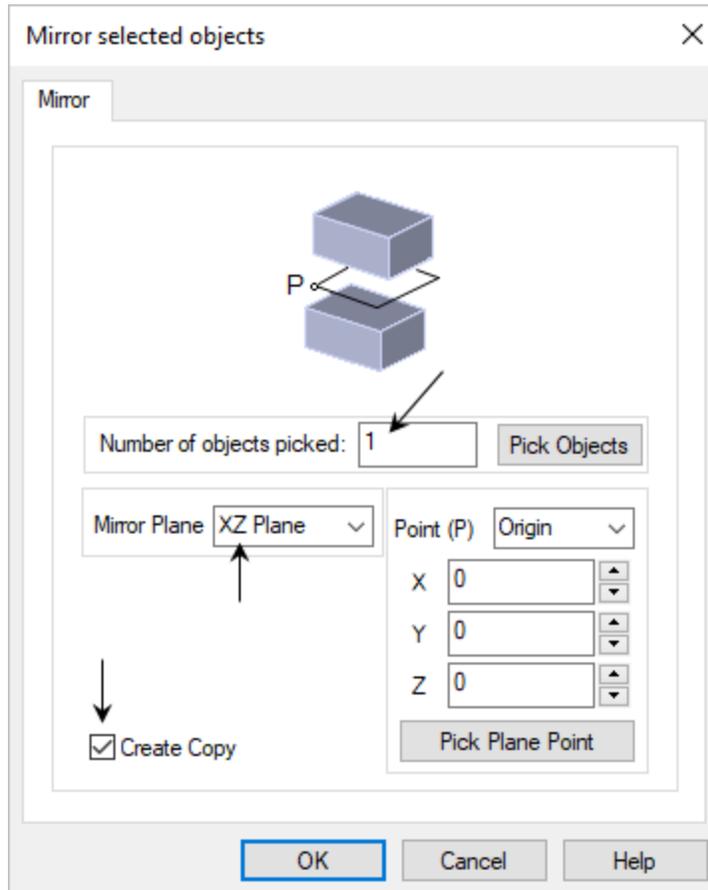
10. With the boss mesh solid selected you will see the **Modify / Transform** tab



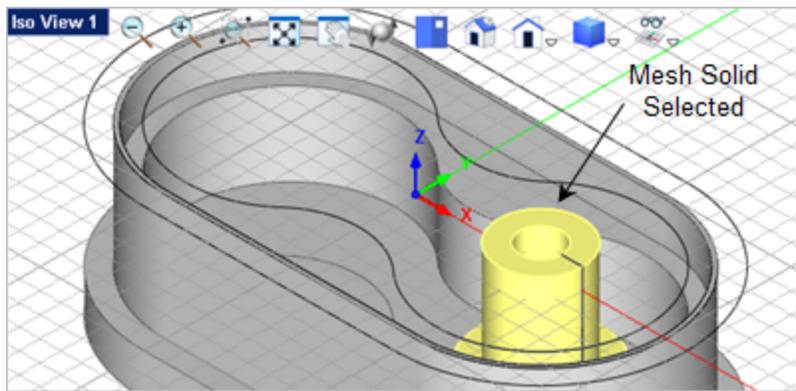
appear on the right end of the **Ribbon Bar**. It ONLY appears when an entity is selected.



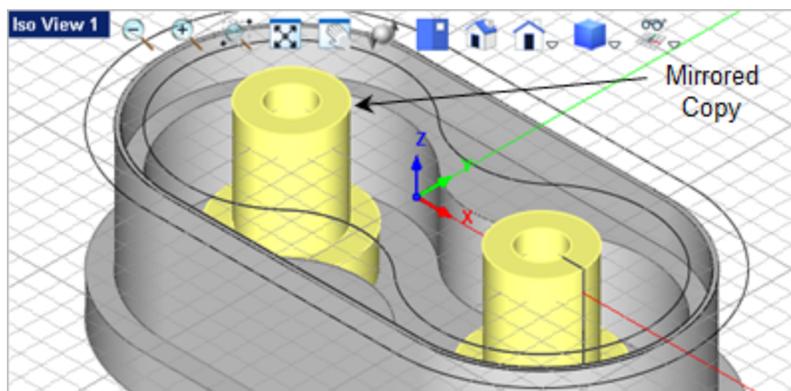
11. From the **Transform** tab, select the **Mirror**  **Mirror ...** command to display the **Mirror** selected objects dialog.
12. In this dialog, set the **Mirror Plane** to **YZ Plane**, check the box next to **Create Copy**, verify that the **Number of objects** picked value is **1** and then pick **OK**.



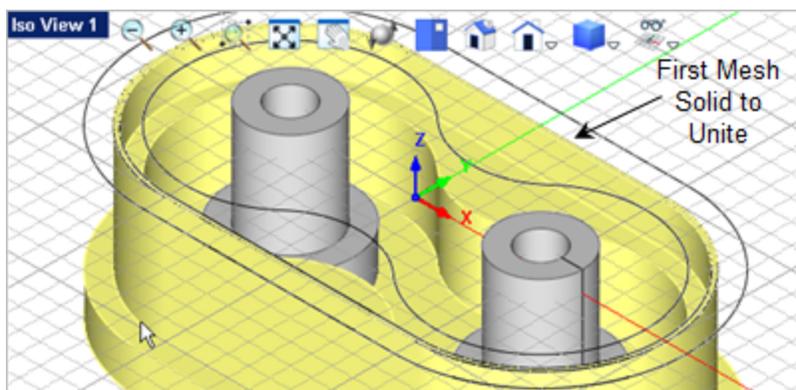
The mesh solid you selected:



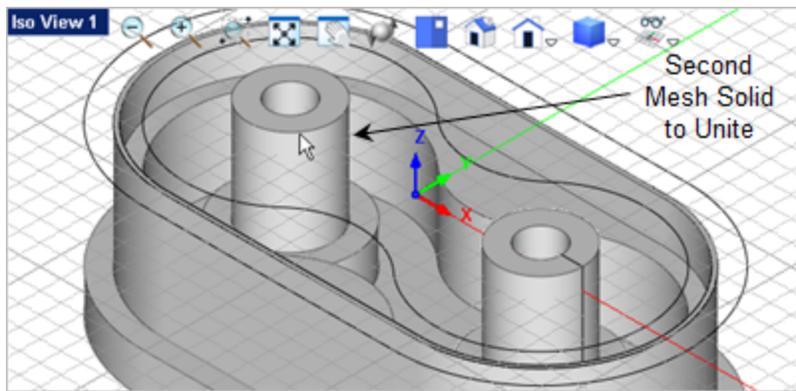
Has now been mirrored about the **YZ plane** creating a second copy as shown.



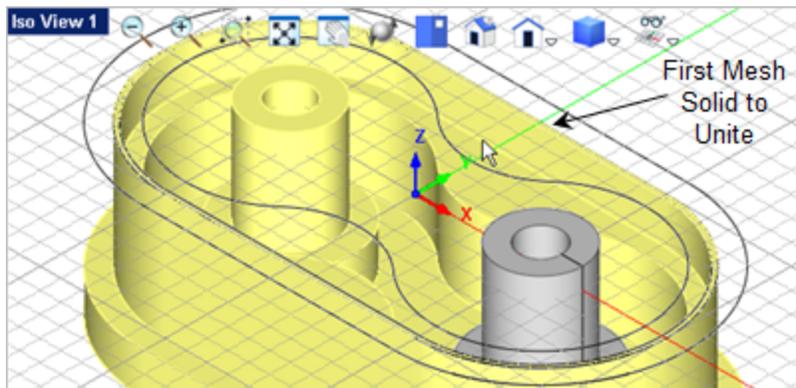
- 13. Now lets unite the two bosses with the body. From the **Mesh Modeling** tab,



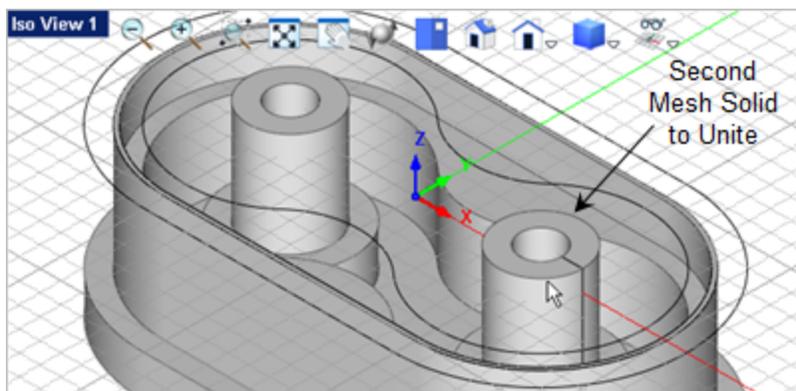
- 14. Now select the second solid to unite (i.,e. the one of the two bosses we just created.)



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.



9.6 Extrude the Connection Bar

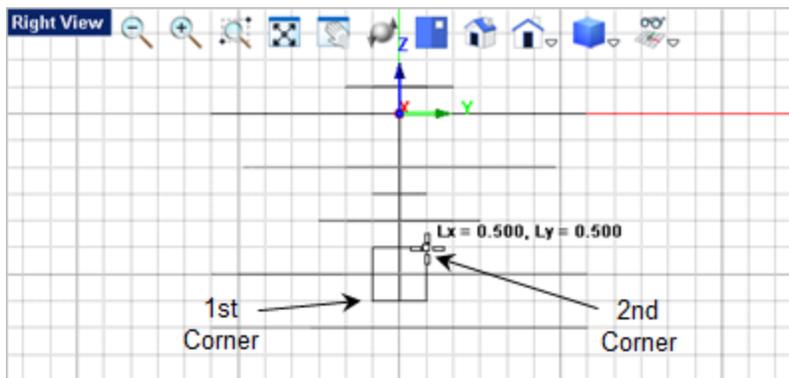
 [Extrude the Connection Bar](#)

In this step we will create the square bar that connects the two bosses at the base of the second pocket.

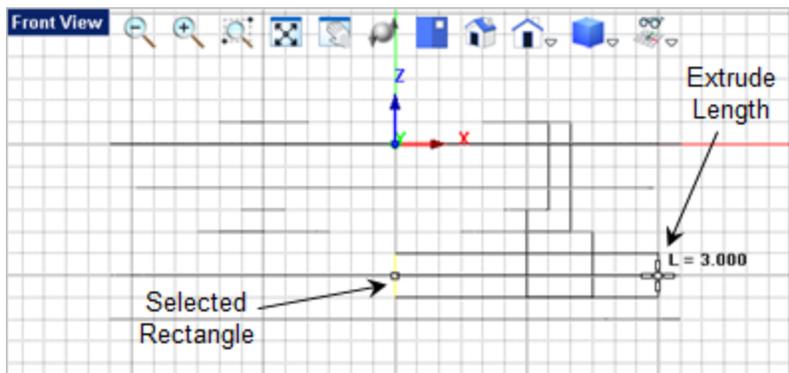
1. First activate the **Right View** and then from the **Display** menu on the **View Toolbar**, select **Wireframe Display**.

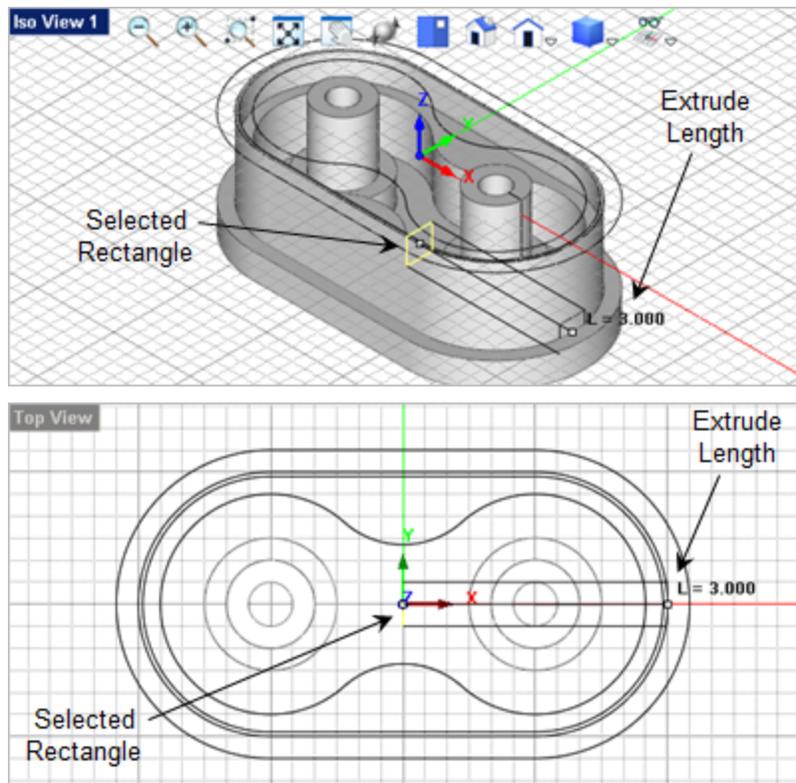


2. From the **Curve Modeling** tab, select the **Rectangle** command.
3. Select the grid points for the 1st and 2nd corners of the rectangle as shown.

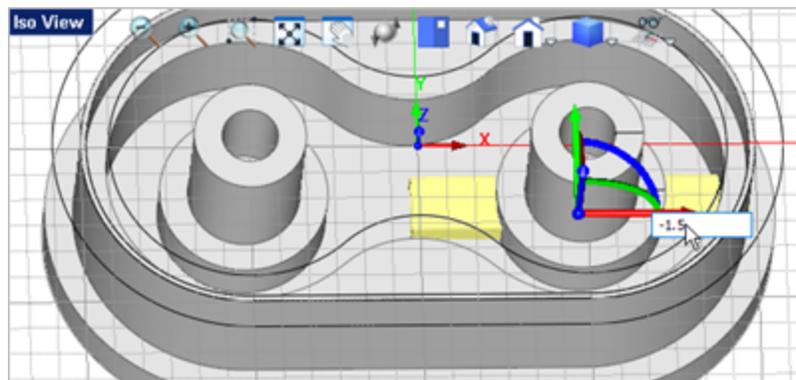


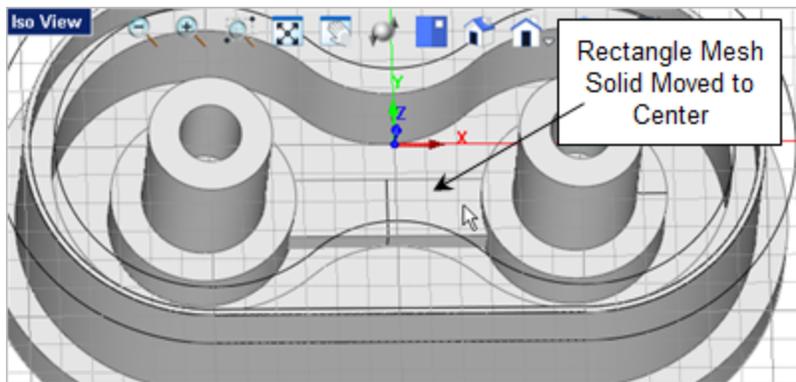
4. Now from the **Mesh Modeling** tab, select the **Extrude Mesh** command.
5. Pick the rectangle we just created and drag the cursor over to the **Front View** and pick the **Grid point** located 3.00" to the right as shown below.





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](#) **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.

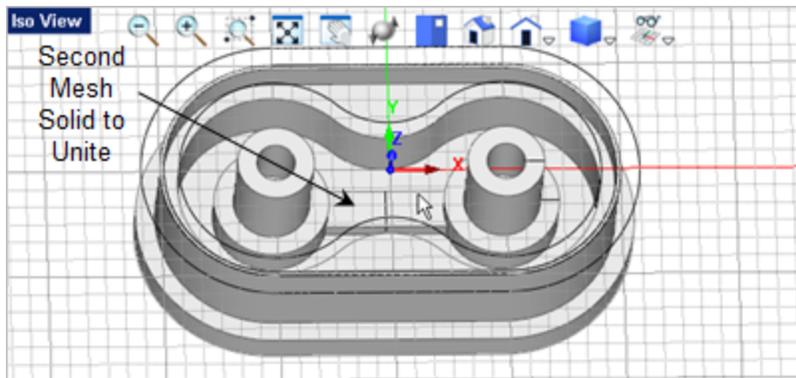
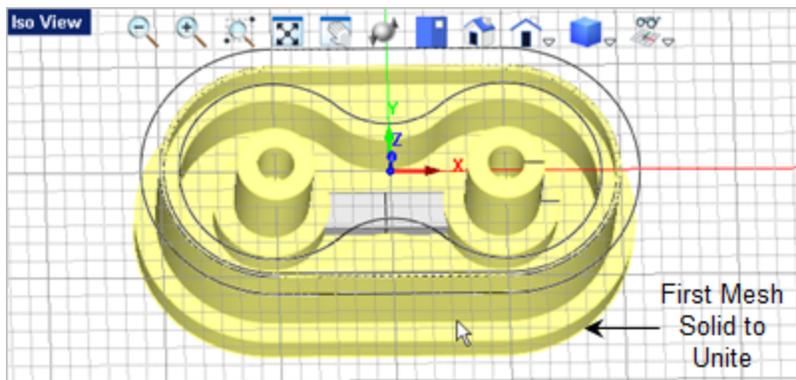




- 9. Now again we will unite the Connector Bar with the Body. From the [Mesh Modeling](#)



tab, [Mesh Modeling](#) select the [Unite Mesh](#) command and select the body for the first mesh solid and the then select the Connector Bar to unite them.

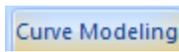


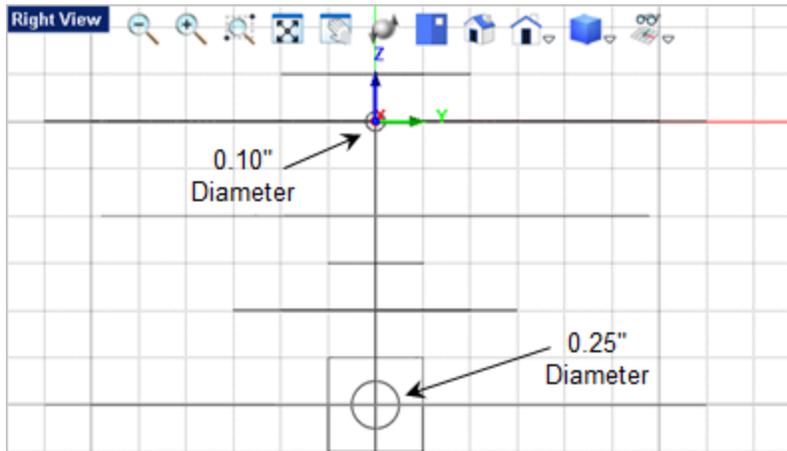
9.7 Extrude the Connection Wall

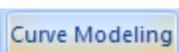
Extrude the Connection Wall

In this step we will create the wall that pass between the two center bosses.

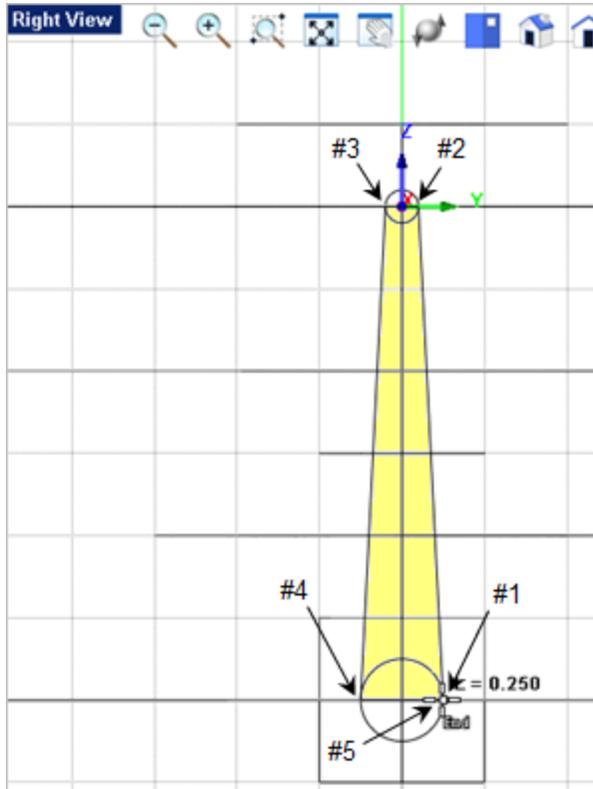
1. Again, activate the **Right View**  and then from the **Display**  menu on the **View Toolbar**, select **Wireframe Display**.

2. From the **Curve Modeling** tab, **Curve Modeling**  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 press **<Enter>**. We will use these two circles to locate the corners of a polyline that defines the cross-section of the connection wall.

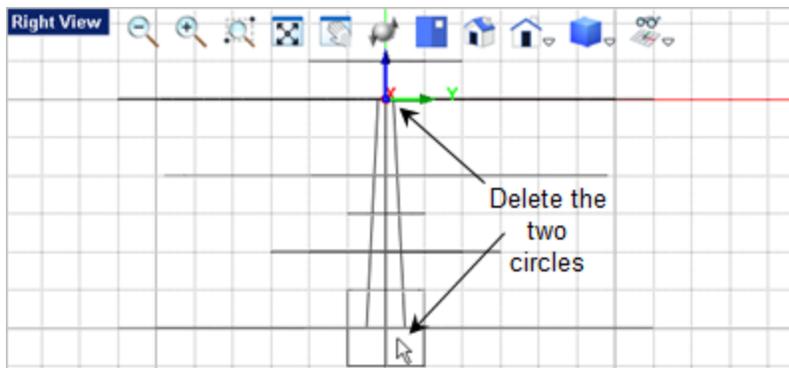


6. Now from the **Curve Modeling** tab, **Curve Modeling**  select the **Polyline**  command.
7. From the **Status Bar** toggle on the **Quad Point Snap**  and toggle off all of the other object snaps.
8. We will be drawing a 4-sided closed polyline that encompasses the yellow area shown in the image 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.

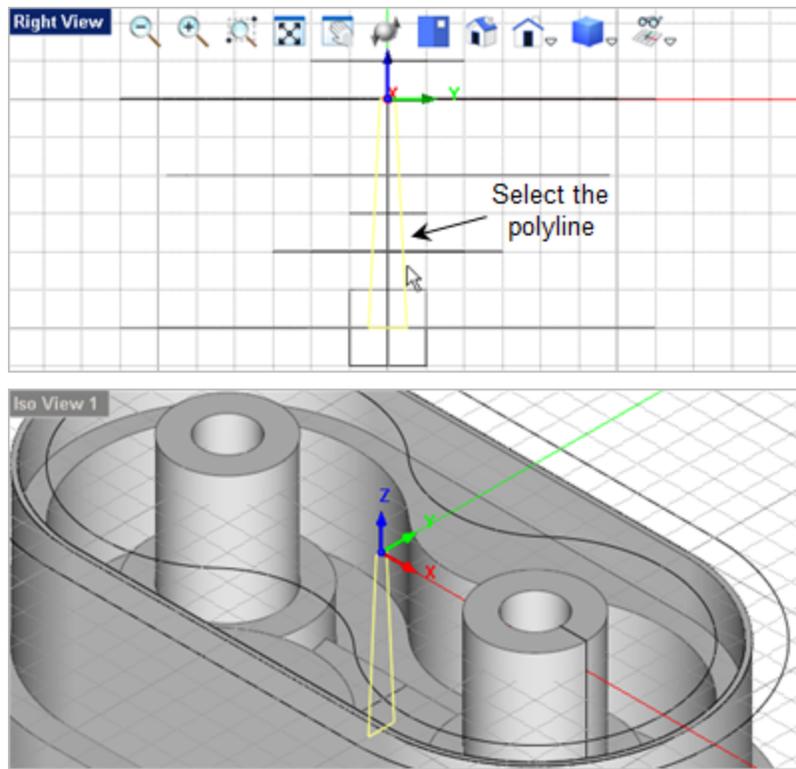
- To complete the polyline select the **Quad** point at the #1 location where the polyline started. The **Polyline**  command will end automatically when the end point coincides with the start point.



- Now you can delete the two circles. Just select them and press the **** key.



- Now we will extrude the 4-sided polyline to create the Connecting Wall. First select the polyline we just created.



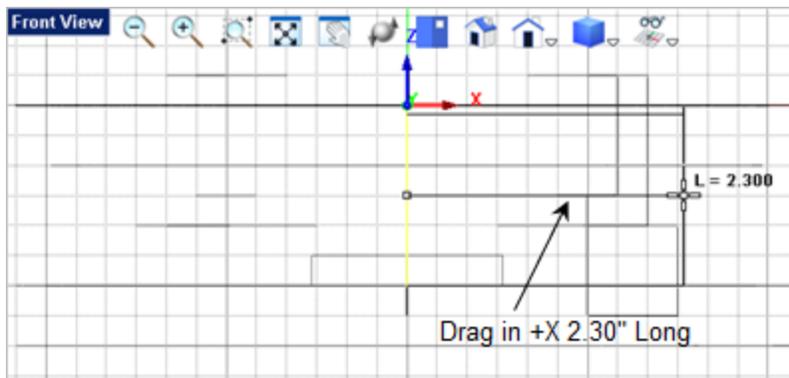
15. Now go to the **Mesh Modeling** tab and select the **Extrude Mesh**



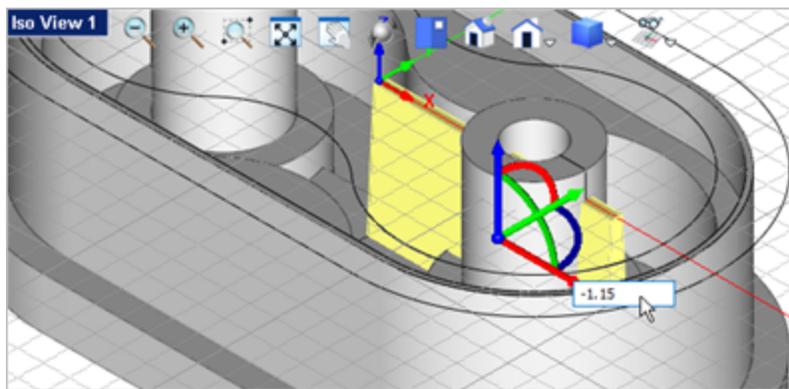
command again.

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.



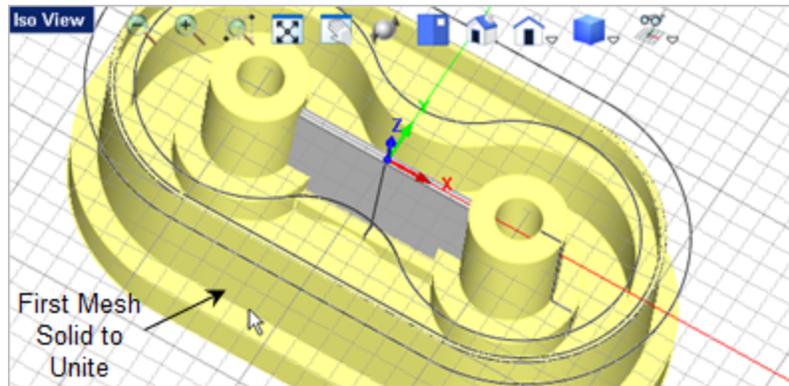
18. Now from **Iso View 1** select the Connection Wall to display the **Graphic Manipulator**. If you do not see it, make sure it is toggled on from the **Status Bar**



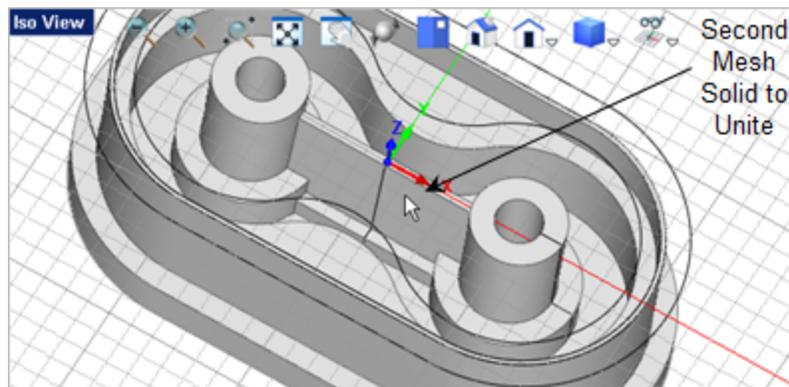
19. Pick the red **X Axis** arrow and type in **-1.15** and press **<Enter>**. The Connection Wall should move into place at the center spanning between the two bosses as shown in the image below.
20. Now lets unite the Connection Wall with the body. Go to the **Mesh Modeling** tab

Mesh Modeling 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.



9.8 Extrude Ejector Pin Holes

Extrude the Ejector Pin Holes

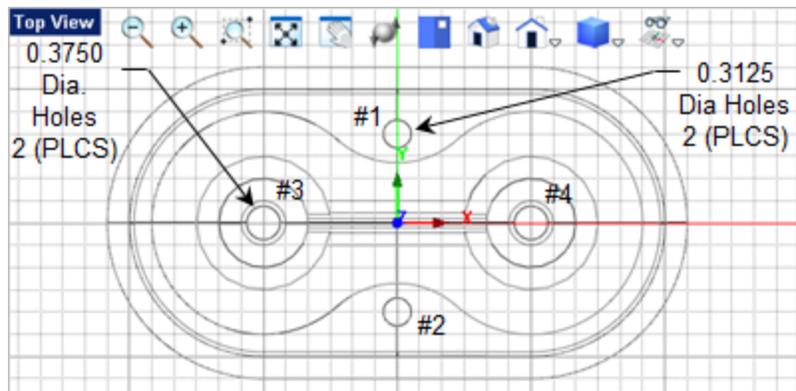
In mold design, ejector pins are used to eject the molded part from the cavity side of the mold. As the mold press opens, the ejector pins slide and protrude through the cavity forcing the plastic part out of the mold where it falls into a catch bin. The ejector pin holes pass complete through the insert.

1. Activate the **Top View**  and then go to the **Curve Modeling** tab

Curve Modeling

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

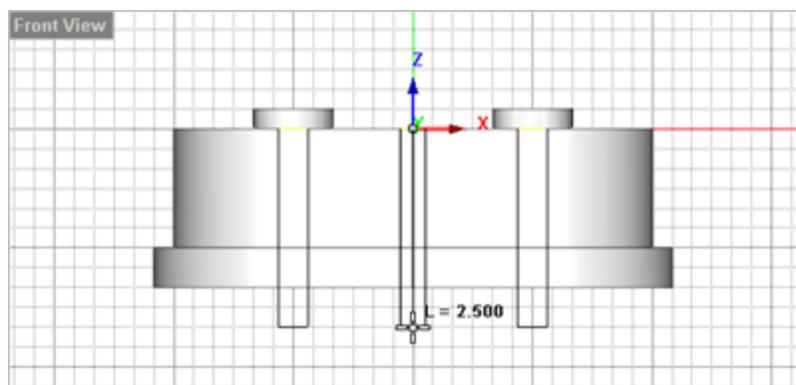
Mesh Modeling

and select the **Extrude Mesh**

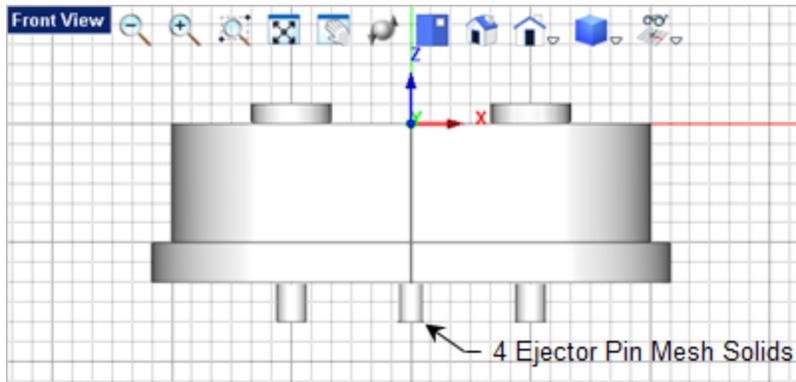


command.

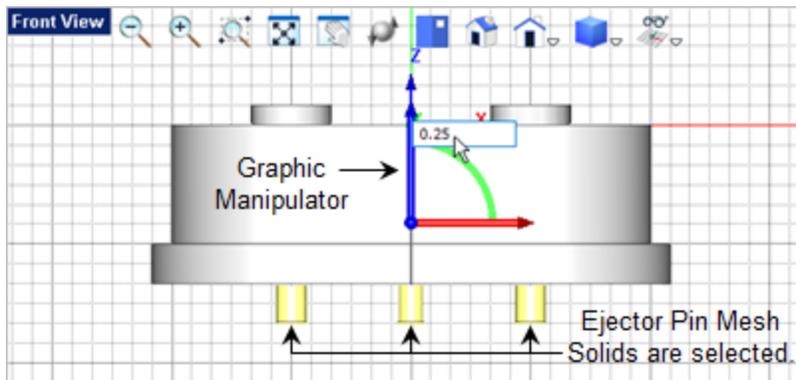
9. Now move the cursor down to the **Front View** **Front View** and drag the extrude length down past the bottom of the body and pick the grid pint located at 2.500" below the part.



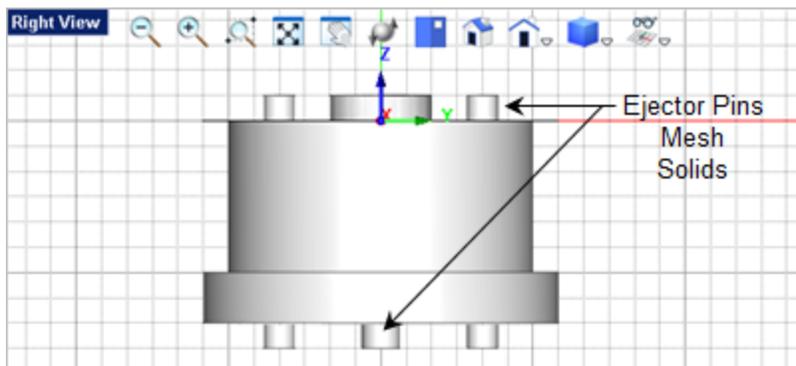
The 4 mesh solids are created:



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** **Right View** and we see the ejector pin mesh solids.

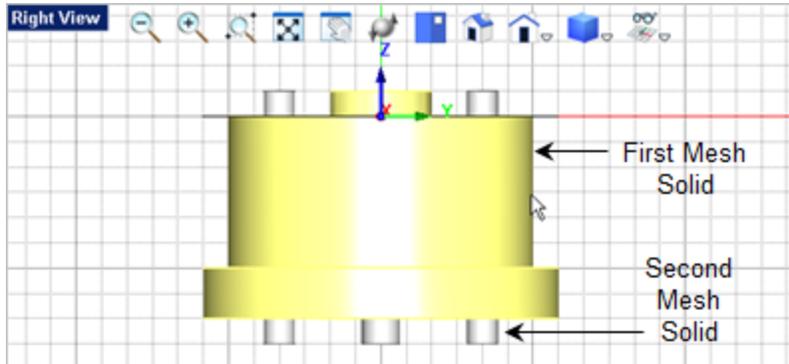


- Now to subtract the ejector pin mesh solids from the body to create the ejector pin holes go to the **Mesh Modeling** tab  and select the **Subtract Mesh**

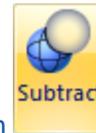


command.

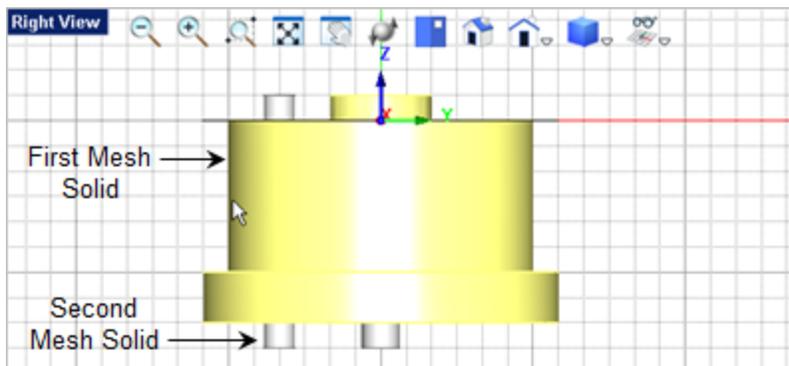
- Select the body as the first mesh solid.



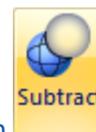
- Then select the 0.3125" ejector pin mesh solid on the right to subtract it from the body.



- Press the **<Enter>** key to repeat the **Subtract Mesh** command and select the body again as the first mesh solid.

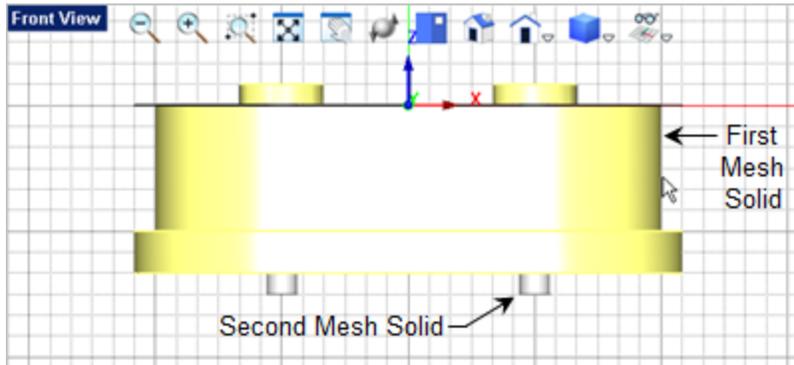


- Then select the 0.3125" ejector pin mesh solid on the left to subtract it from the body.



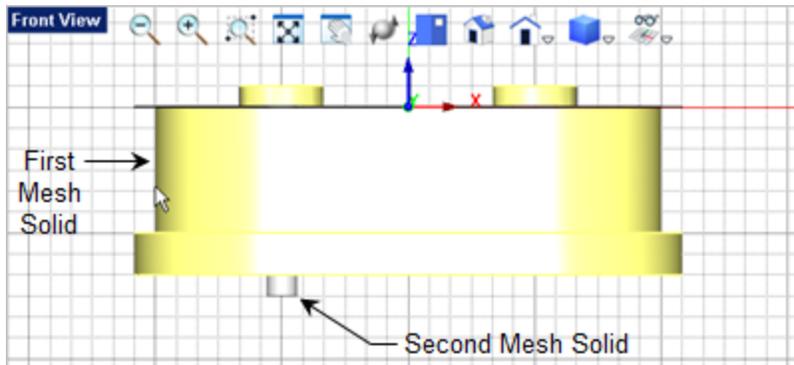
- Press the **<Enter>** key to repeat the **Subtract Mesh** command and this time to subtract the 0.3750" ejector pins move to the **Front View** .

19. Select the body again as the first mesh solid.



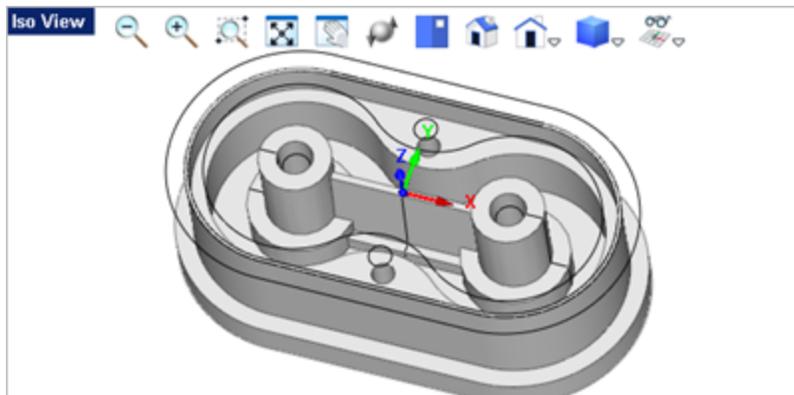
20. This time select the 0.3750" ejector pin mesh solid on the right to subtract it from the body.

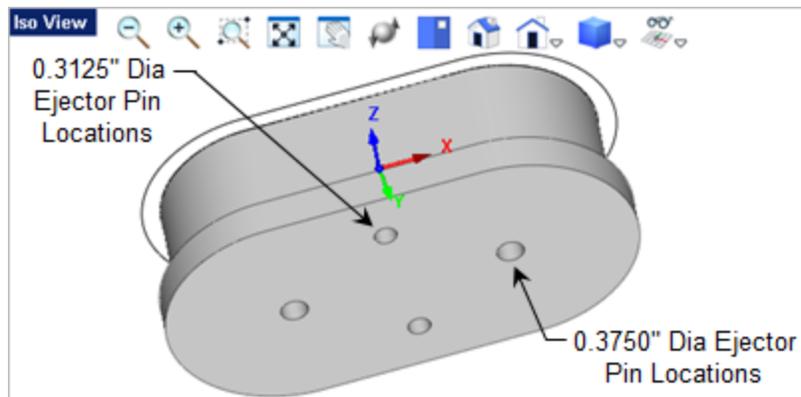
21. Press the <Enter> key again to repeat the command and then select the body mesh solid.



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:



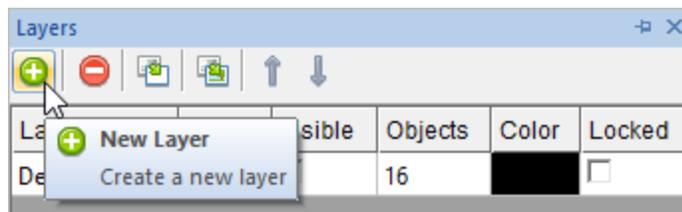


9.9 Change Geometry Layer

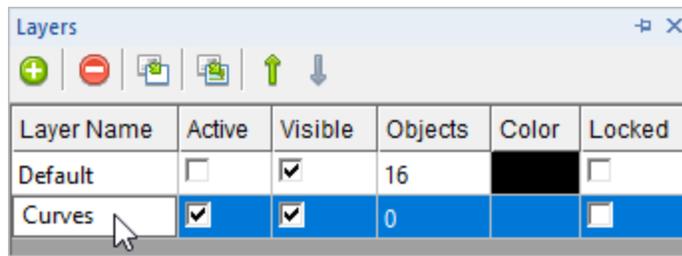
Change Geometry Layer

We have a lot of curves displaying on the part. Let's clean them up.

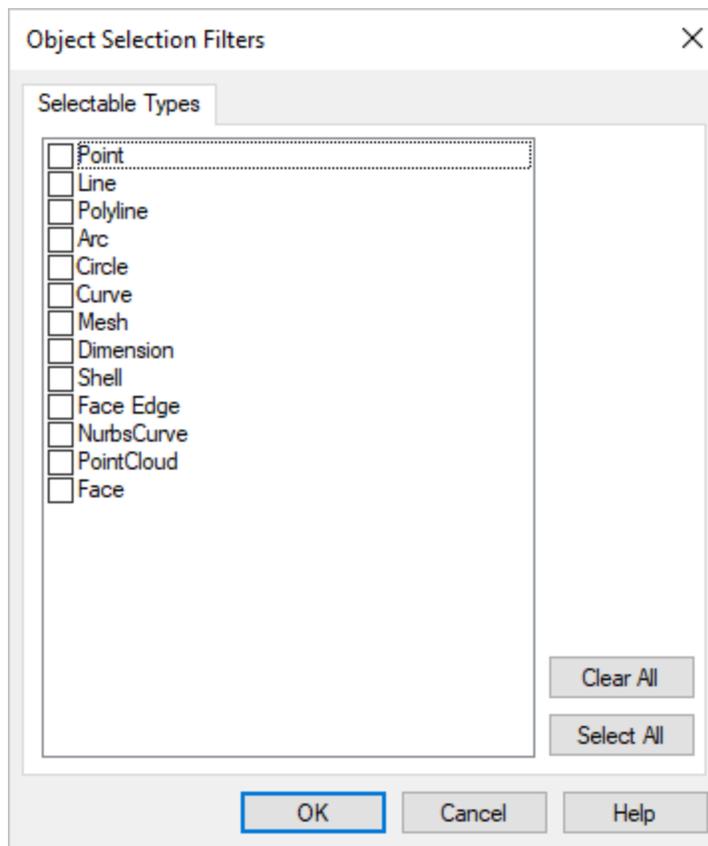
1. Display the **Layer Manager**. You can pick the **Layer Manager** icon  located on the **Status Bar** or select the **Layers** fly-out tab located on the left side of the drawing window. If you recall we unpinned the **Layer Manager** allowing it to collapse to the side of the screen.
2. From the **Layer Manager** select the **Add Layer** icon  to add a new layer.



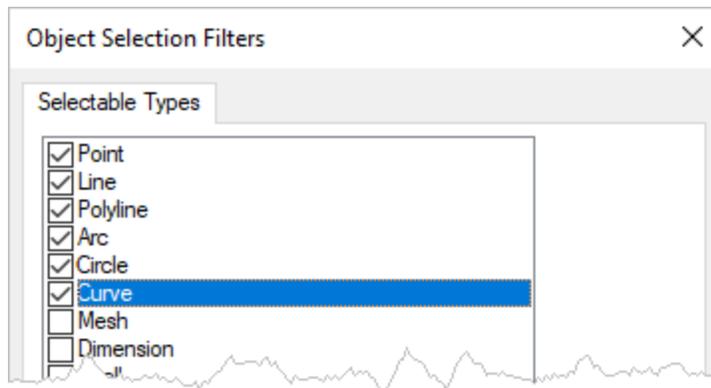
3. Now let's rename the new layer to **Curves**. First double-left-click in the Name field to activate it, and enter **Curves** as the new layer name.
4. Next let's make the **Default** layer the **Active Layer** by checking the box in the **Active** column for the layer **Default**. The Default layer 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.



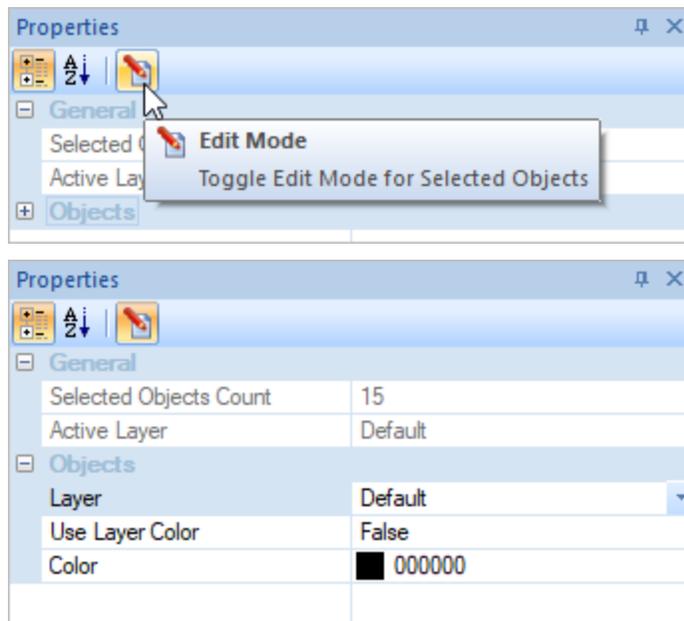
5. Now VisualCAD has a quick way of selecting entities by type. Go to the **Modeling Aids** [Modeling Aids](#) tab and select the **By Type**  icon. This command displays the **Object Selection Filters** dialog that allows you to check mark the entities that you want to select. That dialog is shown below:



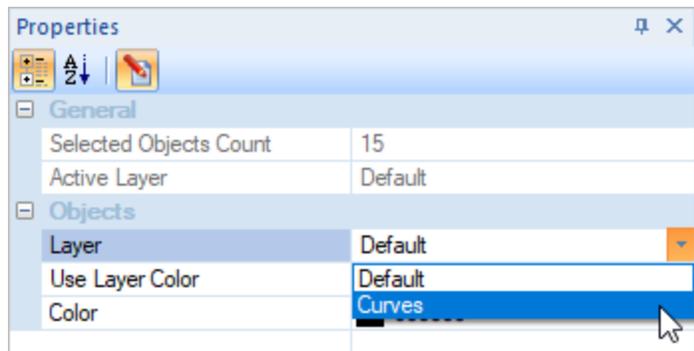
6. Place a check mark in the box for each of the following entity types: **Point**, **Line**, **Polyline**, **Arc**, **Circle**, and **Curve**.



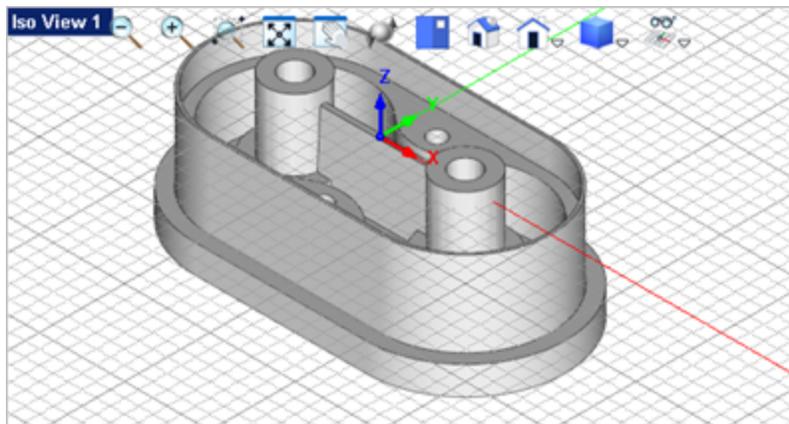
7. Now pick the **OK** button. You will see that all of the reference curves we created in this exercise are highlighted for selection.
8. With the entities selected, select the **Properties**  icon from the **Status Bar**. Its located just to the left of the **Layer Manager**  icon.
9. The **Properties 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.



- 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.



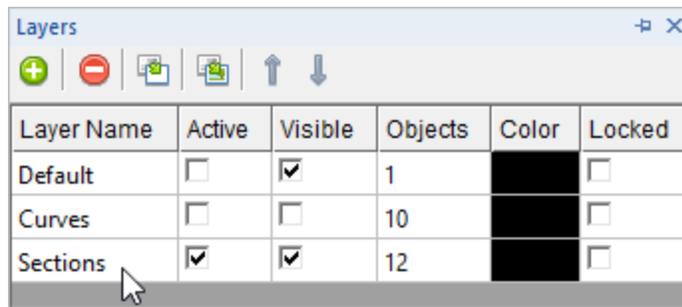
9.10 Create Section Curves

Create Section Curves

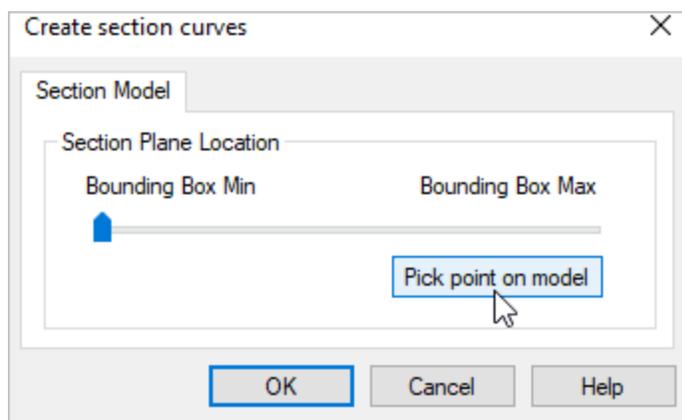
Congratulations, our part is now nearly complete! However, we do want to create cross-section curves that will aid you when it comes time to create a detail drawing of the part.

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**.

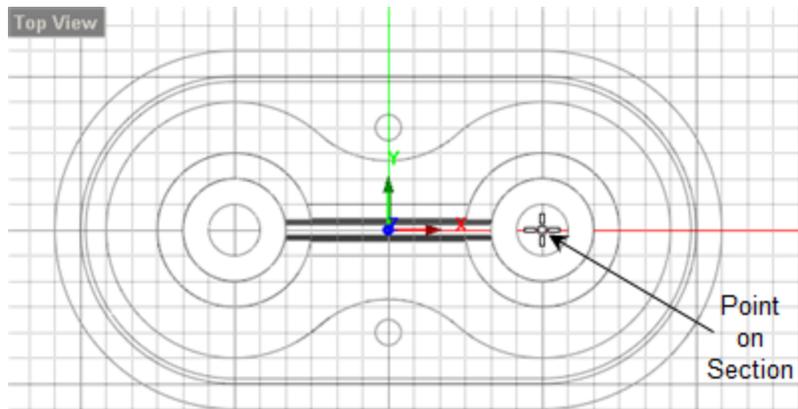
- First lets again create a new layer and make it the active layer so that the section curves will be assigned to that layer automatically when they are created.
- Display the **Layer Manager**  one again.
- Create a new layer and name it **Sections** and also make it the active layer. The **Layer Manager** should look like this:



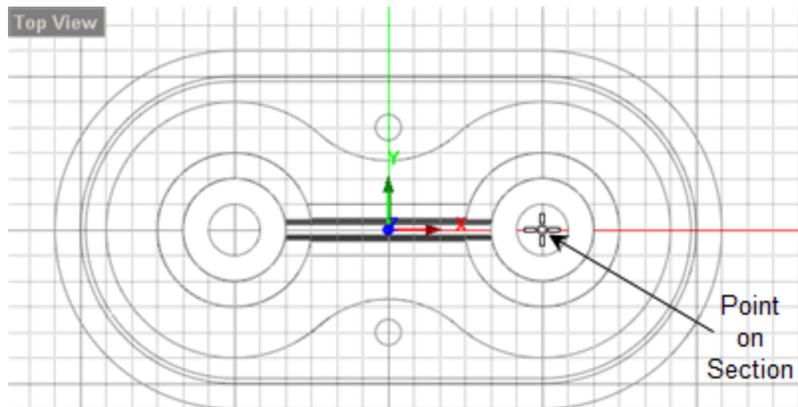
4. Now go to the **Curve Modeling** tab **Curve Modeling** and select the **Section Curve**  command. The Create section curves dialog will display.



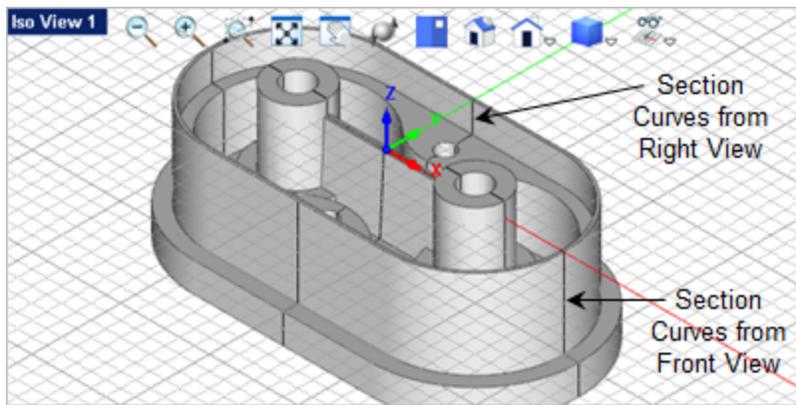
5. First select anywhere in the **Front View** **Front View** to activate it.
6. Now if you move the slider in the dialog you will see the section curves move along the part.
7. We want the location of the section to be precise so lets pick the Button that says "Pick point on model".
8. Now from the **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**.
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.

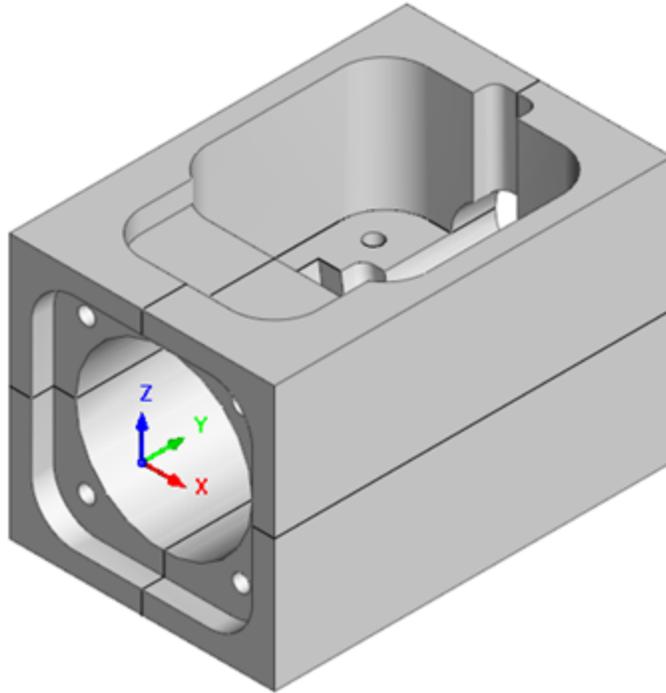


Congratulations!! You modeled your Very first mold insert using [VisualCAD!](#)

14. Save the file as [Insert_Completed.vcp](#).

#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:

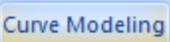
In this exercise you will perform the following [VisualCAD](#) tasks:

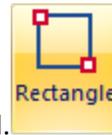
1. [Model the Main Body.](#)
2. [Model the Front Access.](#)
3. [Model the Top Access.](#)
4. [Model the Mounting Holes.](#)
5. [Modify the Top Access.](#)
6. [Create Section Curves.](#)

10.1 Model the Body

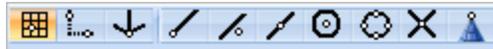


Model the Main Body

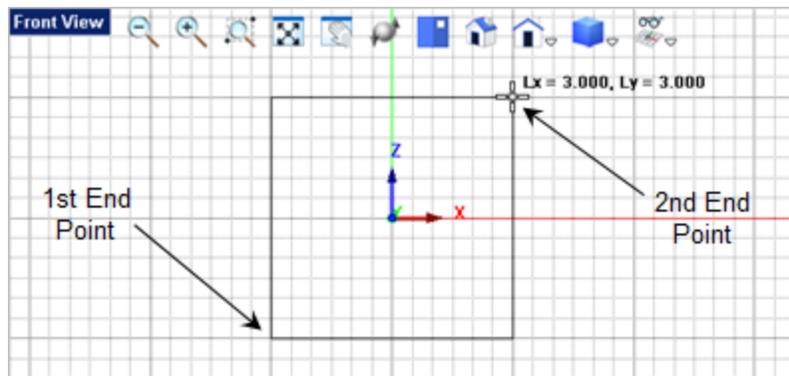
1. Select the **Curve Modeling** tab  from the top **Ribbon Bar**.



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** 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** and press **<Enter>**. This will create a rectangle 3" square.

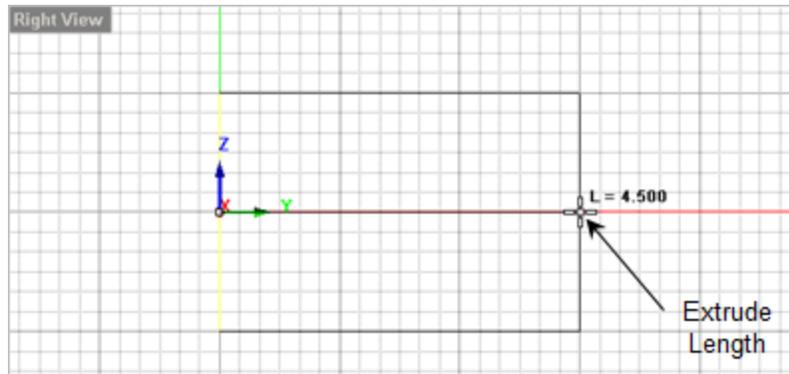


6. Now select the rectangle you just drew and then from the **Mesh Modeling** tab select

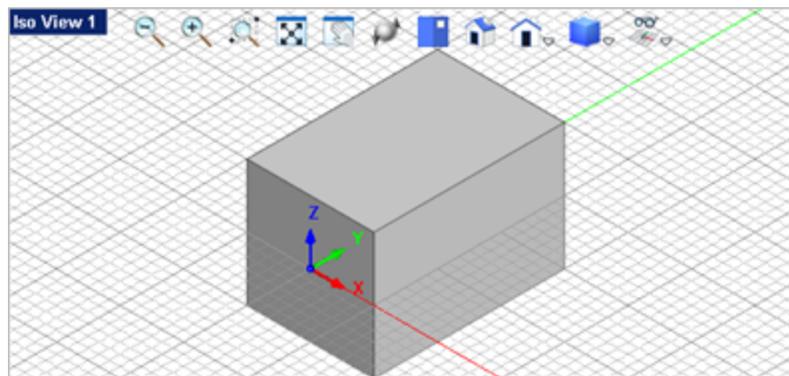


the **Extrude Mesh** command.

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 and press **<Enter>**.



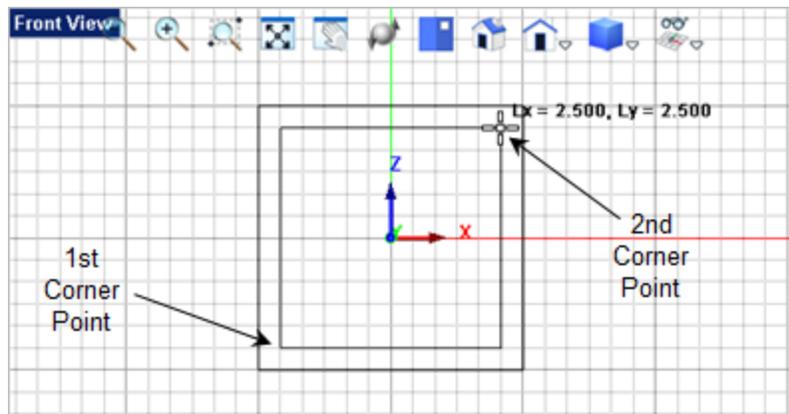
Your part should look like this in Iso View 1:



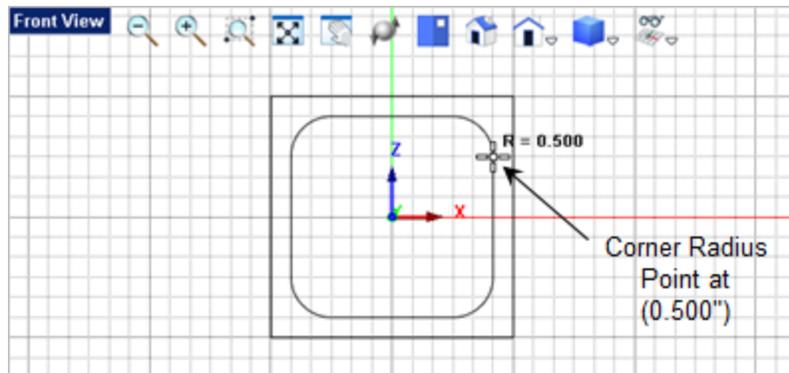
10.2 Model the Front Access

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** 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** 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>`.

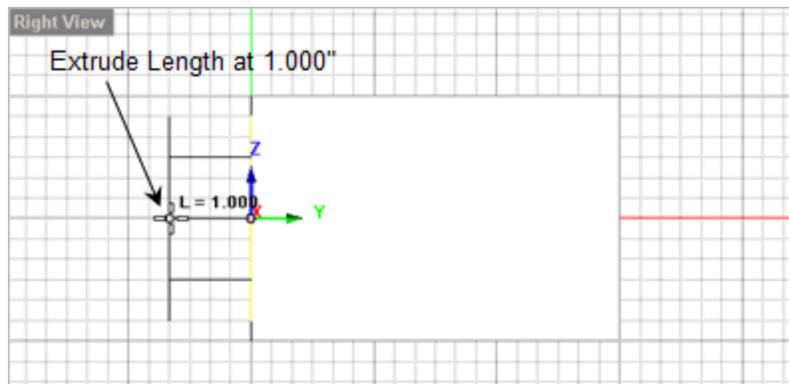


6. Now select the rounded rectangle you just created and from the **Mesh Modeling** tab,

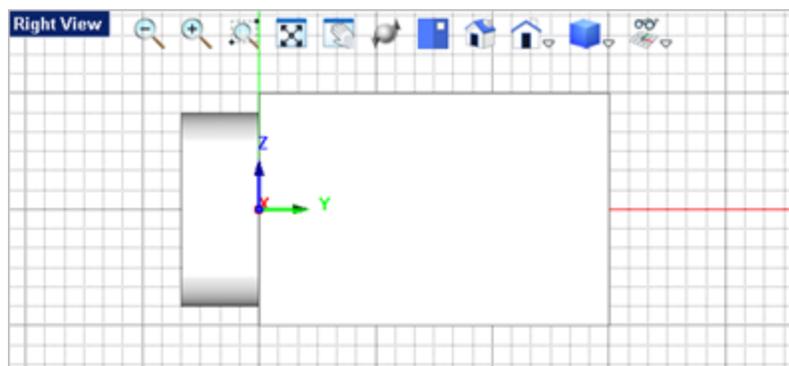


select the **Extrude Mesh** command.

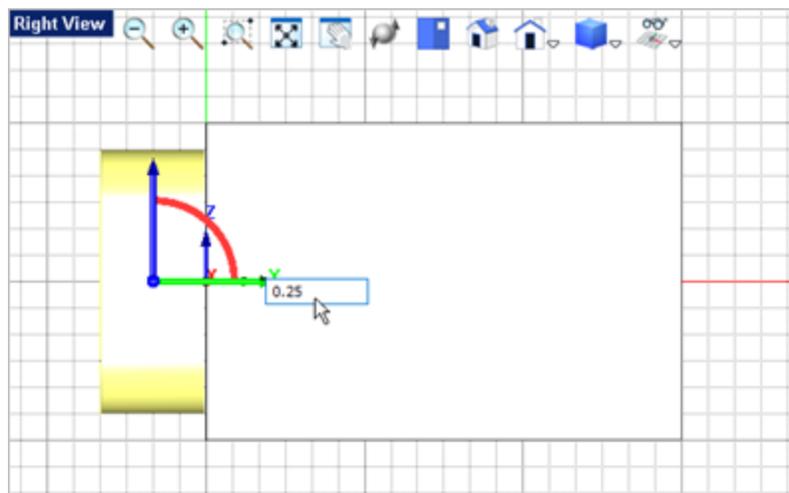
7. Move the cursor over to the **Right View** and select the **Extrude Length** at the **1.000** Grid point or enter `1` in the command window and press `<Enter>`.



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.

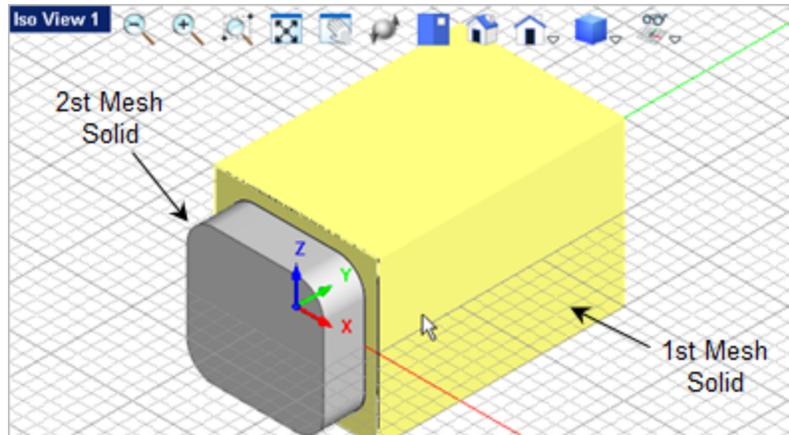


11. Press <Enter> again to accept the move.
12. Now activate the **Iso View 1** window and from the **Mesh Modeling** tab select the

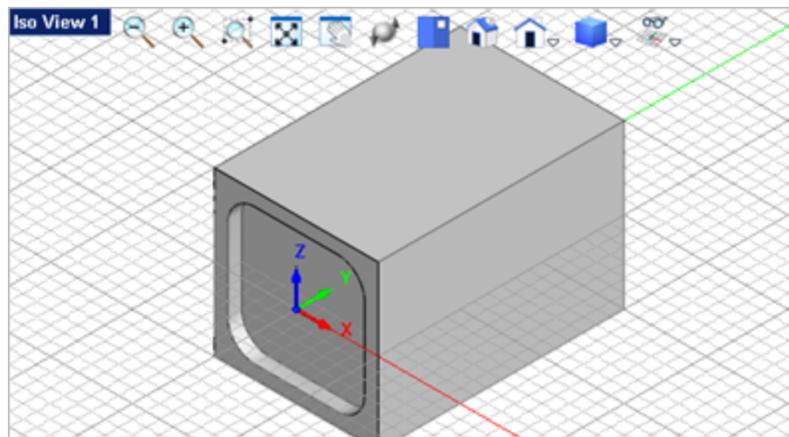


Subtract Mesh command.

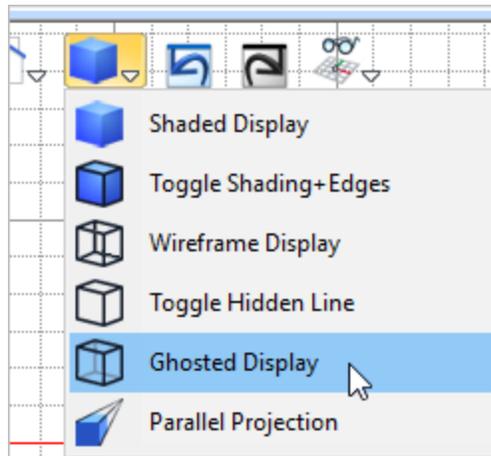
13. For the first mesh solid select the Body.



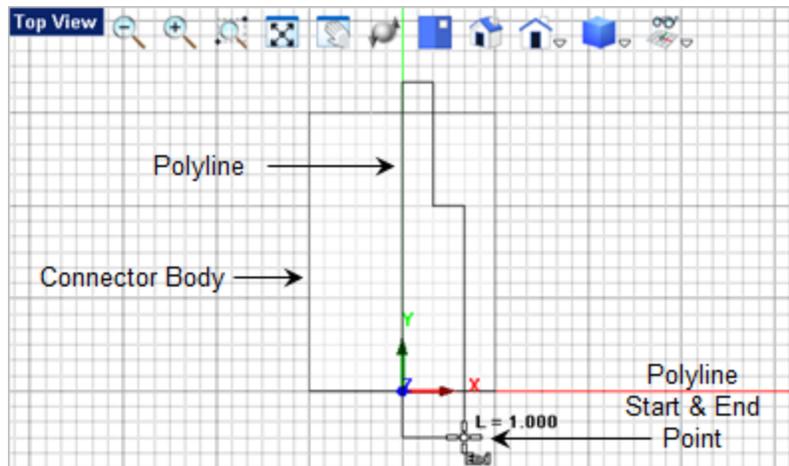
14. For the second mesh solid select the extrusion. The extrusion will be subtracted from the Body as shown below.



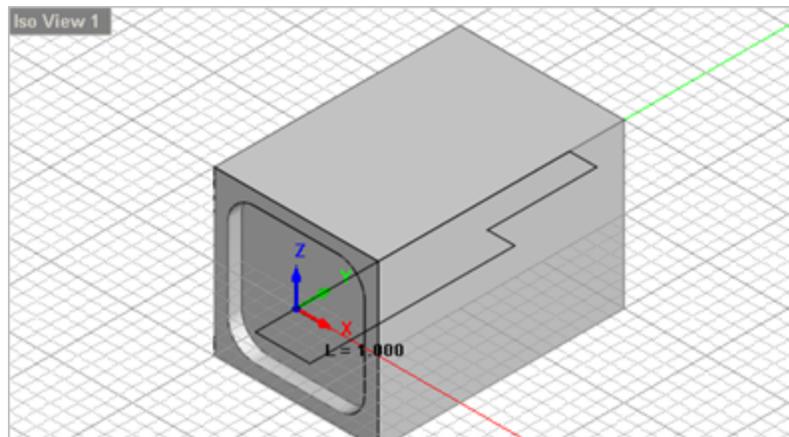
15. Now activate the **Top View**.
16. Select the **View** menu from the **View Toolbar** and then select **Ghosted Display**. Note: The **View Toolbar** will only display in the active viewport.



17. Now from the **Curve Modeling** **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.

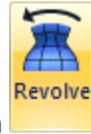


In Iso View 1 the closed curve polyline looks like this:



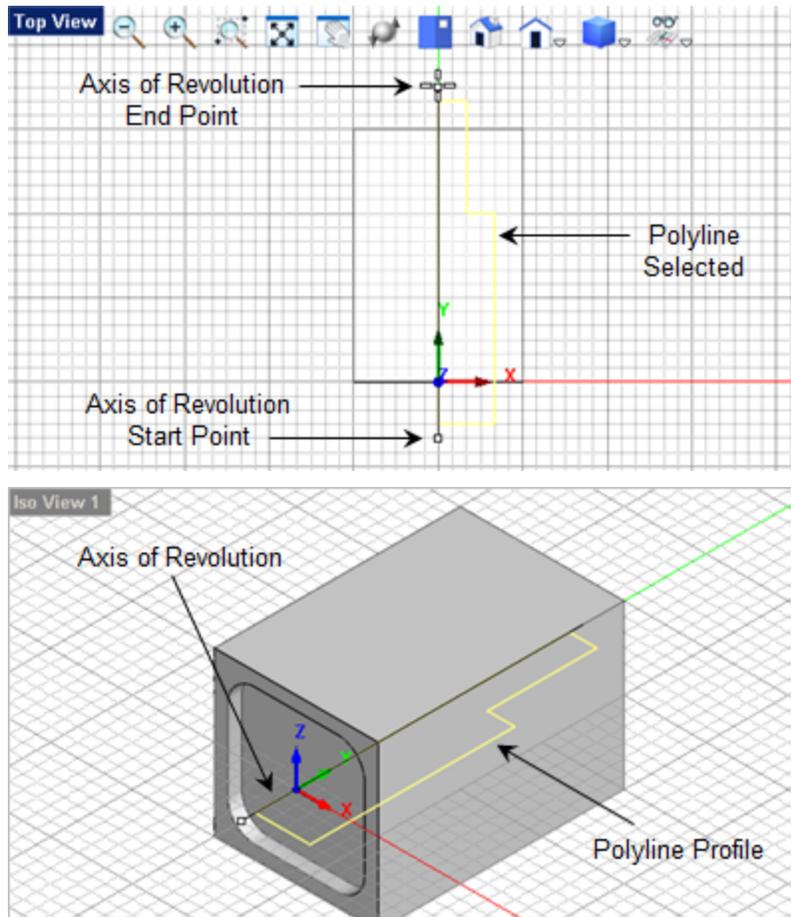
19. Now select the closed curve polyline you just created and from the

Mesh Modeling



tab select the **Revolve Mesh** command.

20. The command line `Command` 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.



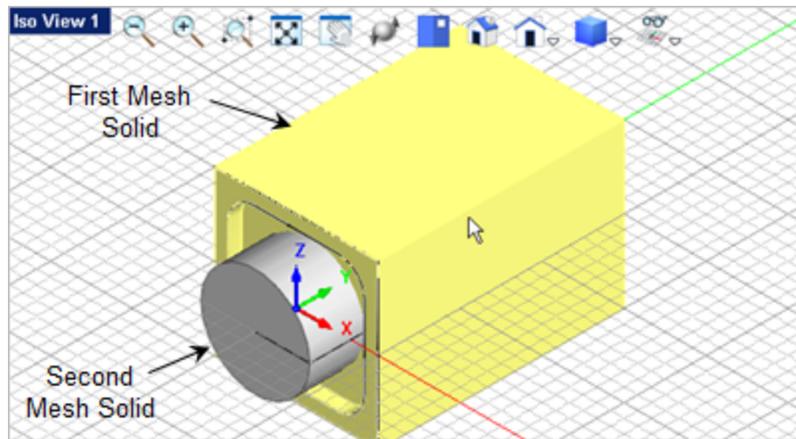
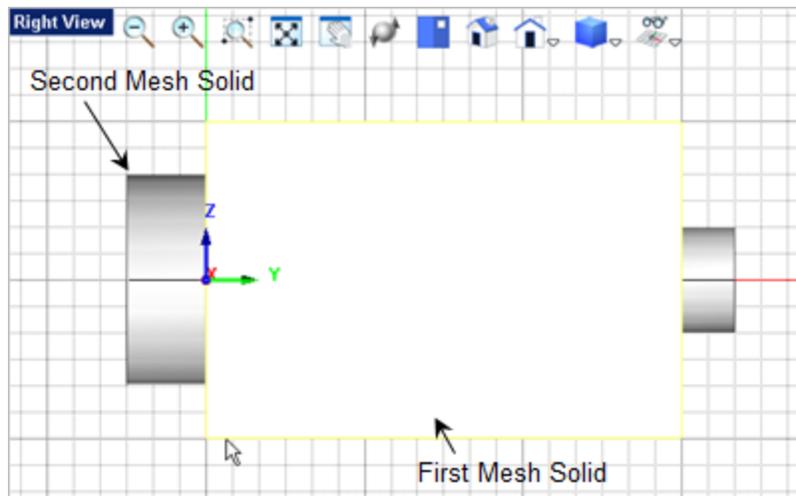
The **Revolve** mesh solid is created.

21. Now you want to subtract it from the Body. From the **Mesh Modeling** tab select the

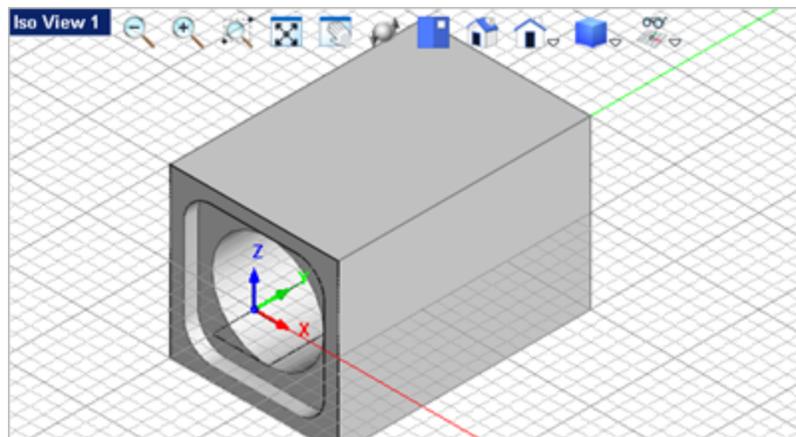


Subtract Mesh command.

22. For the first mesh solid select the Body.
23. For the second mesh solid select the Revolve.



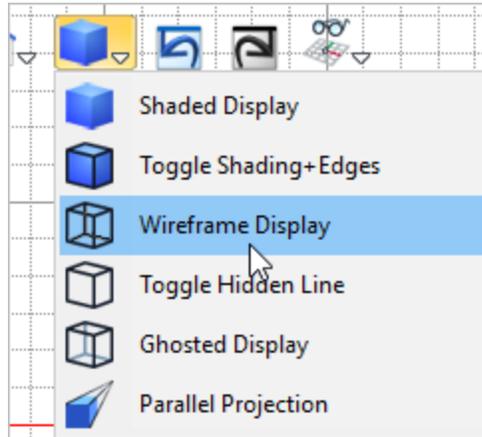
The Body is updated and should look like the image below.



10.3 Model the Top Access

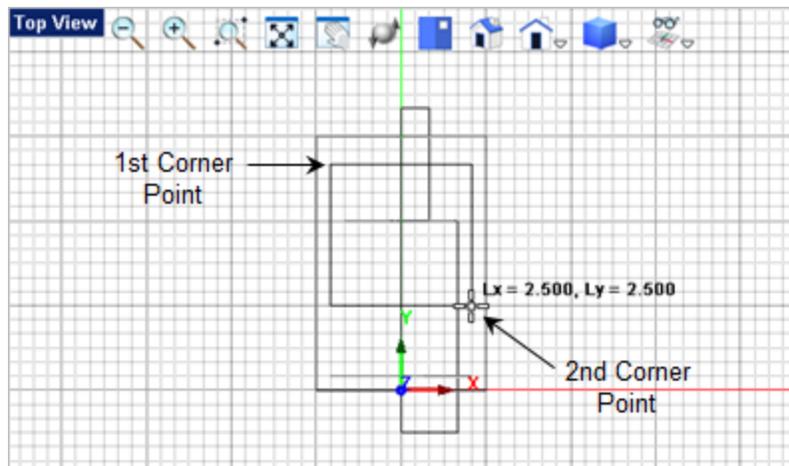
 [Model the Top Access](#)

1. Activate the **Top View**.
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.

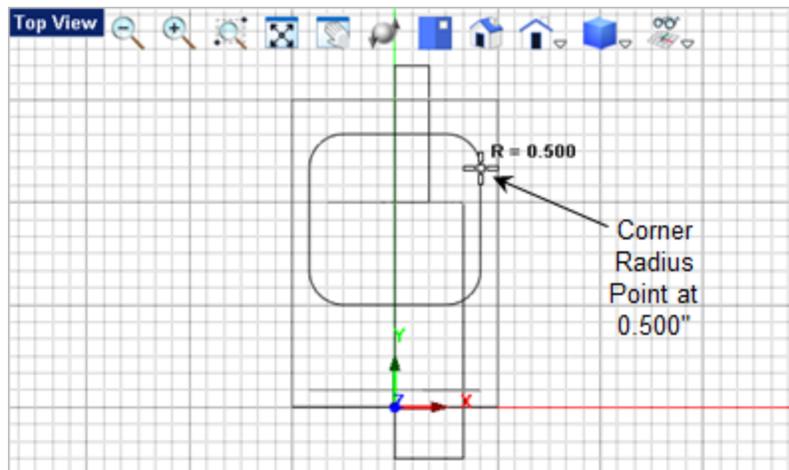


From the **Curve Modeling** tab select the **Rounded Rectangle**  command.

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>**.

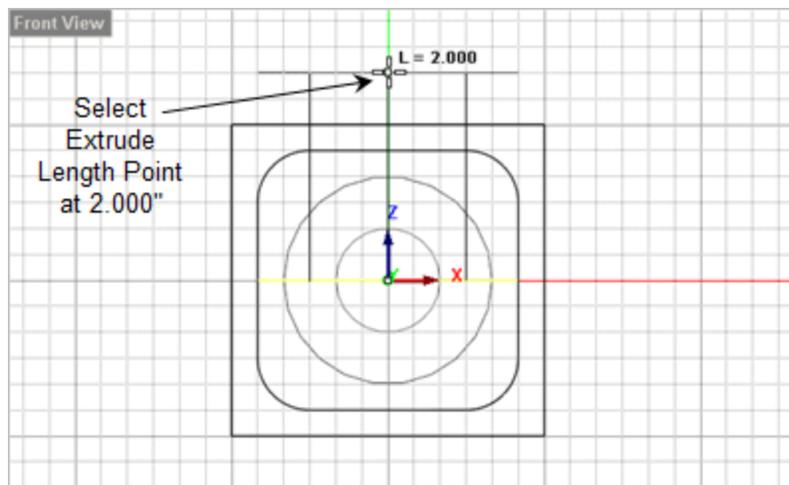


6. Now select the rounded rectangle you just created and from the **Mesh Modeling** tab,

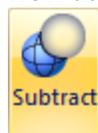


select the **Extrude Mesh** command.

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>**.

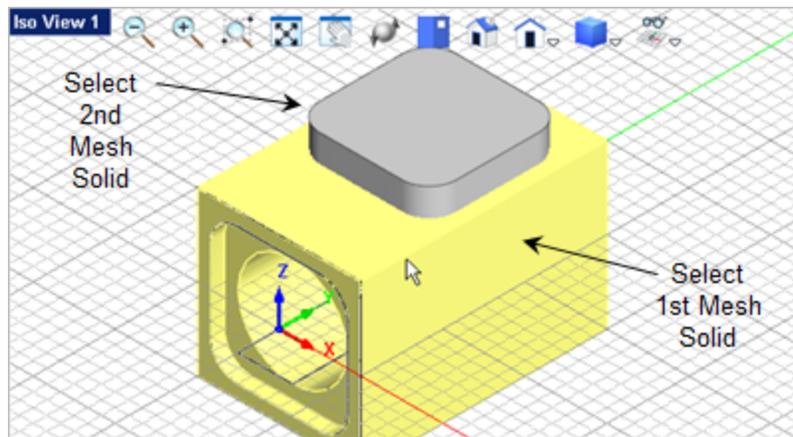


8. Now activate **Iso View 1** and from the **Mesh Modeling** tab select the **Subtract Mesh**

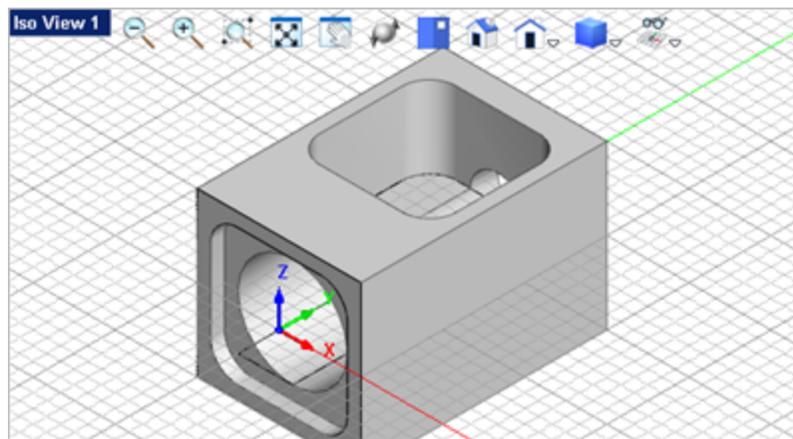


command.

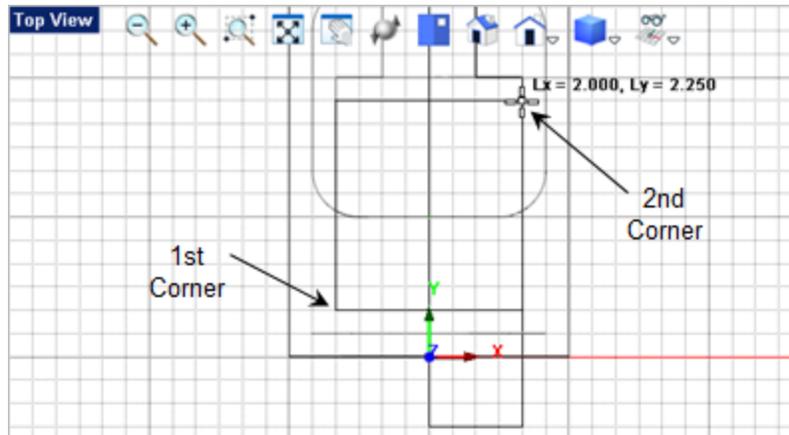
9. For the first mesh solid select the Body.



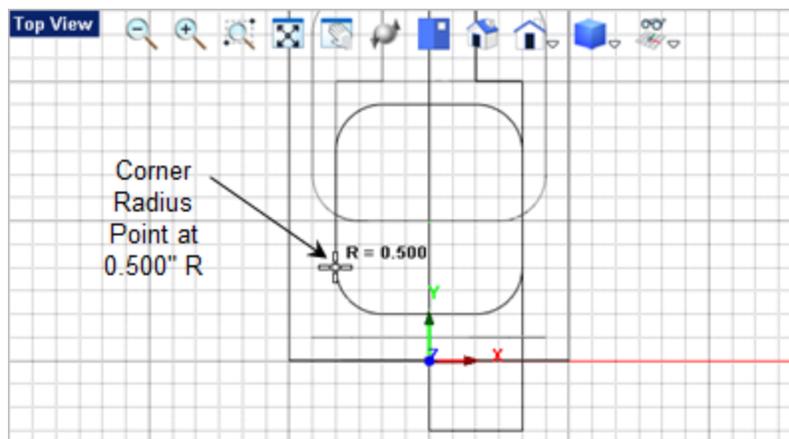
10. For the second mesh solid select the extrusion. The extrusion will be subtracted from the Body as shown below.



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** 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** and press **<Enter>**.
16. Now for the corner radius select the grid point shown below or enter 0.5 in the **Command Window** and press **<Enter>**.

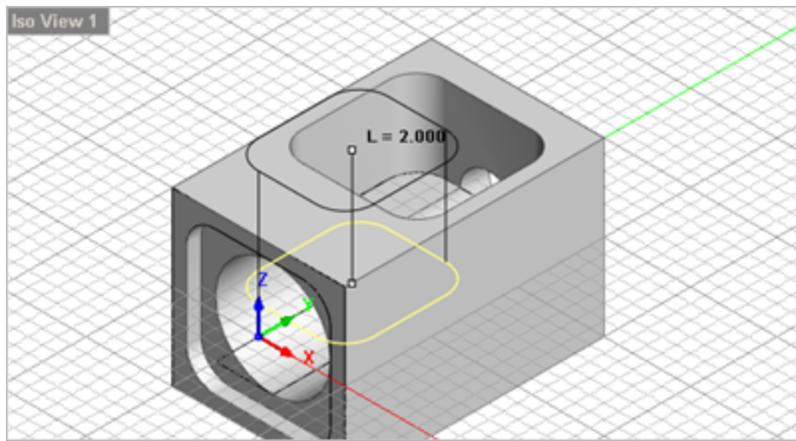
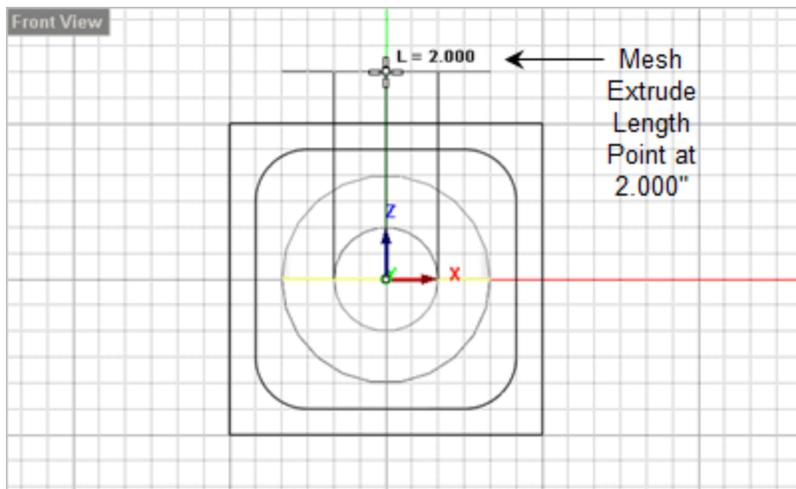


17. Now select the rounded rectangle you just created and from the **Mesh Modeling** tab,

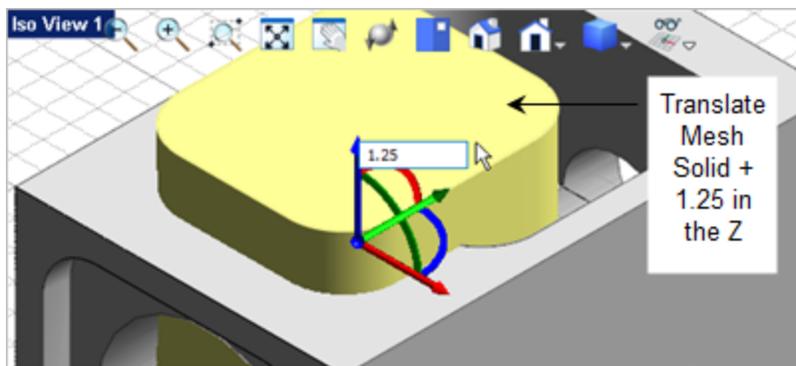


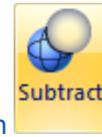
select the **Extrude Mesh** command.

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 and press **<Enter>**.

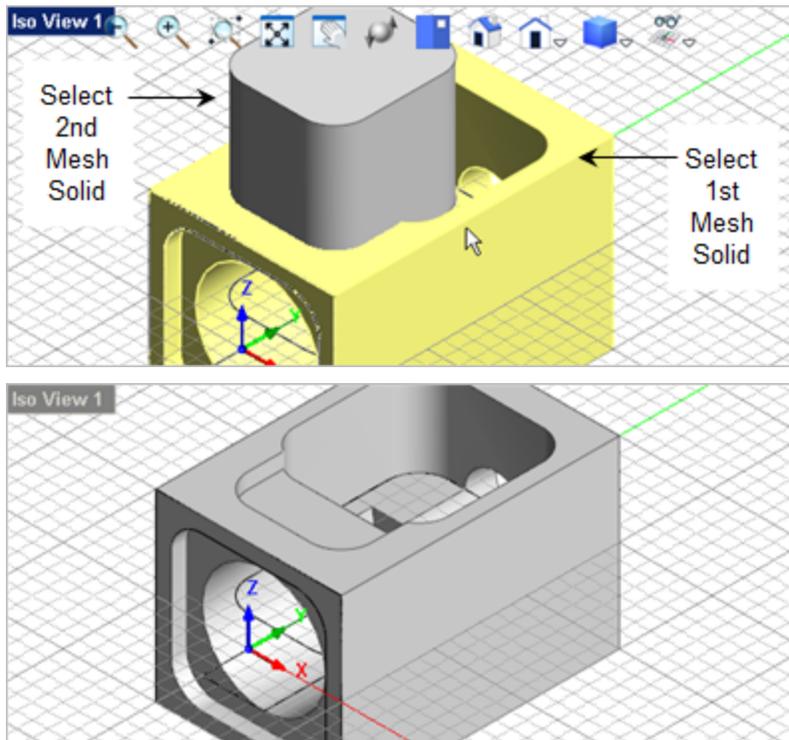


19. Now from **Iso View 1**, select the extrusion you just created to display the **Graphic Manipulator**.
20. Click on the **Blue Z Axis** arrow and enter **1.25** to translate the extrusion upward as shown in the illustration below.





21. From the **Mesh Modeling** 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.



10.4 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 **Front View**.
2. To locate the center of the first hole you will draw two lines on the grid. From the

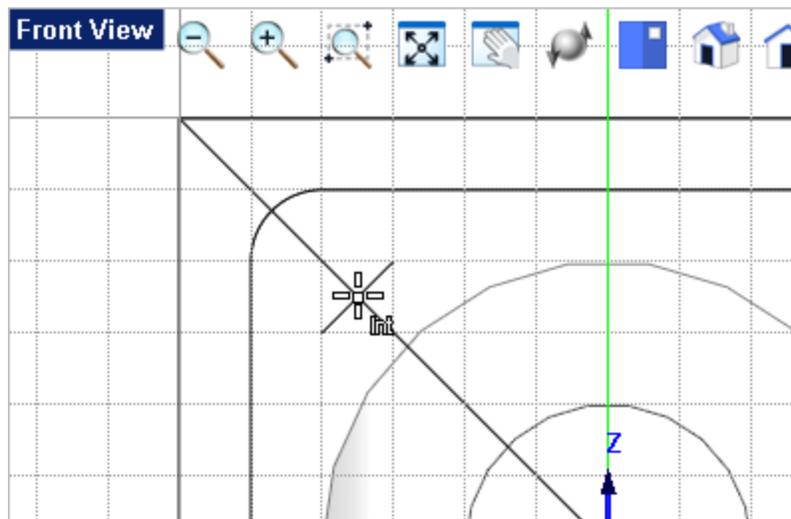
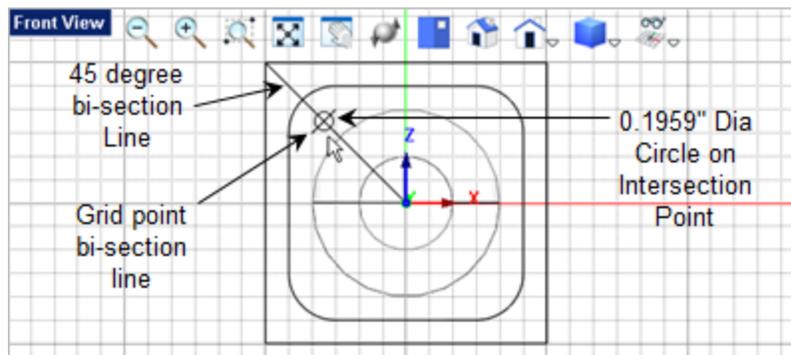


Curve Modeling **Curve Modeling** tab select the **Line** command.

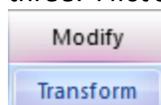
3. Draw the two lines shown below. The first starts at the center and ends at the top left corner. The second passes diagonally across two grid points bisecting the first line at 90 degrees.



4. Now from the **Curve Modeling** **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.



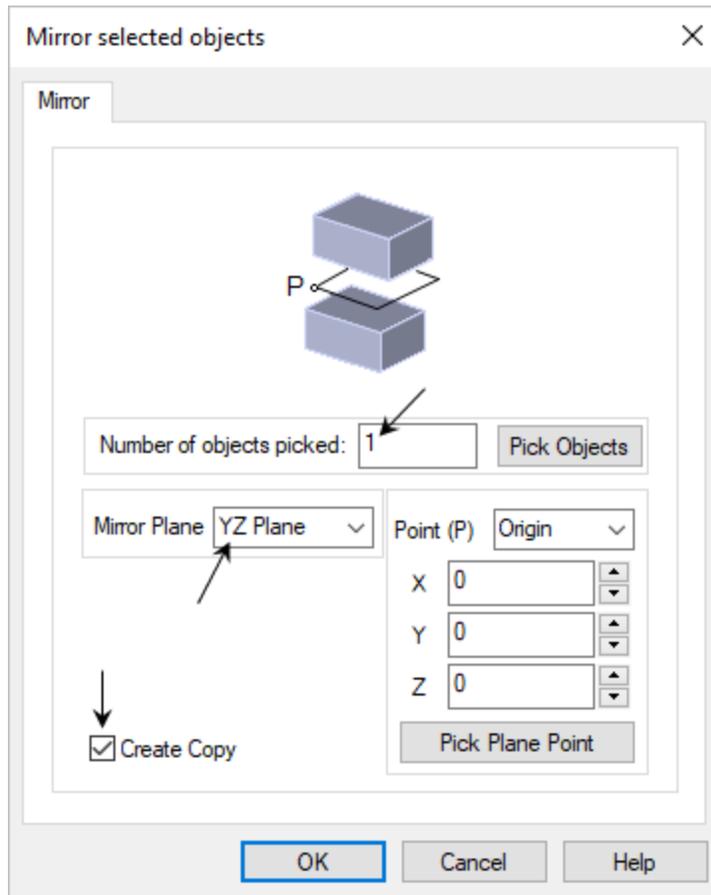
6. For the circle diameter enter **0.1959** in the **Command Window** and press **<Enter>**. This is the #7 drill diameter for a 1/4-20 tap.
7. Now you need 4 of these holes. Let's use the mirror command to create the other three. First select the circle you just created, then go to the **Modify / Transform**



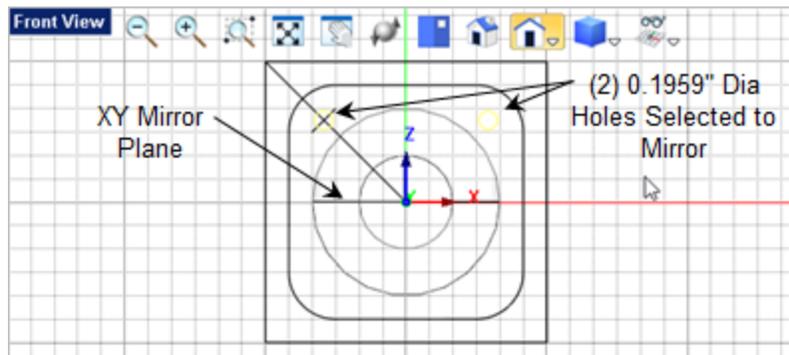
tab and select the **Mirror ...** command.

Note that the [Modify / Transfer](#) tab ONLY appears when an entity is selected!

- 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.

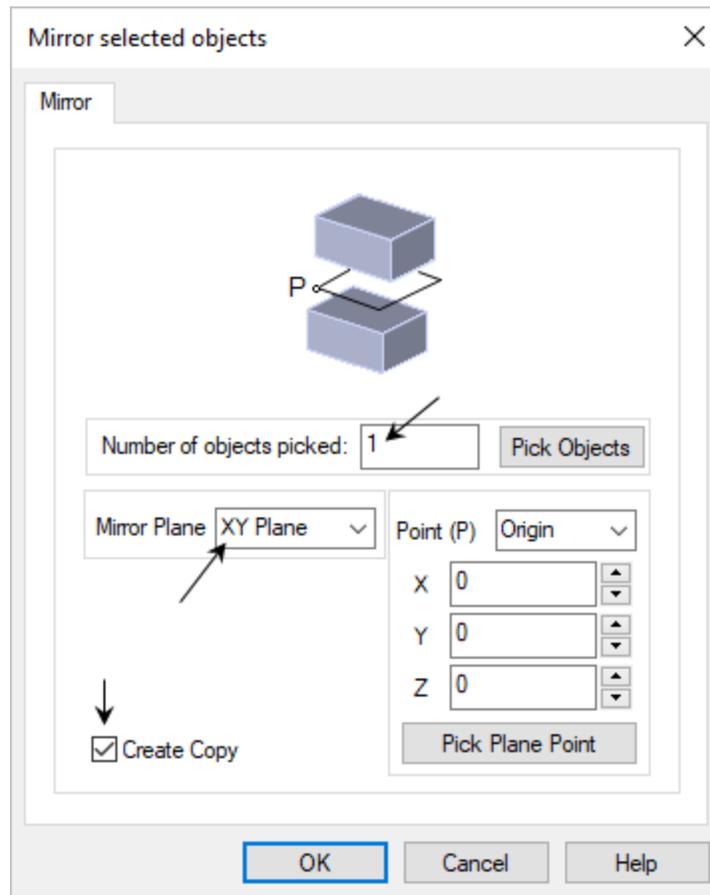


- Pick [OK](#) from the dialog and the circle will be mirrored and both circles should be selected automatically. If not, select both of the circles and pick the [Mirror ...](#) command again.



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.

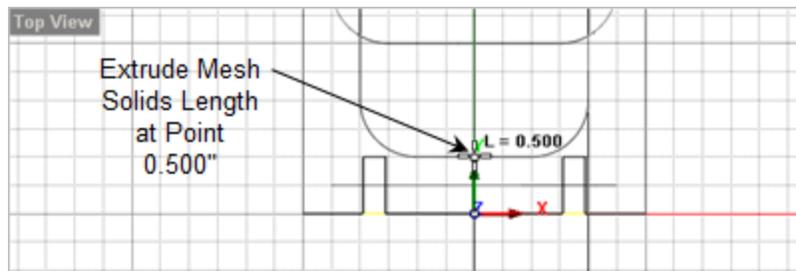
Note: You do not have to select the geometry first, as you can see this dialog has a **Pick Objects** button.



11. Now with all 4 circles drawn, all 4 should be selected automatically. If not select them now.



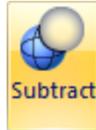
12. Now go to the **Mesh Modeling** tab and select the **Extrude Mesh** command.
13. For the extrude length move the cursor to the **Top View** and select the grid point at **0.500"** in the positive Z direction or simple enter **0.5** in the command window **Command** and press **<Enter>**.



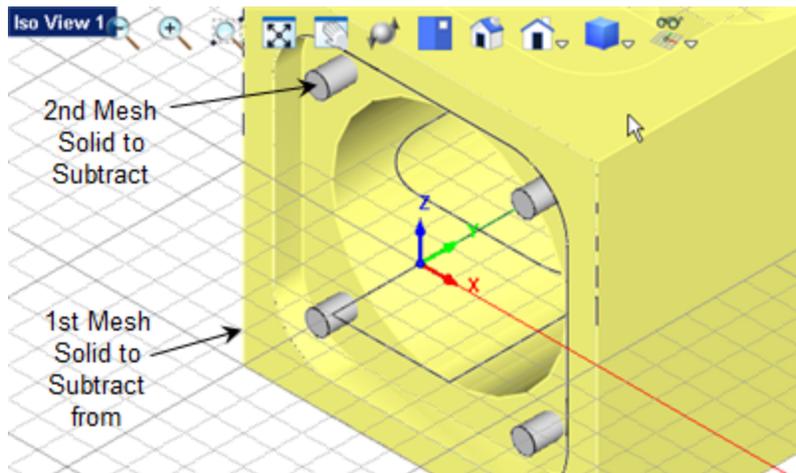
14. Now let's subtract the extrusions from the body to create the holes. Go to

Iso View 1

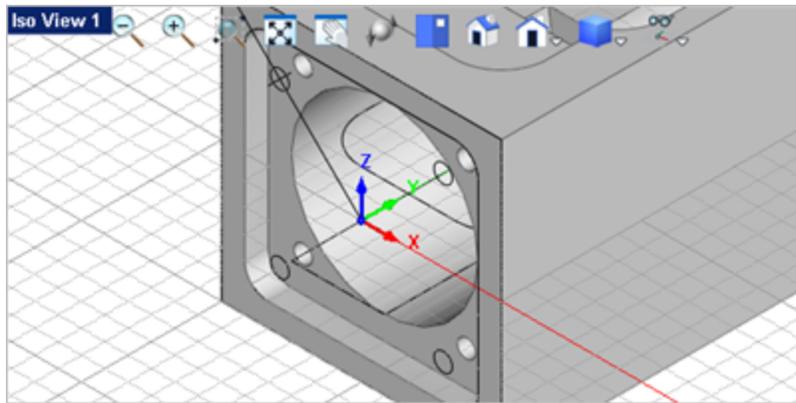
and from the **Mesh Modeling** tab select the **Subtract Mesh** command.



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 **Top View**.

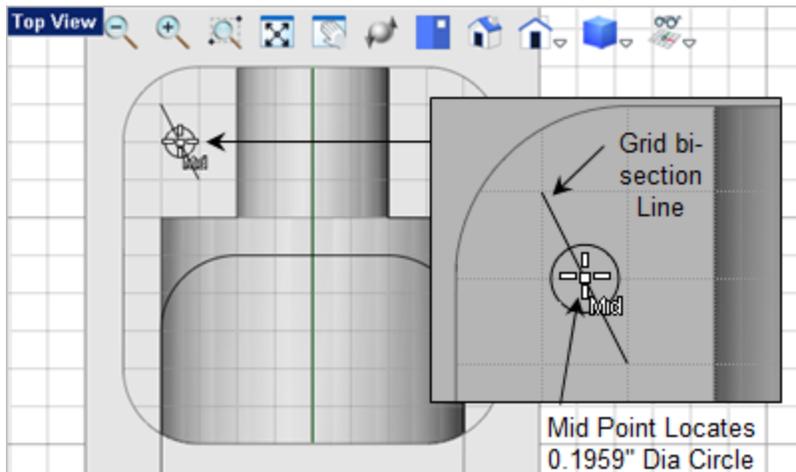
20. Let's change the display mode for the **Top View**. Select the **Display** menu from the **View Toolbar** in the **Top View** and select **Toggle Shaded + Edges**.

21. To locate the first hole, go to the **Curve Modeling** **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.

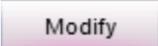
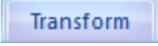


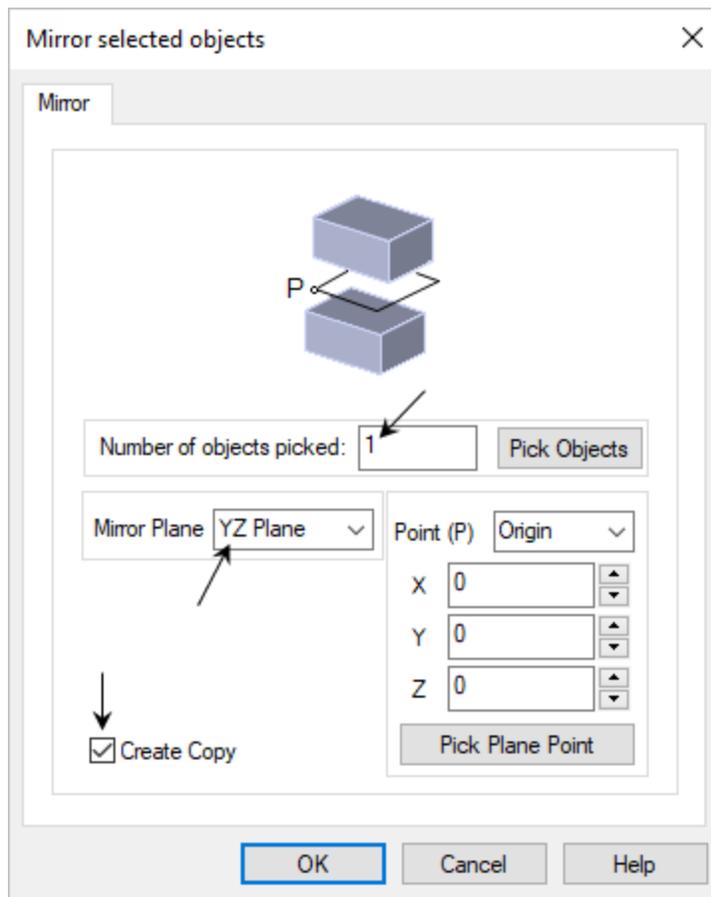
23. Now from the **Curve Modeling** **Curve Modeling** tab select the **Circle** command.

24. From the **Stats Bar** activate the **Mid Point Object Snap**  and deactivate all of the others.

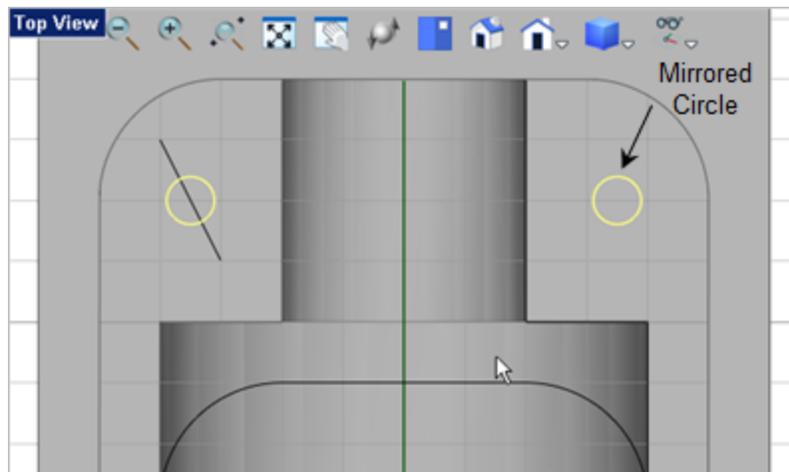


25. For the center point of the circle select the mid point of the line you just created shown in the illustration above.
26. For the circle diameter enter **0.1959** in the **Command Window**  and press **<Enter>**.

27. Now select the circle you just created and go to the **Modify**  **Transform**  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.



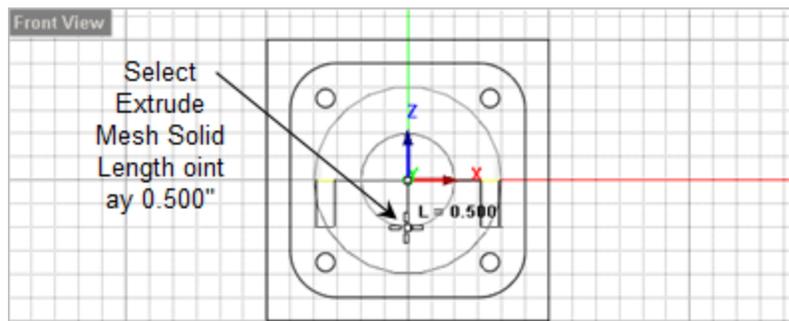
31. With the circles select go to the **Mesh Modeling** tab and select the **Extrude Mesh**



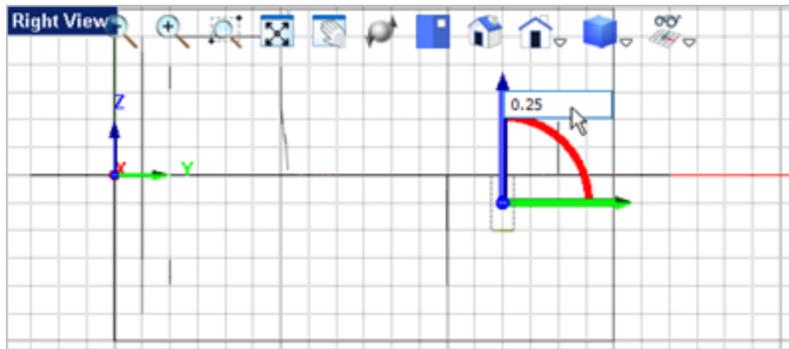
command.

32. From the **Status Bar** activate the **Grid Object Snap**  and deactivate the others.

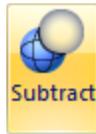
33. Move the cursor down to the **Front View** and select the grid point to set the extrusion length to **0.500"** in the negative Z direction or simply enter **0.5** in the command window **Command** and press **<Enter>**.



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"**.

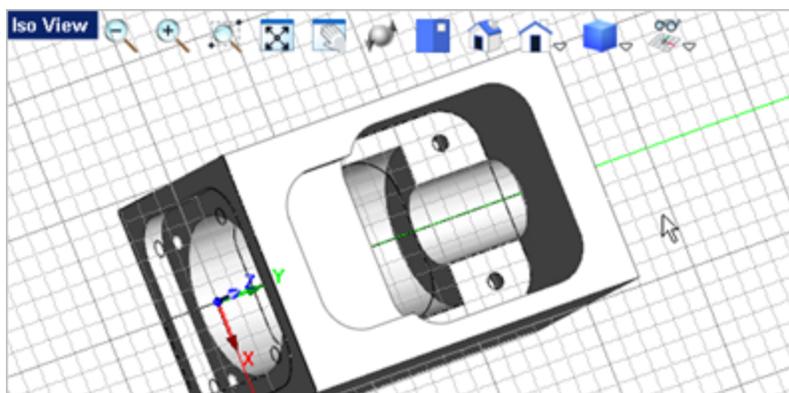
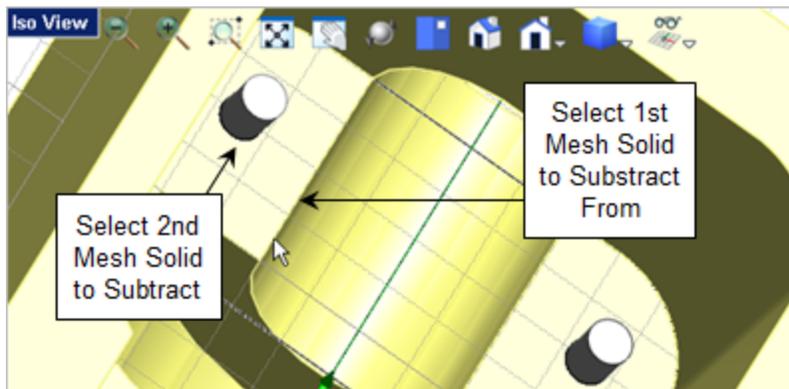


36. Now subtract them from the body. From the **Curve Modeling** **Curve Modeling** tab



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.

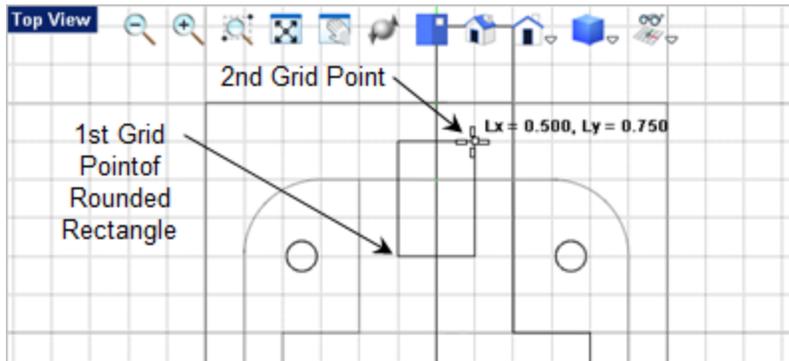


10.5 Modify the Top Access

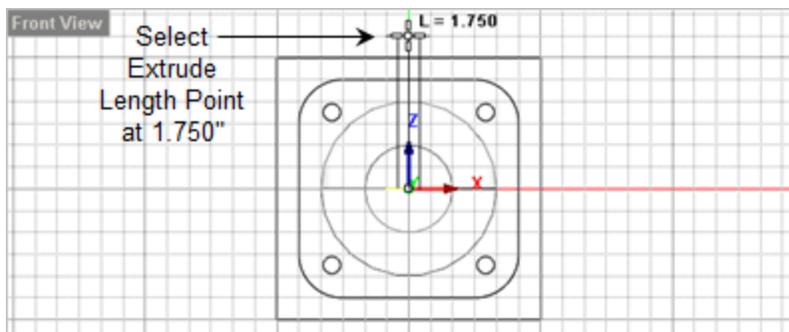
Modify the Top Access

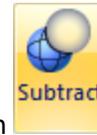
In this step we need to make a few modifications to the top access. The first is an extruded cut at the back of the access. The second is a revolved cut.

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.

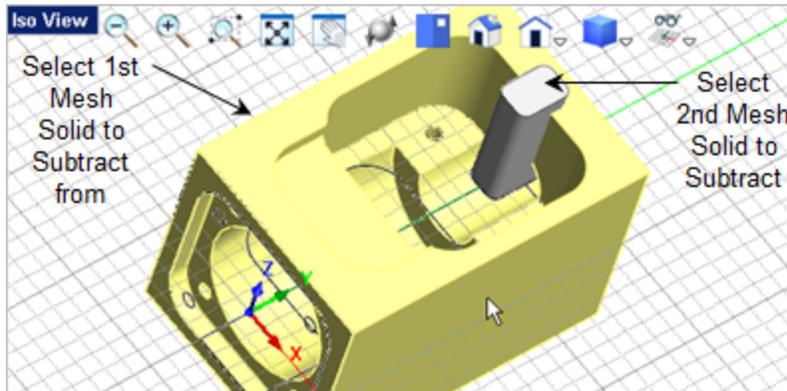


4. For the corner radius, enter **0.125** in the command window **Command** and press **<Enter>**.
5. Now select the rectangle and from the **Mesh Modeling** tab select the **Extrude Mesh**  command.
6. Move to the **Front View** and drag the extrude length to **1.750"**.





7. Now go to the **Mesh Modeling** 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.

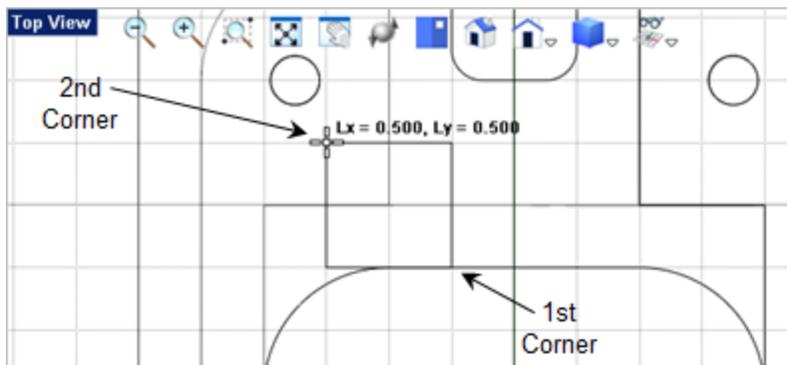


9. For the second modification, go to the **Top View** and from the **Curve Modeling** tab



select the **Rectangle** command.

10. Select the two corner grid points shown below.



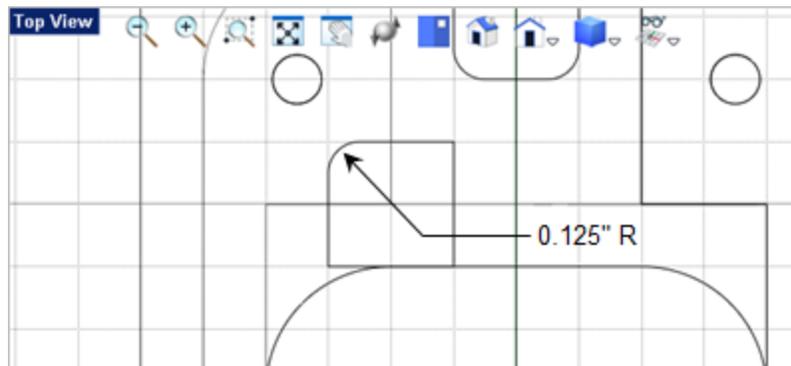
11. Now we need to fillet one of the corners. From the **Curve Modeling** tab select the



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.



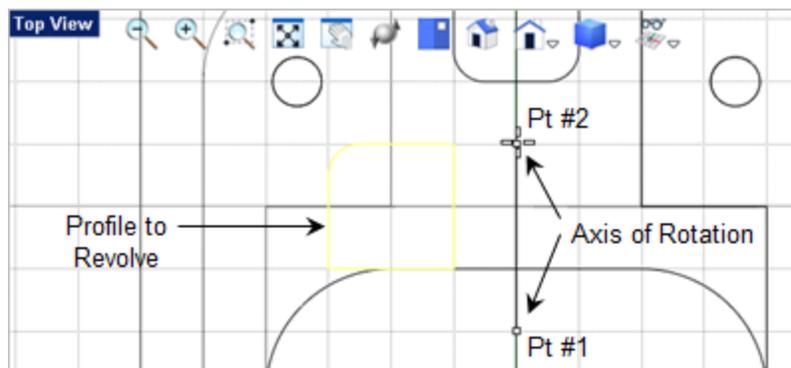
- 14. We need to revolve this, so select the closed rectangular profile, go to the



Mesh Modeling

tab and select the Revolve Mesh command.

- 15. For the axis of revolution, select the two grid points shown below.

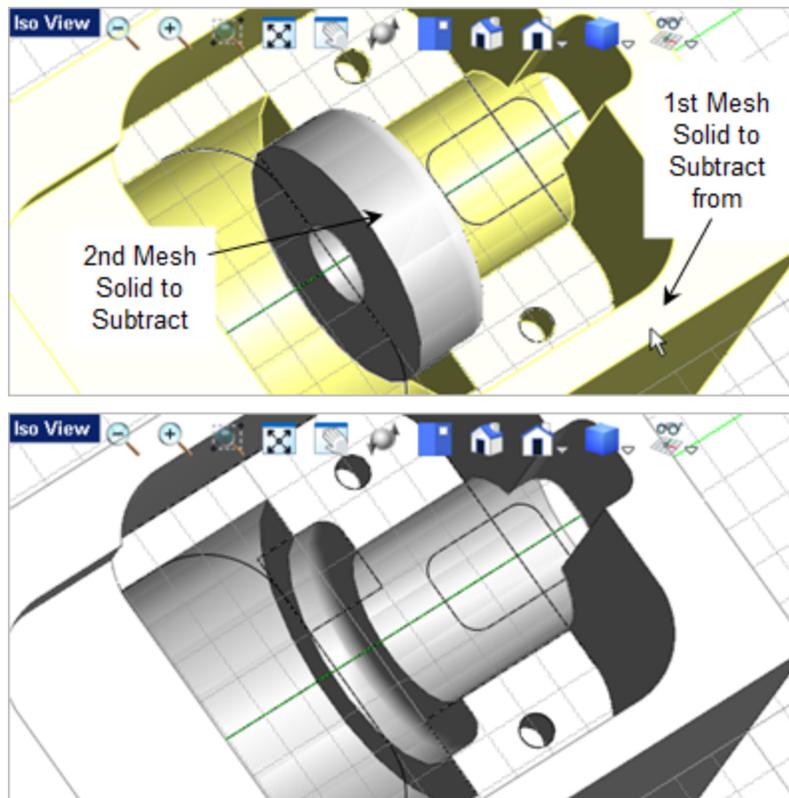


- 16. Now subtract the revolve from the body. From the Mesh Modeling tab select the



Subtract Mesh command.

- 17. For the first mesh solid select the body and for the second select the revolve.

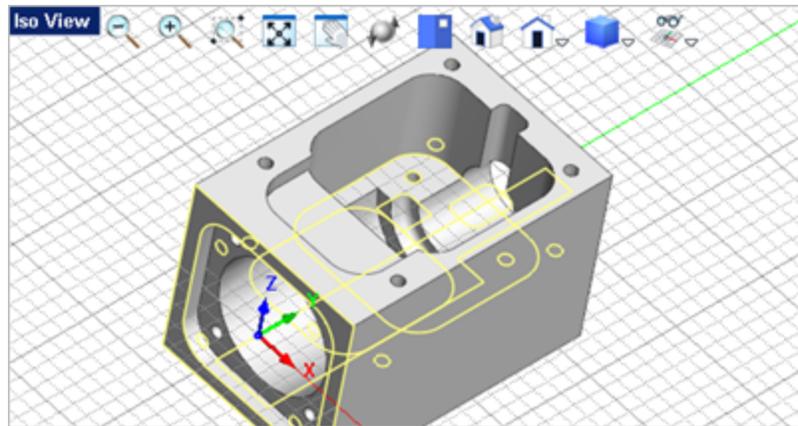
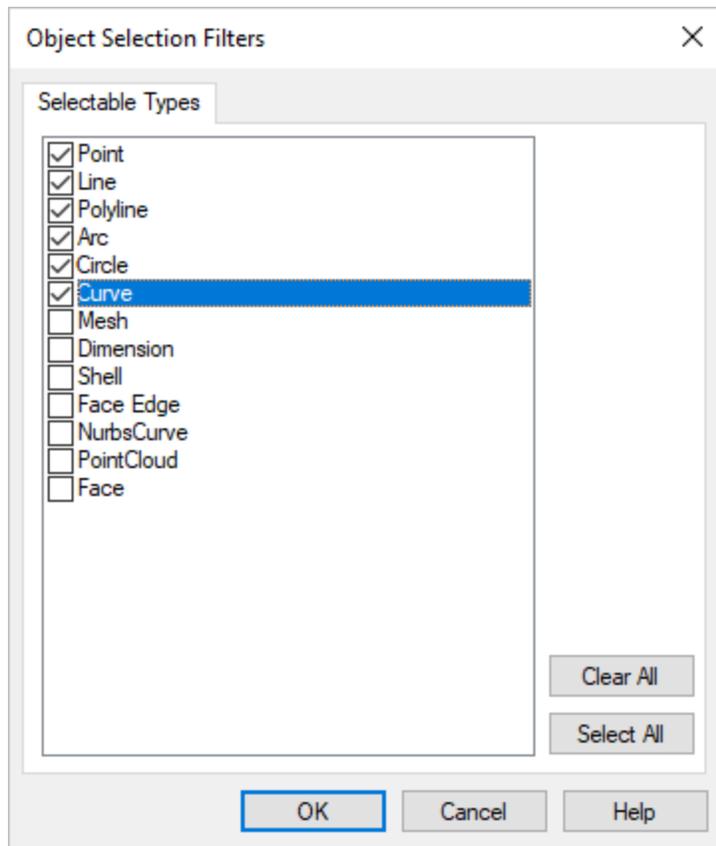


10.6 Create Section Curves

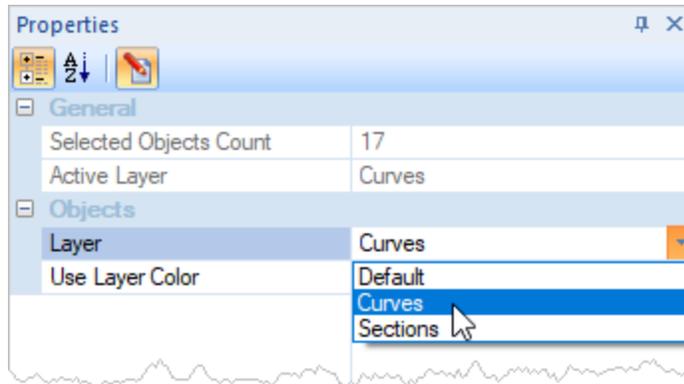
Create Section Curves

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.

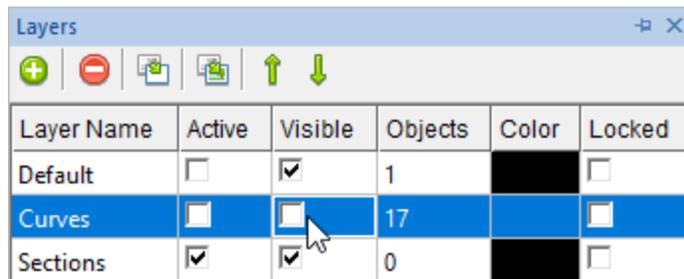


3. From the **Status Bar** select the **Properties** icon  to display the **Properties Manager**.



4. This dialog reports that a total of **17** entities 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 .

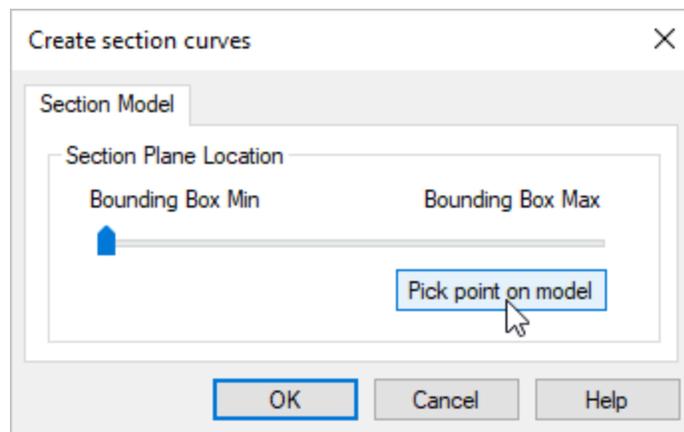


6. From the **Layer Manager**, under the **Active** column, check the box to make the layer named **Sections** the active layer.

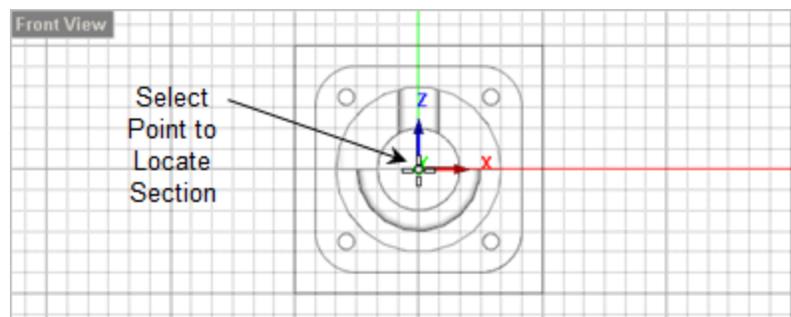
7. You can close the **Properties** and **Layer managers** by allowing them to auto-hide or pick the "X" icon to close them.

8. Now we're ready to create the section curves. First activate the **Top View**. Sections are generated for the currently active view.

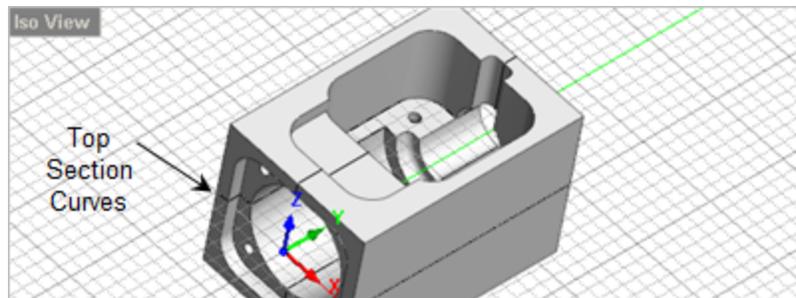
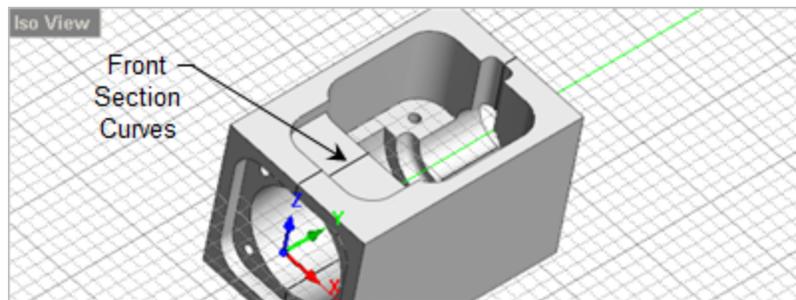
9. Go to the **Curve Modeling** tab and select the **Section Curves**  command. This will display the **Create section curves** dialog.



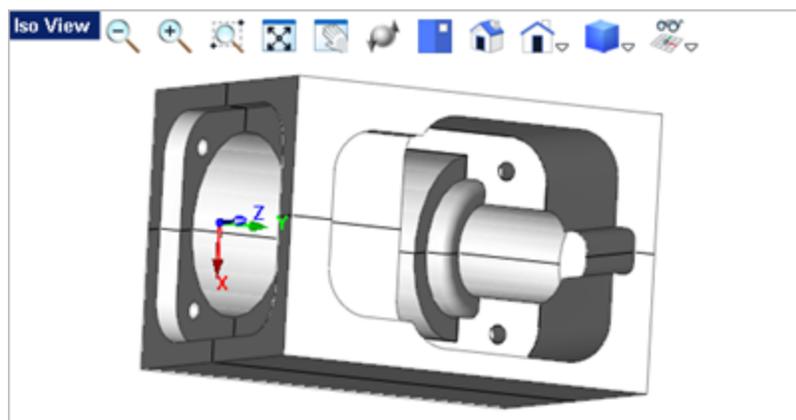
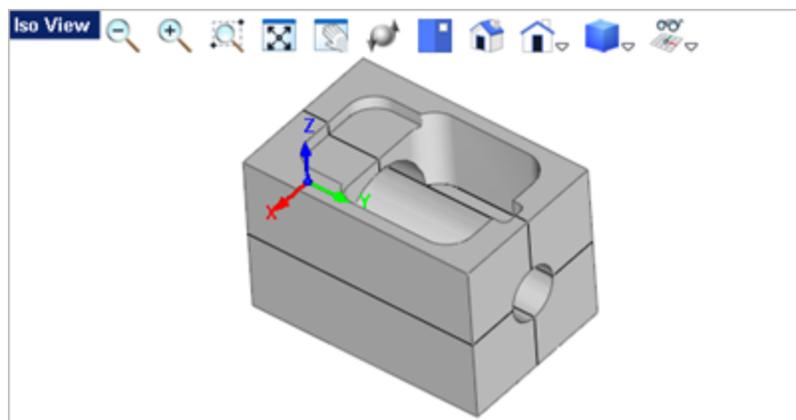
10. If you move the slider you will see a highlight of the section display in all views.
11. Now from the dialog select the **Pick point on model** button. The dialog minimizes allowing you to select a point. From the **Front View**, select the **XY origin** grid point to locate the section.



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**.



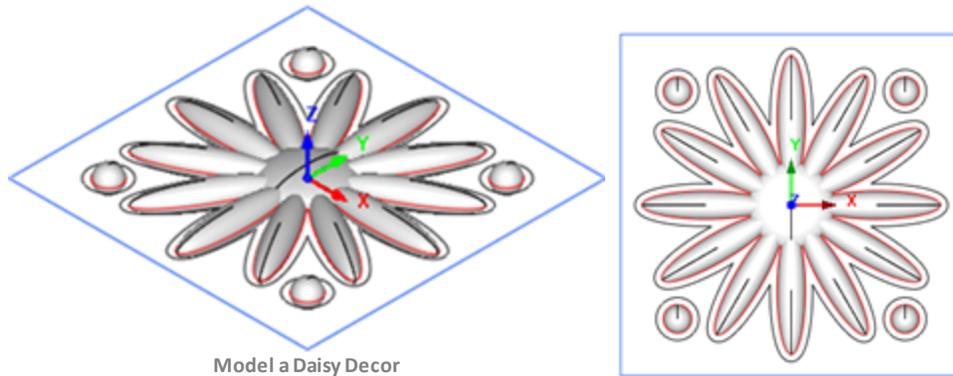
Congratulations!!! You completed the exercise!!



17. Save the file as [Connector_Block_Completed.vcp](#).

#7: 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.



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. [Ghosed 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.](#)

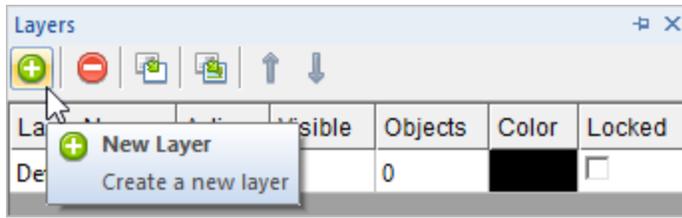
11.1 Create new Layers



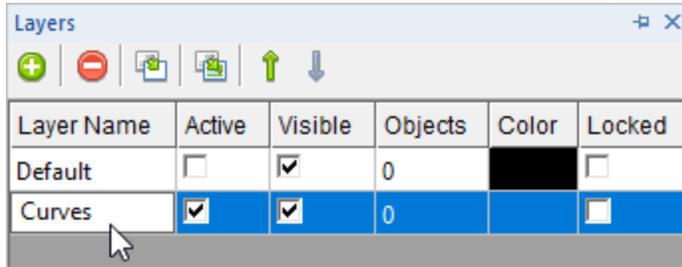
Create new Layers

In this step you will use the [Layer Manager](#) to create some new layers and then set the active layer.

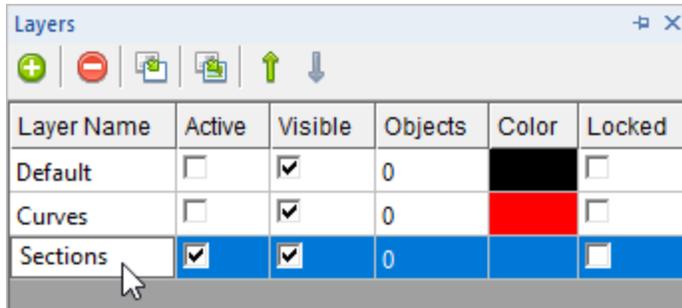
1. Display the [Layer Manager](#). You can pick the [Layer Manager](#) icon  located on the [Status Bar](#).
2. From the [Layer Manager](#) select the [Add Layer](#) icon  to add a new layer.



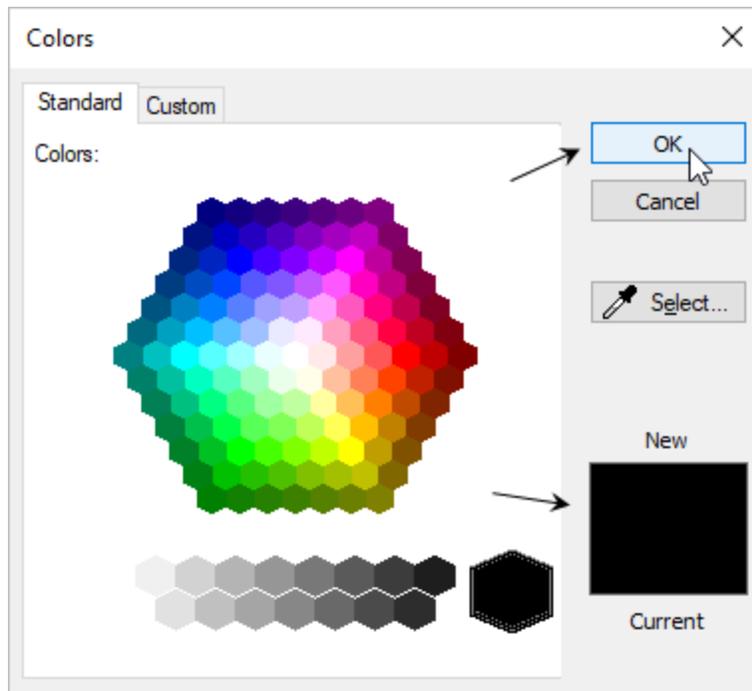
- Now let's rename the new layer to **Curves**. First double-left-click in the **Name** field to activate it, and enter **Curves** as the new layer name.



- Repeat the procedure to create a new layer, this time named **Sections**.



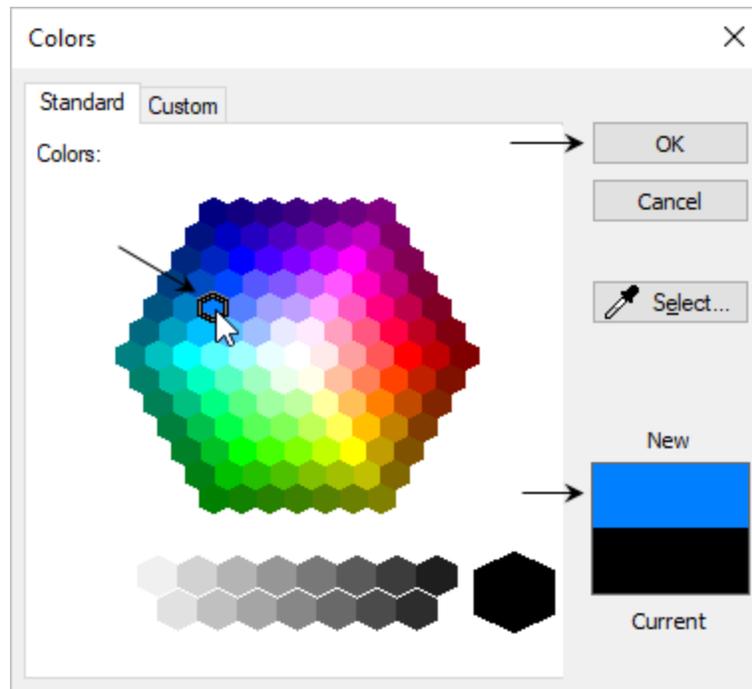
- Now repeat the procedure again, this time named **Stock**.
- Now let's change the color of a couple of the new layers. In the **Color** field left-click on the color box for the layer named **Curves**. This will display the **Colors** dialog.
- Accept the default color **Black** and then pick **OK** 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.

Layer Name	Active	Visible	Objects	Color	Locked
Default	<input type="checkbox"/>	<input type="checkbox"/>	17	Black	<input type="checkbox"/>
Curves	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0	Blue	<input type="checkbox"/>
Sections	<input type="checkbox"/>	<input type="checkbox"/>	5	Red	<input type="checkbox"/>
	<input type="checkbox"/>	<input type="checkbox"/>	0	Blue	<input type="checkbox"/>

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	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0		<input type="checkbox"/>
Curves	<input type="checkbox"/>	<input checked="" type="checkbox"/>	0	Black	<input type="checkbox"/>
Sections	<input type="checkbox"/>	<input checked="" type="checkbox"/>	0	Red	<input type="checkbox"/>
Stock	<input type="checkbox"/>	<input checked="" type="checkbox"/>	0	Blue	<input type="checkbox"/>

11.2 Model & Scale the Body

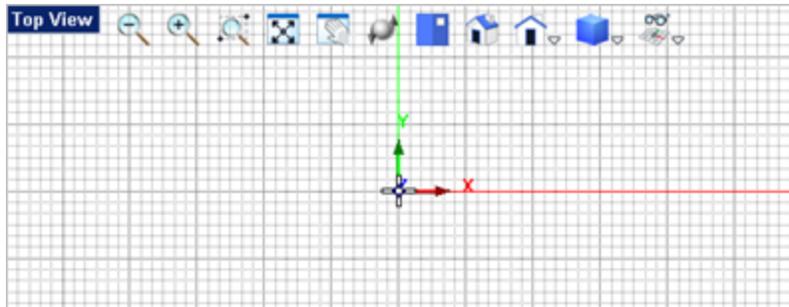
Model & Scale the Body

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.

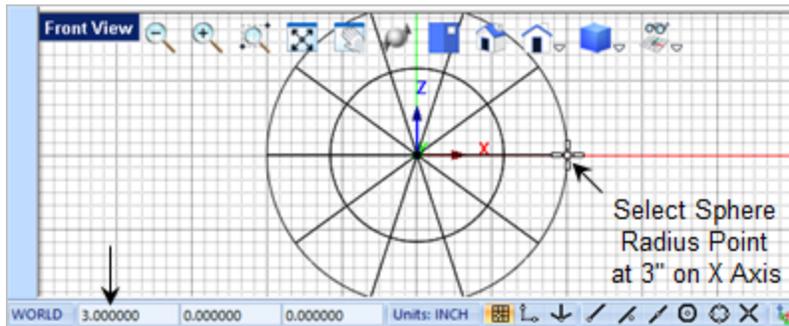
1. Activate the **Top View** **Top View** tab by selecting anywhere in the viewport.
2. Select the **Solid Modeling** **Solid Modeling** tab from the top **Ribbon Bar**.



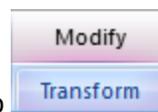
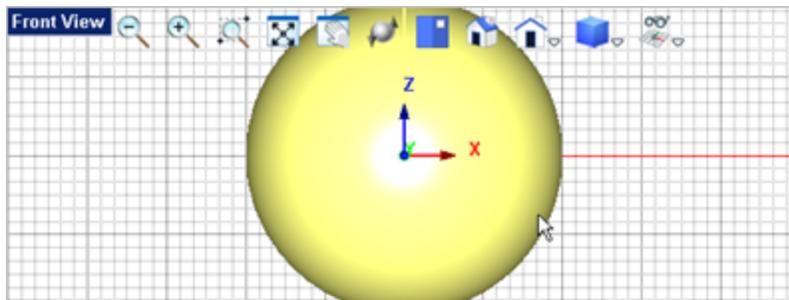
3. Now select the [Sphere](#) command.
4. In the [Status Bar](#) make sure the [Grid Snap](#) is [On](#) .
5. For the center point select the grid point at (0,0,0) or just enter the coordinates 0,0,0 in command input bar .



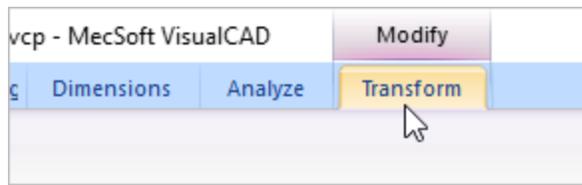
6. For the sphere radius, enter 3,0,0 or select the corresponding grid point and press <Enter>.



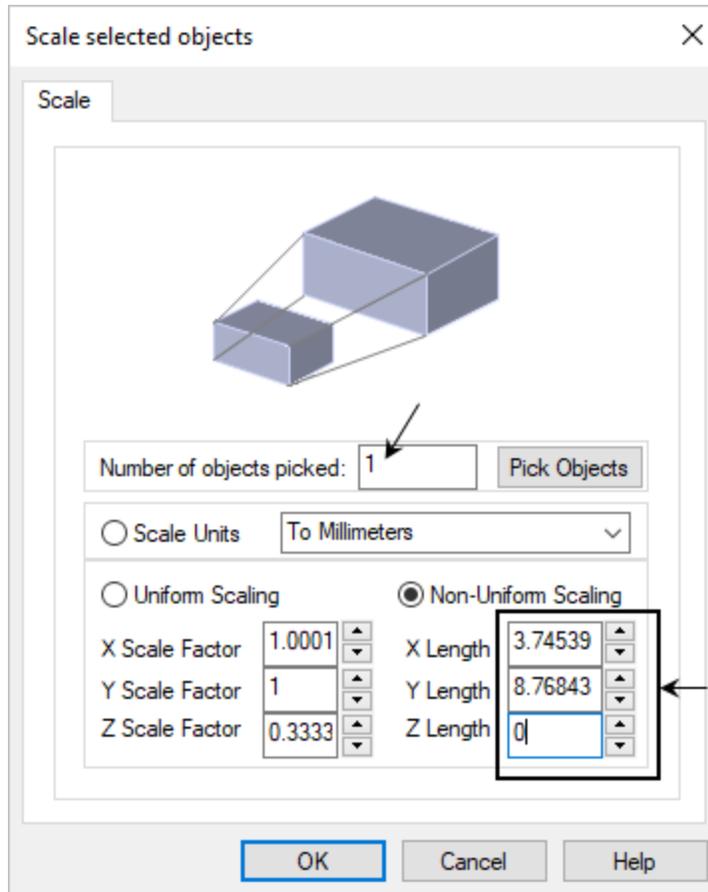
7. Select the solid sphere you just created.



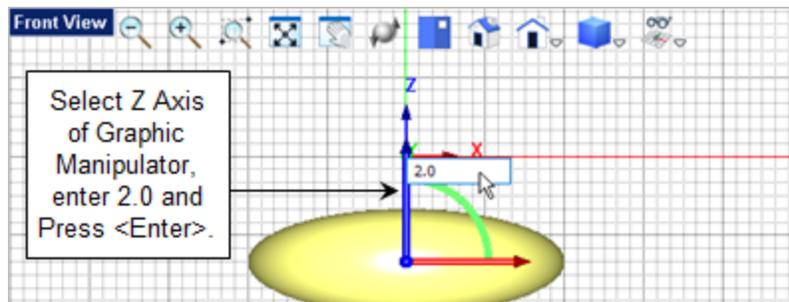
8. You see the [Modify / Transform](#) tab appear on the right end of the [Ribbon Bar](#). It ONLY appears when an entity is selected. Select the [Transform](#) tab.



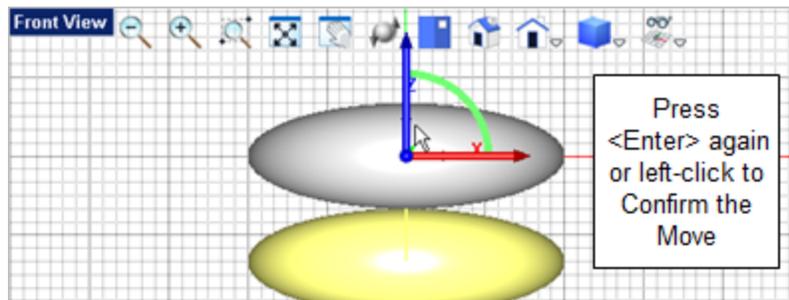
9. Select the [Scale](#)  command.
10. From the [Scale Selected Objects](#) dialog, select the [Non-Uniform Scaling](#) option.



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.



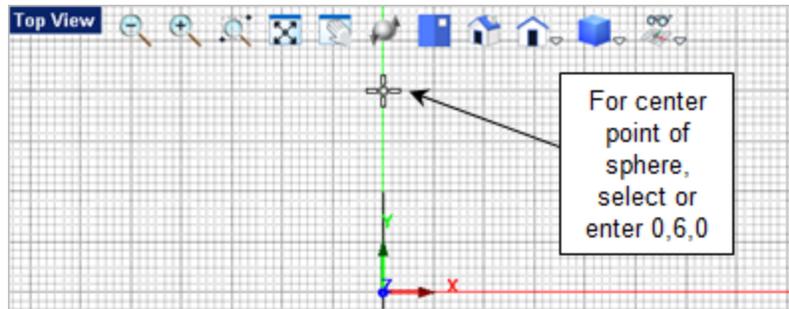
11.3 Model & Scale the Pedal

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.

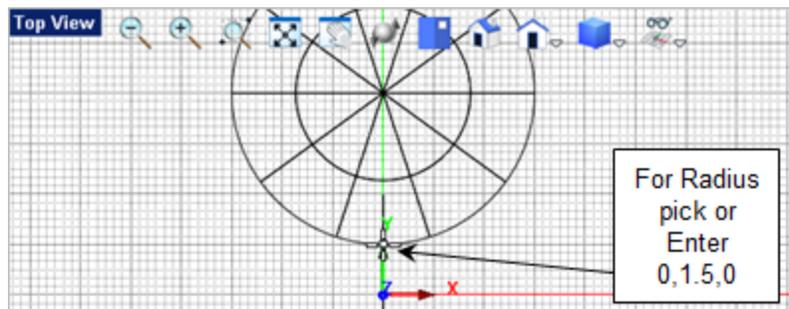
1. Activate the **Top View** **Top View** tab by selecting anywhere in the viewport.

2. From the **Solid Modeling** **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 **Command** and press **<Enter>**.

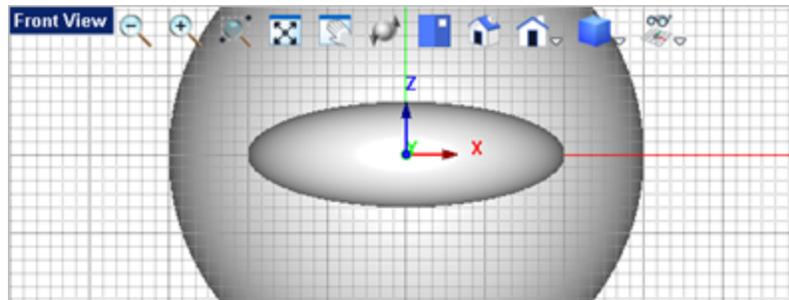


5. For the sphere radius, enter 4.5 in the command input window and press <Enter> or select the grid point located at 0,1.5,0.

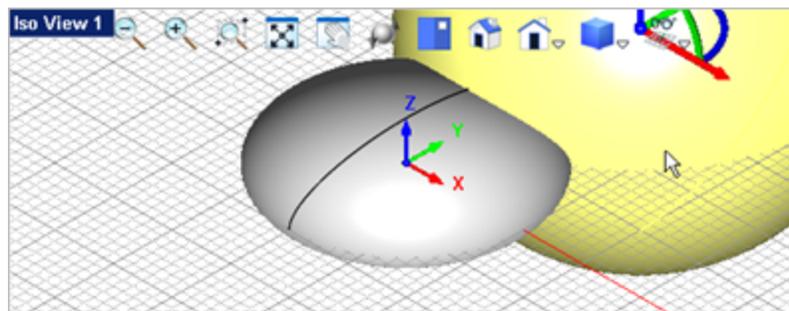
Command

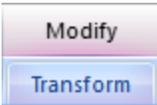


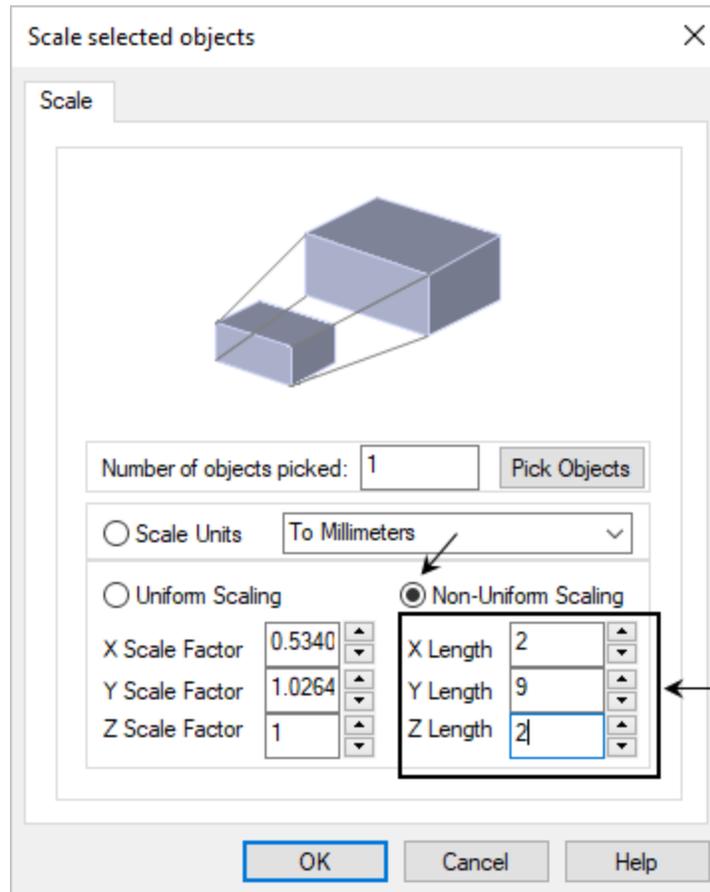
Here is what your part should look like from the **Front View**:



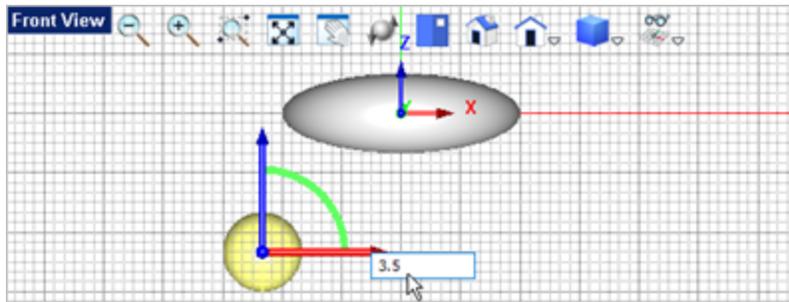
6. From **Iso View 1**, **Iso View 1** select the solid sphere you just created.



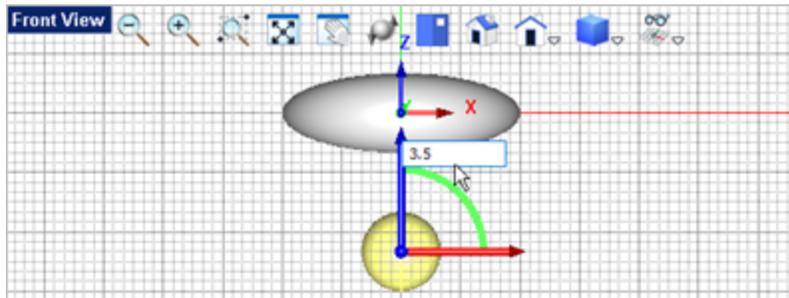
7. Select the **Transform**  tab from the ribbon bar.
8. Select the **Scale**  command.
9. From the **Scale Selected Objects** dialog, select the **Non-Uniform Scaling** option.



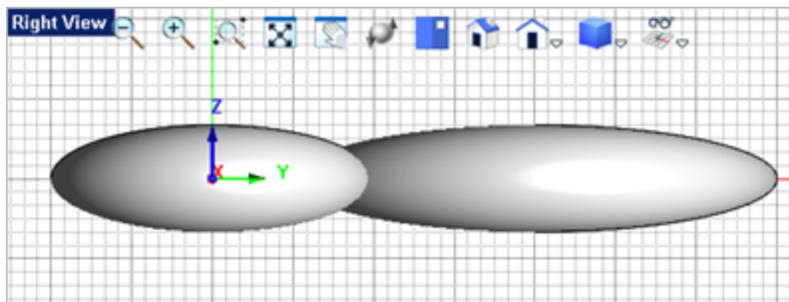
10. Set the **X Length** to 2, **Y Length** to 9 and the **Z Length** to 2.
11. Pick **OK** and the sphere is scaled as shown below.
12. Now press the **<Enter>** key to accept the modification.
13. Let's move the scaled sphere into position along the **Y Axis**. First make sure the **Graphic Manipulator**  is **On**. The icon is located on the **Status Bar**.
14. From the **Front View** , 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** **Right View** your part should look like the image below.



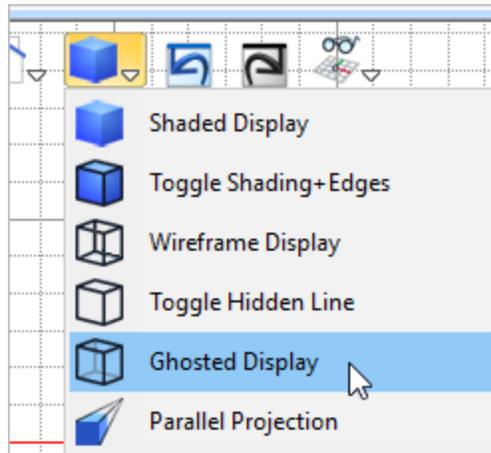
11.4 Ghosted Display Mode

Change the Top View to Ghosted Display

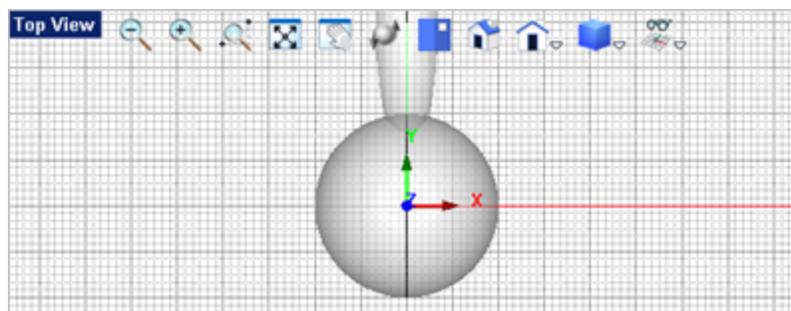
For spheres drawn from the top view, it is best viewed using the ghosted display.

1. Activate the **Top View** **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:

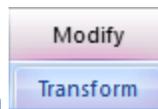
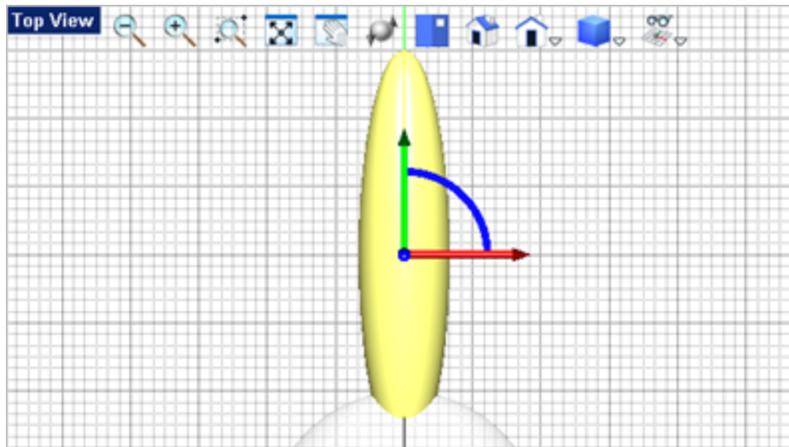


11.5 Polar Array the Pedal

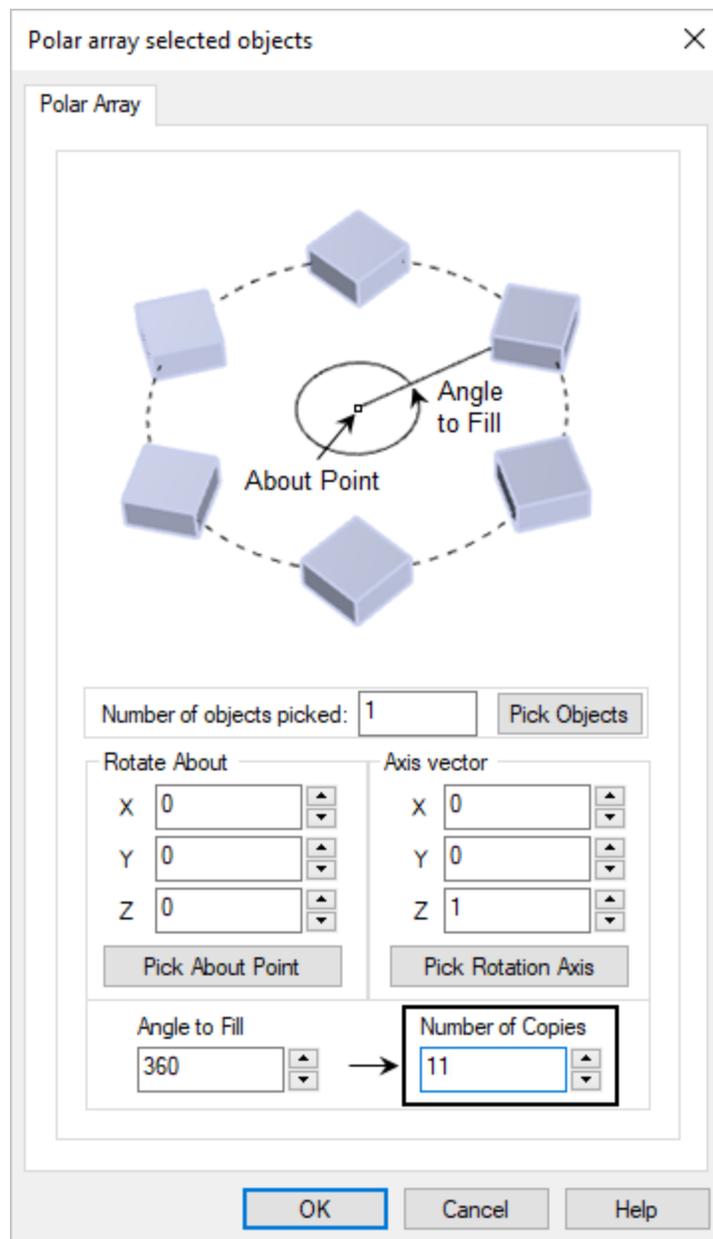
Create the Polar Array

In this step you will create a polar array of the pedal sphere we just created.

1. Activate the **Top View** **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.

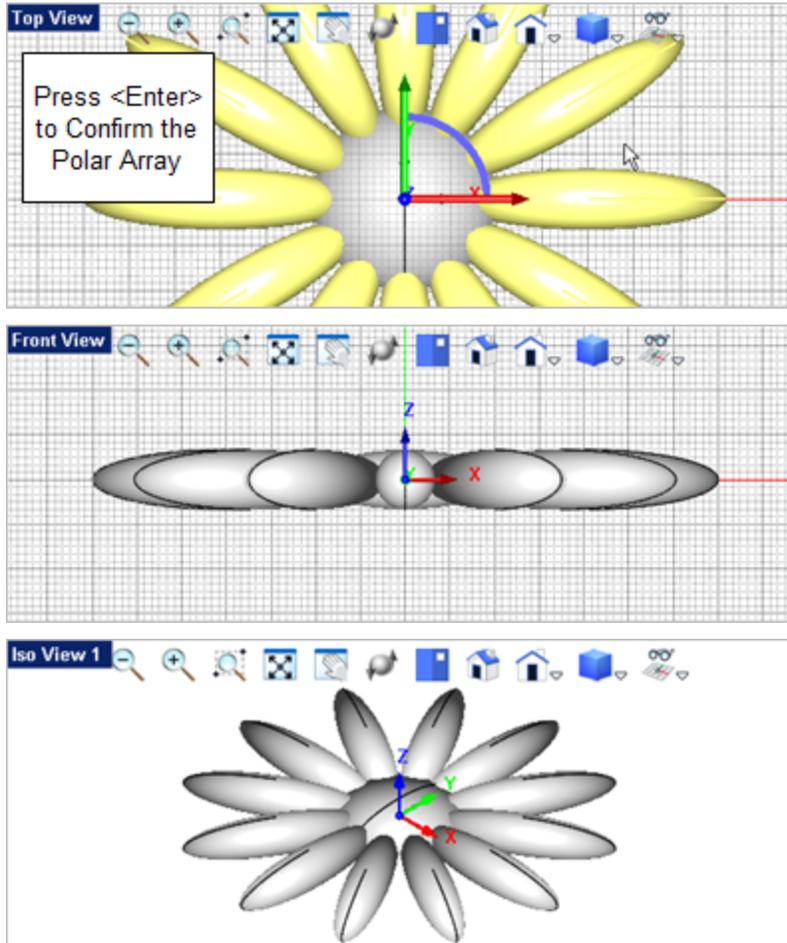


5. In the [Number of Objects](#) field, enter [11](#).
6. The remaining options in the dialog should use the default values shown in the dialog above.
7. Pick [OK](#) from the dialog to create and preview the polar array.

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.



11.6 Additional Spheres

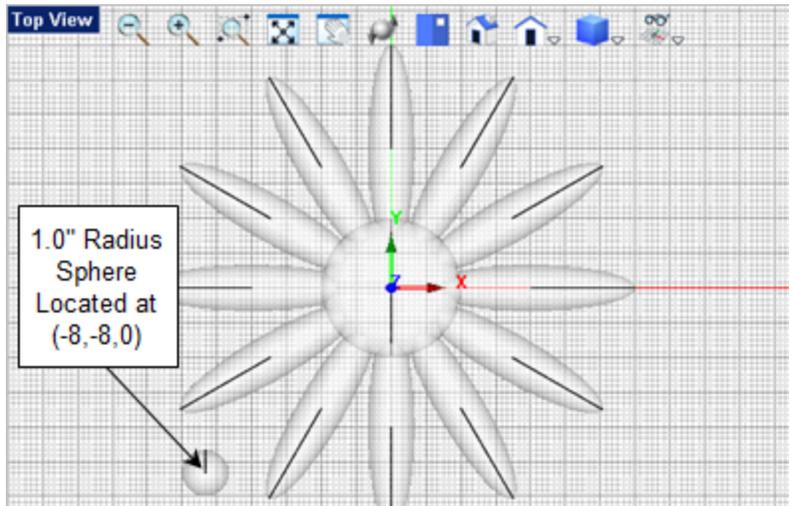
Create the 4 additional Spheres

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.

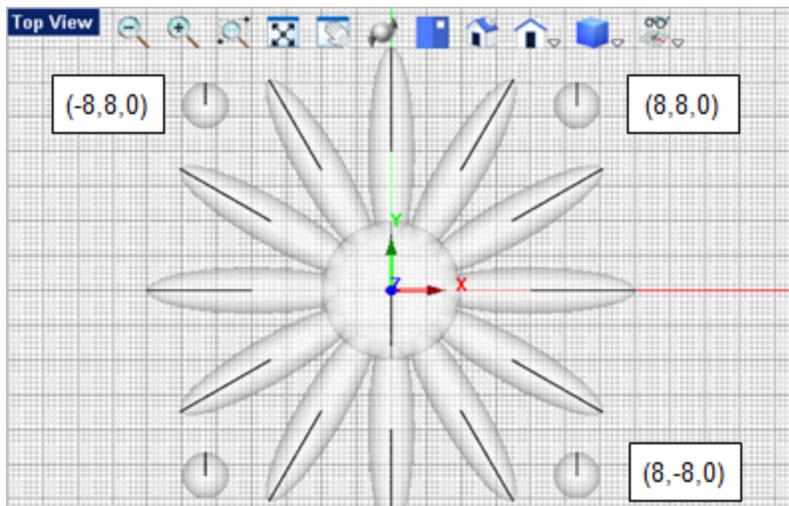
1. Activate the Top View **Top View** tab by selecting anywhere in the viewport.

2. From the Solid Modeling **Solid Modeling** tab select the Sphere  command.

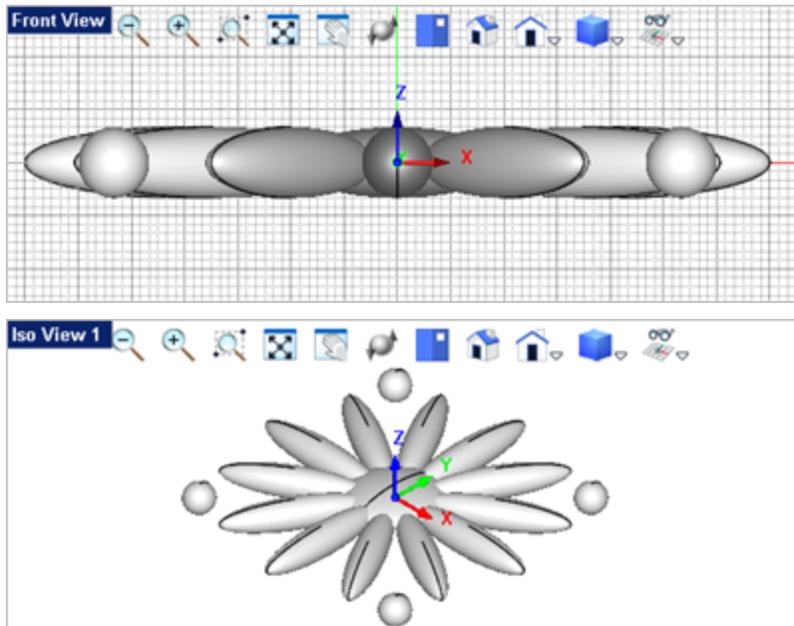
3. In the **Status Bar** make sure the **Grid Snap** is On .
4. For the center point select the grid point at $(-8,-8,0)$ or just enter the coordinates $-8,-8,0$ in command input bar and press **<Enter>**.
5. For the sphere radius, enter 1.0 in the command input bar and press **<Enter>** or select the 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:



11.7 Create & Trim Section Curves

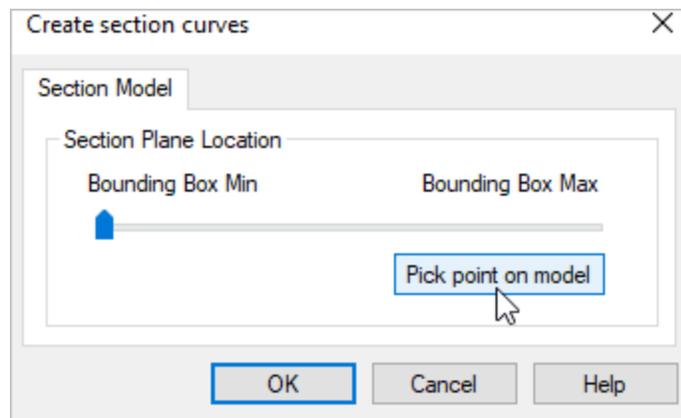
Create Section Curves

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.

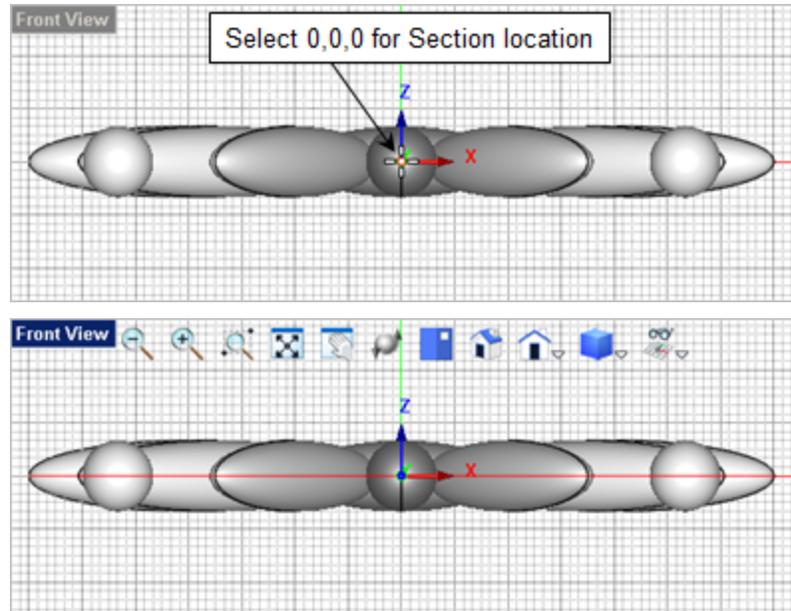
1. Open the **Layer Manager**  and set the **Active Layer** to **Sections**. Refer to the previous steps for displaying and using the **Layer Manager**.

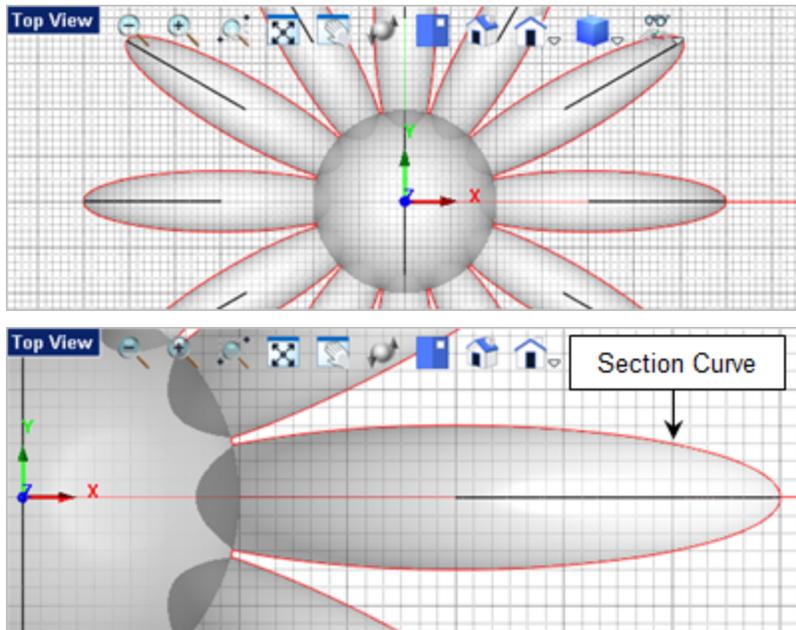
Layer Name	Active	Visible	Objects	Color	Locked
Default	<input type="checkbox"/>	<input checked="" type="checkbox"/>	0	Black	<input type="checkbox"/>
Curves	<input type="checkbox"/>	<input checked="" type="checkbox"/>	0	Black	<input type="checkbox"/>
Sections	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0	Blue	<input type="checkbox"/>
Stock	<input type="checkbox"/>	<input checked="" type="checkbox"/>	0	Blue	<input type="checkbox"/>

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 **Command** and press **<Enter>**. The section curves will be created and displayed in all four viewports.





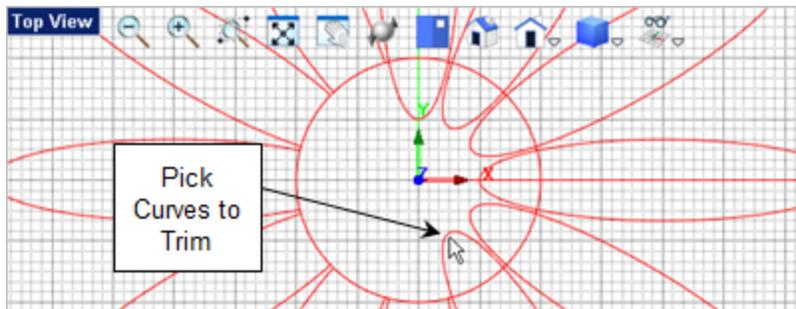
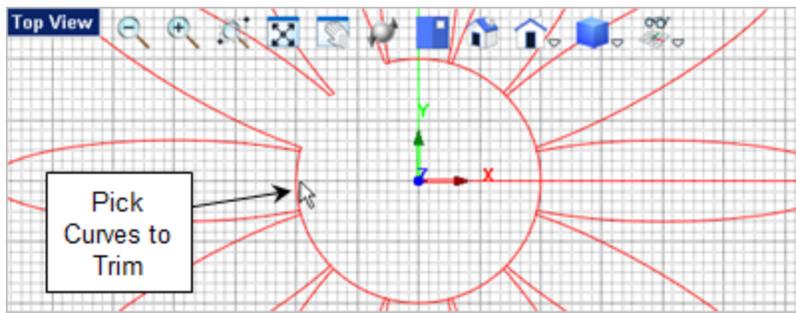
7. Open the [Layer Manager](#) and set the [Active Layer](#) to [Sections](#). Refer to the previous steps for displaying and using the [Layer Manager](#) .

Layer Name	Active	Visible	Objects	Color	Locked
Default	<input type="checkbox"/>	<input type="checkbox"/>	17		<input type="checkbox"/>
Curves	<input type="checkbox"/>	<input type="checkbox"/>	5		<input type="checkbox"/>
Sections	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	5		<input type="checkbox"/>
Stock	<input type="checkbox"/>	<input type="checkbox"/>	1		<input type="checkbox"/>

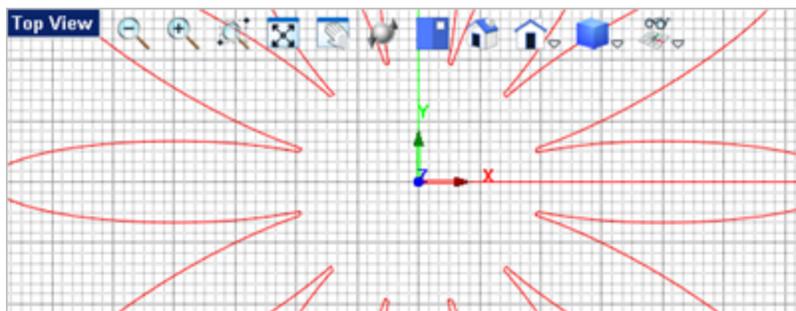
In the following steps we will trim our section curves and merge them.

8. From the [Top View](#) [Top View](#), scroll to zoom in on the center where the section curves from the body and the pedals overlap each other.

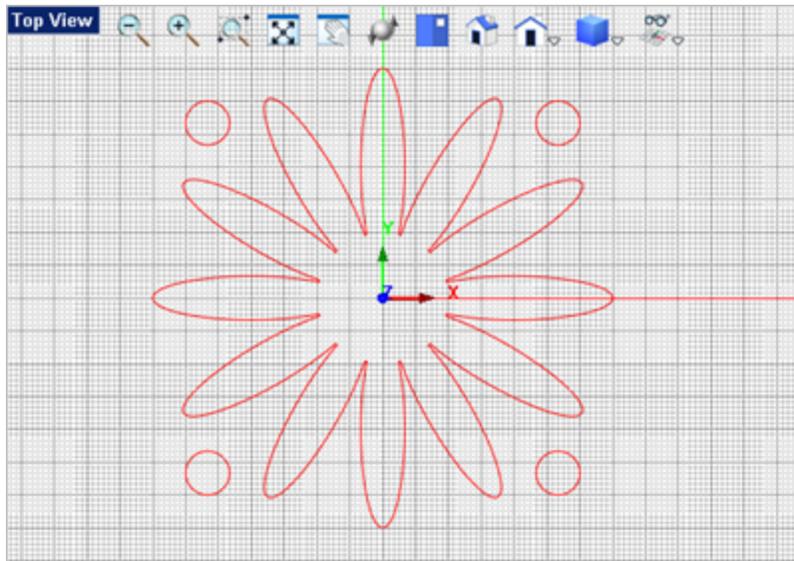
9. From the [Curve Modeling](#) [Curve Modeling](#) tab, select the [Trim](#)  command.
10. Pick on the curve where you want it to be trimmed away.



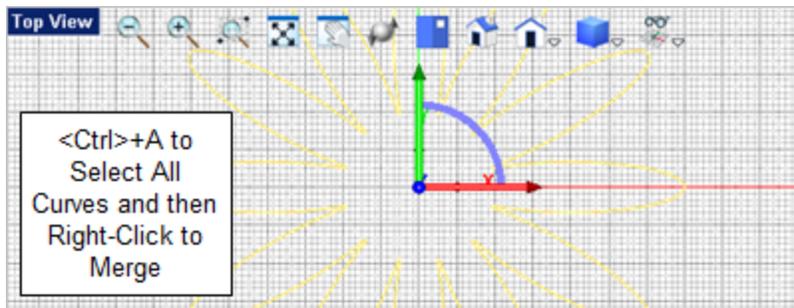
11. Press the <Enter> to repeat the Trim  command and then pick another location to trim.
12. Repeat this procedure until all of the curves are trimmed as shown below.

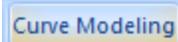


13. Now scroll to zoom out to see all of the curves.



14. Press **<Ctrl+A>** to select all of the curves.



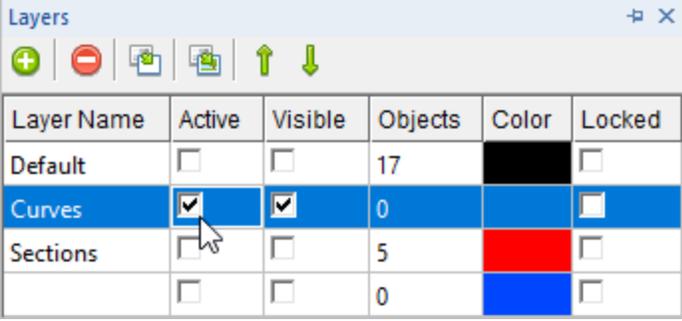
15. Now from the **Curve Modeling**  tab select the **Merge Curves**  command. This will merge all of the curves into one curve. This makes it easier to work with. Only curves that are joined end to end will be merged. Separate curves such as those created from the section of the outer spheres will remain separate curves.

11.8 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.

- 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](#).



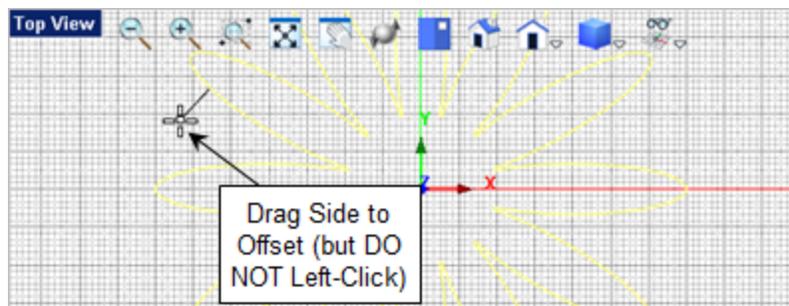
Layer Name	Active	Visible	Objects	Color	Locked
Default	<input type="checkbox"/>	<input type="checkbox"/>	17	Black	<input type="checkbox"/>
Curves	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0	Blue	<input type="checkbox"/>
Sections	<input type="checkbox"/>	<input type="checkbox"/>	5	Red	<input type="checkbox"/>
	<input type="checkbox"/>	<input type="checkbox"/>	0	Blue	<input type="checkbox"/>

- Now, select the main section curve, then go to the [Curve Modeling](#) [Curve Modeling](#)

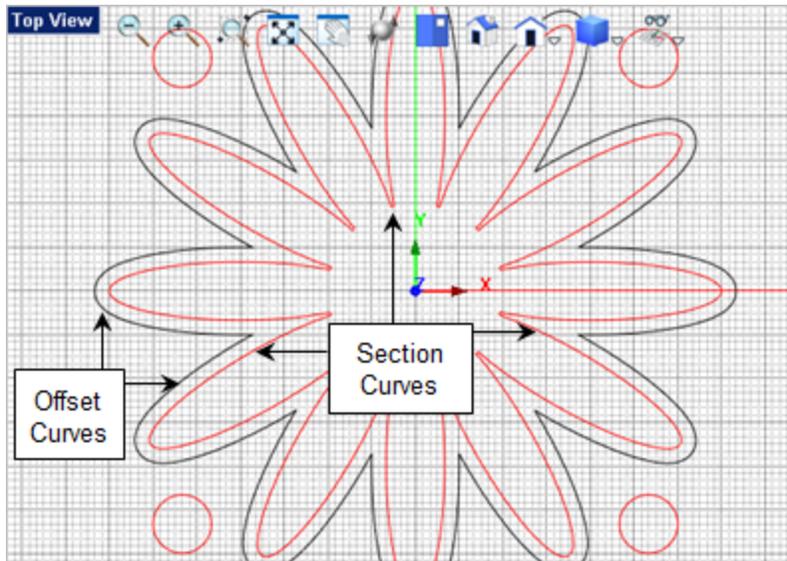


tab and select the [Offset](#) command.

- From the [Top View](#) [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!



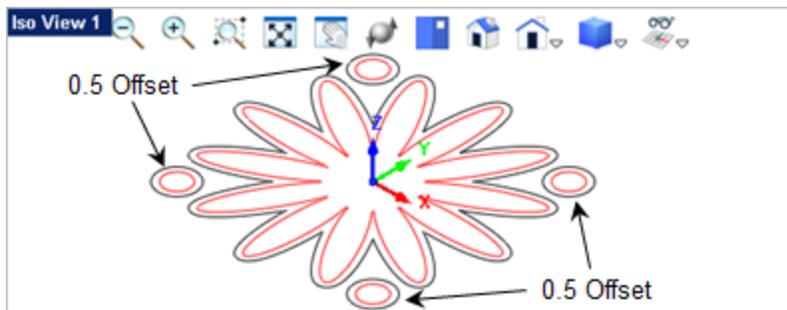
- Now with the curve offset direction displayed enter [0.5](#) in the command input bar and press the [<Enter>](#) key. The curve will be offset toward the outside by [0.5](#) inches.



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 **Offset** command to create offset curves for the other 3 remaining section curves.

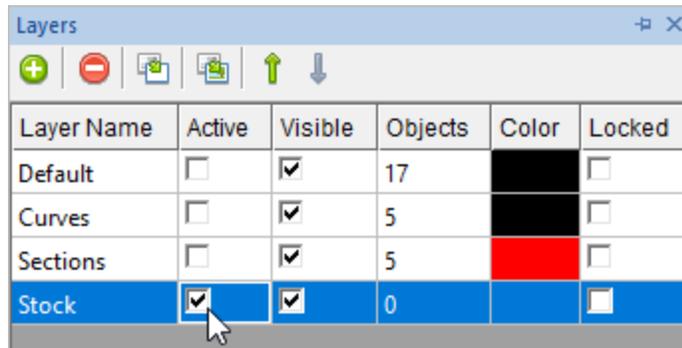


11.9 Create the Stock Boundary

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](#).

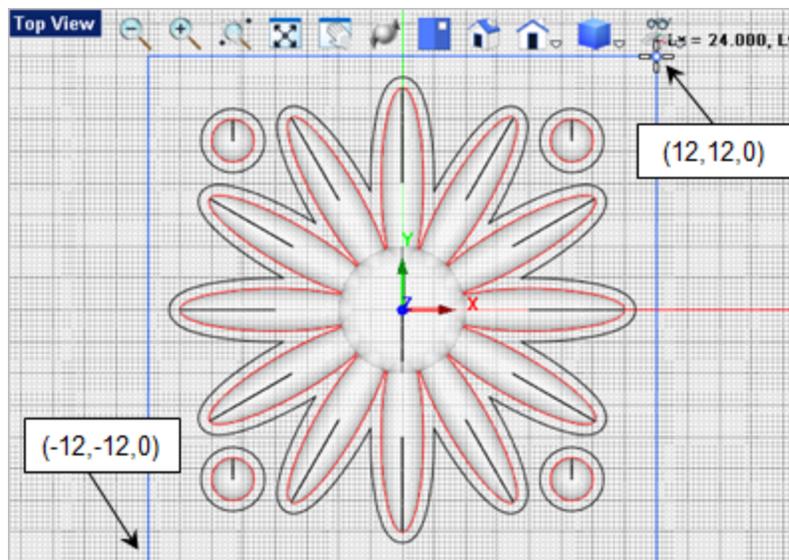


2. Activate the [Top View](#) [Top View](#) tab by selecting anywhere in the viewport.

3. From the [Curve Modeling](#) [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>`.

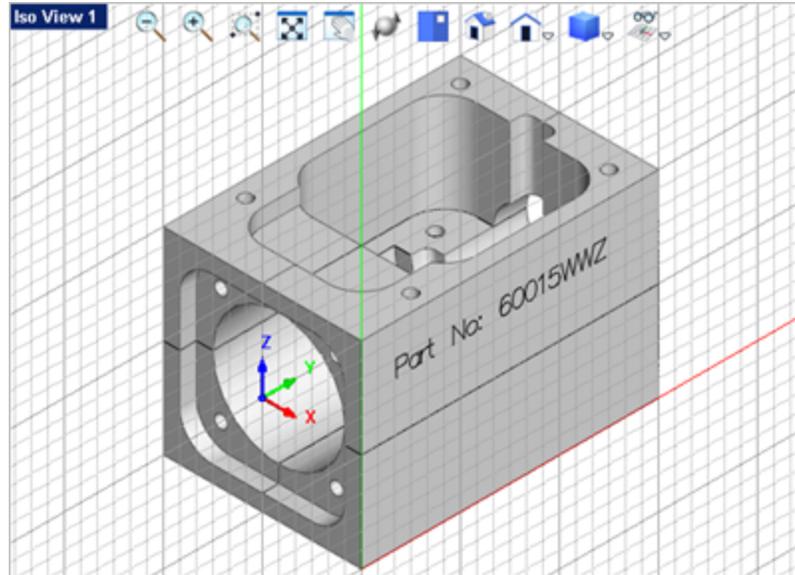


5. For the second corner of the rectangle, you can select the grid point shown below or enter `12,12,0` in the command input bar and press `<Enter>`.
6. Save the file as [Daisy_Decor_Completed.vcp](#).

Congratulations, you have completed this exercise!

#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.](#)

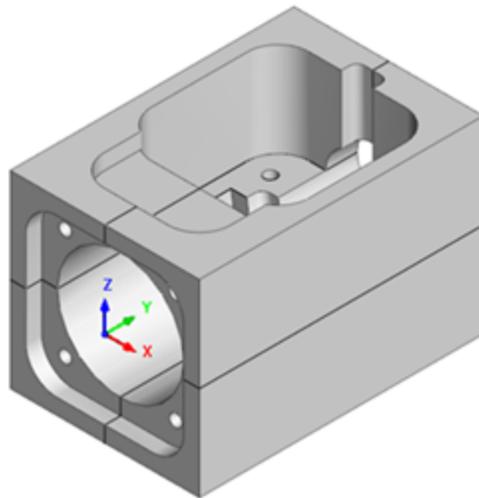
12.1 Orient the C-Plane



Orient the C-Plane

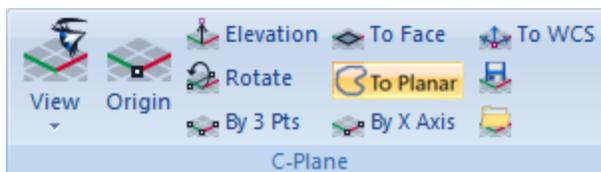
In this step we'll show you how to orient the [Construction Plane \(C-Plane\)](#).

1. First if you have completed and save the part model from [Exercise #5 Connector Block](#), open that part now. It is shown below:



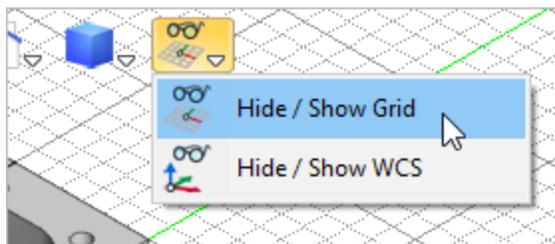
Connector Block

- Now select the **Modeling Aids** tab. On the very right end of the ribbon bar you will find the **C-Plane** pane.



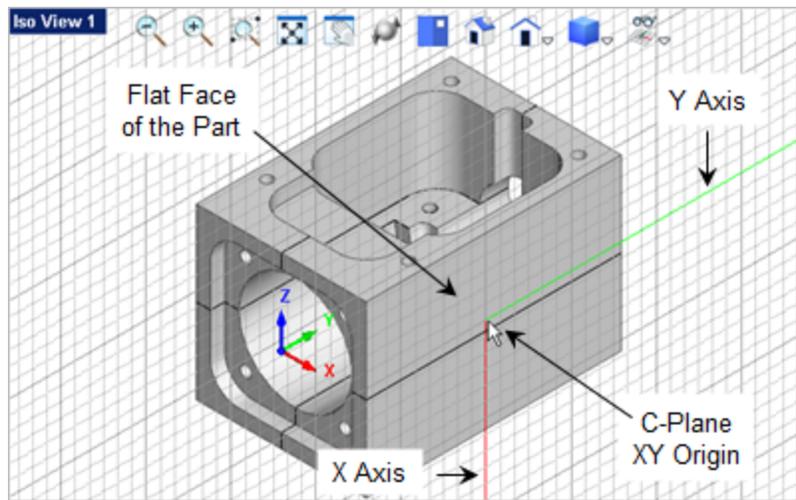
- From the **Modeling Aids** **Modeling Aids** tab select the **C-Plane to 3D Face** **To Face** command.
- The command prompt says: "Pick a flat area". Select the right side face of the part and you will see that the **C-Plane** snaps to that face.

If the **Grid** is not visible, select **Hide / 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.

- 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.

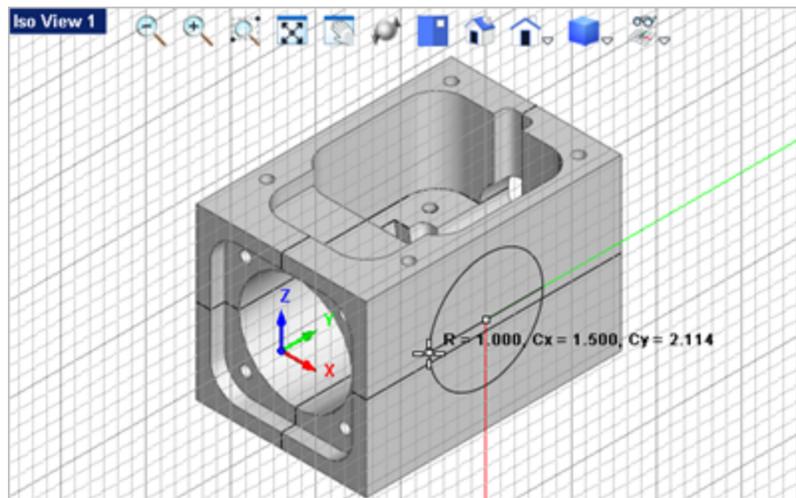


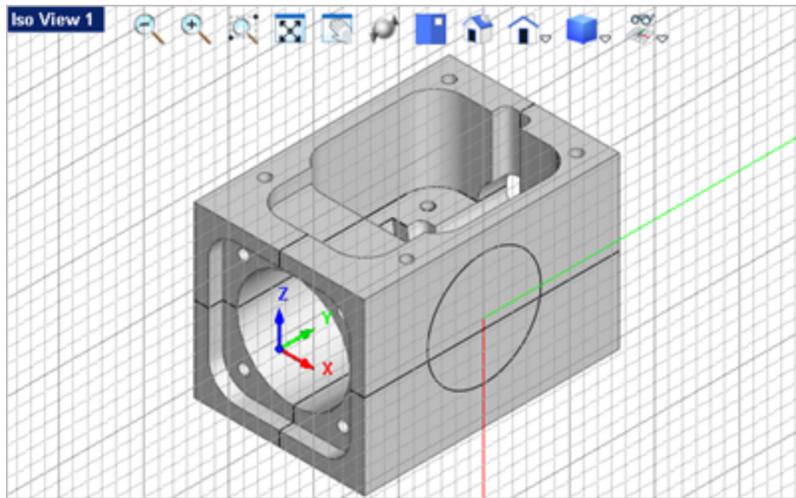
6. You can now draw on the C-Plane (see [Text on a Part Face](#) for more information).



From the **Curve Modeling** **Curve Modeling** tab select the **Circle** **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.





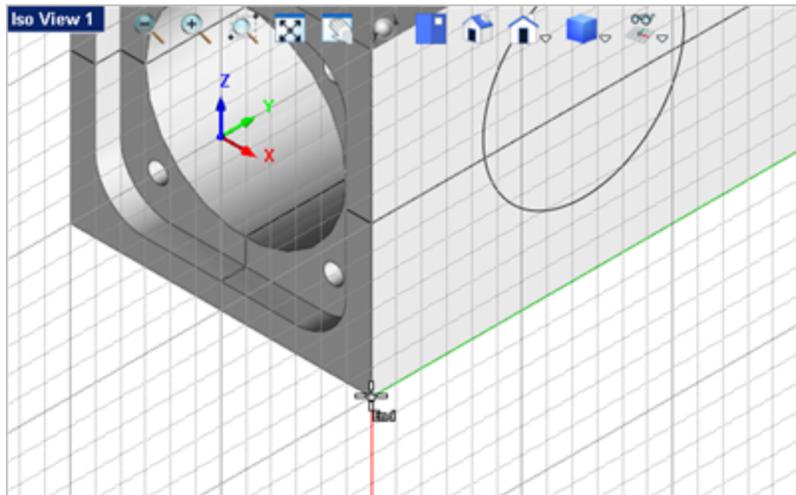
- 9. You can also move the C-Plane origin. From the Modeling Aids **Modeling Aids** tab



select the C-Plane Origin command.

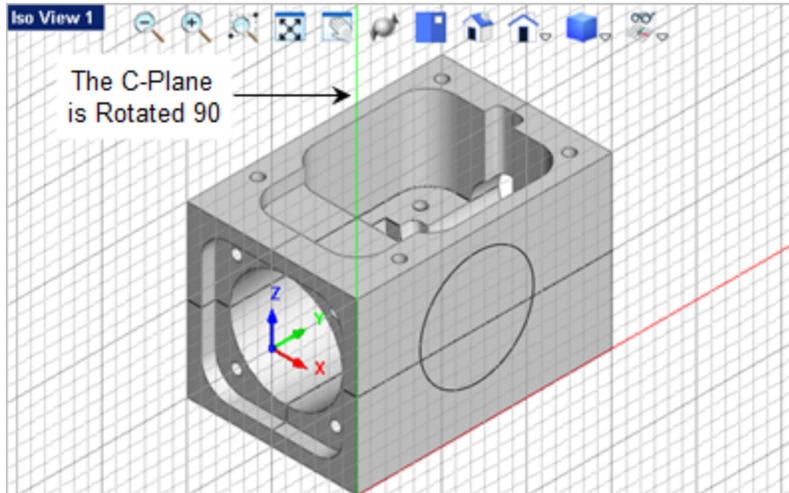
- 10. The command prompt says: "Pick origin point or enter coordinates x,y and Z" Enable the End Point Snap  from the Status Toolbar and then select the bottom right end point of the face.

You will see that the C-Plane origin and the Grid is moved to this point.

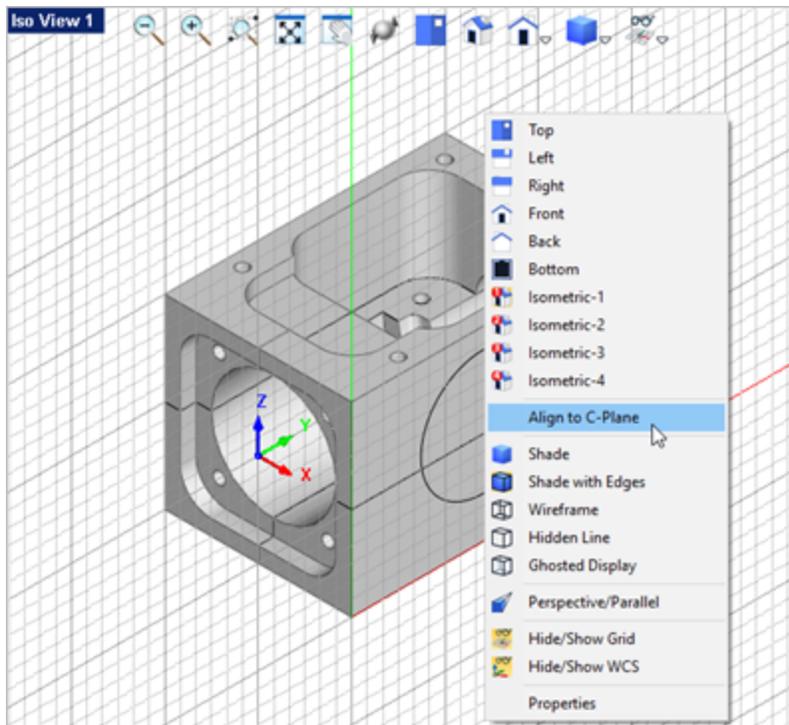


- 11. If you need the X Axis of the C-Plane aligned horizontally across the bottom of the part you can select the Rotate C-Plane about Axis  Rotate command. It is also located on the Modeling Aids **Modeling Aids** tab.

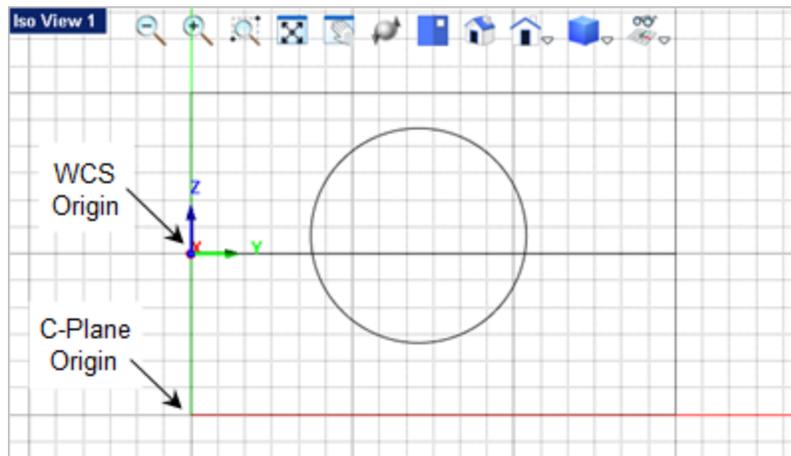
12. The command prompt says: "Enter axis to rotate about and angle to rotate by (eg: X,45)" We want to rotate the Z Axis 90 degrees so enter Z,90 in the command window and press <Enter>.



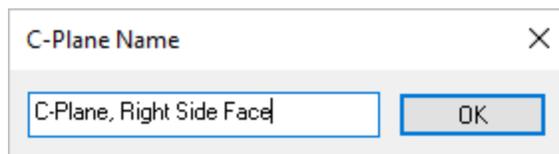
13. Now what if you want the view orientation aligned with the C-Plane?
14. Just right-click anywhere in the view to display the View pop-up menu and select the **Align to C-Plane** command. This menu contains many **Viewport** related commands.



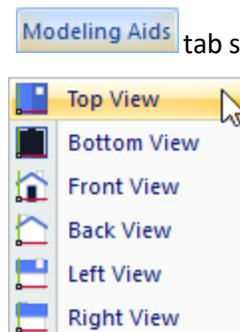
- 15. With the C-Plane aligned with the active Viewport pay close attention to where the origin of the C-Plane is:

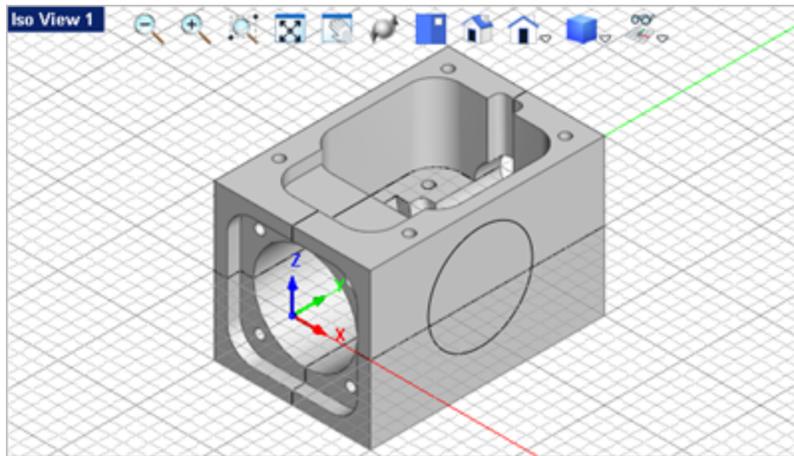


- 16. Now you may want to come back to the C-Plane later so let's save it. From the Modeling Aids **Modeling Aids** tab select the Save C-Plane  command.
- 17. In the C-Plane Name dialog enter a unique name for the active C-Plane and then pick OK.

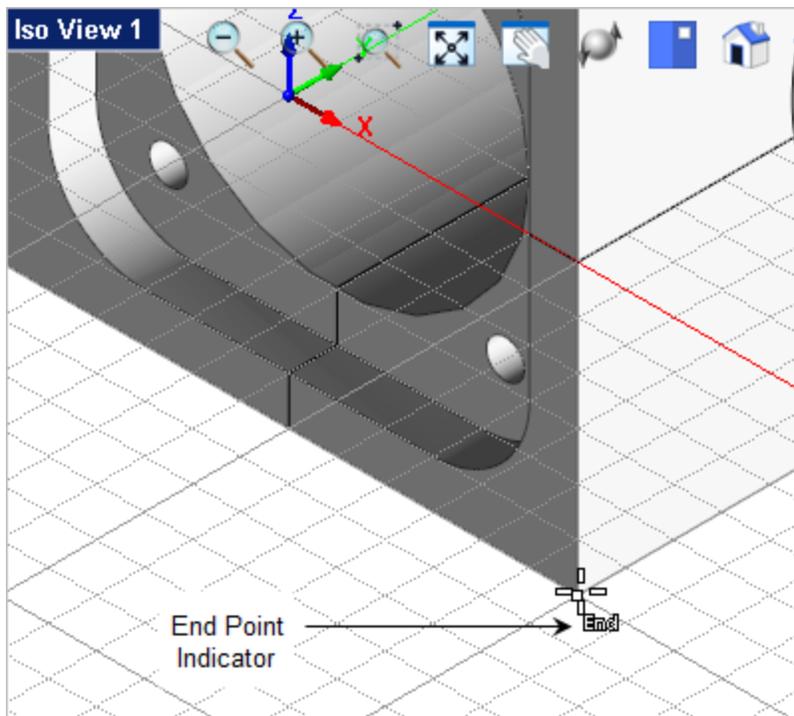


- 18. The active C-Plane is now saved with the part file.
- 19. Now let's change the C-Plane back to the default top view. From the Modeling Aids  tab select the View  menu and select Top View.



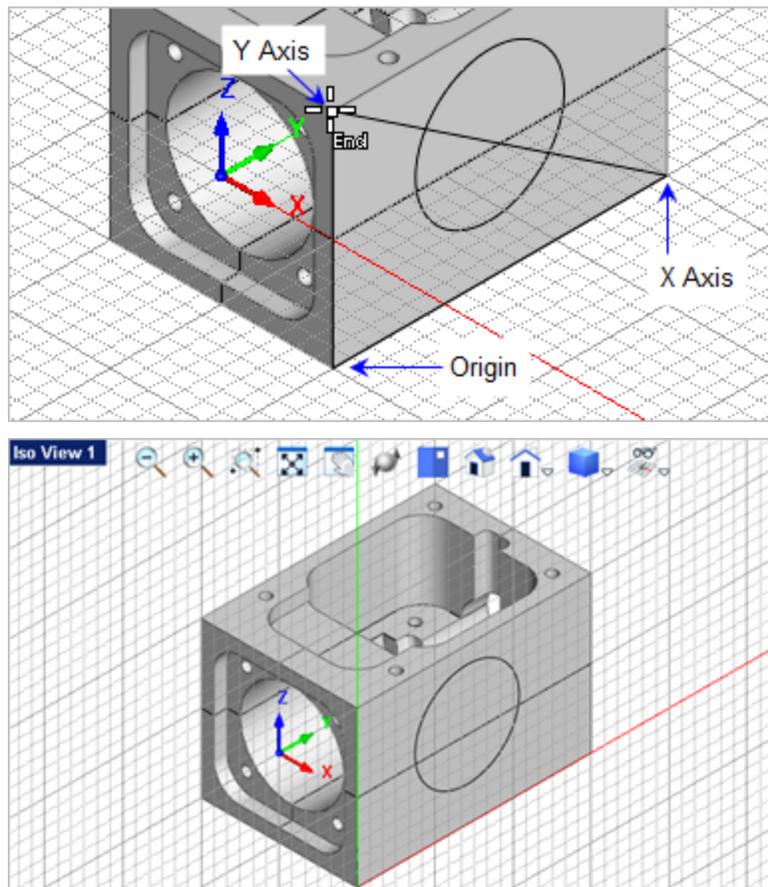


20. Here is an easier way to align the **C-Plane** to the face AND align the **XY Axis** of the **C-Plane** at the same time. From the **Modeling Aids** **Modeling Aids** tab select the **C-Plane by 3 Points** **By 3 Pts** command.
21. The first point selected will be the origin, so make sure the **End 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.

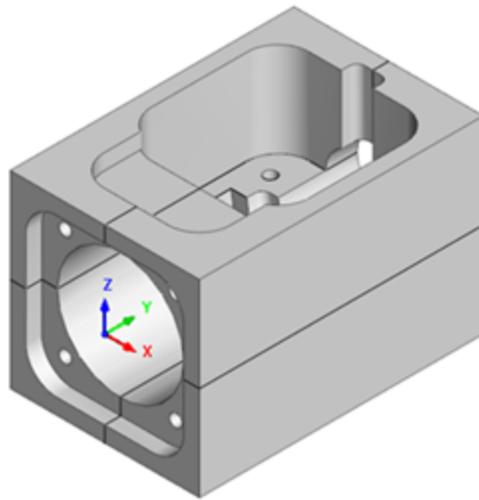


12.2 Text on a Part Face

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:

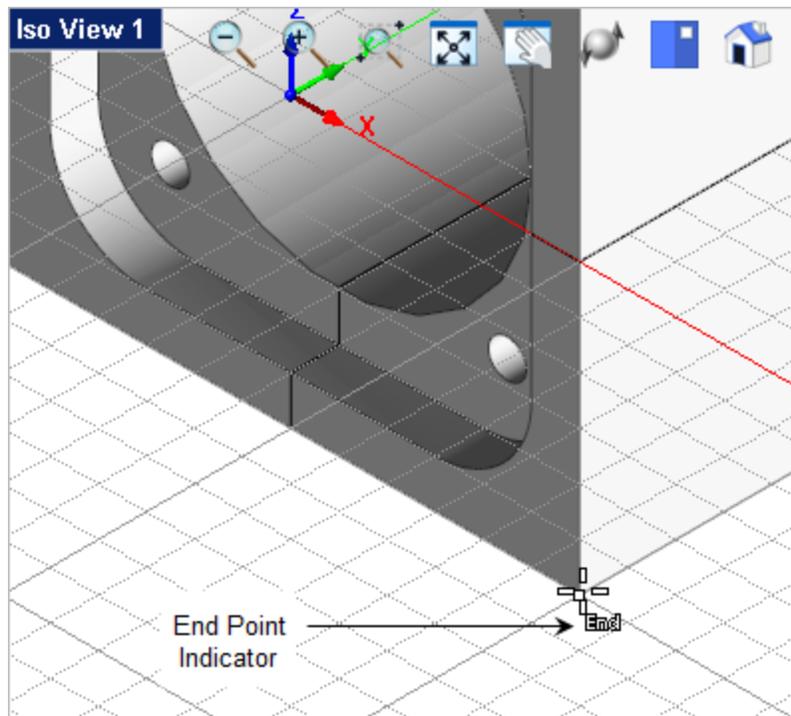


Connector Block

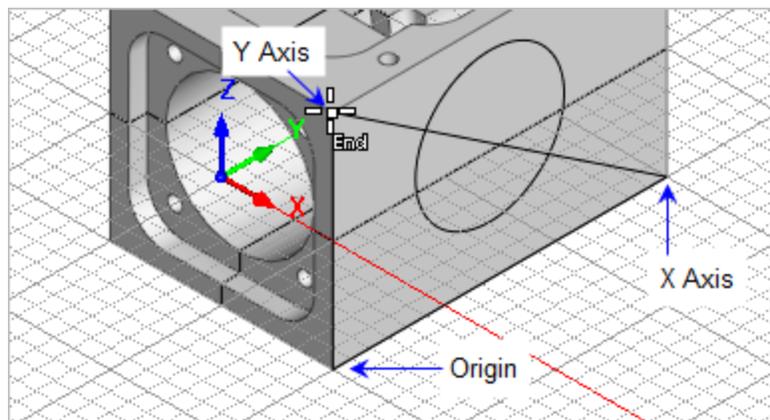
2. Now select the **Modeling Aids** **Modeling Aids** tab. On the very right end of the ribbon bar you will find the **C-Plane** pane.

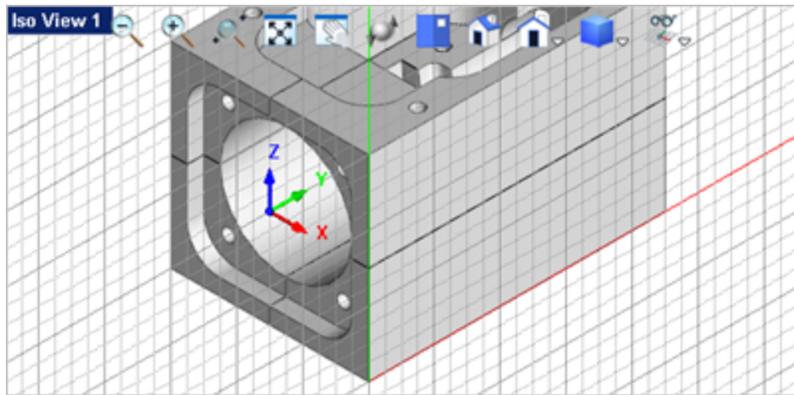


3. From the **Modeling Aids** **Modeling Aids** tab select the **C-Plane by 3 Points** **By 3 Pts** command.
4. The first point selected will be the origin, so make sure the **End Point Snap**  is toggled on.
5. Then select the bottom left corner end point of the right side face of the part. **Note:** Do Not pick until you see the "End" indicated at the cursor location.

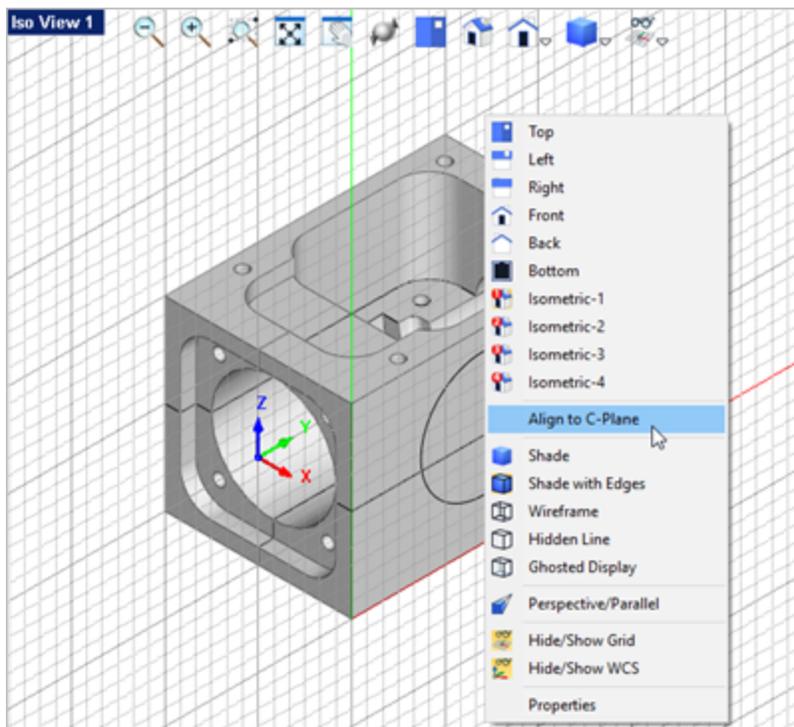


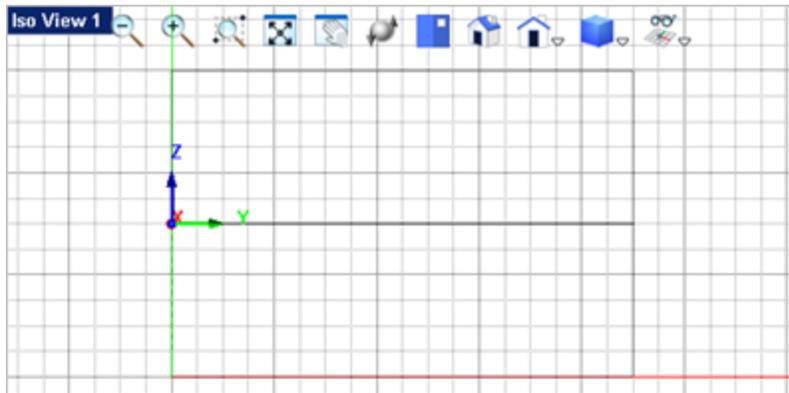
6. For the **X Axis** point select the end point at the back end of the part on the right side face.
7. For the **Y Axis** point select the end point at the top of the part on the right side face.



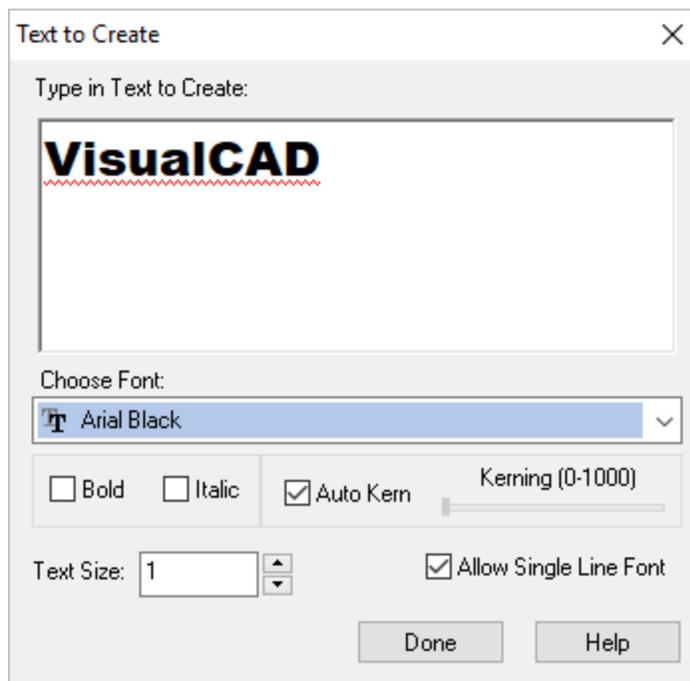


8. Now right-click anywhere in the view to display the **Viewpoint** pop-up menu and select the **Align to C-Plane** command. This menu contains the **Viewport** related commands.

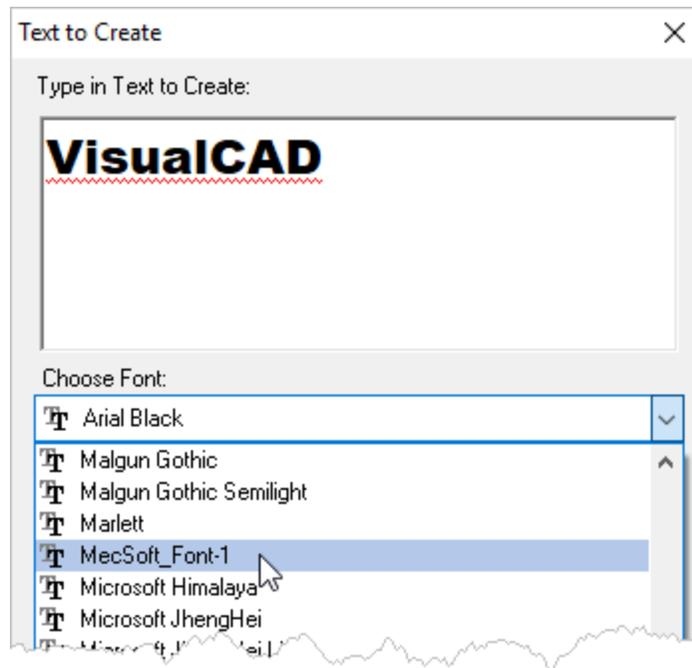




9. If you want to simply label your part you can add dimension text that will lie on the C-Plane. See [Drawing & Dimensioning](#) for that exercise.
10. In this exercise we will create text curves that you can engrave with the 2-1/2 Engraving toolpath strategy, go to the Curve Modeling **Curve Modeling** tab and select the Create Text **T** command.
11. This will display the Text to Create dialog.

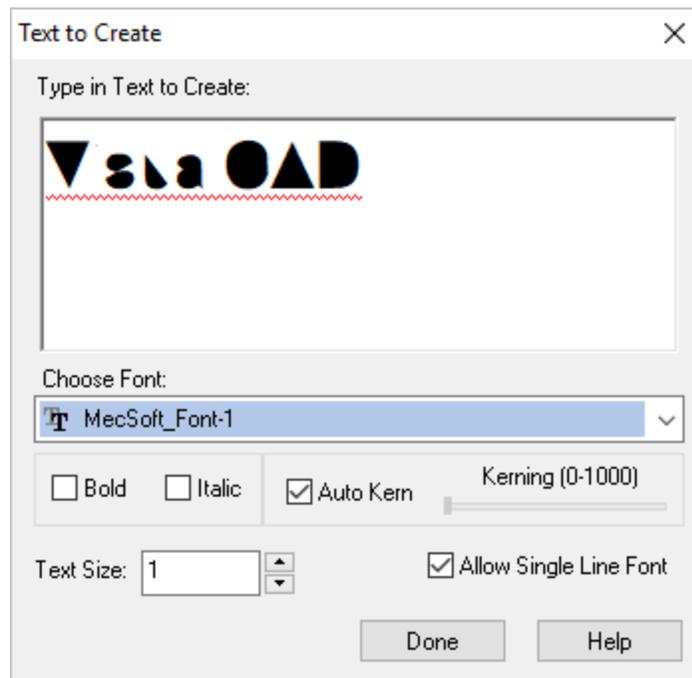


12. For engraving text you may want to use a single stroke font. From the Font drop-down menu select the text font named [MecSoft_Font-1](#).



💡 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.



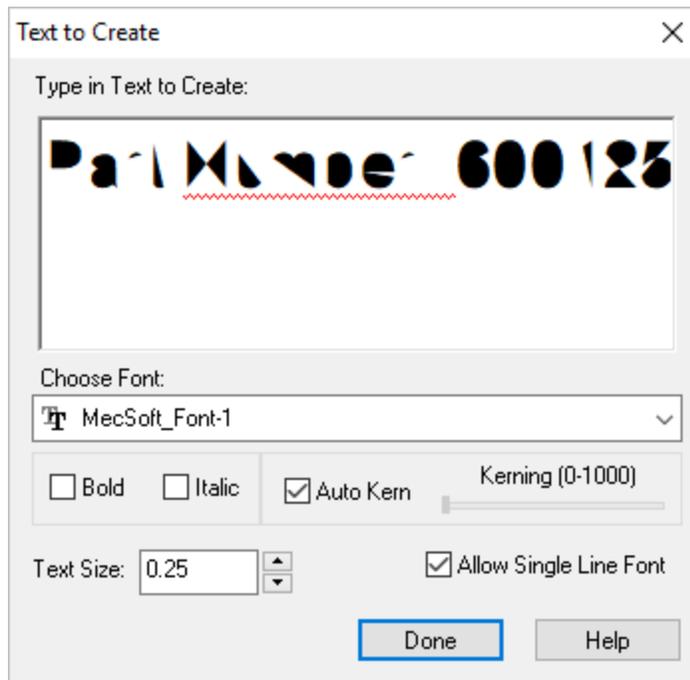
14. Make the following adjustments to the dialog:

Allow Single Line Font: Checked
 Text Size: 0.25

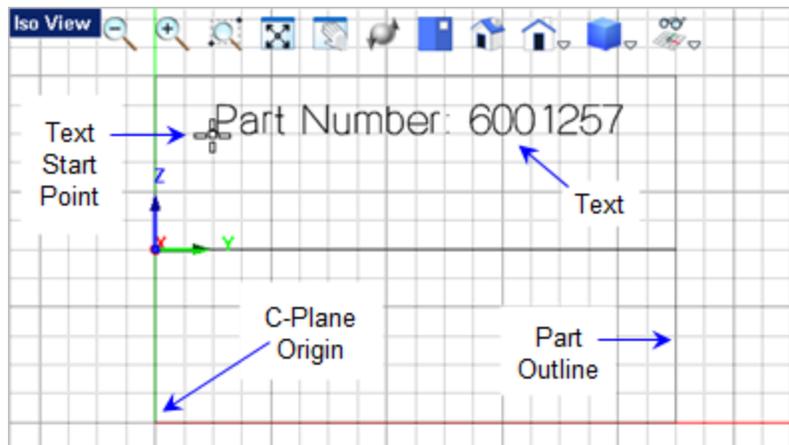
15. Now place in the **Type in Text to Create** window, replace the text **VisualCAD** with the following text:

Part Number: 6001257

16. The dialog should look like this:

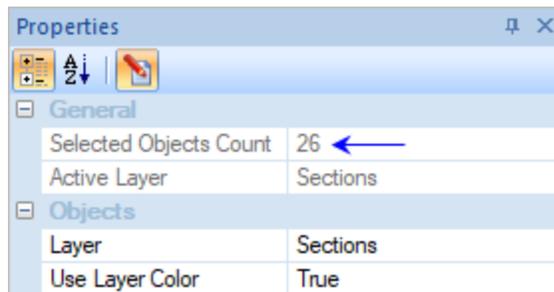


17. Now pick the **Done** button. The dialog will close and command prompt says: **Enter start point of text.**
18. You will see the text as you move the cursor around the **Viewport**.
19. Enable the **Grid Snap**  and then select the grid point to position the text, or **0.5,2.5** in the command window 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.



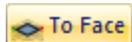
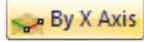
12.3 Other C-Plane Commands

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
	<p>Immediately sets the orientation of the C-Plane to be parallel to the plane of the currently active view. The origin remains the same.</p>
	<p>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.</p>
	<p>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.</p>
	<p>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'.</p>
	<p>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.</p>
	<p>The C-Plane will be oriented to a selected flat plane of a geometry object. The pick point will become the C-Plane origin.</p>
	<p>Two points are entered to re-orient the x-axis of the C-Plane:</p>
	<p>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.</p>
	<p>This option saves the C-Plane orientation from the currently active view. A name must be provided for later reference when loading.</p>
	<p>This option allows the retrieval of a saved C-Plane into the currently active view. A list of previously saved C-Planes is</p>

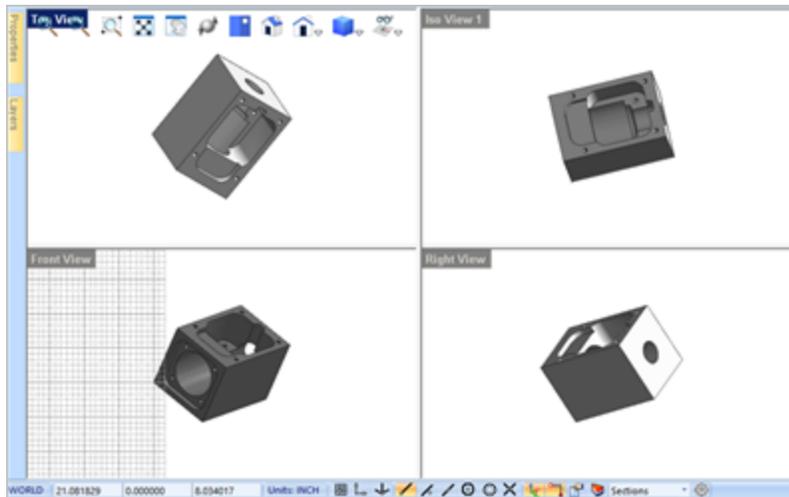
presented. These saved [C-Planes](#) may also be deleted in this option or renamed.

12.4 Orient the Part

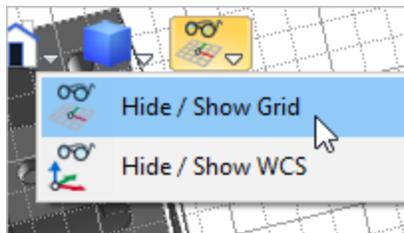
Orient the Part

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:

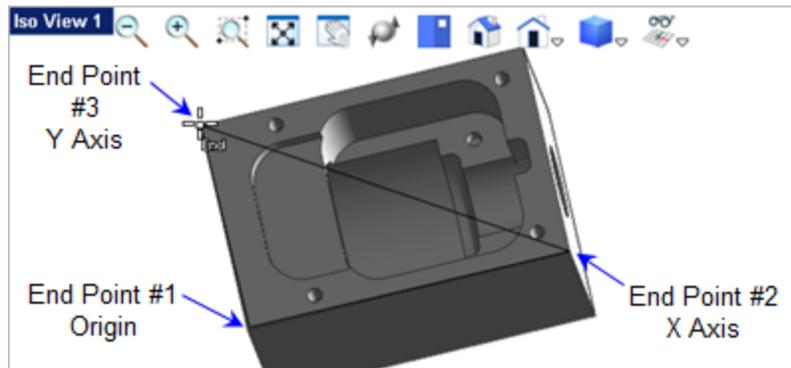


1. The first step toward to orienting this part is to define the [C-Plane](#). For more about this see the topics [Orient the C-Plane](#) and [Text on a Part Face](#) in this exercise.
2. First make sure the [Grid](#) is visible. You can toggle the [Grid](#) on/off by selecting [Hide / Show Grid](#) from the [View Toolbar](#).

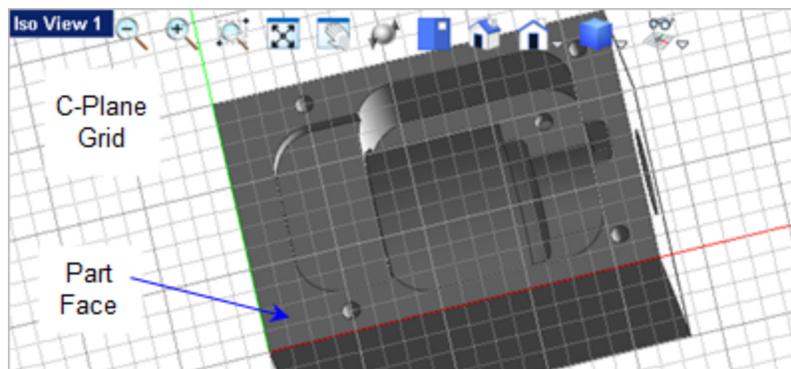


3. Now activate the view that allows you to see the planar face that you want to use to orient the part by. In this example it is [Iso View 1](#) **Iso View 1**.

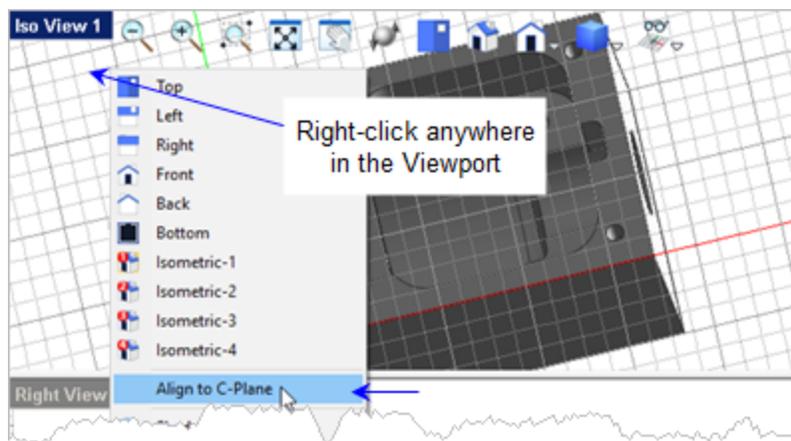
4. Go to the **Modeling Aids** **Modeling Aids** tab and select the **C-Plane by 3 Points** **By 3 Pts** command.
5. Activate the **End Point Snap**  from the **Status Toolbar** and select the end points in the order shown below:



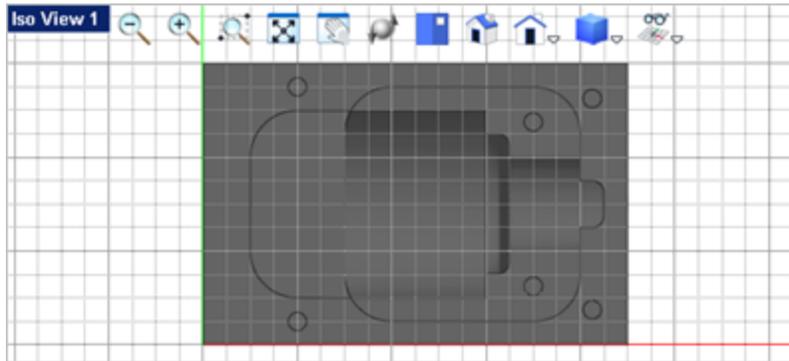
The **C-Plane** will orient to the 3 points selected.



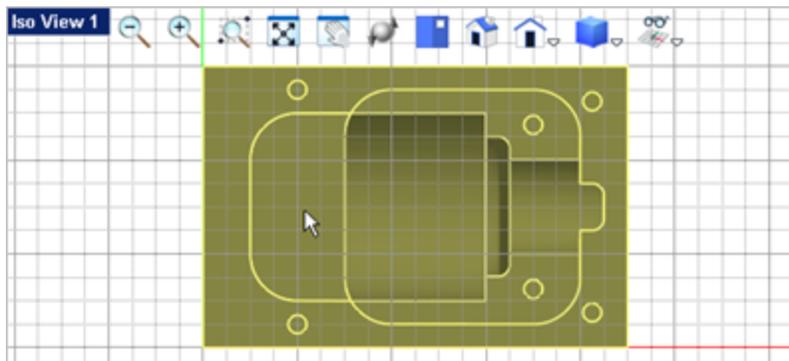
6. Now right-click anywhere in the **Viewport** (do not select any geometry) and select **Align to C-Plane**. This will align **Iso View 1** **Iso View 1** parallel to the **C-Plane**.



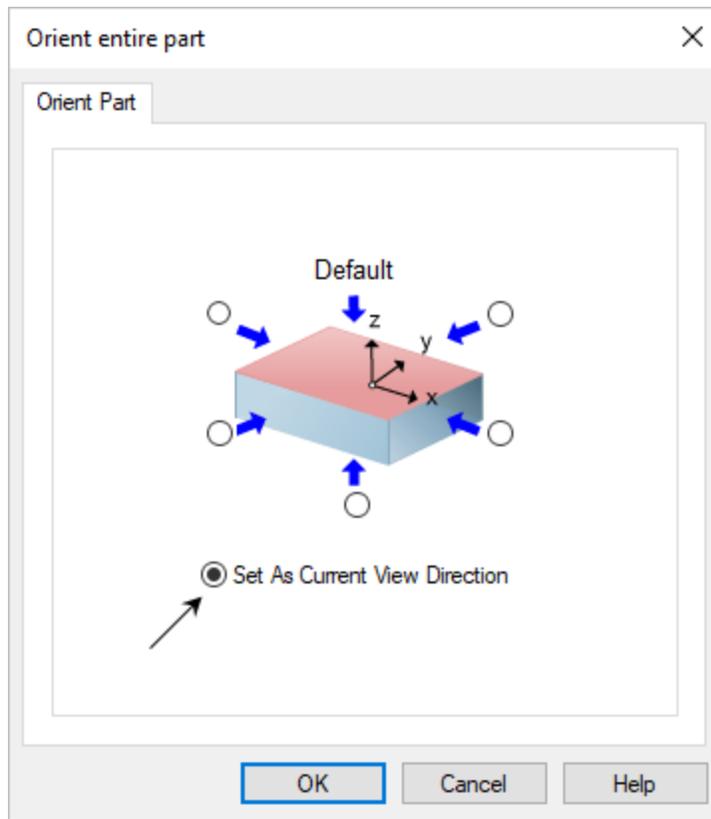
The C-Plane will align to the Viewport.



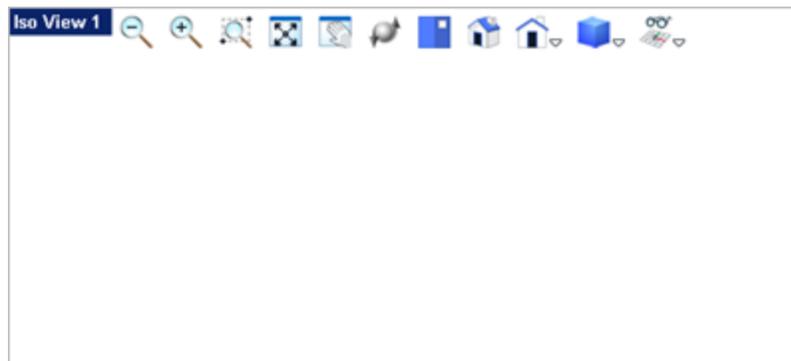
7. Now we can orient the part. Select the part and go to the **Modify / Transform**



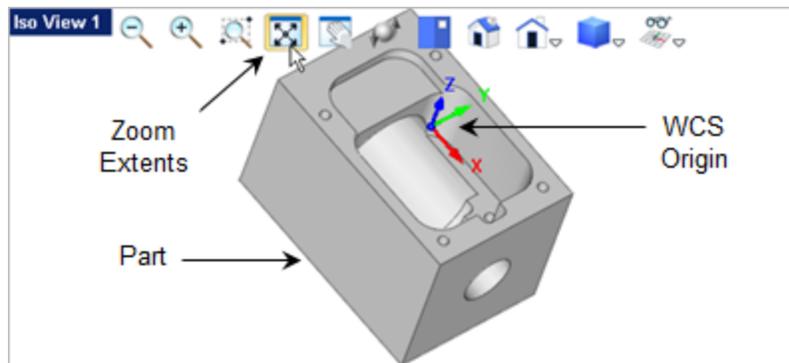
8. This will display the **Orient entire part** dialog. Pick the option to **Set as current view direction**.



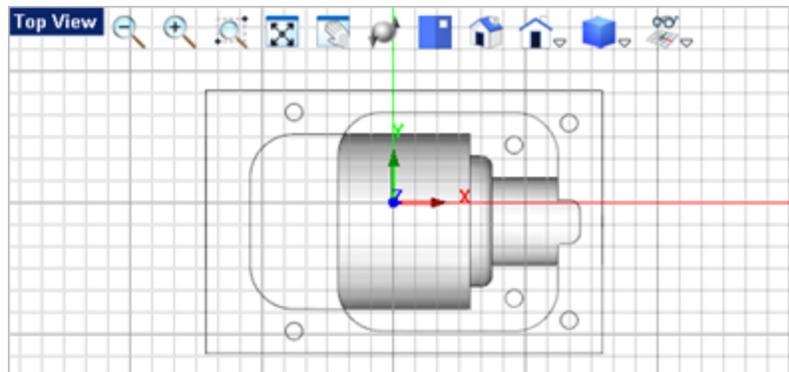
- 9. Pick **OK** and the part appears to disappear from the **Viewport**.



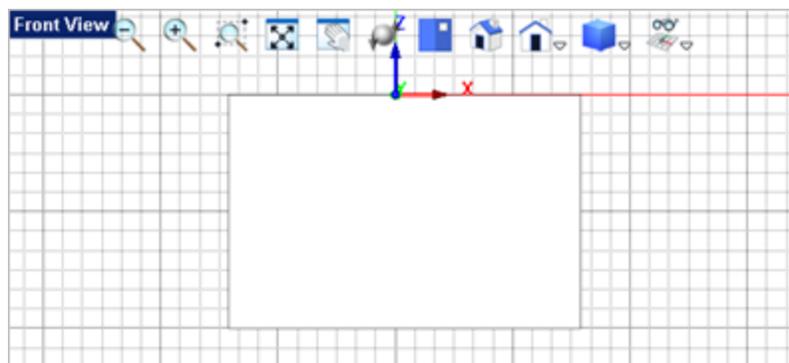
- 10. Now pick the **Fit View**  icon from the **View Toolbar**. The part will appear with the **WCS** oriented in/on the center of the face that we used to align the **C-Plane**

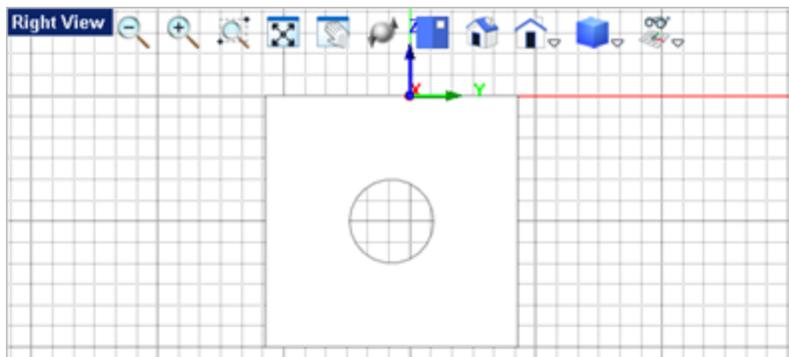


11. Now activate the **Top View** **Top View** and pick the **Fit View**  icon from the **View Toolbar**.

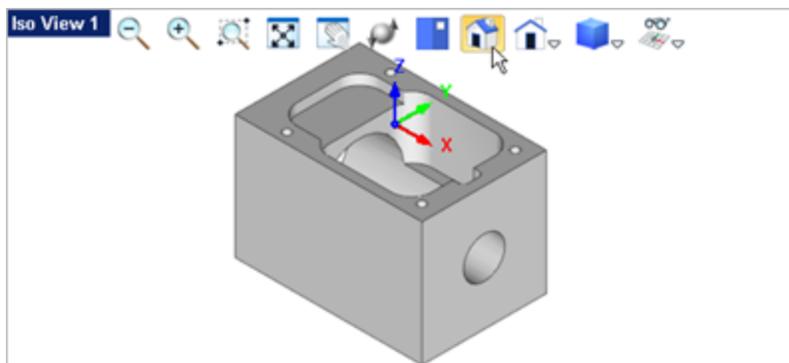


12. Repeat this for the **Front View** **Front View** and the **Right View** **Right View**. Remember that you have to activate the **Viewport** before the **View Toolbar** will display. Left-click anywhere within the **Viewport** to activate it.



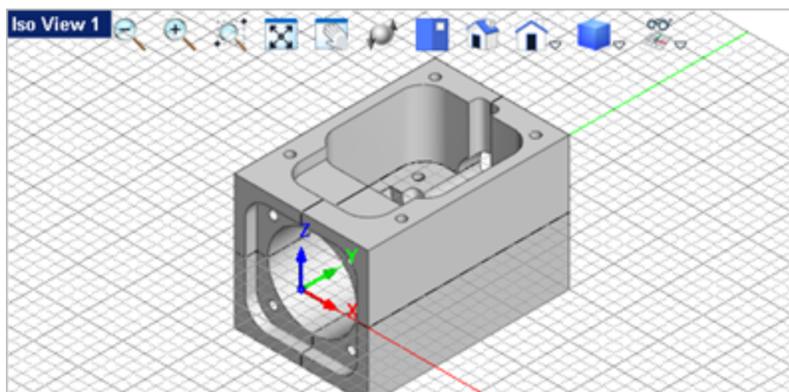


- 13. Activate **Iso View 1** again and this time select the **Iso View** icon from the **View Toolbar** to orient this **Viewport**.

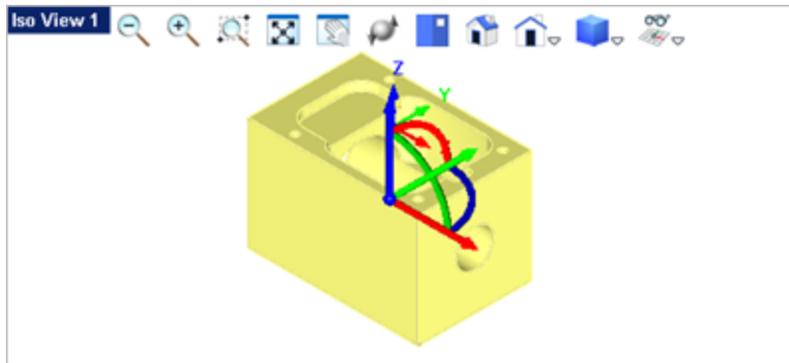


Great! You have the part oriented orthographically (i.e., **Top**, **Front**, **Right**).

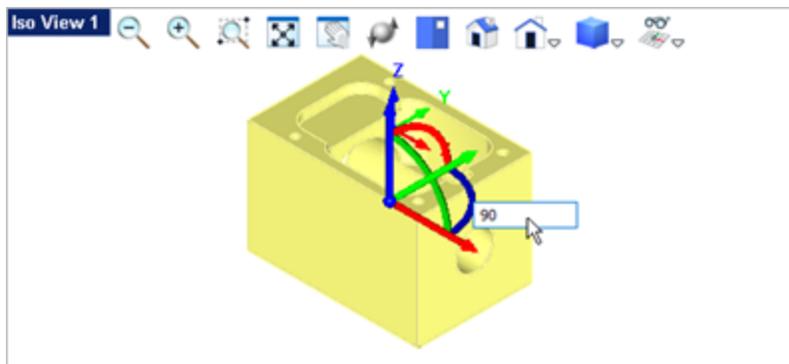
- 14. However, the part is still not oriented like the original part as shown below. You see that the length of the part was oriented along the **Y Axis** and the **WCS** was positioned at the center of the front view. Let's show you some additional commands to fix that.



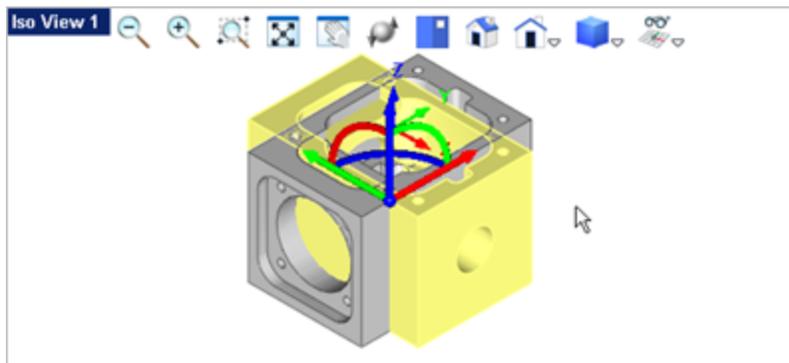
- 15. Activate **Iso View 1** and then from the **Status Toolbar**, toggle the **Graphic Manipulator** **On**.
- 16. Now select the part to see the **Graphic Manipulator** displayed.

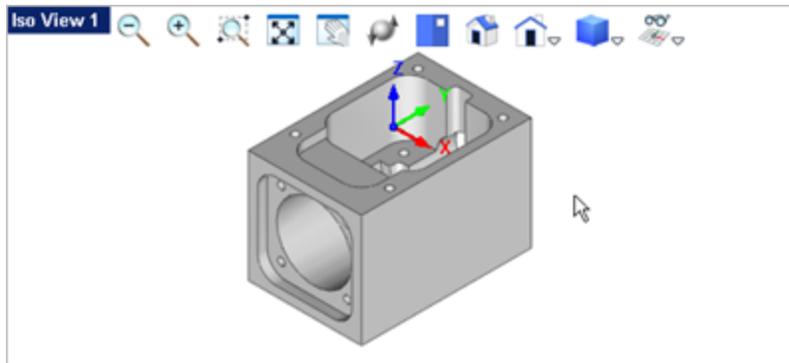


17. Pick the Z Rotation arc of the Graphic Manipulator  to display the input window, enter 90 and then press <Enter>.



18. Now you see the part is rotated so press <Enter> to accept the rotation.

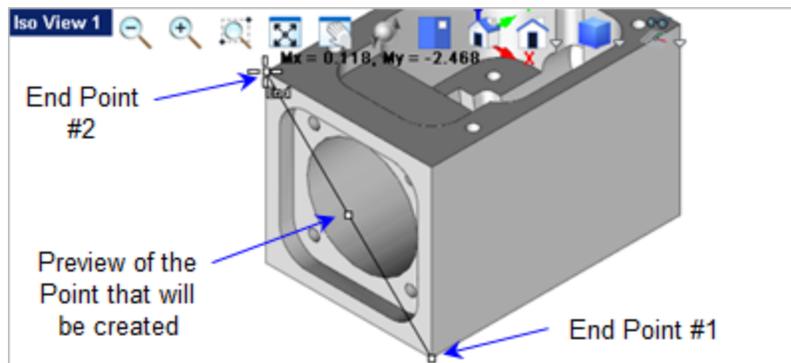




Great where almost there!

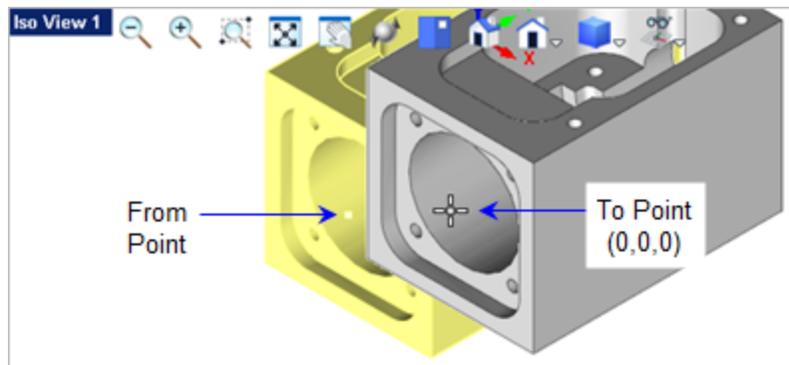
To move the **WCS** to the original location which was centered on the front face of the part we need to create a point there from.

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.



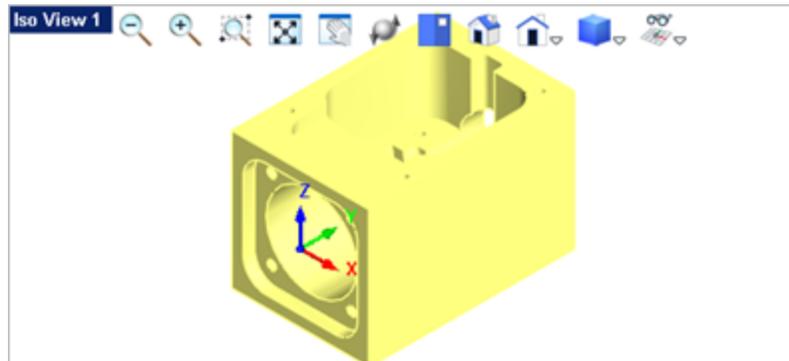
23. You can now toggle the **Graphic Manipulator**  off.
24. With the point created, window select the entire part (and point) or simply press **<Ctrl+A>**.

25. From the **Modify / Transform** **Transform**  tab select the **Move by mouse** 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` and then press **<Enter>**.

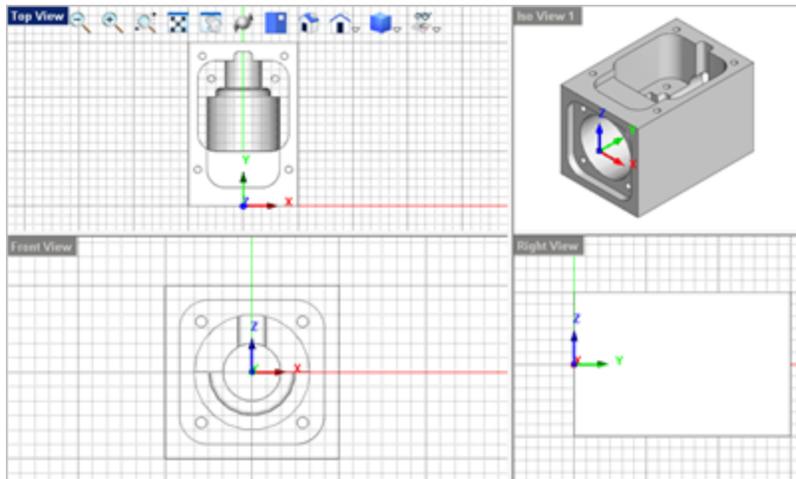
You will see that the part moved to the **WCS** origin. Now press **<Enter>** again to accept the move.



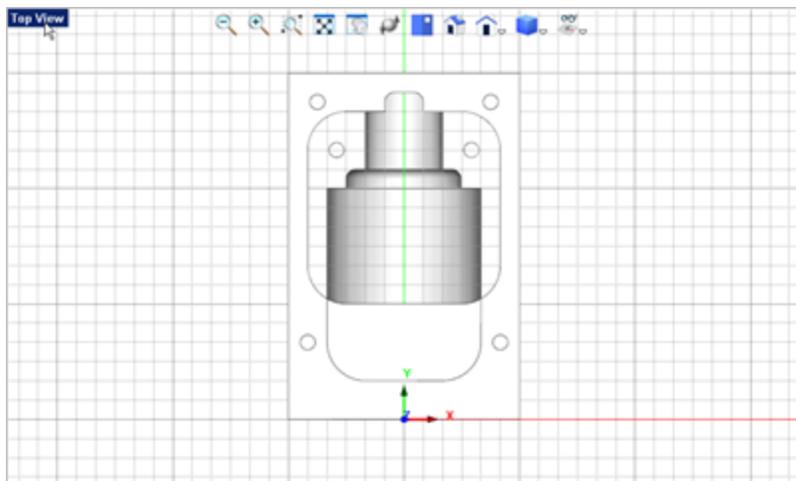
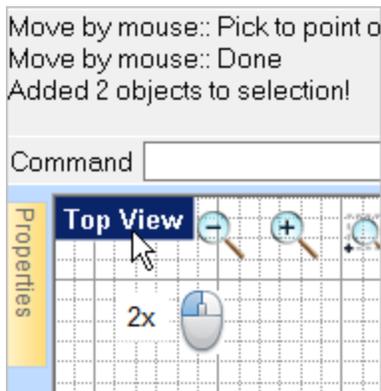
28. Now the part is aligned correctly except for the **C-Plane**. Previously the **C-Plane** was aligned to the **XY** plane of **WCS**.

Here is a neat trick to fix that:

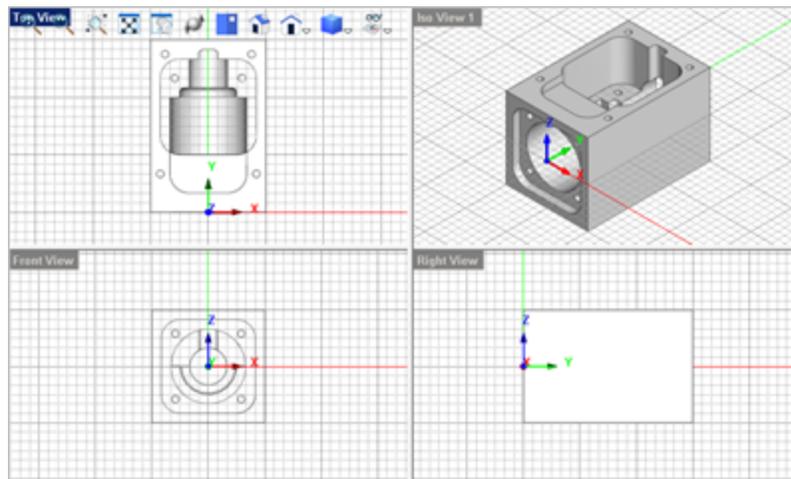
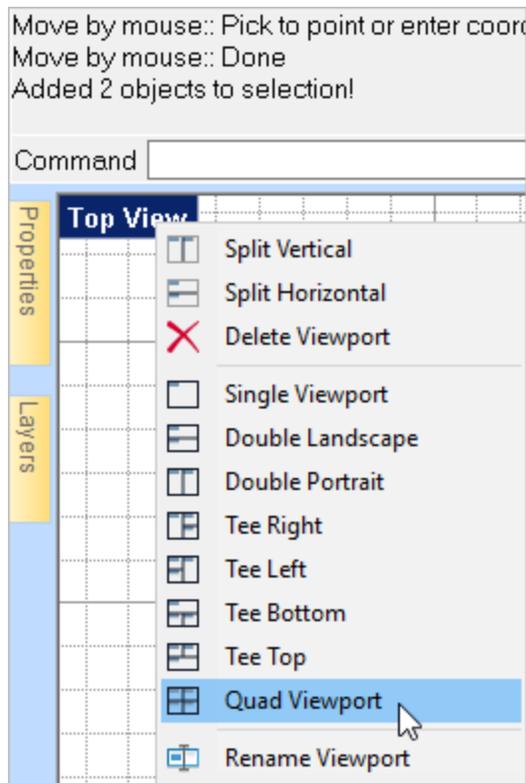
29. Activate the **Top View** .



- 30. Now double-left-click on the **Top View** **Top View** tab. This will change the display to one Viewport set to the Top View.



- 31. Now right-click on the **Top View** **Top View** tab and select Quad Viewport.



You see that **Iso View 1** **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**.

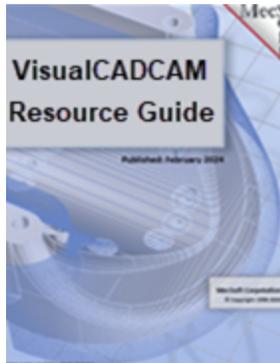
Congratulations on completing this exercise!

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