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Welcome to VisualCAD!



VisualCAD 2025

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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.



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 cplane
- 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

Welcome to VisualCAD/CAM

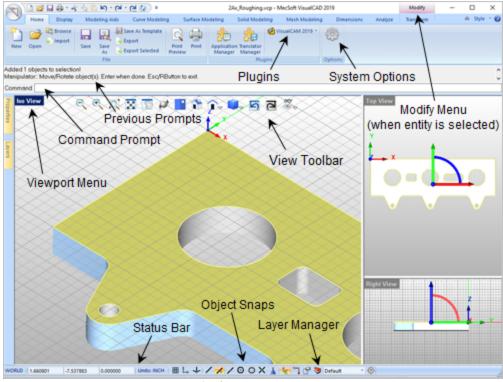
The VisualCAD/CAM Screen Layout

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

Welcome to VisualCAD/CAM

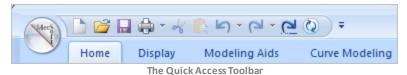
VisualCAD/CAM Features Overview

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.



Quick Access Toolbar's Default Commands



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



Customize Quick Access Toolbar

Customize Keyboard

Customize the Display

Preset < Alt> Accelerator Keys

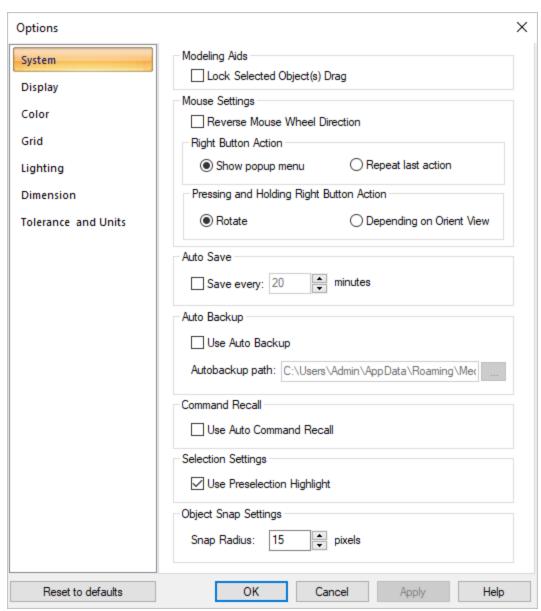
The VisualCAD/CAM Screen Layout

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

System

Display

Color

Grid

Lighting

Dimension

Tolerance and Units

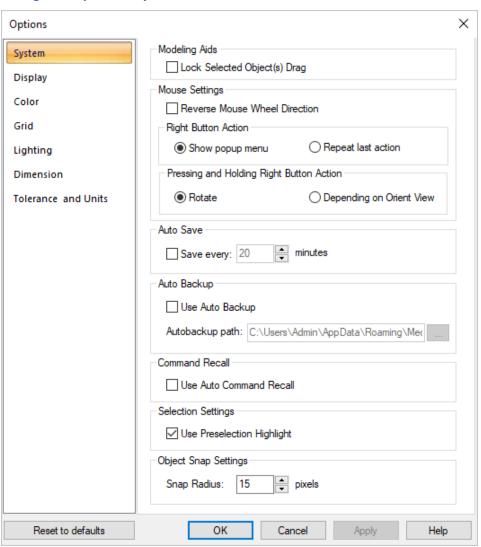
Home Ribbon Bar

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

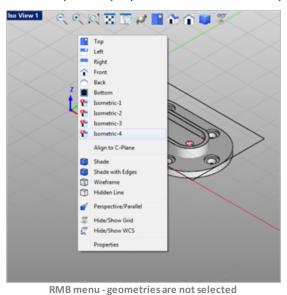
The following Mouse options are supported:

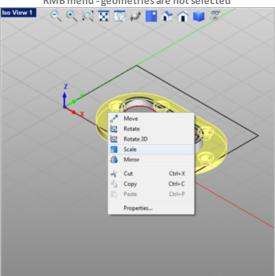
Reverse Mouse Wheel Direction

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 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.

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

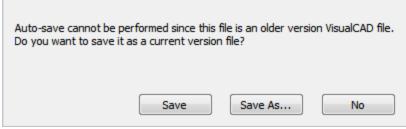
Under the AutoSave function, check the box next to 'Save every xx minutes' to activate the AutoSave function. The time interval can be entered in minutes.

AutoSave saves the file under:

<a href="mailt

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

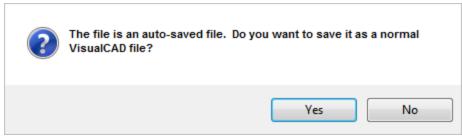
In the event of a computer crash or if VisualCAD/CAM is terminated abruptly by ending the process from task manager, VisualCAD/CAM will recover your file if AutoSave was active and your part file was automatically saved at least once.

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.

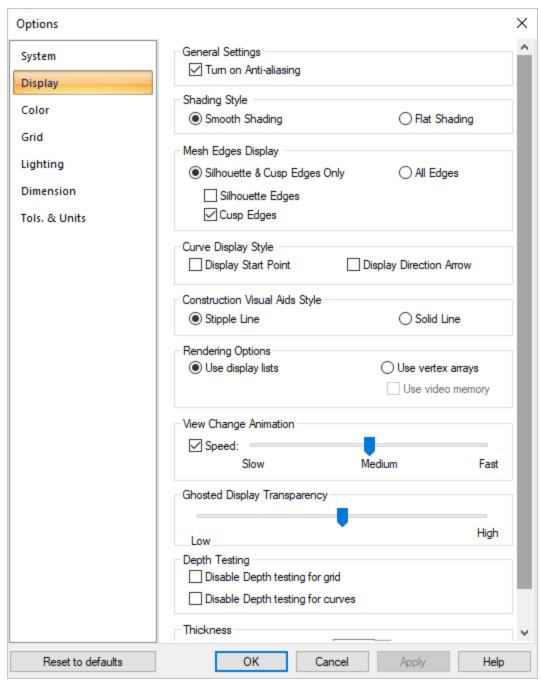
Related Topics

Options

1.5.2 Display

Controls how geometry is displayed. Select from the following:

Dialog Box: Options > Display



Dialog Box: Options > Display

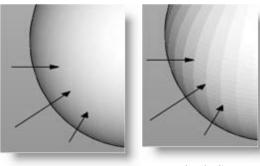
General Settings

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.

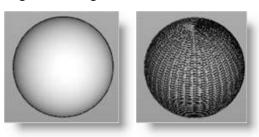


Smooth Shading Flat Shading

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

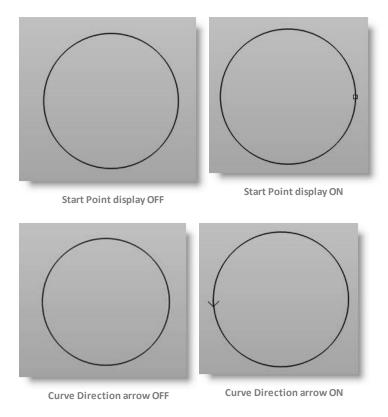
This changes how smooth surfaces are displayed. Since all surfaces are internally represented as flat facets, these facets join up along edges. On smooth surfaces, these edges are not sharp and are not normally displayed. An example of a sharp edge is the edge of a cube.



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

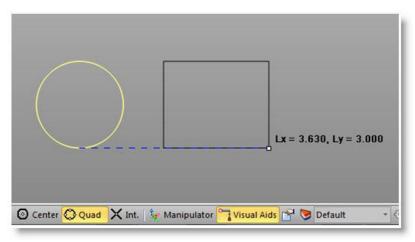
This provides an option to turn on or off display of curve start point and direction arrow.



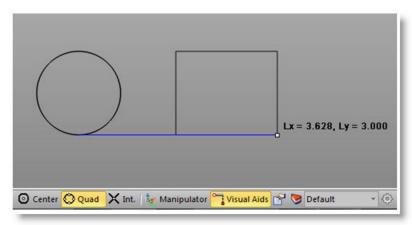
Selecting Start Point and Curve Direction arrow display under options displays both start point and curve direction arrow.

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.

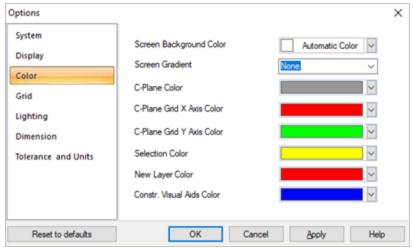
Related Topics

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.

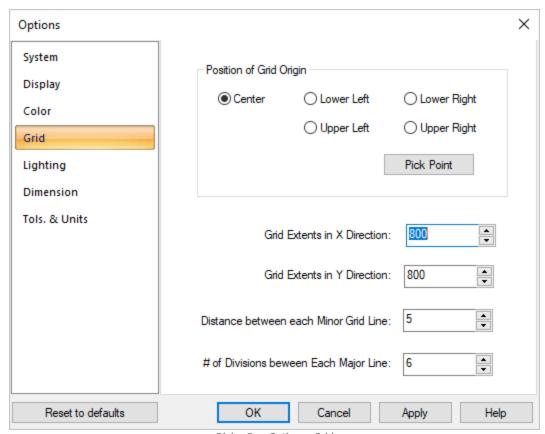
Related Topics

Options

1.5.4 Grid

This controls how the construction plane grid is displayed.

Dialog Box: Options > Grid



Dialog Box: Options > Grid

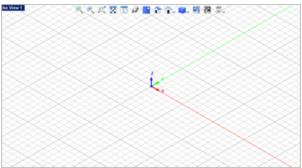
Position of Grid Origin

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

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

Center

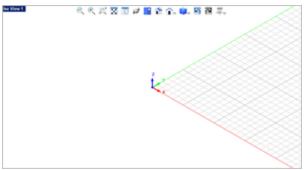
Move the center of the grid to the WCS.



Move the center of the grid to the WCS.

Lower Left

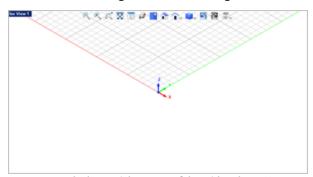
Move the lower left corner of the grid to the WCS.



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

Lower Right

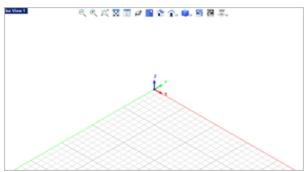
Move the lower right corner of the grid to the WCS.



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

Upper Left

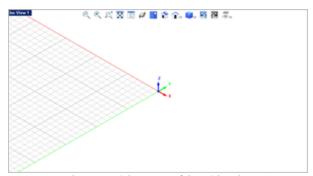
Move the upper left corner of the grid to the WCS.



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

Upper Right

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



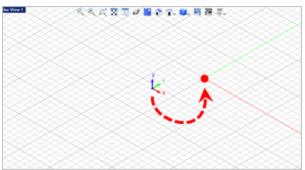
Move the upper right corner of the grid to the WCS.

Pick

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

Notes:

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



Move the the origin of the grid to this location.

Other Grid Options

Grid Extents in X Direction

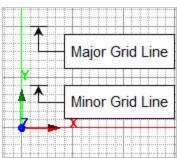
This controls how many major divisions are displayed along the X axis direction.

Grid Extents in Y Direction

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

Distance between each Minor Grid Line

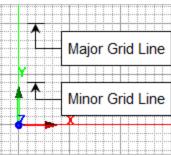
This controls the distance between each minor grid line.



Major/Minor Grid Lines

of Divisions between Each Major Line

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



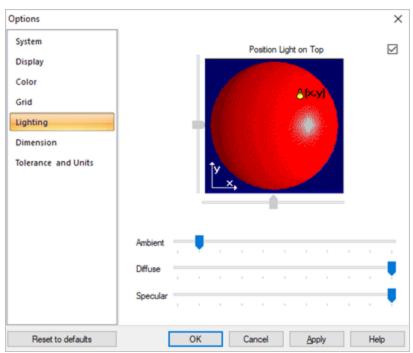
Major/Minor Grid Lines

Related Topics
Options

1.5.5 Lighting

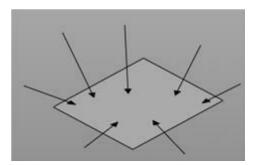
This controls the characteristics of lighting for the model.





Dialog Box: Options > Lighting

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

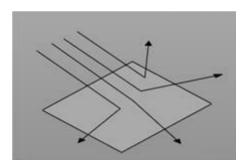


Position Light on Top

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.

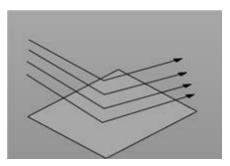
Ambient

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.



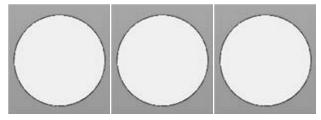
Diffuse

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

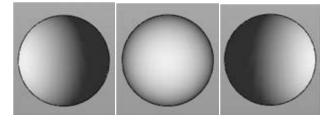


Specular Specular

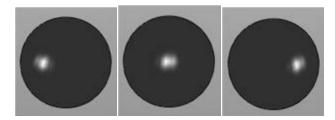
Specular light comes from a single source and is scattered in a single direction. It is reflective light. Shiny metal objects demonstrate good specular reflection. Shading is not a significant consideration with specular light.



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.



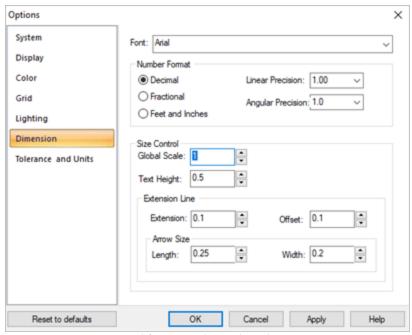
The last set of pictures uses only specular lighting. As the light source moves from left to right, the "shiny" spot follows it. However, there is nothing else lit up, so the effect is not just reflective metal.

Related Topics
Options

1.5.6 Dimension

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

Dialog Box: Options > Dimension



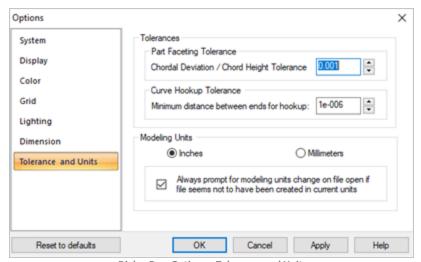
Dialog Box: Options > Dimension

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

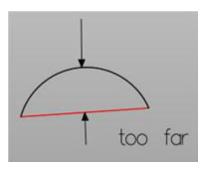
This controls the accuracy of the models. Smaller tolerance numbers often result in significantly big files and slower processing time but are more accurate.

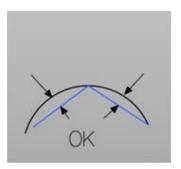
The tolerances are set in a dialog.

Part Faceting Tolerance

Chordal Deviation/Chord Height Tolerance

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



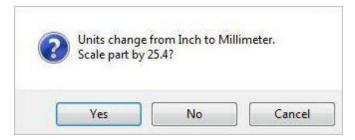


Curve Hookup Tolerance

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.

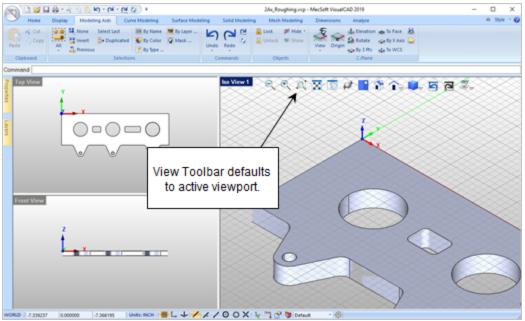
Related Topics
Options

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 Displayed in the Active Viewport



View Toolbar Displays in the Active Viewport by Default

The View Toolbar Commands

Icon	Command/Menu	Command
Q	Zoom Out	
e	Zoom In	
Q	Zoom Box	
\boxtimes	Fit View	_
E	Dynamic Pan	
ۻ	Dynamic Rotate	
	Top View	
	Iso View	

	Top View	Top View	
	Front View	Front View	
	Back View	Back View	
	Left View	Left View	
	Right View	Right View	
	Bottom View		
	Isometric-1	Bottom View	
	Isometric-2		
	Isometric-3	Isometric-1,2,3,4	
	Isometric-4		
	View Menu		
		Shaded Display	
	Shaded Display		
	Toggle Shading+Edges	Toggle Shaded+Edges	
	Wireframe Display	Wireframe Display	
	Toggle Hidden Line	Toggle Hidden Lines	
	Ghosted Display	Ghosted Display	
	Parallel Projection	Toggle Parallel/Perspective	
	Display Menu	Projection	
5	Previous View	These are III de la VI	
		These are Undo/Redo View commands.	
2	Next View		
	Hide / Show Grid	Hide/Show Grid	
(00°)	Hide / Show WCS		
	Show/Hide Menu	Hide/Show WCS	

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



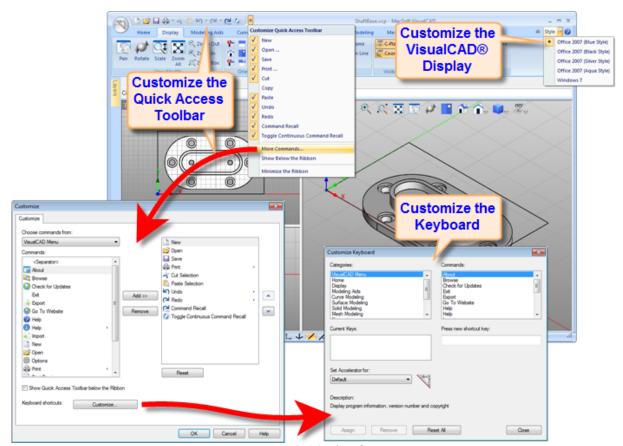
VisualCAD/CAM supports 3D devices from 3Dconnexion®. It can be customized in the 3Dconnexion® Properties dialog by selecting the Button Configuration tab. Button configurations can be set for either the Left mouse button, the Right mouse button, or both and other buttons depending on the type of 3D device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



Customize VisualCAD®

Related Topics

Customize Quick Access Toolbar

Minimize Ribbon Bar

Preset < Alt> Accelerator Keys

Customize Dialog

Customize the Keyboard

Styles Menu

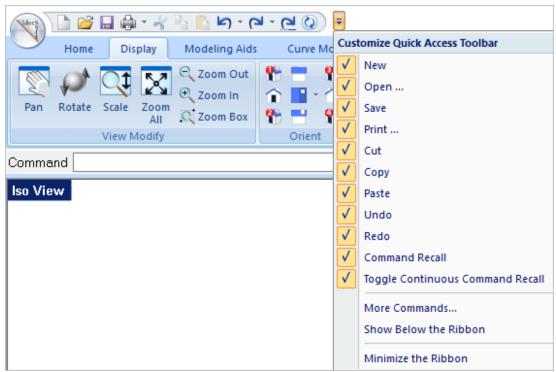
1.7.1 Customize Quick Access Toolbar



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

Menu: Customize Quick Access Toolbar

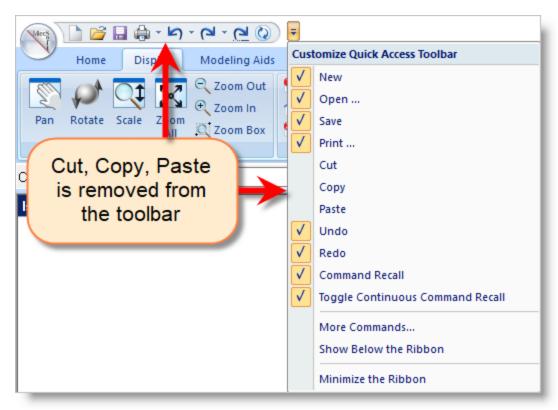
Select the indicator to the right of the Quick Access Toolbar to display the menu:



Displaying the Quick Access Toolbar Manu

Quickly Turn Commands On/Off

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



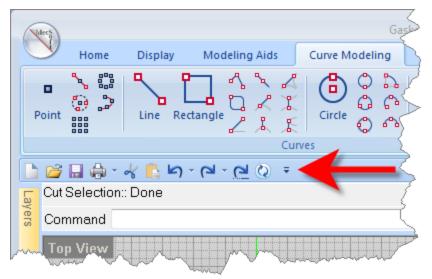
Quickly Turn Commands On/Off from the Quick Access Toolbar

More Commands...

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

Show below the Ribbon

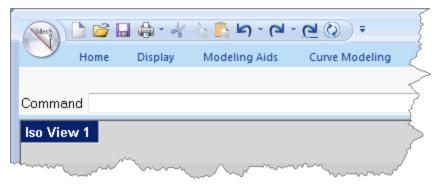
You can move the Quick Access Toolbar to reside below the Ribbon bar by selecting Show Below Ribbon from the Customize Quick Access Toolbar menu. You can also do this by selecting appropriate check box in the <u>Customize</u> dialog.



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

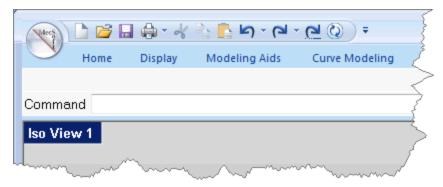
Customize Dialog

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

<u>Customize Quick Access Toolbar</u>

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

- 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.

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

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

Preset <Alt> Keys: Menus

The following preset <Alt> Accelerator key combinations are assigned to commands on the VisualCAD/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: Menus

Preset <Alt> Keys: Home Ribbon Bar

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



Preset <Alt> Keys: Home Ribbon Bar

Preset <Alt> Keys: Display Ribbon Bar

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



Preset <Alt> Keys: Display Ribbon Bar

Preset <Alt> Keys: Modeling Aids Ribbon Bar

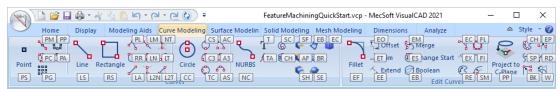
The following preset <Alt> Accelerator key combinations are assigned to commands on the Modeling Aids Ribbon Bar.



Preset < Alt> Keys: Modeling Aids Ribbon Bar

Preset <Alt> Keys: Curve Modeling Ribbon Bar

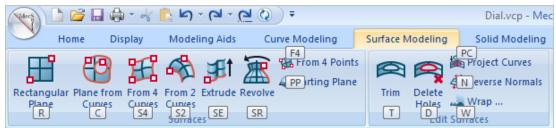
The following preset <Alt> Accelerator key combinations are assigned to commands on the Curve Modeling Ribbon Bar.



Preset < Alt> Keys: Curve Modeling Ribbon Bar

Preset <Alt> Keys: Surface Modeling Ribbon Bar

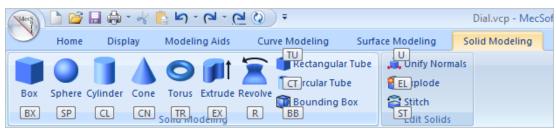
The following preset <Alt> Accelerator key combinations are assigned to commands on the Surface Modeling Ribbon Bar.



Preset <Alt> Keys: Surface Modeling Ribbon Bar

Preset <Alt> Keys: Solid Modeling Ribbon Bar

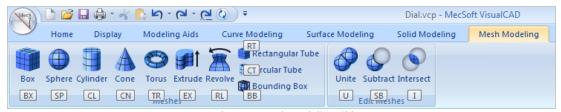
The following preset <Alt> Accelerator key combinations are assigned to commands on the Solid Modeling Ribbon Bar.



Preset <Alt> Keys: Solid Modeling Ribbon Bar

Preset <Alt> Keys: Mesh Modeling Ribbon Bar

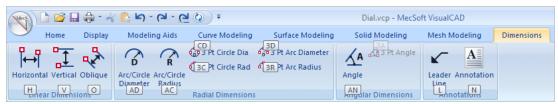
The following preset <Alt> Accelerator key combinations are assigned to commands on the Mesh Modeling Ribbon Bar.



Preset <Alt> Keys: Mesh Modeling Ribbon Bar

Preset <Alt> Keys: Dimensions Ribbon Bar

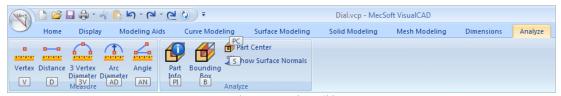
The following preset <Alt> Accelerator key combinations are assigned to commands on the Dimensions Ribbon Bar.



Preset <Alt> Keys: Dimensions Modeling Ribbon Bar

Preset <Alt> Keys: Analyze Ribbon Bar

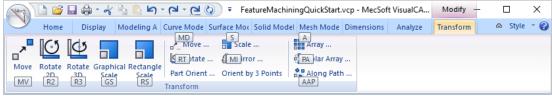
The following preset <Alt> Accelerator key combinations are assigned to commands on the Analyze Ribbon Bar.



Preset <Alt> Keys: Analyze Ribbon Bar

Preset <Alt> Keys: Transform Ribbon Bar

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



Preset < Alt> Keys: Transform Ribbon Bar

Related Topics

Customize VisualCAD/CAM

Customize Quick Access Toolbar

Minimize Ribbon Bar

Customize Dialog

Customize the Keyboard

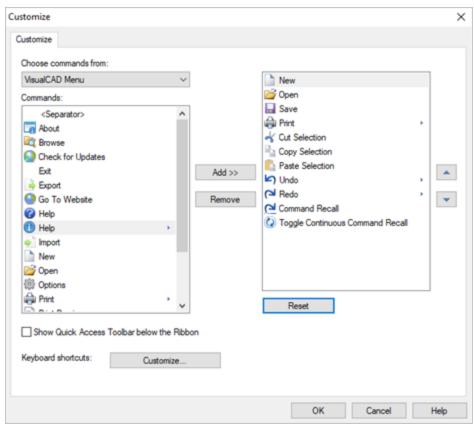
Styles Menu

1.7.4 Customize Dialog

The Customize dialog shown below allows you to add new command icons to the <u>Quick Access</u> <u>Toolbar</u>. From this dialog you can also <u>Customize the Keyboard</u> 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 <u>Quick Access</u> <u>Toolbar</u> 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:

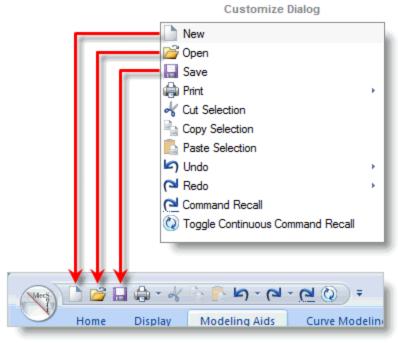
This window lists all of the commands on the Ribbon bar selected from the Choose commands from: list. Select a command from the list and then pick the Add >> button to add it to the list on the right.

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.

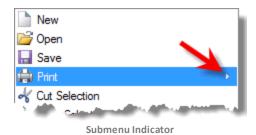
Selected Commands List

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



Quick Access Toolbar

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



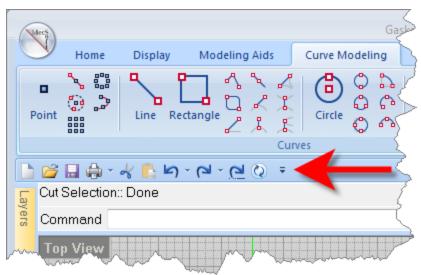
Pick the Up or Down buttons to move a selected command higher (to the left) or lower (to the right) on the Quick Access Toolbar.

Reset

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

Show Quick Access Toolbar below Ribbon

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



Show the Quick Access Toolbat below the Ribbon Bar

Keyboard Shortcuts / Customize

Select the Customize... button to display the <u>Customize Keyboard</u> 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

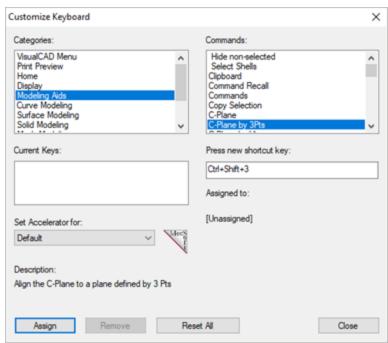
1.7.5 Customize the Keyboard

Invoked by the <u>Customize dialog</u>, the <u>Customize Keyboard</u> dialog shown below allows you to redefine the <u>Default Shortcut Keys</u> assigned to your keyboard. Keyboard shortcuts are combinations of keys that can be pressed to execute a <u>VisualCAD/CAM</u> 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

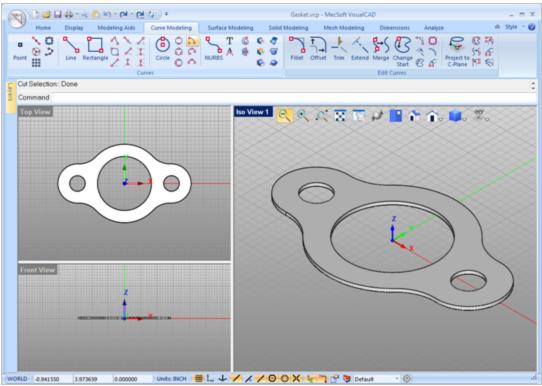
1.7.6 Styles Menu

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.



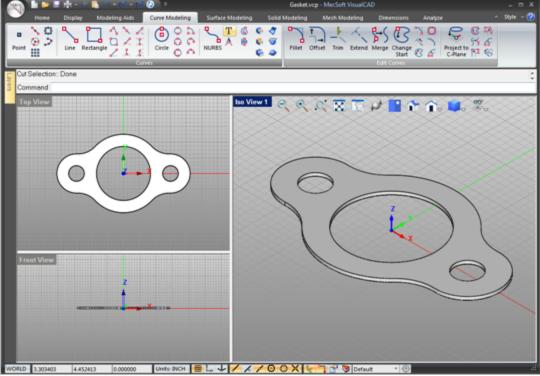


Office 2007 (Blue Style)



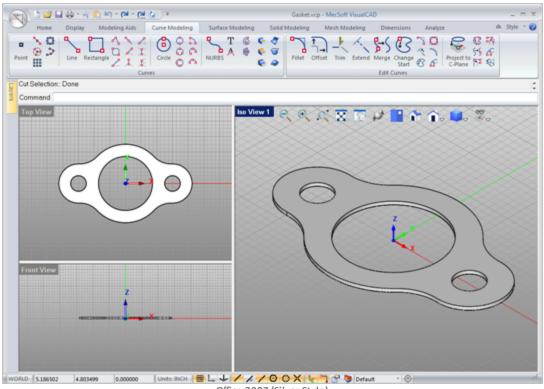
Office 2007 (Blue Style)

Office 2007 (Black Style)



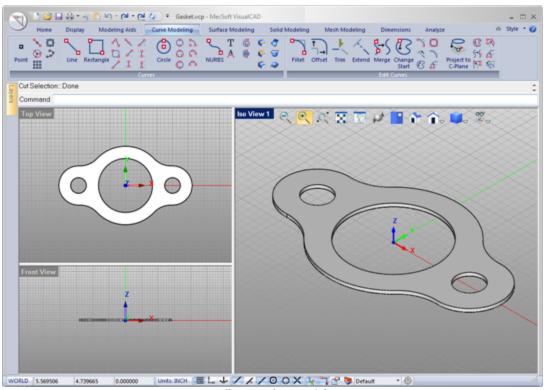
Office 2007 (Black Style)

Office 2007 (Silver Style)



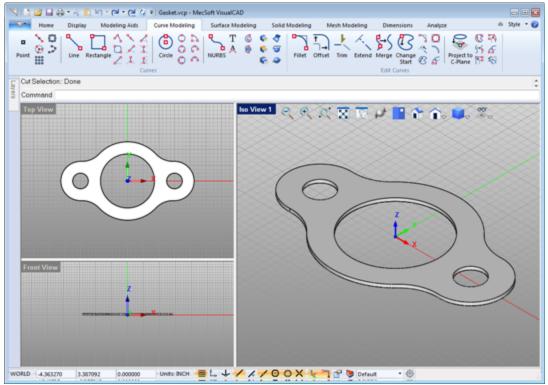
Office 2007 (Silver Style)

Office 2007 (Aqua Style)



Office 2007 (Aqua Style)

Windows 7



Windows 7

Related Topics

Customize VisualCAD/CAM

<u>Customize Quick Access Toolbar</u>

Minimize Ribbon Bar

Preset <Alt> Accelerator Keys

Customize Dialog

Customize the Keyboard

Quick Start



VisualCAD 2025

Prefer Printed Documentation? Click Here!

What's New | Quick Start Play List

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

Training Guides (Click on Image to Download)

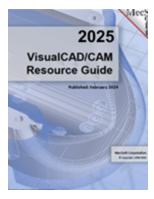


Related Topics
Find More Resources

Resource Guide

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

2025 VisualCAD/CAM Resource Guide



The 2025 VisualCAD/CAM Resource Guide!

18 Pages

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

<u>Prefer Printed Documentation? Click Here!</u>
What's New | Quick Start Play List

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. <u>VisualCAD Preferences</u>
- 2. 2D Drawing & Dimensioning
- 3. Model a Spanner
- 4. Model a Base Plate
- 5. Model a Mold Insert
- 6. Model a Connector Block

- 7. <u>Model a Daisy Decor</u>
- 8. <u>Using Construction Planes</u>

#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:

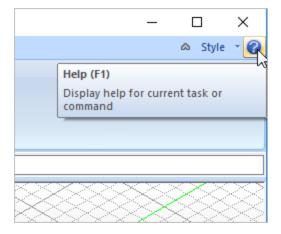
In this exercise you will perform the following VisualCAD tasks:

- 1. Access the online help.
- 2. Set the Display Style.
- 3. <u>Set the Units to Inches.</u>
- 4. <u>Set System Options.</u>
- 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

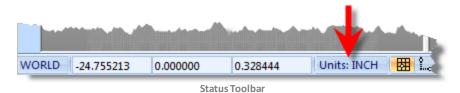
VisualCAD allows you to change the Style of the display. At the very top right of the VisualCAD display window, select the Style drop-down menu and select Office 2007 (Blue Style).



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.



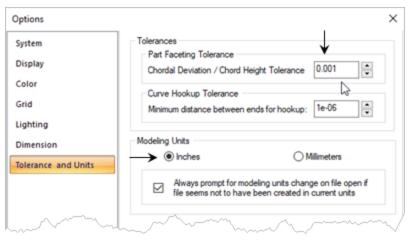
If Units is set to MM, double-click on it to change to INCH.

5.4 Set Systems Options

Set Systems Options

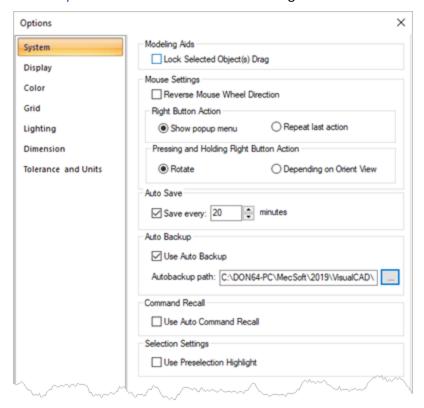
In this exercise you will learn how to set some of the systems options in VisualCAD.

- 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 <u>Tolerance and Units</u> from the left side of the dialog.
- 4. Under Part Faceting Tolerance set the Chordal Deviation / Chord Height Tolerance to 0.001".
- 5. Set the Units to Inches.

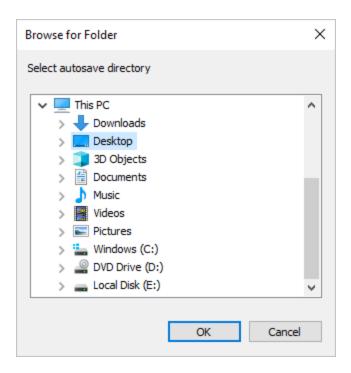


System Options Dialog

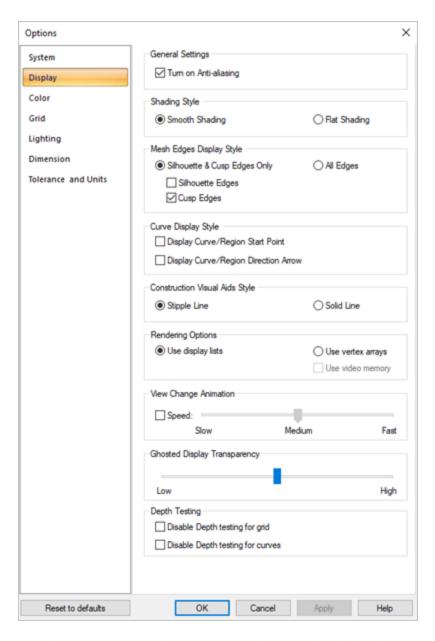
6. Select System from the left side of the dialog.



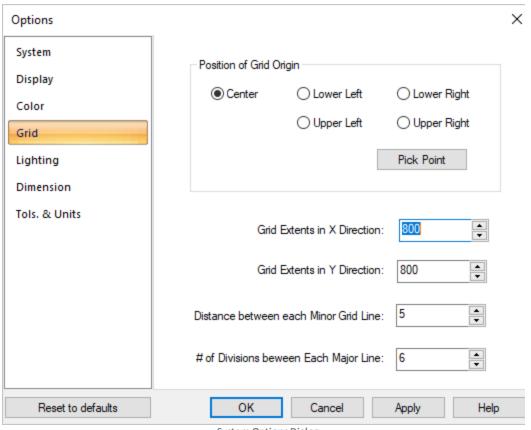
- 7. Under Auto Save, check the box and then set the value to save every 10 minutes.
- 8. Check the box to use Auto Backup and then select the ... box to display the Browse for Folder dialog.
- 9. Select the folder where you want backup files to be saved and then pick OK.



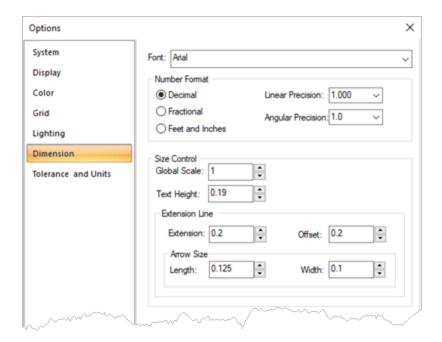
- 10. Set the remaining System Options as shown in the dialog above.
- 11. Now select Display from the left side of the dialog.
- 12. Under View Change Animation uncheck the box next to Speed.



- 13. Set the remaining Display Options as shown in the dialog above.
- 14. Select Grid from the left side of the dialog. In these exercises we use the following Grid options.



- System Options Dialog
- 15. Select Dimension from the left side of the dialog.
- 16. Set the Dimension Preferences to those shown in the dialog below.
- 17. Pick OK to close the Options dialog.



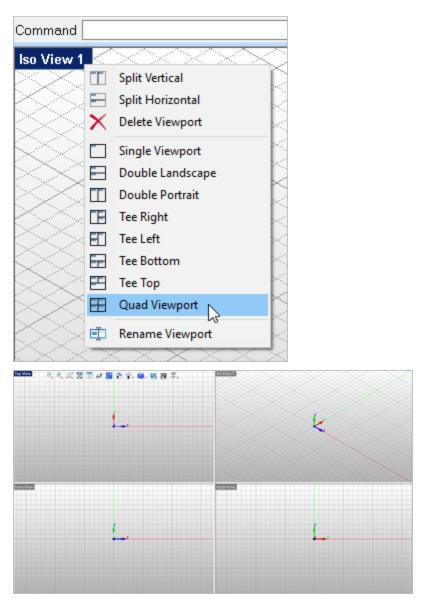
5.5 Set to Quad Viewports

Set to Quad Viewports

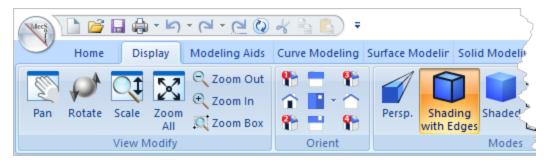
It is much faster to draw and model in VisualCAD if you have multiple Viewports displayed.

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.

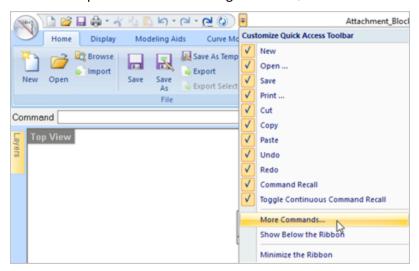


5.6 The Quick Access Toolbar

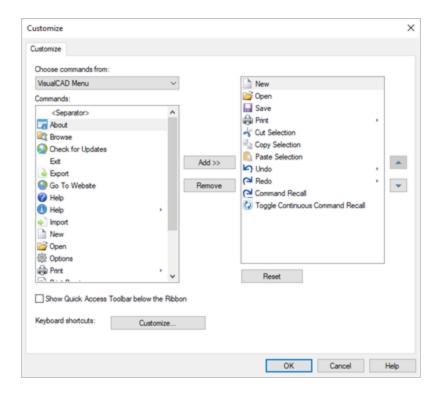
The Quick Access Toolbar

Located at the very top left of the VisualCAD window you will find the <u>Quick Access</u> <u>Toolbar</u>. 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 <u>Customize</u> dialog. You can use this dialog to add or remove commands to the Quick Access Toolbar.

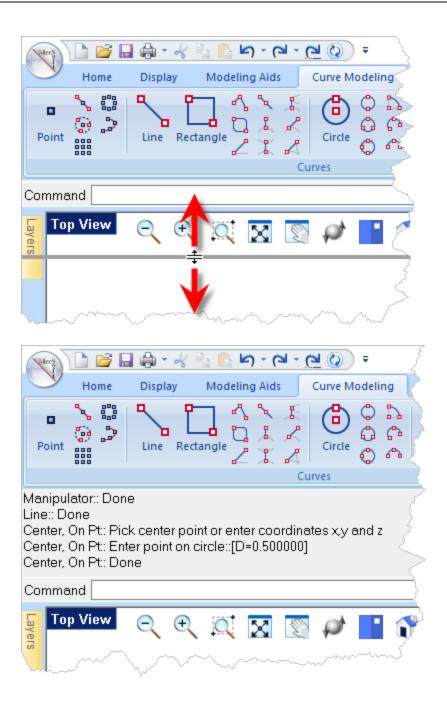


5.7 Viewing the Command Prompts

Viewing the Command Prompts

The Command Input bar Command 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.

- Move the cursor onto the bottom of the Command Input bar Command until it changes.
- 2. Then left-click and drag the Command Input bar Command down to reveal to review the history of commands.



5.8 The Status Bar

The Status Bar

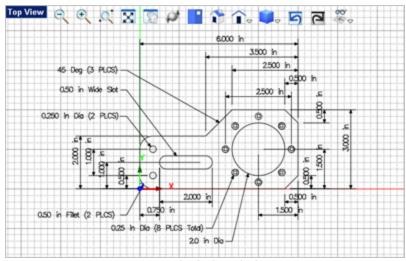
Located across the very bottom of the VisualCAD display you will find the Status Bar. It contains information and quick access to some very helpful commands. From left to right on the Status Bar you will find This is where you access the Layer Manager, Properties dialogs, Graphic Manipulator and more. 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. <u>Using Visual Aids</u>.
- 4. <u>Dimensioning</u>.

6.1 Create Layers

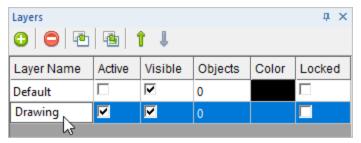
Create new Layers

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

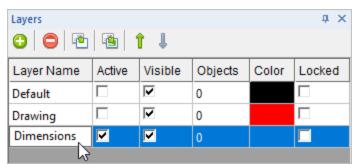
- 1. Display the Layer Manager. You can pick the Layer Manager icon located on the Status Bar.
- 2. From the Layer Manager select the Add Layer icon to add a new layer.



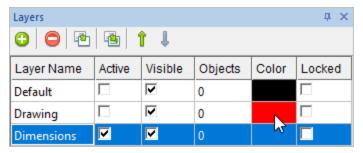
3. Now lets rename the new layer to Drawing. First double-left-click in the Name field to activate it, and enter Drawing as the new layer name.



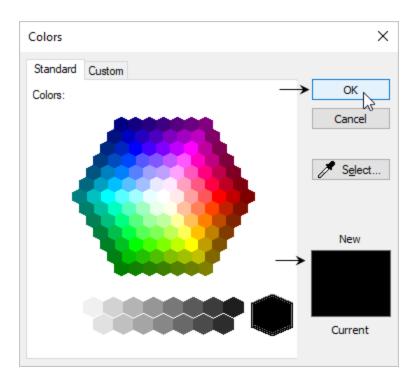
4. Repeat the procedure to create a new layer, this time named Dimensions.



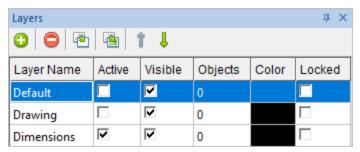
- 5. If the Layer Manager keeps Auto-Hiding to the left side of the display, you can select the Auto Hide pin icon at the top of the Layer Manager. to toggle this feature. The pin down icon means that the dialog will stay displayed. The pin left icon means that the dialog will auto-hide.
- 6. Now let's change the color of these two layers. In the Color field left-click on the color box for the layer named Drawing. Again, this will display the Colors dialog.



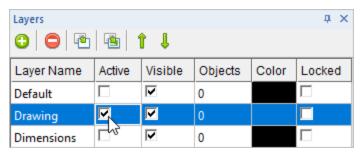
7. From the Colors dialog that displays, accept the default color black by pick OK to close the dialog.



8. Repeat the procedure for the layer named Dimensions.



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



6.2 Curve Drawing

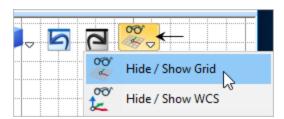
Curve Drawing

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

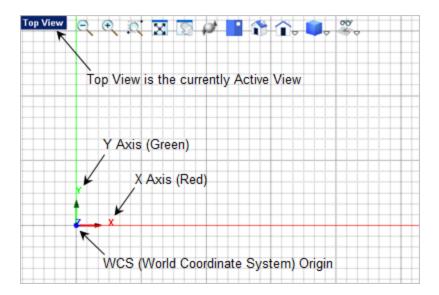
- 1. You only need the Top View Top View displayed for this exercise so if you have multiple Viewports displayed, double-left-click on the Top View tab to change the display to that Viewport.
- 2. Alternately you can right-click on the Top View tab and select Single Viewport.
- 3. If you do not see the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



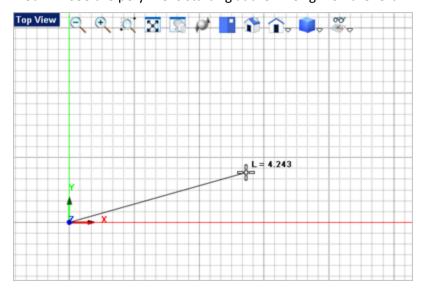
- 7. Now we can start curve drawing. From the Curve Modeling Curve Modeling tab, select the Polyline command.
- 8. The command prompt says:

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

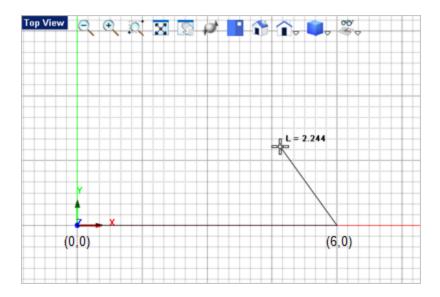
 For the start point of the polyline, enter 0,0 in the command window

 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 Command and press <Enter>. This is in Absolute World Coordinates, 6 units in X and 0 units in Y measured from the World Origin.



10. Now if you are drawing and you do not know the exact absolute coordinate values you can enter Relative Coordinate values in the command window.

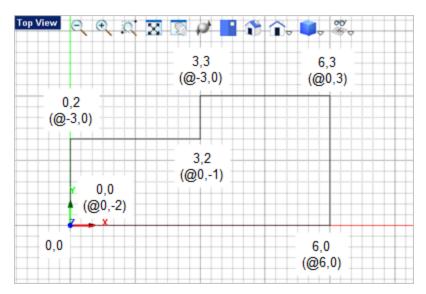
For the next coordinate enter @0,3 in the command window and press <Enter>. This will place the next point in the polyline 3" from the last point in the positive Y Axis direction.

Listed below are the absolute and relative coordinate values for the entire polyline. Use them to complete the polyline, pressing <Enter> after each coordinate entered. If you make a mistake, just Undo and start again.

Polyline Absolute (Relative) coordinate values

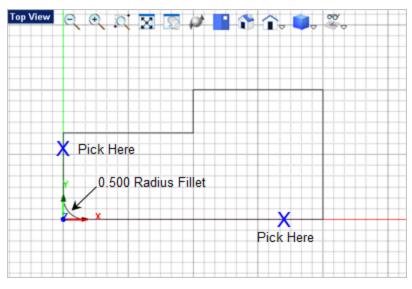
- 0,0
- 6,0 (@6,0)
- 6,3 (@0,3)
- 3,3 (@-3,0)
- 3,2 (@0,-1)
- 0,2 (@-3,0)
- 0,0 (@0,-2)

Your polyline should like this:



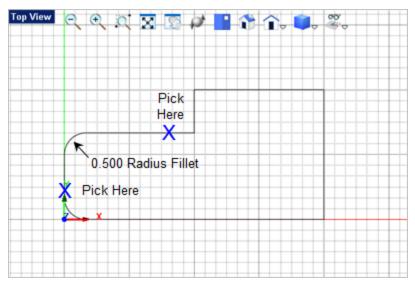
Note: In the future you can also specify a point at an angle and length from the previous point using the following format: @6<45 where the length of the line segment is 6 and the direction is 45 degrees from the previous point.

- 11. Always Save your work! Select the Save or Save As command from the Home tab or the VisualCAD Main Menu or press <Ctrl+S>.
- 12. Now let's add some fillets. From the Curve Modeling tab select the Fillet command.
- 13. For the fillet radius, enter 0.5 in the command window and press <Enter>.
- 14. For the two lines to fillet, select the lines marked in the illustration below. For filleting it is best to select the line on the end furthest from the end you want to fillet.

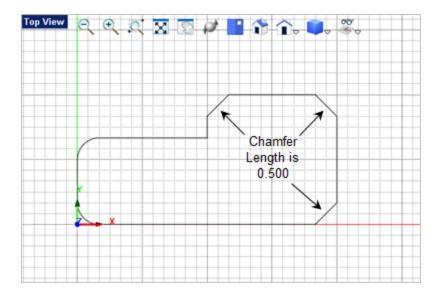




- 15. Press <Enter> to repeat the Fillet command.
- 16. The default fillet radius is now 0.5 so proceed directly to picking the two lines to fillet and press <Enter>.



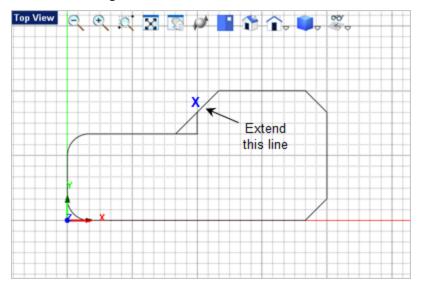
- 17. Next we will chamfer the three remaining outer corners. From the Curve Modeling tab select the Chamfer command.
- 18. For the chamfer length, enter 0.5 in the command window Command and press <Enter>.
- 19. For the lines to chamfer, select lines marked in the illustration below. For chamfers it 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.

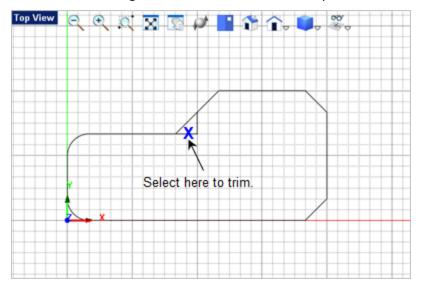


- 22. Now from the Curve Modeling Curve Modeling tab select the Extend Curves command.
- 23. Select the change line marked below to extend it to the horizontal line.

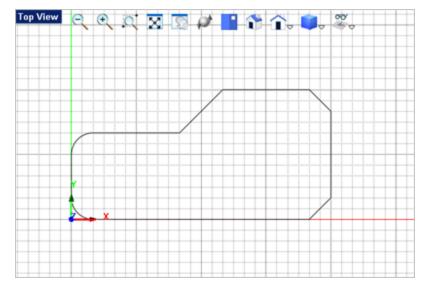




- 24. Now from the Curve Modeling Curve Modeling tab select the Trim Curves command.
- 25. Pick the line segment to trim. When using the Trim command, you do not need to select the trimming curve. Just select what curve you want trimmed.



26. Next, delete the unneeded line. Just select it and then press the key or select the Cut Cut command icon from the Quick Access Toolbar.



6.3 Using Visual Aids

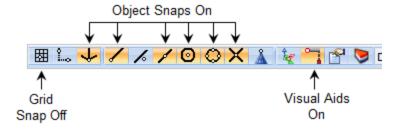
Using Visual Aids

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.

1. Curve creation using Visual Aids can be enabled by selecting the Visual Aids icon on the Status Toolbar.

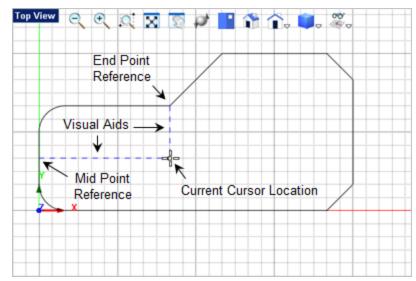
While Visual Aids is toggled On, the currently active Object Snaps are referenced visually depending on where the cursor is located on the drawing.

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

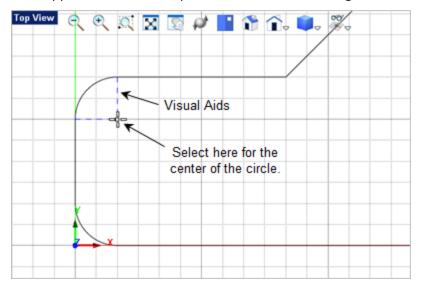




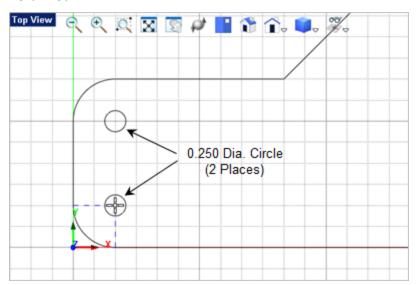
- 4. Now from the Curve Modeling Curve Modeling tab select the Circle on Point command.
- 5. For the circle center point move the cursor near geometry in your drawing. You will see Visual Aids appear (i.e., reference lines).



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

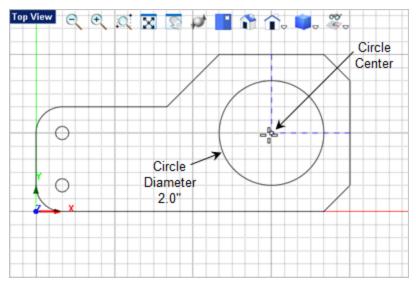
More Drawing Tools

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

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

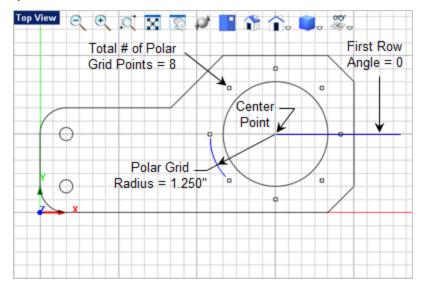


- 2. For the center point of the circle use the Visual Aids to locate and select the Mid Point intersection of the upper and right side lines.
- For the circle diameter enter 2.0 in the command window Command and press <Enter>.



- 4. Next we will draw a polar array of points around the circle we just drew.
 - Why am I drawing Points? Because when machining Hole Making operations in CAM such as Drill, Tap, Bore, etc., you can select points as Control Geometry! The hole geometry DOES NOT have to be present in the 2D drawing or 3D model in order to drill at that location.
- 5. From the Curve Modeling command.
- 6. For the total number of points on the circumference, enter 8.
- 7. For the angle of the first row about the X Axis enter 0.
- 8. Pick the center point for the polar grid or enter the coordinate values. We want the points centered around the 2" circle we just drew, so set you Object Snap to Center Point and move the cursor over the circle until the center point is highlighted and the select it.

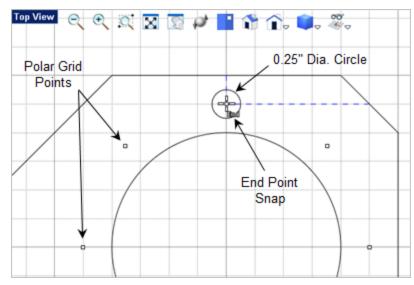
9. For the polar grid radius, enter 1.250 and then press <Enter>. The polar grid of points will be created around the circle as shown below.



10. For this exercise we will also add the polar array of circles. From the Curve



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



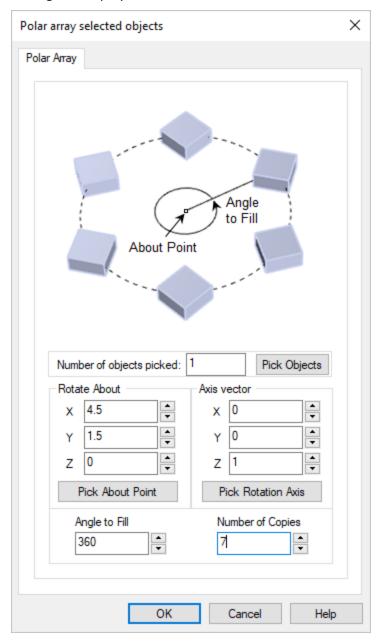
13. Now we will duplicate the circle by creating a polar array. First select the circle you just created.

Modify

Transform tab.

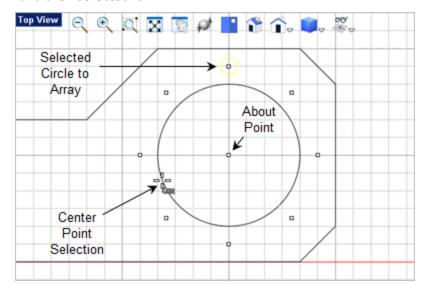
14. Then from the Modify / Transform Transform

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

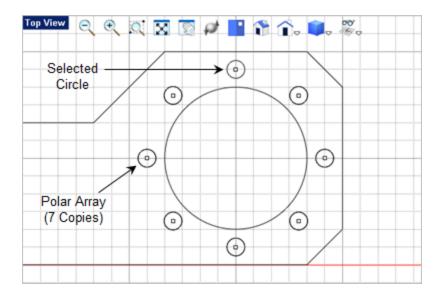


- 16. The Number of Objects field should be 1. This represents the circle that you have selected.
- 17. Under the Axis Vector, the values should be X0, Y0, Z1.

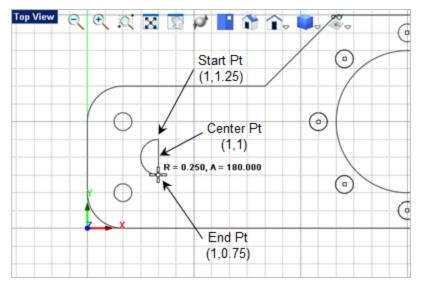
- 18. Under the Rotate About section, select the Pick About Point button. The dialog will minimize and wait for you to select a point.
- 19. From the <u>Status Bar</u> activate the Center Point Object Snap .
- 20. Now move the cursor over the 2" diameter circle until the center point is displayed and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

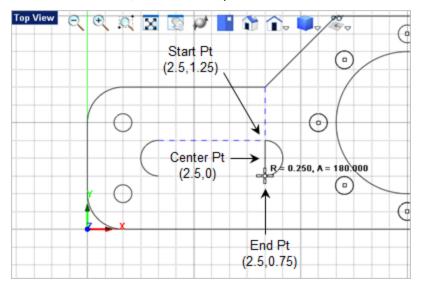


- 26. Now we will drawing a slot. From the Curve Curve Modeling tab select the Arc Center, Start, Angle command.
- 27. In the Status Bar make sure the Grid Snap is On .
- 28. For the center point select the grid point shown below or enter 1,1 in the command window and press <Enter>.
- 29. Following the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

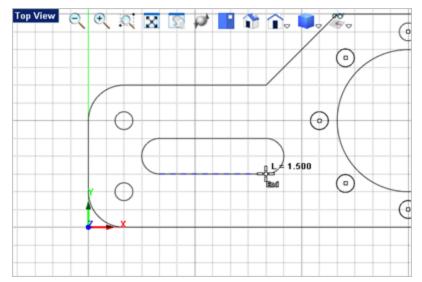
31. Now we want to draw another arc, press <Enter> to repeat the command and select or enter the center, start and end points shown below.



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:



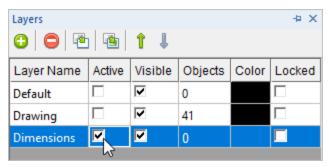
6.5 Dimensioning



Dimensioning

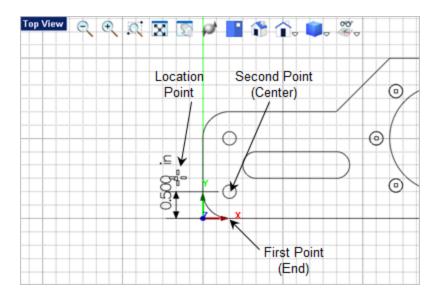
In this step you will learn how to place dimensions on the drawing.

- 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.

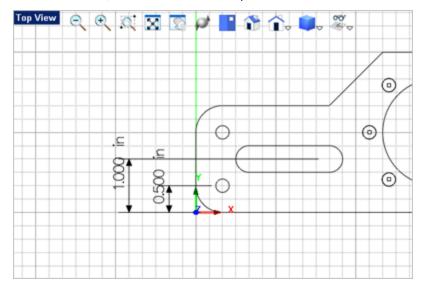




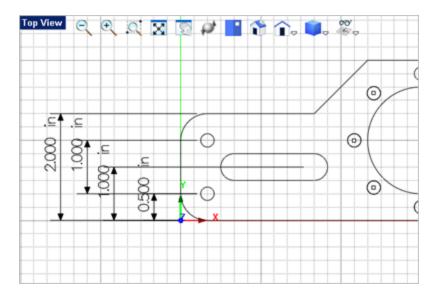
- 5. Activate the Object Snaps required (End , Center and Grid)
- 6. Select the first and second points that you wish to dimension.
- 7. Then select a point to locate the dimension. refer to the illustration below:



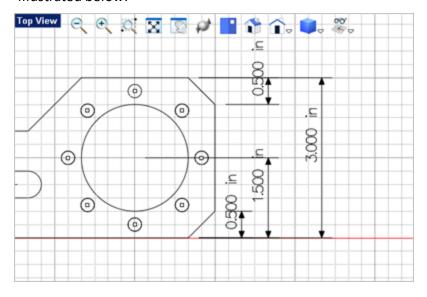
- 8. Press < Enter > to repeat the command.
- 9. Select the first, second and location point for the dimension as shown below:



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



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



12. Now we will create the horizontal dimensions. From the Dimension

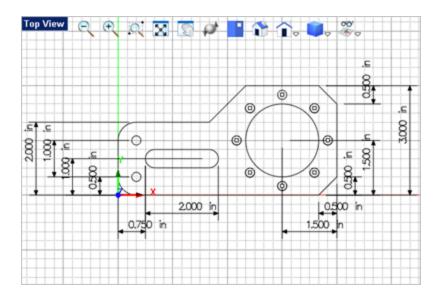
Dimensions



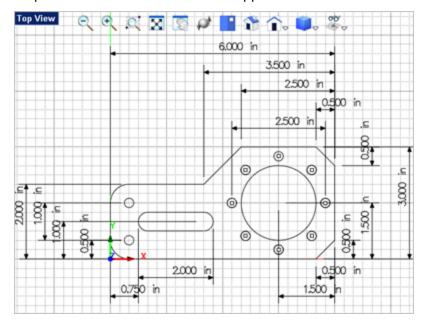
tab select the Horizontal Dimension

command.

13. Select the first second and location points to create the lower horizontal dimensions shown below:



14. Repeat the command to create the upper horizontal dimensions shown below:

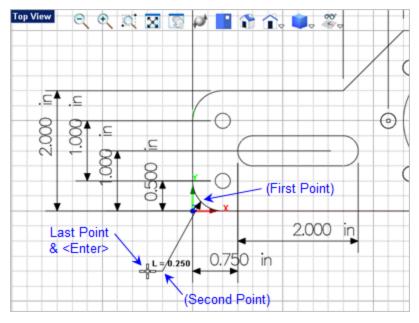


15. Now we will create some leader lines. From the Dimension tab select

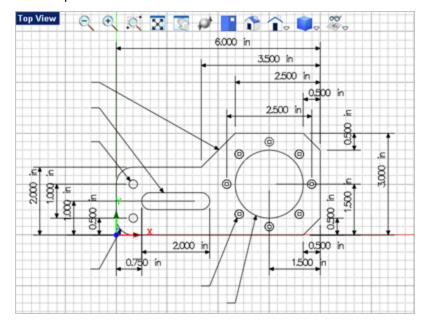


- 16. The first command prompts says: "Enter the text string." We will add the annotation separately so you can press <Enter> to continue.
- 17. The next prompt says: "Pick point or enter coordinates x,y, and z" Change the Object Snap to Near and select a point on the lower fillet for the first point.

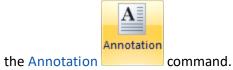
18. Then pick additional points to define the leader line and the press <Enter> or right-click to end the command. The points to pick are shown in the illustration below:



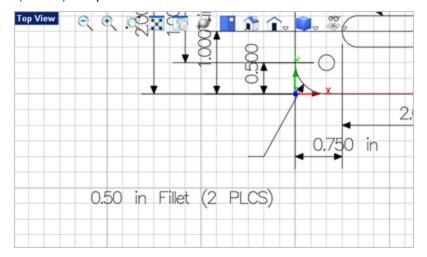
19. Now repeat the Leader Line command and add the leader lines shown below:



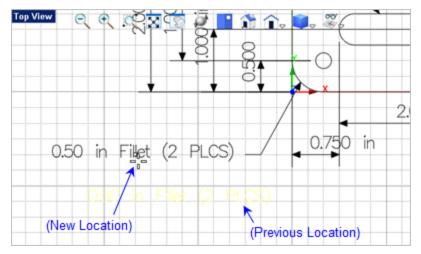
20. Now we will add the annotation text. From the Dimension tab select



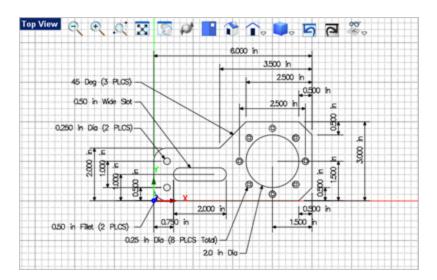
21. For the text string, at the command prompt Command enter 0.50 in Fillet (2 PLCS) and press <Enter>



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.



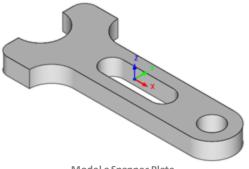
- 23. Repeat the Annotation command to create the additional leader text shown below:
- 24. Save the file as Drawing and Dimensioning Completed.vcp.



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, Fillets, Mirroring, Merging and Extruding curves.



Model a Spanner Plate

What you will learn:

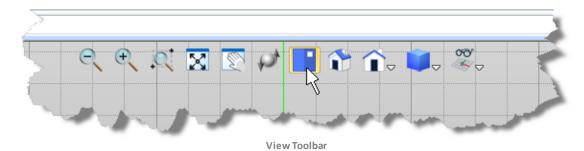
In this exercise you will perform the following VisualCAD tasks:

- 1. Set to the Top View.
- 2. Create Reference Points.
- 3. Create Inner Cutouts.
- 4. Trim Curves.
- 5. Offset, Extend & Trim Curves.
- 6. <u>Fillet Curves.</u>
- 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 <u>View Toolbar</u> or the Orient pane of the <u>Display Ribbon Bar</u>.



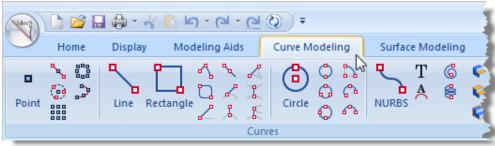


Display Ribbon Bar - Set to the Top View

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 Command
- 5. As you move the mouse on the viewport, the co-ordinate value at the mouse location is displayed under the status bar.

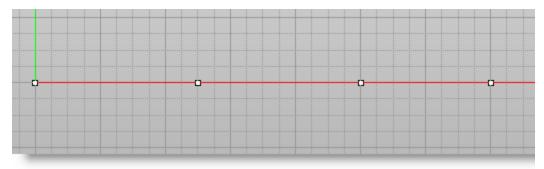
Create additional points by repeating the Create Point command at the following coordinate locations.

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 Curve Modeling 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 Command 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

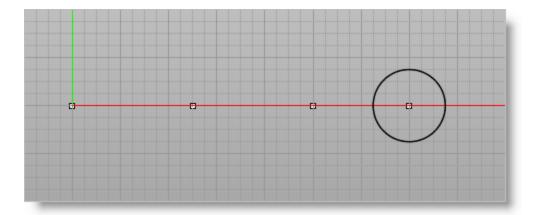
4. For the center point, pick last created point or type 7,0

Command Prompt:

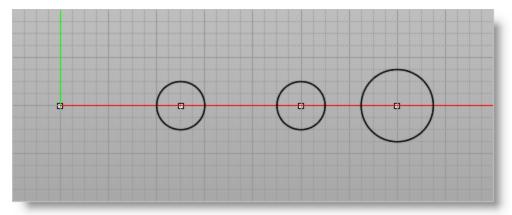
Center, On Pt:: Enter point on circle::[D=0.500000]

5. Specify 1.50 for Diameter in the command bar Command \(\sqrt{and} \) and press Enter.

The created circle is as shown below.



6. Repeat the Circle command to create 2 more circles at the center points (2.5,0 and 5,0) enter a Diameter of 1.00 for each circle.



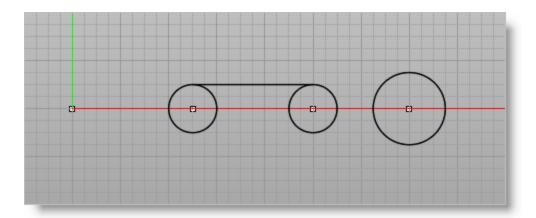
7. From the Status Bar, toggle Grid Snap off and then toggle Quad Point Snap on.



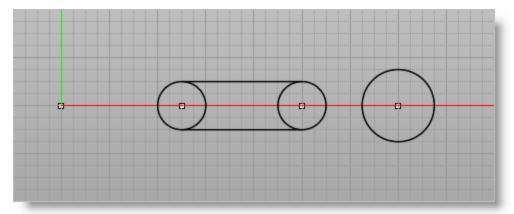
8. Select the Create Single Line icon.



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



10. Repeat the line for lower quadrant points.



7.4 Trim Curves

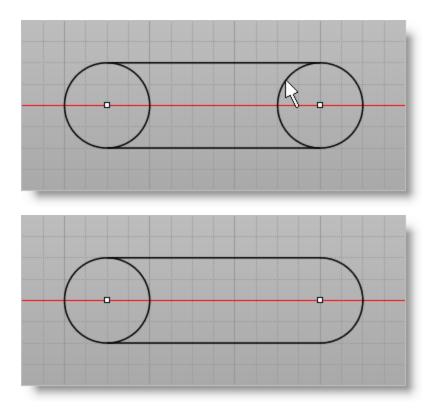
Trim Curves

The circles must be trimmed.

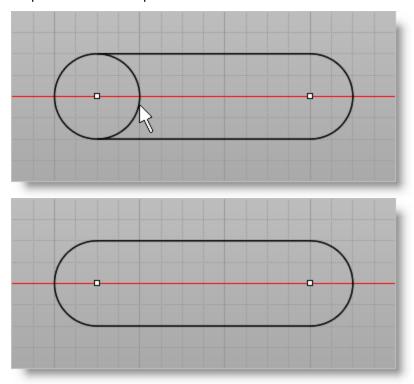
1. Select the Curve Modeling tab Curve Modeling from the top Ribbon Bar.



- 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.



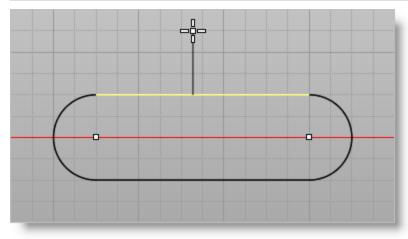
7.5 Offset, Extend & Trim Curves

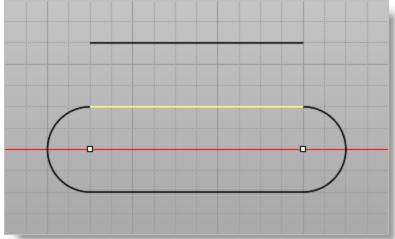
Offset, Extend & Trim Curves

- 1. Select the Curve Modeling tab Curve Modeling from the top Ribbon Bar.
- 2. Keep the Curve Modeling Ribbon Bar displayed and then select the top horizontal line.
- 3. Pick the Offset Curve icon from the Edit Curves pane.

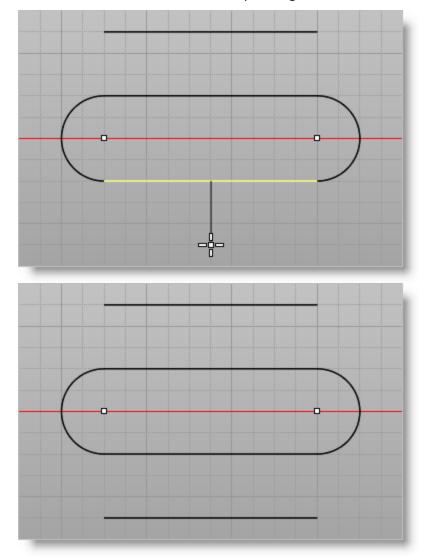
 A preview line will appear, indicating the direction of offset. Point the preview line to offset the curve outward and then type 0.75 from the keyboard and press Enter.
 - For Offset Curve, your cursor points to the offset direction, so do not move the cursor to the Command Prompt Command field.

 Simply type 0.75 and press Enter to input the value.

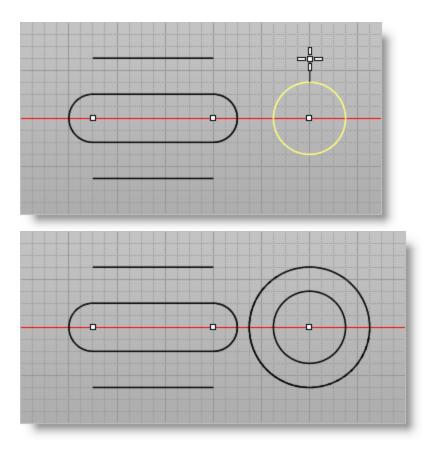




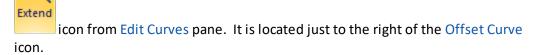
4. Repeat the offset for the horizontal line at the bottom, making sure to keep the cursor and directional indicator line pointing downward.



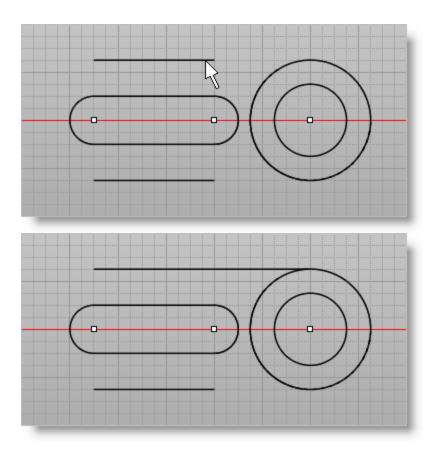
- 5. Select the larger circle on the right and then pick the Offset Curve icon again.
- 6. With the cursor, point the offset direction outward from the circle, type 0.50 and press Enter.



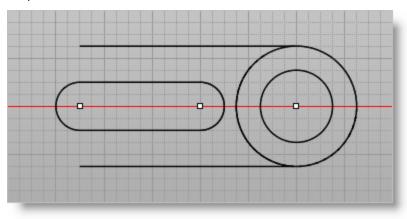
7. To modify the offset lines so that they meet the larger circle, pick the Extend Curve



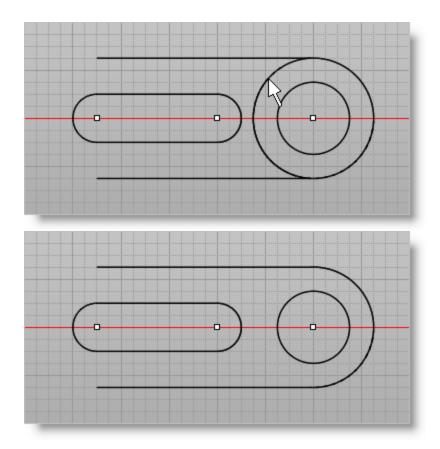
8. Click one of the offset lines towards the right. The line extends to the point where it meets the circle.



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

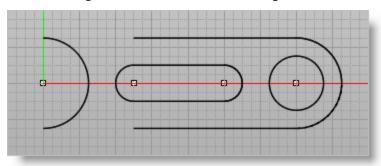
We will now create the left section of the spanner using the filleting and mirroring tools. We will start by creating the arc at the end of the spanner.

- 1. Select the Curve Modeling tab Curve Modeling 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 Command and then press Enter.
- 4. For Start, type 0,1.25 in the Command Prompt Command and then press Enter.

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

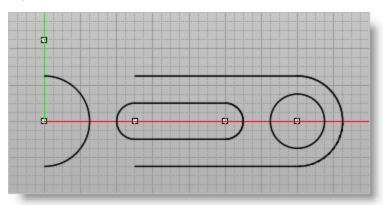
Your drawing should look like the following:



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.

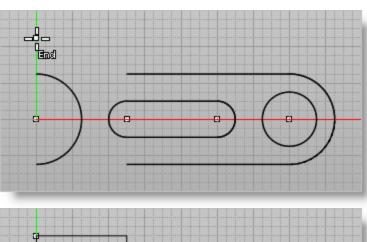


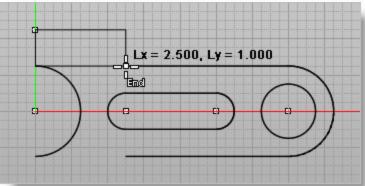
7. From the Status Bar, toggle End Point Snap On.



- 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.

Point





- 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.

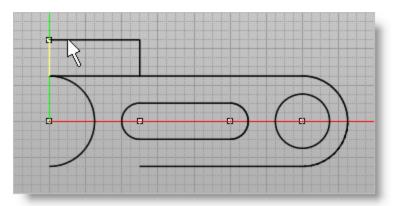


12. Select Fillet

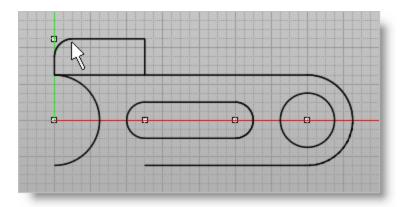
Command Prompt:

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.

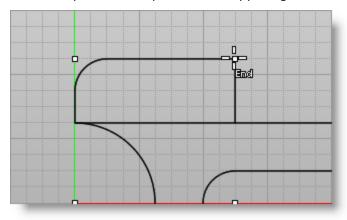


15. Let's now create another arc. Again, from the Curve Modeling Ribbon Bar, select Center, Start, Angle Pts.

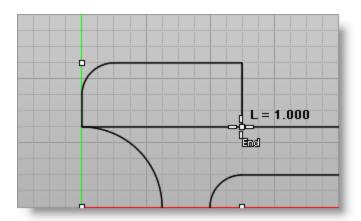
Command Prompt:

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.



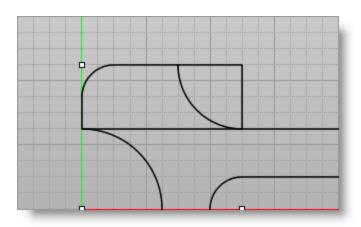
18. For Angle, type in -90 and then press Enter to create an arc in the top right corner of the rectangle.

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]

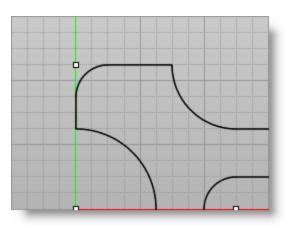
Command -90

The created Arc is shown below:





- 19. To clean up the corners we select Trim
- 20. Select trimming curves to obtain the shape as shown below:

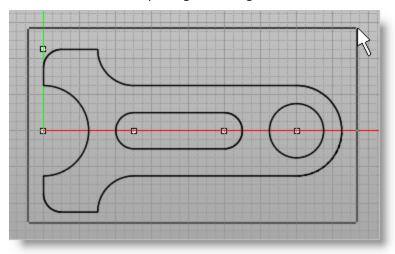


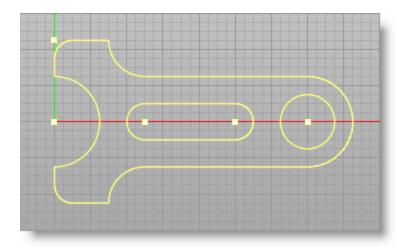
7.7 Merge Curves

Merge Curves

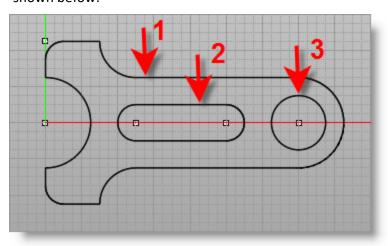
Now we will merge our curves into three separate closed curves.

1. Select all the curves by using a crossing window selection:





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.



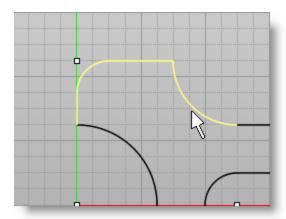
Mirror Curves 7.7.1



Mirror Curves

We will now select the 4 curves that form the head of the spanner and mirror them.

Select the 4 curves as shown below. Hold down the Ctrl key to select multiple objects.



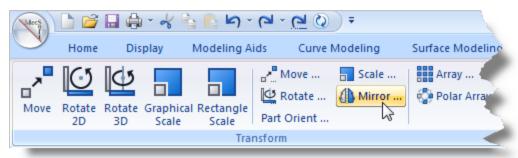
 With the 4 curves selected, you will notice that VisualCAD's Modify mode automatically becomes active and that the Transform Ribbon Bar becomes available. Refer to the menu image below:



The Transform Ribbon Bar is Activated

3. Now there are two methods to activate the Mirror command. Select the method that suits you best.

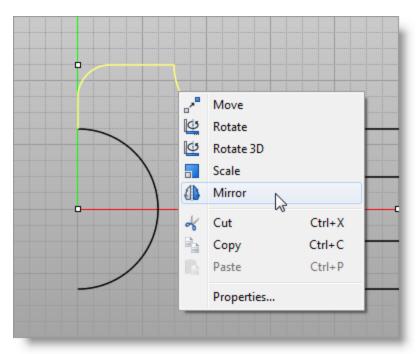
You can select the Transform Ribbon Bar tab and then select the Mirror icon as shown:



The Mirror icon on the Transform Ribbon Bar

- or -

While the 4 curves are selected, you can simply Right-click and select Mirror from the pop-up menu as shown below:

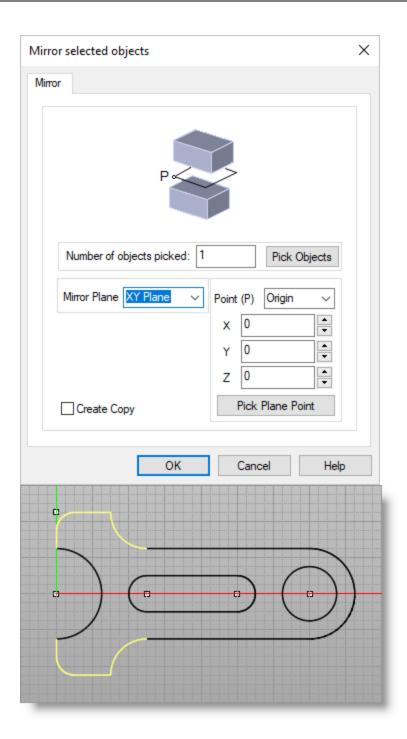


Right-clcik Menu for Editing Objects

4. From the Mirror selected objects dialog box:

Set the Mirror Plane to XZ Plane Leave Point (P) as Origin since the origin lies on the center line of the part. Check Create Copy Pick OK.

The curves are mirrored as shown below.

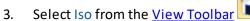


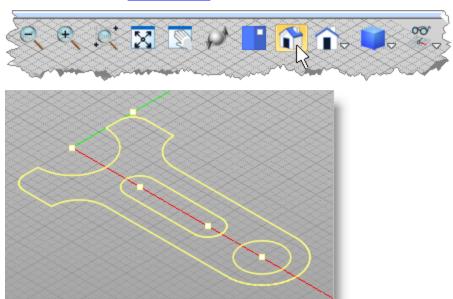
7.8 Extrude Curves

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.



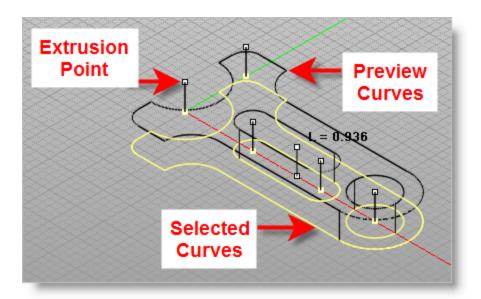


- 4. From the Solid Modeling Ribbon Bar, select Extrude.
- Preview curves will appear to determine the direction of extrusion.
 Set the extrusion to point in the positive Z direction by moving the cursor above the geometry.

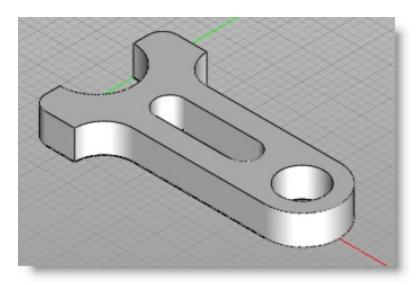
Then type 1.0 in the command input bar and press the <Enter> key.

Command Prompt:

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



The curves are now extruded and displayed as shown below.

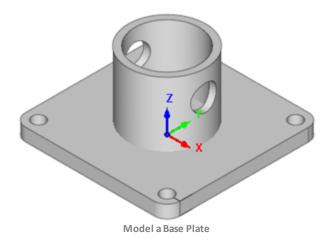


6. Save the file as Spanner_Completed.vcp.

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.



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

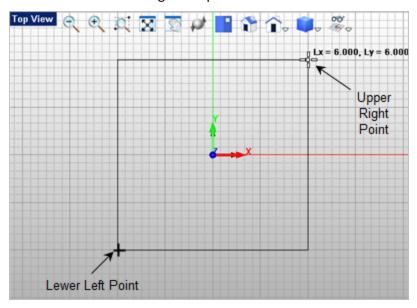
In this exercise you will learn the following VisualCAD concepts:

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

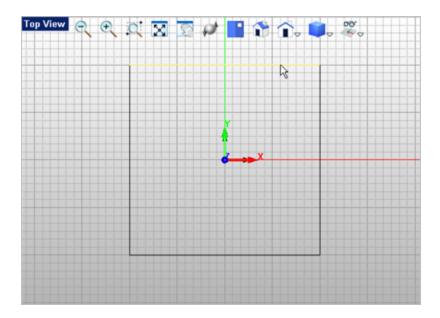


- 2. Now select the Rectangle command.
- 3. For the first corner of the rectangle, select the lower left grid point location shown below or type -3,-3 in the Command Prompt Command and press <Enter>.

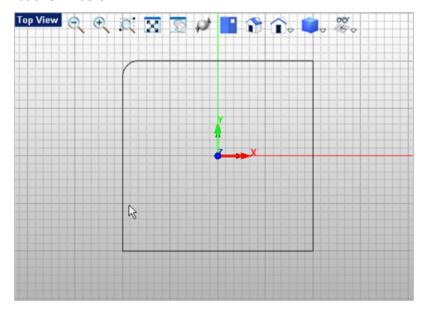
4. For the second corner of the rectangle, select the upper right grid point shown below or type 3,3 in the Command Prompt Command and press <Enter>. This will create a rectangle 6" square.



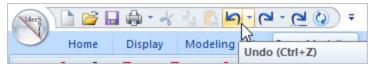
- 5. To fillet the corners, select the Fillet command from the Curve Modeling tab.
- 6. For the fillet Radius, enter 0.5 in the command prompt Command and press <Enter>.
- 7. Now select the first edge to fillet as shown below. **Note**: When selecting curves to fillet, it is best to select furthest away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

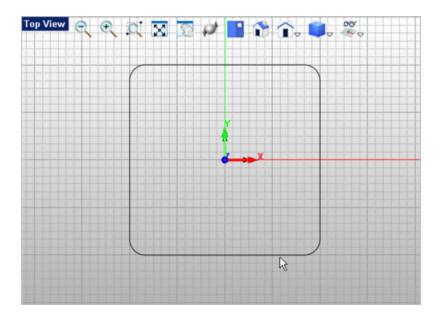


8. Now select the second edge to fillet and a 0.5 radius fillet is added to the rectangle as shown below.



9. Now press <Enter> to repeat the Fillet command and fillet the remaining 3 corners of the rectangle as shown below. If you make a mistake, you can select the Undo command located on the Quick Access toolbar and try again.

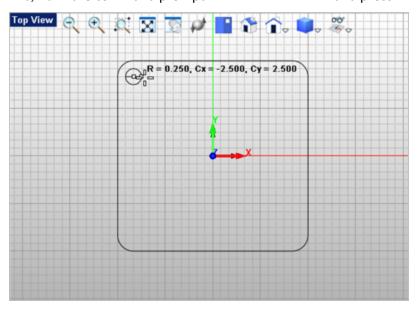




10. Now lets add the mounting holes. From the Curve Modeling tab,

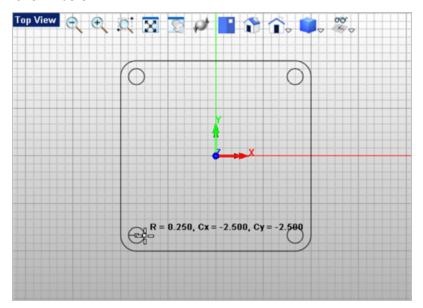
Circle

- select the Circle on Pt command.
- 11. For the center point of the first hole select the grid point shown below or enter-2.5,2.5 in the command prompt Command and press <Enter>.



12. Now for the circle radius you can select the next grid point to the right of the center point you just selected or enter 0.5 for the diameter and press <Enter>.

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



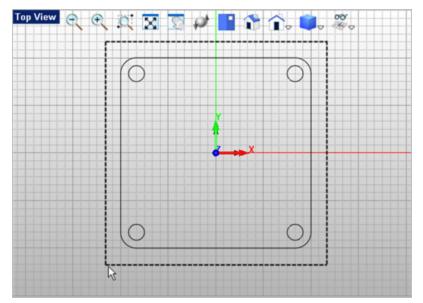
8.2 Extrude the Base Plate



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



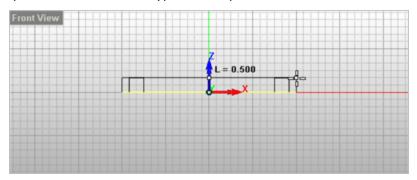
After your drawing is selected, go to the Mesh Modeling tab 2.

Mesh Modeling from



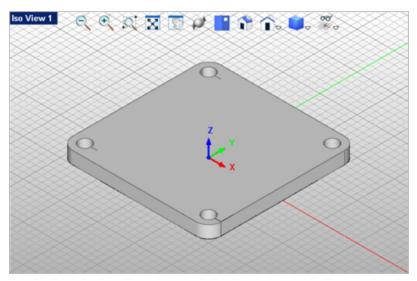
the Ribbon Bar and select the Extrude command.

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 view is shown below.





Save your work by selecting the Save icon from the Quick Access toolbar.

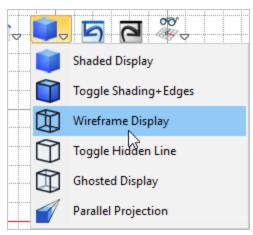


8.3 Model the Tube

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 Top View to activate it.
- 2. From the <u>View Toolbar</u> drop down the Display menu and select Wireframe Display. This will allow us to better see our next drawing.

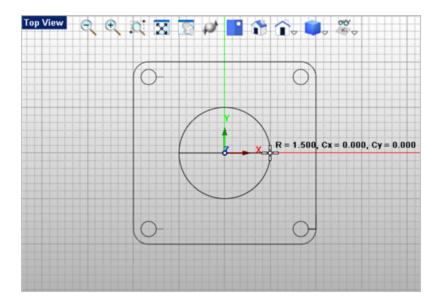


3. Now from the Curve Modeling Curve Modeling tab select the Circle on Point

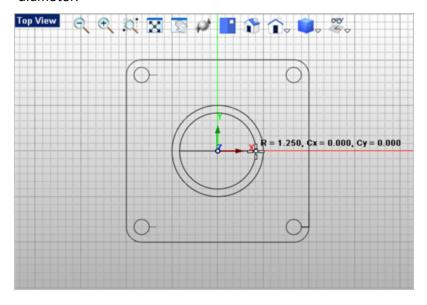


command again.

- 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.



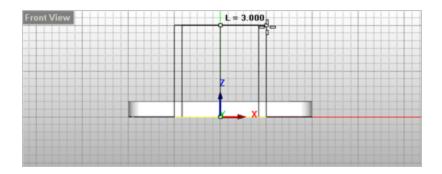
6. Now repeat the command, this time drawing a circle with a diameter of 2.5" in diameter.



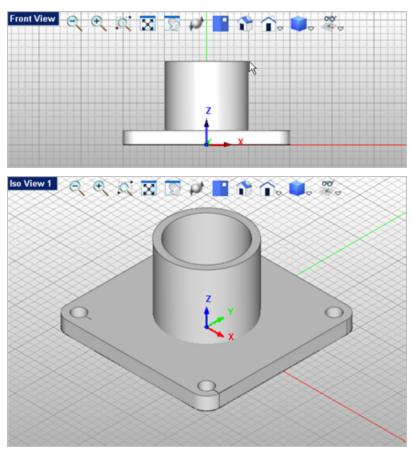
- 7. Once the two concentric circles are drawn, select them both. You can window select them or while holding down the <Ctrl> key, just pick both of them.
- 8. With the two circles selected, go to the Mesh Modeling Mesh Modeling tab and



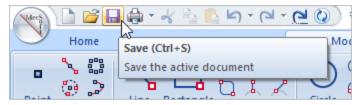
9. Now move the cursor to the Front View and select the grid point located at 3" in the positive Z direction as shown below.



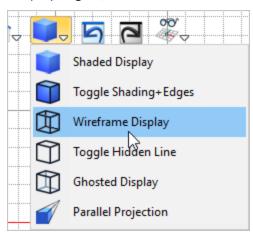
10. The center tube is now created and shown in each window.



11. Save your work by selecting the Save icon from the Quick Access Toolbar.

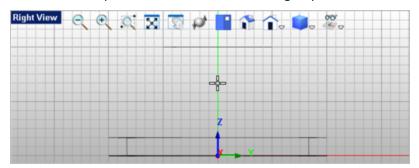


- 12. Activate the Right View by selecting anywhere on the grid.
- 13. From the View Toolbar, drop down the Display menu and select Wireframe Display. Again, this will allow us to better see our next drawing.

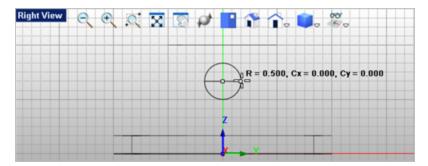




- 14. From the Curve Modeling Curve Modeling tab, select the Circle on Pt command again.
- 15. For the center point of the circle, select the grid point shown below.



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



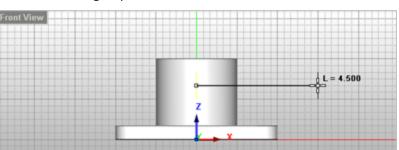
17. Now select the circle you just created, then go to the Mesh Modeling Mesh Modeling

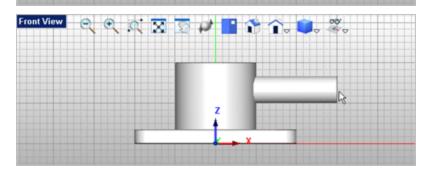


tab and select the Extrude

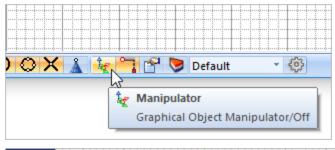
command again.

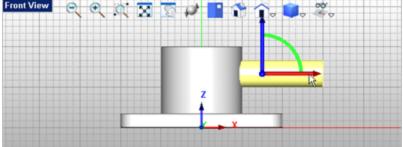
18. For the length of the cylinder move the cursor over to the Front View Front View and select the grid point shown below.



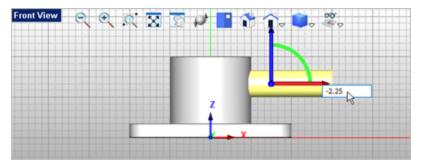


19. Now select the cylinder and toggle the display of the Graphic Manipulator on. This toggle icon is located on the Status Bar at bottom of the display.

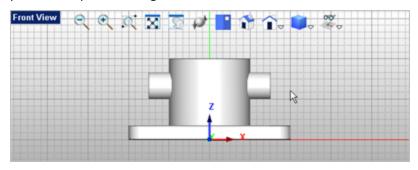




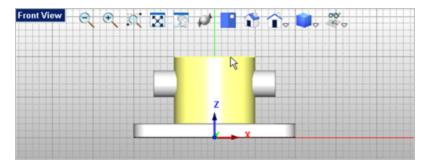
20. With the cylinder selected pick on the red X axis of the Graphic Manipulator to display the input box and enter -2.25 Command and press <Enter> as shown below.



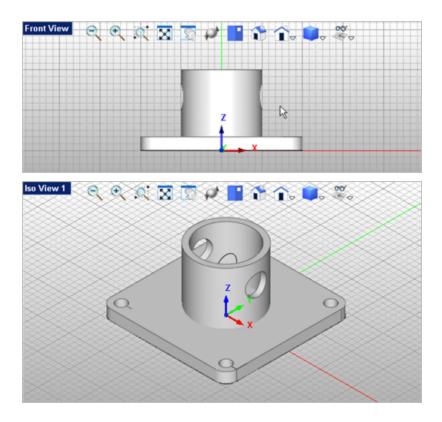
The cylinder is translated to the left negative X direction by 2.25 inches so that it passes complete through the tube on both sides.



- 21. You can toggle the Graphic Manipulator 🗽 off.
- 22. From the Mesh Modeling 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.



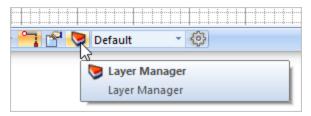
8.4 **Create a New Layer**

Create a New Layer

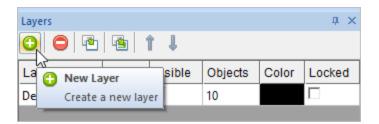
In this step we will create a new layer, turn it off and the move our curves to it to hide them from the display.

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

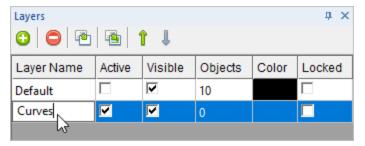




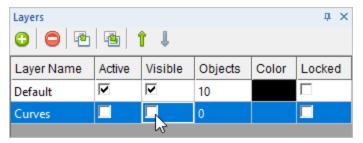
2. At the top of the Layer Manager select the New Layer icon and a new layer will be added to the list.



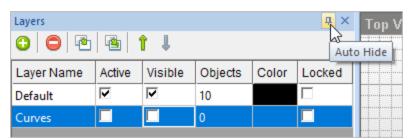
3. Now double-left click on the New Layer name and type in the names "Curves" and press <Enter>.

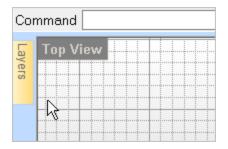


- 4. Now to hide the Curves layer first check the box to make the Default layer the Active layer.
- 5. Then uncheck the box to make the Curves layer hidden.



6. Now let's Auto Hide the Layer Manager. Select the Auto Hide pin icon located at the top right of the Layer Manager and will see that the dialog disappears to the left side of the screen.



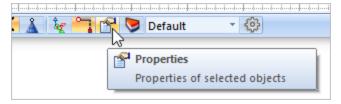


8.5 Edit Geometry Properties

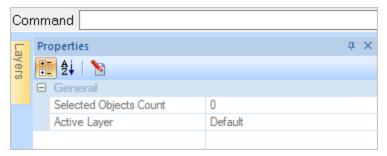
Edit Properties of Geometry

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.

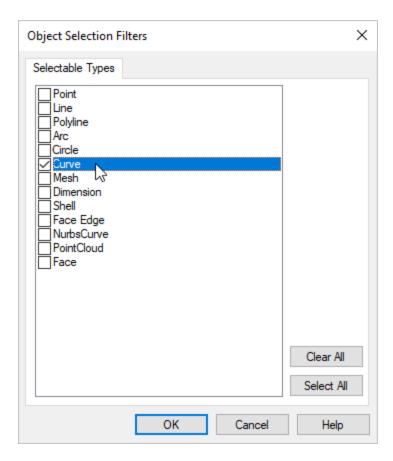
1. Let's start by selecting the Properties icon located on the Status Toolbar at the bottom of the display (just to the left of the Layer Manager local).



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

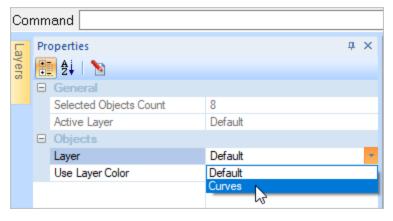


3. Now we will automatically select all of our curve geometry. Go to the Modeling Aids tab and select pick the Select by Type icon. This will display the Object Selection Filters dialog.

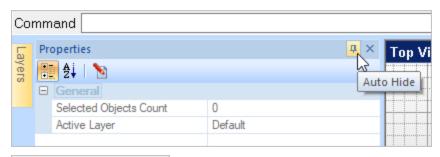


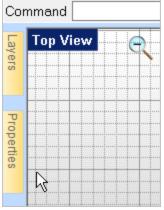
4. From the dialog check the box for Curves and uncheck all other boxes and then pick OK to close the dialog. You will see that all of the curves are highlighted for selection.

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

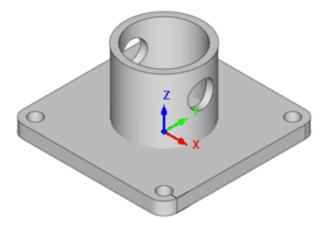


5. Now close the Properties Manager by selecting the Auto Hide pin located at the top right of the Properties Manager.



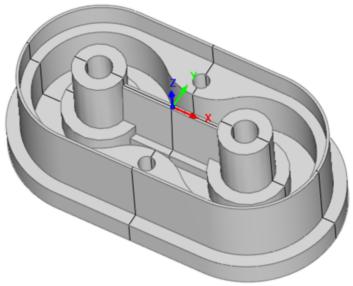


- 6. This exercise is complete and your part now looks like this:
- 7. Save the file as Base_Plate_Completed.vcp.



#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:

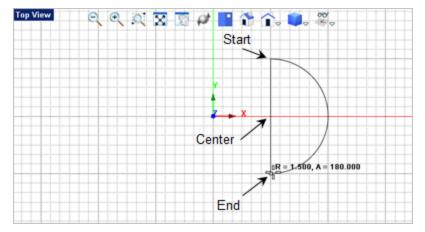
- 1. Extrude the Body.
- 2. Extrude the Flange.
- 3. Extrude the Upper Pocket.
- 4. Extrude the Lower Pocket.
- 5. Revolve the Center Bosses.
- 6. Extrude the Connection Bar.
- 7. Extrude the Connection Wall.
- 8. Extrude Ejector Pin Holes.
- 9. Change Geometry Layer.
- 10. Create Section Curves.

9.1 Extrude the Body

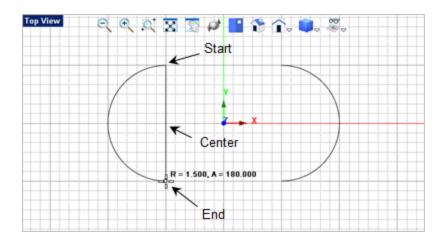
Extrude the Body

In this step we will model the body by creating an extrusion.

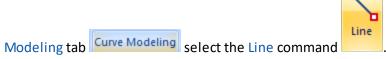
- 1. Activate the Top View Top View tab by selecting anywhere in the viewport.
- 2. Select the Curve Modeling tab Curve Modeling from the top Ribbon Bar.
- 3. Now select the Arc Center, Start, Angle command.
- 4. In the Status Bar make sure the Grid Snap is On ...
- 5. For the center point select the 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.



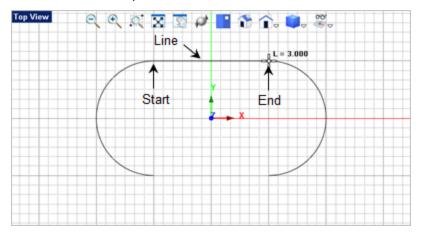
- 7. For the arc end point move the cursor clockwise and pick the end point shown. You will see that the arc is previewed as you move the cursor.
- 8. Now we want to draw another arc, press <Enter> to repeat the command and select the center, start and end points shown below.



9. Now we want to draw a line to connect the two arcs across the top. From the Curve

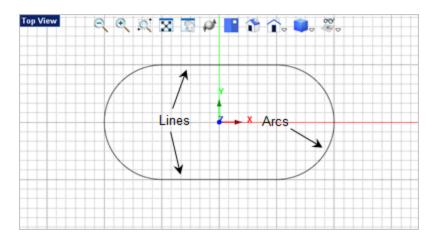


10. For the start point for the line select the grid point located at the top end of the arc of the left and then for the end point of the line select the top end of the arc on the right. Optionally, you can set the Object Snap to End Point from the Status Bar and select the end points of the arcs to 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.

Line



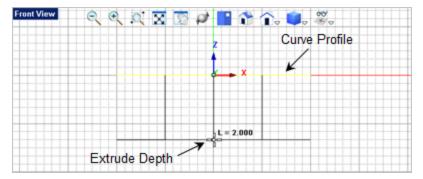
- 12. For modeling its best that you merge your profile geometry into a single curve entity.
- 13. Window select the geometry (both arcs and both lines) and from the Curve

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.

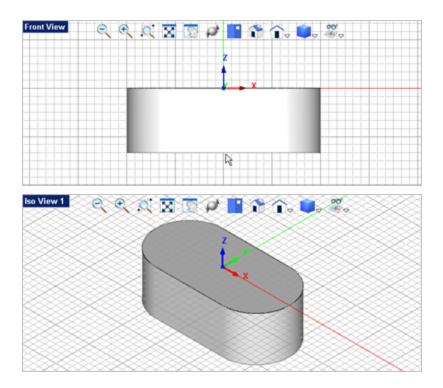
14. Now lets create the body by extruding our profile curve. With the curve selected, change to the Mesh Modeling tab Mesh Modeling and select the Extrude Mesh



15. For the extrusion distance, drag the cursor down to the Front View and select the grid point shown below. Its located at 2.0" below the selected curve.



The Mesh Solid is created and shown below.



9.2 Extrude the Flange

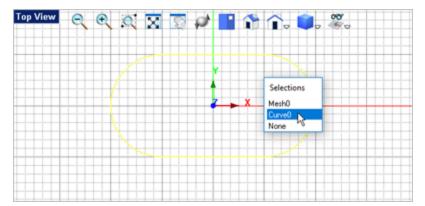


Extrude the Flange

Now we will extrude the flange.

1. Activate the Top View by selection the profile curve you just created. You will notice that a selection list pops up because there is now more than one entity in the same located.

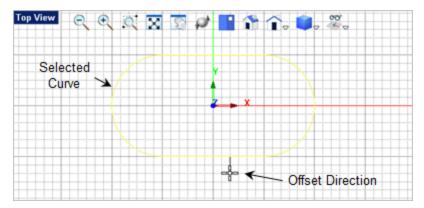
Note: If the Selections menu shown below does not display, go to the Home tab in VisualCAD and select Options. Then under System clear the check box next to Use Preselection Highlight.



- 2. From the selection list pick Curve. The number value to the right of the curve may change depending on when the entity is created.
- 3. With the curve selected, from the Curve Modeling tab Curve Modeling select the



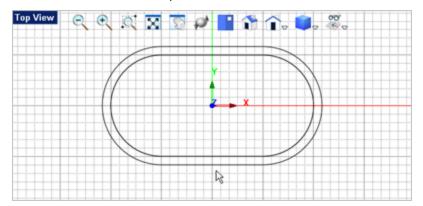
4. If you move the cursor you see that an offset direction is indicated. Move the cursor below and on the outside of the curve as shown below.



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

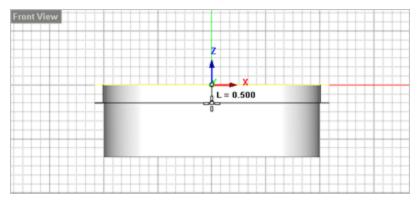
Command and press <Enter> to create the offset curve shown below.



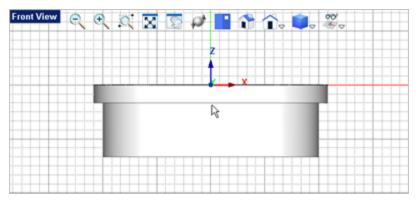
6. Now select the offset curve you just created, go to the Mesh Modeling tab



7. Drag the cursor to the Front View the selected curve. The cursor indicates that the extrusion length is 0.500".



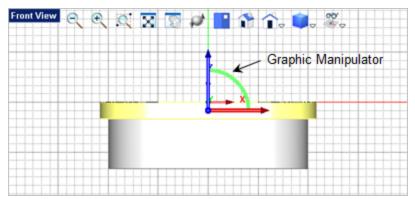
The flange is created and displayed.



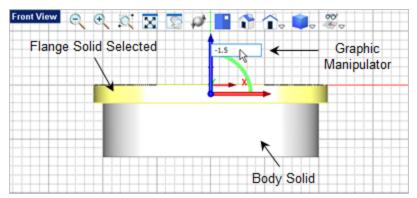
8. Now lets move the flange to the bottom of the body. To do this we will use the Graphic Manipulator. To toggle the Graphic Manipulator on, go to the Status Bar and select the icon .

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.

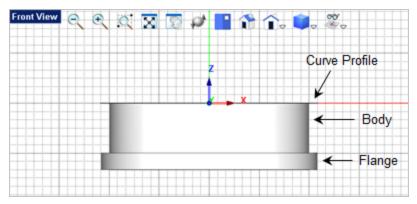
9. Now select the flange and you will see that the Graphic Manipulator is displayed on top of the selected entity.



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



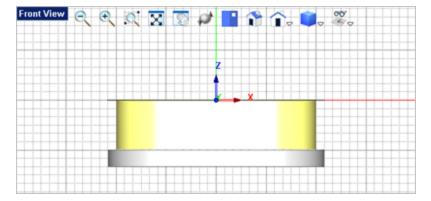
The flange is moved 1.5" in the negative Z direction.



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



13. For the first mesh solid select the body.



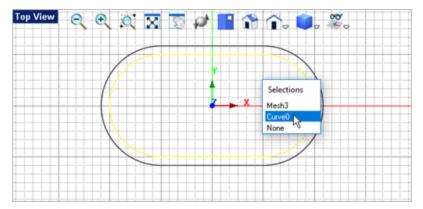
14. For the second mesh solid select the flange. The two will be joined into one mesh solid.

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.

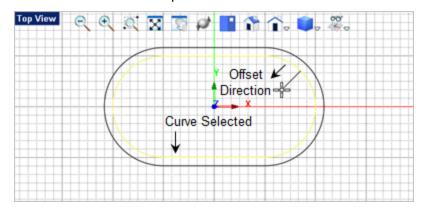
1. From the Top View select the profile curve of the body by clicking on it and selecting Curve from the selection list.

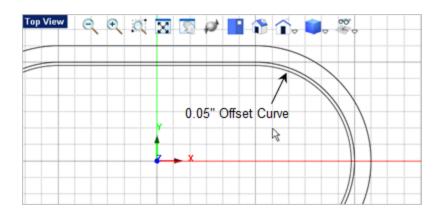


2. With the curve select, go to the Curve Modeling tab Curve Modeling and select the

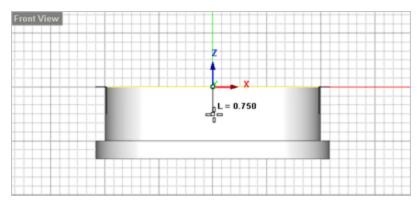


3. Drag the offset direction to the inner side of the curve and then enter 0.05 Command and press <Enter>.

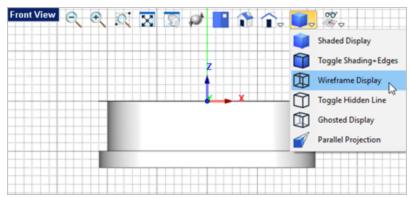




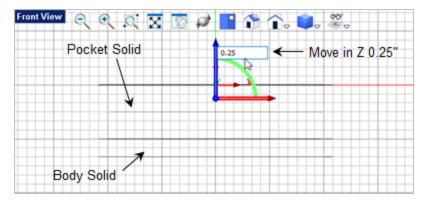
- 4. Now select the offset curve, go to the Mesh Modeling tab Mesh Modeling and select
 - the Extrude Mesh command.
- 5. Drag the cursor down to the Front View on the Front View and select the grid point at 0.750" as shown below.



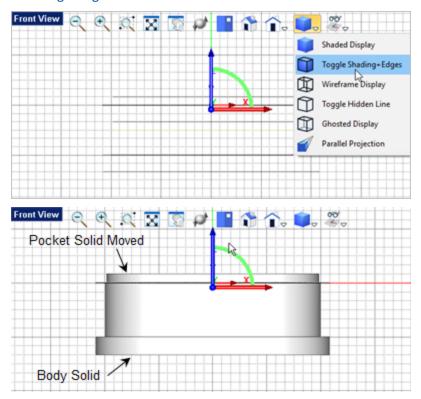
6. Now to see the pocket mesh solid, from the <u>View Toolbar</u> select the <u>Wireframe Display</u> item from the display menu.



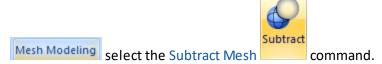
- 7. We want to move the pocket mesh solid so make sure the Graphic Manipulator icon is still toggled on (it's located on the Status Bar remember?).
- 8. With the Front View set to Wireframe display you can see to select the pocket mesh solid. Select it now and the Graphic Manipulator will display on it.
- 9. Select the blue Z Axis arrow and enter 0.25 Command and press <Enter>. The pocket mesh solid will move upward 0.25".



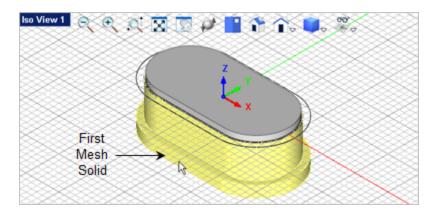
10. From the <u>View Toolbar</u> drop down the display menu and select Toggle Shading + Edges.



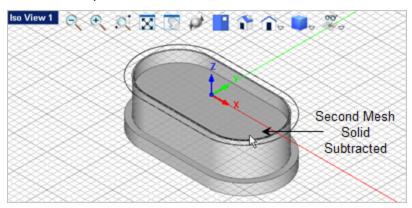
11. Now lets subtract the pocket from the body. From the Mesh Modeling tab



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
Iso View 1



13. Now for the second mesh solid, select the pocket solid and it will be subtracted from the body.



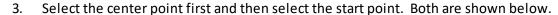
9.4 Extrude the Lower Pocket

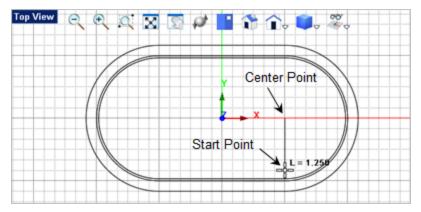


In this step we will create the lower pocket mesh solid and subtract it from the body.

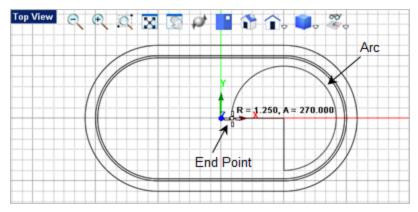
- 1. We will start in the Top View so select anywhere in that view to activate
- 2. Now we will create two partial arcs, each 270 degrees. From the Curve Modeling tab

 Curve Modeling, select select the Arc Center, Start, Angle command again.

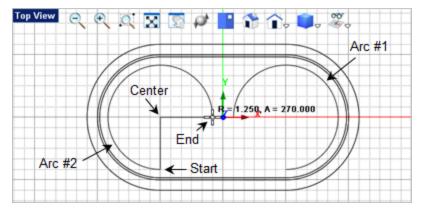




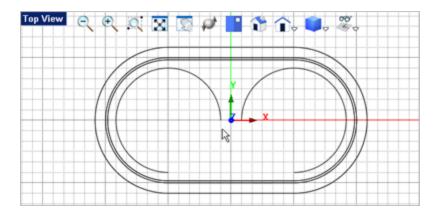
4. Now for the end point of the arc, drag the cursor counter clockwise until the arc preview is at 270 degrees as shown below.



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

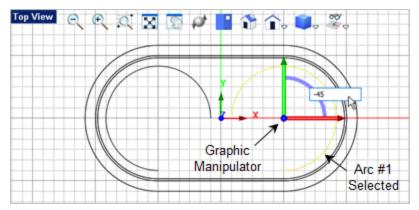


You drawing should now look like this:

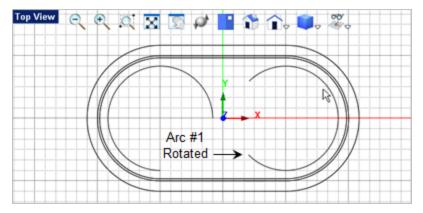


- 6. Now we will rotate each arc 45 degrees to position them properly. Toggle the Graphic Manipulator on , select the first arc.
- 7. This time we will use the Graphic Manipulator's rotate feature. Select the blue Z

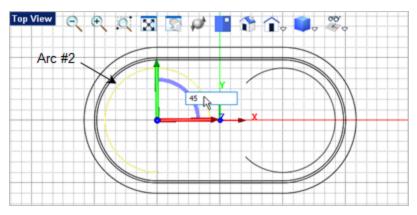
 Axis arc, enter -45 Command and then press <Enter>.



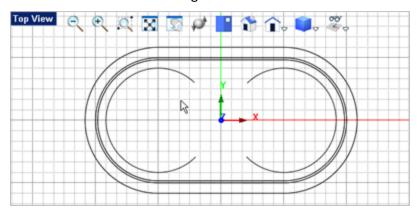
The arc is rotated negative 45 degrees about the Z axis as shown.



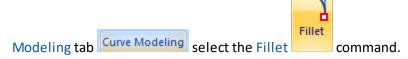
8. Now select the second arc, pick the Z axis arc, enter 45 and press < Enter >.



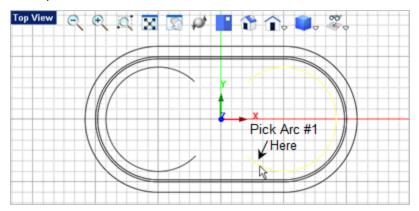
The second arc is rotate 45 degrees about the Z axis as shown.



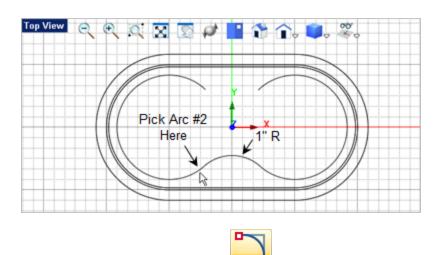
9. To complete the second pocket profile we will fillet the two arcs. From the Curve



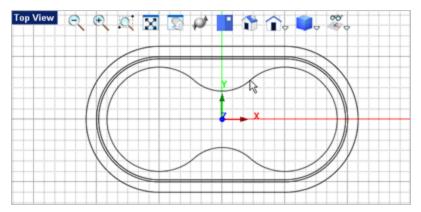
- 10. Enter 1.0 for the fillet radius and press <Enter>.
- 11. Now pick the first arc near the end shown.



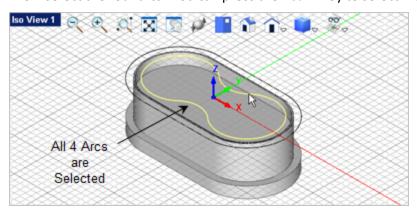
And then pick the second arc near the end shown to create the 1" fillet between the two arcs.



- 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.



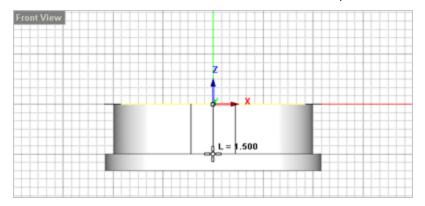
- 15. Now from the Curve Modeling tab Curve Modeling select the Merge Curves command and then right-click or press <Enter> to merge them into one curve.
- 16. With the curve still selected go the Mesh Modeling tab Mesh Modeling and select



the Extrude Mesh

command one again.

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

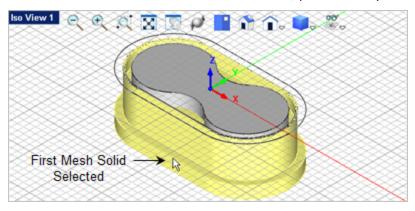


18. Now once again from the Mesh Modeling tab Mesh Modeling select the Subtract

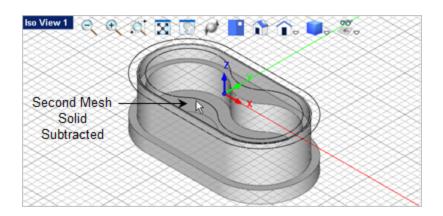


Mesh

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.

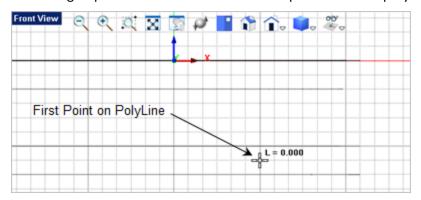


9.5 Revolve the Center Bosses

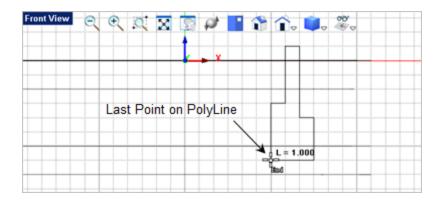
Revolve the Center Bosses

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.

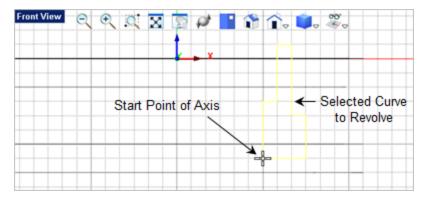


Now select the polyline, and then go to the Mesh Modeling tab Mesh Modeling and 5.

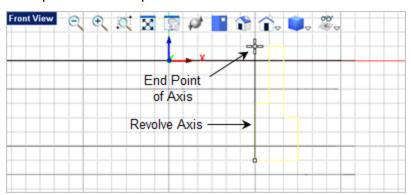


select the Revolve Mesh command. You can toggle off the Graphic Manipulator **a** as we do not need it here.

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

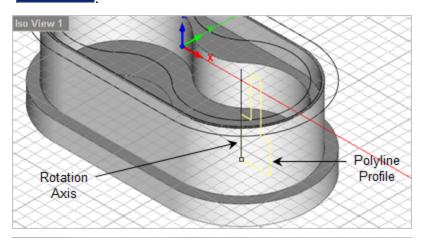


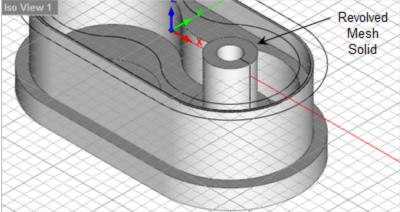
Now pick the second point on the axis of rotation as shown.



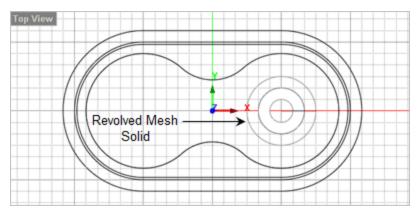
8. The revolved mesh solid is created and can be best viewed from Iso View 1

Iso View 1





9. Now go to the Front View and select the boss mesh solid we just created.

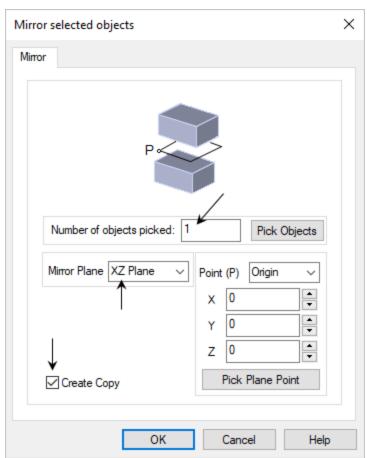


10. With the boss mesh solid selected you will see the Modify / Transform tab

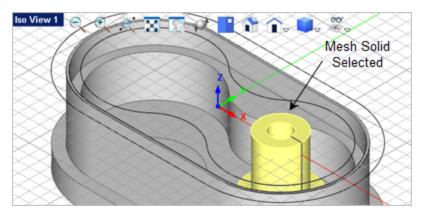
Transform appear on the right end of the Ribbon Bar. It ONLY appears when an entity is selected.



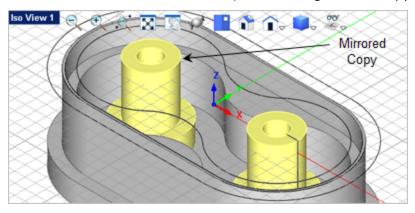
- 11. From the Transform tab, select the Mirror command to display the Mirror selected objects dialog.
- 12. In this dialog, set the Mirror Plane to YZ Plane, check the box next to Create Copy, verify that the Number of objects picked value is 1 and then pick OK.



The mesh solid you selected:

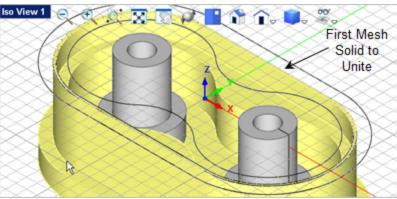


Has now been mirrored about the YZ plane creating a second copy as shown.

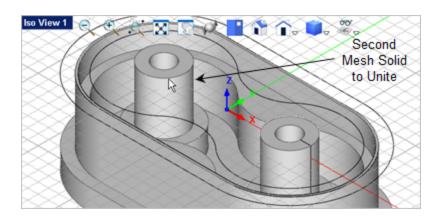


13. Now lets unite the two bosses with the body. From the Mesh Modeling tab,



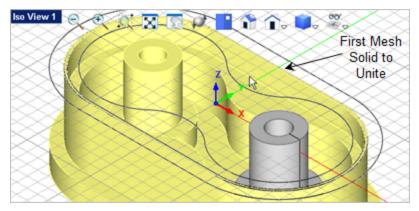


14. Now select the second solid to unite (i.,e. the one of the two bosses we just created.)

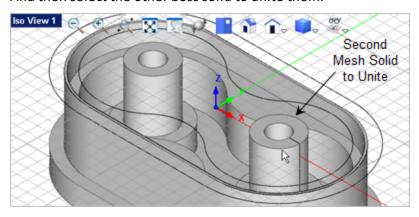


15. Now press <Enter> to repeat the Unite Mesh solid one again.

command and select the body



And then select the other boss solid to unite them.



9.6 Extrude the Connection Bar

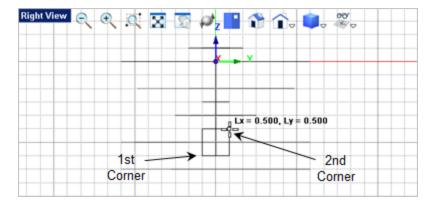
Extrude the Connection Bar

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

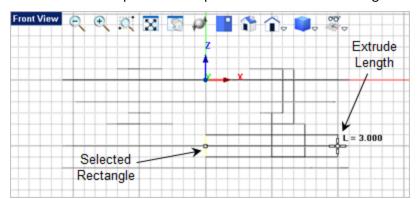
First activate the Right View and then from the Display menu on the View Toolbar, select Wireframe Display.

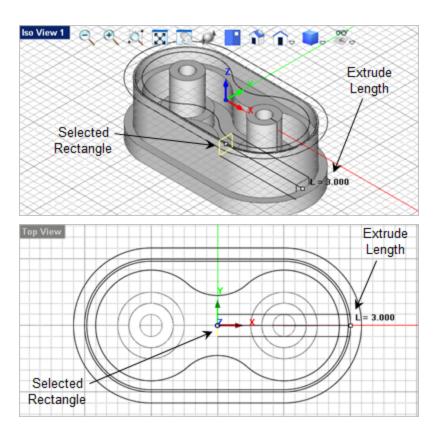


- From the Curve Modeling tab, Curve Modeling select the Rectangle 2. command.
- Select the grid points for the 1st and 2nd corners of the rectangle as shown.



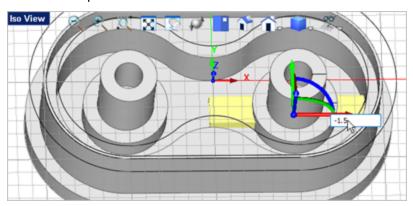
- Now from the Mesh Modeling tab, Mesh Modeling select the Extrude Mesh command.
- Pick the rectangle we just created and drag the cursor over to the Front View Front View and pick the Grid point located 3.00" to the right as shown below.

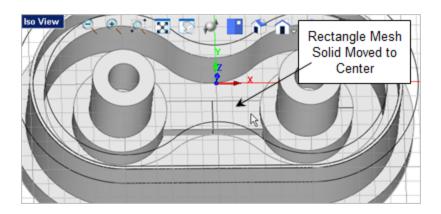




- 6. Now we want to use the Graphic Manipulator again so go to the Status Bar and make sure it is toggled On .
- 7. Select Iso View 1 Iso View 1 and right-click the mouse and drag rotate the part until you get a good view of the Connection Bar and select it.
- 8. From the Graphic Manipulator pick the red X axis arrow, enter -1.5

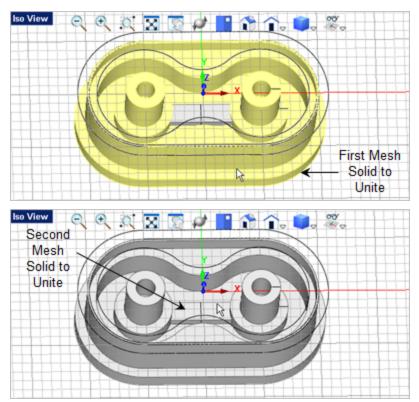
 Command and then press <Enter> to see the Connector Bar move to the left to line up on center between the two bosses as shown.





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

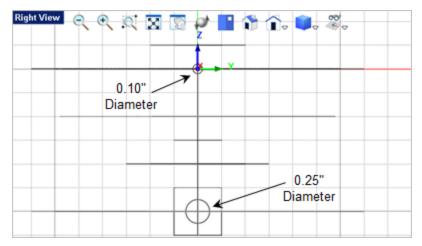
Extrude the Connection Wall

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.

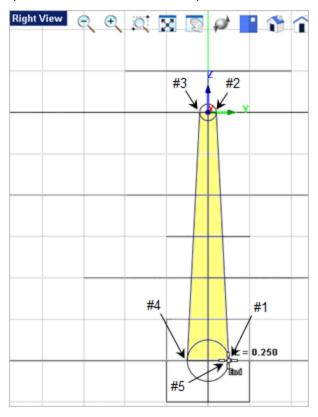


- 2. From the Curve Modeling tab, Curve Modeling select the Circle on Point command.
- 3. For the center point select the grid point shown below.
- 4. Enter 0.25 Command for the diameter and press <Enter>.
- 5. Press <Enter> again to repeat the command. For the second circle select the grid point shown below and then enter 0.10 Command for the diameter and pres <Enter>. We will use these two circles to locate the corners of a polyline that defines the cross-section of the connection wall.

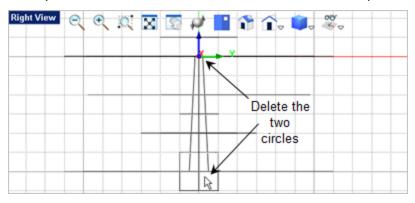


- 6. Now from the Curve Modeling tab, Curve Modeling select the Polyline command.
- 7. From the Status Bar toggle on the Quad Point Snap and toggle off all of the other object snaps.
- 8. We will be drawing a 4-sided closed polyline that encompasses the yellow area shown in the image below. For the start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

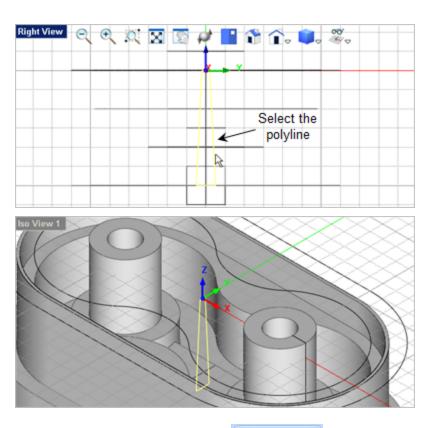
12. To complete the polyline select the Quad point at the #1 location where the polyline started. The Polyline command will end automatically when the end point coincides with the start point.



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



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

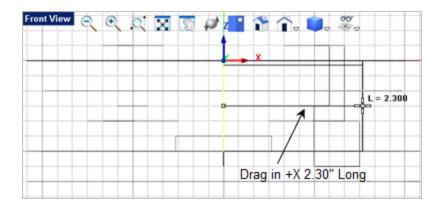


15. Now go to the Mesh Modeling tab Mesh Modeling and select the Extrude Mesh

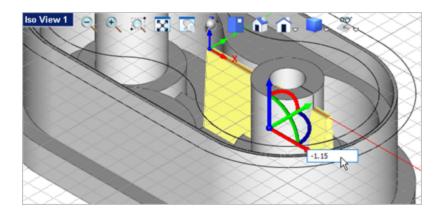


- 16. Drag the cursor to the Front View Front View and drag the length of the extrusion to the right of the center line where the selected polyline resides. Refer to the image below.
- 17. The point we want is not on a grid point, so with the extrusion previewed (do not select a point), go to the command prompt command and type in the distance 2.300 and press <Enter>.

The Connection Wall is created.



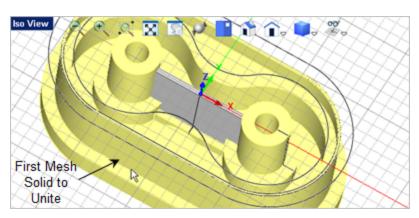
18. Now from Iso View 1 Iso View 1 select the Connection Wall to display the Graphic Manipulator. If you do not see it, make sure it is toggled on from the Status Bar



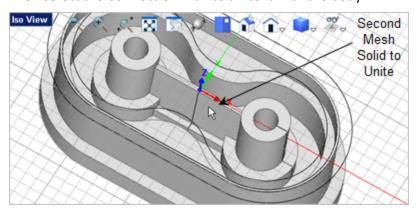
- 19. Pick the red X Axis arrow and type in -1.15 Command and press <Enter>. The Connection Wall should move into place at the center spanning between the two bosses as shown in the image below.
- 20. Now lets unite the Connection Wall with the body. Go to the Mesh Modeling tab



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

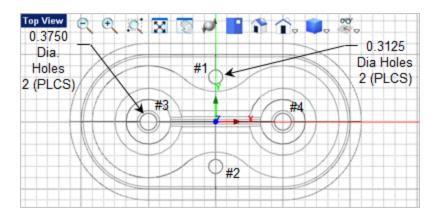
Extrude the Ejector Pin Holes

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.

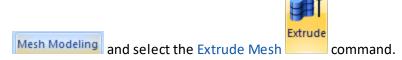
1. Activate the Top View Top View and then go to the Curve Modeling tab



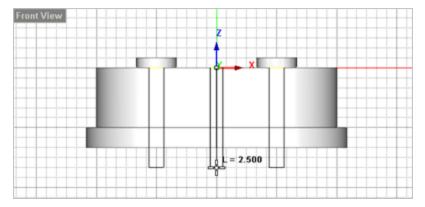
2. We will draw a total of 4 holes. Two will be 0.3125" in diameter (located at #1 and #2) and two will be 0.3750" diameter (located at #3 and #4).



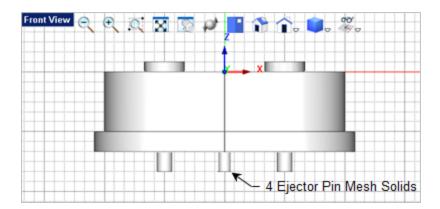
- 3. Select the grid point at the #1 location and then enter 0.3125 in the command prompt Command and then press <Enter>.
- 4. Press <Enter> to repeat the command.
- 5. Select the grid point at the #2 location and press <Enter> again to accept the new default diameter.
- 6. Now press <Enter> to repeat the command, select the grid point at the #4 location and then enter 0.375 in the command prompt Command 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



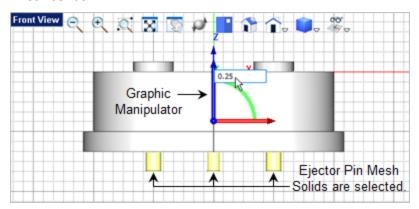
9. Now move the cursor down to the Front View length down past the bottom of the body and pick the grid pint located at 2.500" below the part.



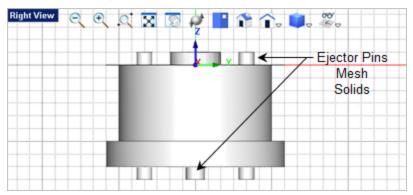
The 4 mesh solids are created:



- 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 Command 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.

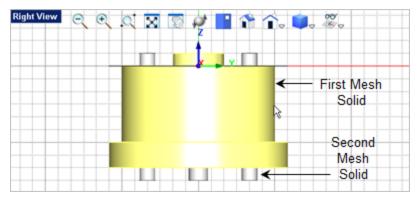


13. Now to subtract the ejector pin mesh solids from the body to create the ejector pin holes go to the Mesh Modeling tab Mesh Modeling and select the Subtract Mesh



command.

14. Select the body as the first mesh solid.

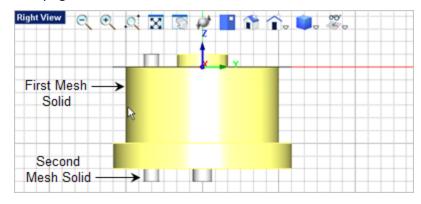


15. Then select the 0.3125" ejector pin mesh solid on the right to subtract it from the body.



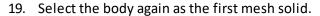
16. Press the <Enter> key to repeat the Subtract Mesh body again as the first mesh solid.

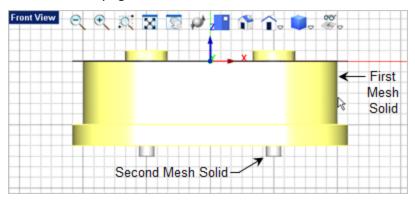
command and select the



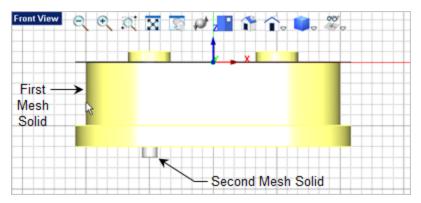
17. Then select the 0.3125" ejector pin mesh solid on the left to subtract it from the body.

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



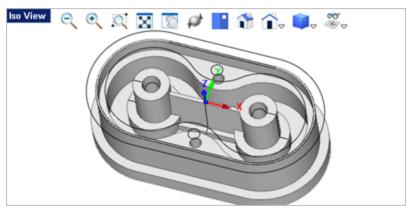


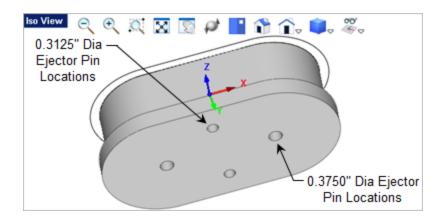
- 20. This time select the 0.3750" ejector pin mesh solid on the right to subtract it from the body.
- 21. Press the <Enter> key again to repeat the command and then select the body mesh solid.



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:



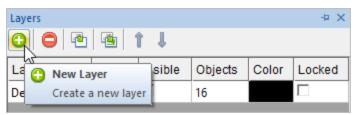


9.9 Change Geometry Layer

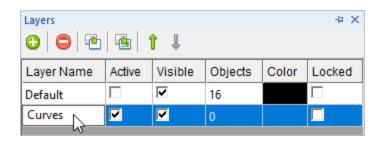
Change Geometry Layer

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

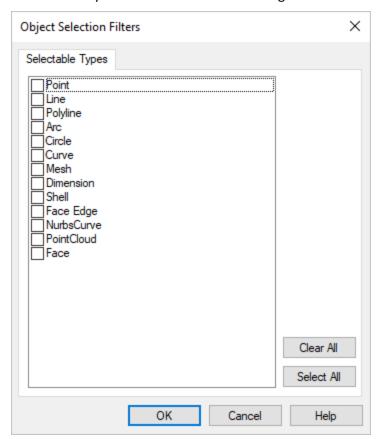
- 1. Display the Layer Manager. You can pick the Layer Manager icon located on the Status Bar or select the Layers flay-out tab located on the left side of the drawing window. If you recall we unpinned the Layer Manager allowing it to collapse to the side of the screen.
- 2. From the Layer Manager select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



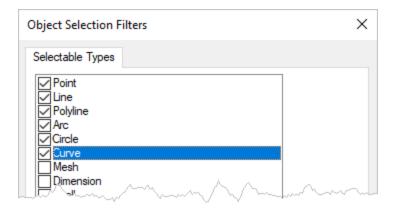
- 3. Now lets rename the new layer to Curves. First double-left-click in the Name field to activate it, and enter Curves as the new layer name.
- 4. Next lets make the Default layer the Active Layer by checking the box in the Active column for the layer Default. The Default layer is now the active layer. Let's also uncheck the box in the visible column for the Curves layer. Now any geometry placed on the Curves layer will be hidden from view. Your Layer Manager should look like the one shown below.



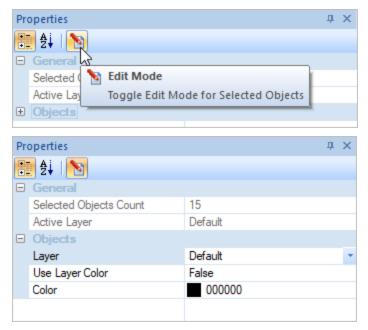
5. Now VisualCAD has a quick way of selecting entities by type. Go to the Modeling Aids Modeling Aids tab and select the By Type icon. This command displays the Object Selection Filters dialog that allows you to check mark the entities that you want to select. That dialog is shown below:



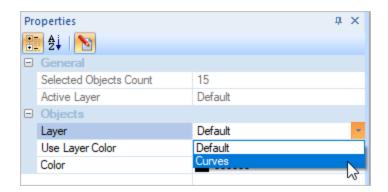
6. Place a check mark in the box for each of the following entity types: Point, Line, Polyline, Arc, Circle, and Curve.



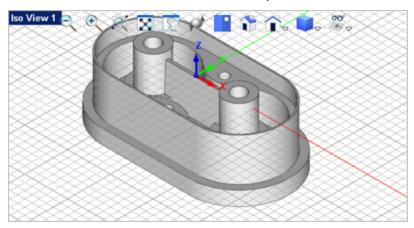
- 7. Now pick the OK button. You will see that all of the reference curves we created in this exercise are highlighted for selection.
- 8. With the entitles selected, select the Properties icon from the Status Bar. Its located just to the left of the Layer Manager icon.
- 9. The Properties Manager will display on the left side of the screen similar to the Layer Manager.
- 10. At the top of the Properties Manager select the Edit icon. This will display additional fields that you can edit.



11. We want to change the layer of the curves so drop down the layer menu by selecting the menu indicator on the right side in the Layer row and select the new layer named Curves that we recently created and hidden.



12. Now when you click anywhere in the drawing window you will see that all of the curves have been moved to the Curves layer and are not hidden from view.



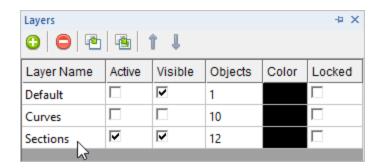
9.10 Create Section Curves

Create Section Curves

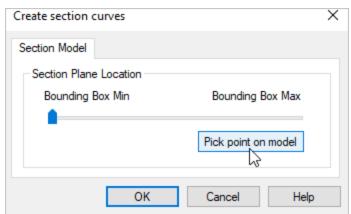
Congratulations, our part is now nearly complete! However, we do want to create cross-section curves that will aid you when it comes time to create a detail drawing of the part.

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.

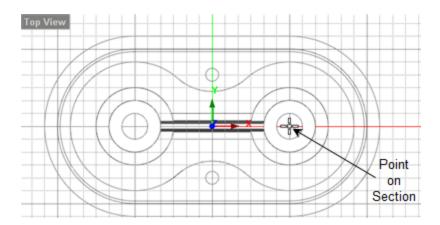
- 1. First lets again create a new layer and make it the active layer so that the section curves will be assigned to that layer automatically when they are created.
- 2. Display the Layer Manager one again.
- 3. Create a new layer and name it Sections and also make it the active layer. The Layer Manager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



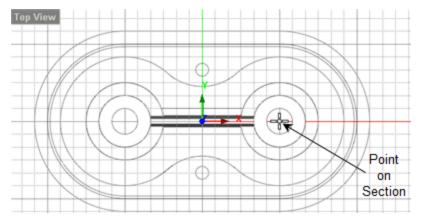
Now go to the Curve Modeling tab Curve Modeling and select the Section Curve command. The Create section curves dialog will display.



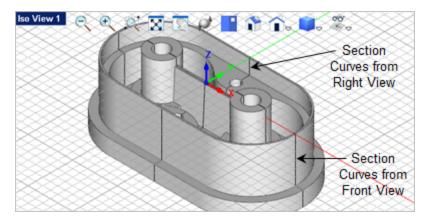
- 5. First select anywhere in the Front View to activate it.
- 6. Now if you move the slider in the dialog you will see the section curves move along the part.
- 7. We want the location of the section to be precise so lets pick the Button that says "Pick point on model".
- 8. Now from the Top View pick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

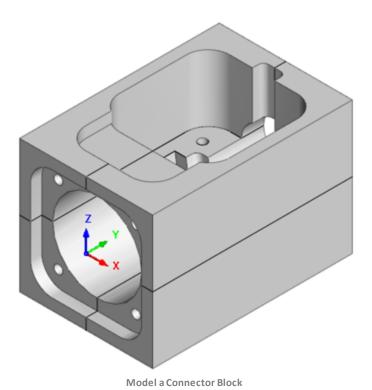


Congratulations!! You modeled your Very first mold insert using VisualCAD!

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.



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.

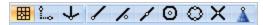
10.1 Model the Body

Model the Main Body

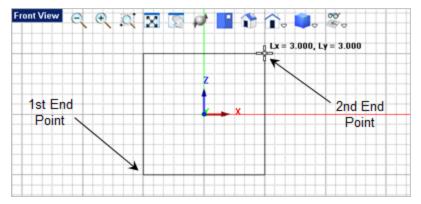
1. Select the Curve Modeling tab Curve Modeling from the top Ribbon Bar.



- 2. Now select the Rectangle command.
- 3. Turn on Grid snap and turn off all other object Snaps. Object Snap toggles are located on the Status Toolbar at the bottom of the display.



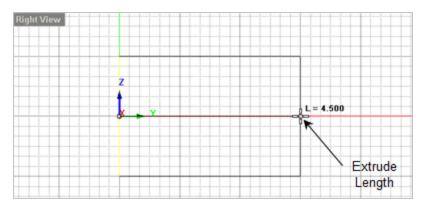
- 4. For the first corner of the rectangle, select the lower left grid point location shown below or type -1.5, 0, -1.5 in the Command Prompt Command and press <Enter>.
- 5. For the second corner of the rectangle, select the upper right grid point shown above or type 3,3 in the Command Prompt Command and press <Enter>. This will create a rectangle 3" square.



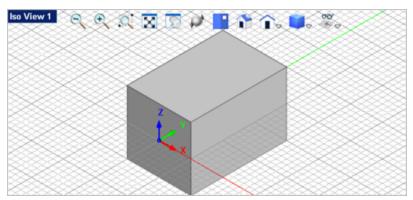
6. Now select the rectangle you just drew and then from the Mesh Modeling tab select



7. Now drag the cursor over to the Right View and select the grid point at 4.5 or just enter 4.5 in the command prompt Command and press <Enter>.

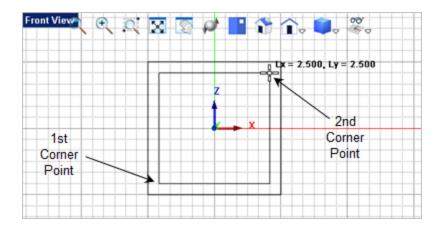


Your part should look like this in Iso View 1:

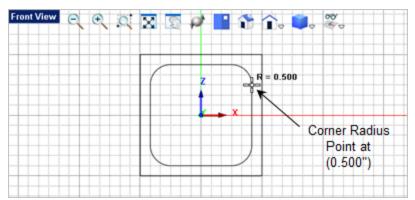


10.2 Model the Front Access

- 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 Command and press <Enter>.

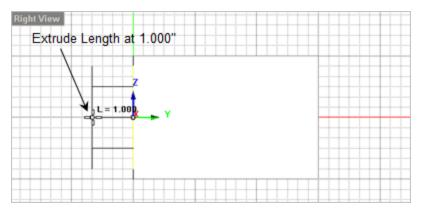


- 4. For the second corner of the rectangle, select the lower left grid point location shown below or type 1.25,1.25,0 in the Command Prompt Command and press <Enter>.
- 5. Now for the corner radius select the Grid point shown below or enter 0.5 in the Command Window Command and press <Enter>.

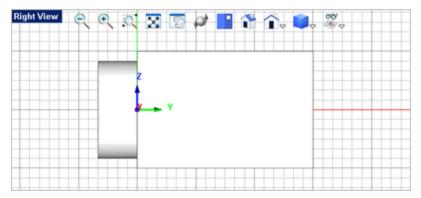


6. Now select the rounded rectangle you just created and from the Mesh Modeling tab,

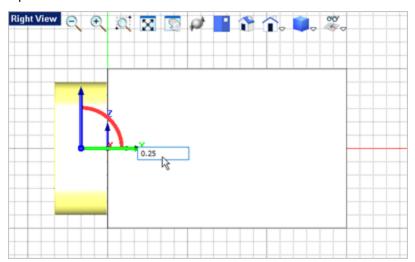
7. Move the cursor over to the Right View and select the Extrude Length at the 1.000 Grid point or enter 1 in the command window Command and press <Enter>.



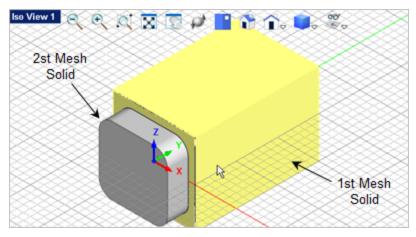
The extruded profile is shown below.



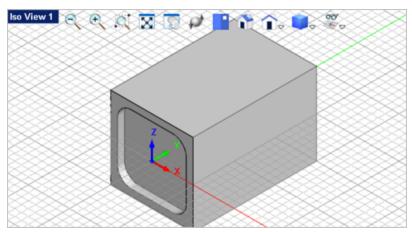
- 8. Now we will move the extrusion so that it intersects the body. Activate the Graphic Manipulator from the Status Bar.
- 9. Select the extrusion mesh solid to display the Graphic Manipulator 🔽.
- 10. Pick the Green Y axis of the Graphic Manipulator and enter 0.25 in the input field and press <Enter>. The extrusion will move in the Y direction the amount specified.



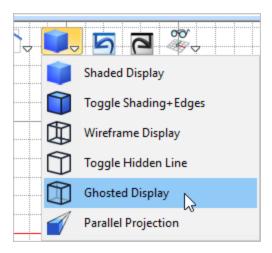
- 11. Press <Enter> again to accept the move.
- 12. Now activate the solview 1 window and from the Mesh Modeling tab select the Subtract Mesh
- 13. For the first mesh solid select the Body.



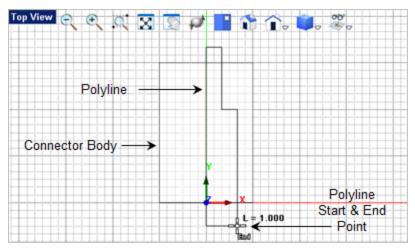
14. For the second mesh solid select the extrusion. The extrusion will be subtracted from the Body as shown below.



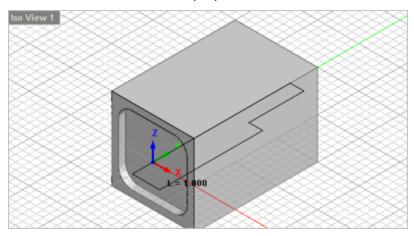
- 15. Now activate the Top View.
- 16. Select the View menu from the View Toolbar and then select Ghosted Display. Note: The View Toolbar will only display in the active viewport.



- 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

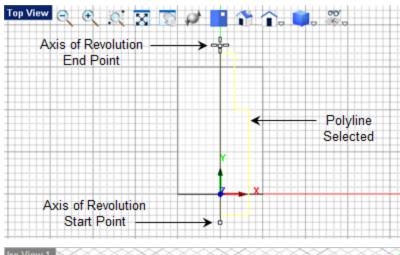
Mesh Modeling

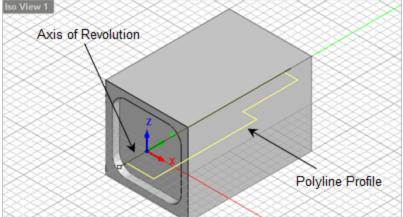


tab select the Revolve Mesh

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.





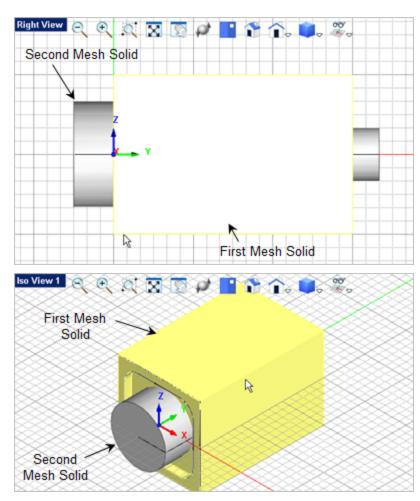
The Revolve mesh solid is created.

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

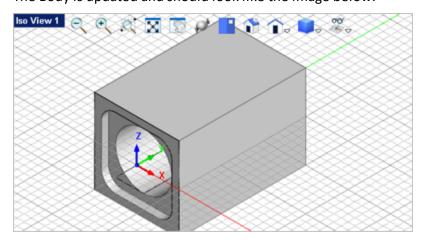


Subtract Mesh

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



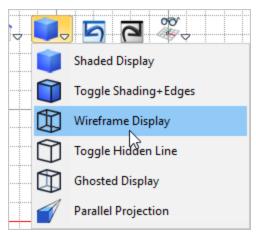
The Body is updated and should look like the image below.



10.3 Model the Top Access

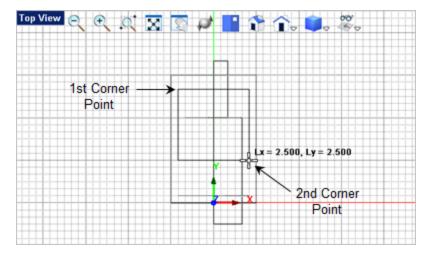
Model the Top Access

- 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.

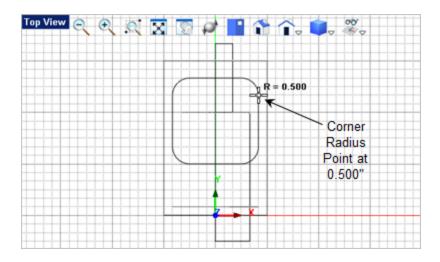


From the Curve Modeling tab select the Rounder Rectangle Command.

3. For the first corner of the rectangle, select the upper right grid point location shown below or type 1.25,1.25,0 in the Command Prompt Command and press <Enter>.



- 4. For the second corner of the rectangle, select the lower left grid point location shown below or type 1.25,1.5,0 in the Command Prompt Command press <Enter>.
- 5. Now for the corner radius select the grid point shown below or enter 0.5 in the Command Window Command and press <Enter>.



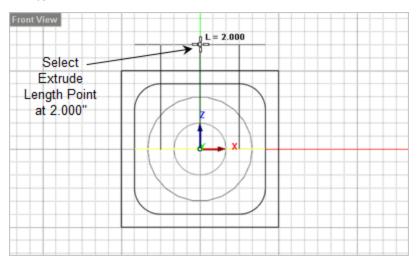
6. Now select the rounded rectangle you just created and from the Mesh Modeling tab,



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 Command and press <Enter>.

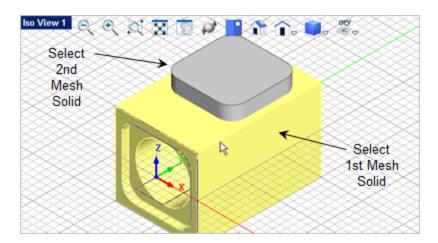


8. Now activate Iso View 1 and from the Mesh Modeling tab select the Subtract Mesh

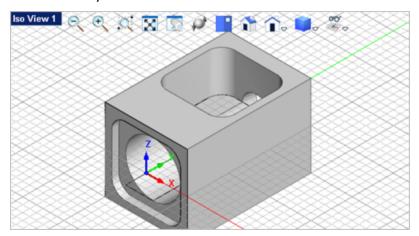


command.

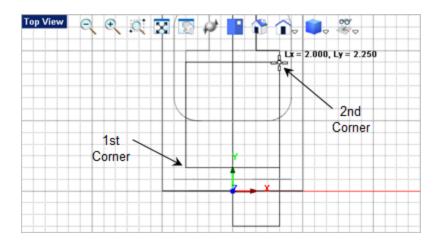
9. For the first mesh solid select the Body.



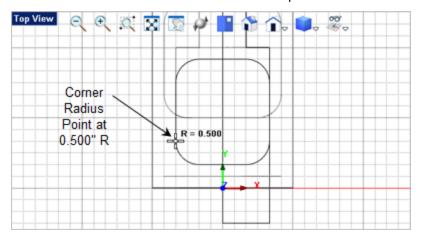
10. For the second mesh solid select the extrusion. The extrusion will be subtracted from the Body as shown below.



- 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 Command and press <Enter>.



- 15. For the second corner of the rectangle, select the upper right grid point location shown above or type 1,2.75,0 in the Command Prompt Command and press <Enter>.
- 16. Now for the corner radius select the grid point shown below or enter 0.5 in the Command Window and press <Enter>.

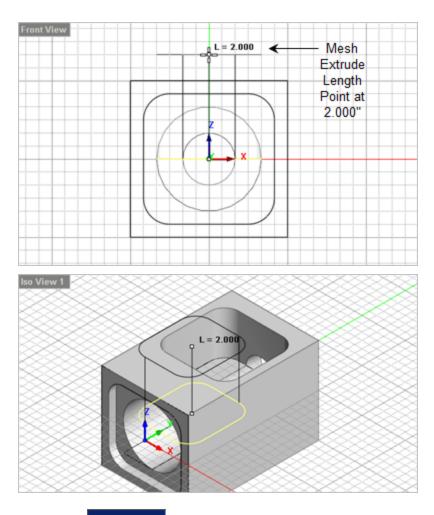


17. Now select the rounded rectangle you just created and from the Mesh Modeling tab,

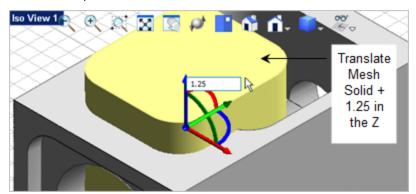
18. Move the cursor down to the Front View and select the Extrude Length at the 2.000 Grid point or enter 0,0,2 in the command window Command and press <Enter>.

command.

select the Extrude Mesh

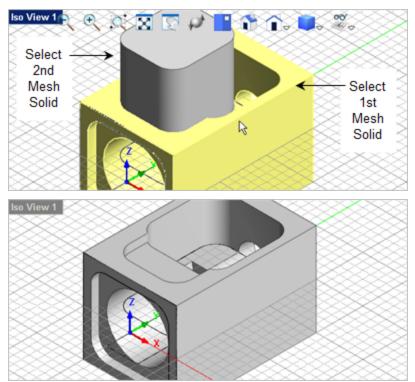


- 19. Now from Iso View 1, select the extrusion you just created to display the Graphic Manipulator .
- 20. Click on the Blue Z Axis arrow and enter 1.25 Command to translate the extrusion upward as shown in the illustration below.





- 21. From the 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



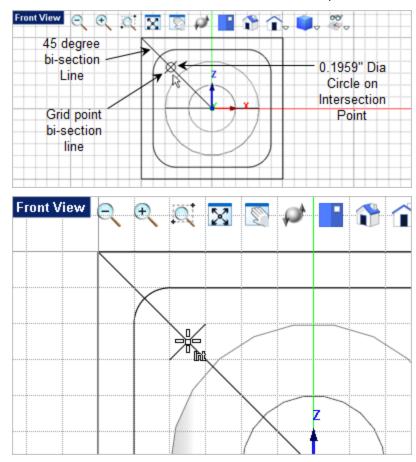
Draw the two lines shown below. The first starts at the center and ends at the top left corner. The second passes diagonally across two grid points bisection the first line at 90 degrees.



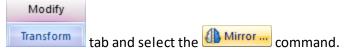
Now from the Curve Modeling Curve Modeling tab select the Circle

command.

For the circle center, activate the Intersect Object Snap K 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.

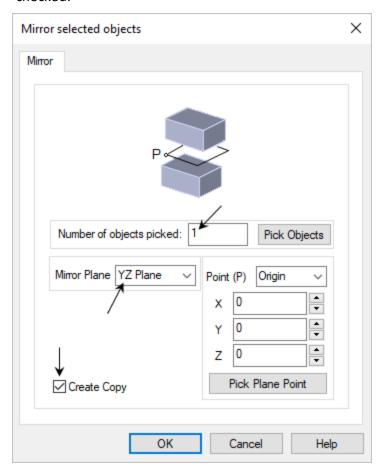


- For the circle diameter enter 0.1959 in the Command Window Command and press <Enter>. This is the #7 drill diameter for a 1/4-20 tap.
- Now you need 4 of these holes. Let's use the mirror command to create the other three. First select the circle you just created, then go to the Modify / Transform

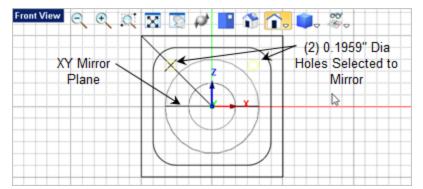


Note that the Modify / Transfer tab ONLY appears when an entity is selected!

8. From the Mirror selected objects dialog, the Number of selected objects should be 1, The Mirror Plane should be YZ Plane and box next to Create Copy should be checked.

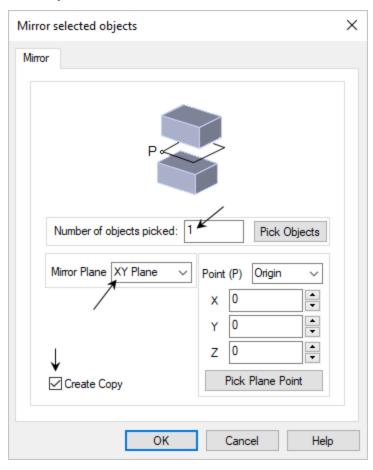


9. Pick OK from the dialog and the circle will be mirrored and both circles should be selected automatically. If not, select both of the circles and pick the command again.



 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.

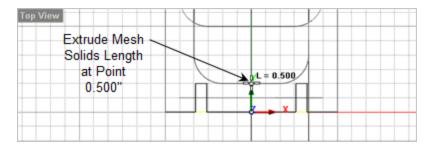
Note: You do not have to select the geometry first, as you can see this dialog has a Pick Objects button.



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



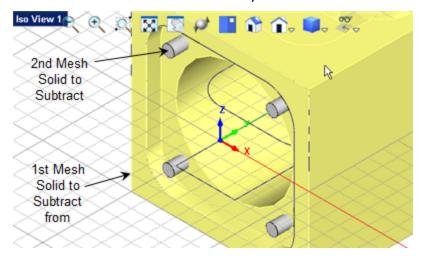
- 12. Now go to the Mesh Modeling tab and select the Extrude Mesh
- 13. For the extrude length move the cursor to the at 0.500" in the positive Z direction or simple enter 0.5 in the command window and press <Enter>.



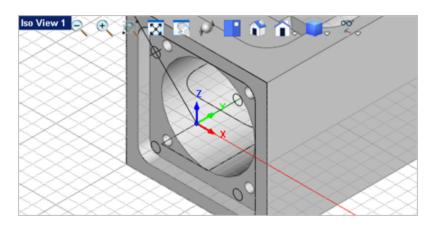
14. Now let's subtract the extrusions from the body to create the holes. Go to



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.

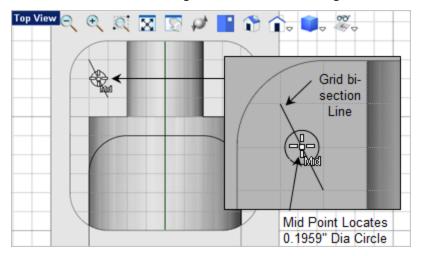


- 19. Now we have 2 additional holes of the same diameter located in the top access. Activate the Top View
- 20. Let's change the display mode for the Top View. Select the Display menu from the View Toolbar in the Top View and select Toggle Shaded + Edges.
- 21. To locate the first hole, go to the Curve Modeling Curve Modeling tab and select the



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

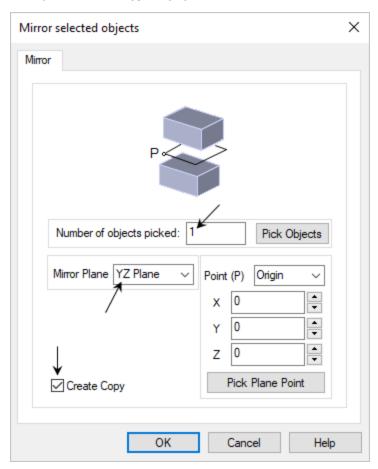
command.

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

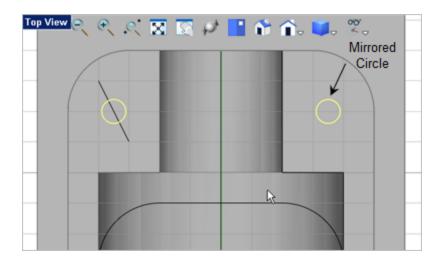


- 25. For the center point of the circle select the mid point of the line you just created shown in the illustration above.
- 26. For the circle diameter enter 0.1959 in the Command Window and press <Enter>.
- 27. Now select the circle you just created and go to the Mirror command.

Modify



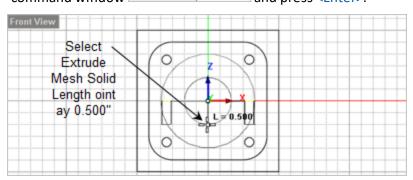
- 28. From the Mirror select objects dialog the Number of selected objects should be 1, the Mirror Plane should be YZ Plane and the Create Copy box should be checked.
- 29. Now pick OK to mirror the circle.
- 30. Both circles should now be selected. If not select them both now.



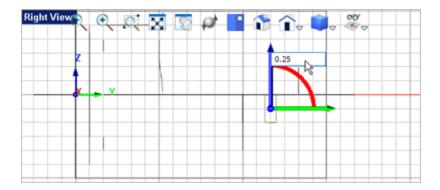
31. With the circles select go to the Mesh Modeling tab and select the Extrude Mesh



- 32. From the Status Bar activate the Grid Object Snap and deactivate the others.
- 33. Move the cursor down to the extrusion length to 0.500" in the negative Z direction or simply enter 0.5 in the command window 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".



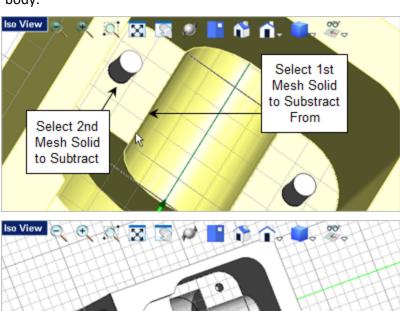
36. Now subtract them from the body. From the Curve Modeling Curve Modeling tab



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.



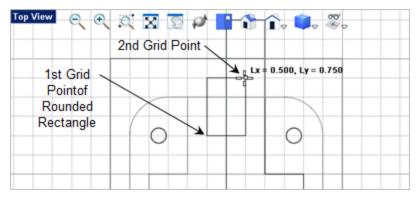
13

10.5 Modify the Top Access

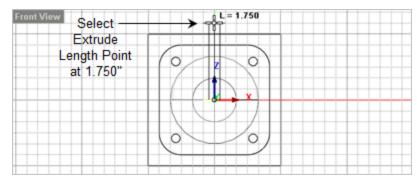
Modify the Top Access

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.

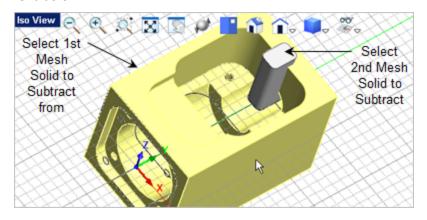


- For the corner radius, enter 0.125 in the command window Command and press <Enter>.
- 5. Now select the rectangle and from the Mesh Modeling tab select the Extrude Mesh command.
- 6. Move to the Front View and drag the extrude length to 1.750".



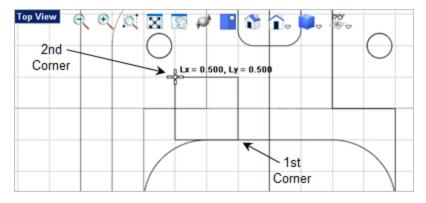


- 7. Now go to the Mesh Modeling tab and select the Subtract Mesh command again.
- 8. For the first mesh solid select the body. For the second mesh solid select the extrusion



9. For the second modification, got to the select the Rectangle command.

10. Select the two corner grid points shown below.

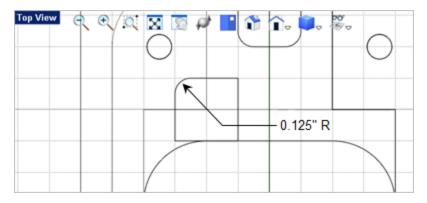


11. Now we need to fillet one of the corners. From the Curve Modeling tab select the

Fillet Curve command.

12. In the command prompt command enter 0.125 for the fillet radius and press <Enter>.

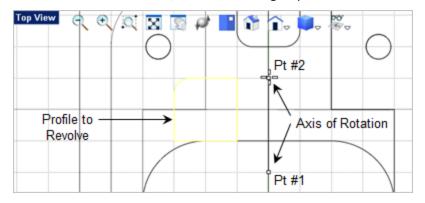
13. Now select the top and left side of the rectangle to fillet the upper left corner as shown below.



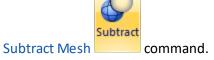
14. We need to revolve this, so select the closed rectangular profile, go to the



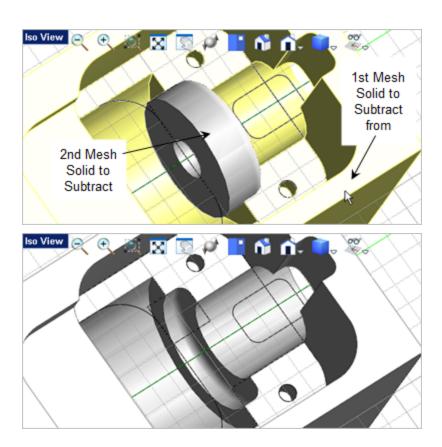
15. For the axis of revolution, select the two grid points shown below.



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



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

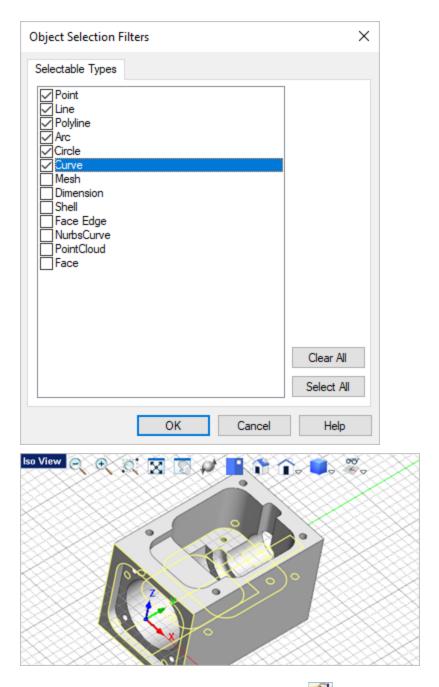


10.6 Create Section Curves

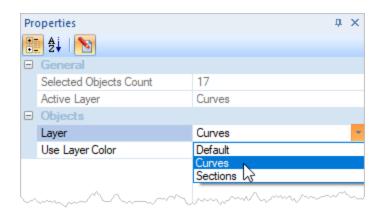
Create Section Curves

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.

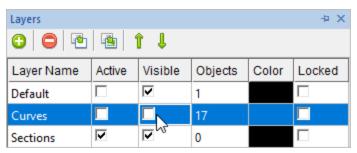
- First lets move all of our curves onto a separate layer and hide them. From the Modeling Aids tab, select the Selection pane of the Modeling Aids tab.
- From the Object selection filters dialog, check the boxes next to Point, Line, Polyline, Arc, Circle and Curve and then pick OK. All of the curve related entities will now be selected.



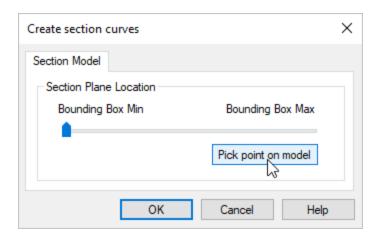
3. From the Status Bar select the Properties icon to display the Properties Manager.



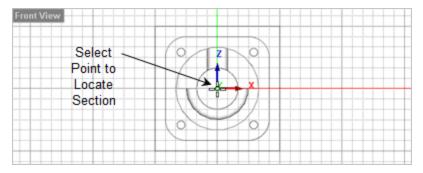
- 4. This dialogs reports that a total of 17 entitles are selected. In the Objects section drop down the menu for Layer and select Curves from the list of layers.



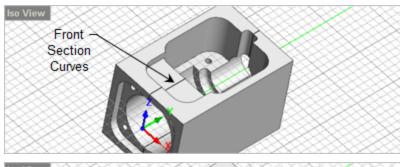
- 6. From the Layer Manager, under the Active column, check the box to make the layer named Sections the active layer.
- 7. You can close the Properties and Layer managers by allowing them to auto-hide or pick the "X" icon to close them.
- 8. Now we're ready to create the section curves. First activate the 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.

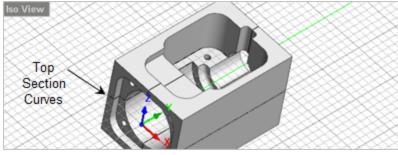


- 10. If you move the slider you will see a highlight of the section display in all views.
- 11. Now from the dialog select the Pick point on model button. The dialog minimizes allowing you to select a point. From the Front View, select the XY origin grid point to locate the section.

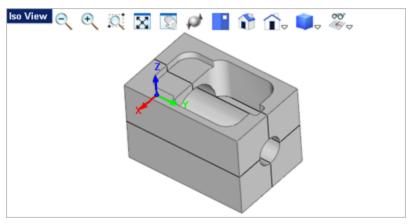


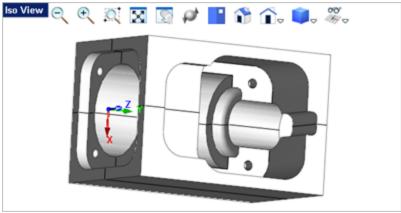
- 12. The dialog reappears and you can pick OK to close it.
- 13. Now activate the Right View
- 14. Go to the Curve Modeling tab and select the command again.
- 15. From the Create section curves dialog, select the Pick button and then select the same grid point, 0,0,0 origin in the Front View.
- 16. You will now notice that section curves were created in both the XY Plane and the YZ Plane.





Congratulations!!! You completed the exercise!!

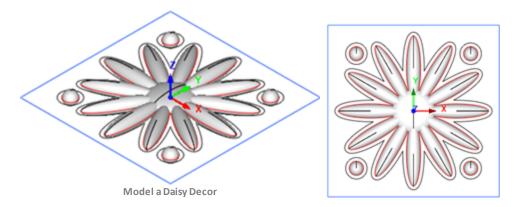




17. Save the file as Connector_Block_Completed.vcp.

#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:

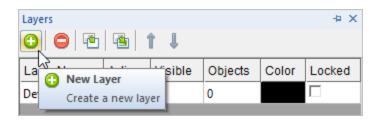
- 1. Create new Layers.
- 2. Model & Scale the Body.
- 3. Model & Scale the Pedal.
- 4. Ghosted Display Mode.
- 5. Polar Array the Pedal.
- 6. Additional Spheres Accents.
- 7. Create & Trim Section Curves.
- 8. <u>Create Offset Curves.</u>
- 9. Create the Stock Boundary.

11.1 Create new Layers

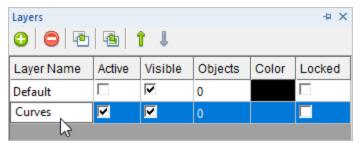
Create new Layers

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

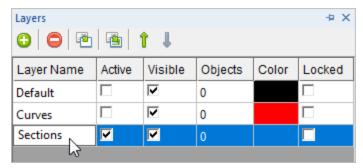
- 1. Display the Layer Manager. You can pick the Layer Manager icon located on the Status Bar.
- 2. From the Layer Manager select the Add Layer icon to add a new layer.



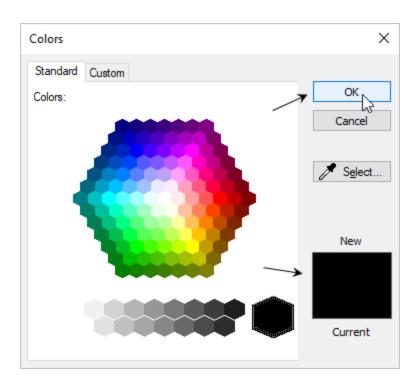
3. Now lets rename the new layer to Curves. First double-left-click in the Name field to activate it, and enter Curves as the new layer name.



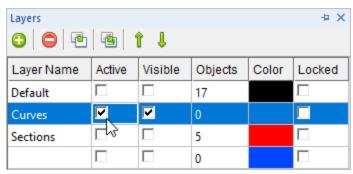
4. Repeat the procedure to create a new layer, this time named Sections.



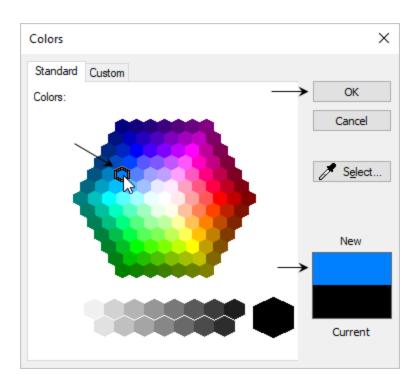
- 5. Now repeat the procedure again, this time named Stock.
- 6. Now let's change the color of a couple of the new layers. In the Color field left-click on the color box for the layer named Curves. This will display the Colors dialog.
- 7. Accept the default color Black and then pick OK to close the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



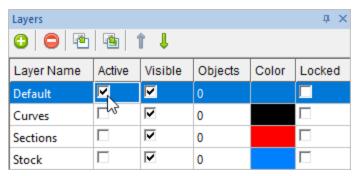
- 8. If the Layer Manager keeps Auto-Hiding to the left side of the display, select the Auto Hide pin icon at the top of the Layer Manager. You can also see that the layer named Curves was changed to color black.
- 9. Now let's change the color of another layer. In the Color field left-click on the color box for the layer named Stock. Again, this will display the Colors dialog.



 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.



11.2 Model & Scale the Body

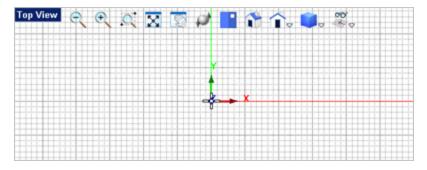
Model & Scale the Body

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.

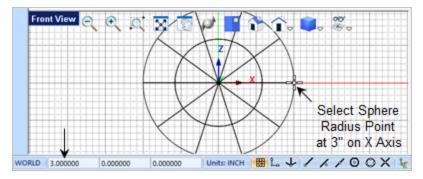
- 1. Activate the Top View Top View tab by selecting anywhere in the viewport.
- 2. Select the Solid Modeling Solid Modeling tab from the top Ribbon Bar.



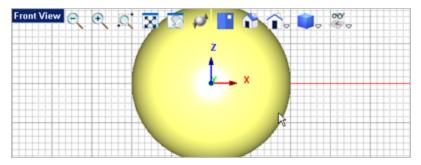
- 3. Now select the Sphere command.
- 4. In the Status Bar make sure the Grid Snap is On .
- 5. For the center point select the grid point at (0,0,0) or just enter the coordinates 0,0,0 in command input bar Command.



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



7. Select the solid sphere you just created.

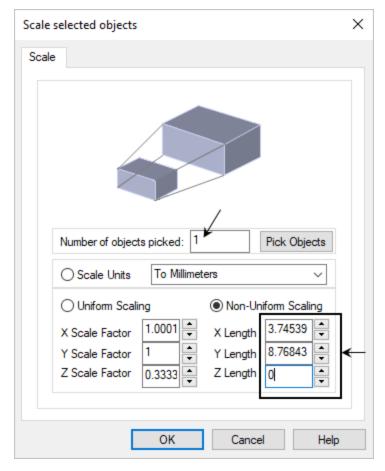


8. You see the Modify / Transform tab appear on the right end of the Ribbon Bar. It ONLY appears when an entity is selected. Select the Transform tab.

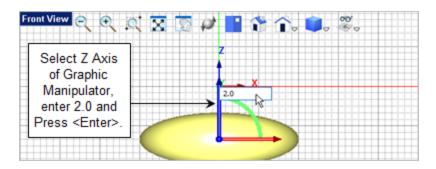
Modify



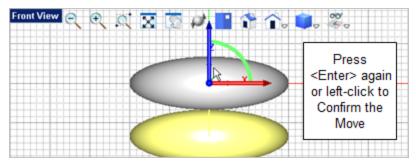
- 9. Select the Scale Scale command.
- 10. From the Scale Selected Objects dialog, select the Non-Uniform Scaling option.



- 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.

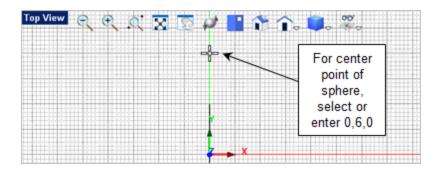


11.3 Model & Scale the Pedal

Model & Scale the outer Pedal

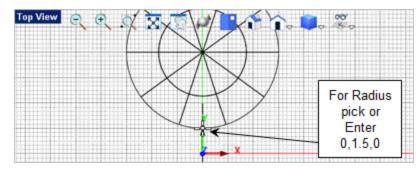
In this step you will model and scale one of the outer pedals, again using a solid sphere and non-uniform scaling. Note: You can replace the Solid commands with Mesh commands if desired.

- 1. Activate the Top View Top View tab by selecting anywhere in the viewport.
- 2. Form 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>.

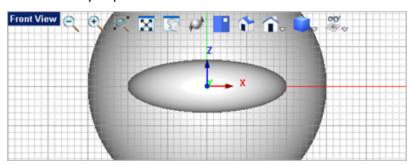


For the sphere radius, enter 4.5 in the command input window Command 5. and press <Enter> or select the grid point located at 0,1.5,0.

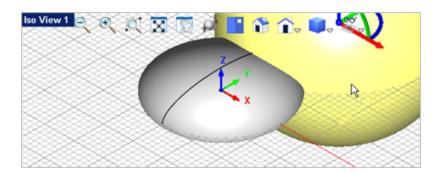




Here is what you part should look like from the Front View:

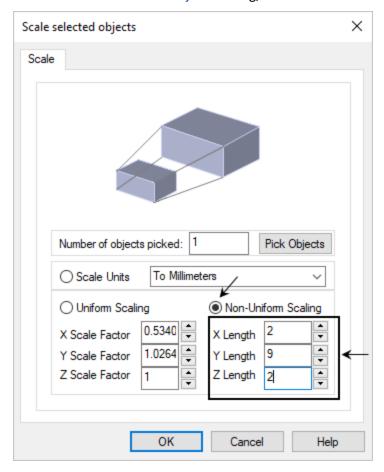


From Iso View 1, Iso View 1 select the solid sphere you just created. 6.

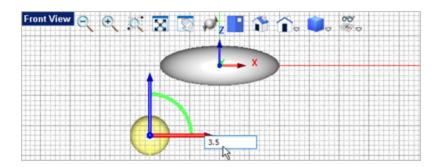




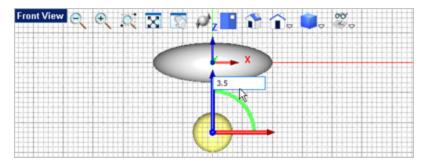
- 7. Select the Transform tab from the ribbon bar.
- 8. Select the Scale Scale command.
- 9. From the Scale Selected Objects dialog, select the Non-Uniform Scaling option.



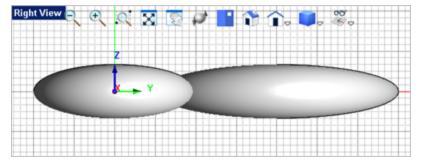
- 10. Set the X Length to 2, Y Length to 9 and the Z Length to 2.
- 11. Pick OK and the sphere is scaled as shown below.
- 12. Now press the <Enter> key to accept the modification.
- 13. Let's move the scaled sphere into position along the Y Axis. First make sure the Graphic Manipulator is On. The icon is located on the Status Bar.
- 14. From the Front View Front View, select the Red X arrow to display the input field, enter 3.5 and press <Enter>.



15. Press <Enter> again to confirm the move and the sphere will move up to the Z 0 location as shown below.



- 16. Now select the sphere again.
- 17. Select the Blue Z arrow to display the input field, enter 3.5 Command and press <Enter>.
- 18. Press <Enter> again to confirm the move and the sphere will move to the X 0 location. If you activate the Right View your part should look like the image below.



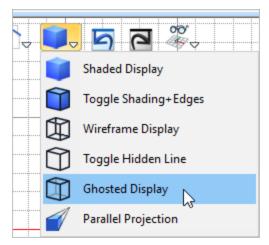
11.4 Ghosted Display Mode

Change the Top View to Ghosted Display

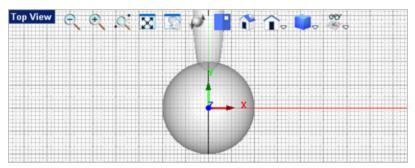
For spheres drawn from the top view, it is best viewed using the ghosted display.

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:

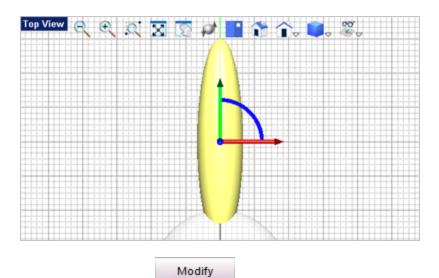


11.5 Polar Array the Pedal

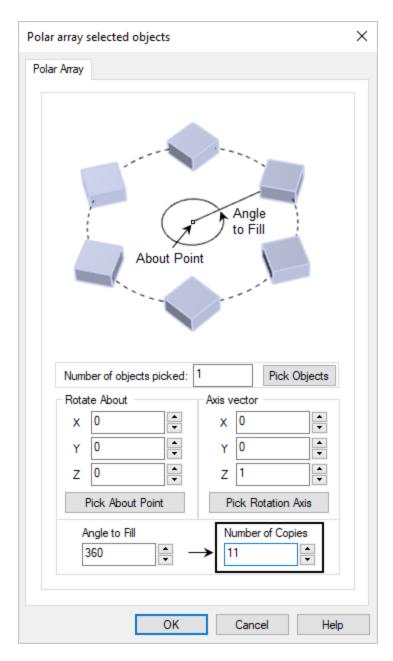
Create the Polar Array

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.



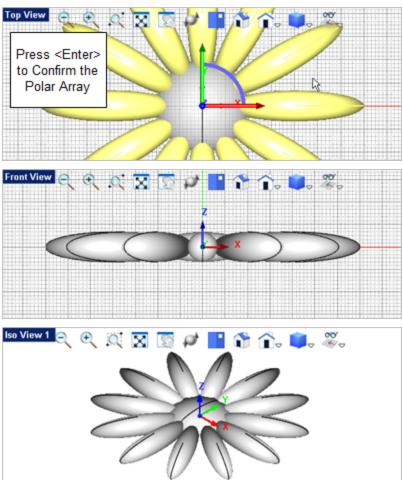
- 3. Select the Transform Transform tab from the ribbon bar.
- 4. Select the Polar Array Polar Array command. The Polar array selected objects dialog will display.



- 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





11.6 Additional Spheres

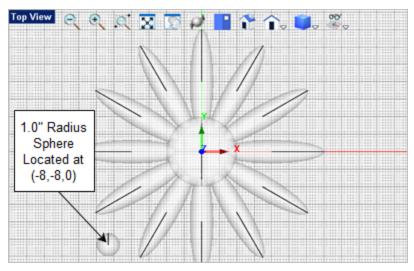
Create the 4 additional Spheres

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.

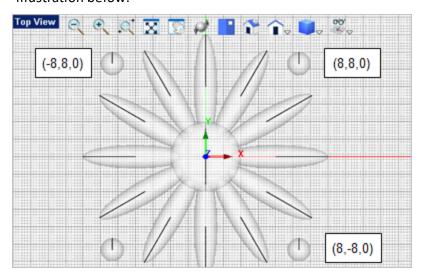
1. Activate the Top View Top View tab by selecting anywhere in the viewport.

2. Form 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 command and press <Enter>.
- 5. For the sphere radius, enter 1.0 in the command input bar Command and press <Enter> or select the grid point located at -8,-8,0 shown below.

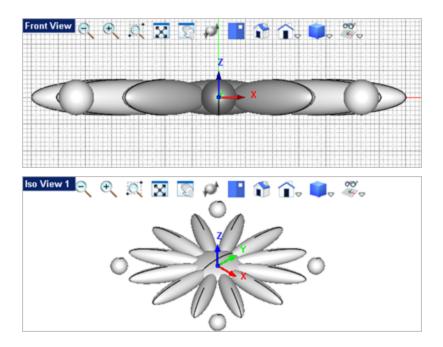


- Press <Enter> to repeat the Solid Sphere command.
- 7. Create the three additional spheres located at the coordinates shown in the illustration below.



Your drawing should look like this:

6.

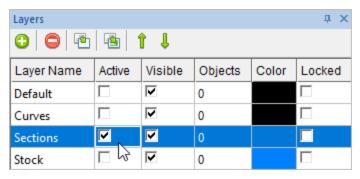


11.7 Create & Trim Section Curves

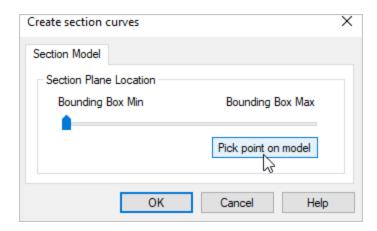
Create Section Curves

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.

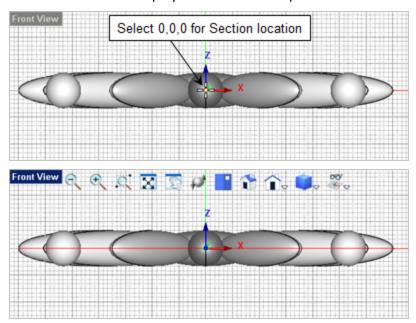
1. Open the Layer Manager and set the Active Layer to Sections. Refer to the previous steps for displaying and using the Layer Manager.

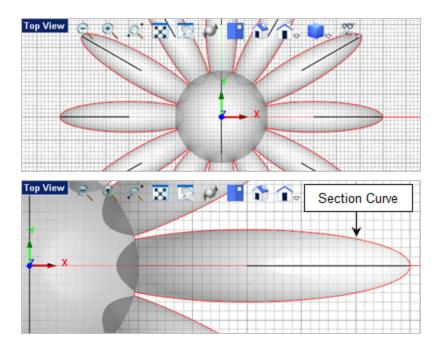


- Section curve are created in the plane of the active layer. We want the section curves to be generated on the XY plane so activate the Top View tab by selecting anywhere in the viewport.
- 3. Form the Curve Modeling Curve Modeling tab select the Section Curve command. The Create section curves dialog will display.

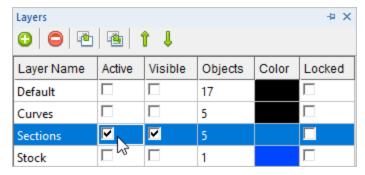


- 4. If you move the slider you will see dynamically where the section curves lie. In this case, with the Top View active, moving the slider will display section curves dynamically in the XY plane and in the Z Axis.
- 5. We want to precisely locate the section plane so select the Pick point on model 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.

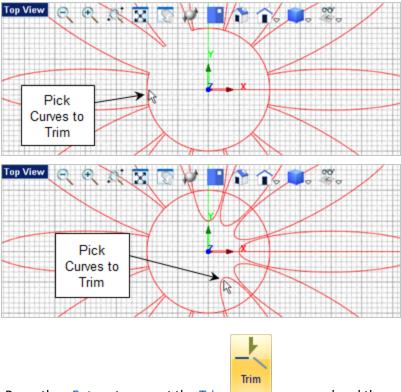


In the following steps we will trim our section curves and merge them.

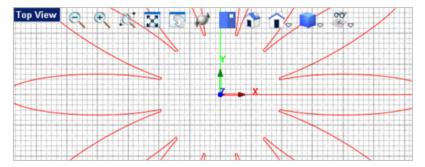
8. From the Top View, scroll to zoom in on the center where the section curves from the body and the pedals overlap each other.



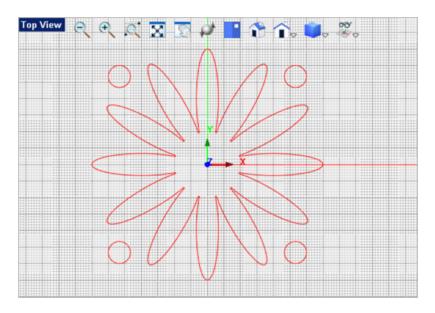
- 9. From the Curve Modeling Curve Modeling tab, select the Trim
- 10. Pick on the curve where you want it to be trimmed away.



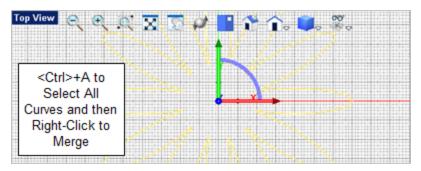
- 11. Press the <Enter> to repeat the Trim command and then pick another location to trim.
- 12. Repeat this procedure until all of the curves are trimmed as shown below.



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



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



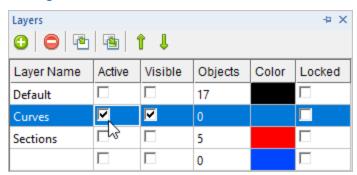
15. Now from the Curve Modeling command. This will merge all of the curves into one curve. This makes it easier to work with. Only curves that are joined end to end will be merged. Separate curves such as those created from the section of the outer spheres will remain separate curves.

11.8 Create Offset Curves

Create Offset Curves

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 2. other layers hidden. Refer to the previous steps for displaying and using the Layer Manager.



Now, select the main section curve, then go to the Curve Modeling 3.

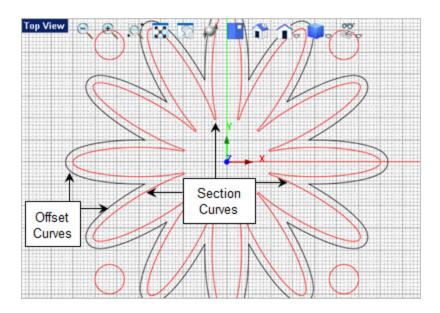


Offset tab and select the Offset command.

From the Top View move the cursor towards the outside of the curve so that you see the offset direction indicator point toward the outside. **DO NOT** pick a point!



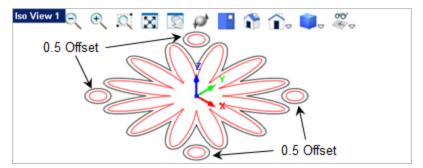
Now with the curve offset direction displayed enter 0.5 in the command input bar Command sand press the <Enter> key. The curve will be offset toward the outside by 0.5 inches.



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.

Offset

7. Repeat the command to create offset curves for the other 3 remaining section curves.

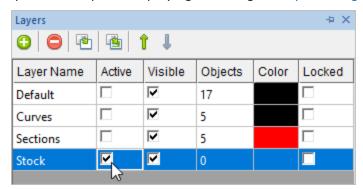


11.9 Create the Stock Boundary

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.

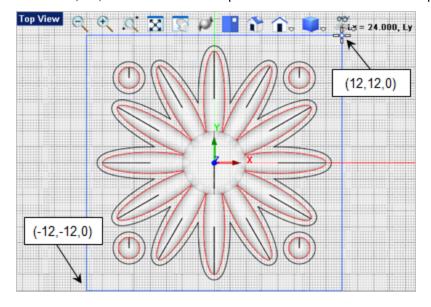
1. Open the Layer Manager and set the Active Layer to Stock. Refer to the previous steps for displaying and using the Layer Manager.



2. Activate the Top View Top View tab by selecting anywhere in the viewport.



- 3. Form 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 Command and press <Enter>.

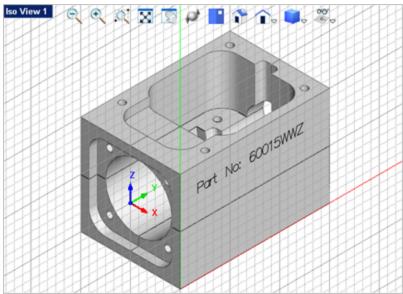


- 5. For the second corner of the rectangle, you can select the grid point shown below or enter 12,12,0 in the command input bar Command and press <Enter>.
- 6. Save the file as Daisy_Decor_Completed.vcp.

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:

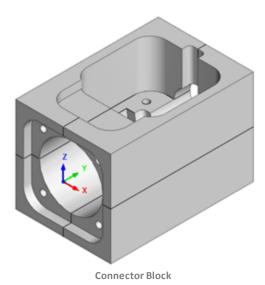
- 1. Orient the C-Plane.
- 2. Draw on a Face.
- 3. Other C-Plane Options.
- 4. Orient the Part.

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 <u>Exercise #5 Connector</u>
Block, open that part now. It is shown below:



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

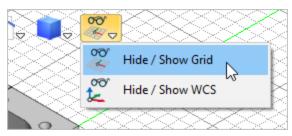


- 3. From the Modeling Aids Modeling Aids tab select the C-Plane to 3D Face

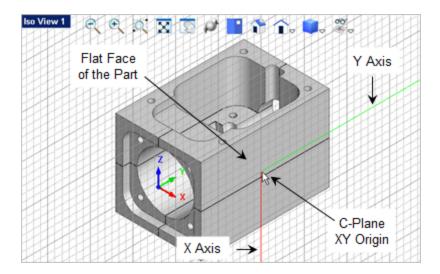
 command.

 ✓ To Face
- 4. The command prompt says: "Pick a flat area". Select the right side face of the part and you will see that the C-Plane snaps to that face.

If the Grid is not visible, select Hid / Show Grid from the View Toolbar.



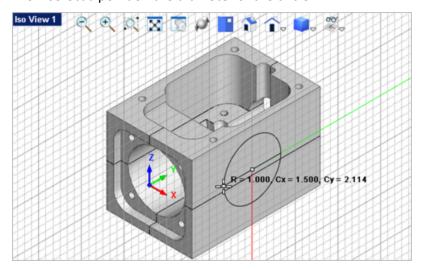
- It is best to have the Construction Plane Grid visible. This allows you to better keep track of where the C-Plane is located and oriented.
- 5. You can now see that the C-Plane and the Grid is alined with the right side flat face of the part. You will notice that the C-Plane origin is position at the point on the flat face you selected.

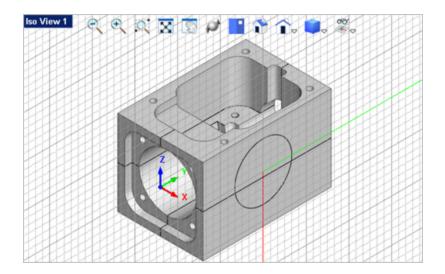


6. You can now draw on the C-Plane (see <u>Text on a Part Face</u> for more information).



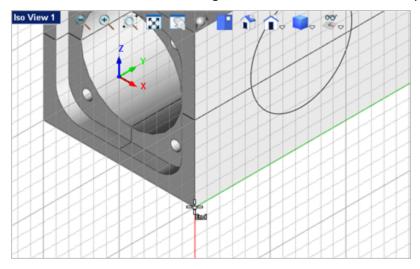
- 7. For the center point, toggle on the Grid Object Snap and select the origin point or simply enter 0,0 in the command window and press <Enter>.
- 8. Now select a point on the diameter of the circle.





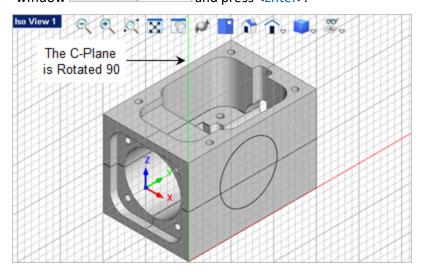
- 9. You can also move the C-Plane origin. From the Modeling Aids Modeling Aids tab
 - select the C-Plane Origin command.
- 10. The command prompt says: "Pick origin point or enter coordinates x,y and Z" Enable the End Point Snap from the Status Toolbar and then select the bottom right end point of the face.

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

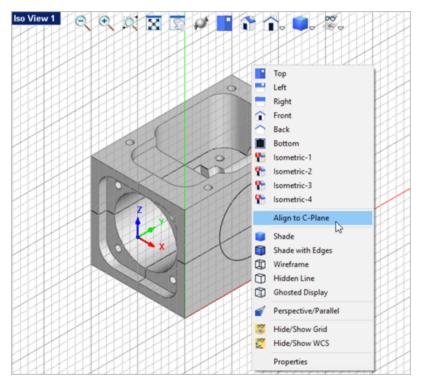


11. If you need the X Axis of the C-Plane aligned horizontally across the bottom of the part you can select the Rotate C-Plane about Axis Rotate command. It is also located on the Modeling Aids Modeling Aids tab.

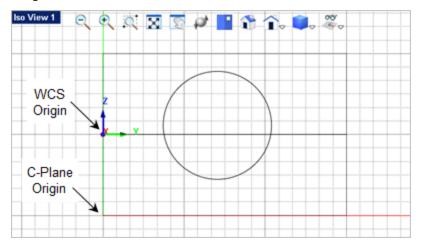
12. The command prompt says: "Enter axis to rotate about and angle to rotate by (eg: X,45)" We want to rotate the Z Axis 90 degrees so enter Z,90 in the command window Command and press <Enter>.



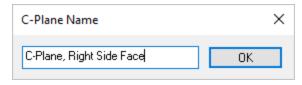
- 13. Now what if you want the view orientation aligned with the C-Plane?
- 14. Just right-click anywhere in the view to display the View pop-up menu and select the Align to C-Plane command. This menu contains many Viewport related commands.



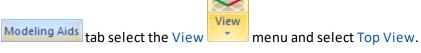
15. With the C-Plane aligned with the active Viewport pay close attention to where the origin of the C-Plane is:

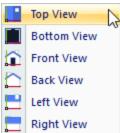


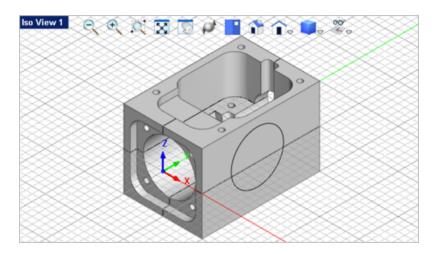
- 16. Now you may want to come back to the C-Plane later so let's save it. From the Modeling Aids tab select the Save C-Plane command.
- 17. In the C-Plane Name dialog enter a unique name for the active C-Plane and then pick OK.



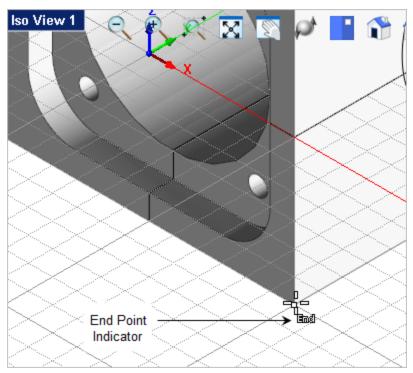
- 18. The active C-Plane is now saved with the part file.
- 19. Now let's change the C-Plane back to the default top view. From the Modeling Aids



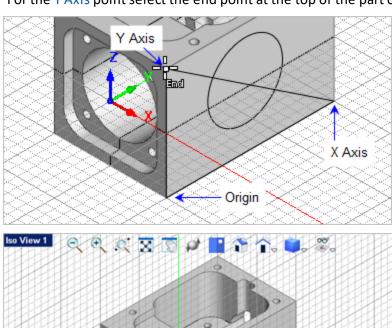




- 20. Here is an easier way to align the C-Plane to the face AND align the XY Axis of the C-Plane at the same time. From the Modeling Aids Modeling Aids tab select the C-Plane by 3 Points By 3 Pts command.
- 21. The first point selected will be the origin, so make sure the End Point Snap is toggled on.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



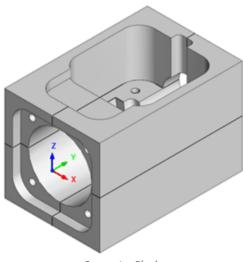
24. For the Y Axis point select the end point at the top of the part on the right side face.

12.2 Text on a Part Face

Drawing on a Face

In the last step (see <u>Orient the C-Plane</u>) 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:

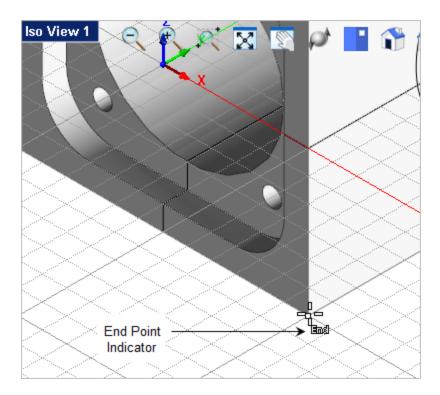


Connector Block

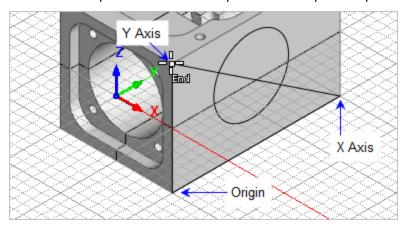
2. Now select the Modeling Aids Modeling Aids tab. On the very right end of the ribbon bar you will find the C-Plane pane.

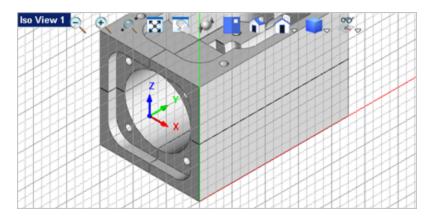


- 3. From the Modeling Aids Modeling Aids tab select the C-Plane by 3 Points Command.
- 4. The first point selected will be the origin, so make sure the End Point Snap is toggled on.
- 5. Then select the bottom left corner end point of the right side face of the part. **Note**: Do Not pick until you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

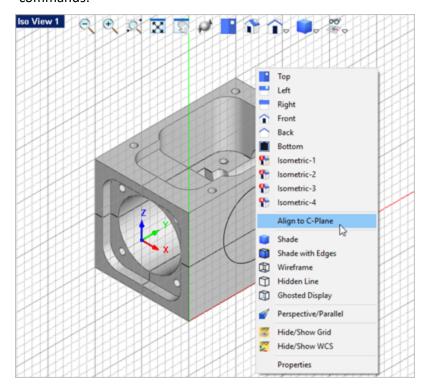


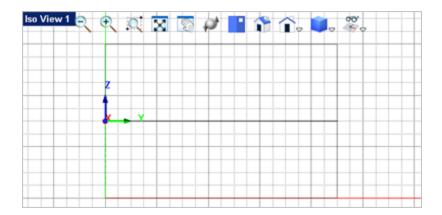
- 6. For the X Axis point select the end point at the back end of the part on the right side face.
- 7. For the Y Axis point select the end point at the top of the part on the right side face.





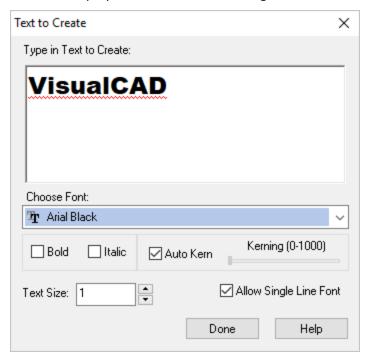
8. Now right-click anywhere in the view to display the Viewpoint pop-up menu and select the Align to C-Plane command. This menu contains the Viewport related commands.



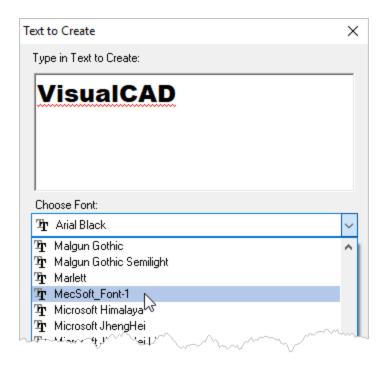


- 9. If you want to simply label your part you can add dimension text that will lie on the C-Plane. See <u>Drawing & Dimensioning</u> for that exercise.
- 10. In this exercise we will create text curves that you can engrave with the 2-1/2

 Engraving toolpath strategy, go to the Curve Modeling Curve Modeling tab and select the Create Text Command.
- 11. This will display the Text to Create dialog.

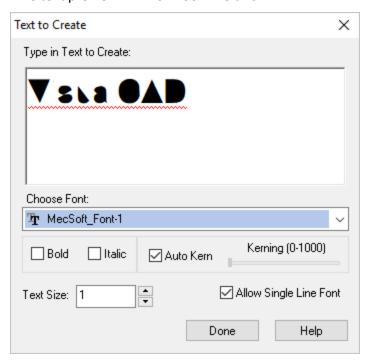


12. For engraving text you may want to use a single stroke font. From the Font drop-down menu select the text font named MecSoft_Font-1.



 If you do not see the MecSoft Font, close VisualCAD and <u>click here to Download</u> <u>the single stroke font</u>. Unzip and copy the file MecSoft_Font_1.ttf to the C: \Windows\Fonts folder.

13. The text preview will now look like this.



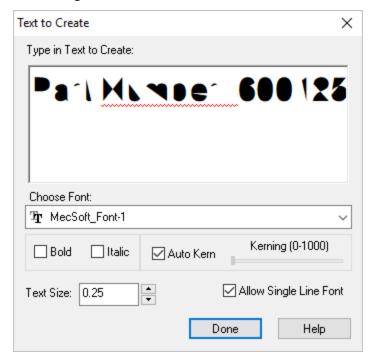
14. Make the following adjustments to the dialog:

Allow Single Line Font: Checked Text Size: 0.25

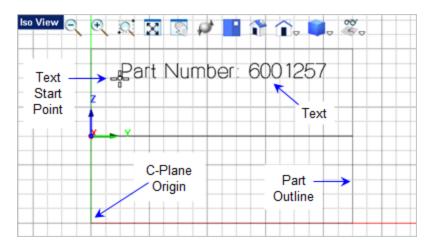
15. Now place in the Type in Text to Create window, replace the text VisualCAD with the following text:

Part Number: 6001257

16. The dialog should look like this:

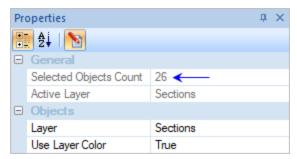


- 17. Now pick the Done button. The dialog will close and command prompt says: Enter start point of text.
- 18. You will see the text as you move the cursor around the Viewport.
- 19. Enable the Grid Snap and then select the grid point to position the text, or 0.5,2.5 in the command window and press <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 <u>Set System Options</u> 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
View	Immediately sets the orientation of the C-Plane to be parallel to the plane of the currently active view. The origin remains the same.
Origin	Moves the origin of the C-Plane to a selected point or entered coordinates. The current configuration of the construction plane is saved whenever the model is saved.
Elevation	Moves the elevation of the C-Plane along its normal (perpendicular to the plane). Input is a single distance and may be positive or negative. A positive number moves the plane towards the positive end of the coordinate axes even if the viewing location is along the negative axis. All input values are treated as an incremental distance relative to the current location of the plane.
Rotate	Rotates the C-Plane about one of the principle axes of the current C-Plane, X, Y, or Z. The center of rotation is the center of the plane and not necessarily the origin. Input is 'axis,degrees'.
•ॢ• By 3 Pts	Three points are selected (origin, x-axis and y-axis) or entered to re-orient the C-Plane. The y-axis point defines the C-Plane and the general direction of the positive y-axis.
To Face	The C-Plane will be oriented to a selected flat plane of a geometry object. The pick point will become the C-Plane origin.
By X Axis	Two points are entered to re-orient the x-axis of the C-Plane:
↓ To WCS	Resets the C-Plane back to its default orientation relative to the WCS and the currently active viewport. The origin will be the WCS origin.
	This option saves the C-Plane orientation from the currently active view. A name must be provided for later reference when loading.
	This option allows the retrieval of a saved C-Plane into the currently active view. A list of previously saved C-Planes is

presented. These saved C-Planes may also be deleted in this option or renamed.

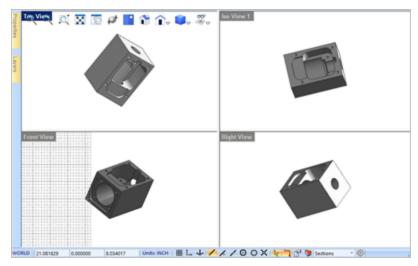
12.4 Orient the Part



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:

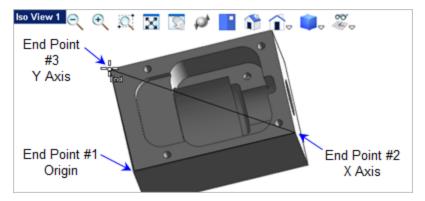


- The first step toward to orienting this part is to define the C-Plane. For more about this see the topics Orient the C-Plane and Text on a Part Face in this exercise.
- First make sure the Grid is visible. You can toggle the Grid on/off by selecting Hide / Show Grid from the View Toolbar.

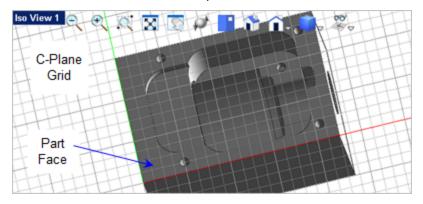


Now activate the view that allows you to see the planar face that you want to use to 3. orient the part by. In this example it is Iso View 1 Iso View 1

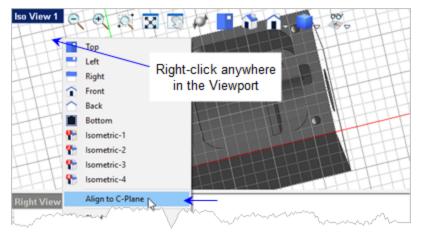
- 4. Go to the Modeling Aids Modeling Aids tab and select the C-Plane by 3 Points command.
- 5. Activate the End Point Snap from the Status Toolbar and select the end points in the order shown below:



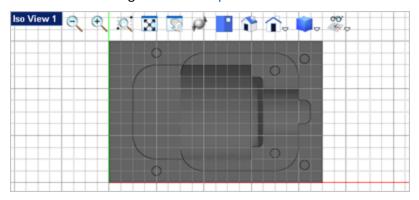
The C-Plane will orient to the 3 points selected.



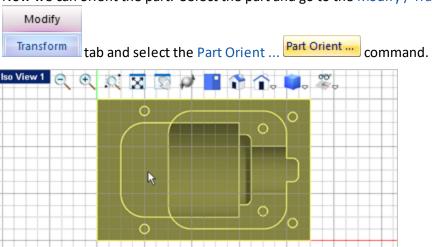
6. Now right-click anywhere in the Viewport (do not select any geometry) and select Align to C-Plane. This will align Iso View 1 parallel to the C-Plane.



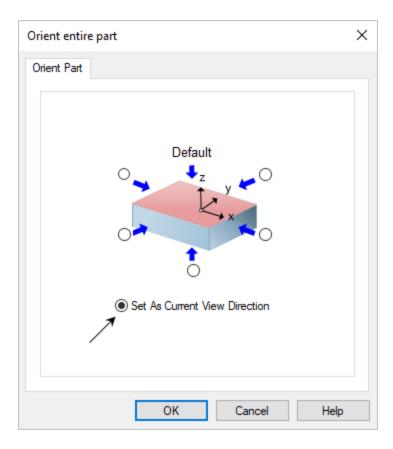
The C-Plane will align to the Viewport.



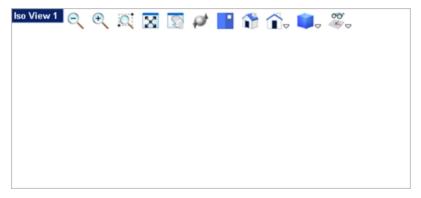
7. Now we can orient the part. Select the part and go to the Modify / Transform



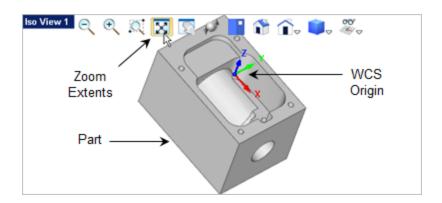
8. This will display the Orient entire part dialog. Pick the option to Set as current view direction.



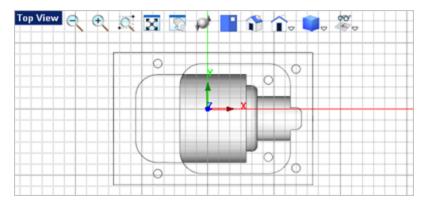
9. Pick OK and the part appears to disappear from the Viewport.



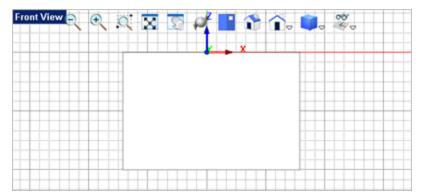
10. Now pick the Fit View icon from the View Toolbar. The part will appear with the WCS oriented in/on the center of the face that we used to align the C-Plane

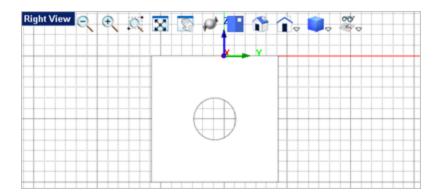


11. Now activate the Top View Top View and pick the Fit View icon from the View Toolbar.

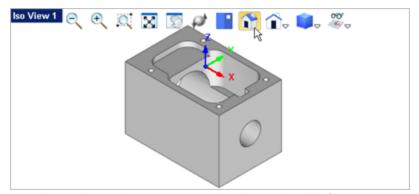


12. Repeat this for the Front View Right View and the Right View Remember that you have to activate the Viewport before the View Toolbar will display. Left-click anywhere within the Viewport to activate it.



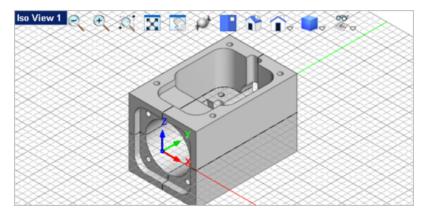


13. Activate Iso View 1 again and this time select the Iso View icon from the View Toolbar to orient this Viewport.

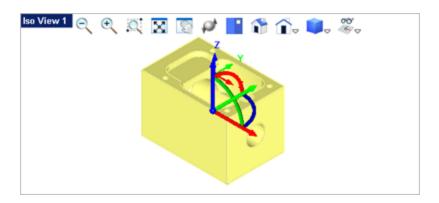


Great! You have the part oriented orthographically (i.e., Top, Front, Right).

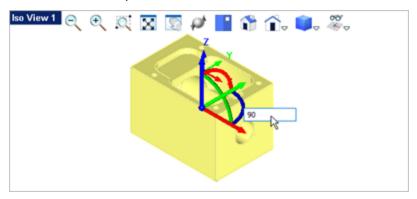
14. However, the part is still not oriented like the original part as shown below. You see that the length of the part was oriented along the Y Axis and the WCS was positioned at the center of the front view. Let's show you some additional commands to fix that.



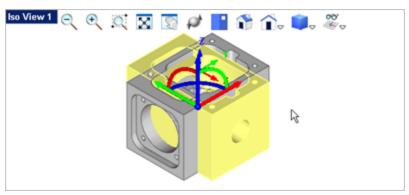
- 15. Activate Iso View 1 and then from the Status Toolbar, toggle the Graphic Manipulator On.
- 16. Now select the part to see the Graphic Manipulator displayed.

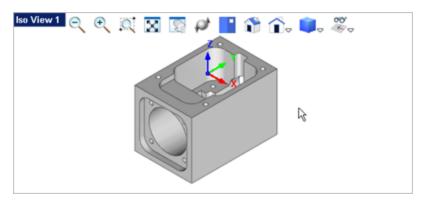


17. Pick the Z Rotation arc of the Graphic Manipulator to display the input window, enter 90 and then press <Enter>.



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

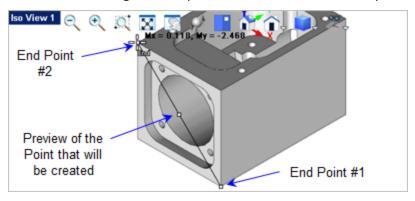




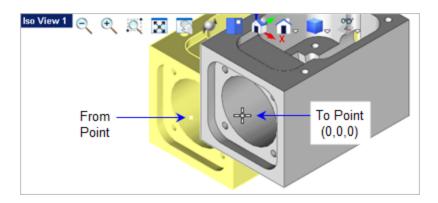
Great where almost there!

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

- 19. Zoom in on the front face of the part.
- 20. From the Curve Modeling 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.

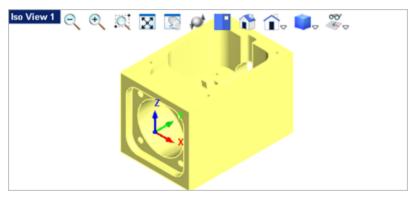


- 23. You can now toggle the Graphic Manipulator 🗽 off.
- 24. With the point created, window select the entire part (and point) or simply press <Ctrl+A>.
- 25. From the Modify / Transform tab select the Move by mouse command.
- 26. The command prompt says: Pick from point or enter coordinates x,y and 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 Command and then press <Enter>.

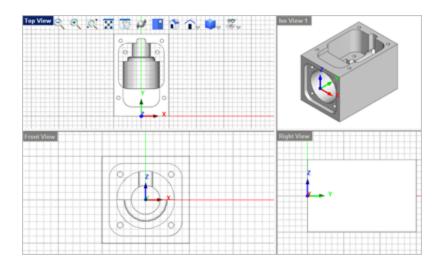
You will see that the part moved to the WCS origin. Now press <Enter> again to accept the move.



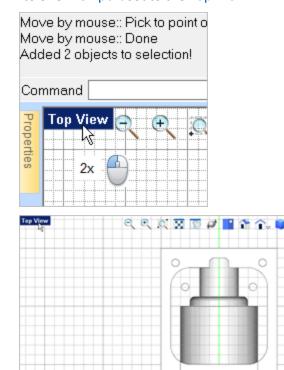
28. Now the part is aligned correctly accept for the C-Plane. Previously the C-Plane was aligned to the XY plane of WCS.

Here is a neat trick to fix that:

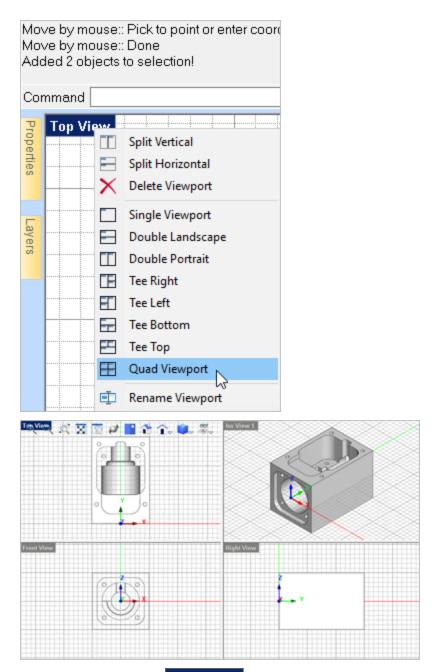
29. Activate the Top View Top View



30. Now double-left-click on the Top View to one Viewport set to the Top View.



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



You see that Iso View 1 Iso View 1 inherited the C-Plane of the Top View because that was the active View when you selected Quad Viewport.

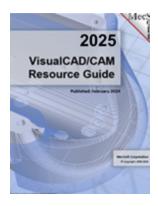
32. Save the file as Construction_Planes_Completed.vcp.

Congratulations on completing this exercise!

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