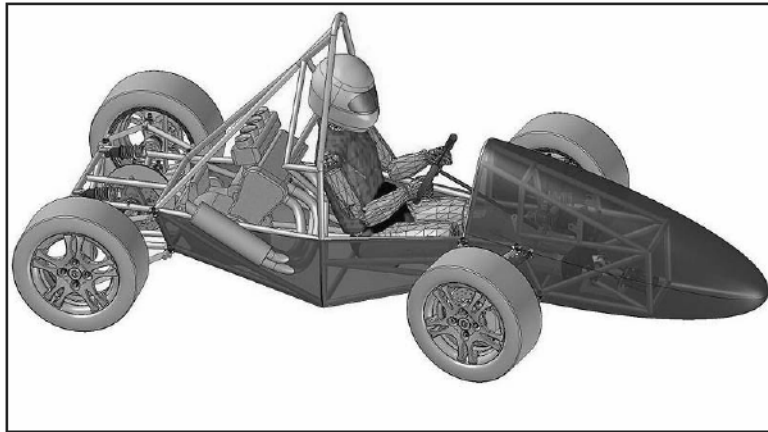


Engineering and Technical Drawing Using Solid Edge Version 17

By Jerry Craig



Images from Solid Edge Gallery



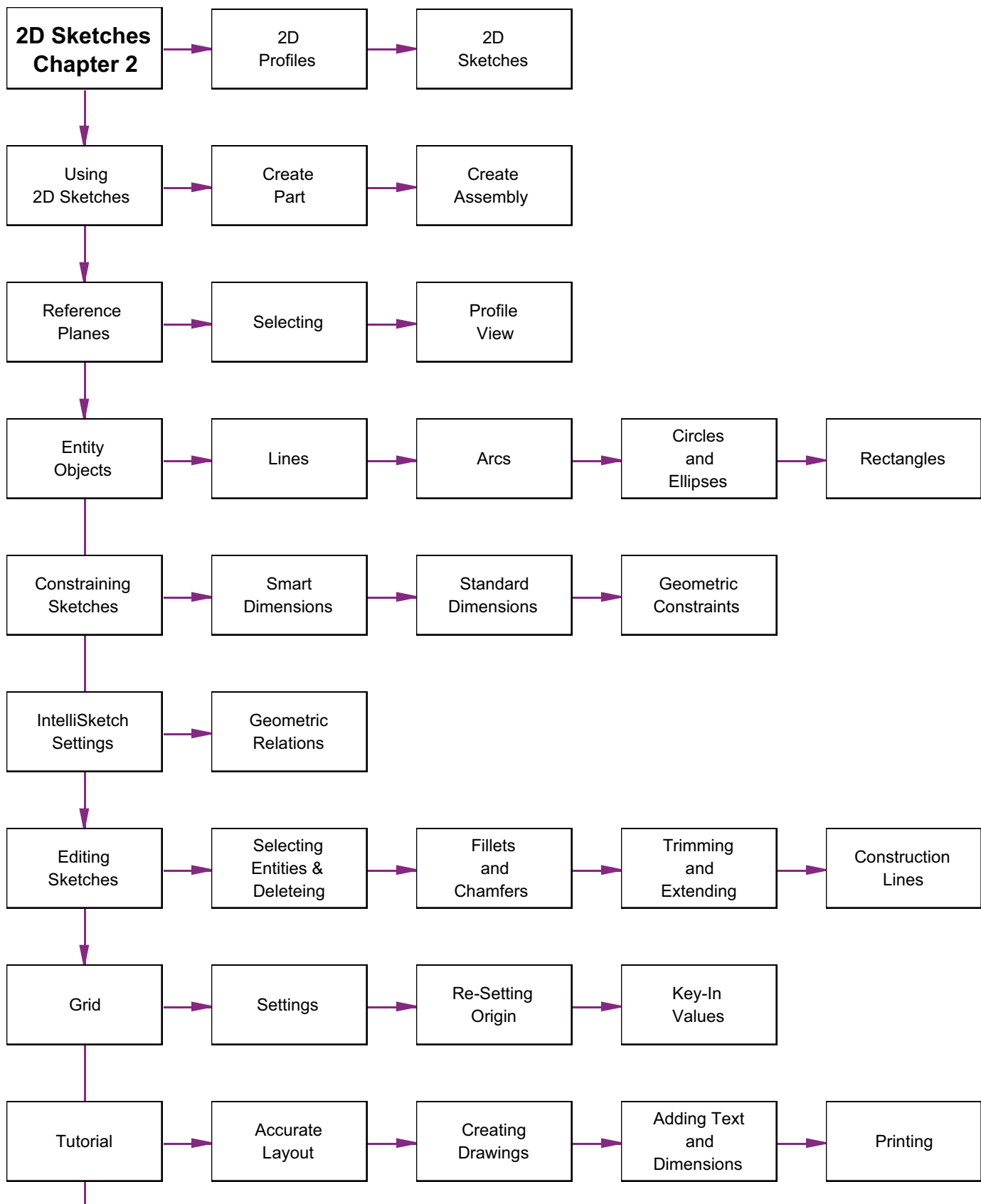
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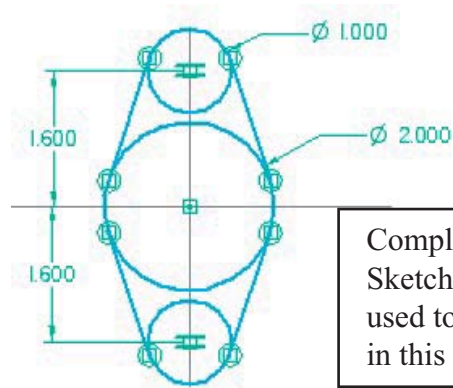




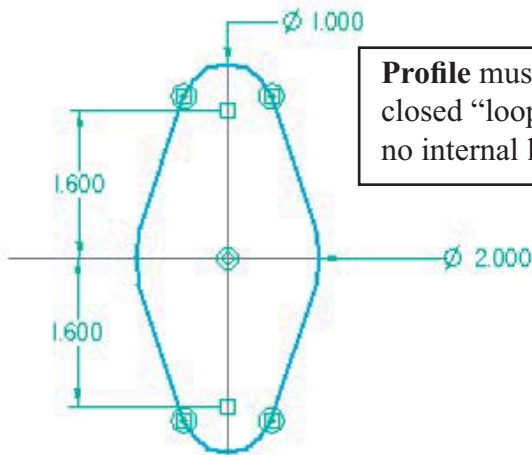
2D Drawings may take two forms:

2D Sketches show the outlines and locations of multiple shapes or parts to be designed.

2D Profiles must be closed shapes with no entities inside or crossing the sketch. The sketch at the right may be used to create a part by selecting specific shapes then “cleaning up” the outline by trimming inner curves. The final shape must be a single series of connected lines and arcs.



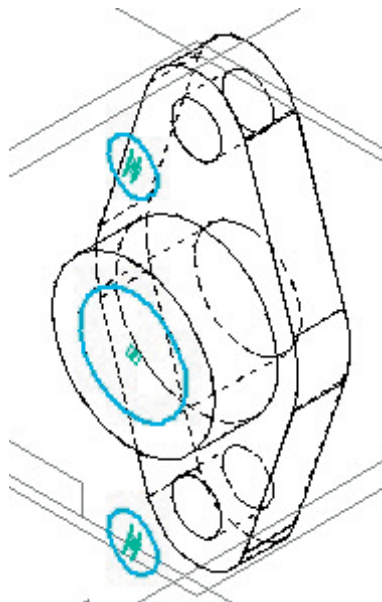
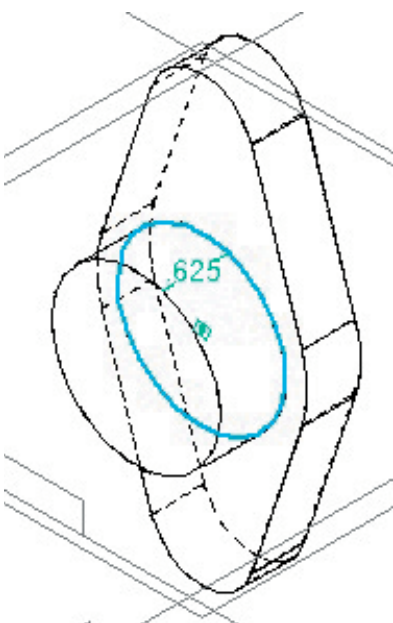
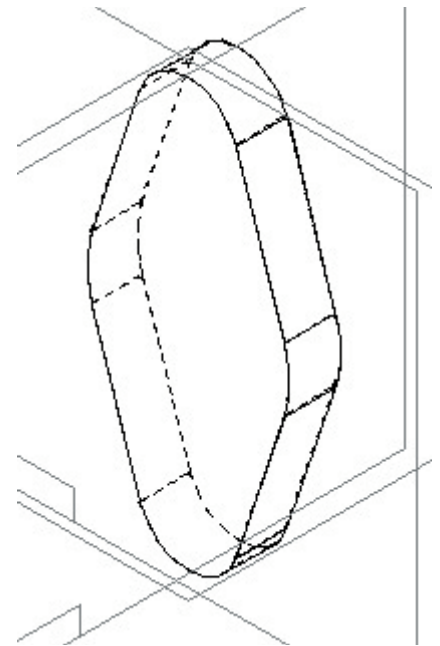
Complete Master Sketch. (Cannot be used to create a part in this form).



Profile must be a closed “loop” with no internal lines.

Portions of the Master Sketch are copied when creating individual part features.

Resulting 3D object with depth added.



Assembly Sketches (Layouts)

These are used to define the critical geometry in a series of mating parts. Elements from the master sketch may be selected to create the individual parts. This assures that the parts are the correct size and are located in exactly the right place.

Elements of the part sketches which are derived from the master sketch may be associated (linked) back to the master sketch. This means that any change in the master sketch will update all the associated parts.

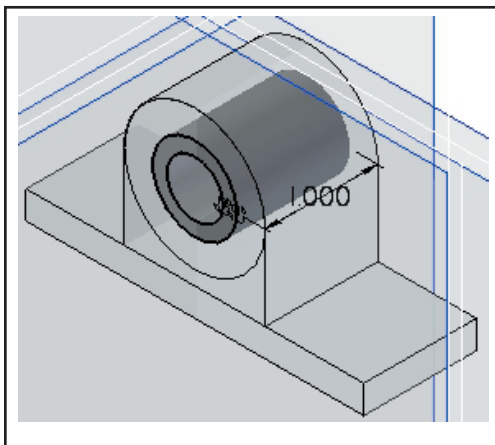
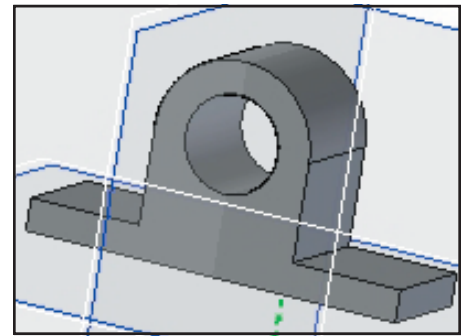
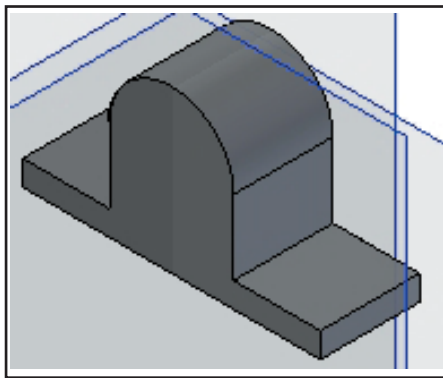
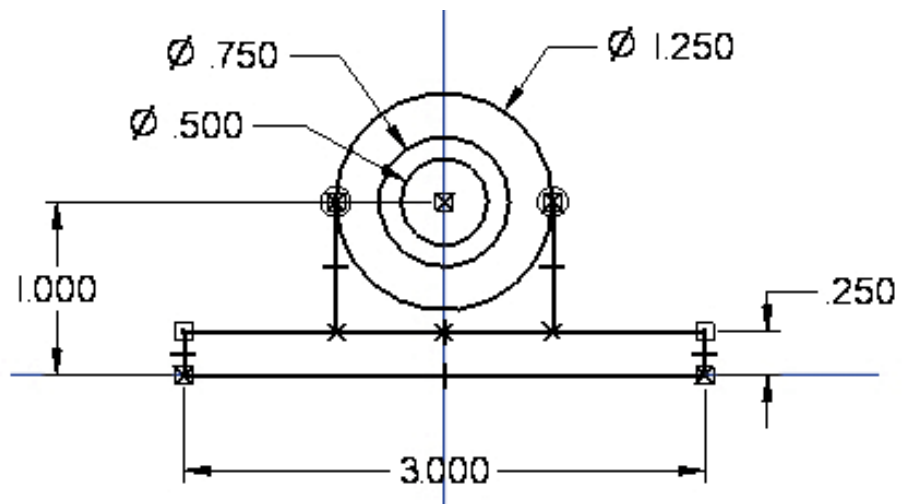
Portions of the sketch were used to create the base of the pillow block assembly.

Other portions of the same sketch will be used to create the opening for the bearing.

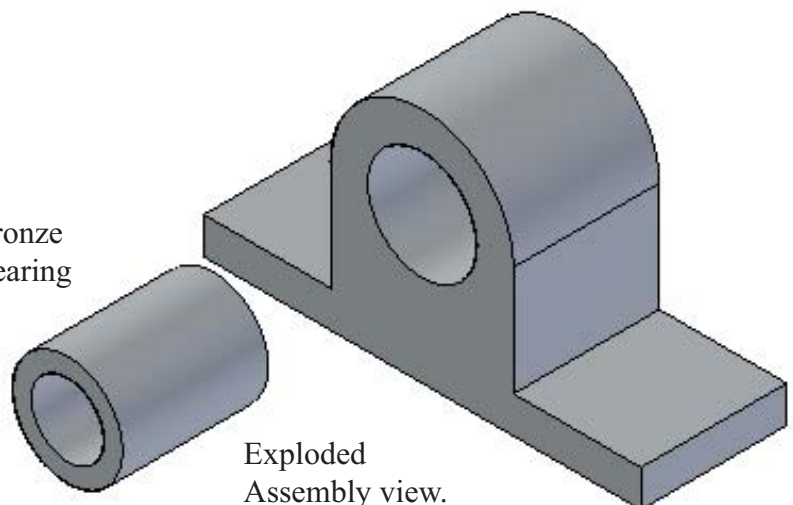
Modeling the bearing is done by selecting the inner circles. The bronze bearing is created as a separate part.

The final assembly is shown with the parts modeled exactly as they fit.

As you can see, 2D sketches are the basis for much of the design and part creation process. These sketches can be very general at first then, exact sizes can be assigned later. All parts associated to the master sketches will re-size to the new dimensions. Each part will be the right size and in the right place.



Bronze
Bearing



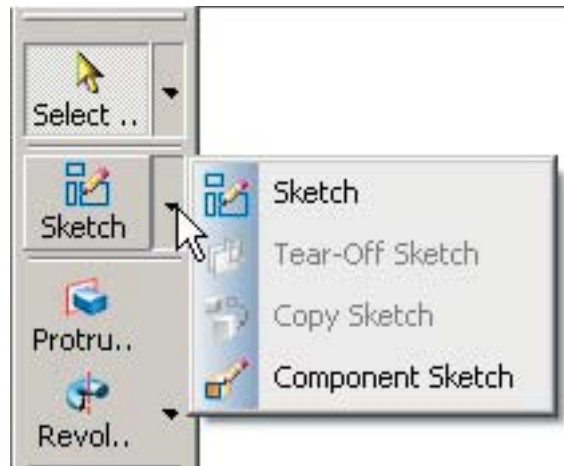
Exploded
Assembly view.

Starting a 2D Drawing

Any new object starts with a 2D sketch. Complex objects can be designed more efficiently by creating a master sketch with part geometry defined by a series of overlapping lines. As the part shape evolves the 2D lines are used to add material or cut away material.

Simpler shapes may be designed by starting directly with the Protrusion or revolved Protrusion icons.

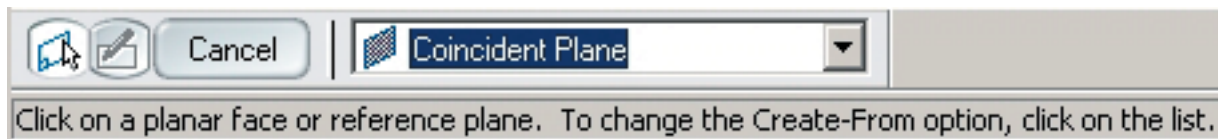
Master sketches give more control over the modeling process.



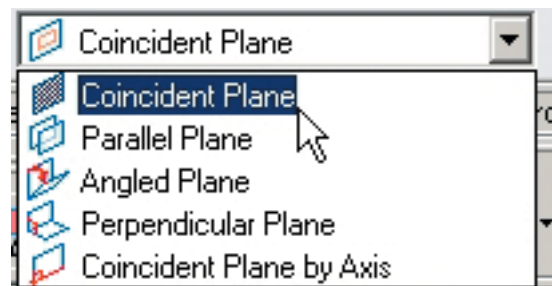
Component Sketches are placeholders for existing parts or for related items yet to be designed.

Once **Sketch** is started, The **Ribbon Bar** displays prompts and options.

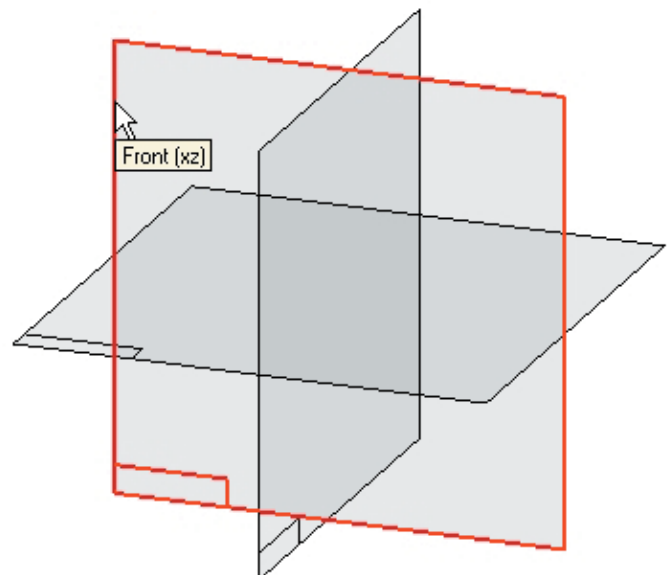
A **Reference Plane** must be selected. Or, a new reference plane must be defined as a 2D drawing plane.



A pop down dialog box allow the selection of alternate planes for starting the sketch.



Coincident Option starts a sketch on one of the existing reference planes.



A 2D drawing screen appears.

- The **Draw** toolbar replaces the Main toolbar.

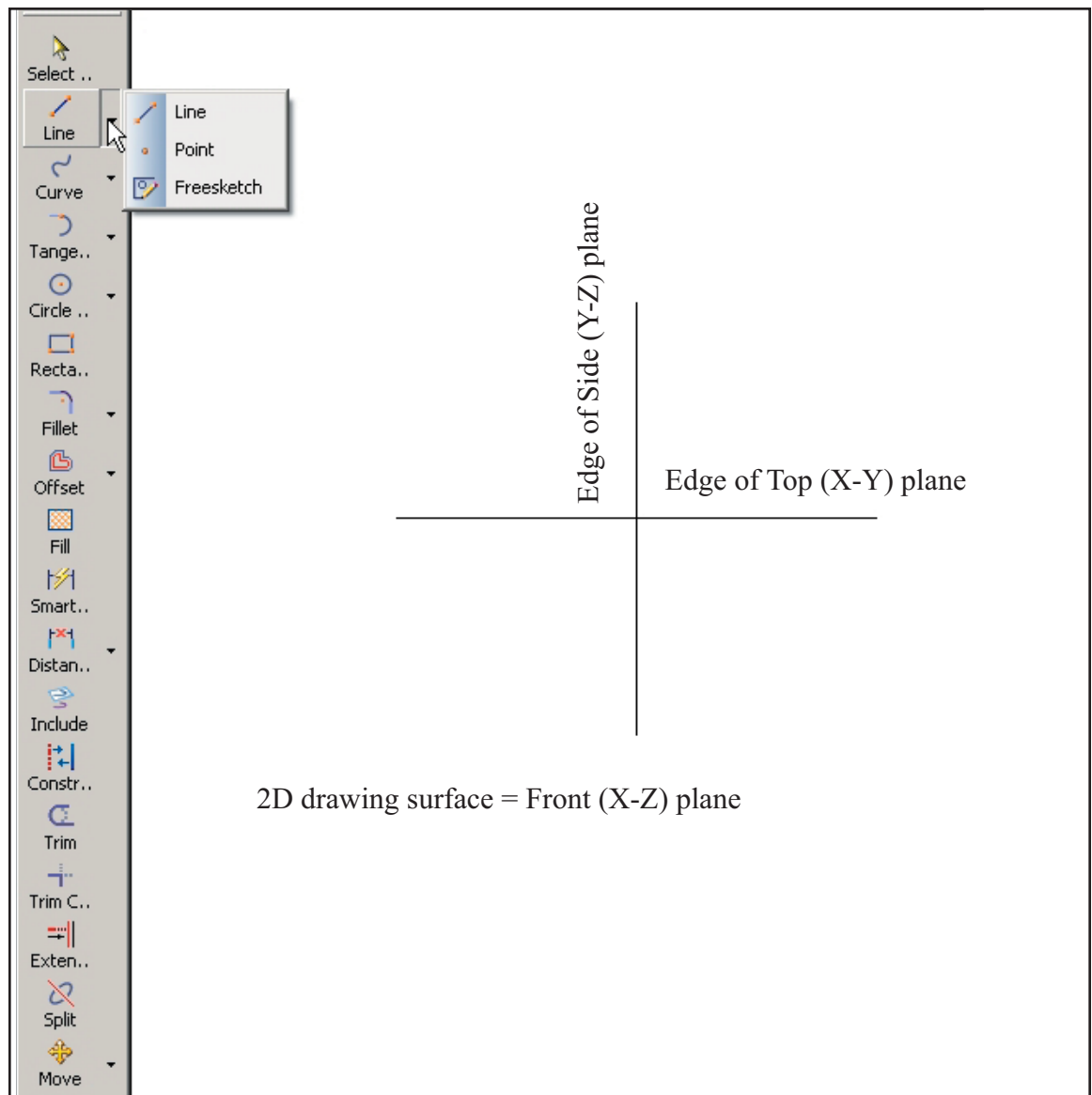
- The drawing surface is now the Front (X-Z) plane.

- The horizontal line is the edge view of the Top (X-Y) plane.

- The vertical line is the edge of the Side (Y-Z) plane.

Draw menu items may have several options selected from pop down menus.

The first tutorial (chapter 1) introduced the 2D drawing process.



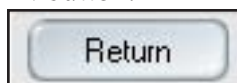
Exercise:

- ✓ Start Solid Edge.
- ✓ Click Sketch.
- ✓ Click the Front reference plane.

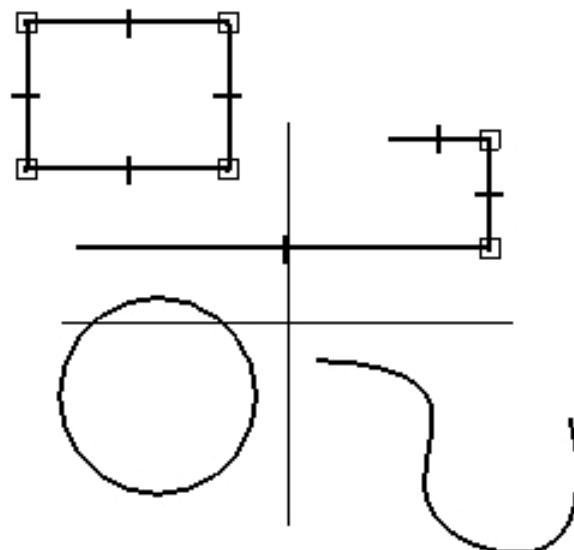
Draw some shapes.
Explore the pop downs

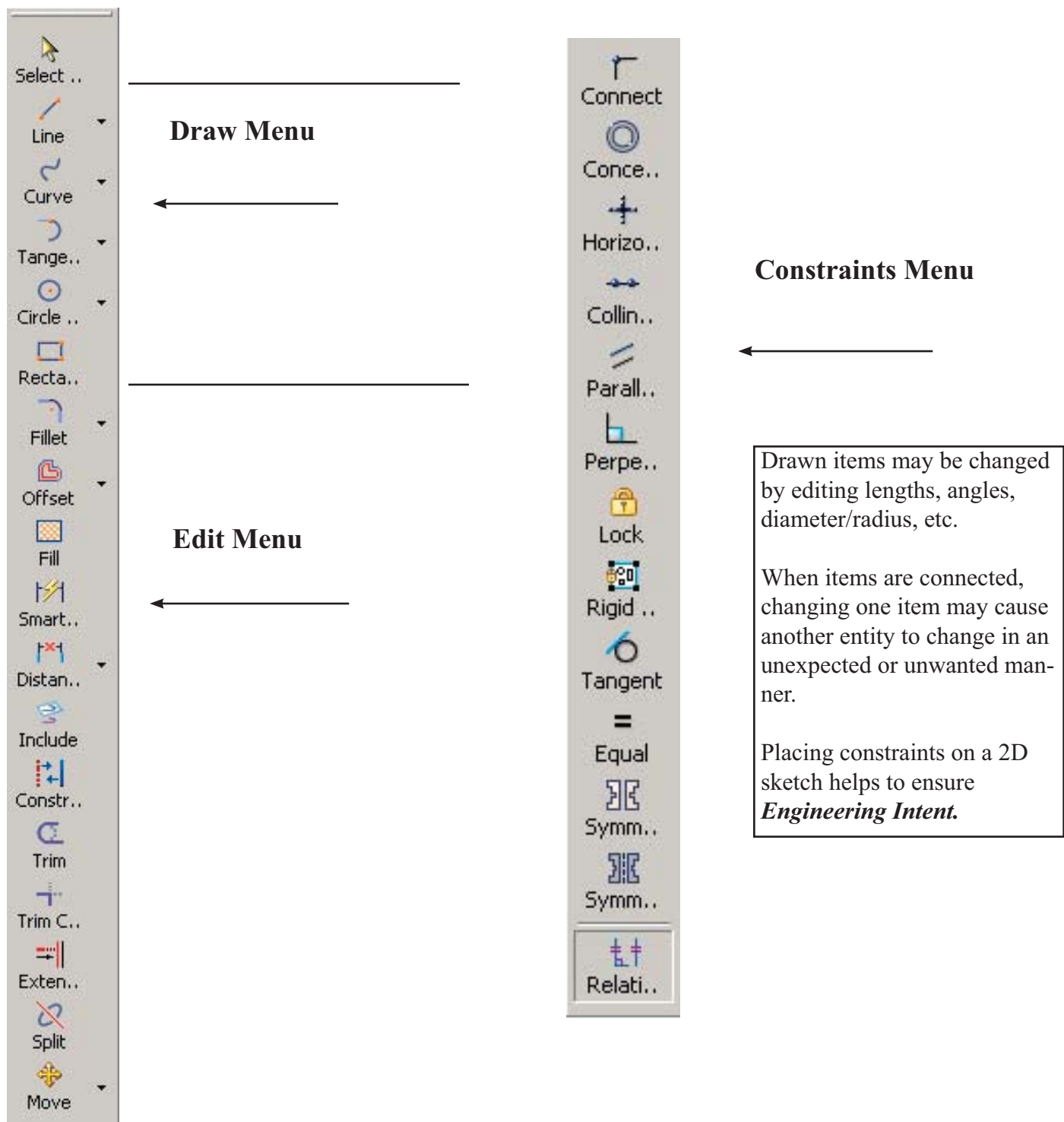
Press ESC key to stop a command.

When finished, click the RETURN button.



Exit Do not save the drawing.





Constraining sketches is an important step in the design process. Many features may be changed as the design evolves. Making sure the master sketch changes in a predictable manner is critical.

Constraints are a combination of geometric (parallel, vertical, concentric, tangent, etc.) and dimensional lengths. The first tutorial (chapter 1) introduced some of these ideas.

Line command. Starting a line displays the **Line Ribbon Bar**.

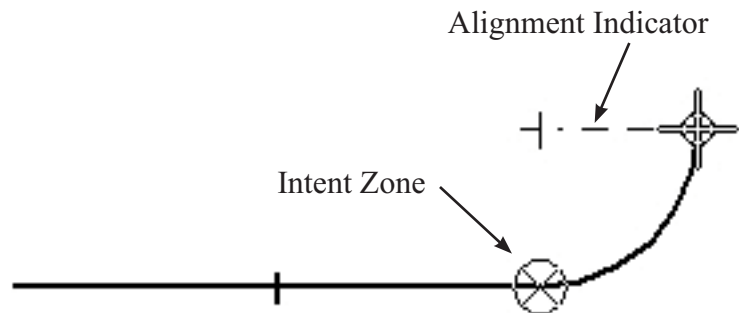


Active
Line
Style

Toggle between line and arc

Many indicators will appear as a drawing is created. These help to insure good geometry.

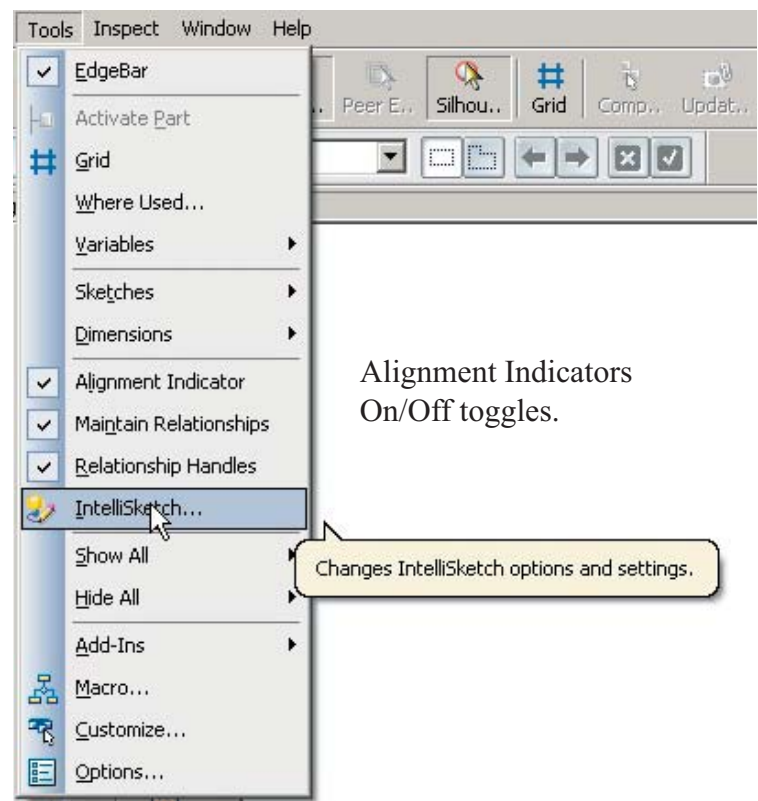
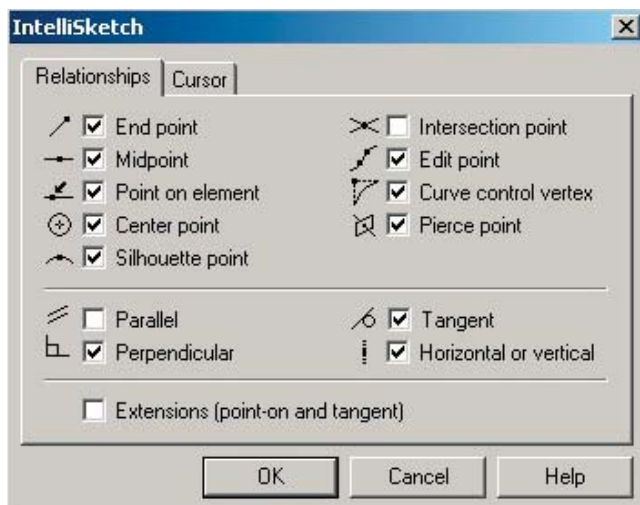
A straight line was drawn then the arc toggle was selected. The **intent zone** creates the arc based on *how the mouse moves out of the intent zone*.

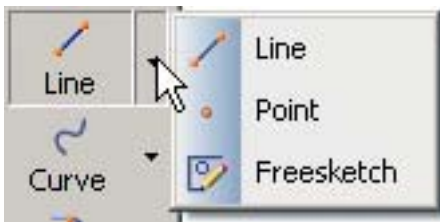


Alignment indicators flash when horizontal, vertical, extensions or other alignments are sensed.

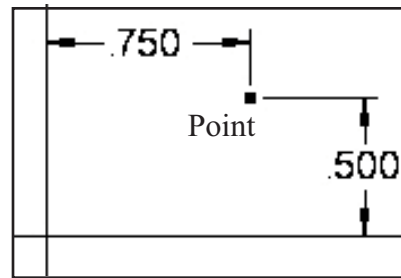
Indicators may be turned On/Off by clicking **Tools** menu and clicking the selection buttons.

Tools ... Intellisketch ... Relationships toggles various geometric indicators on/off.





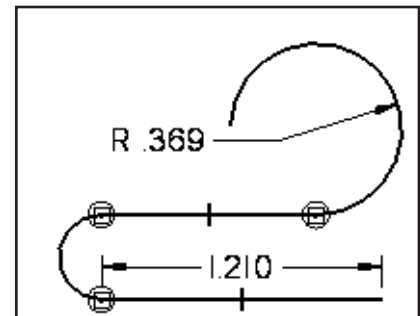
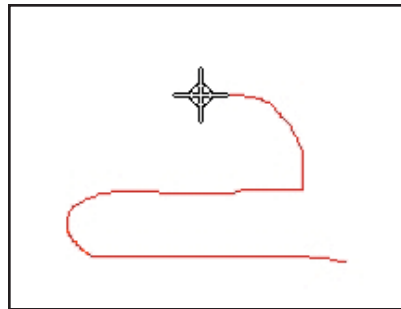
Distance Between
dimension option
was used to fix
point location.



Other options for Line are Point (used as a marker or place holder) and Freesketch.

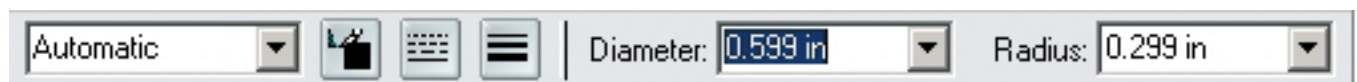
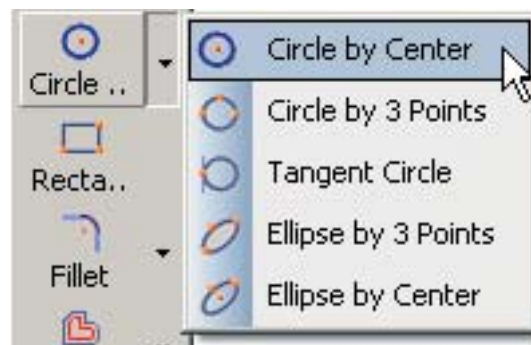
Freesketch. Hold left mouse button down and draw shape. Solid Edge will create straight lines and arcs.

✓ Place **smart dimensions** to constrain the shape.



Circle. Pop down provides options for drawing circles and ellipses.

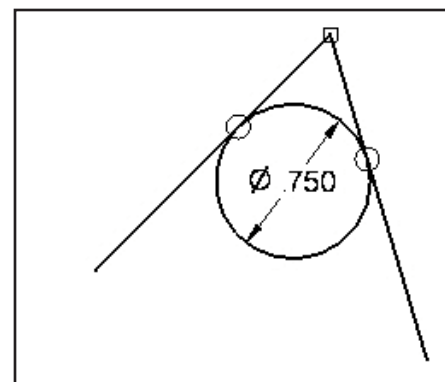
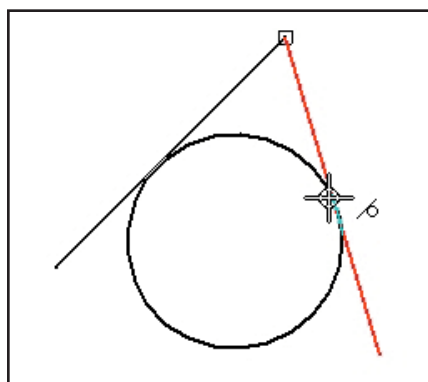
The circle ribbon bar provides options for color, linetype and line thickness. Click either diameter or radius box to input size.



Tangent Circle

A circle was placed at random tangent to two lines.

The circle size may be set using the circle command or by placing a **smart dimension**.



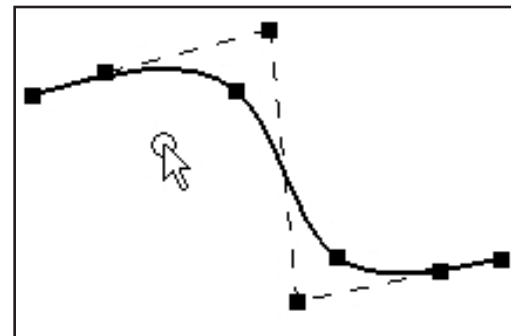
Arc.

Three options are shown.

The arc ribbon bar gives slightly different options for each arc type.

**Curve.**

Click a series of points.
A smooth curve is drawn.

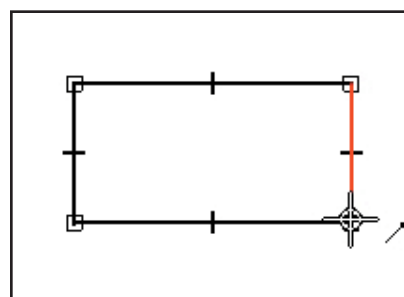
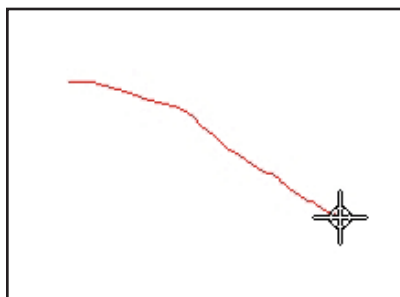


Curve has many settings and options.
Check **Help** for more information.

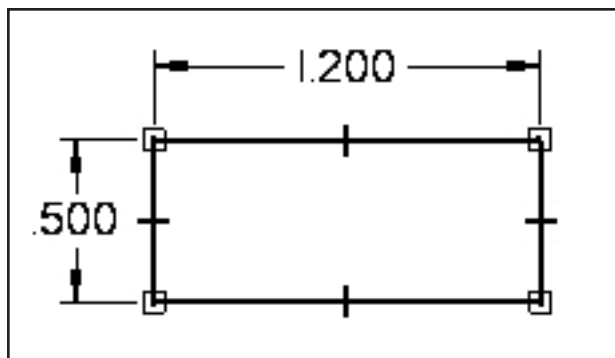
Rectangle.

Quick method:

Select **Rectangle**.
Hold down left mouse button and sketch diagonal line.



Use
smart dimensions
to set size.



Constraining Sketches.

Constraints help to insure that changes to a sketch behave in a predictable manner. Editing causes shapes to change and parts often must be re-sized for weight, strength, appearance or other reasons. All of the changes can result in some weird shapes unless constraints are in place to control the changes.

Two types of constraints are used: **Dimension Constraints** and **Geometric Constraints**.

Dimension Constraints

Smart Dimensions are applied to individual elements. A single pick will dimension a line, circle, etc.

A pop down menu for other dimension types is shown. Distance between and angle between relate two items.

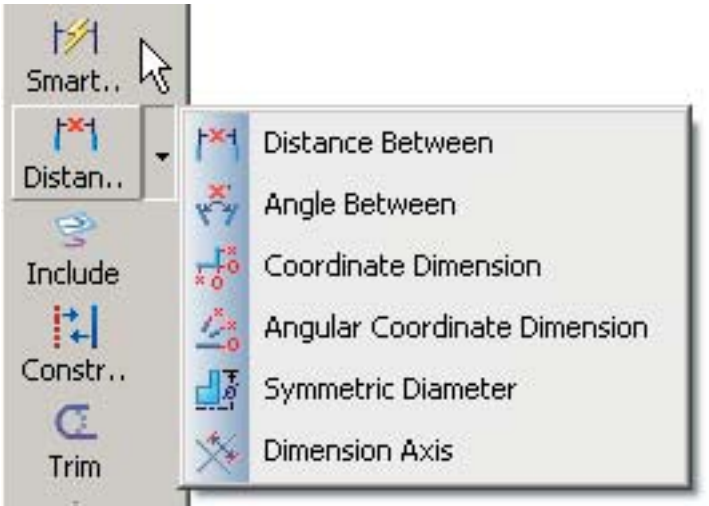
Driving dimensions control a part feature.
Driven dimensions are controlled by other dimensions. Each type is a different color on the screen.

Geometric Constraints



The geometric constraint icons are shown. Choosing and applying constraints is a matter of planning which elements must remain fixed in relation to other elements that are allowed to change.

Turns display of geometric icons Off/On



Relationship	Handle
Collinear	
Connect (1 degree of freedom)	
Connect (2 degrees of freedom)	
Concentric	
Equal	
Horizontal/Vertical	
Tangent	
Tangent (Tangent + Equal Curvature)	
Tangent (Parallel Tangent Vectors)	
Tangent (Parallel Tangent Vectors + Equal Curvature)	
Symmetric	
Parallel	
Perpendicular	
Fillet	
Chamfer	
Link (local)	
Link (peer-to-peer)	
Link (sketch to sketch)	
Rigid Set (2-D elements)	

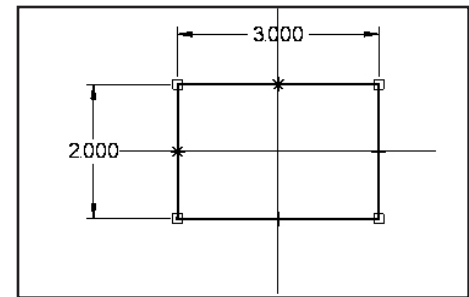
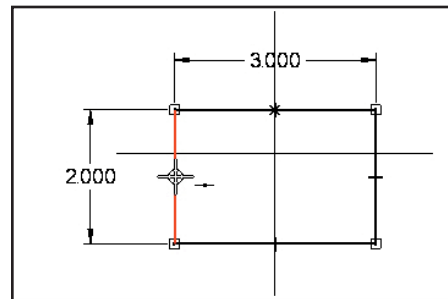
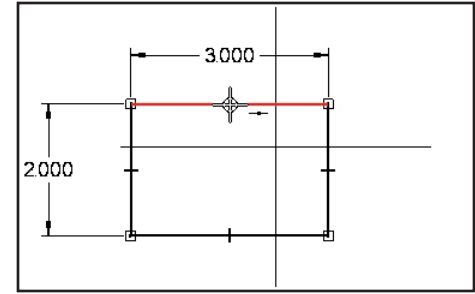
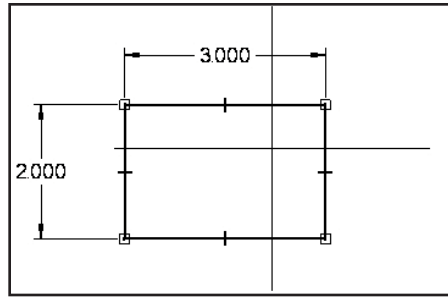
Aligning Geometry

The rectangle is centered on the reference planes using **connect constraint**.

(The first tutorial in chapter 1 used horizontal and vertical to do the same thing).



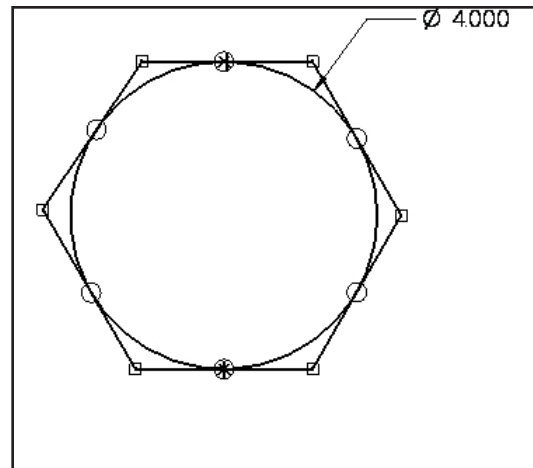
Connect the mid-point of the horizontal line to the vertical (Y-Z) plane.
Connect the mid point of the vertical line to the horizontal (X-Y) plane.



Changing the 2.000 and 3.000 dimensions will keep the rectangle centered.

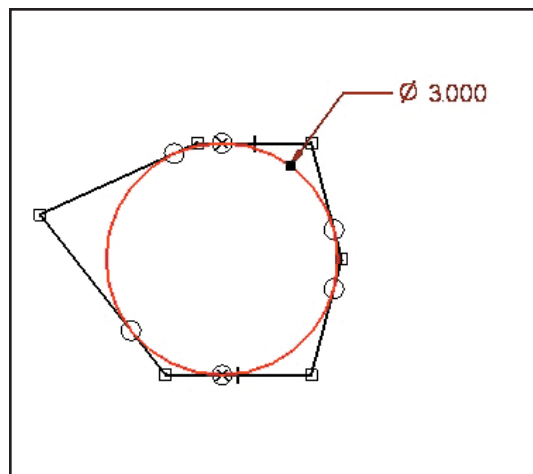
A hexagon has been circumscribed about a circle. We want to be able to change the circle size and keep the hexagon shape.

Lines are **connect constrained** and **tangent constrained**.



Changing the circle size produces in an unexpected result!

What other constraints need to be added to fix the problem?



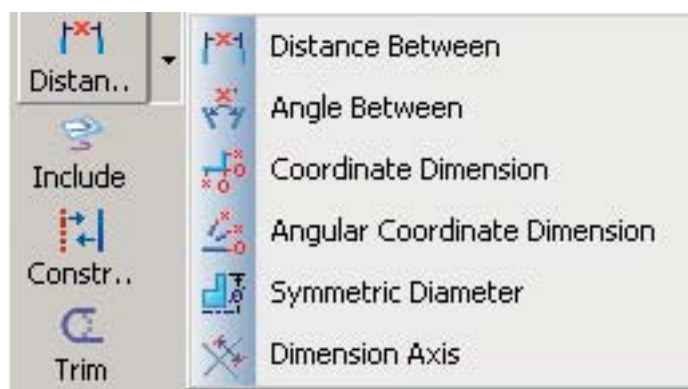
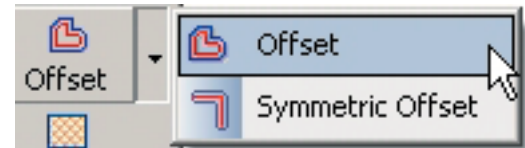
Editing 2D sketches.

2D drawings are often constructed using basic shapes then editing the shapes to get the final forms.



Fillet is a rounded corner. **Chamfer** is a beveled corner.

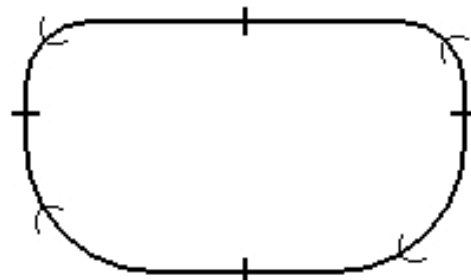
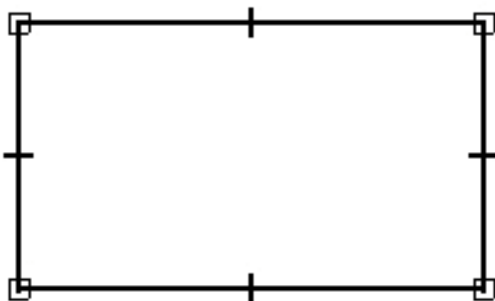
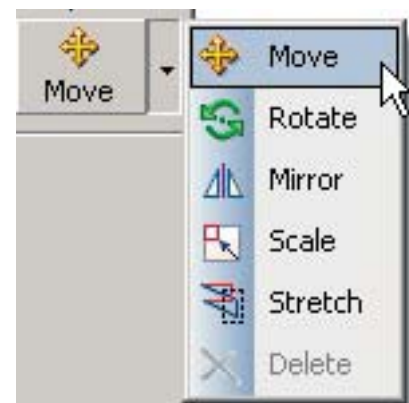
Offset may be one side or both sides.



Sizes and locations may be fixed by placing dimensions.

Objects may be moved or re-sized by changing the dimension value.

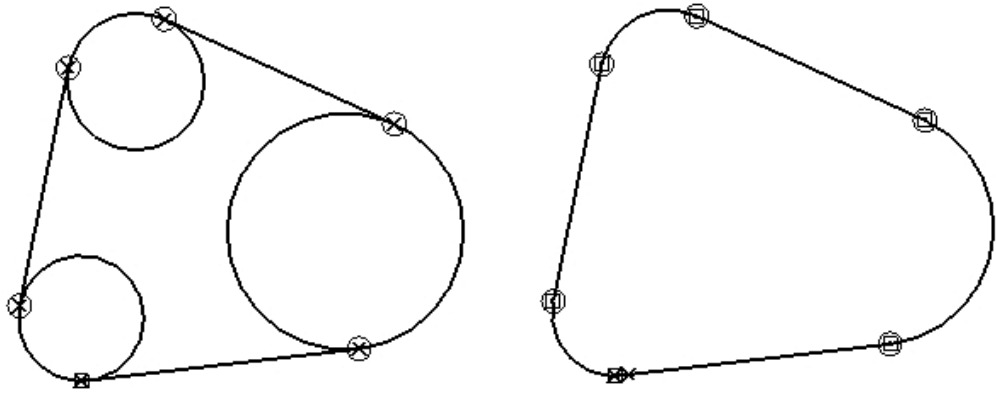
Move pop-down displays several related editing commands.



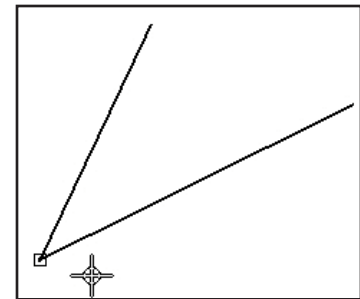
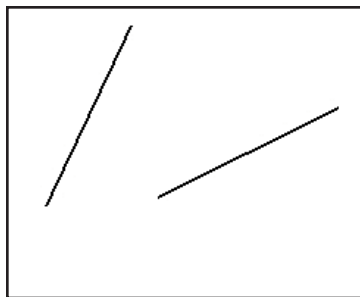
The rectangle shape was modified using the fillet editing command.

Trim

Inner circular arcs were removed using **trim**.



Extend will only work if there is a “stopper” boundary.

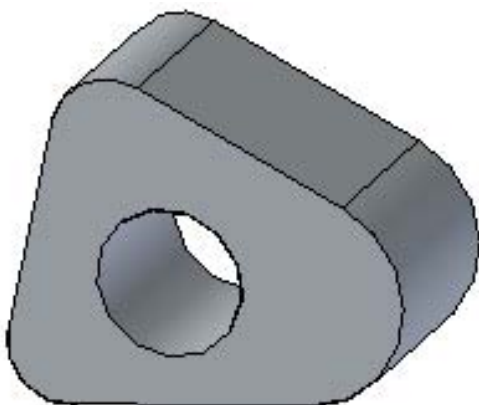
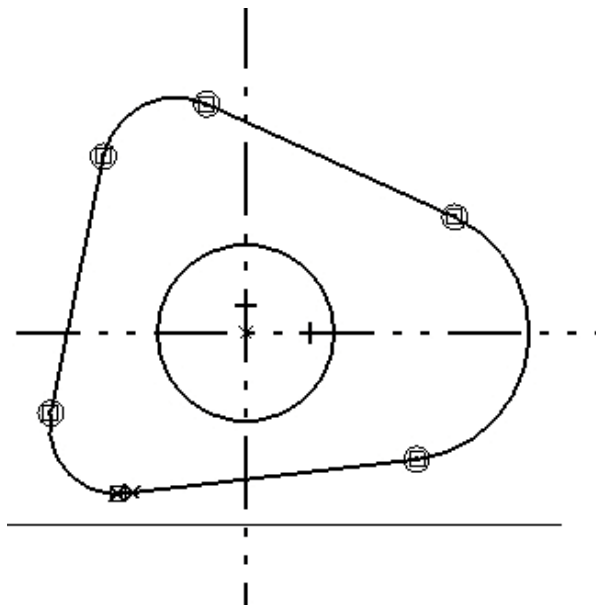


Extend would not work but **Trim** would work.

Construction Lines

Regular lines may be toggled to construction line status (and back).
Construction lines are not considered as part of a *profile* sketch.

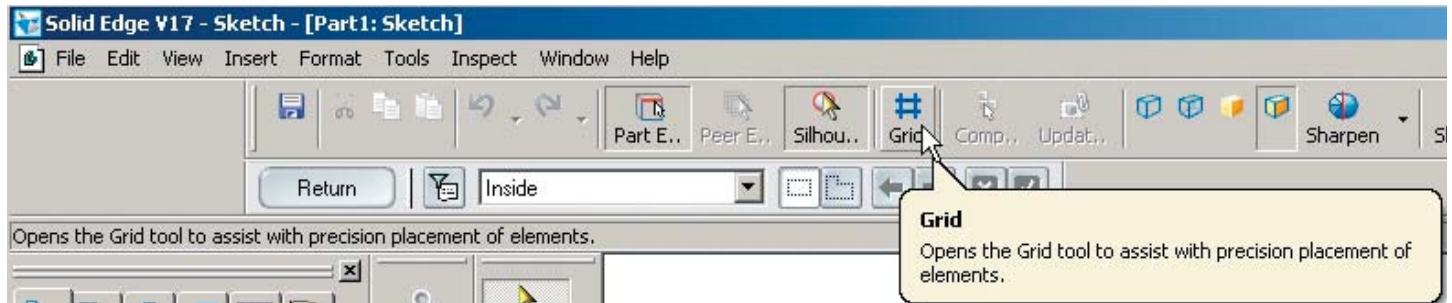
Using construction lines is a good way to anchor geometry and locate circle centers, etc.



Grid and Grid Snap

Placing a grid on the screen enables additional drawing accuracy controls. By setting a useful grid spacing and toggling **Snap To Grid**, the start and end points of lines, centers of circles, etc. can be placed accurately.

Grid is very useful when an item must be placed relative to an existing point. Re-set the grid zero then measure over to the new point.



Toggle Grid On/Off with the screen icon or use **Tools ...Grid**

Grid Ribbon Bar is shown.

Grid settings dialog

Grid display On/Off

Snap to Grid On/Off

Set Grid to Origin

Set Grid 0,0 to new location

Grid Options Dialog Box

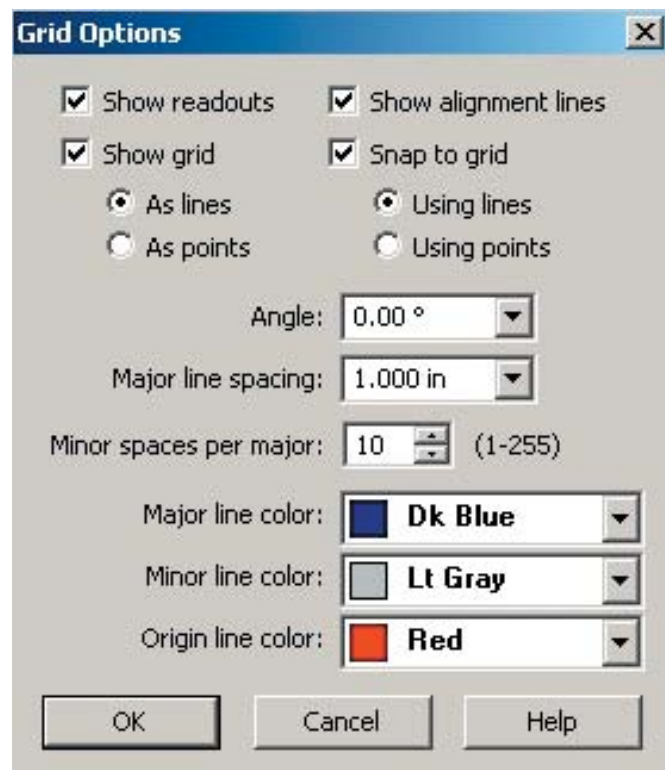
Sets major line spacing, Minor spaces per major, colors, etc.

A typical decimal inch setting is shown.

Working with architectural layouts, a typical setting might be:

Major = 1 foot

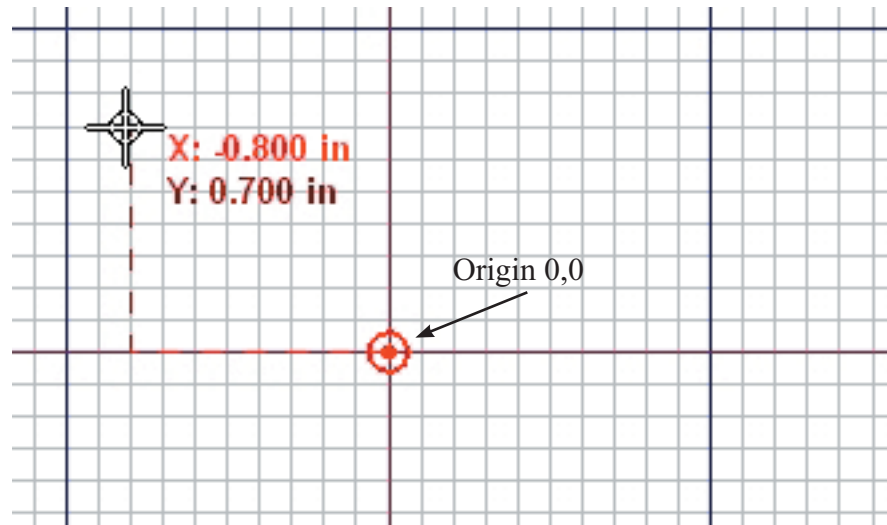
Minor = 12 (1 inch)



Decimal inch grid setting shown.

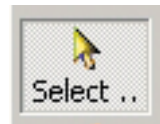
Grid Snap enabled so start and end points are on grid intersections.

X,Y locations may be set by clicking left mouse button or by keying in points.



Re-Sizing the Grid Area

✓ Click the Select Tool

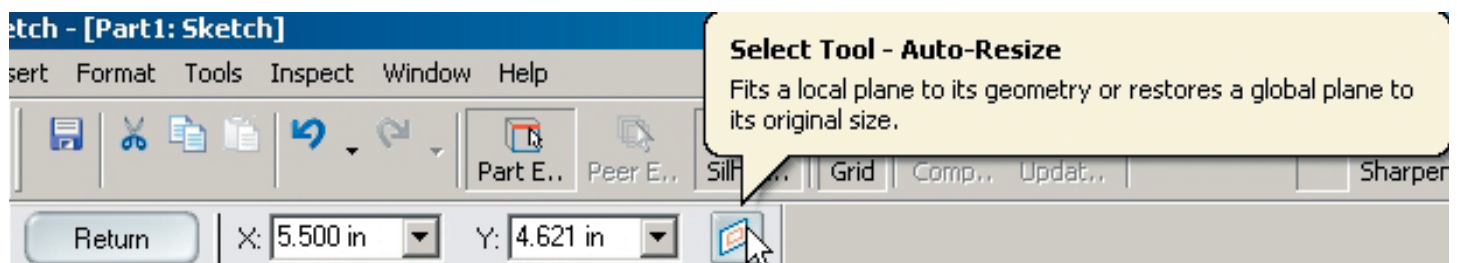
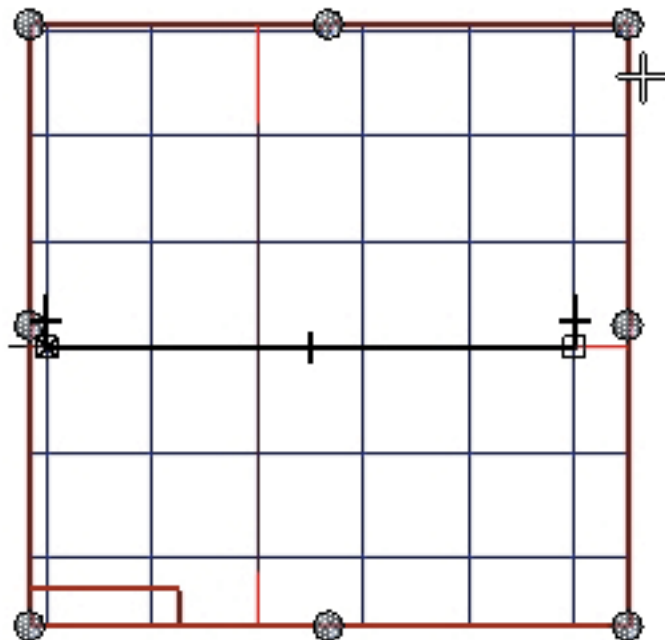


✓ Click near the outside of the grid.

✓ Drag the grid to a new area.

A new ribbon bar for **Grid Size Setting** will appear at the top of the screen.

An auto-resize option may be enabled.



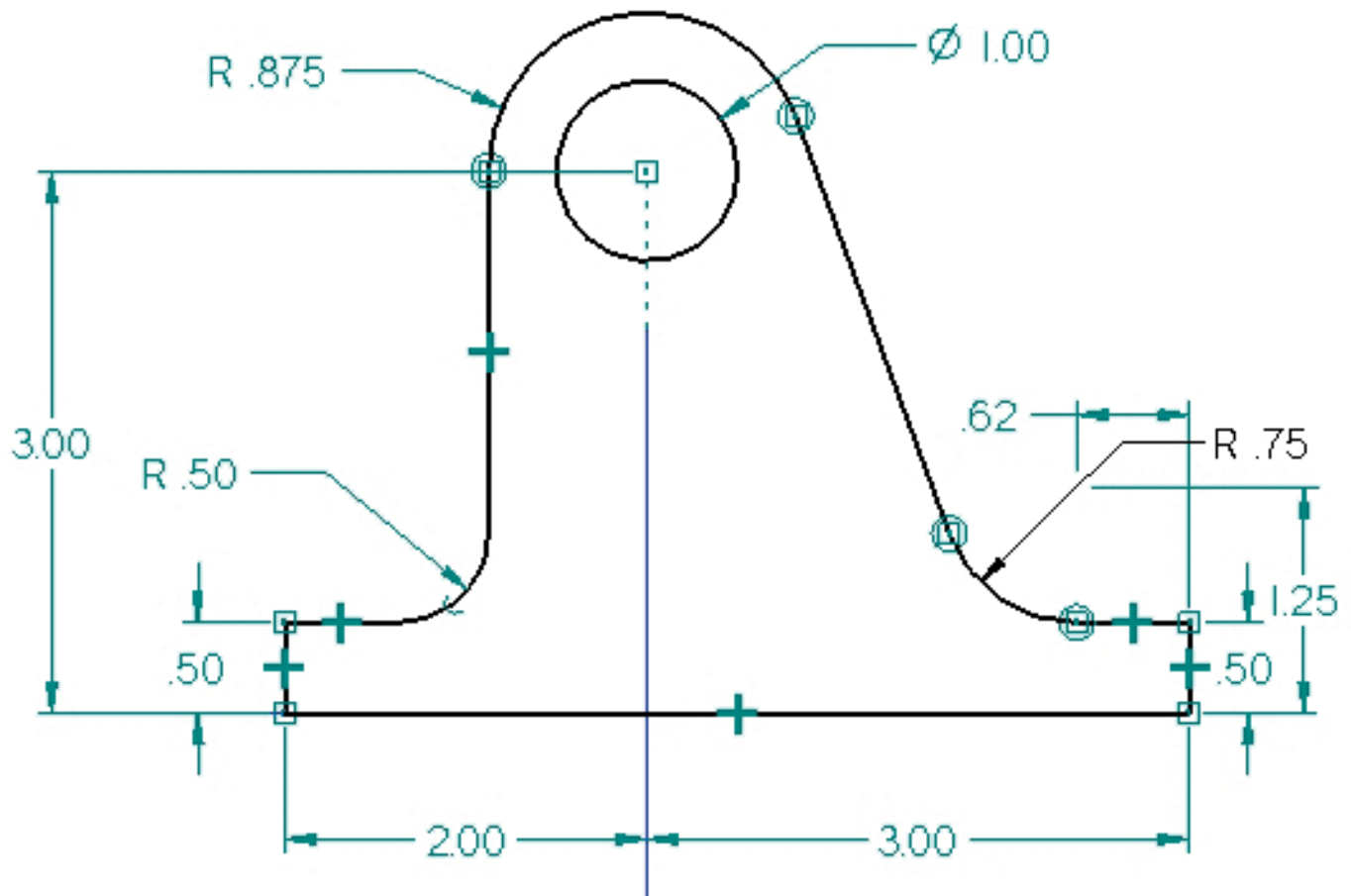
Tutorial - Shaft Support

✓ Start a new Part. Be sure to use **More ... Normeng.par** as the startup file

✓ Click Sketch

✓ Pick the Front (X-Z) reference plane.

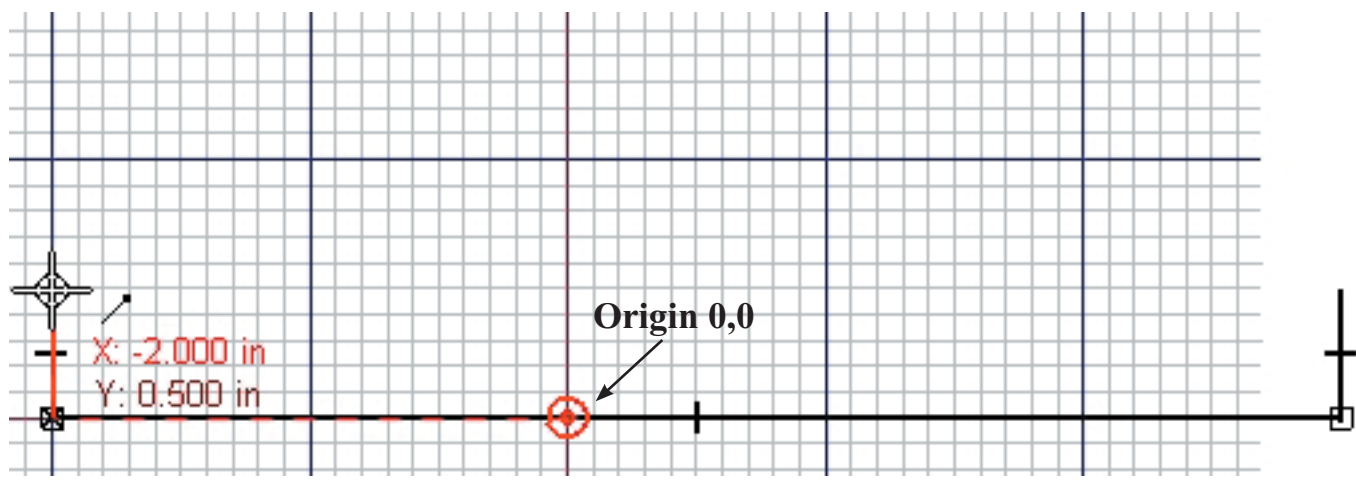
Refer back to this drawing for dimensions



1. Set up a decimal inch grid as shown in the previous section. 1.000 inch major lines and 10 minor lines.
2. Draw the base of the part as shown.

Start **Line** at -2.000,-0500 to -2.000, 0.000 to 3.000,0.000 to 3.000, 0.500

Re-size the grid if needed.



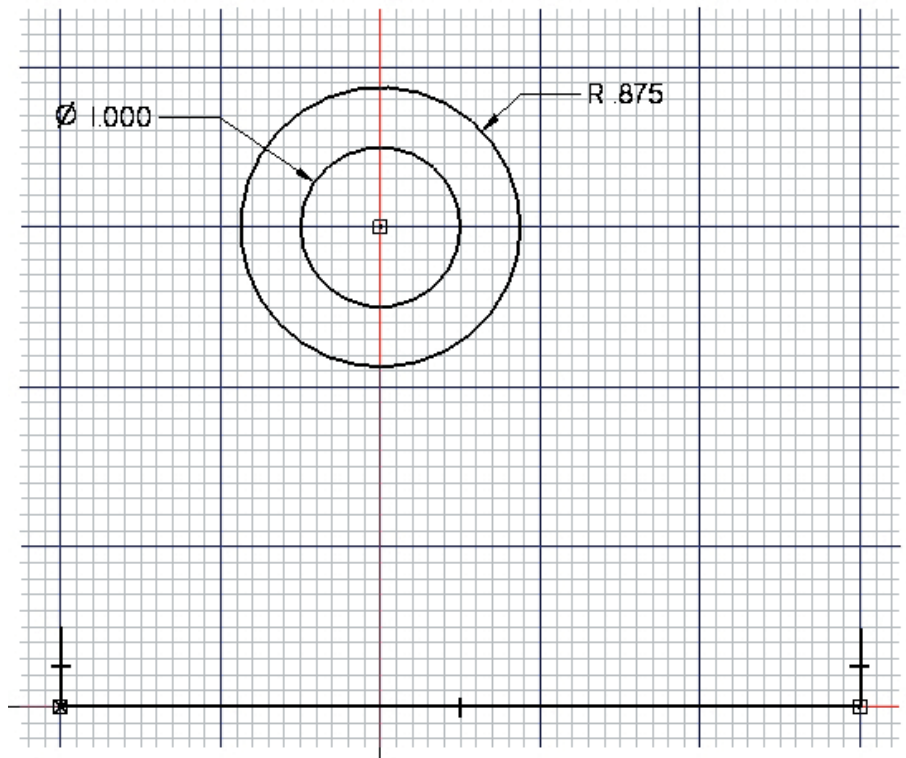
3. Draw two circles with center at

$X = 0.000, Y = 3.000$

The small circle is 1.000 diameter.
(\emptyset symbol = diameter)

The large circle is .875 radius.
(R = radius)

Use the **Tab Key** to move the input to radius in the circle ribbon bar.

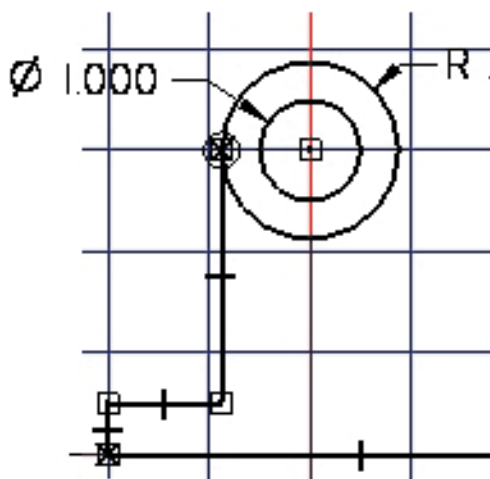
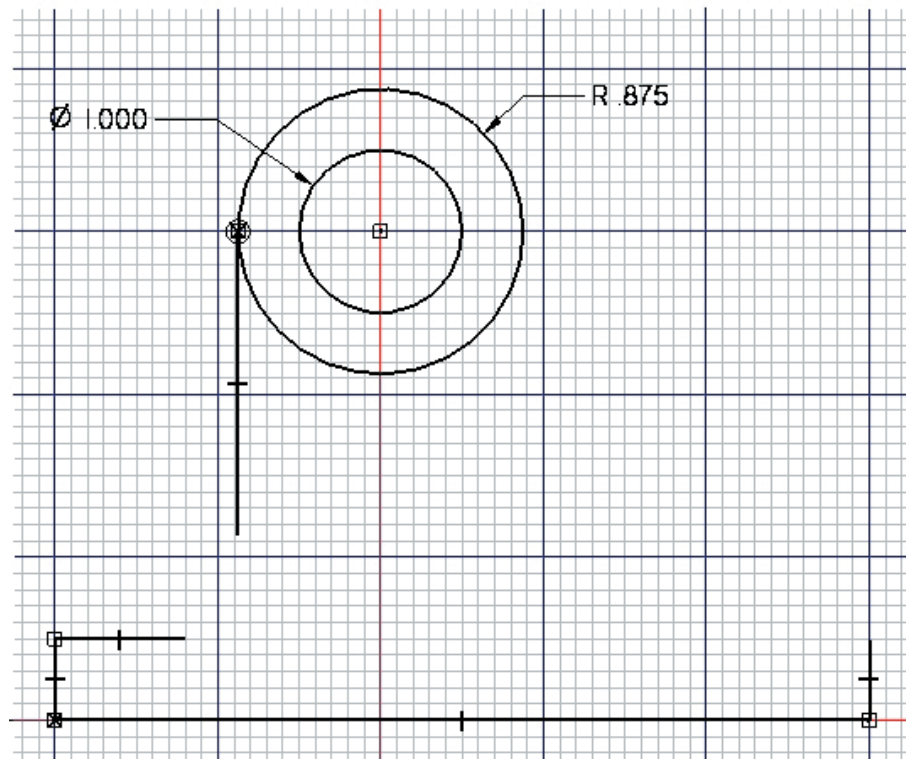


4. Draw a line straight down from the side of the large circle.

Make sure the line is tangent to the circle and is indicated vertical.

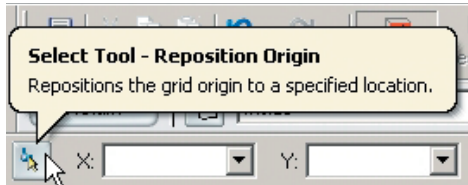
5. Draw a short line across from -2.000, 0.500

6. Use the **Trim Corner** tool to close the corner.



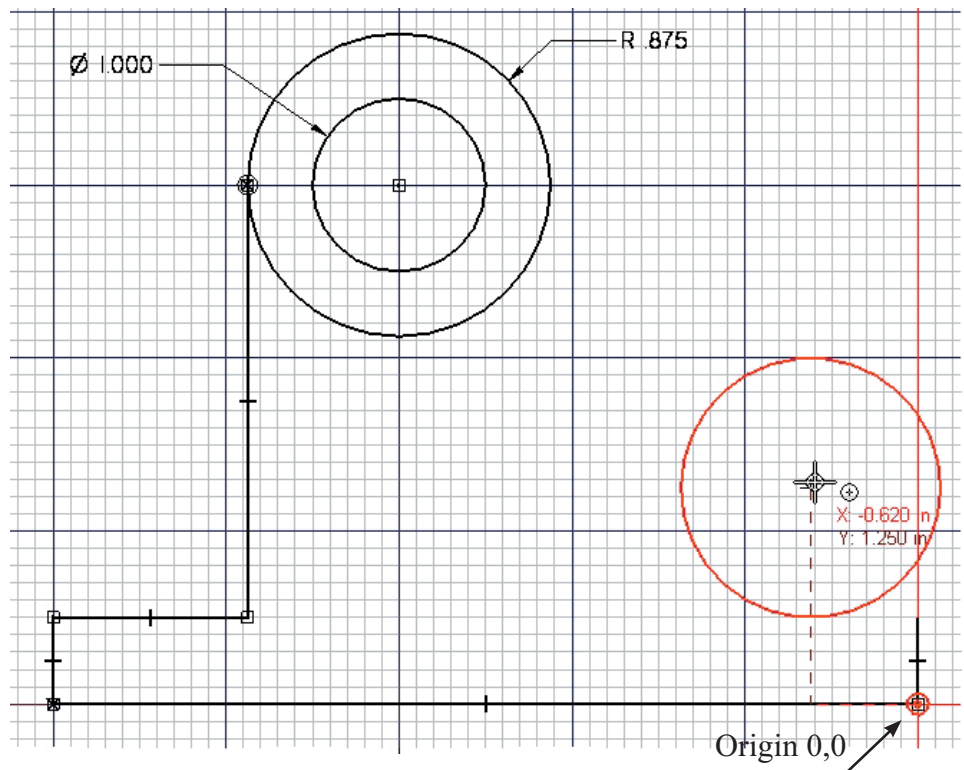
7. Re-set the grid origin to the end of the line at 3.000, 0.000

Click the **re-position origin** icon on the **grid ribbon bar**.



Click the end of the line at 3.000, 0.000.

8. Draw the circle at -0.620, 1.250
Key in the center location in the X: and Y: boxes in the **grid ribbon bar**.
Point to the center location.

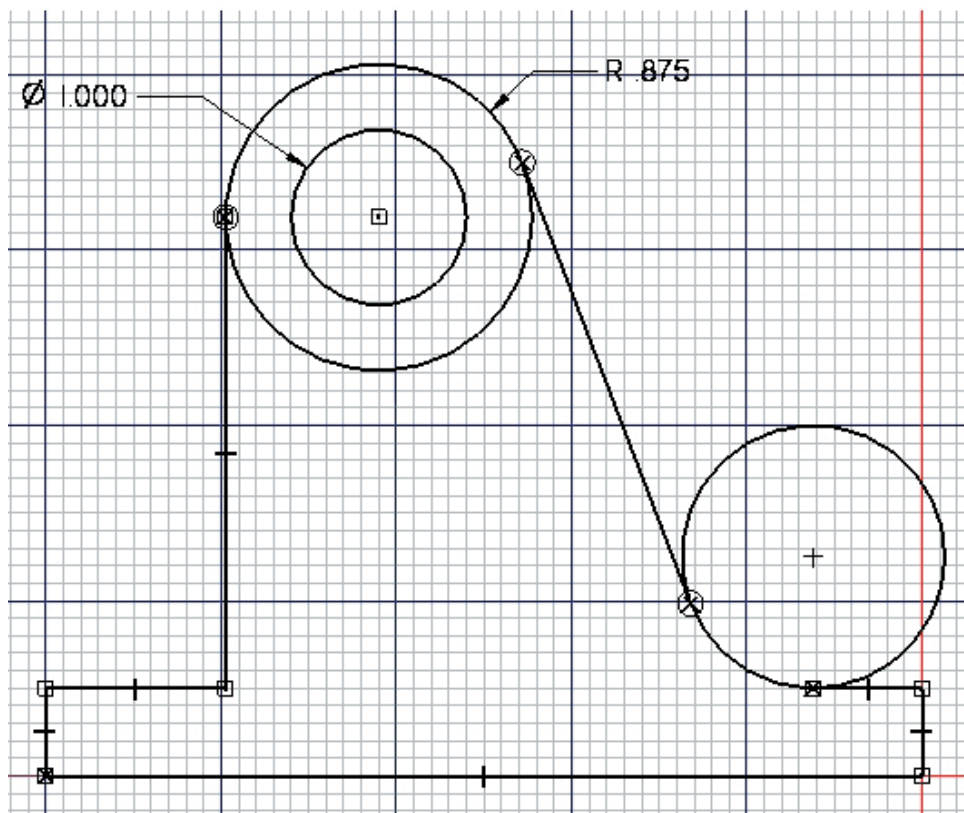


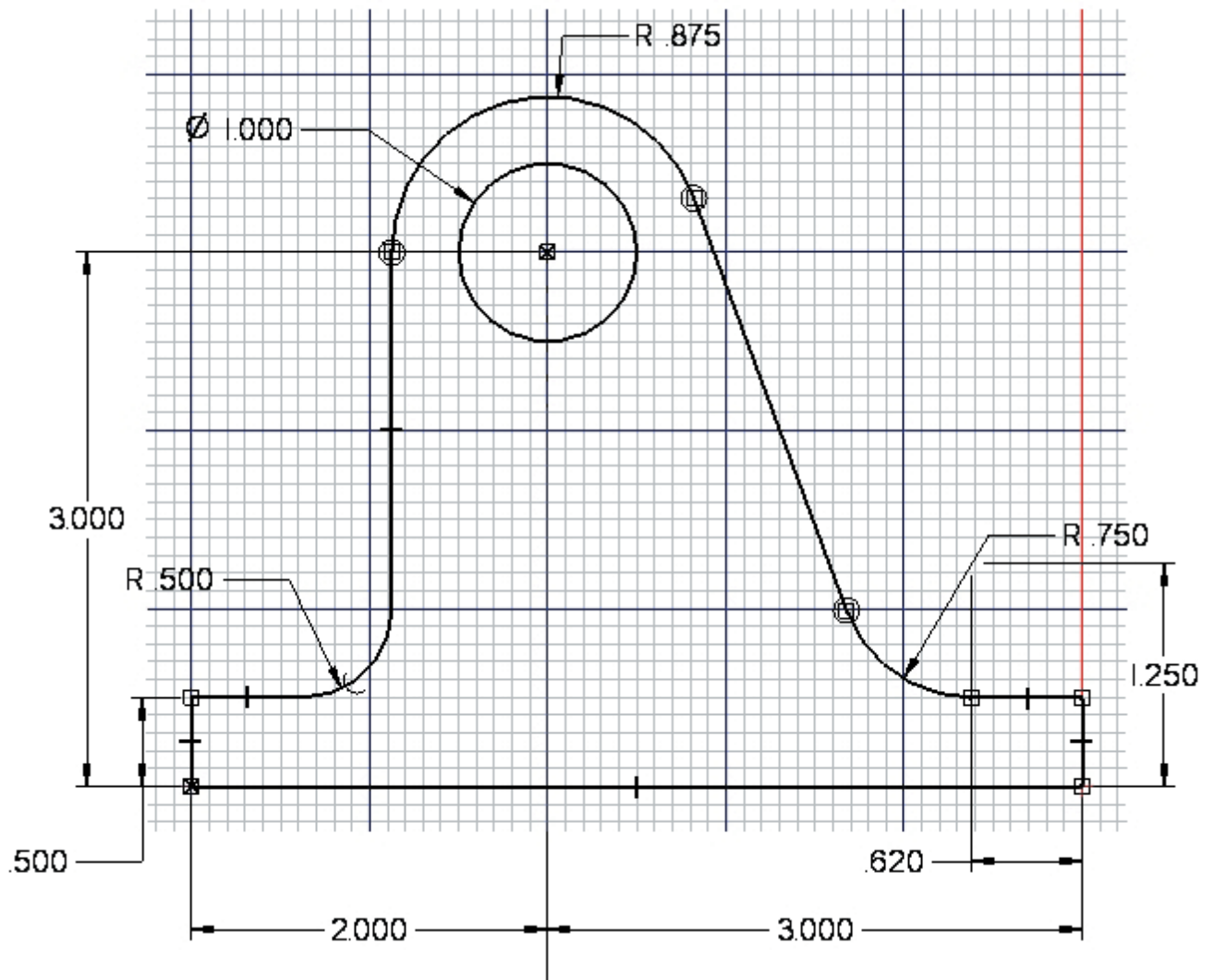
9. Draw line tangent to 2 circles as shown.

Use **tangent constraint** to be sure line is tangent to both circles.

10. Draw line from 0.000, 0.500 tangent to the bottom of the right circle.

Save the drawing as **ShaftSupport**





The bottom of the large circle and the upper side of the right circle.



The $.500$ radius at the left side of the drawing. (We could have used Fillet at step 6.)

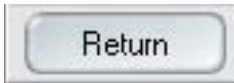
Place dimensions on the sketch as shown. Use dimension constraints to correct any location errors.

Be sure the center of the two circles is in vertical alignment with the Side (Y-Z) reference plane.

Save the drawing.

(Next, we will make a part and a drawing).

13. Click the **Return** icon



The sketch can be seen on the Front (X-Z) plane.

14. Click **Protrusion**

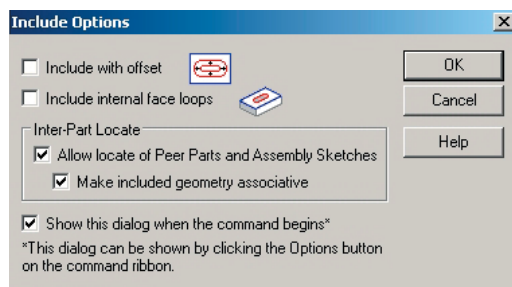


15. Select the **Front (X-Z)** plane



16. Click **Include**

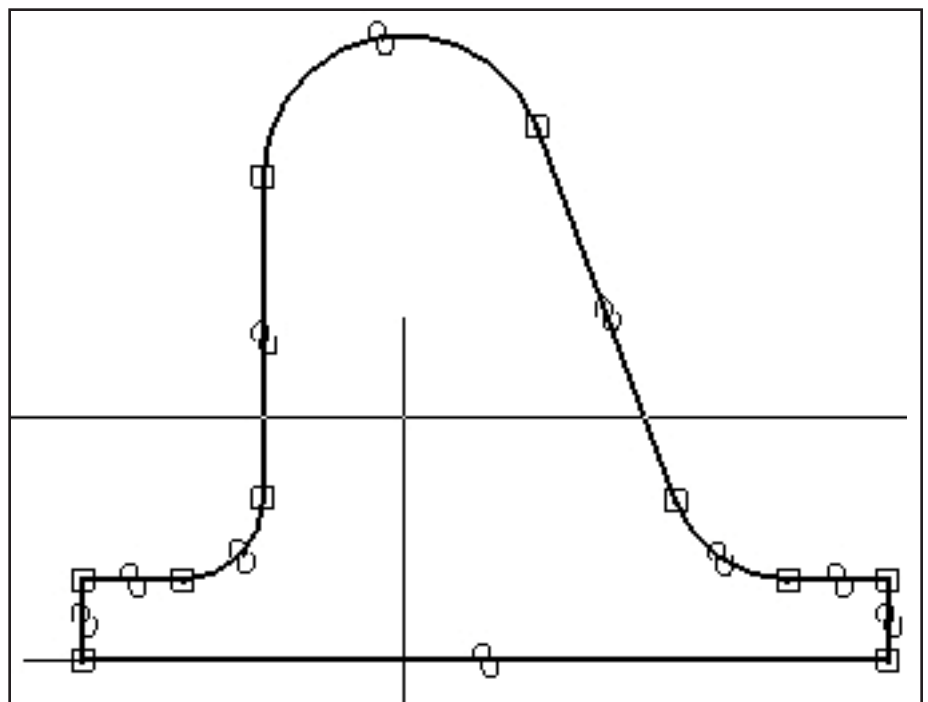
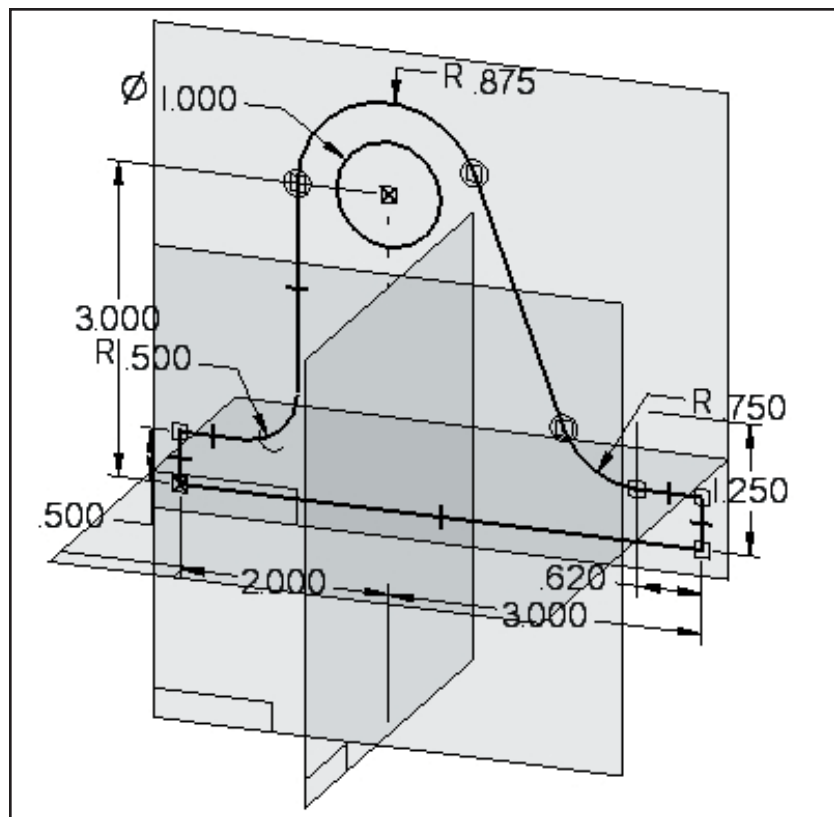
Just O.K the dialog box.



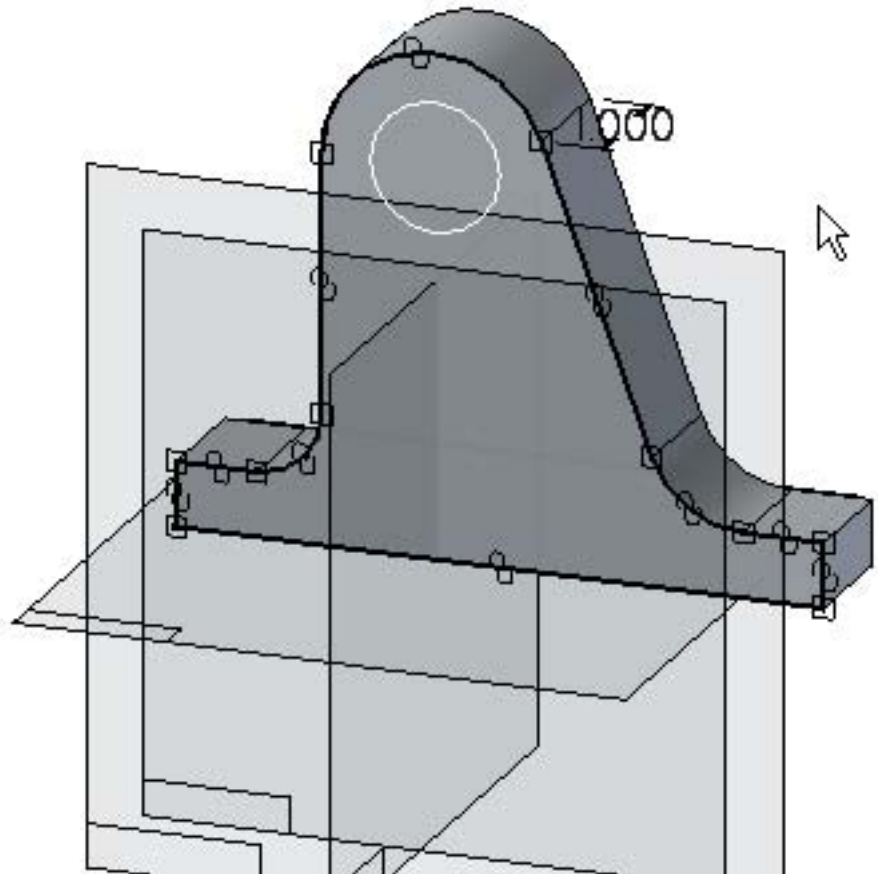
17 Click each of the lines and arcs that define the outline of the part. A Include icon will appear on each item.

Do NOT include the reference planes.

The final outline should be a closed loop.



18. Click **RETURN** and set the depth of the part to 1.000



19. Click **Cutout**

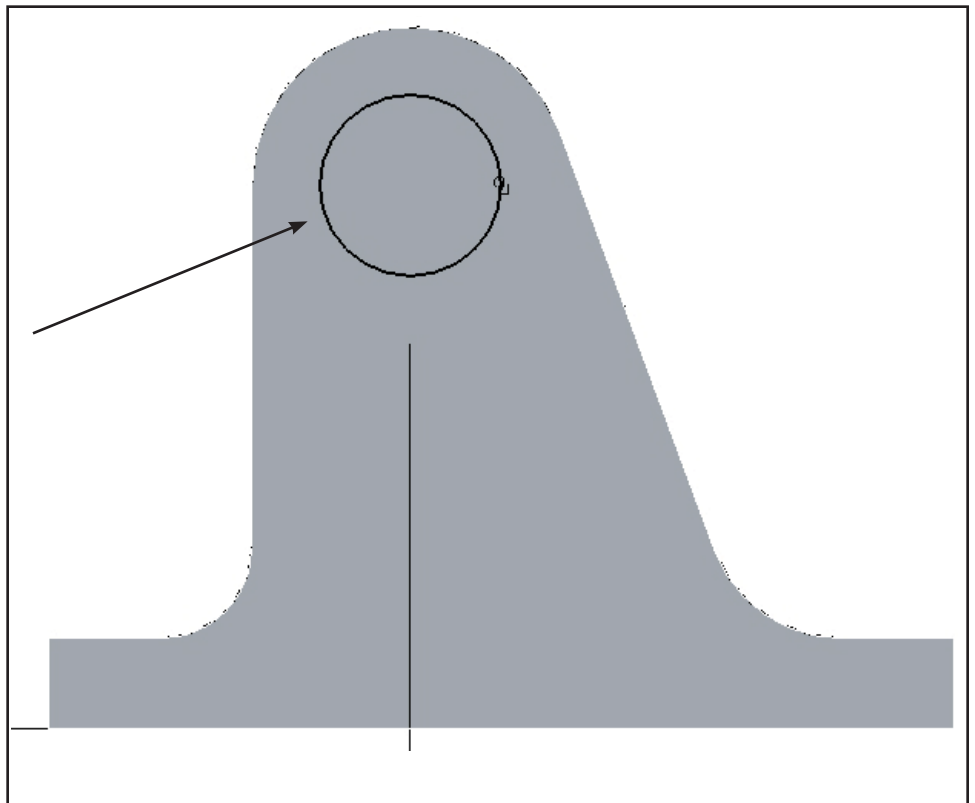


Select the **Front (X-Z)** plane.

20. Click **Include**
Just OK the dialog box.



Include the 1.00 diameter center circle.

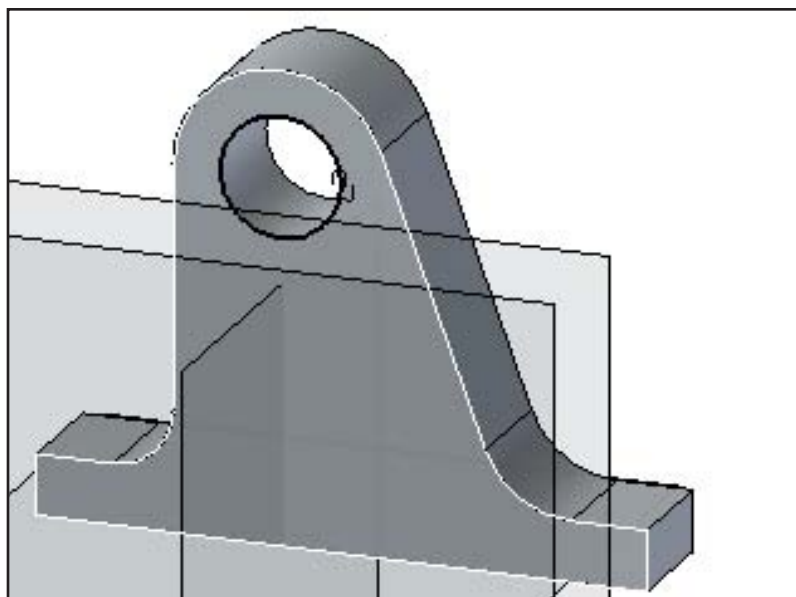
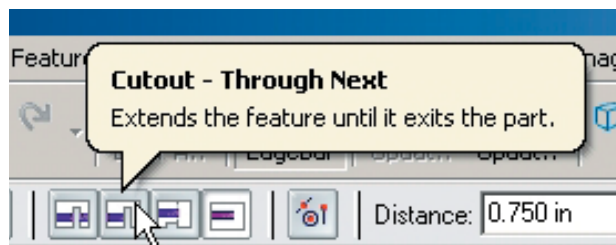


20. Click **RETURN**

Click the **Through Next** option in the **cutout ribbon bar**.

Use the mouse to point the cutout arrow toward the back of the part.

Save the model.

**Part Properties.**

Collecting data for Bill of Material lists is an important documentation step. If your instructor provides data, fill in the dialog boxes. If not, just **Cancel**.

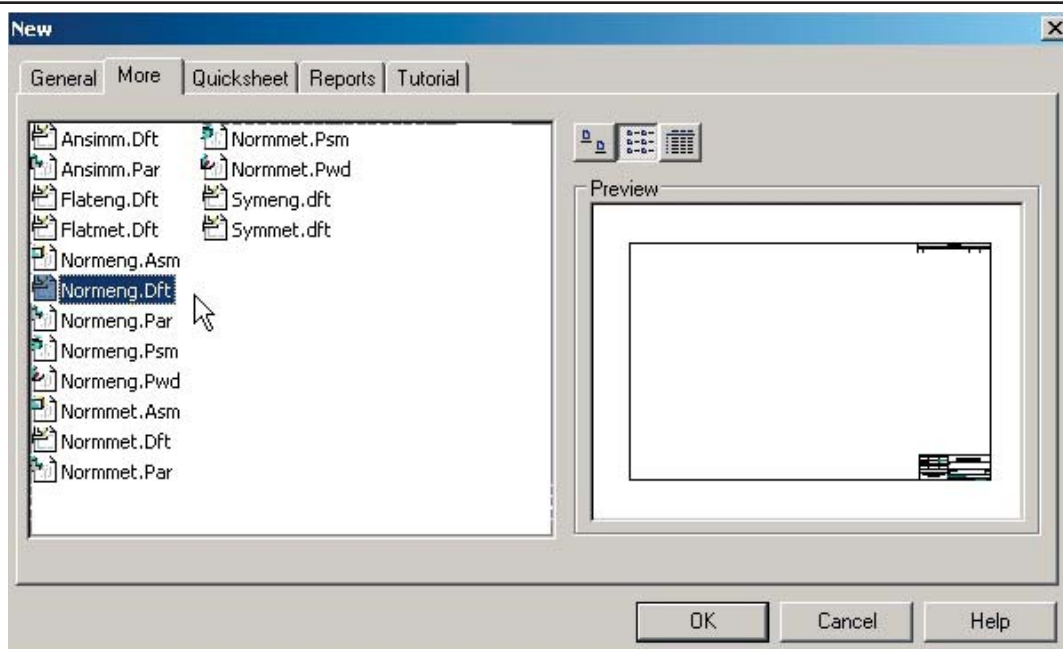
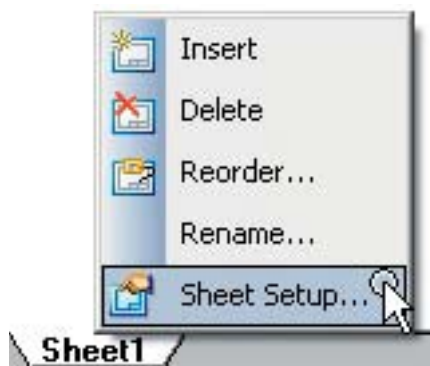
Making a drawing.

Be sure the part is saved from the previous steps.

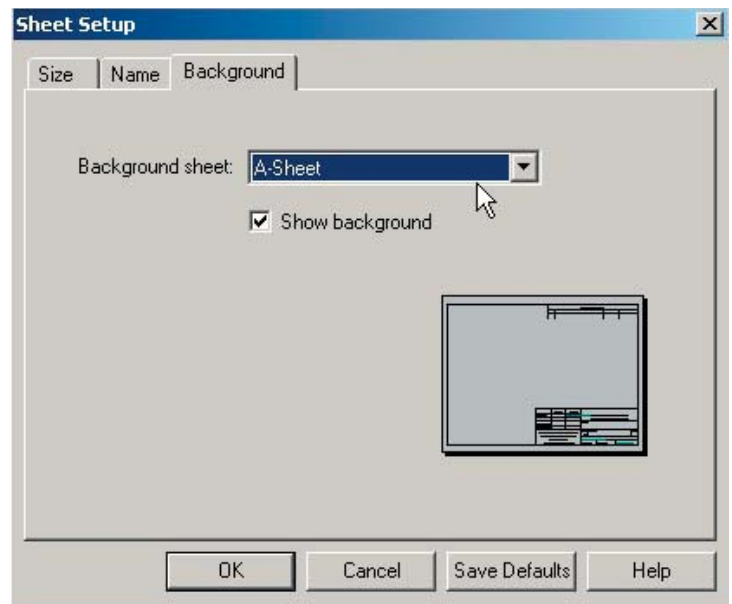
21. Click **New** on the File Menu.

Click the **More** Tab and select **Normeng.dft**

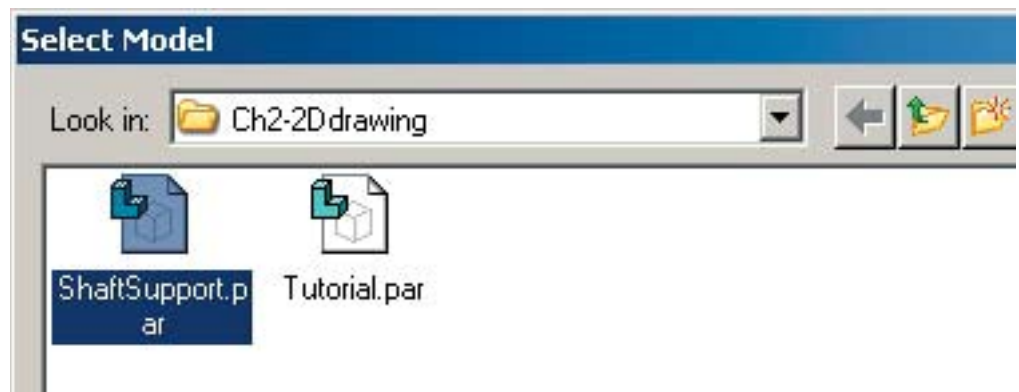
A default drawing sheet will show.

22. At the bottom of the screen, **Right Click** the **Sheet** option.

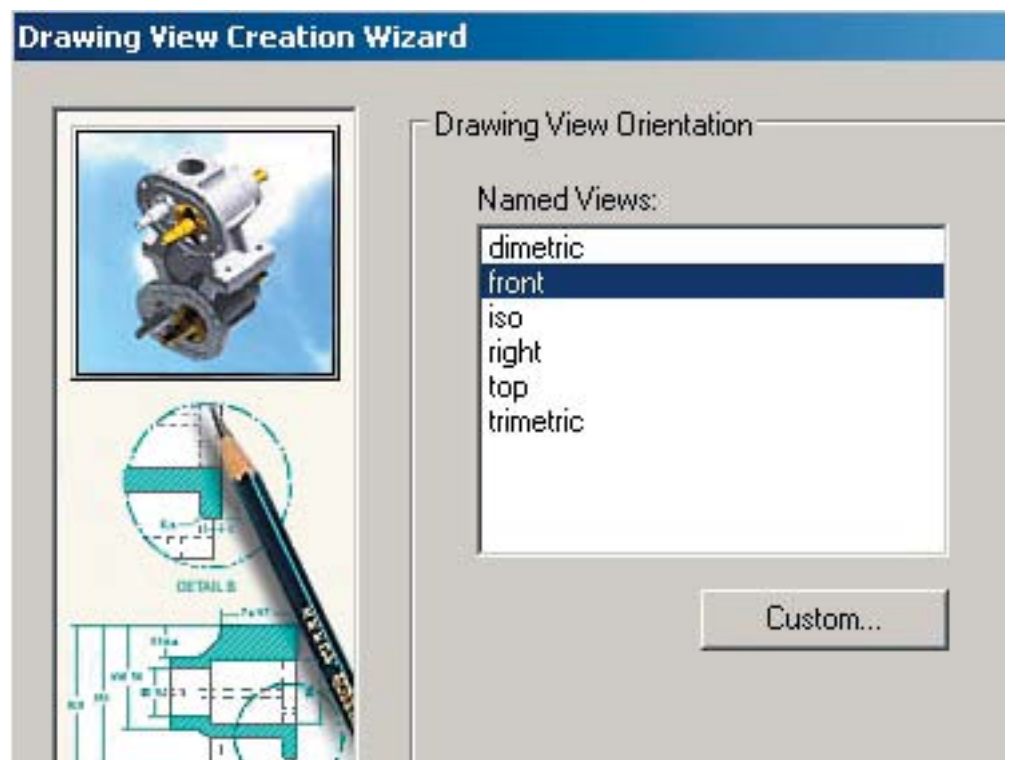
23. Click the **Background** Tab
and select **A-Sheet** (8 1/2 x 11 size)



24. Select the
ShaftSupport.par model
from the folder where you
save it.



25. Select
Front
for the key view.



26. The **Front** view is selected by default.

Click **TOP**, **RIGHT** and **ISOMETRIC** as shown.

Click **Finish**.

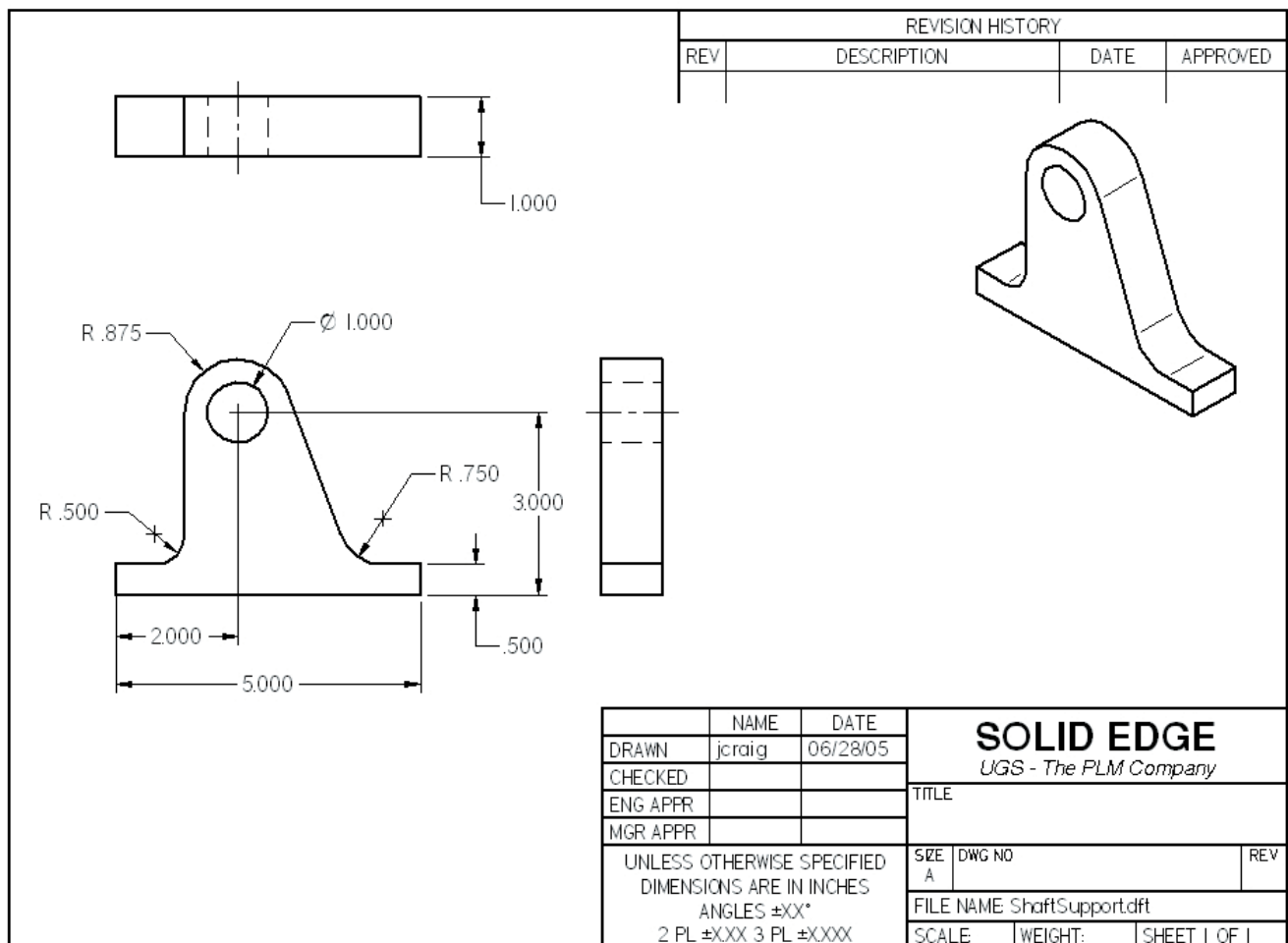
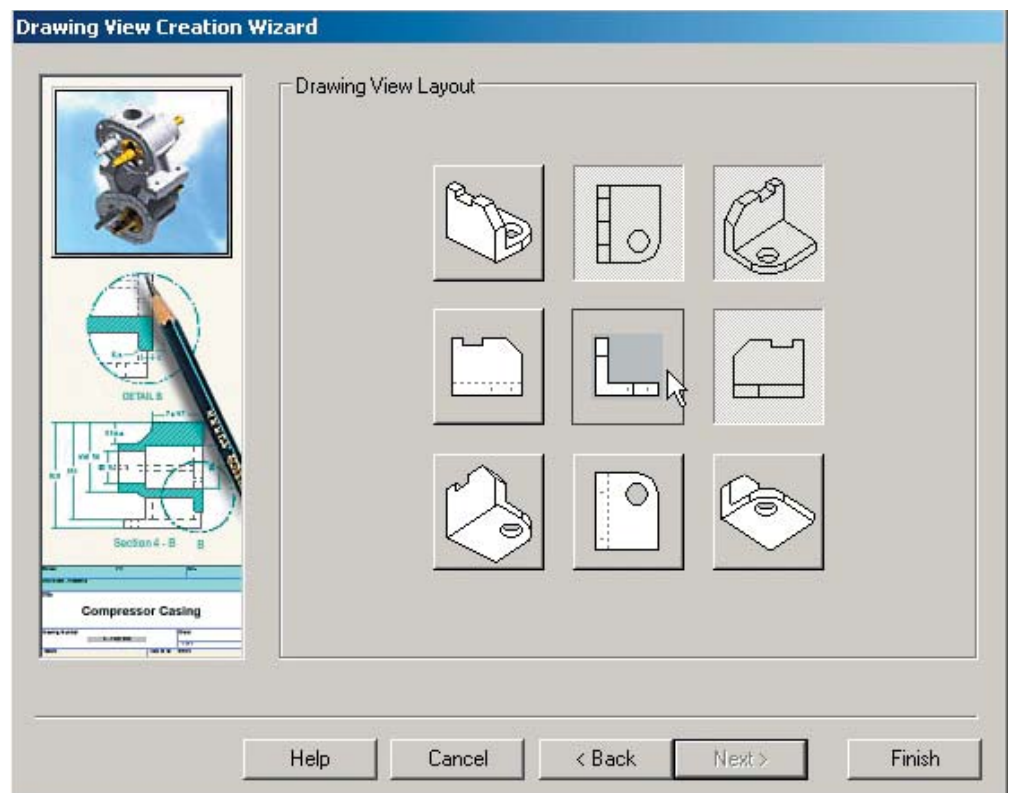
The drawing will fit at a scale of 1:2. Select that option before placing the views.

Click a view to move it on the sheet.

Place some dimensions as shown.

Save and **Print** the drawing.

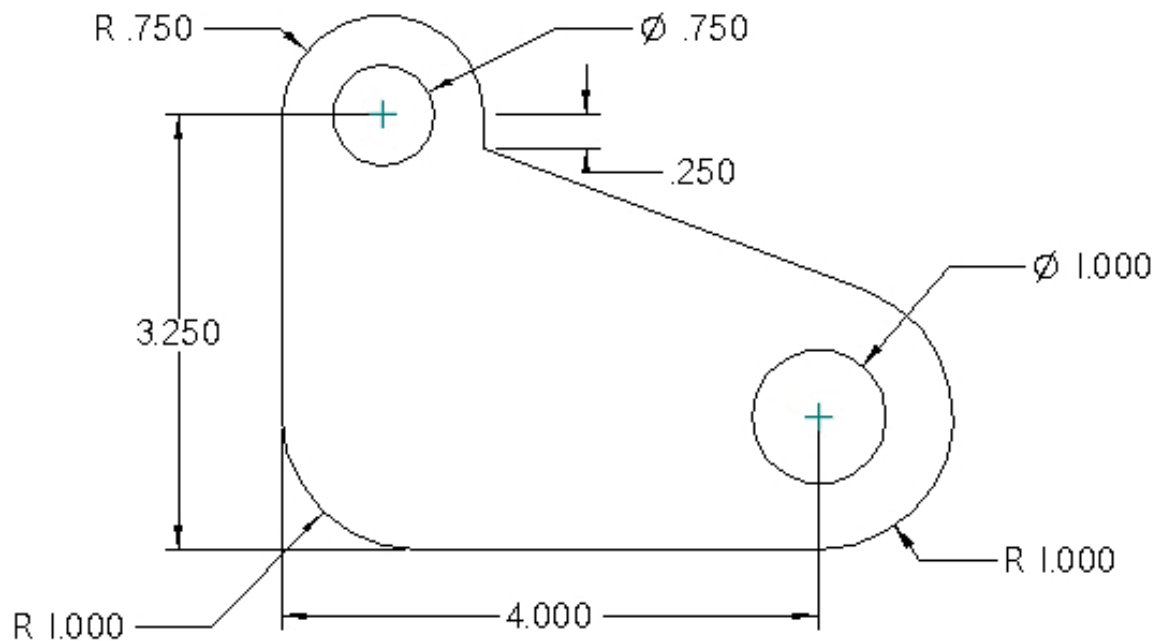
End of tutorial



Problem 1.

- ___ Layout the sketch in the Part Environment.
- ___ Dimension the sketch.
- ___ Set up an “A” size sheet in the Draft Environment.
- ___ Place the views of the part as shown in the previous tutorial.
- ___ Fill in the Title Block as shown. Use your name in the DR BY: box.
- ___ Dimension the drawing.
- ___ **Save** and **Print** the drawing.

Thickness = 1.000



		COMPANY OR SCHOOL NAME	
		PART NAME : BAFFLE PLATE	
	DR BY: CRAIG, J		

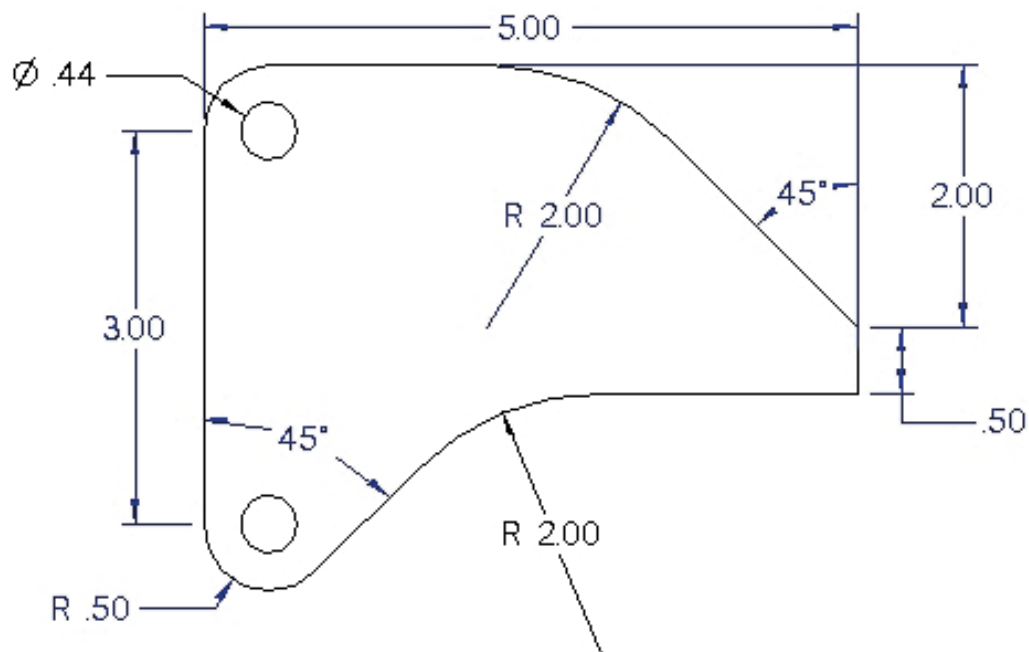
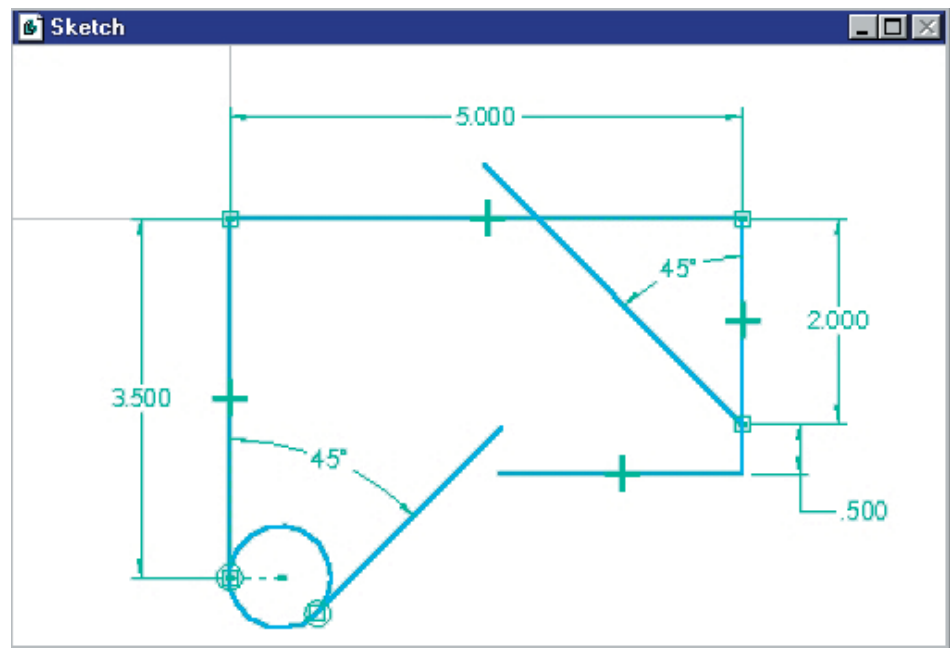
Problem 2.

Create the sketch for the part shown below.

Follow the steps as shown in Tutorial 1.

Create a drawing,

dimension save and print.

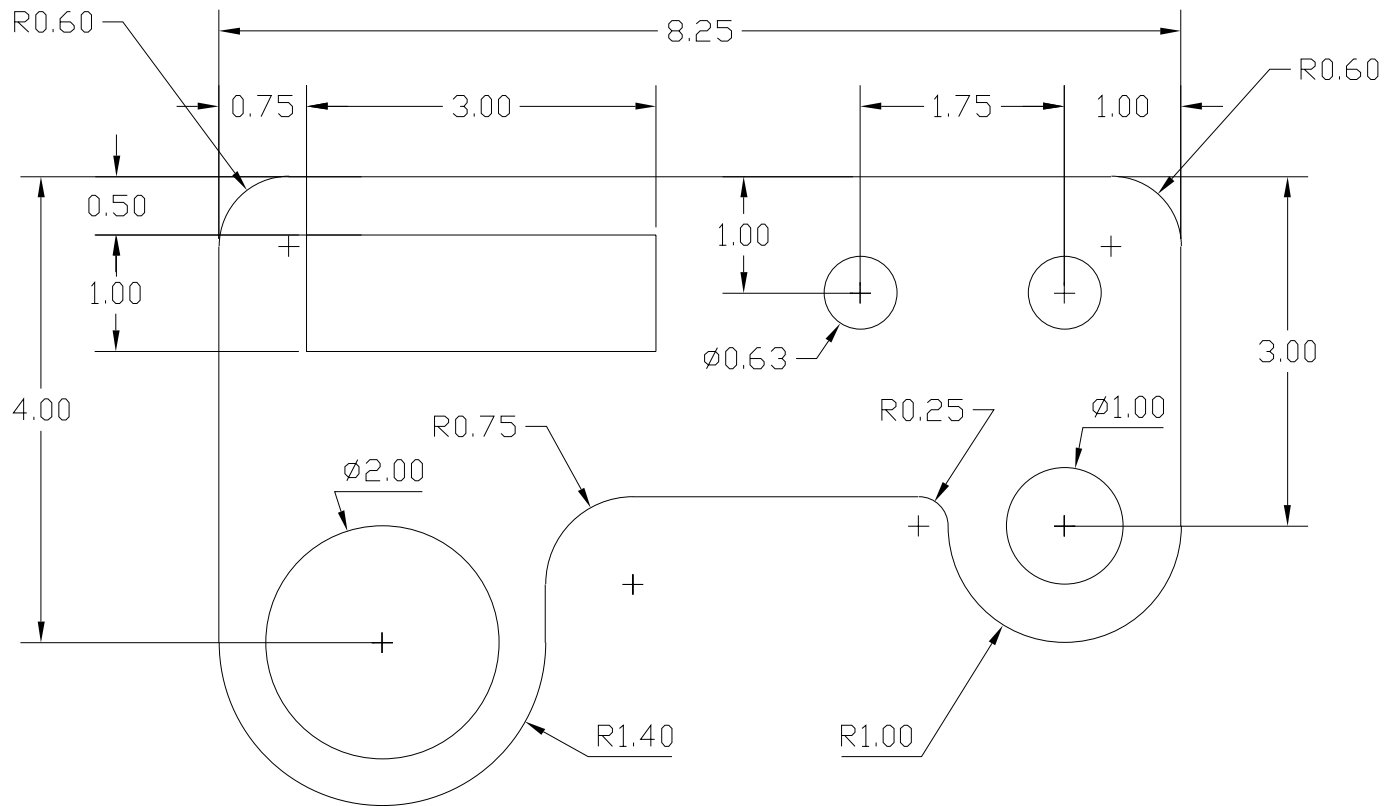


Thickness = .125

		SCHOOL OR COMPANY NAME	
		PART: SCRAPER PLATE	
		DR BY: LAST, F	

Problem 3.

Create sketch and model. Make a drawing as shown in Tutorial 1. Dimension, Save and Print.



Part = Template

Thickness = .25

Problem 4.

Do the “Modeling Parts with Sketches” tutorial. Make a drawing as shown in Tutorial 1. Save and Print the model and the drawing.

Solid Edge Tutorials

Tutorials

Part Tutorials

- Introduction to Part Modeling
- Modeling Pattern Features in Solid Edge
- Modeling Mounting Boss and Vent Features
- Modeling Parts With Sketches**
- Modeling Thin-Region Features
- Modeling a Yoke
- Modeling a Part Using Surfaces
- Repairing Disconnected Features
- Modeling a Family of Parts in Solid Edge
- Modeling Helical Features
- Modeling Lofted Features
- Modeling Molded Parts
- Modeling Swept Features
- Using Variables in Part Models

Assembly Tutorials

- Building a Slider Assembly in Solid Edge
- Modeling a Sheet Metal Part in the Context of an Assembly
- Building a Grinder Assembly in Solid Edge
- Creating a Tube Part with XpresRoute
- Creating a Wire Part with XpresRoute
- Building a Family of Assemblies
- Modeling a Cover Associatively within an Assembly
- Creating and Publishing Virtual Components

Draft Tutorials

- Detailing Drawings with Solid Edge Draft
- Producing an Exploded Assembly View
- Modeling Parts from Drawing Views
- Creating Hole Tables in Detailed Drawings

Sheet Metal Tutorials

- Introduction to Sheet Metal Modeling
- Creating a Sensor
- Creating a Feature Library
- Modeling Contour Flanges in Solid Edge

Weldment Tutorials

- Constructing a Weldment

Other Tutorials

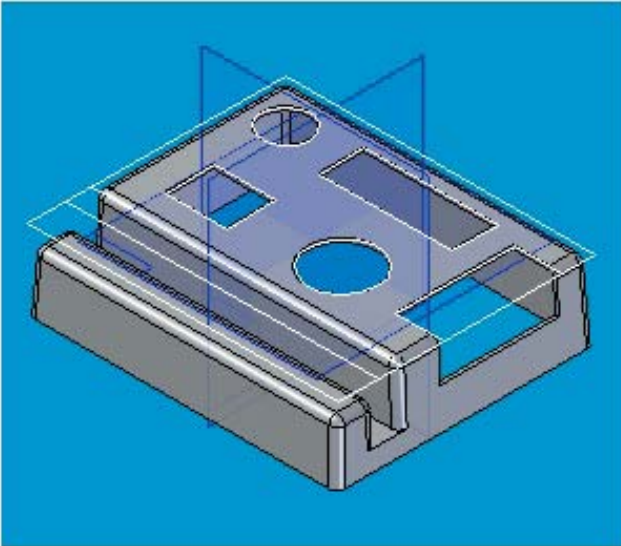
- Rendering with Virtual Studio+

Modeling Parts With Sketches

This tutorial demonstrates a typical workflow for using a sketch to model a part with Solid Edge. It demonstrates the advantages of drawing several profiles in one sketch, and using those profiles later to construct other features.

This is an intermediate tutorial. You might find it easier to follow if you first complete the Introduction to Part Modeling tutorial.

Estimated time to complete: 45 minutes



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