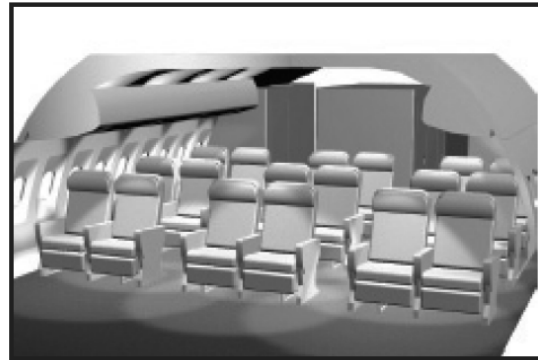# Engineering and Technical Drawing
Using Solid Edge
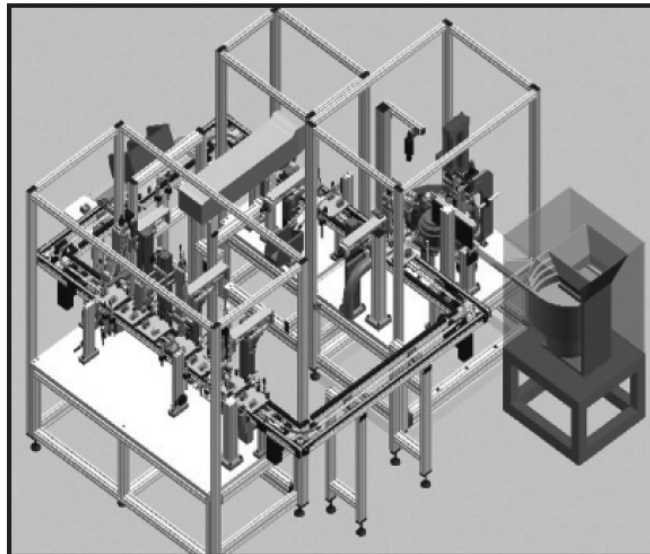### Version 19

Jerry W. Craig
Washington University – School of Engineering
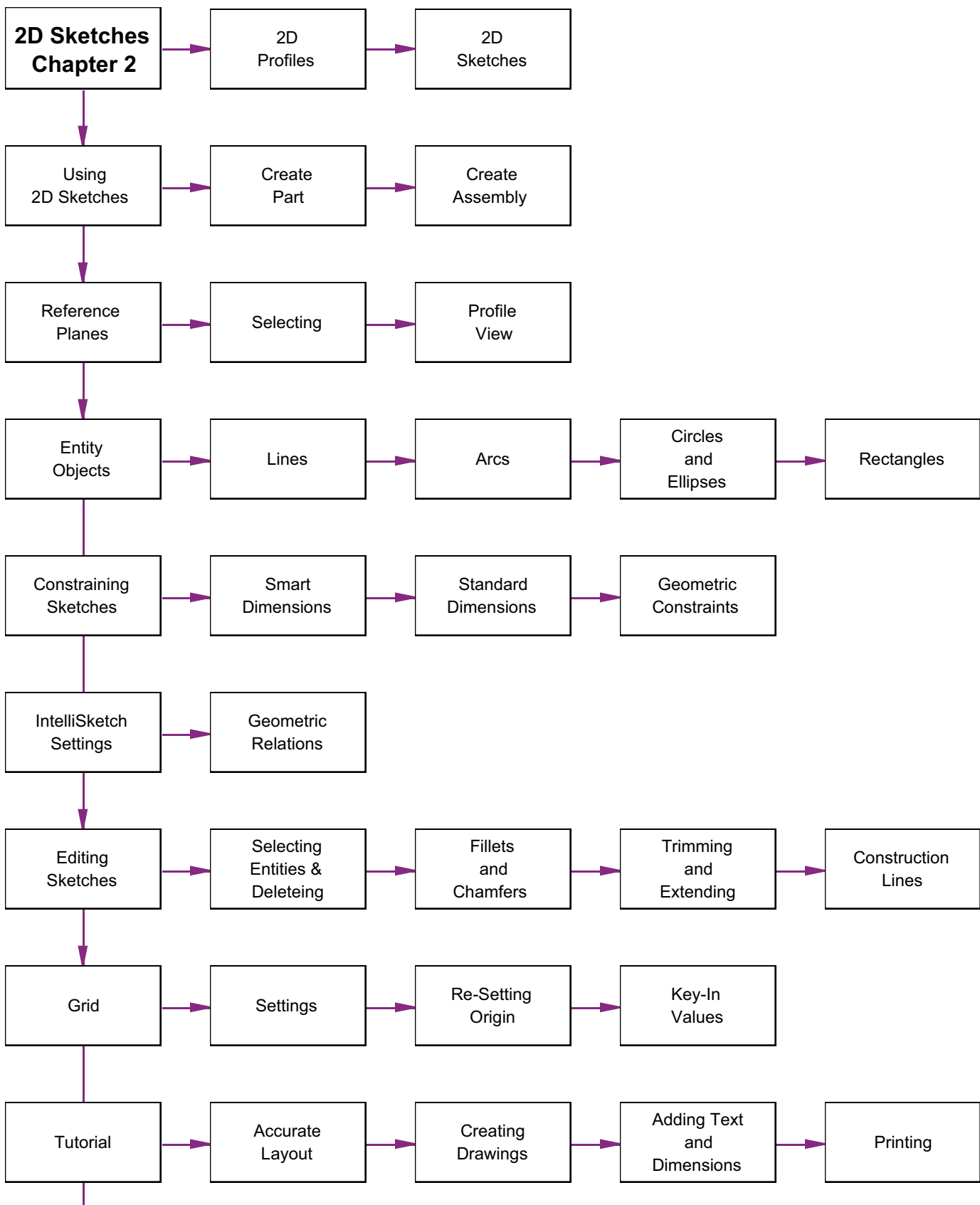**St. Louis, Missouri**

Images from
Solid Edge
Gallery

2D Sketches Chapter 2	→	2D Profiles	→	2D Sketches

Using 2D Sketches	→	Create Part	→	Create Assembly

Reference Planes	→	Selecting	→	Profile View

Entity Objects	→	Lines	→	Arcs	→	Circles and Ellipses	→	Rectangles

Constraining Sketches	→	Smart Dimensions	→	Standard Dimensions	→	Geometric Constraints

IntelliSketch Settings	→	Geometric Relations

Editing Sketches	→	Selecting Entities & Deleteing	→	Fillets and Chamfers	→	Trimming and Extending	→	Construction Lines

Grid	→	Settings	→	Re-Setting Origin	→	Key-In Values

| Tutorial	→	Accurate Layout	→	Creating Drawings	→	Adding Text and Dimensions	→	Printing

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.
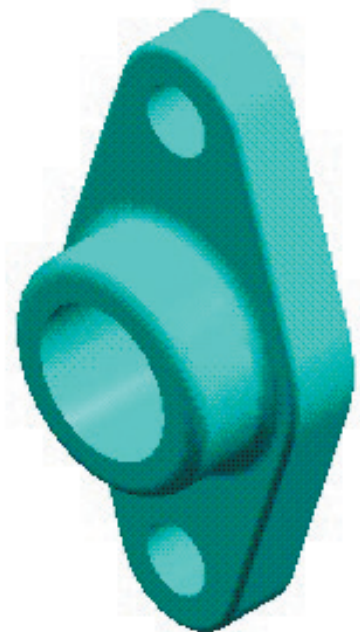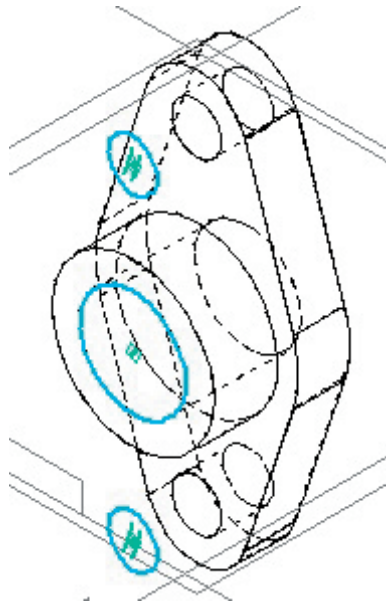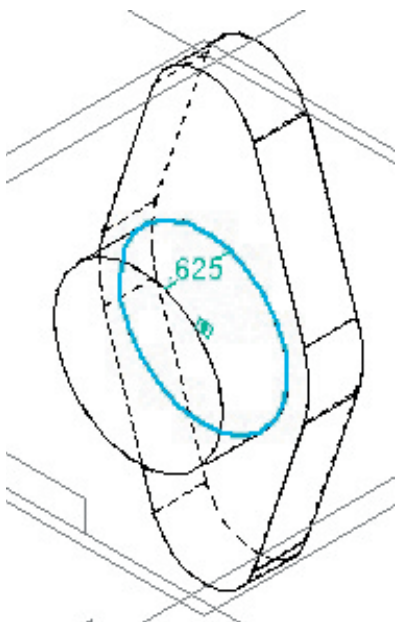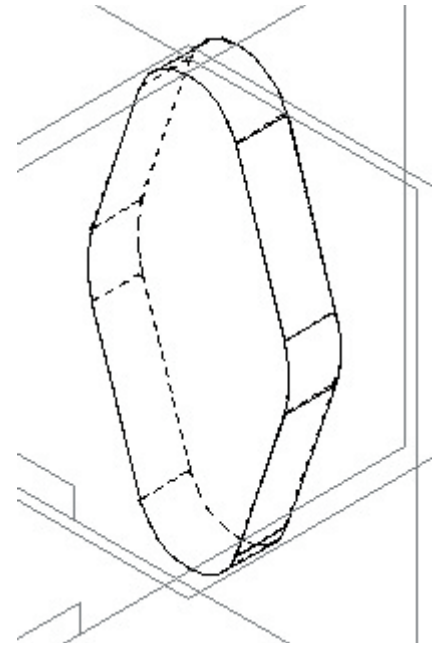
Ø 1.000

1.600

Ø 2.000

1.600
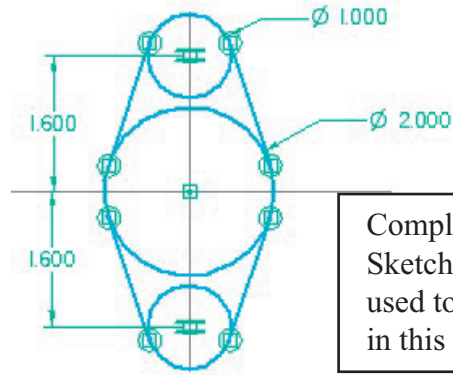
Complete Master Sketch. (Cannot be used to create a part in this form).

Ø 1.000

1.600

Ø 2.000

1.600
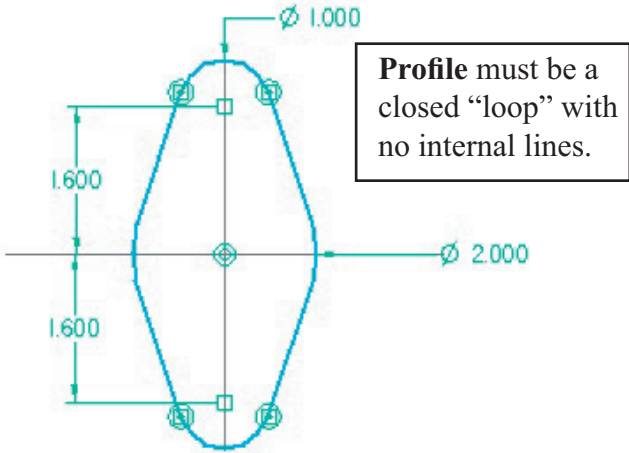
Profile must be a closed "loop" with no internal lines.

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

Resulting 3D object with depth added.

.625

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 resize to the new dimensions. Each part will be the right size and in the right place.
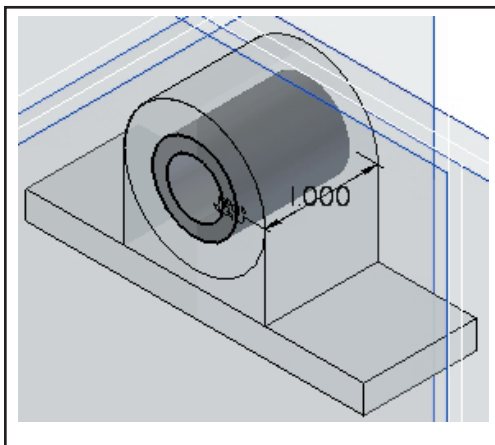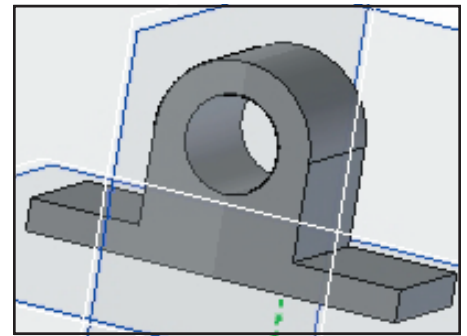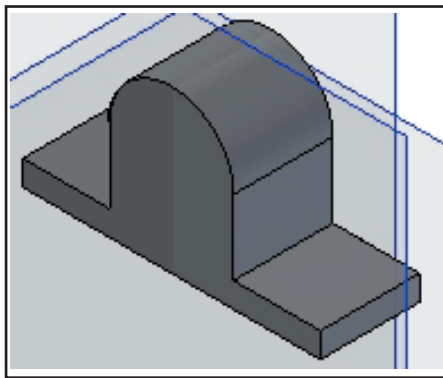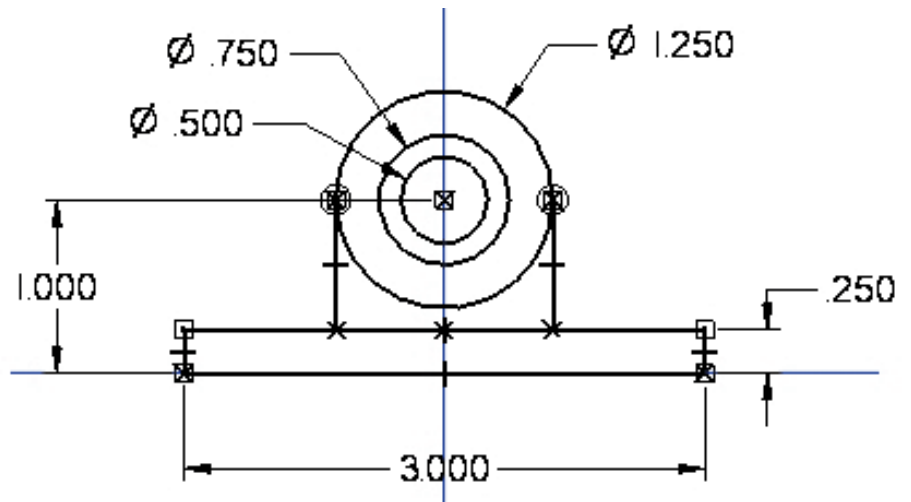
Bronze
Bearing
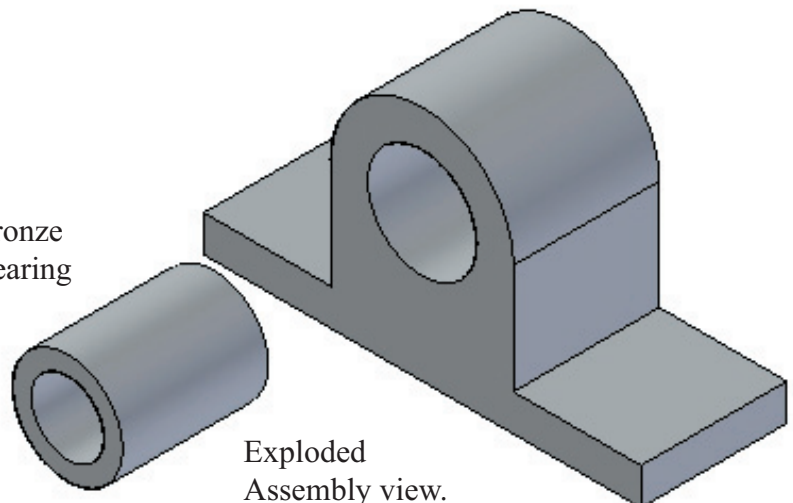
Exploded
Assembly view.

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 staring directly with the Protrusion or revolved Protrusion icons. *Master sketches give more control over the modeling process.*
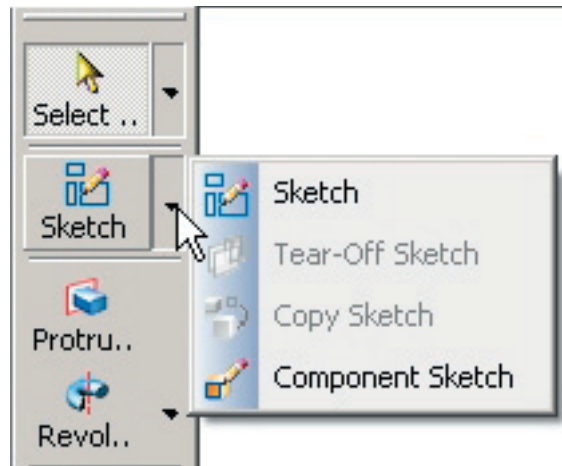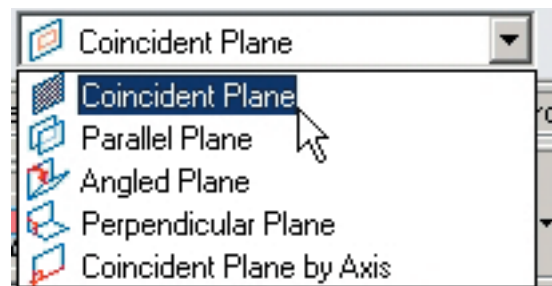
Component Sketches are placeholders for existing parts or for related items yet to be designed.

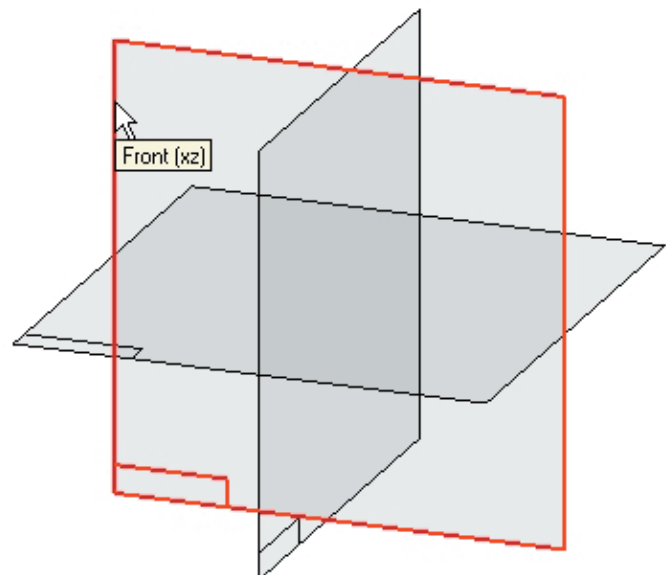Once **Sketch** is started, The **Ribbon Bar** displays prompts and options.
A **Reference Plane** must be selected. Or, a new reference plane must be defined as a 2D drawing plane.
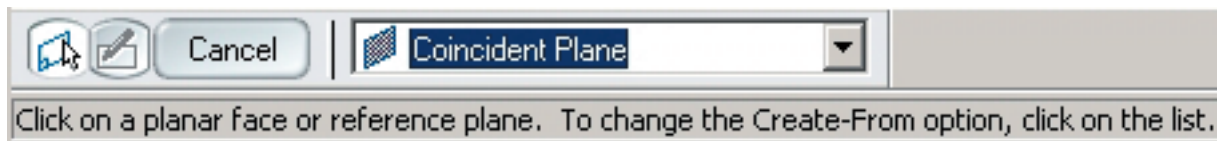
A pop down dialog box allow the selection of alternate planes for starting the sketch.

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 hve several options selected from pop down menus.

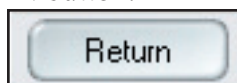The first tutorial (chapter 1) introduced the 2D drawing process.

Edge of Side (Y-Z) plane

Edge of Top (X-Y) plane

2D drawing surface = Front (X-Z) plane
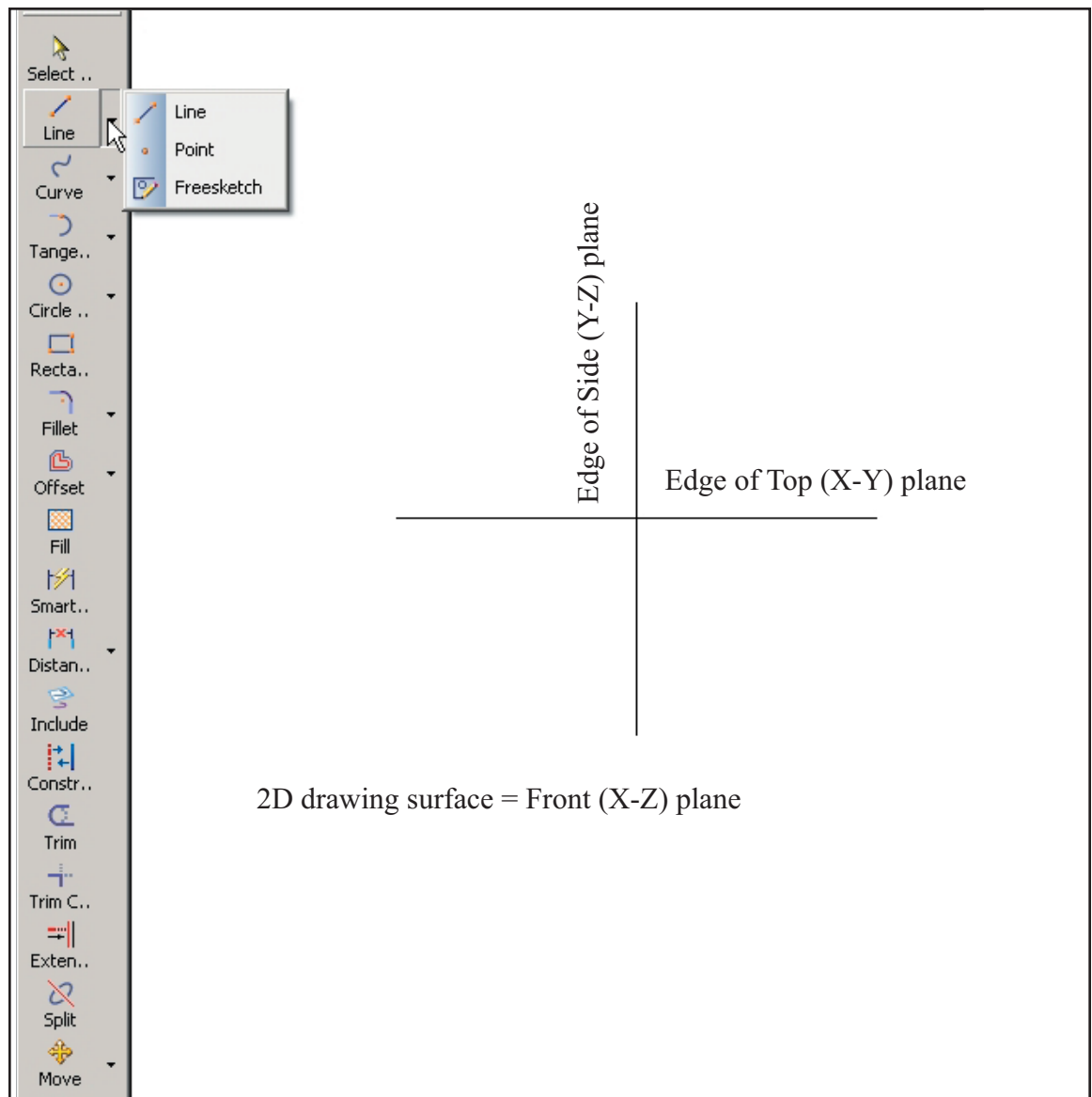
Exercise:

√ Start Solid Edge.
√ Click Sketch.
√ Click the Front reference plane.

Draw some shapes.
Explore the pop downs

Press ESC key to stop a command.
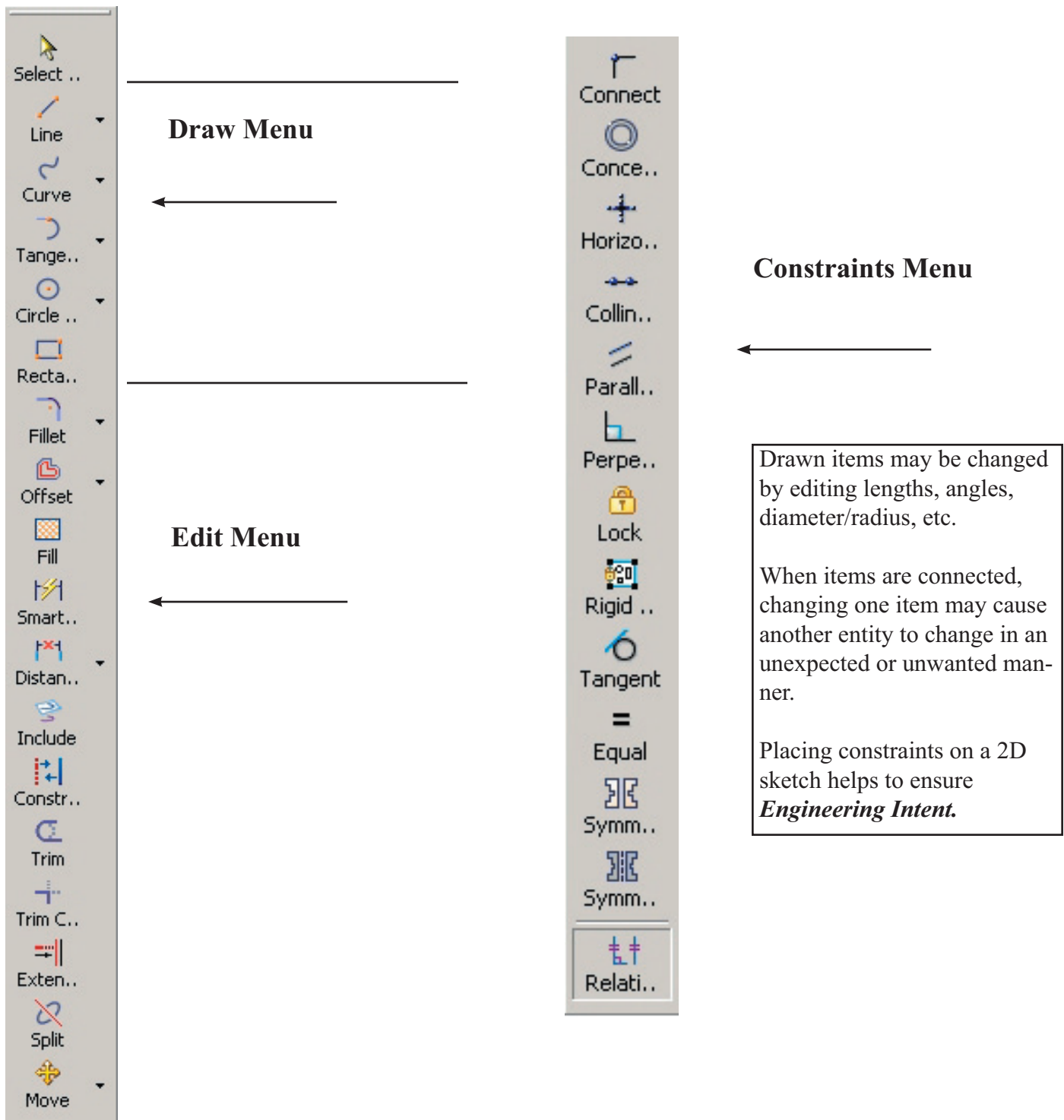
When finished, click the RETURN button.

Return

Exit Do not save the drawing.

Draw Menu

Edit Menu

Constraints Menu

Drawn items may be changed by editing lengths, angles, diameter/radius, etc.

When items are connected, changing one item may cause another entity to change in an unexpected or unwanted manner.

Placing constraints on a 2D sketch helps to ensure *Engineering Intent.*
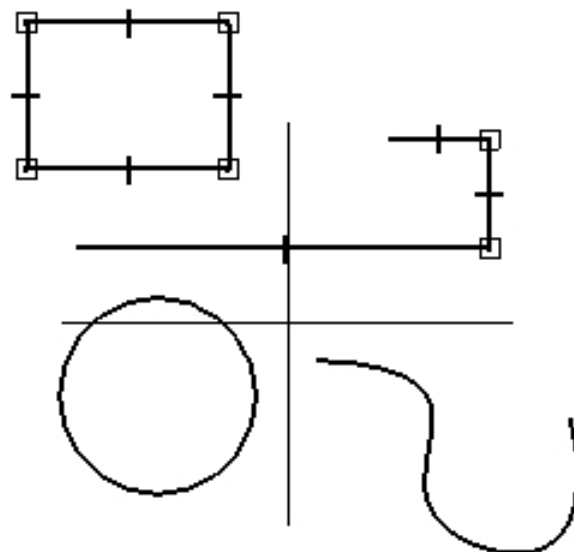
Constraining sketches is an important step in the design process. Many features may be changed as the design evolves. Making sure the master sketch changes in a predictable manner is critical.

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.**

Color   Type   Thickness

Active
Line
Style

Toggle between line and arc

Alignment Indicator

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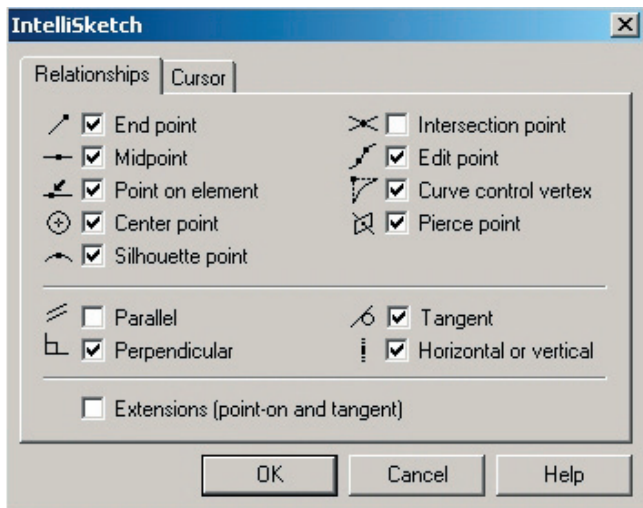
Intent Zone
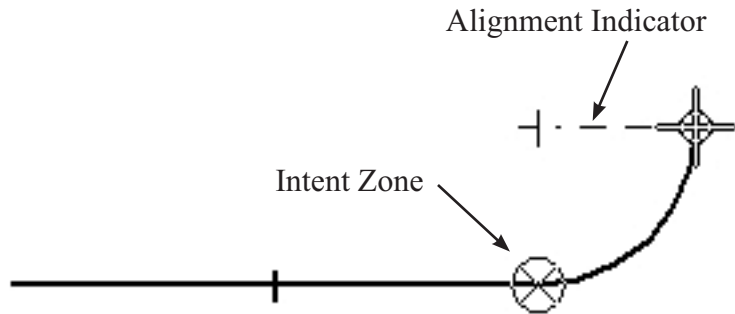
Alignment indicators flash when horizontal. vertical, extensions or other alignments are sensed.

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

Alignment Indicators
On/Off toggles.

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

Distance Between dimension option was used to fix point location.
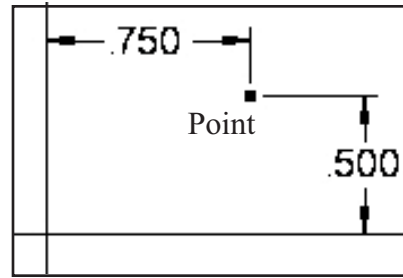
Other options for Line are Point (used as a marker or place holder) and Freesketch.

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

√ Place **smart dimensions** to constrain the shape.
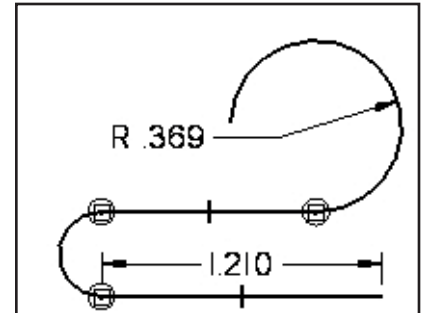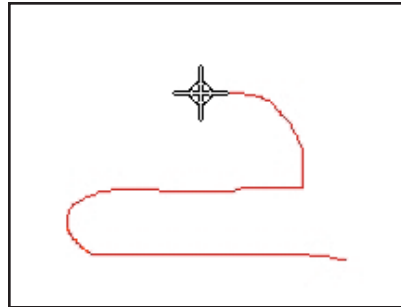
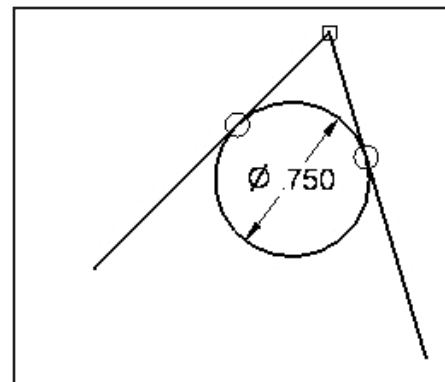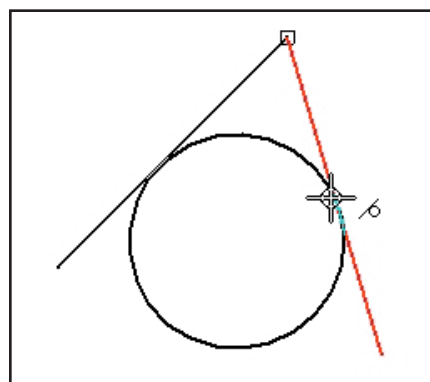**Circle**. Pop down provides options for drawing circles and ellipses.

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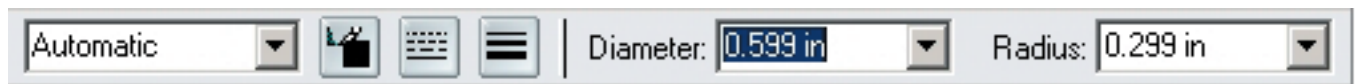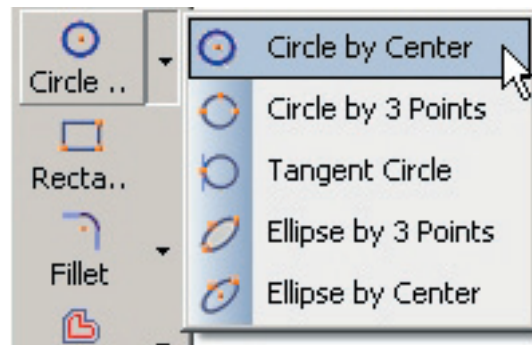
Tangent Circle

A circle was placed at random tangent to two lines.

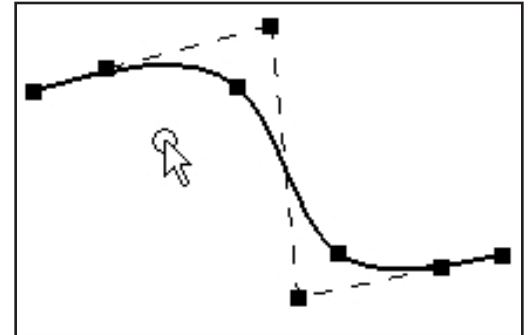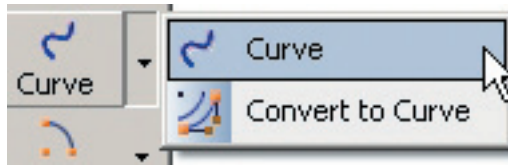The circle size may be set using the circle command or by placing a **smart dimension**.

Arc.

Three options are shown.
The arc ribbon bar gives slightly
different options for each arc type.
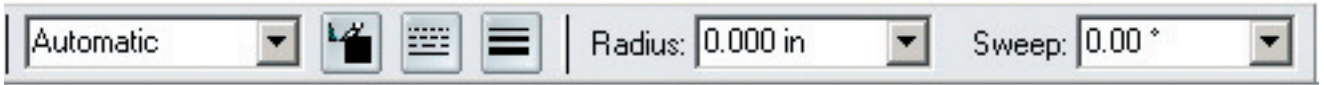
Curve.

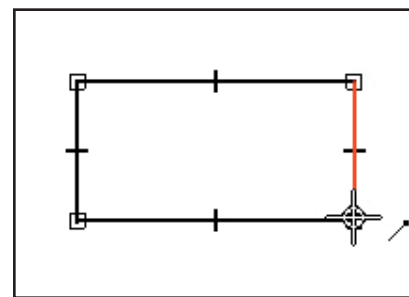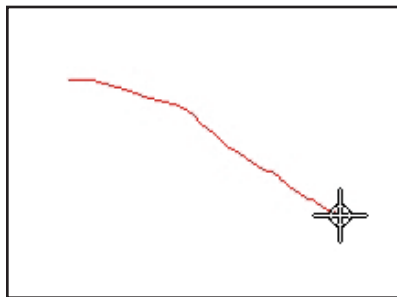Click a series of points.
A smooth curve is
drawn.

Curve has many set-
tings and options.
Check **Help** for more
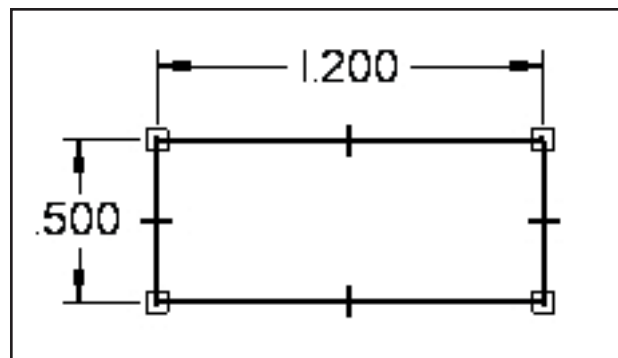information.

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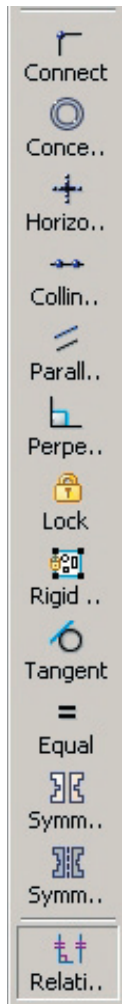
The geometric constraint icons are shown. Choosing and applying constraints is a matter of planning which elements must remain fixed in relation to other elements that are allowed to change.

Turns display of geometric icons Off/On
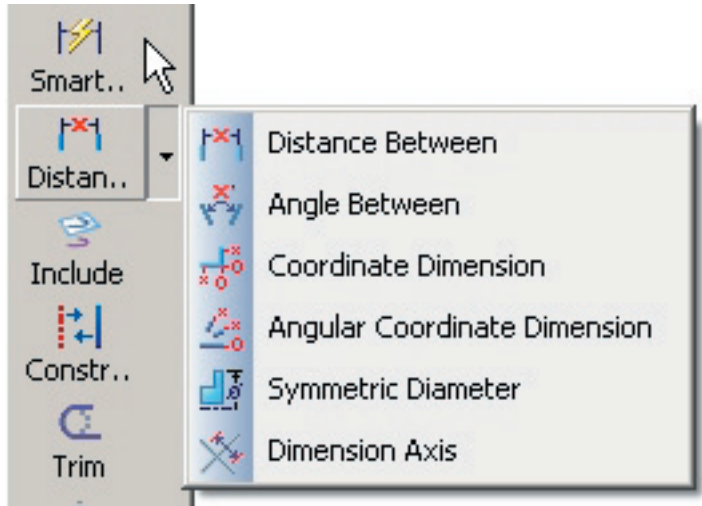
| Relationship	Handle
Collinear	○
Connect (1 degree of freedom)	✗
Connect (2 degrees of freedom)	⊡
Concentric	◎
Equal	=
Horizontal/Vertical	┼
Tangent	○
Tangent (Tangent + Equal Curvature)	⊖
Tangent (Parallel Tangent Vectors)	♀
Tangent (Parallel Tangent Vectors + Equal Curvature)	♀
Symmetric	] [
Parallel	//
Perpendicular	⌐
Fillet	⌒
Chamfer	⟨
Link (local)	⊗
Link (peer-to-peer)	⊗
Link (sketch to sketch)	⊠
Rigid Set (2-D elements)	□

Aligning Geometry

The rectange 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

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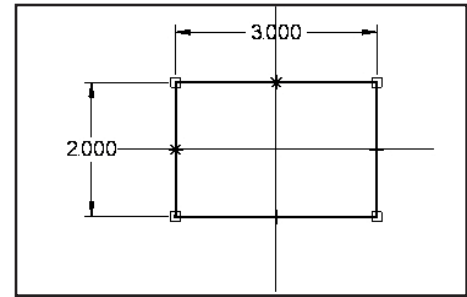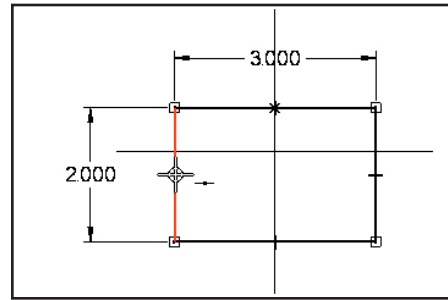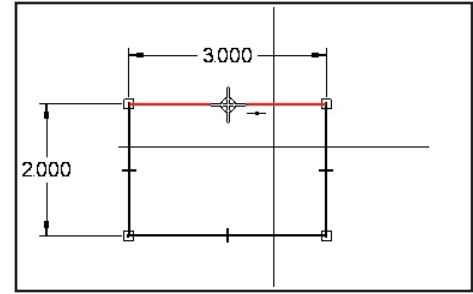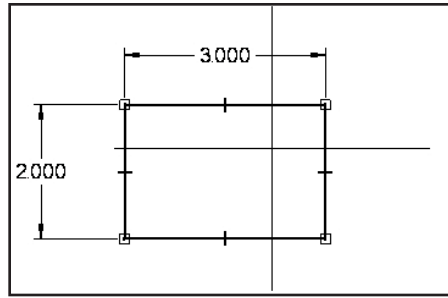
Changing the 2.000 and 3.000 dimensions will keep the rectangle centered.
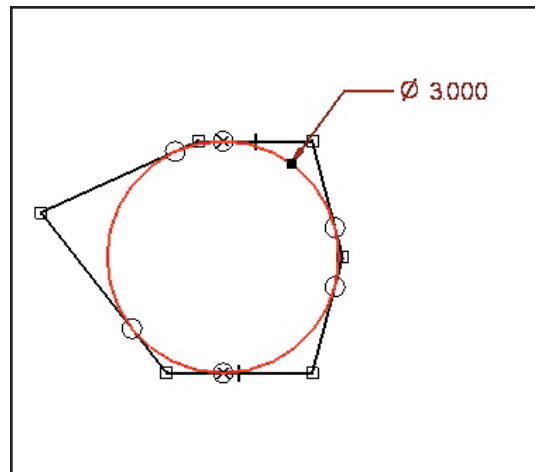

A hexagon has been circumscribed about a circle. We want to be able to change the circle size and keep the hexagon shape.

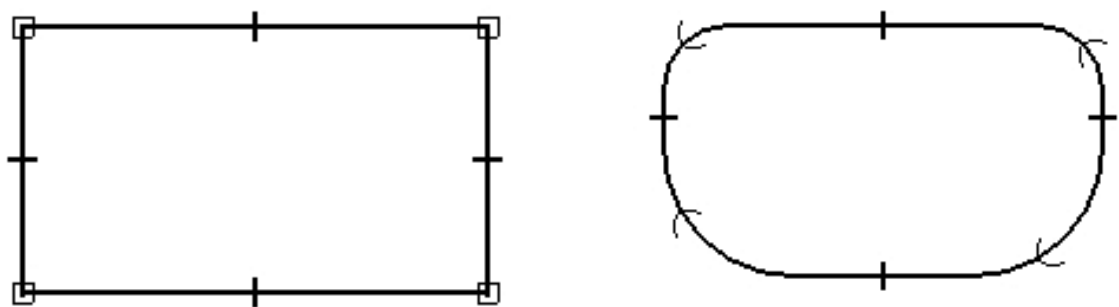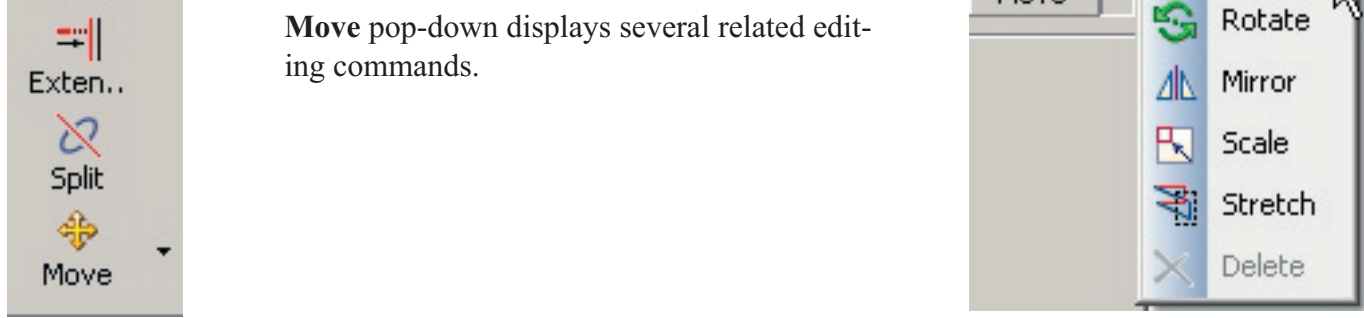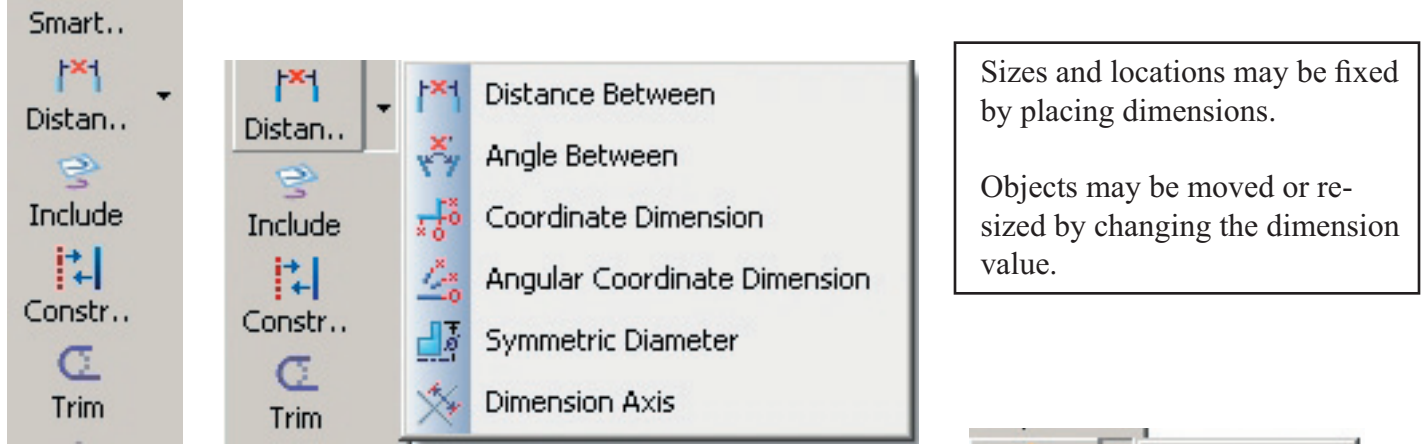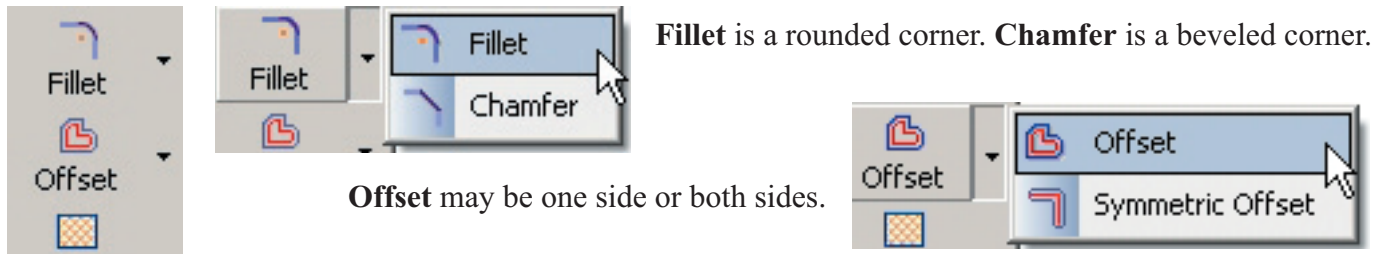Lines are **connect constrained** and **tangent constrained**.
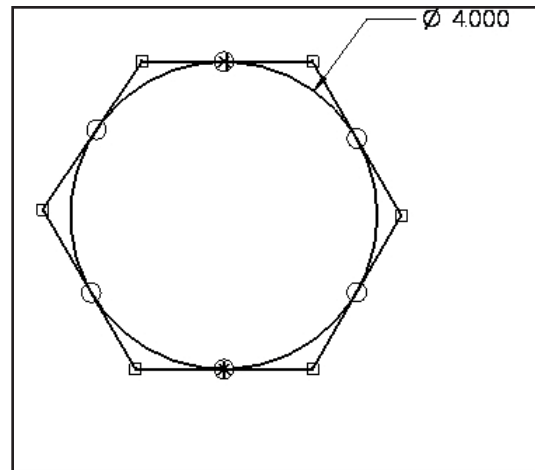
Changing the circle size produces in an unexpected result!

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.

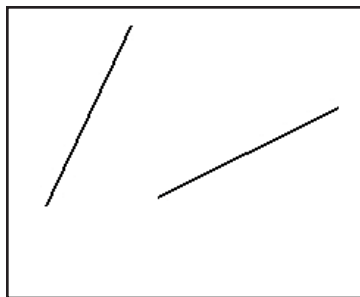**Move** pop-down displays several related editing commands.

The rectangle shape was modified using the fillet editing command.

Trim

Inner circular arcs were
removed using **trim.**
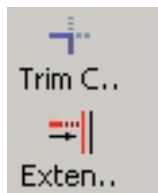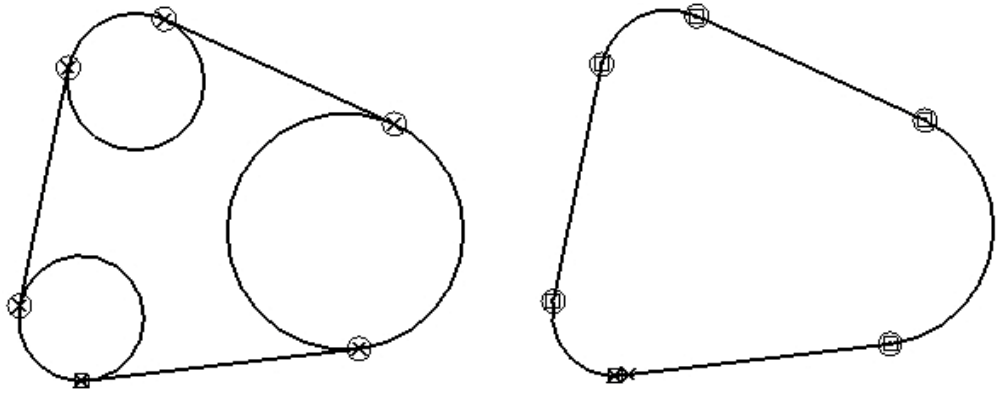
Extend will only work
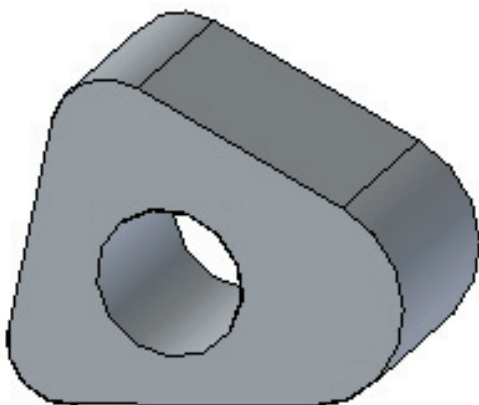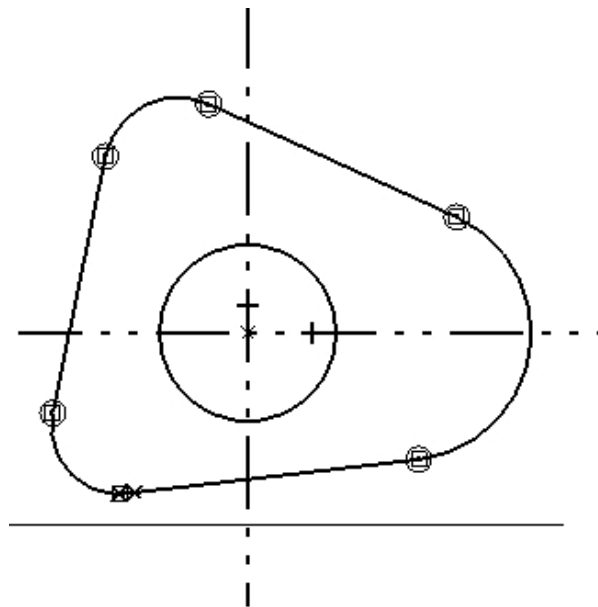if there is a "stopper"
boundry.
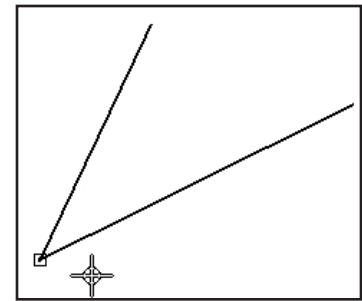
Extend would not work but
Trim would work.

Construction Lines

Regular lines may be toggled to con-
struction 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.
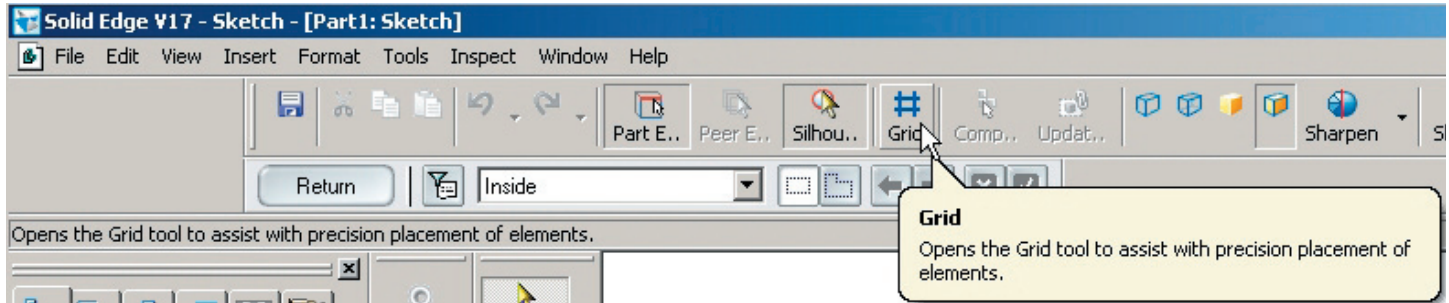
Toggle Grid
On/Off with the
screen icon or use
Tools ...Grid

Grid settings dialog

Grid display On/Off

Grid Ribbon Bar
is shown.

Snap to Grid
On/Off

Set Grid
to Origin

Set Grid 0,0 to new
location
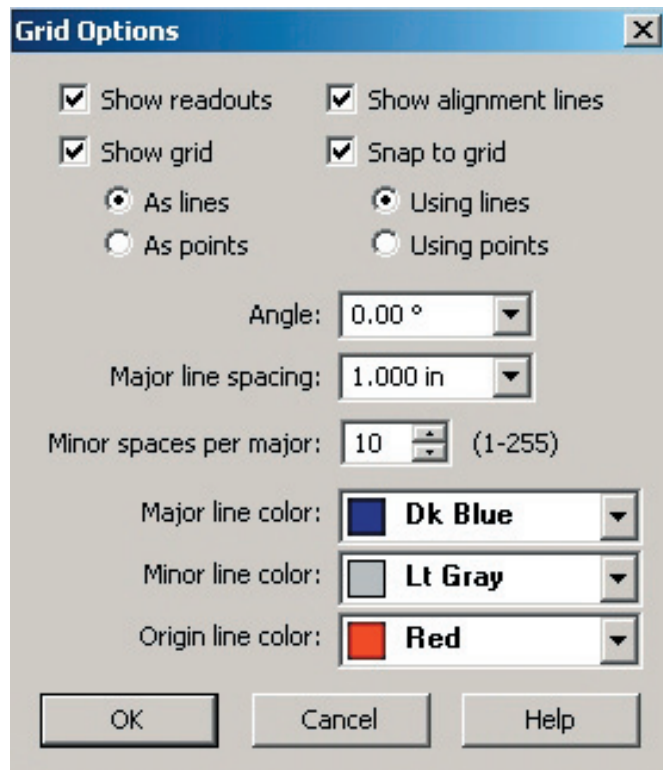

Grid Options Dialog Box

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:
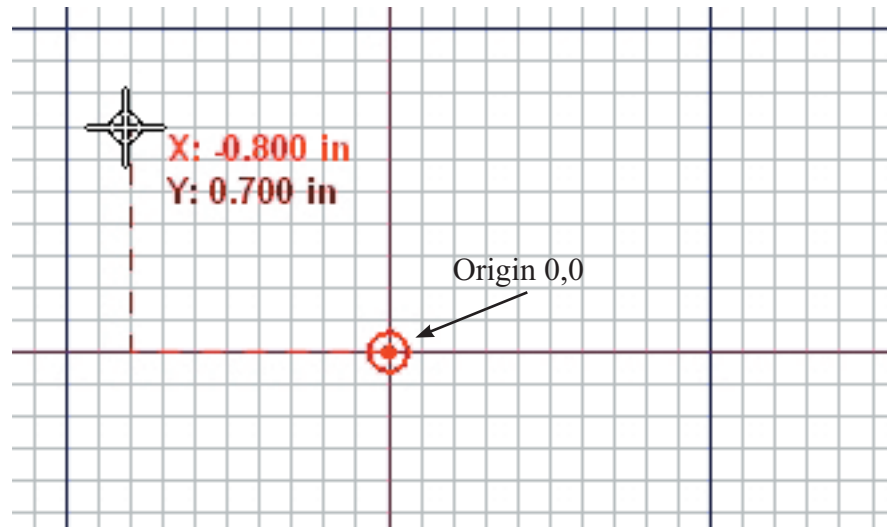Major = 1 foot
Minor = 12 (1 inch)

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.

X: -0.800 in
Y: 0.700 in
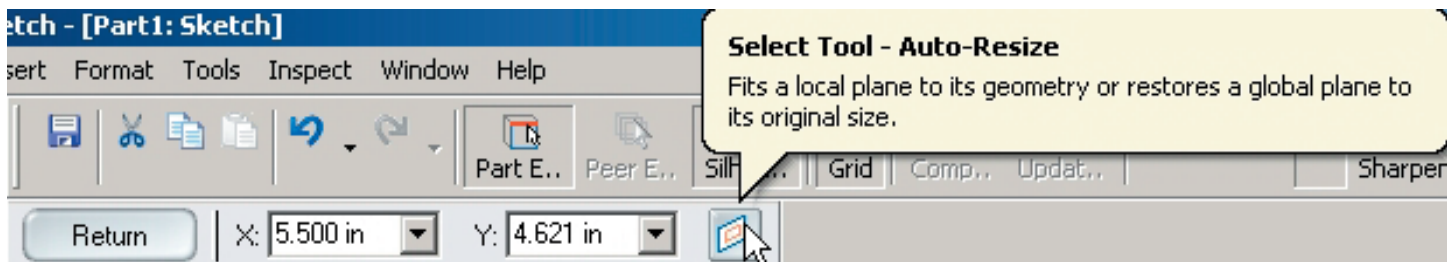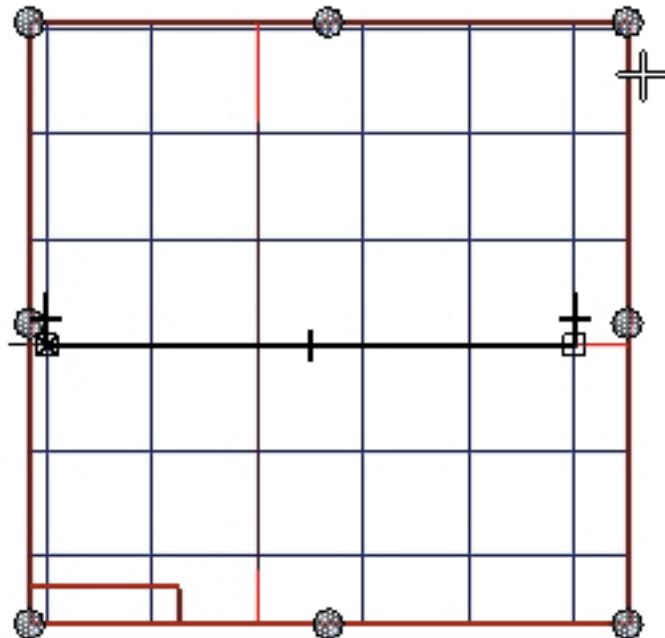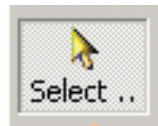
Origin 0,0

Re-Sizing the Grid Area

√ Click the Select Tool

Select ..

√ 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.

etch - [Part1: Sketch]

sert Format Tools Inspect Window Help

Select Tool - Auto-Resize
Fits a local plane to its geometry or restores a global plane to its original size.

Part E.. Peer E.. Silh ..   Grid   Comp.. Updat..   Sharper

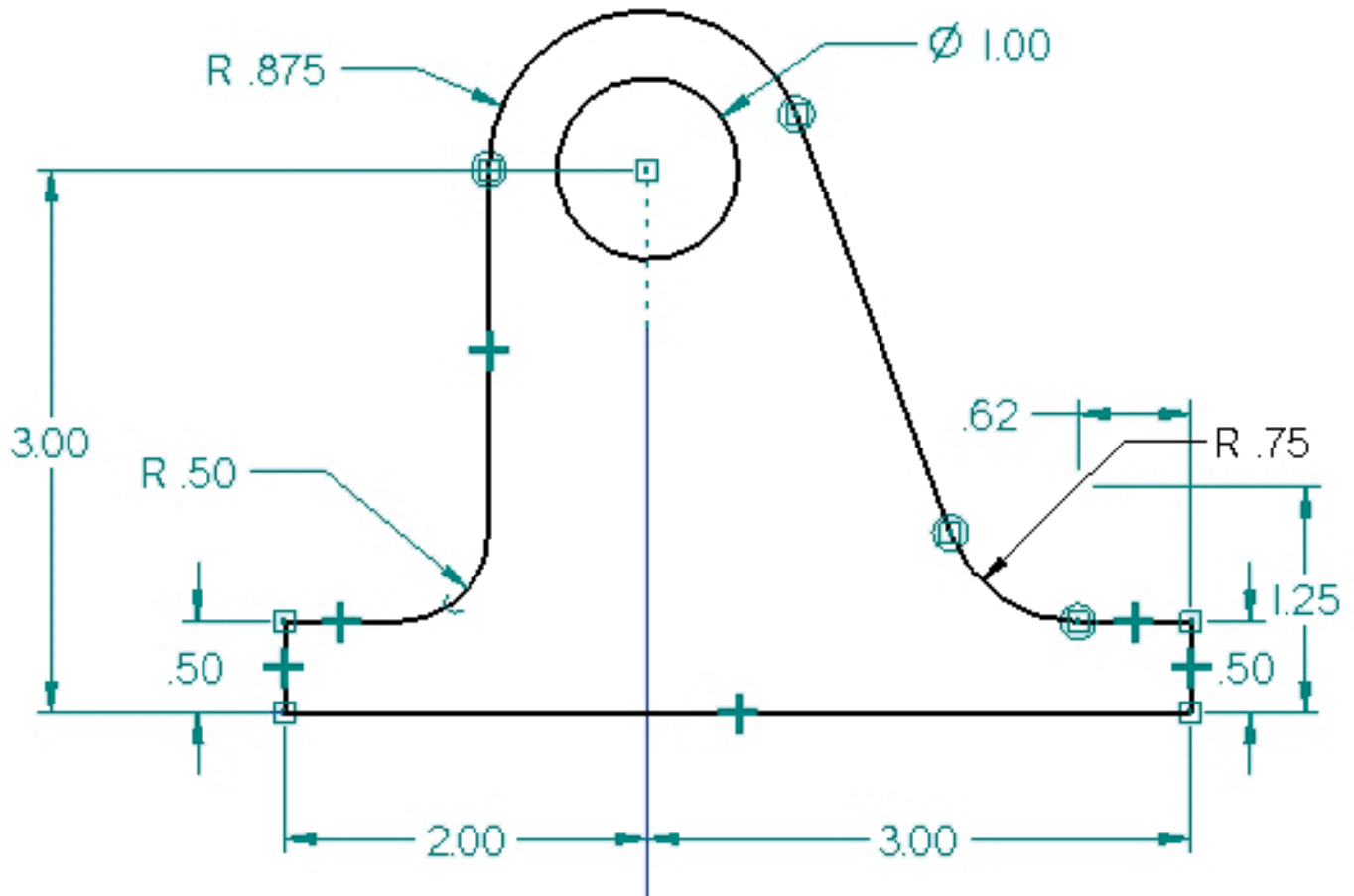Return   X: 5.500 in ▼   Y: 4.621 in ▼

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.
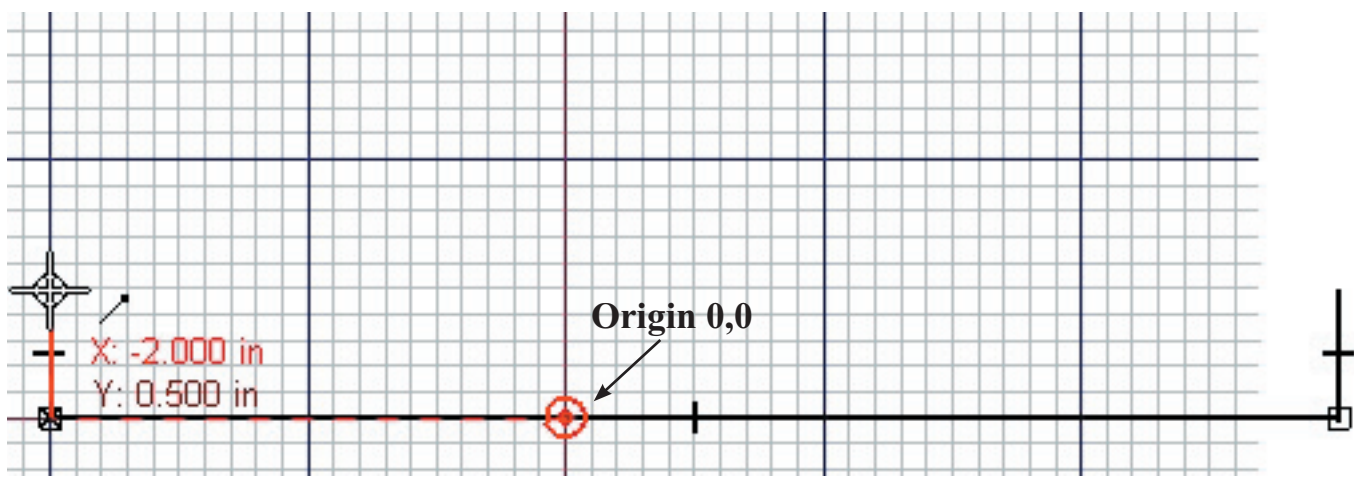Refer back to this drawing for dimensions

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, 0.500 to -2.000, 0.000 to 3.000,0.000 to 3.000, 0.500
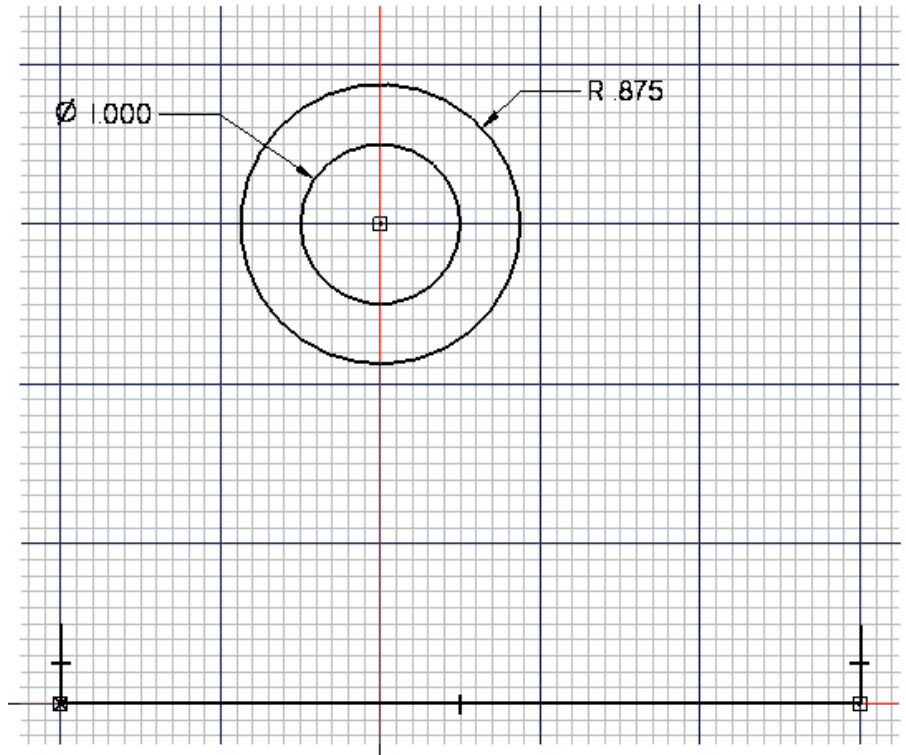Re-size the grid if needed.

3. Draw two circles with center at

X = 0.000, Y = 3.000

The small circle is 1.000 diameter.
(Ø 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.
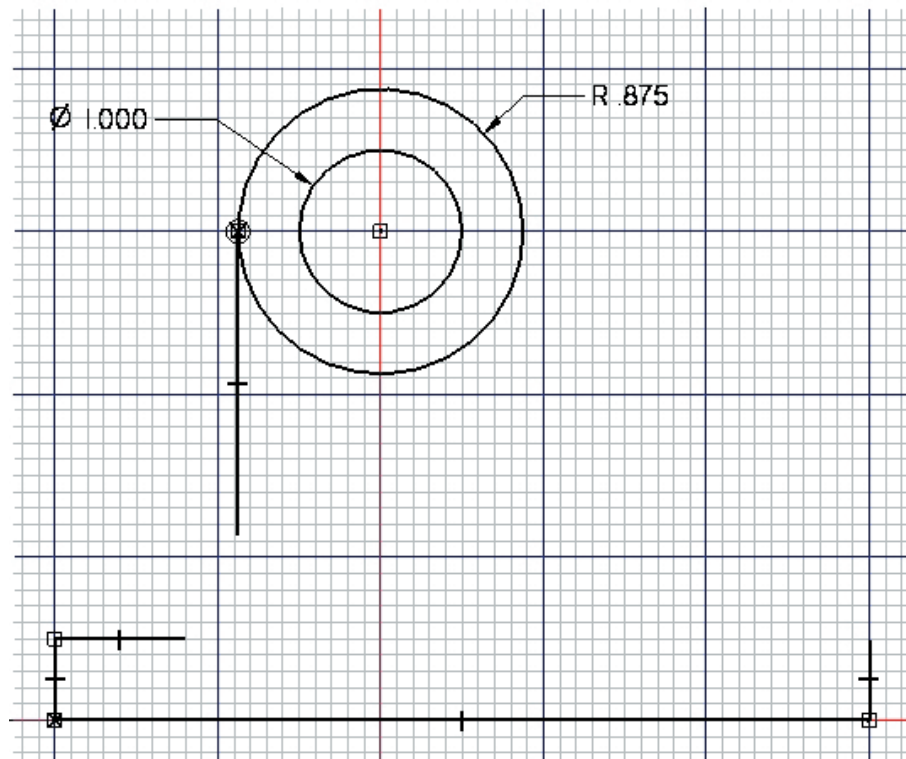
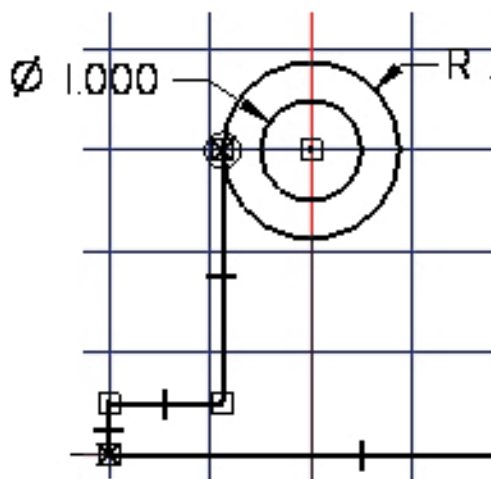Diameter: 1.750 in   ▼   Radius: 0.875 in I ▼

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.

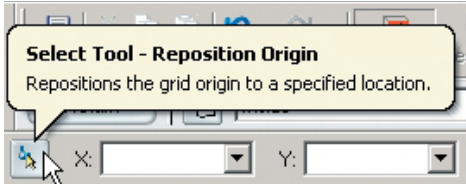5. Draw a short line across from
-2.000, 0.500

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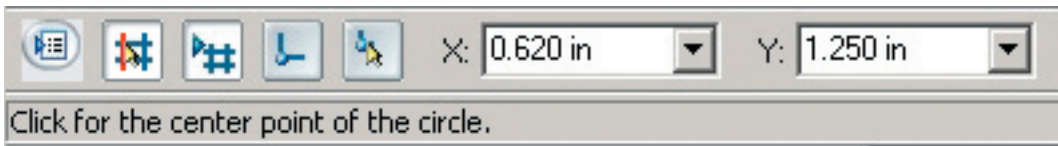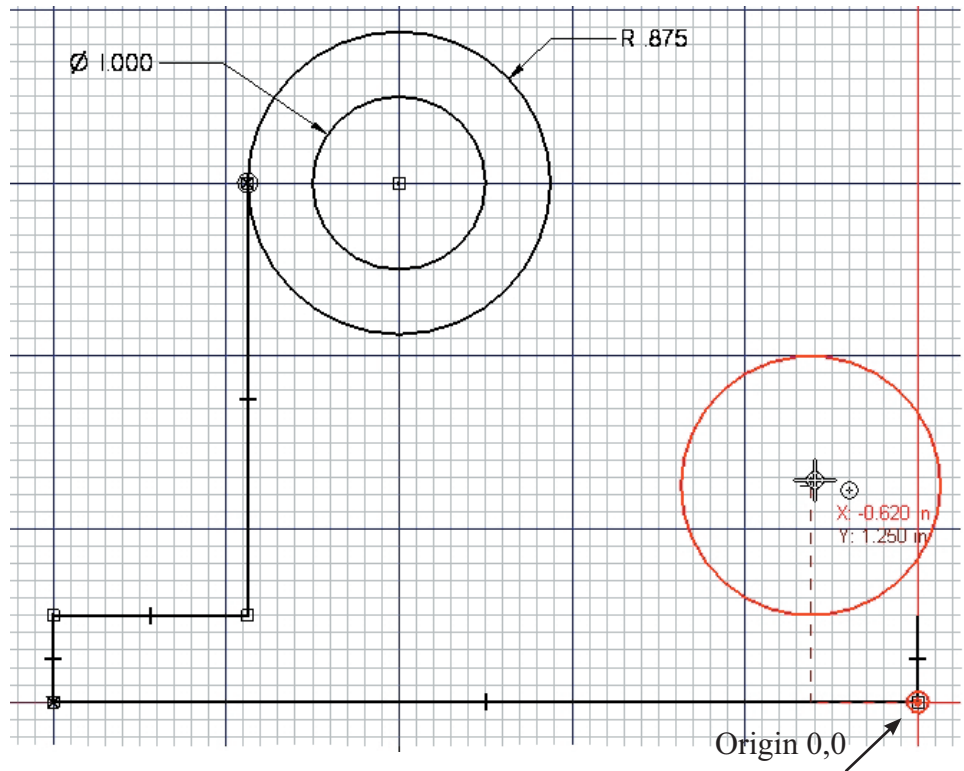the corner.

Trim C..

Ø 1.000   R .

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**.

Select Tool - Reposition Origin
Repositions the grid origin to a specified location.

X: [▼]   Y: [▼]

Click the end if 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.

Ø 1.000

R .875

X: -0.620 in
Y: 1.250 in

Origin 0,0

X: [0.620 in   ▼]   Y: [1.250 in   ▼]
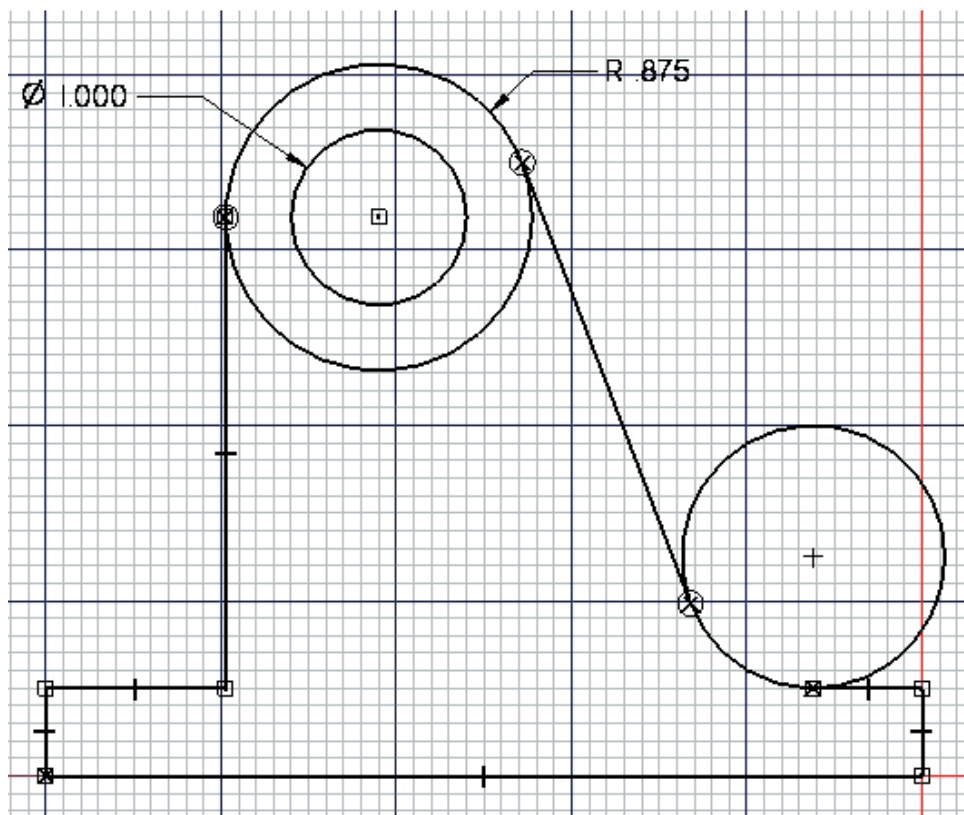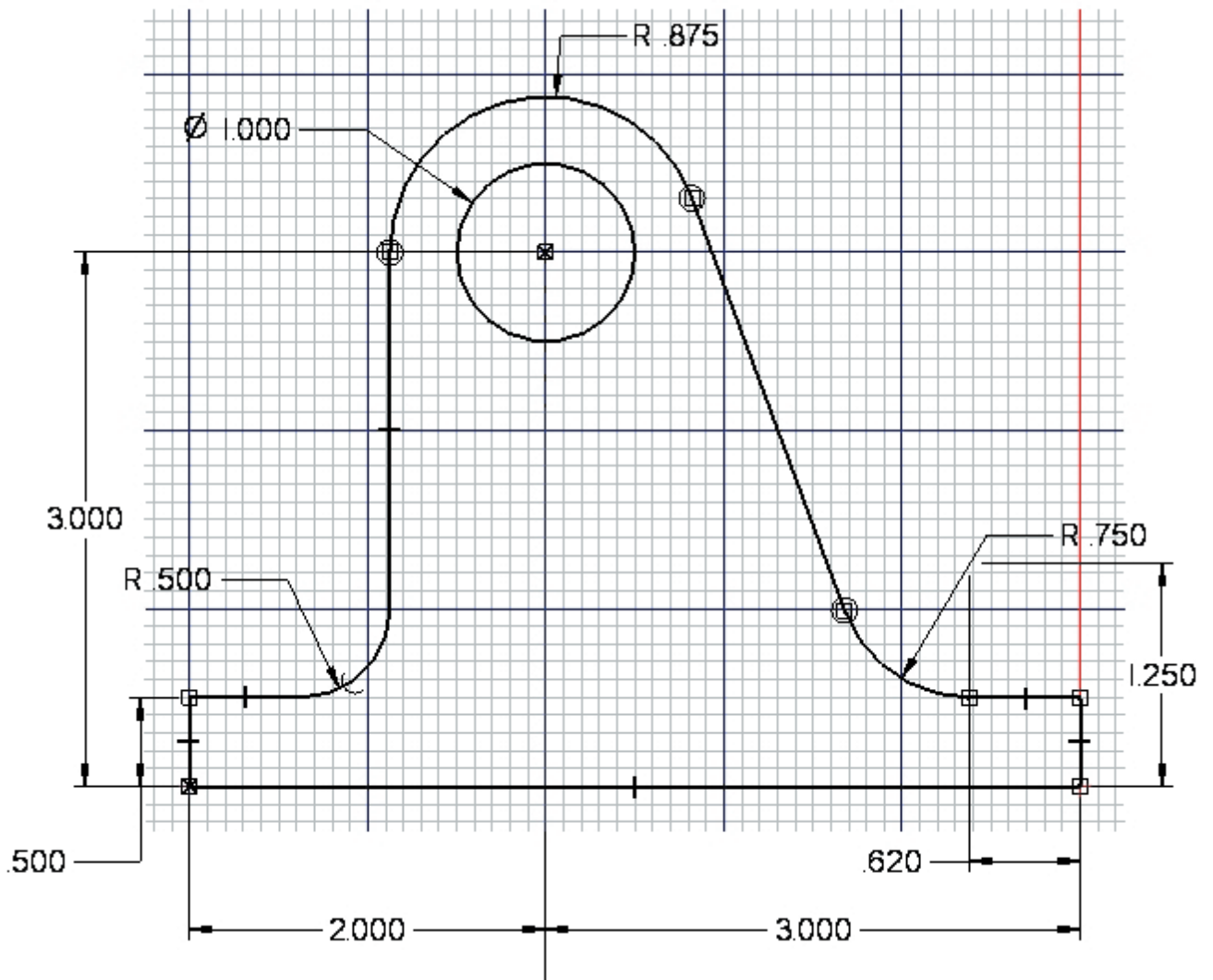
Click for the center point of the circle.

9. Draw line tanget 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**

Ø 1.000

R .875

R .875

Ø 1.000

R .750

3.000

R .500

1.250

.500

.620

2.000

3.000

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

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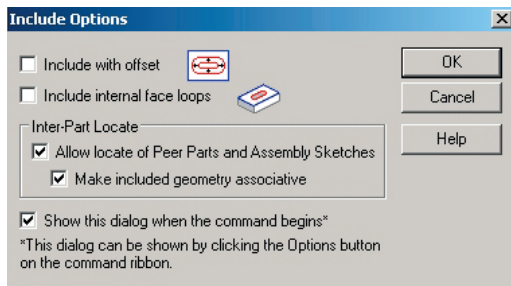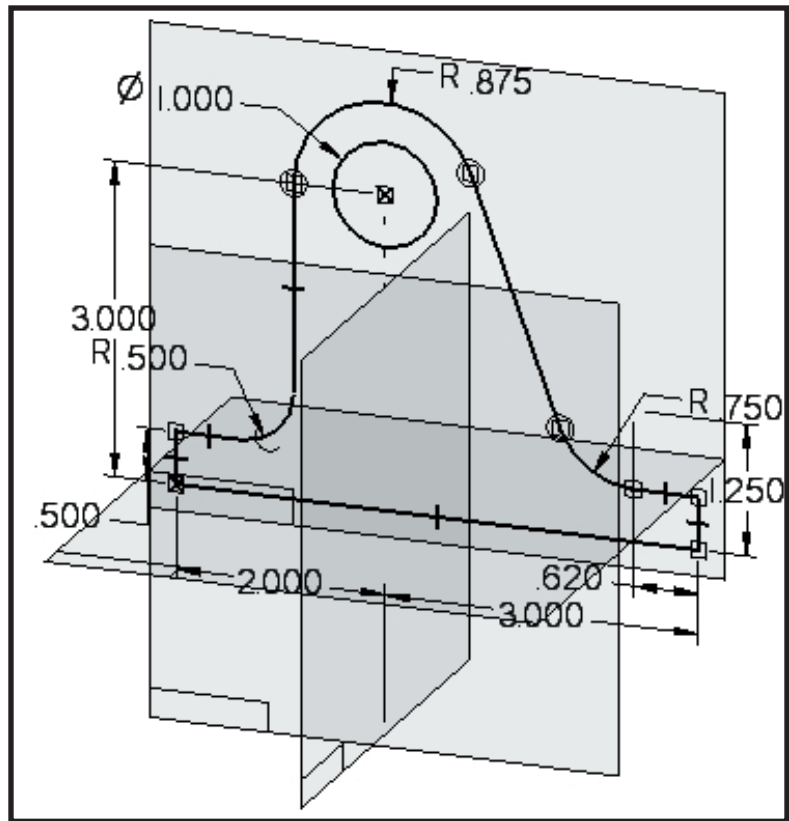
Return
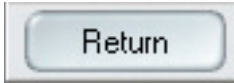
The sketch can be seen on the Front
(X-Z) plane.

14. Click **Protrusion**

Protru..

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

Include

16. Click **Include**
Just O.K the dialog box.

Include Options

☐ Include with offset
☐ Include internal face loops
┌ Inter-Part Locate ─────
☑ Allow locate of Peer Parts and Assembly Sketches
   ☑ Make included geometry associative

☑ Show this dialog when the command begins*

*This dialog can be shown by clicking the Options button
on the command ribbon.

OK
Cancel
Help

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.
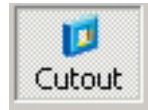Click each segment only one time.*

The final outline should be a closed
loop.
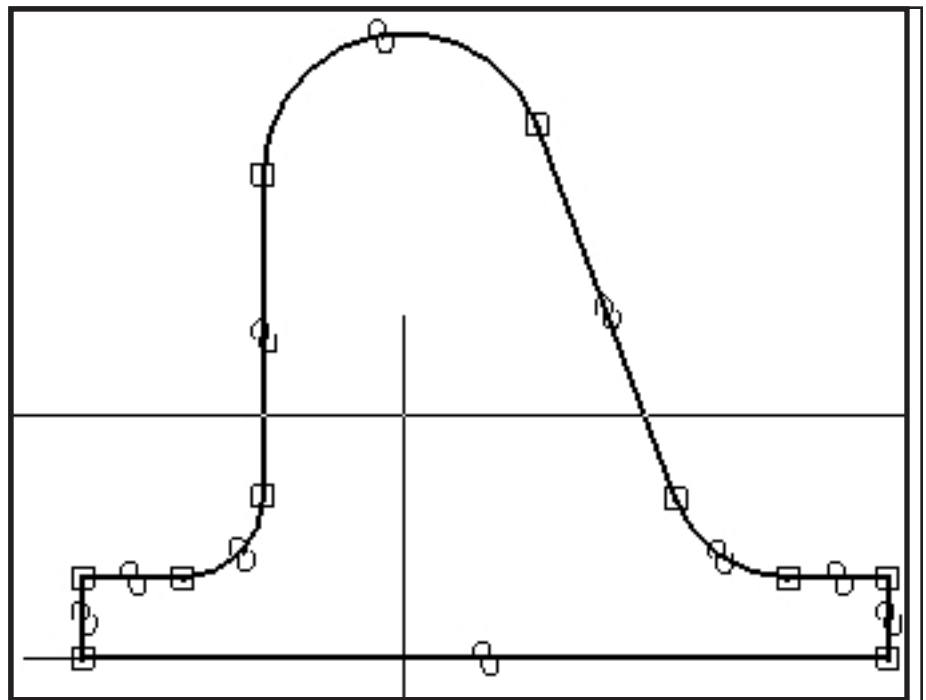
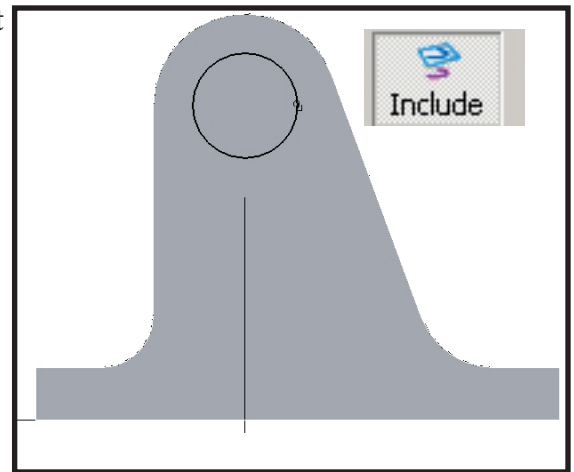18. Click **RETURN** and set the depth of the part to 1.000
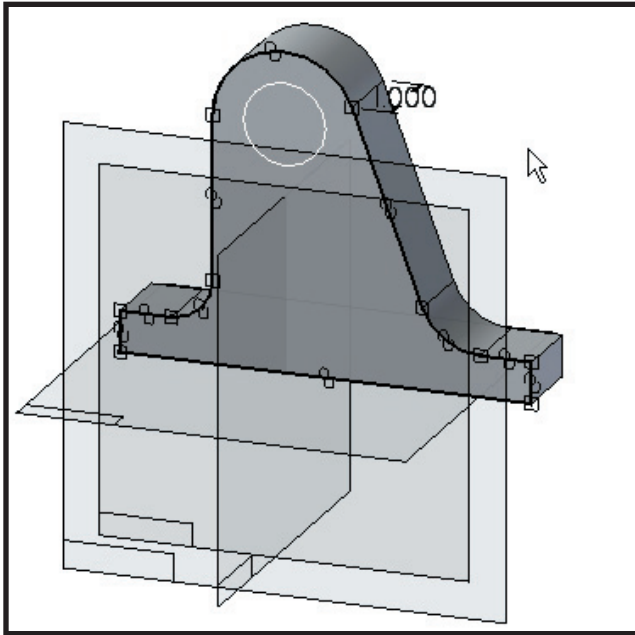
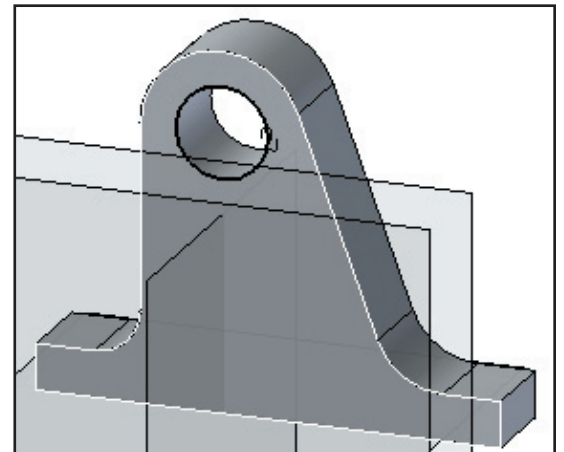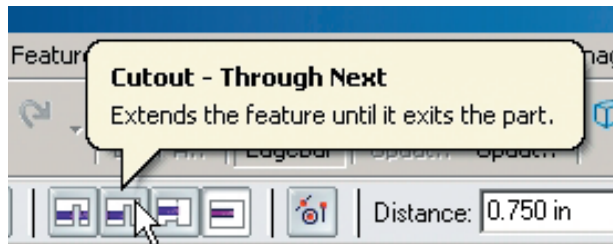19. Click **Cutout**

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.
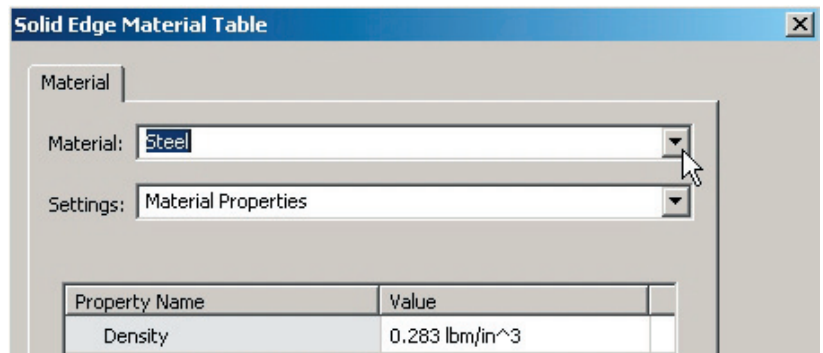

Part Properties.

Collecting data for Bill of Material lists is an important documentation step. It is best to input the data as the part is being modeled.

__ Pop down **Tools ...Material Table**

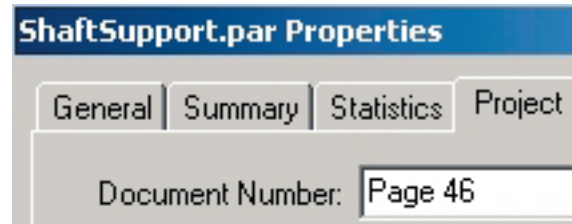__ Scroll down and select **Steel**

Save the model.

__ Pop down
File .. File properties

__ Type in the information to
the two tabs as shown

The material, part name and
document number will appear
in the **Bill of Material** when
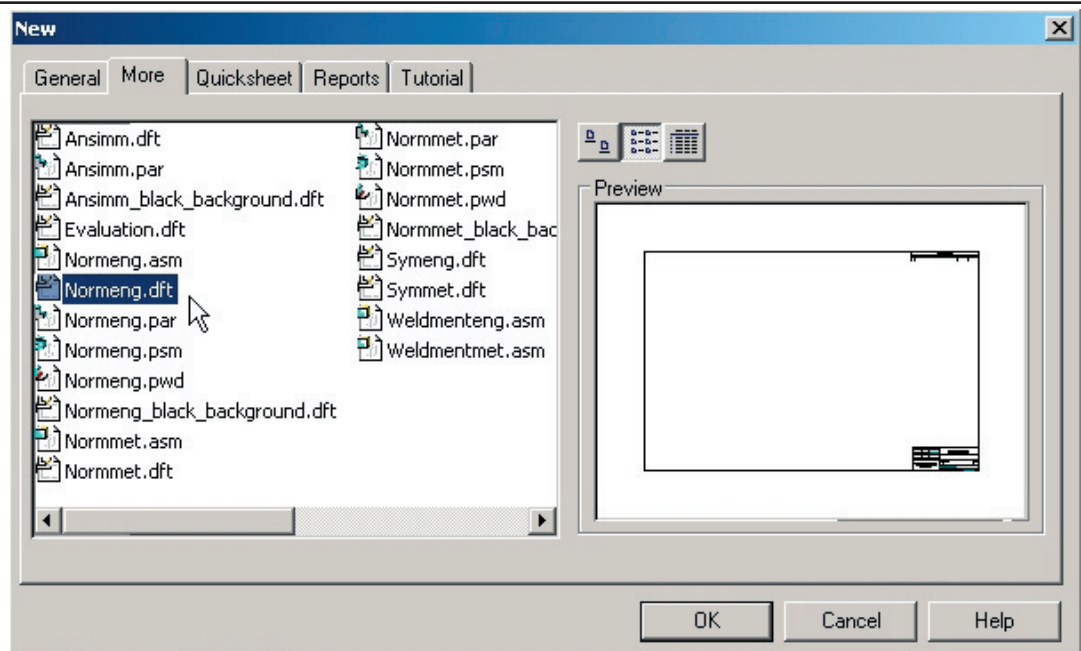inserted into the drawing.

Save the model.

Making a drawing.
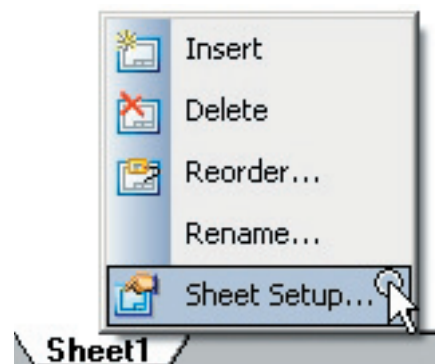
Be sure the part is saved
from the previous steps.

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

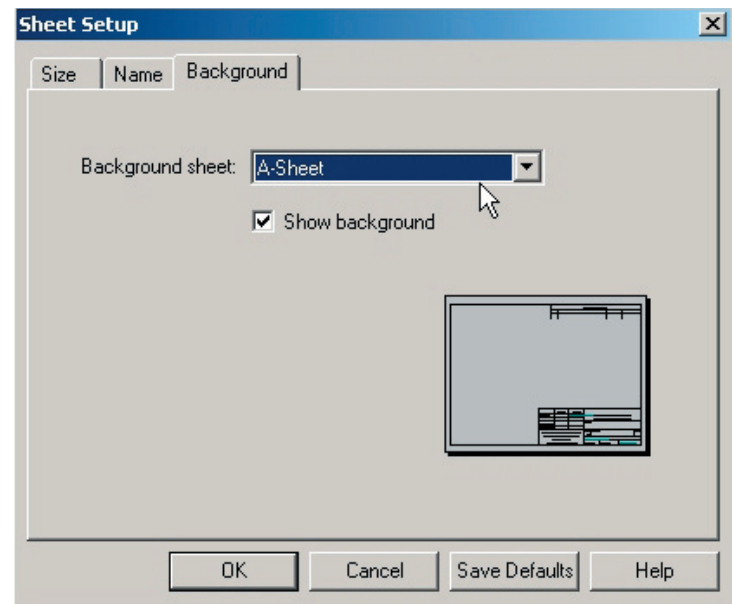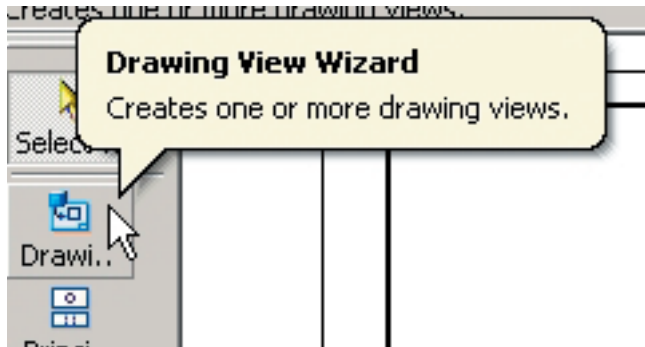Click the **More** Tab
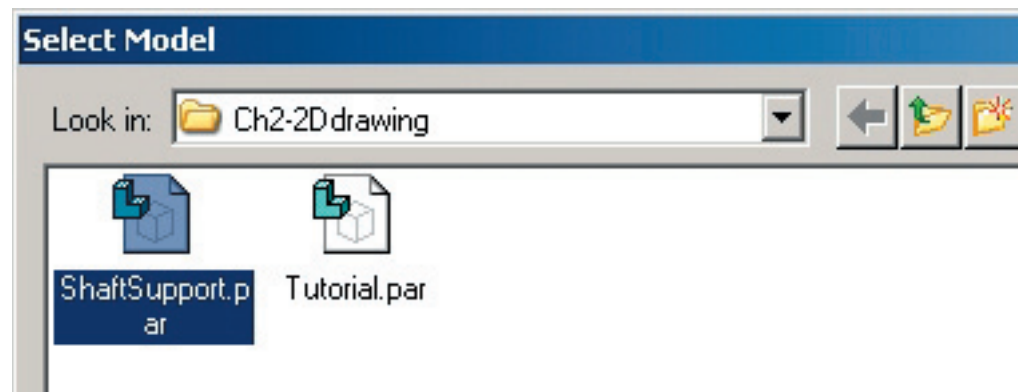and select
Normeng.dft

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)

Sheet Setup

Size | Name | Background

Background sheet: A-Sheet

☑ Show background

OK   Cancel   Save Defaults   Help

Drawing View Wizard
Creates one or more drawing views.

Select

Drawi..

24. Click the
Drawing View Wizard.
Select the
ShaftSupport.par model from
the folder where you saved it.

Select Model

Look in: Ch2-2D drawing

ShaftSupport.p   Tutorial.par
ar

25. Select
Front
for the key view.

Drawing View Creation Wizard

Drawing View Orientation

Named Views:

dimetric
front
iso
right
top
trimetric

Custom...
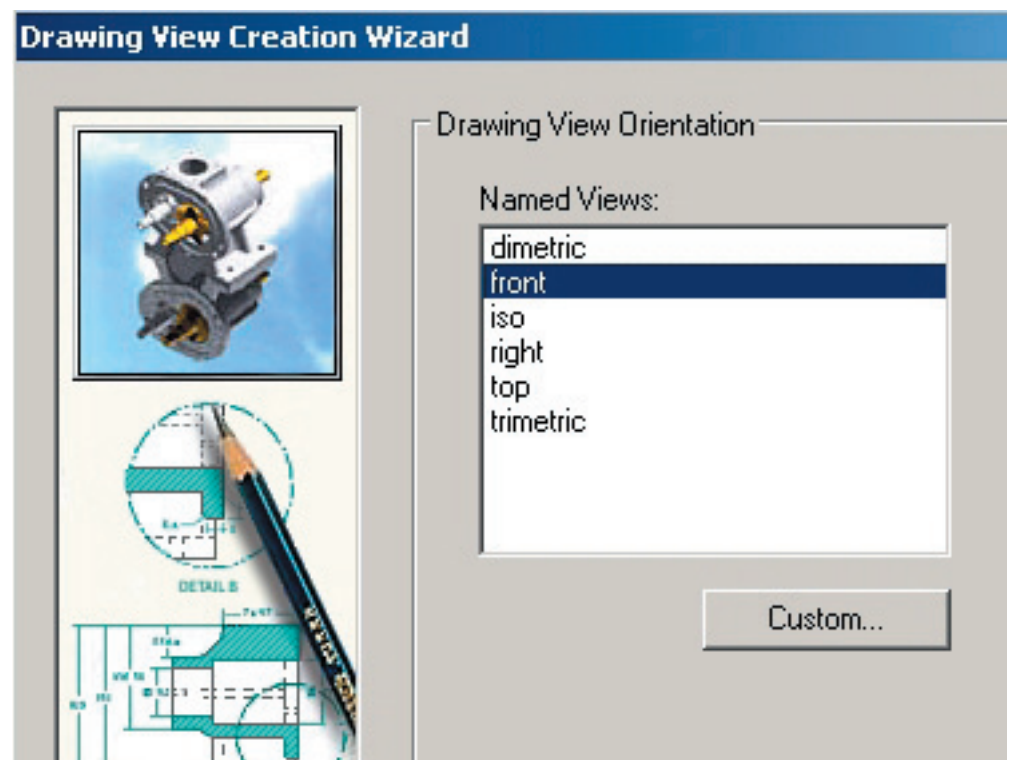
26. The **Front** view is selected by default.
Click **TOP , RIGHT** and **ISOMETRIC** as shown.
Click **Finish.**
The drawing will fit at a scale of 1:2 . Select that option before placing the views.

Click a view to move it on the sheet.
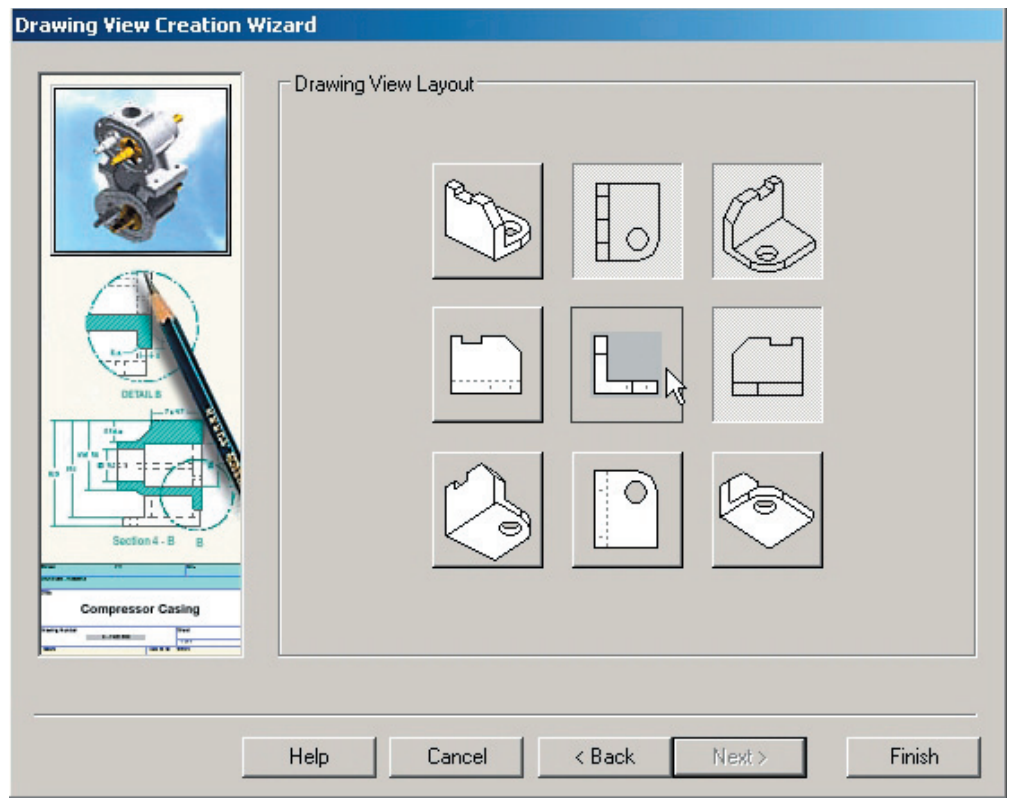__ Place dimensions as shown.

__ Click the **Parts List** icon
__ Click the front view.
Place the Parts list on the drawing.

Save and **Print** the drawing.
End of tutorial

Drawing View Creation Wizard

Drawing View Layout

Compressor Casing

| Help | Cancel | < Back | Next > | Finish |

REVISION HISTORY

REV	DESCRIPTION	DATE	APPROVED

R .875
Ø 1.000
R .750
3.000
R .500
2.000
.500
1.000
5.000

Detail ..
Updat..
Parts ..

Item Number	Document Number	Title	Material	Quantity
I	Page 46	Shaft Support		I

	NAME	DATE	**SOLID EDGE**		
DRAWN	jcraig	06/28/05	UGS - The PLM Company		
CHECKED					
ENG APPR			TITLE		
MGR APPR					

UNLESS OTHERWISE SPECFED
DIMENSIONS ARE IN INCHES
ANGLES ±XX°
2 PL ±XXX 3 PL ±XXXX

SIZE A	DWG NO		REV
	FILE NAME: ShaftSupport.dft		
SCALE:	WEIGHT:	SHEET I OF I	

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.

__Set Part Properties, material, etc.   Dimension the drawing.
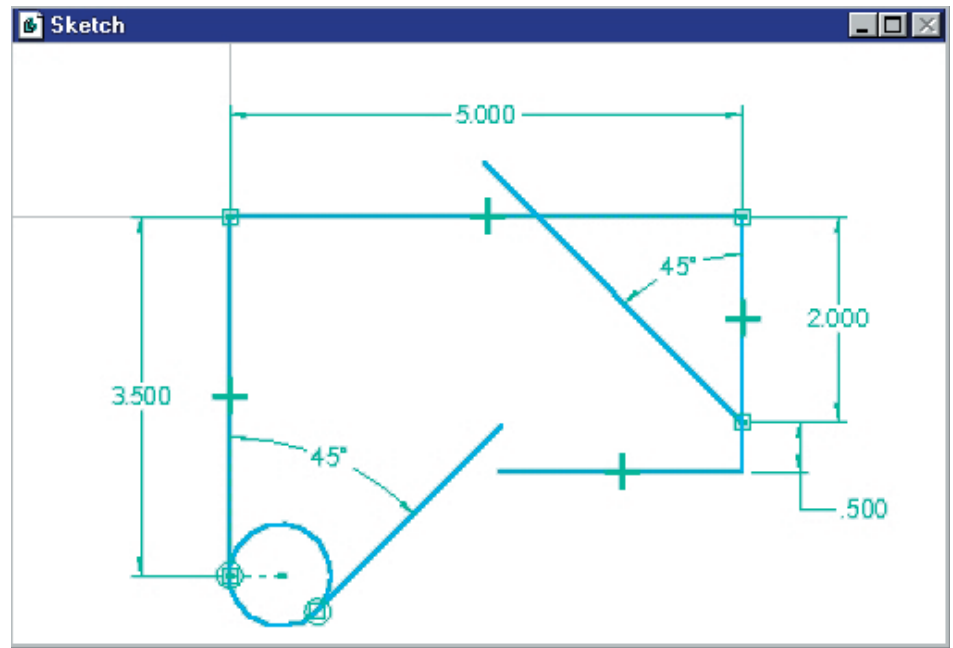
__ **Save** and **Print** the drawing.

Thickness = 1.000
Material = 2024
Aluminum

R .750

Ø .750

.250

Ø 1.000

3.250

R 1.000

R 1.000

4.000

COMPANY OR SCHOOL NAME

PART NAME : BAFFLE PLATE

DR BY: CRAIG, J
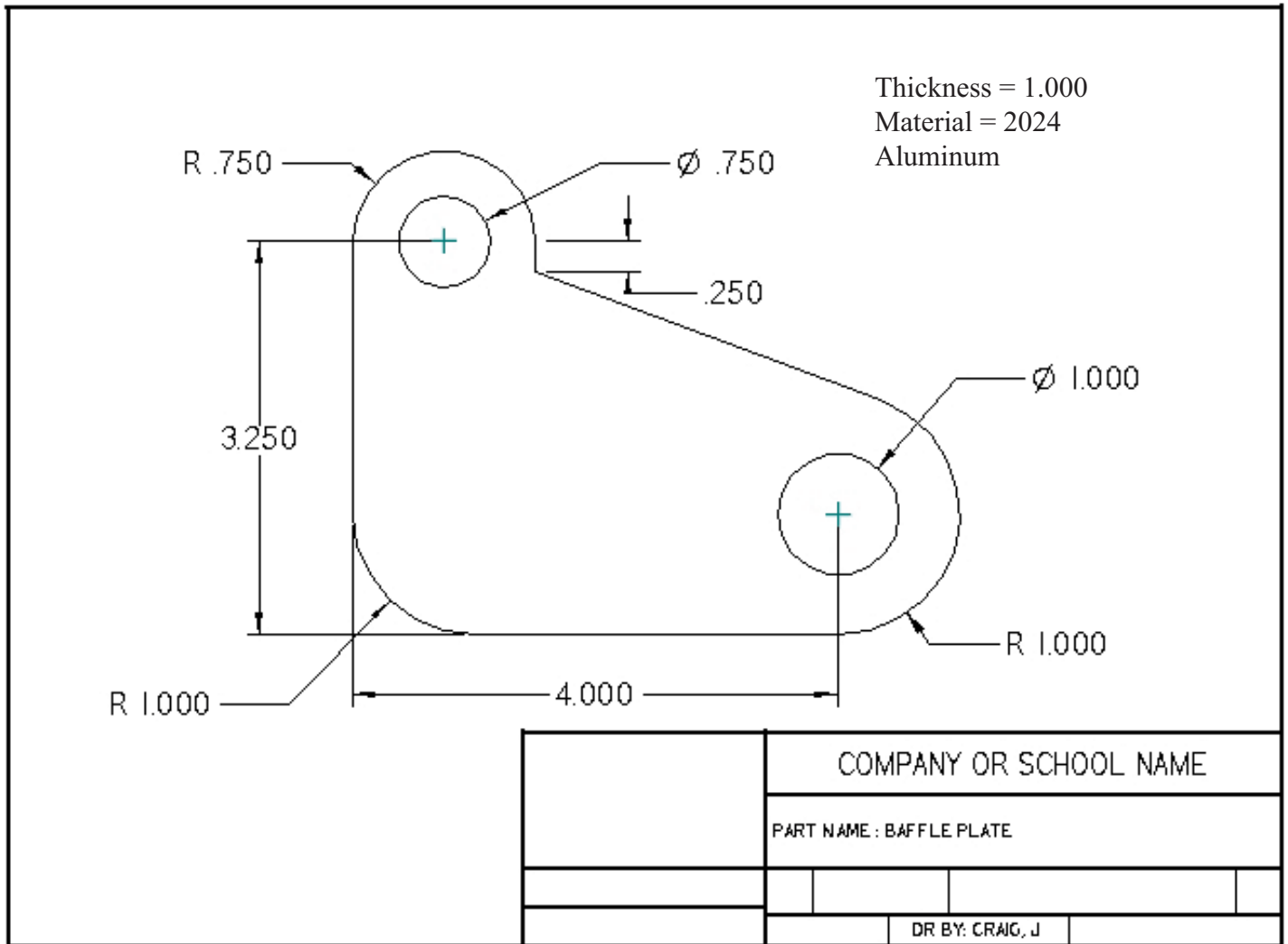
Problem 2.

Create the sketch for the part shown below.

Follow the steps as shown in Tutorial 1.

Create a drawing with dimensions and parts list.

dimension save and print.

Thickness = .125
Material = Brass

SCHOOL OR COMPANY NAME

PART: SCRAPER PLATE

DR BY: LAST, F
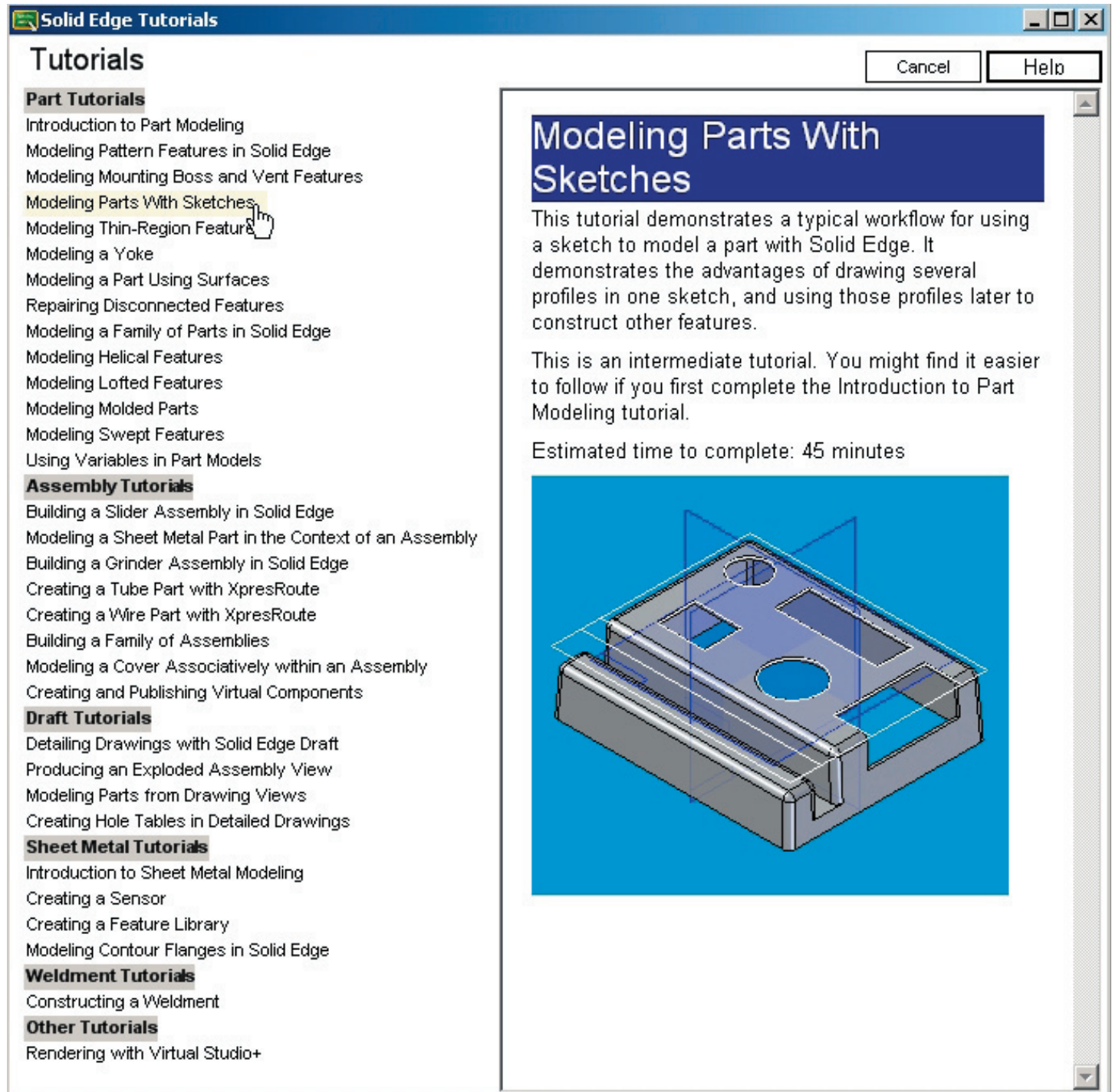
Problem 3.

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

Part = Template

Thickness = .25
Material = Steel
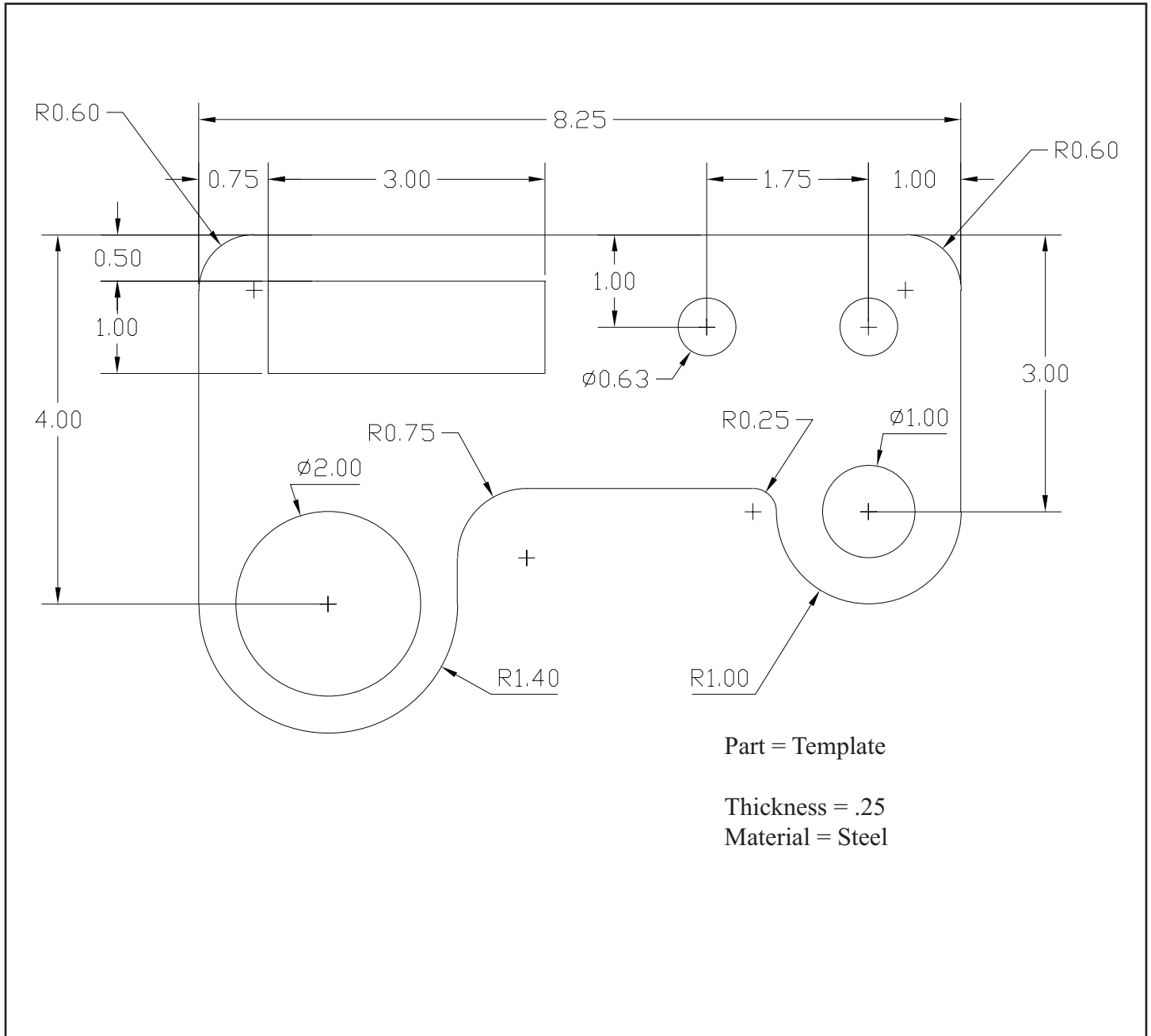
Problem 4.

Do the "Modeling Parts with Sketches" tutorial. Make a drawing as shown in Tutorial 1.
Save and Print the model and the drawing. (No dimensions).