

# Autodesk Inventor

## Module 10

### 2D Sketching Planes

#### Learning Outcomes:

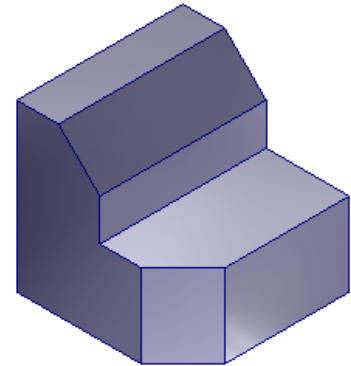
When you have completed this module, you will be able to:

1. Describe the three predefined 2D sketching planes and the view of the model that they are on.
2. Construct solid models by drawing the base sketch on either the front or right side instead of the default top view.
3. Describe a consumed and an unconsumed sketch.

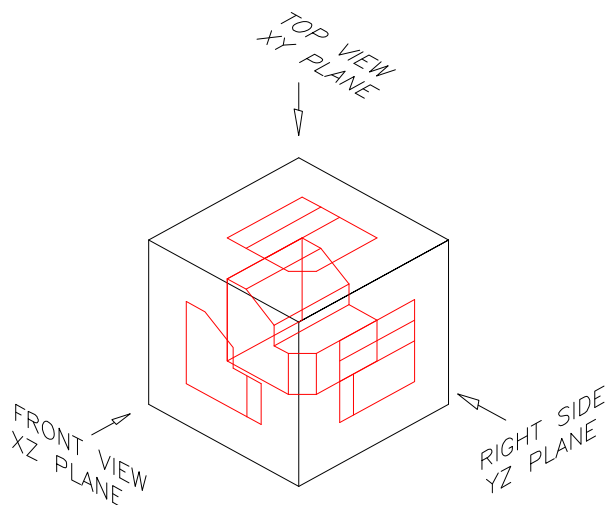
#### 2D Sketching Planes

Up to this point in the modules, the base sketch has been drawn on the XY plane. The XY plane is the top view of the model and the default plane as configured in the templates that are being used to complete the workalongs and lab exercises in the Inventor Modules. The models that have been constructed up to this point in the modules were all designed so that the base sketch was drawn on the XY plane or the top view. In this module, learning how to construct solid models by drawing the base sketch on either the front or right side planes will be taught.

Inventor has three predefined planes that can be used to draw the base sketch. They are the XY, XZ, and YZ planes. The XY plane is the top view, the XZ plane is the front view and the YZ is the right side view.



**Figure 10-1**  
The 3D Model



**Figure 10-2**  
The Glass Box

Keep in mind the rule that was taught in Module 4. "It is best to draw the base sketch on the plane that has the most complex contour. Contours with arcs and curves should be avoided".

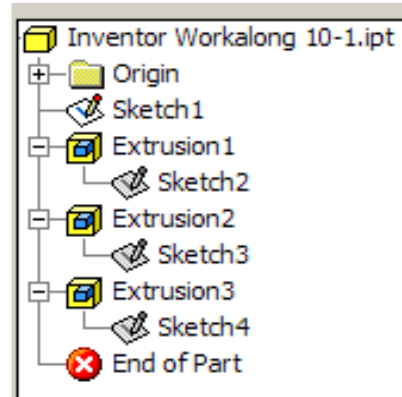
#### The Three Predefined Planes

To help visualize the *three predefined planes* used in Inventor, the 3D model shown in Figure 10-1 is used in this module. The glass box principle that was taught in Module 8 is used to help visualize the three planes. See Figure 10-2.

## Consumed and Unconsumed Sketches

A *consumed sketch* is a 2D sketch that has been extruded or revolved to create a 3D solid model. An *unconsumed sketch* is a 2D sketch that is blank or one that has not been extruded or revolved.

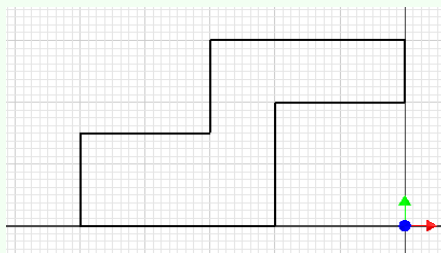
The Browser Bar will display which sketches have been consumed and which ones are unconsumed. See Figure 10-3. Sketch1 is unconsumed while Sketch2, Sketch3 and Sketch4 have been extruded and are consumed.



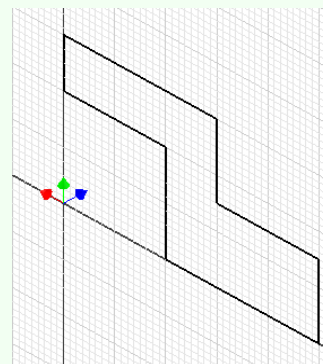
**Figure 10-3**  
Consumed and Unconsumed Sketches

### USER TIP

A 2D sketch can be drawn in the orthographic view or the home/isometric view. In fact, any orbited or rotated view can be used. When drawing on a orbited or rotated view, it is always best to draw in the home or isometric view since this helps the operator maintain a good mental picture of the model.



Orthographic View



Home or Isometric View

### MUST KNOW

Inventor has three predefined planes that can be used to draw the base sketch on. They are the XY, XZ and YZ planes. The XY plane is the top view, the XZ plane is the front view and the YZ is the right side view.

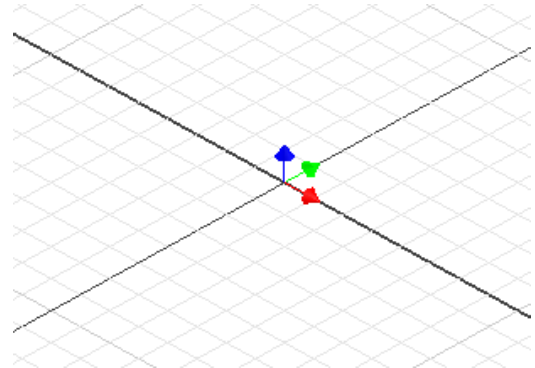
**WORK  
ALONG**

**Working with 2D Sketching Planes**

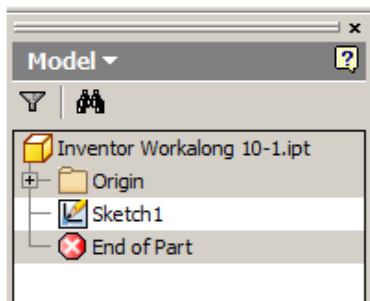
**Step 1** Start a new part file using the template English - Modules Part (in).ipt.

**Step 2** In 2D sketch mode, press F6 to change to the home/isometric view. The graphic window and the Browser Bar will appear as shown in the figures. (Figure Step 2A and 2B)

**Author's Comments:** This is the top view or the XY plane since that is Inventor's default plane. Note that Sketch1 is always on the default plane.

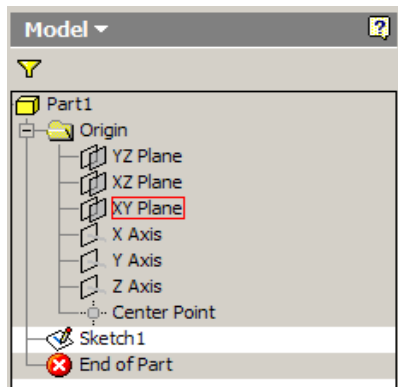


**Figure Step 2A**

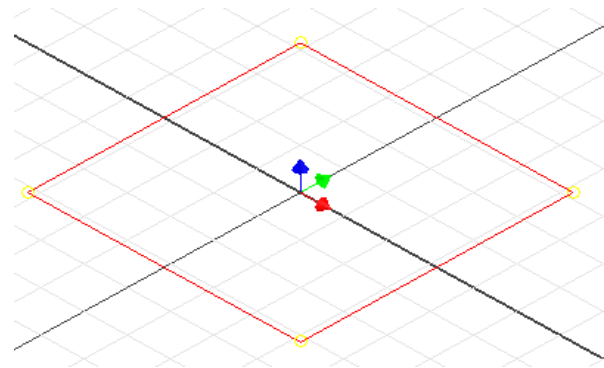


**Figure Step 2B**

**Step 3** In the Browser Bar, expand the Origin folder as shown in the figure. Place the cursor on the XY Plane. Note the orientation of the plane on the sketch. (Figure Step 3A and 3B)



**Figure Step 3A**

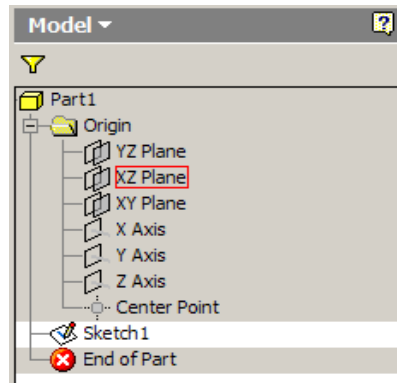


**Figure Step 3B**

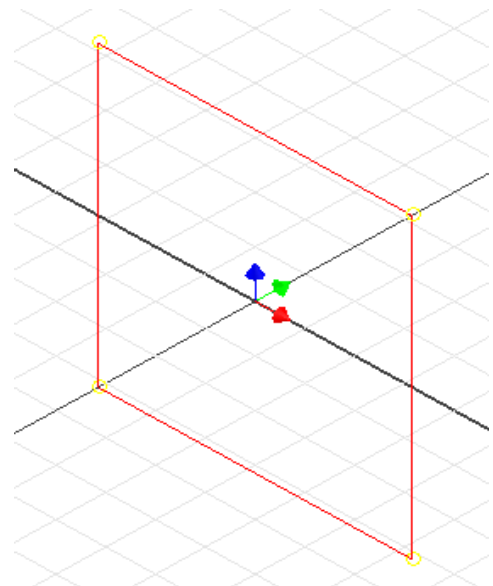
**Author's Comments:** The XY Plane is the top view.

**Step 4** Place the cursor on the XZ Plane in the Browser Bar. Note the orientation of the plane on the sketch. (Figure Step 4A and 4B)

**Author's Comments:** The XZ Plane is the front view.

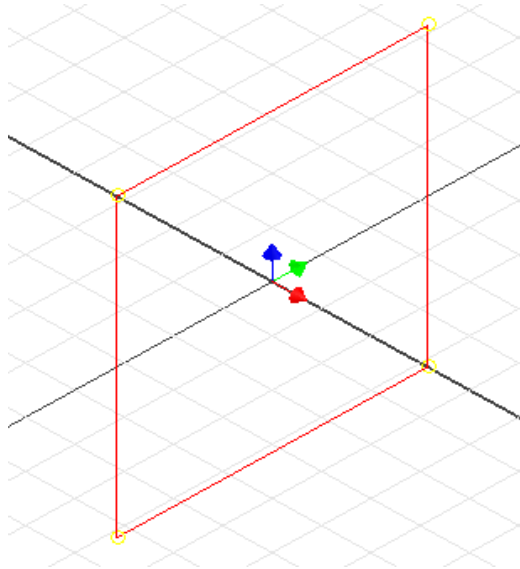


**Figure Step 4A**

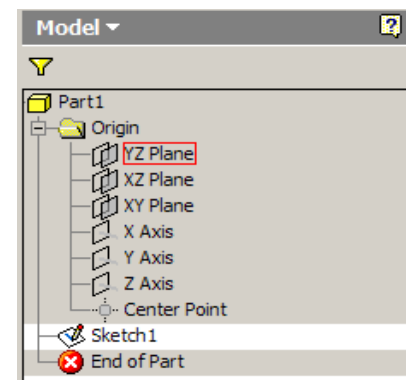


**Figure Step 4B**

**Step 5** Place the cursor on the YZ Plane in the Browser Bar. Note the orientation of the plane on the sketch. (Figure Step 5A and 5B)



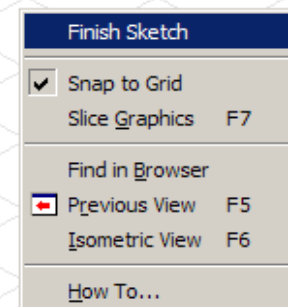
**Figure Step 5B**



**Figure Step 5A**

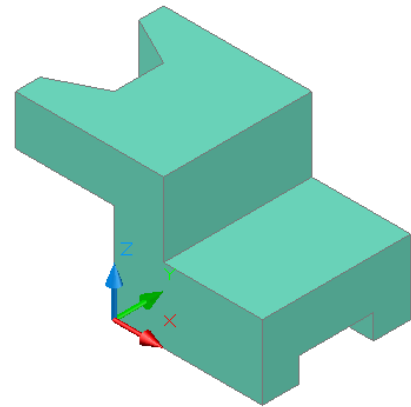
**Author's Comments:** The YZ plane is the right side view.

**Step 6** Place the cursor anywhere in the graphic window and right click the mouse. In the right-click menu, click Finish Sketch. Save the part with the name Inventor Workalong 10-1. (Figure Step 6)

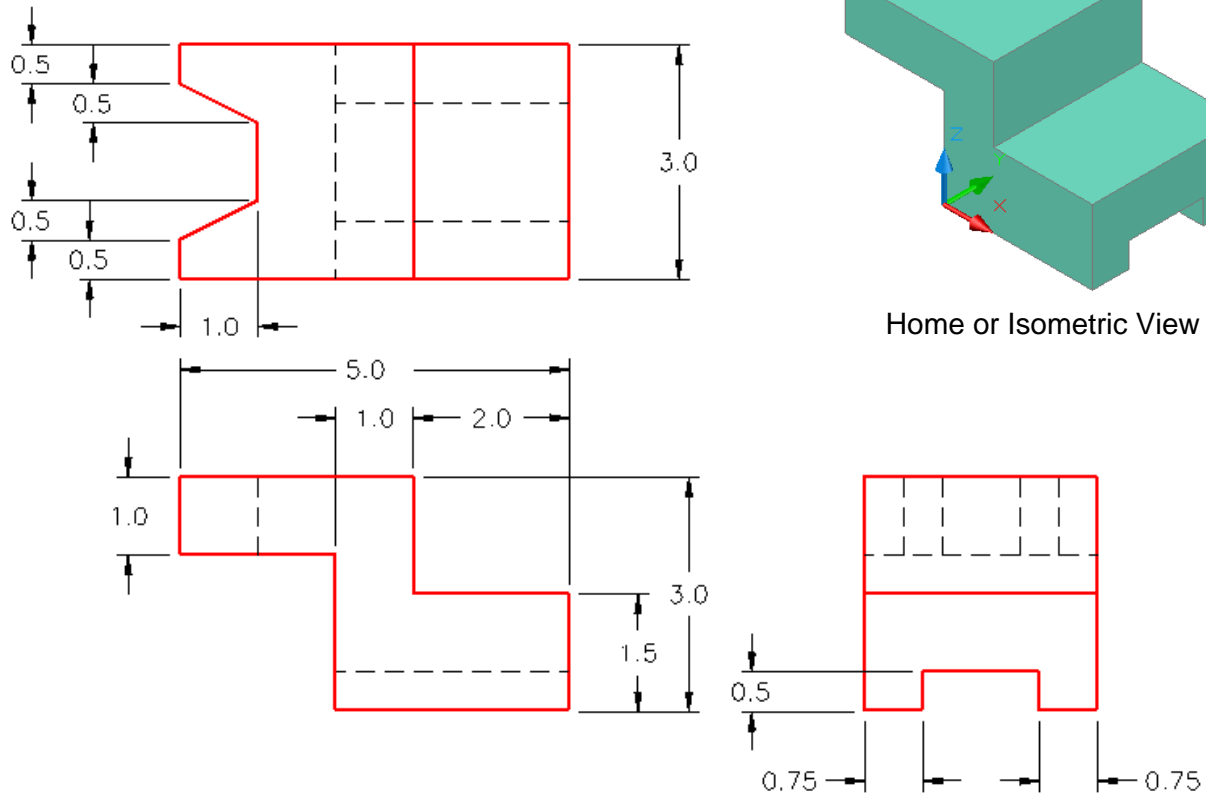


**Figure Step 6**

**Author's Comments:** The object shown below is the model that you will be constructing in this workalong.



Home or Isometric View



Dimensioned Multiview Drawing

**Author's Comments:** Before starting the base sketch you must pick the best view to draw it on. It should, in most cases, be the view with the most complex contour. For this object, the best view to use is the front view or the XZ plane.

**Step 7** While In model mode, expand the Origin folder in the Browser Bar and right-click the XZ plane. In the right-click menu, click New Sketch as shown in figure. (Figure Step 7)

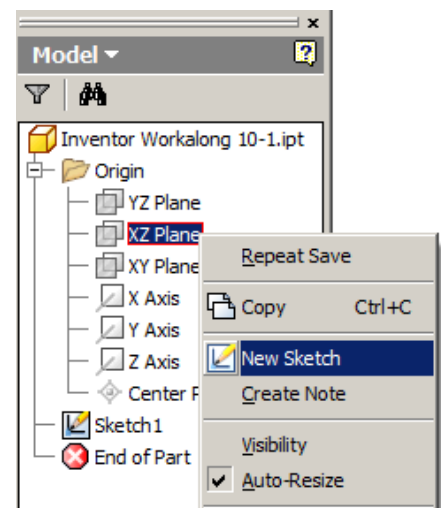
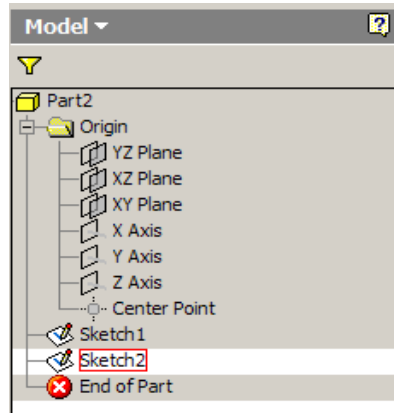
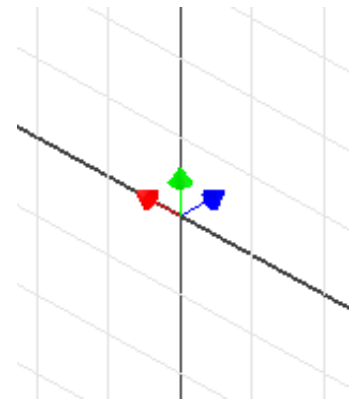


Figure Step 7

**Step 8** The graphic window will change to sketch mode in the XZ plane. Note that in the Browser Bar a new sketch will appear named Sketch2. (Figure Step 8A and 8B)



**Figure Step 8B**

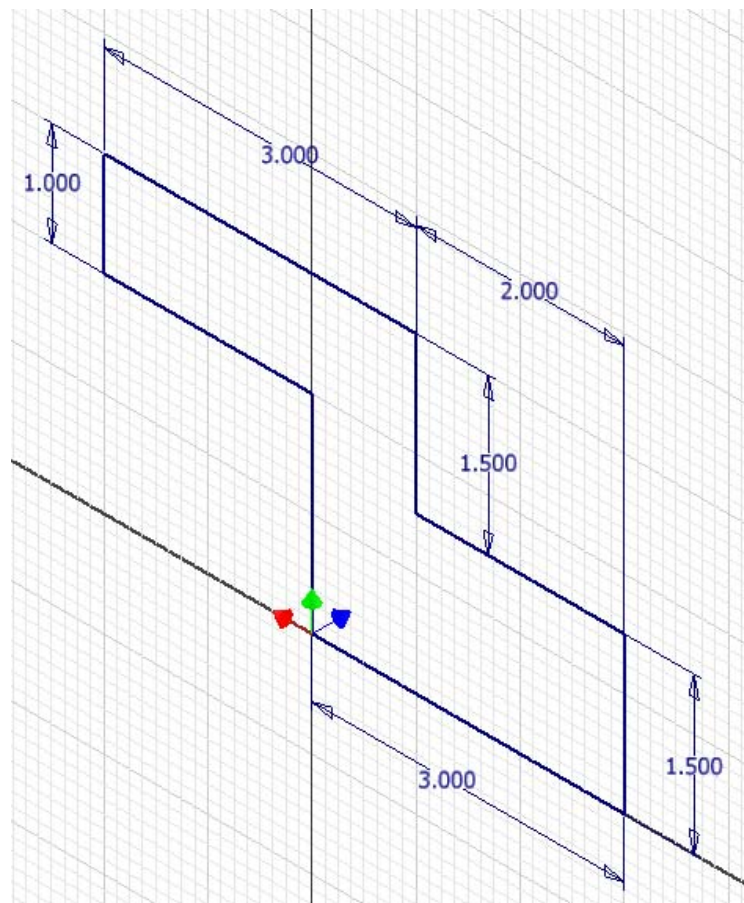


**Figure Step 8A**

**Author's Comments:** Since Sketch1 was not used, it is blank and is an unconsumed sketch.

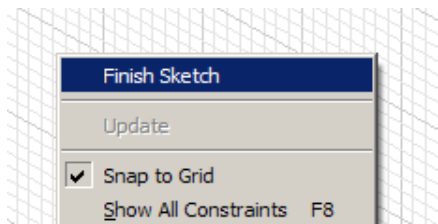
**Step 9** Project the Center Point onto the sketch plane. Draw the base sketch for the object applying all of the necessary geometrical constraints to maintain the shape of the sketch. Note the location of X0Y0Z0. Insert the necessary driving dimensions to fully constrain the sketch. (Figure Step 9A and 9B)

**Author's Comments:** All the lines in the sketch should display purple. If they do not, start the workalong over again.

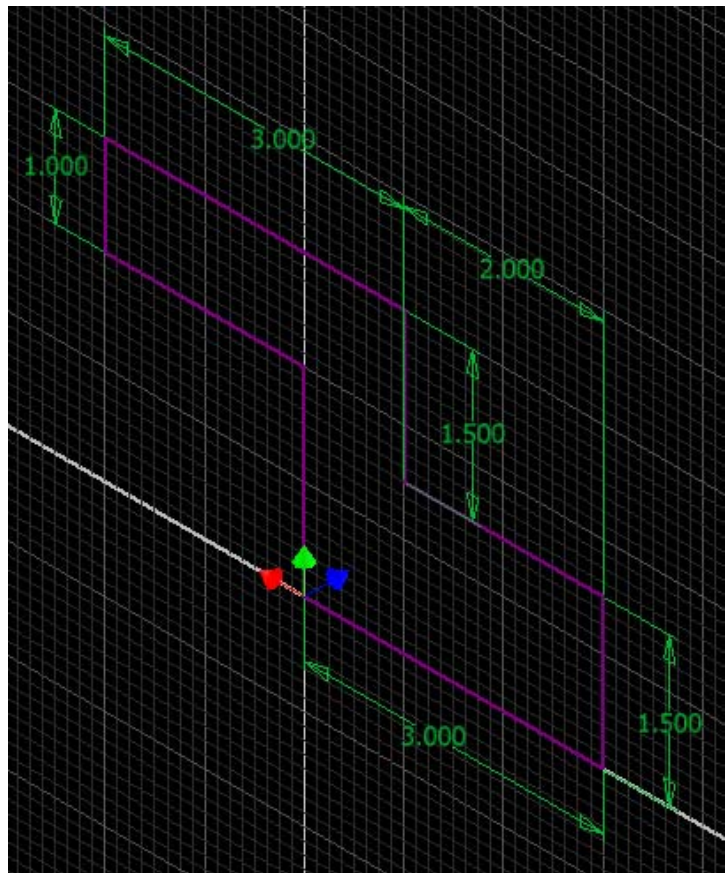


**Figure Step 9A**

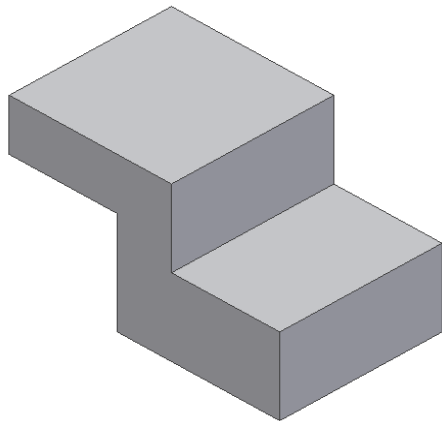
**Step 10** Right-click anywhere in the graphic window. In the right-click menu, click Finish Sketch to return to model mode. (Figure Step 10)



**Figure Step 10**



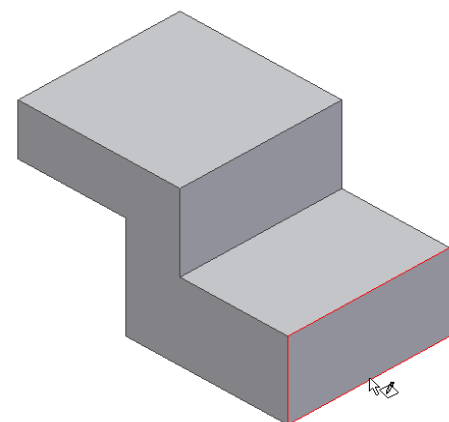
**Figure Step 9B**



**Figure Step 11**

**Step 11** Extrude the sketch to create the solid model as shown in figure. (Figure Step 11)

**Step 12** Using the 2D SKETCH command, or even better the shortcut S, start a new sketch and select the right side as the plane to draw it on. (Figure Step 12)



**Figure Step 12**

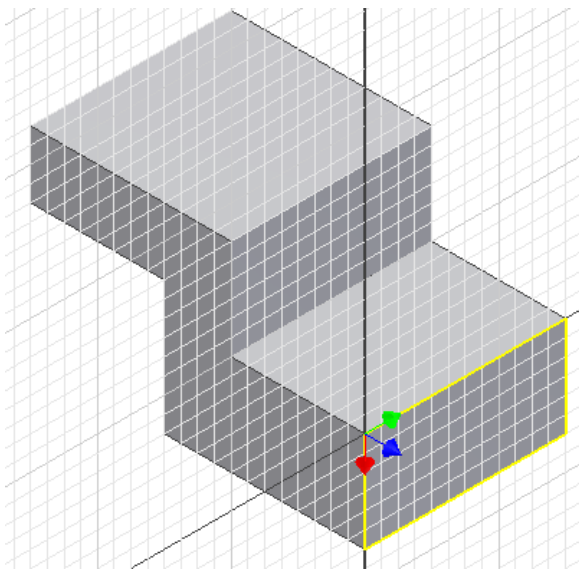


Figure Step 13

**Step 13** The grid will display on the right side. It will be Sketch3 in the Browser Bar. (Figure Step 13)

**Step 14** Draw three lines for the slot. Apply all of the necessary geometrical and dimensional constraints to fully constrain the sketch. (Figure Step 14A and 14B)

**Author's Comments:** Ensure that you snap onto the edge when you start the first line and end the third line. By doing that, you only are required to draw 3 lines to fully constrain the sketch. See Module 7 if you have trouble doing this.

**St**

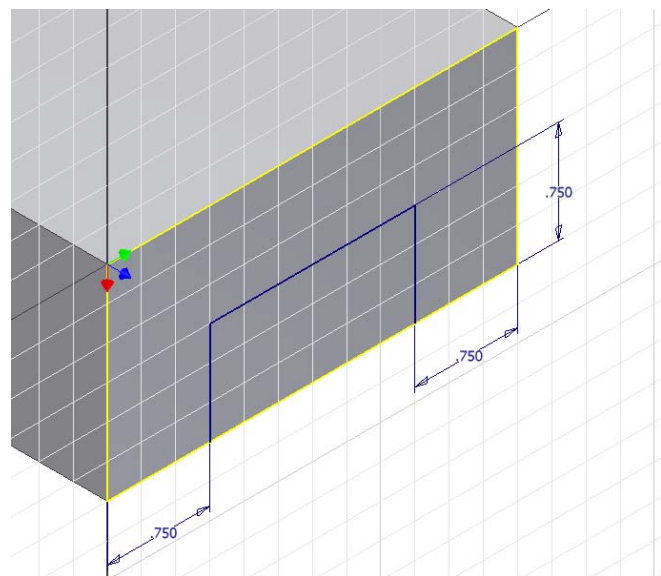


Figure Step 14A

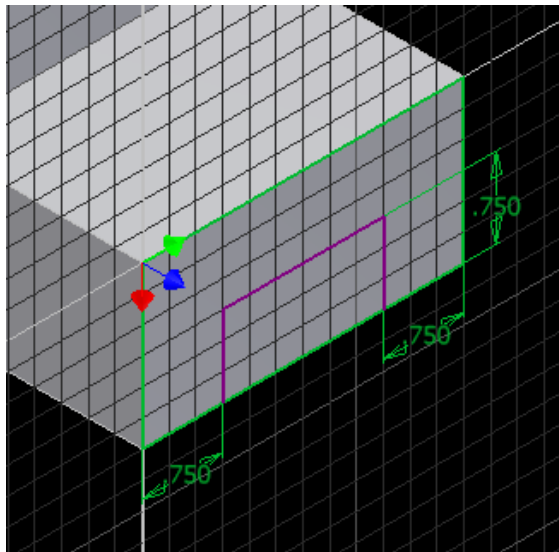


Figure Step 14B

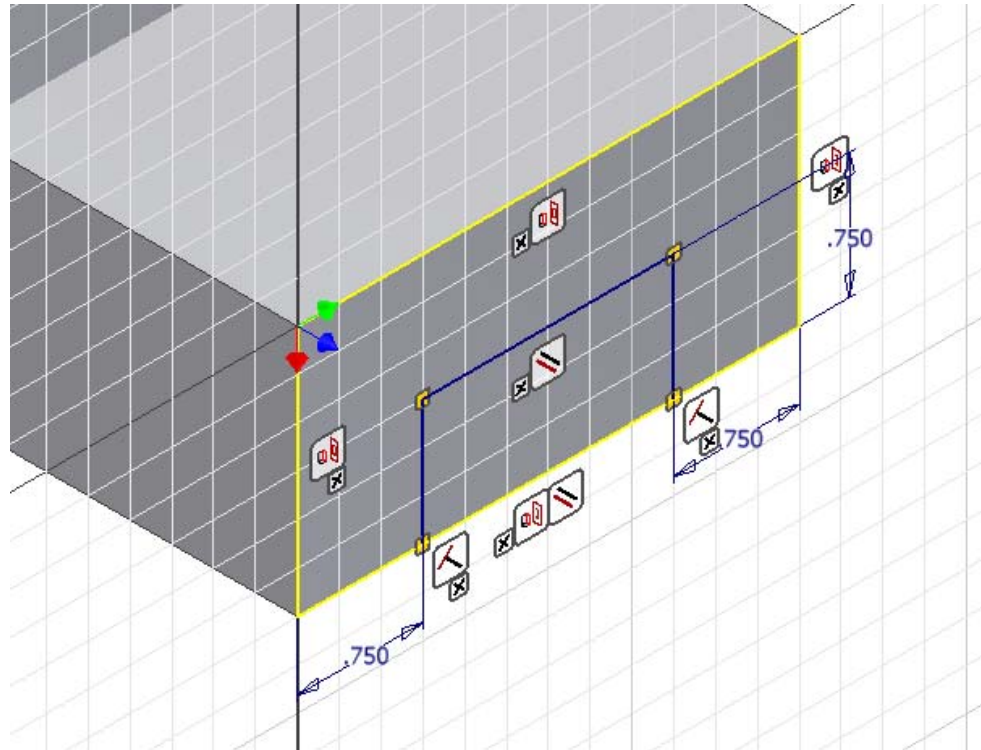
**Author's Comments:** All three lines in the sketch should display purple to indicate the sketch is fully constrained. Do not continue on with this workalong until the sketch is fully constrained.



**Step 15** Press F8 to enable the display of the constraint icons. They should be similar to the figure. (Figure Step 15)

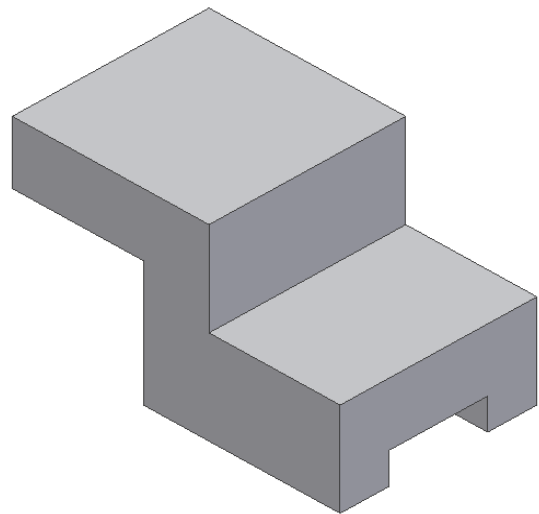
**Author's Comments:**

Inventor 2008 will display the icons slightly different than 2009. 2009 is shown here.



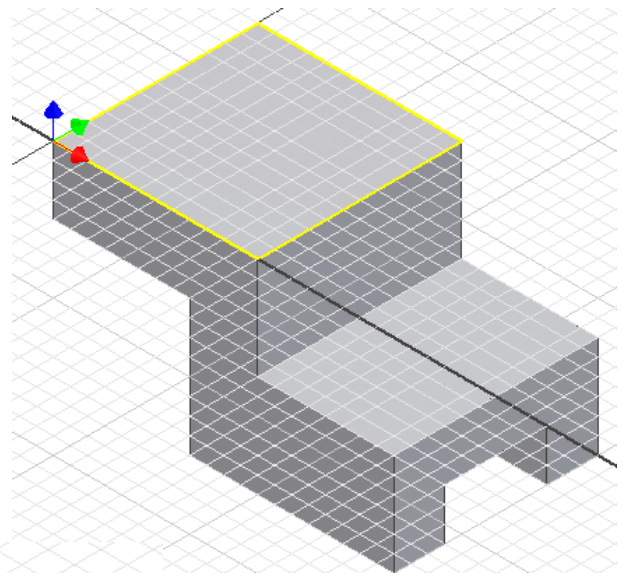
**Figure Step 15**

**Step 16** Press F9 to disable the display of the constraint icons. Extrude the sketch using the To extents. (Figure Step 16)



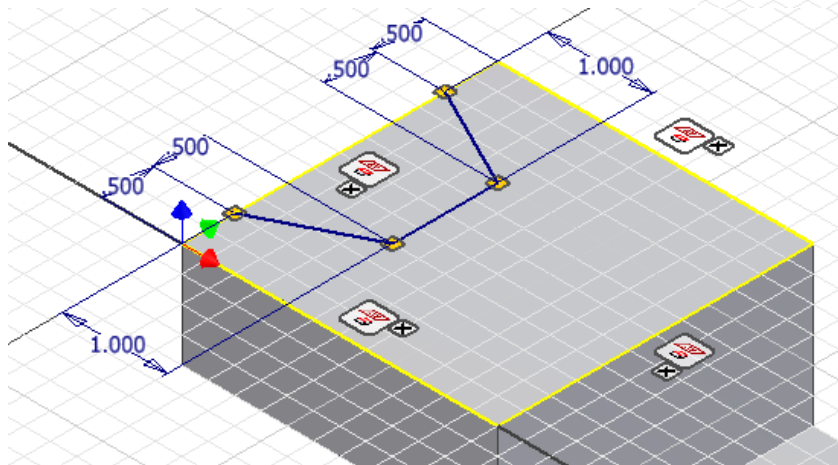
**Figure Step 16**

**Step 17** Start a new sketch on the top plane as shown in the figure. (Figure Step 17)



**Figure Step 17**

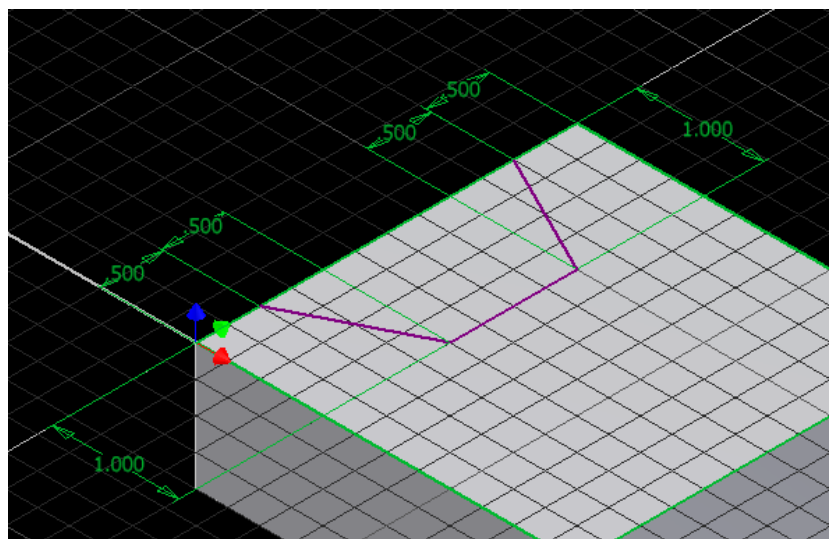
**Step 18** Draw the 2D sketch on the new sketching plane and insert the necessary dimensions to fully constrain it. (Figure Step 18A and 18B)



**Figure Step 18A**

**Author's Comments:** Inventor 2008 will display the icons slightly different than 2009. 2009 is shown here.

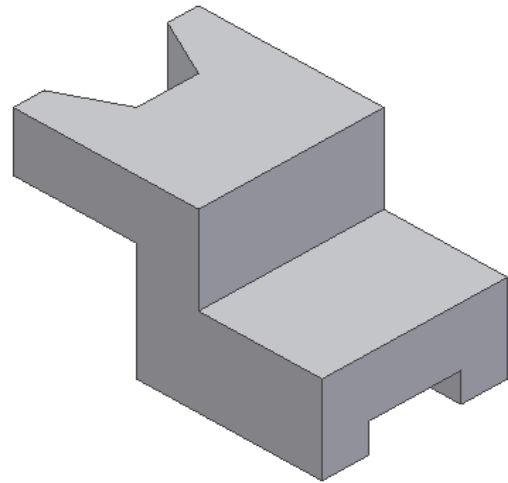
**Author's Comments:** Ensure that you snap onto the edge. That way, you will only be required to draw three lines to fully constrain the sketch.



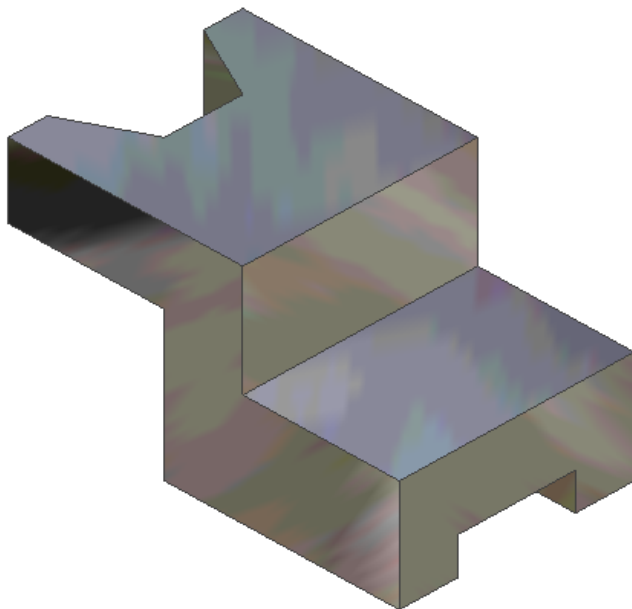
**Figure Step 18B**

**Step 19** Extrude the top sketch to complete the solid model. (Figure Step 19)

**Step 20** Change the view to the home or isometric view and apply the color Black Chrome. Orbit or rotate the view to check the bottom of the solid model. (Figure Step 20)



**Figure step 19**



**Figure Step 20**

**Step 21** Save and close the file.

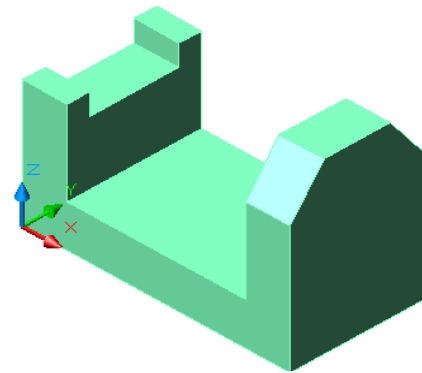
### **The Key Principles in Module 10**

1. Inventor has three predefined planes that can be used to draw the base sketch on. They are the XY, XZ and YZ planes. The XY plane is the top view, the XZ plane is the front view and the YZ is the right side view.
2. A consumed sketch is a 2D sketch that has been extruded or revolved. An unconsumed sketch is a 2D sketch that is blank or one that has not been extruded or revolved.

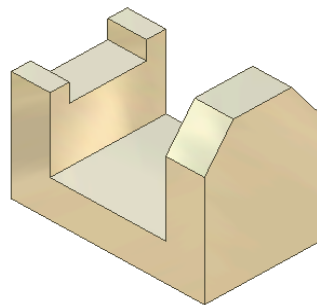
<b>Lab Exercise 10-1</b>		<b>Time Allowed: 45 Min.</b>
<b>Part Name:</b> Inventor Lab 10-1	<b>Template:</b> English - Modules Part (in).ipt	<b>Units:</b> Inches
<b>Project:</b> Inventor Course	<b>Color:</b> Metal-Titanium (Polished)	<b>Material:</b> N/A

**Instructions:**

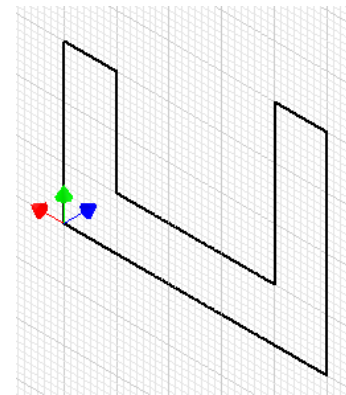
1. Project the center point onto the base sketching plane.
2. Note the location of X0Y0Z0.
3. Draw the necessary sketches and extrude them to produce the solid model shown below. Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches.
4. Apply the color as shown above.
5. Save the file with the name Inventor Lab 10-1 as shown above.



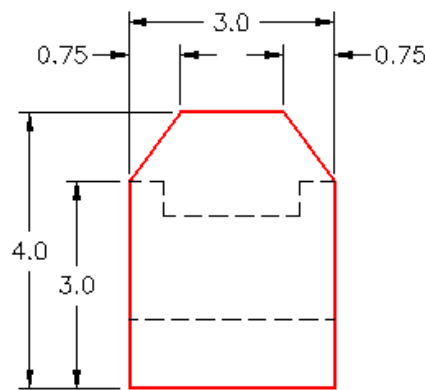
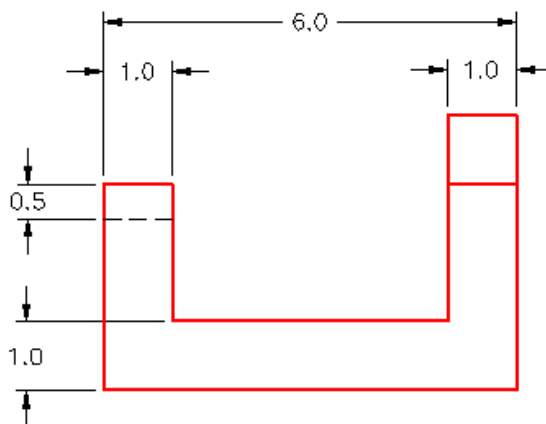
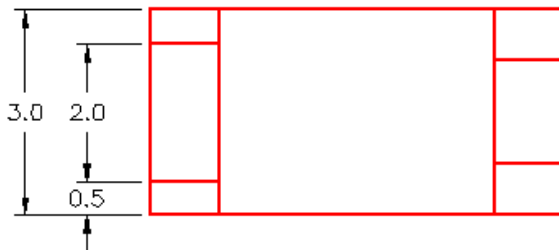
3D Model - Home or Isometric View



Completed Solid Model - Home or Isometric View



Suggested Base View - Front XZ Plane

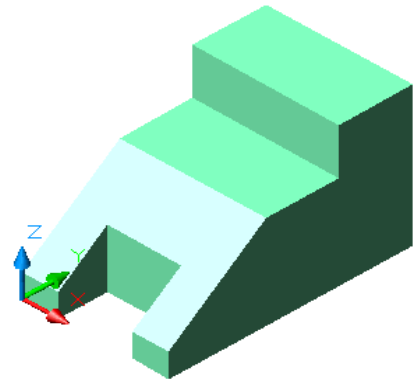


Dimensioned Multiview Drawing

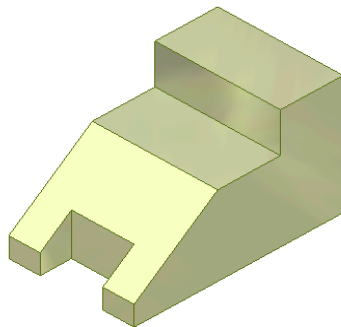
<b>Lab Exercise 10-2</b>		<b>Time Allowed: 45 Min.</b>
<b>Part Name:</b> Inventor Lab 10-2	<b>Template:</b> Metric - Modules Part (mm).ipt	<b>Units:</b> Inches
<b>Project:</b> Inventor Course	<b>Color:</b> Zinc Chromate	<b>Material:</b> N/A

**Instructions:**

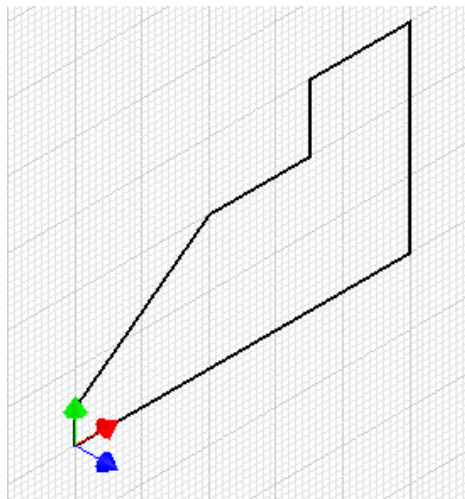
1. Project the Center Point onto the base sketching plane.
2. Note the location of X0Y0Z0.
3. Draw the necessary sketches and extrude them to produced the solid model shown below. Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches.
4. Apply the color shown above.
5. Save the file with the name Inventor Lab 10-2 as shown above.



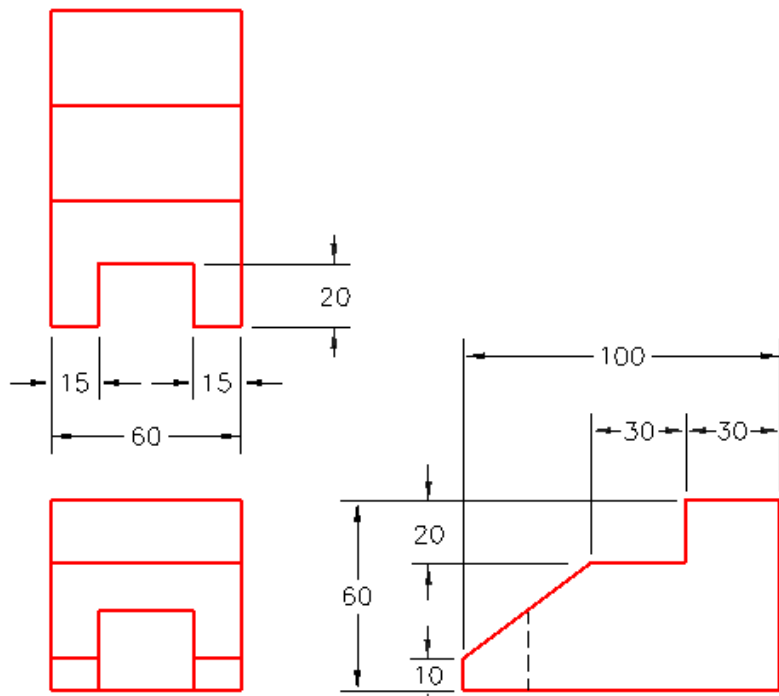
3D Model - Home or Isometric View



Completed Solid Model - Home or Isometric View



Suggested Base View - Right Side - YZ Plane



Dimensioned Multiview Drawing

## CHAPTER 2

### Tutorial

# Applying Colors to Parts

#### Learning Objectives

---

*After Completing this chapter, you will be able to:*

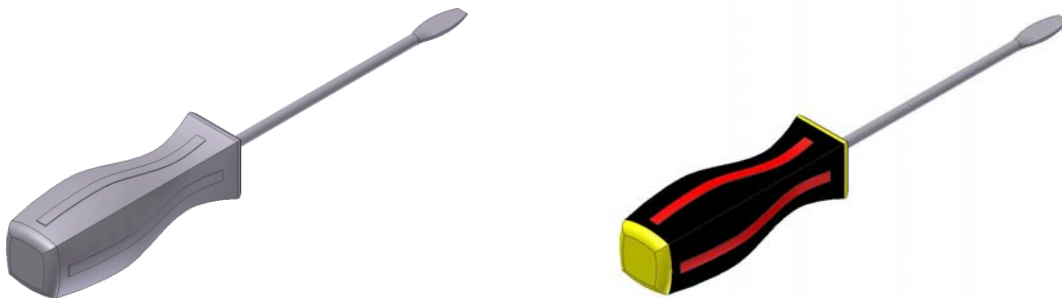
- Apply colors to entire parts, features and surfaces
- Override part colors with feature and face colors

#### Required Objectives

---

*Before stating this tutorial you should have been able to:*

- Select features in the object browser
  - Understand the concept of colors
- 



**Figure 1**

1. Open the file **Screwdriver1.ipt**. Note that the screwdriver's default color is gray (As Material).
2. Find **Loft1** in the browser and right click on it. Select **Properties** from the context menu as shown in Figure 2.
3. In the **Feature Properties** dialogue, select **BlackRubber** in the **Feature Color Style** pulldown and click OK. The body of the screwdriver will change to black .
4. Select **Emboss1** in the browser and right click on it. Select **Properties** from the context menu.

- In the **Feature Properties** dialogue, select **Red** in the pulldown and click OK. Only the first ridge on the body of the screwdriver will change to red. The rest will remain gray as shown in Figure 3.

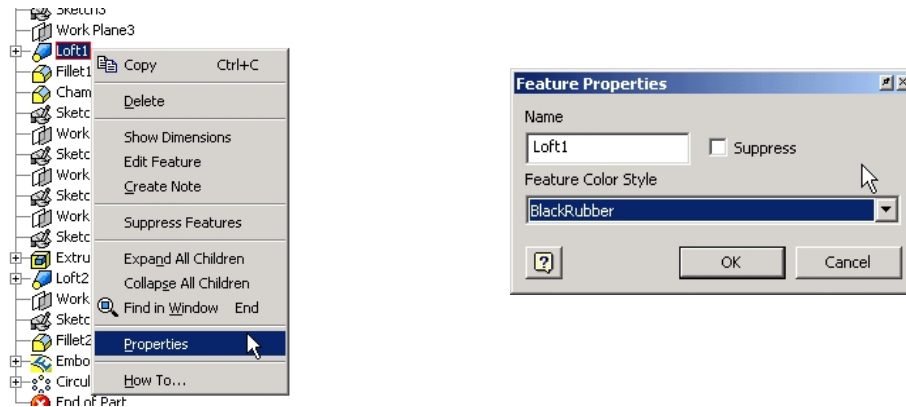


Figure 2

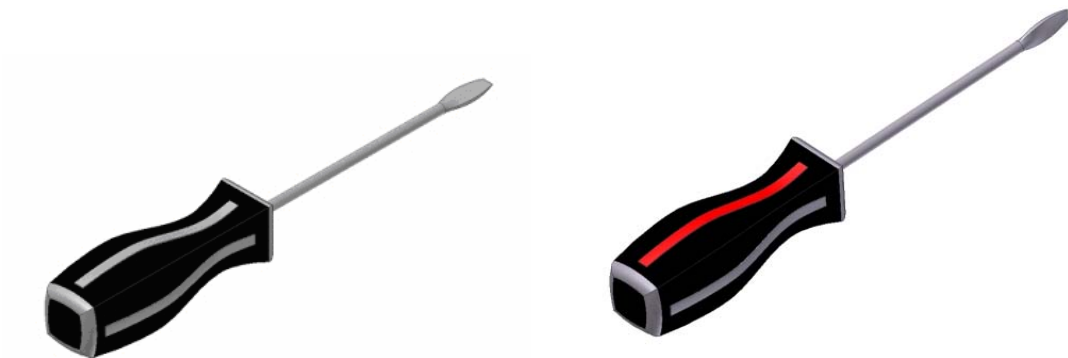


Figure 3

- Select **CircularPattern1** in the browser and right click on it. Select **Properties** from the context menu.
- In the **Feature Properties** dialogue, select **Red** in the pulldown and click OK. The remaining ridges on the body of the screwdriver will change to red.
- Zoom in on the base of the screwdriver and select the four fillets and face that make up the base. After all five faces have been selected, as shown in Figure 4, right click and select **Properties** from the context menu to open the **Face Properties** dialogue. In the **Face Color Style**, select **Yellow** and click OK.

Select multiple faces by holding down the **Control** key. To remove a face or line, hold **Shift** and select the line.

- Repeat the same process for the chamfers that make up the part of the screwdriver that connects to the blade as shown in Figure 5.

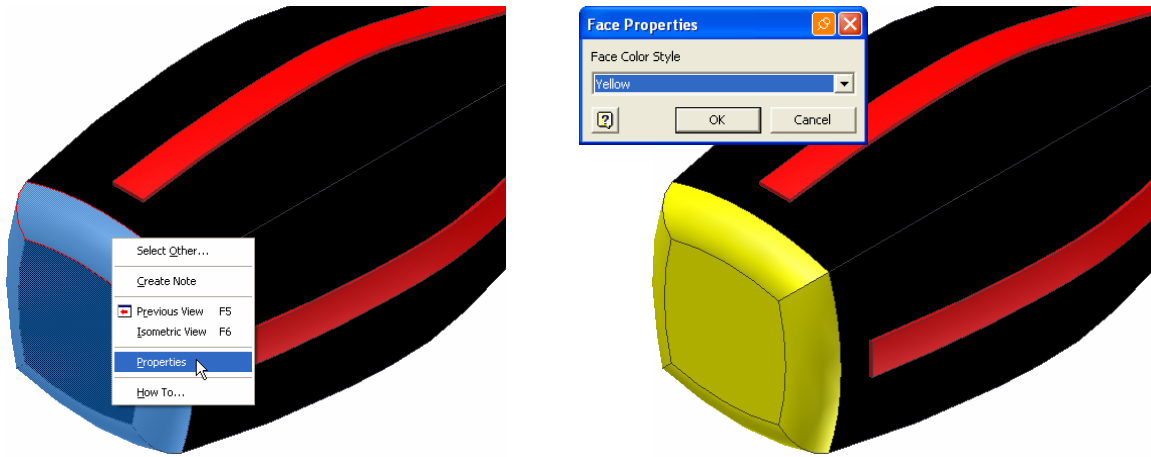


Figure 4

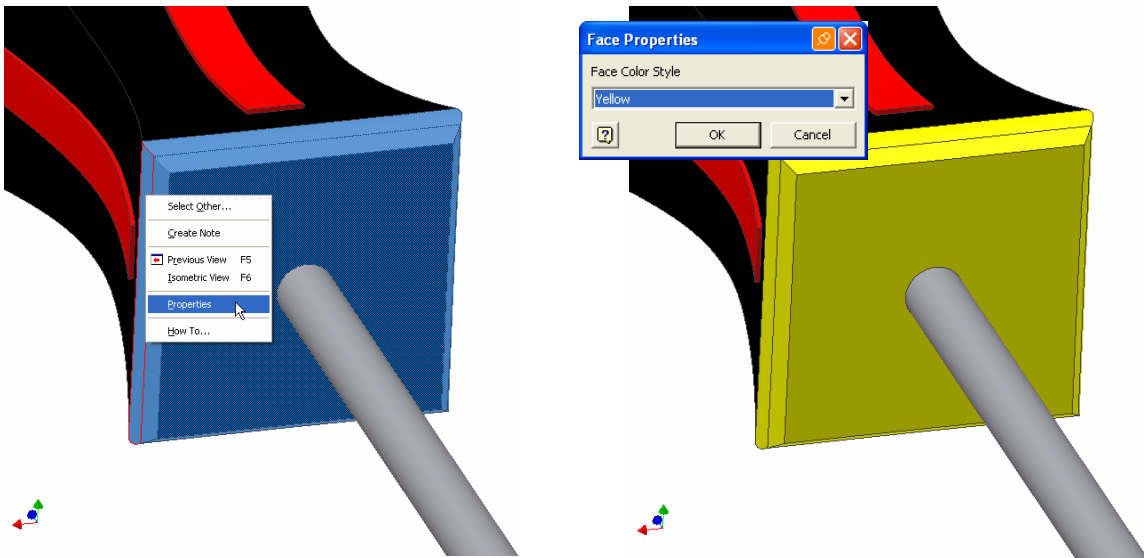
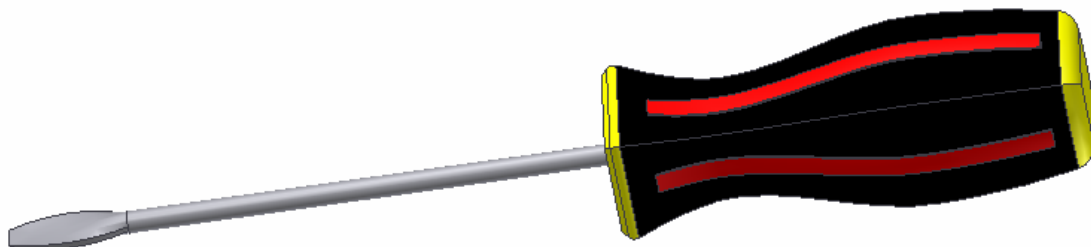


Figure 5

- Change the default color of the part via the color pulldown on the standard toolbar to any other color. Notice that only the color of the shaft and blade change. This is because the blade extrusion and loft's properties were not overridden and their color styles were set to "as part".





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# CHAPTER 2

## Tutorial

# Defining Materials

### Learning Objectives

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After Completing this chapter, you will be able to:

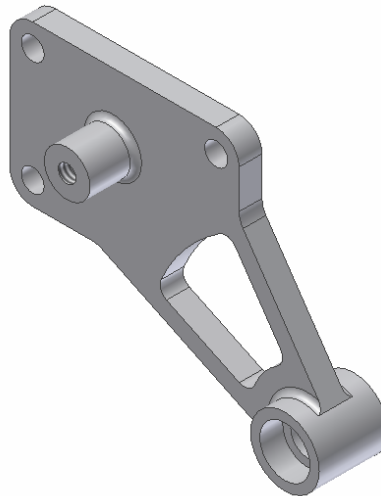
- Define materials and their associated colors

### Required Objectives

---

Before starting this tutorial you should have been able to:

- Apply colors to parts  
Create new colors
- 



**Figure 1:** Idler Bracket

1. Open the file **IdlerBracket1.ipt**.
2. Select **Format>Materials** and click **New**. Name the new material **Aluminum-6061**.
3. Enter the properties of the material as shown in Figure 2. While Inventor only utilizes values for density and render color, it is good practice to enter all values at creation time for use in 3<sup>rd</sup> party applications.

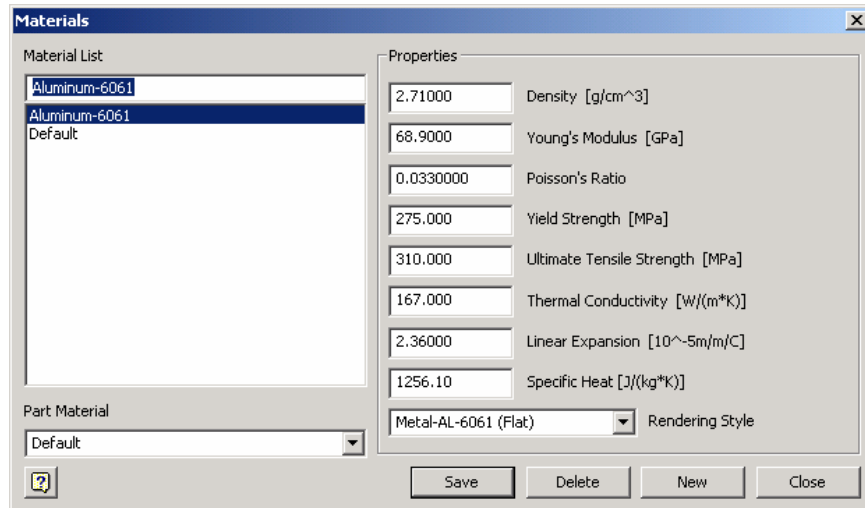
#### Professional Tip

*All values for physical properties have metric units. Be sure to convert your values if your source is stated in English units.*

- Assign the **Rendering Style** of **Metal-AL-6061 (Flat)** to the new material.

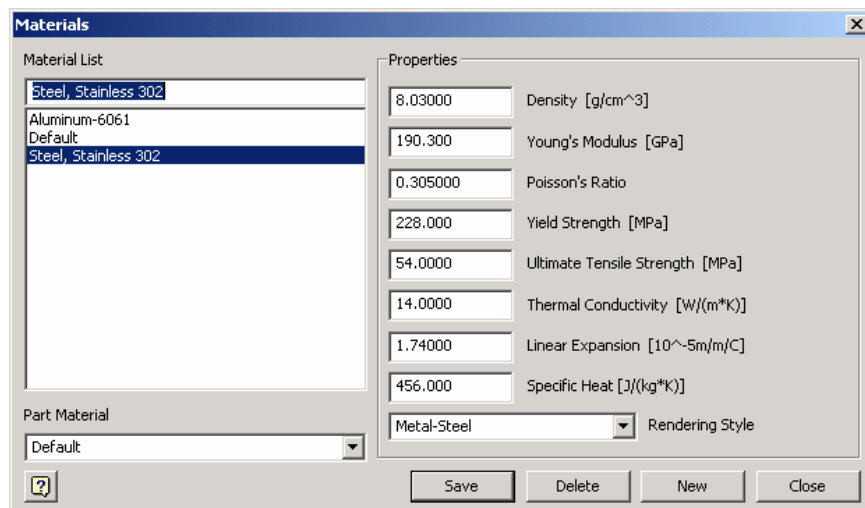
*Note: This color was predefined in the part file. If you want to apply a custom color, you must create it before creating the material.*

- Click **Save**.



**Figure 2:** Physical Properties of Aluminum 6061

- Click **New** once again and create a new material **Steel, Stainless 302**, enter the properties of the material as shown in Figure 3.
- Assign the **Rendering Style** of **Metal-Steel** to the material.



**Figure 3:** Physical Properties of Stainless Steel 302

- Click **Save**, then **Close** and return to the model.
- In the model environment, select **File>iProperties** from the main menu. Click on the **Physical** tab.

10. Select **Aluminum-6061** from the **Material** pulldown. Note the mass of the part. Click **Apply**, note how the color of the part has changed.
11. Change the material to **Steel, Stainless 302** and note the mass of the part. Click **OK** and return to the model. Again, note that the color of the model has changed.

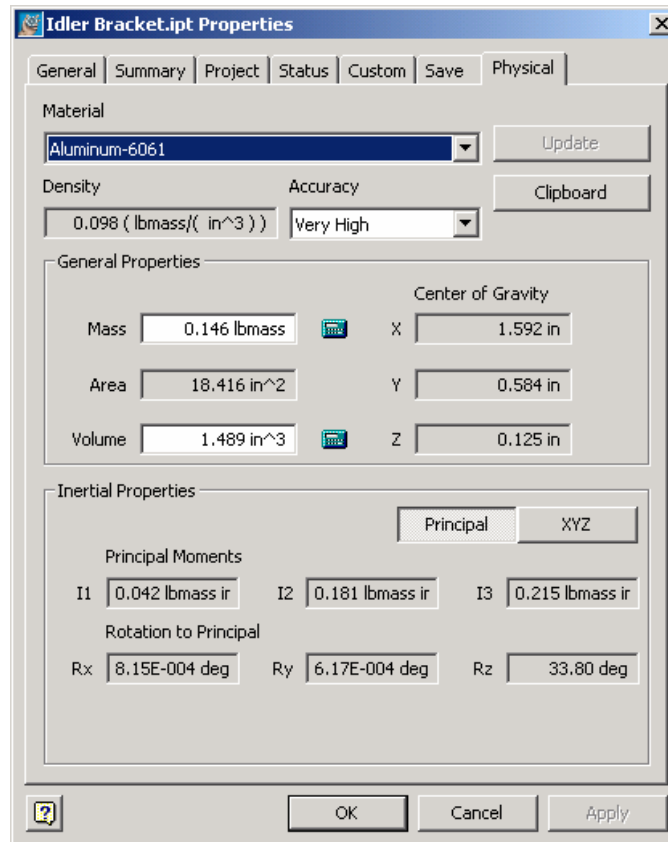


Figure 4

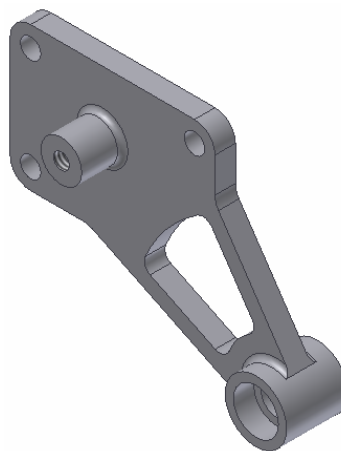


Figure 5

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# CHAPTER 3

## TUTORIAL

# Simple Extruding of a Sketch

### Learning Objectives

---

*After completing this tutorial, you will be able to:*

- Sketch an object
- Extrude a sketch

### Required Competencies

---

*Before starting this tutorial, you should have been able to:*

- Construct, constraint and dimension sketches
- Make work planes visible

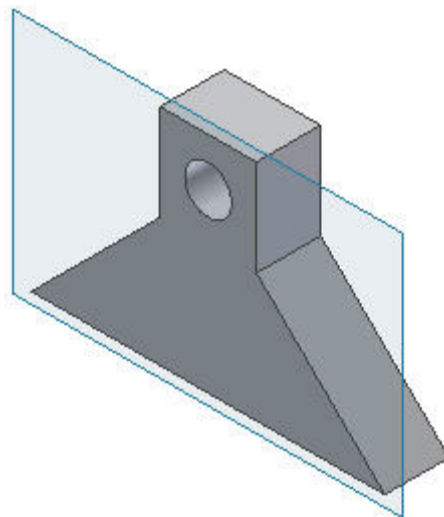


Figure 1

1. Open a new Standard.ipt
2. Click on the plus sign next to origin in the **Model** panel bar as shown in Figure 2. Click the right mouse button over the **XY Plane** and select visibility as shown in Figure 3.
- 3.

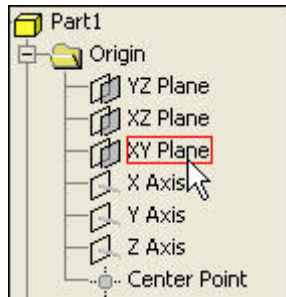


Figure 2

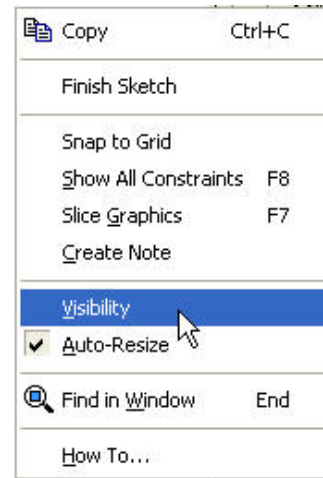


Figure 3

3. Select the **Line** Button in the 2D sketch panel and create the sketch shown in Figure 4.

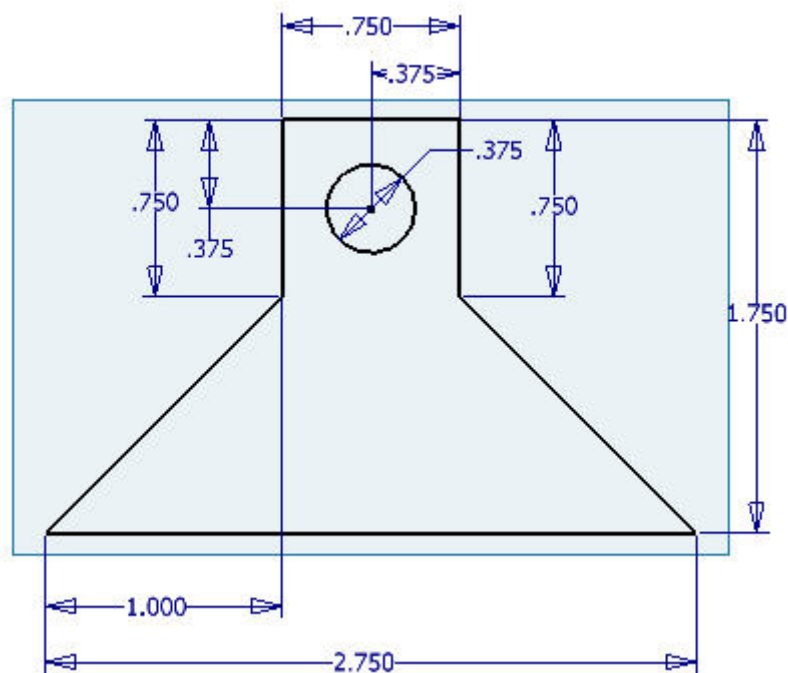


Figure 4

4. Once the sketch is completed and fully dimensioned click on **Return** in the Inventor Standard menu at the top of the screen.

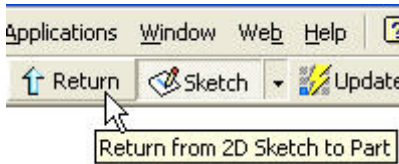


Figure 5

5. Move the cursor into the workspace and right click to open the options menu. Select **Isometric View** as shown in Figure 6.



Figure 6

6. Select **Extrude** in the Part Features menu as shown in Figure 7. This opens the Extrude dialog box as shown in Figure 8.



Figure 7

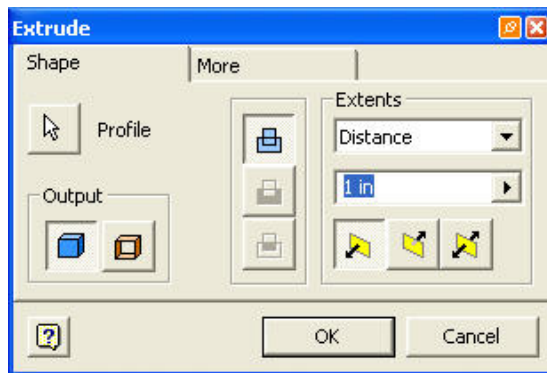


Figure 8



7. The **Profile** button should be selected as shown in Figure 9.

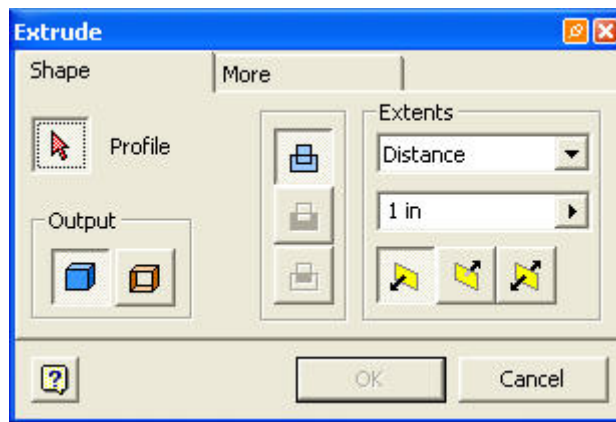


Figure 9

8. Move the cursor over the sketch till it becomes highlighted and left click, making sure that the circle is not included, as shown in Figure 10.

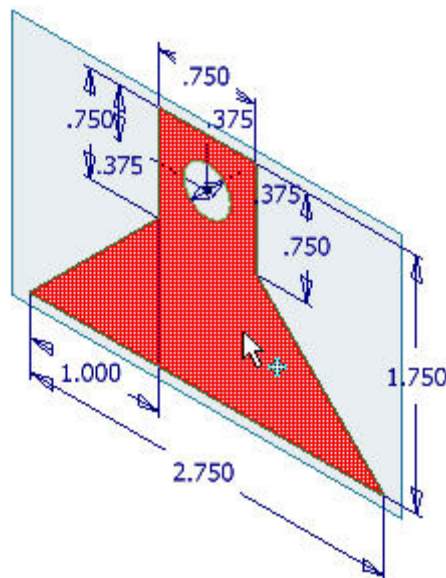


Figure 10

9. Set the distance as .5 in and select the middle button with the arrow pointing up and away from the yellow plane as shown in Figure 11.

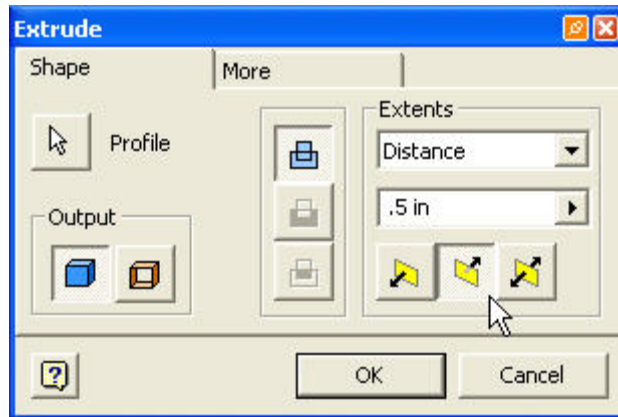


Figure 11

10. Click on the **OK** button and the finished part should look like Figure 12.

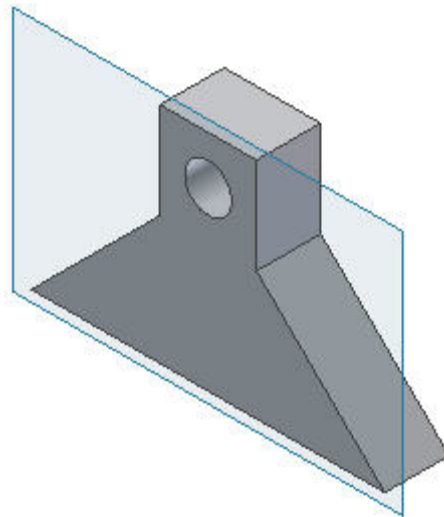


Figure 12

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# CHAPTER 3

## TUTORIAL

# Simple Extruding of a Sketch

### Learning Objectives

---

*After completing this tutorial, you will be able to:*

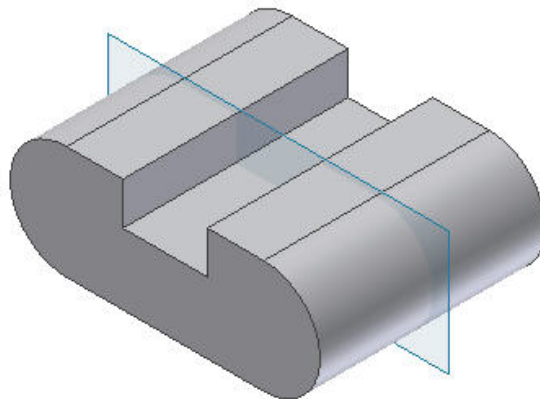
- Sketch an object
- Extrude a sketch

### Required Competencies

---

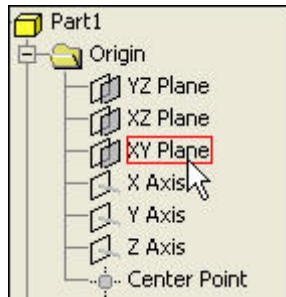
*Before starting this tutorial, you should have been able to:*

- Construct, constraint and dimension sketches
- Make work planes visible

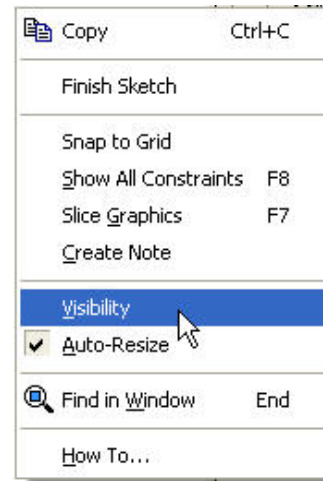


**Figure 1**

1. Open a new Standard.ipt
2. Click on the plus sign next to origin in the **Model** panel bar as shown in Figure 2. Click the right mouse button over the **XY Plane** and select visibility as shown in Figure 3.

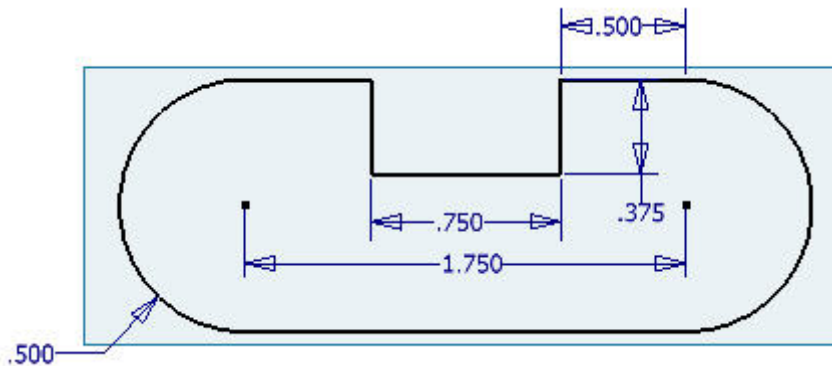


**Figure 2**



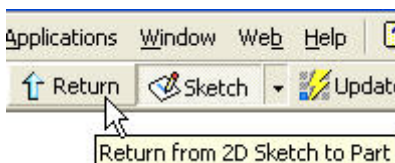
**Figure 3**

3. Select the **Line** Button in the 2D sketch panel and create the sketch shown in Figure 4.



**Figure 4**

4. Once the sketch is completed and fully dimensioned click on **Return** in the Inventor Standard menu at the top of the screen.



**Figure 5**

5. Select **Extrude** in the Part Features menu as shown in Figure 6. This opens the Extrude dialog box as shown in Figure 7.



Figure 6

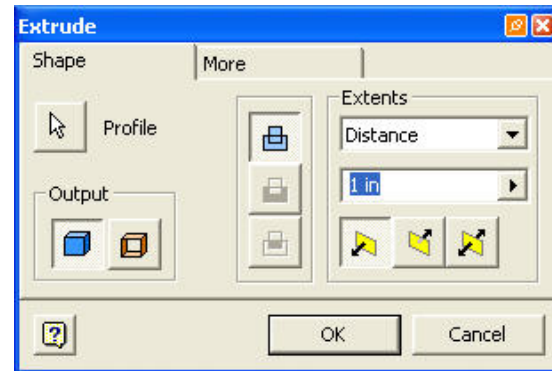


Figure 7

6. The profile should already be highlighted in your work space. In the area highlighted under the Extents category type in a depth of 2 in as shown in Figure 8.

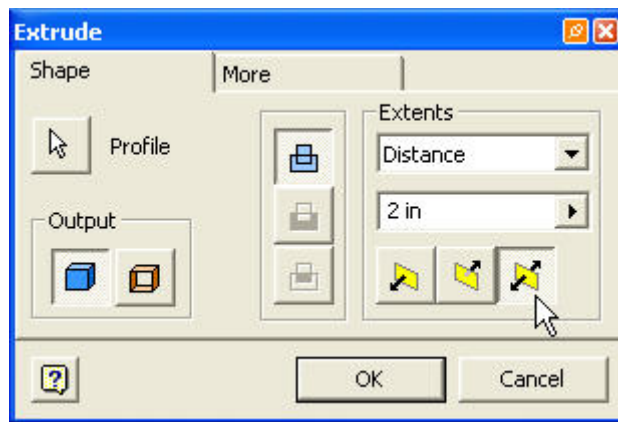


Figure 8

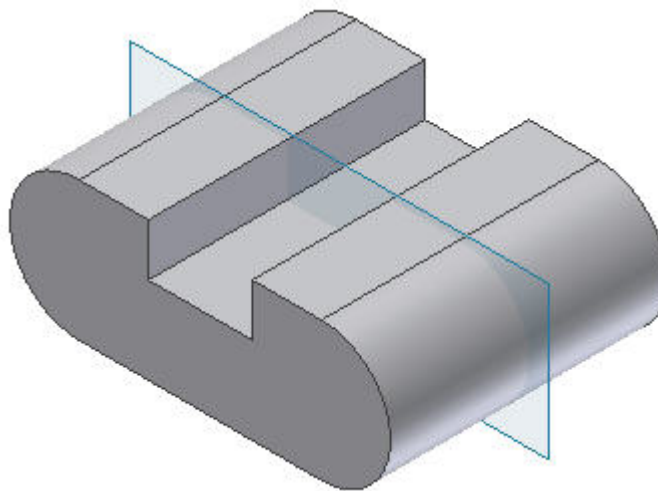
7. Select the button that has arrows pointing in both directions from the yellow plane at the bottom of the dialog box; this is considered a midplane extrusion. Click the OK button.

8. Move the cursor into the workspace and right click to open the options menu. Select **Isometric View** as shown in Figure 9.



**Figure 9**

9. The finished part should look as shown in Figure 10.



**Figure 10**

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# CHAPTER 6

## Placed Features

### Learning Objectives

---

After Completing this chapter, you will be able to:

- Understand and create the following features:
  - Holes
  - Fillets
  - Chamfers
  - Shell
  - Patterns (Rectangular, Circular)
- 

Placed features are predefined features that require specific values as well as the desired positioning. In order to use placed features there must be a sketched base feature already created. Editing of placed features is done in the *Browser* like sketched features. The use of these placed features is usually more efficient than the use of sketched features. Please refer to “Learning Inventor” Chapter 6.

### HOLES

#### Creating Holes

There are three types of holes to choose from in Inventor: *drilled, counterbore, and countersink*. Each of these hole types also have the option of being *tapped*. Holes can have the diameter and depth or termination determined in the *Hole* menu box.

#### Steps to creating holes

1. Have a base feature created in which the hole will be placed on.
2. Choose the sketch plane (surface) that the hole will be placed.
3. In order to use the hole command you must place a *Point, Hole Center* from the Sketch Panel Bar, Figure 1 .

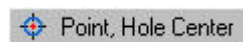


Figure 1



Figure 2

4. Once the point has been placed in the desired position return to the Features Panel Bar and select the Hole command shown in Figure 2.
5. The Hole dialog box will then open providing the different options for hole creation. There are four tabs with various setting options available for each.

## Type Tab

This tab allows you to select the hole type, termination, centers, and hole diameter as shown in Figure 3.

- Hole Types include: drilled, counterbore, and countersink.
- There are three types of Termination to choose from:
  - Distance: allows you to give a specific depth for the hole.
  - Through All: the hole will cut through the entire part.
  - To: allows you to choose a plane in which the hole will stop.
- Centers: this button will allow you to choose multiple hole centers that will share the same hole type.
- Hole Diameter: allows you to change the diameter of the hole by selecting the dimension in the dialog box and typing in the desired dimension. See Figure 3.

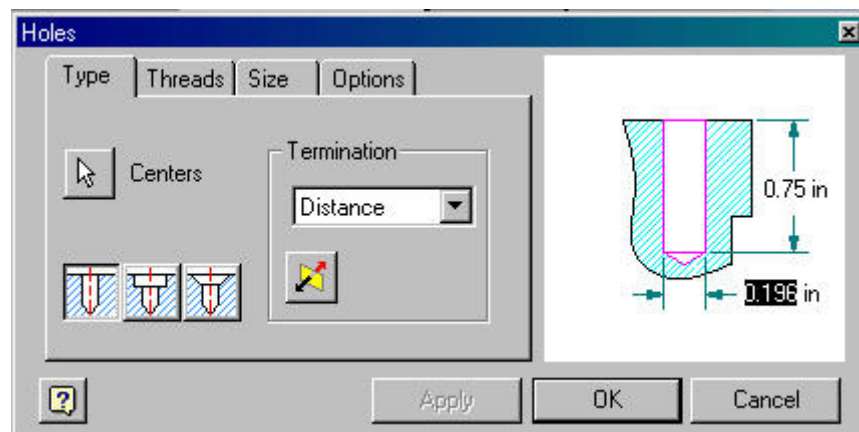


Figure 3

## Threads Tab

This tab allows you to choose whether you want the hole to be tapped and set the various values that can be defined. Values that can be changed are the thread depth, thread type, and thread direction (Right Hand, Left Hand). See Figure 4.

## Size Tab

This tab allows you to set the thread's nominal size, pitch, class, and diameter.

## Options Tab

This tab allows you to set the drill point and countersink angles.

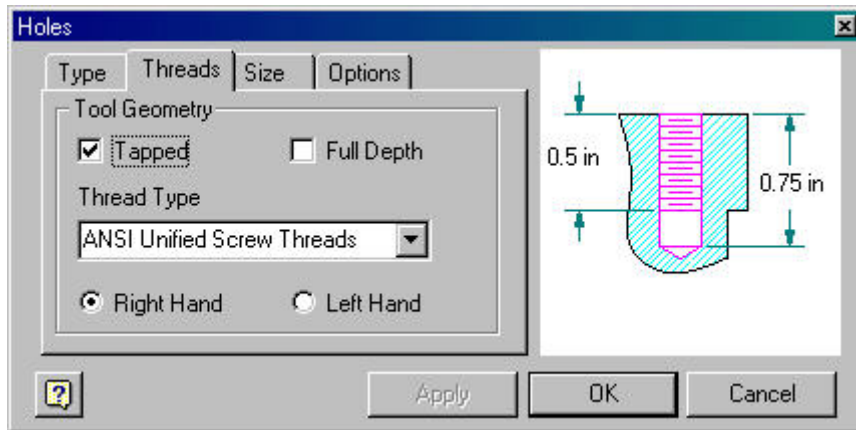


Figure 4

**NOTE**

*Any attribute of a hole can be edited by accessing the edit feature command in the Model Panel.*

**Example 1**

- Open the file “Lever.ipt”

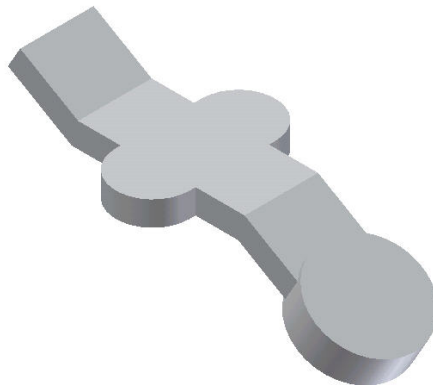


Figure 5

- Select **Sketch** from the Command Bar
  - Click on the top surface of Extrusion 1

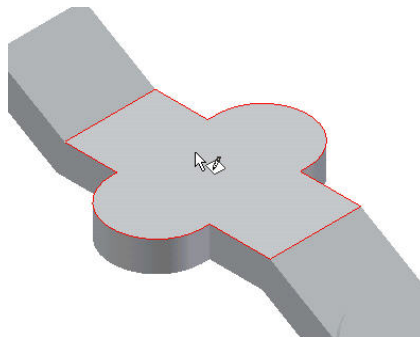
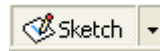
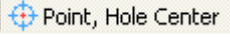


Figure 6

- Choose **Point, Hole Center**  and click on the center points of the two arcs of Extrusion 1 as shown in Figure 7.

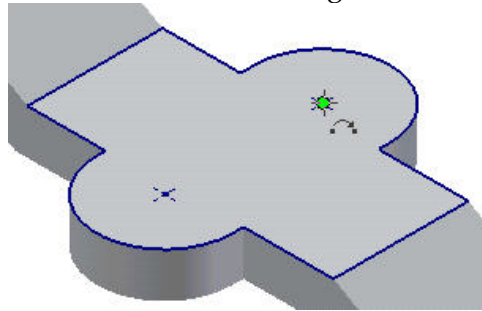
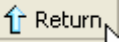



Figure 7

- Once you have placed the hole centers click **Return**. 

- Select **Hole** from the Features Panel Bar 
  - A Dialog box will appear with the hole options.
- Edit the Hole termination and diameter as shown in Figure 8.

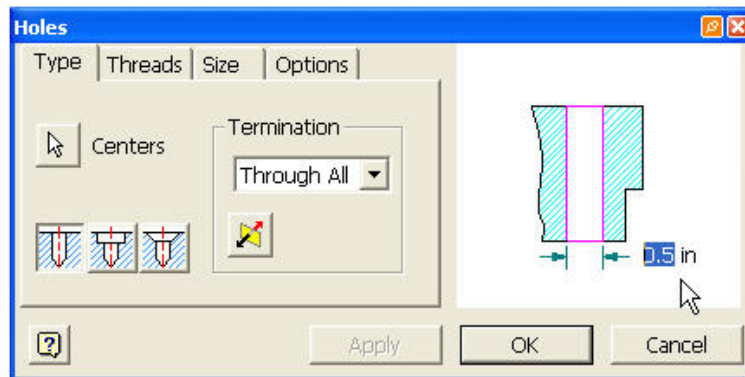


Figure 8

- Next choose the Threads tab

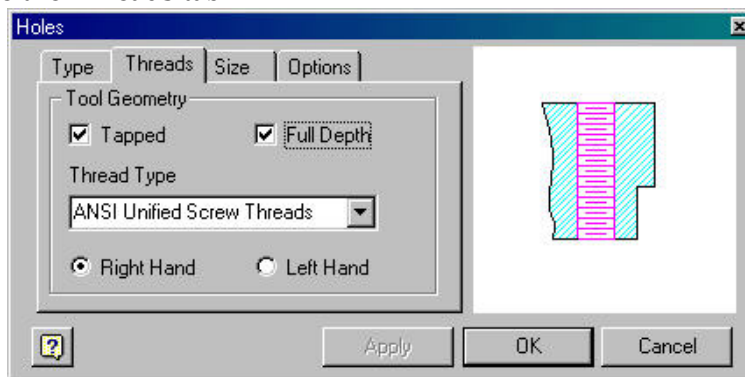


Figure 9

- ❑ Select Tapped and Full Depth
- ❑ Click the OK button

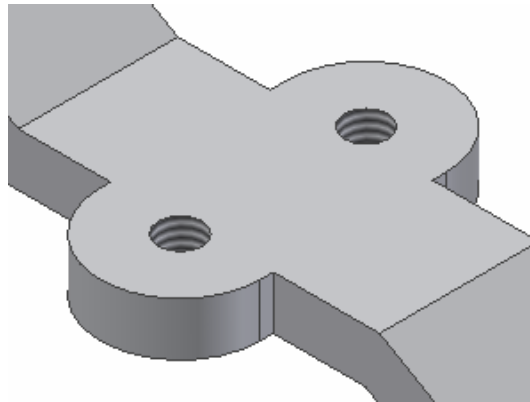


Figure 9

- ❑ Select the sloped circular surface as the new sketch plane.

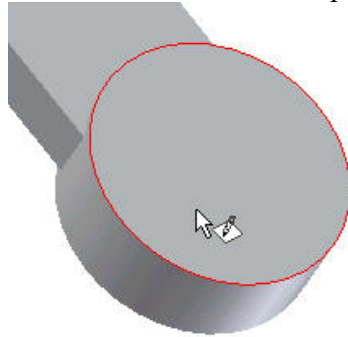


Figure 10

- ❑ Place the **Point, Hole Center** at the center point of the circle.

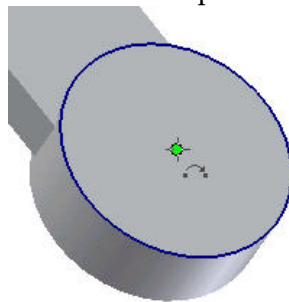


Figure 11

- ❑ Click **Return**
- ❑ Select the **Hole** command.
- ❑ Select the Threads Tab and turn off Tapped.

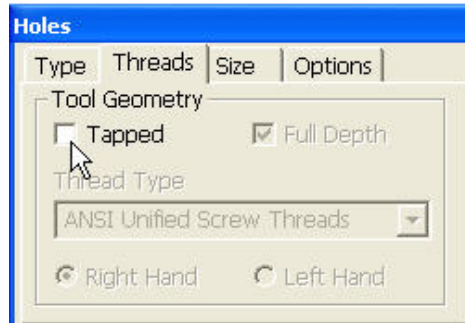


Figure 12

- Choose Counterbore for the hole type and set the dimensions as shown in Figure 13.

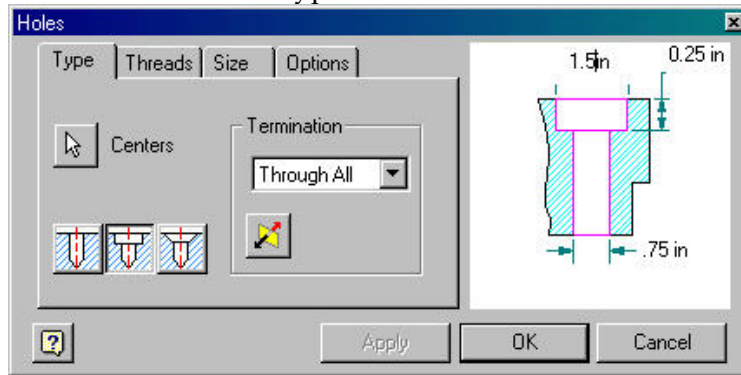


Figure 13

- Click the **OK** button

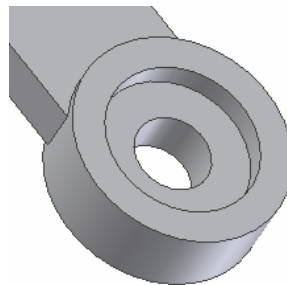


Figure 14

- Select the sloped rectangular surface as your new sketch plane.

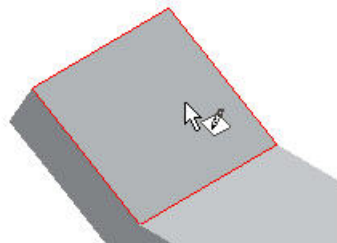


Figure 15

- Put a **Point, Hole Center** some where on the sketch plane.

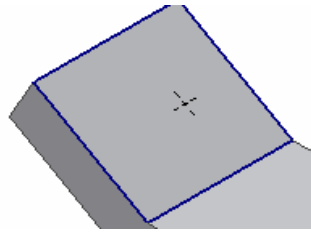


Figure 16

- ❑ Locate the Point by dimensioning it .75 in from the right edge, and .625 from the top edge. Refer to Chapter 5

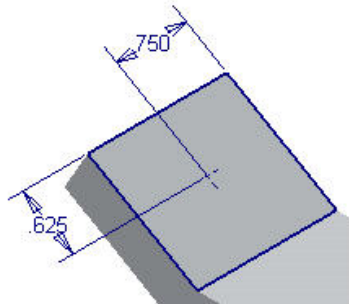


Figure 17

- ❑ Click Return
- ❑ Select the Hole feature.
- ❑ Choose Countersink for the hole type and set the dimensions as shown.

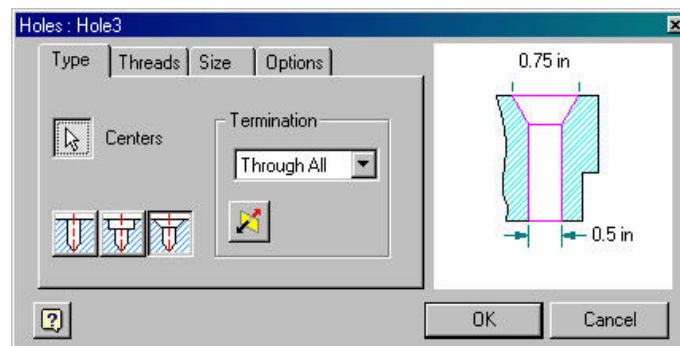


Figure 18

- ❑ Click the **OK** button.
- ❑ The finished part with all the placed holes.

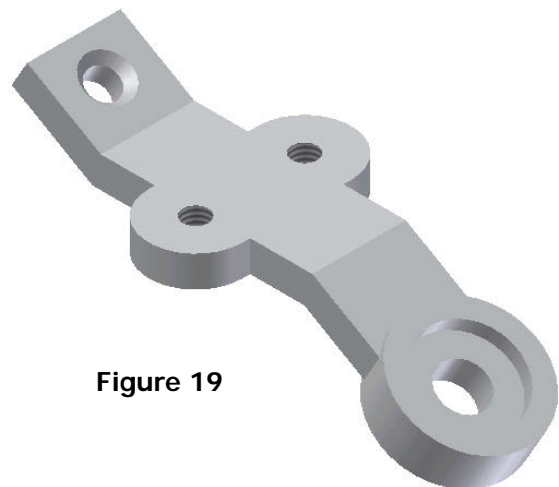


Figure 19

## FILLETS & CHAMFERS

### Creating Fillets

Inventor has three different styles of *Fillets* that can be placed from the feature menu on a given part. The first style, constant, creates fillets with the same radius from start to finish of the edge selected. Constant fillets have the following three selection options: Edge, Loop, Feature.

#### Edge

This allows you to select individual edges of an object that need to be filleted. If there are tangent edges they will be selected as one.

#### Loop

Loop selects all edges of a surface that creates a closed loop.

#### Feature

The feature option will place fillets on all edges of a selected object.

### Steps to creating a constant fillet

1. Have a base feature created in which the fillets will be placed on.
2. Choose the selection option and set the desired radius.
3. Select the edges on the object that will be filleted using whichever selection option desired.
4. Once all edges that will have the fillet placed on them are selected click the OK button and the fillets will be placed.

### Example 2

- Open the file "holder.ipt"

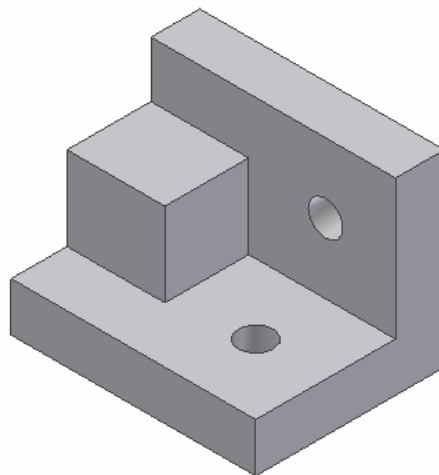


Figure 20



- Select **Fillet**  from the Features Bar which will bring up the Fillet Dialog box.

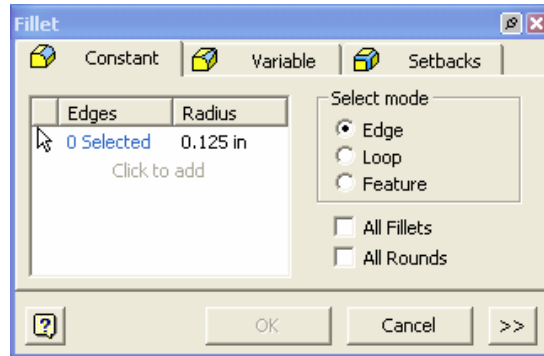


Figure 21

- Set the fillet radius to 6mm and select the four corners as shown in Figure 21 and click **OK** or press **Enter** on the keyboard.

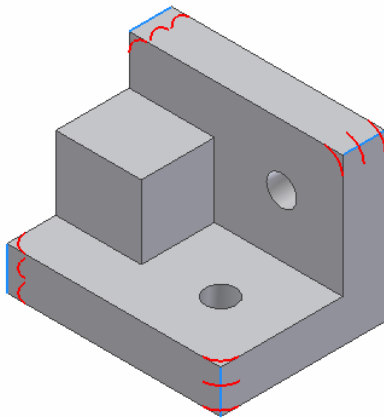


Figure 22

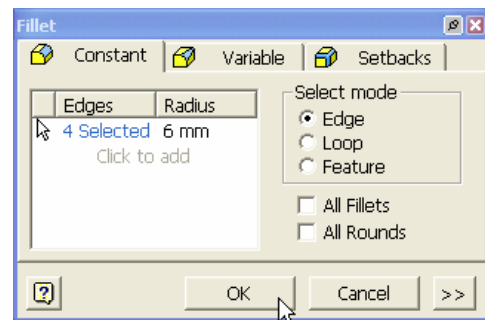


Figure 23

- Once again select **Fillet**.
- Set the fillet radius to 3 and choose loop for the select mode.

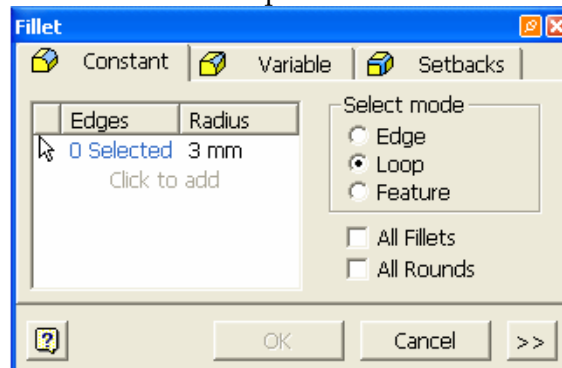


Figure 24

- Choose all the inner surfaces of the part.

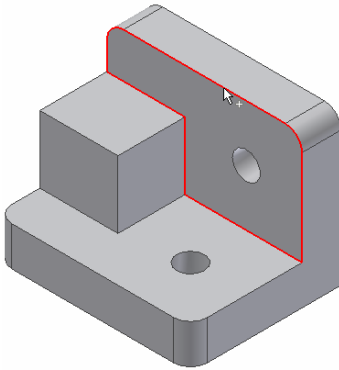


Figure 25

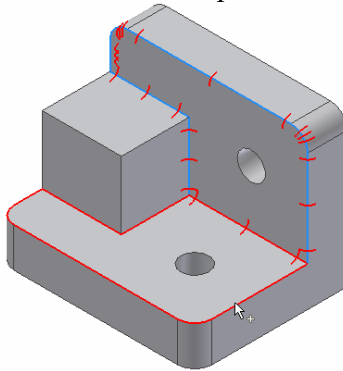


Figure 26

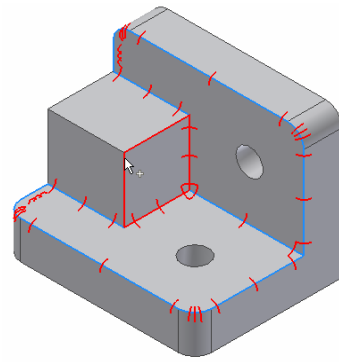


Figure 27

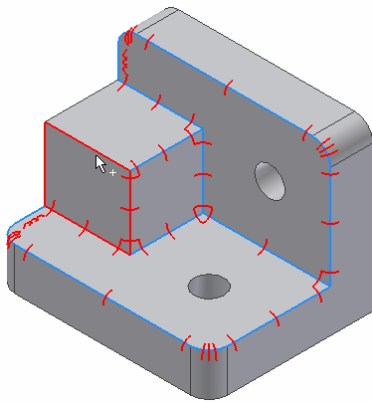


Figure 28

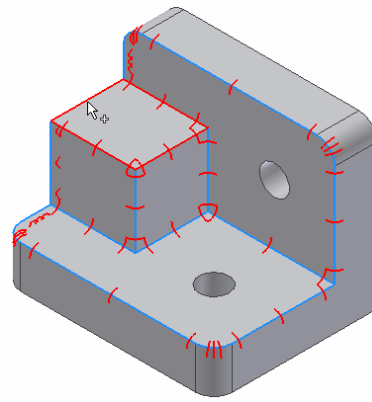


Figure 29

- Once all areas are selected click **OK** or press **Enter** on the keyboard.
- The finished part should look like Figure 30.

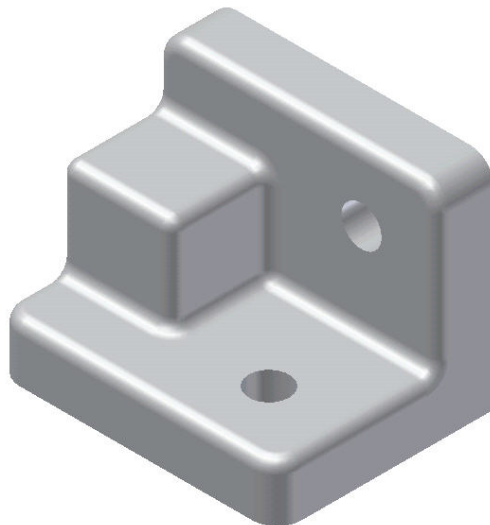


Figure 30

### Example 3

- Open the file “variable fillet.ipt”

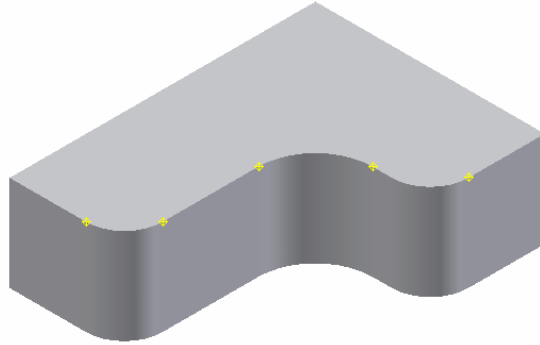


Figure 31

- Select **Fillet** from the Features Bar and click on the **Variable** tab

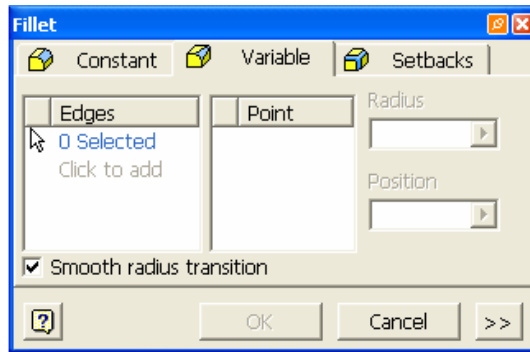


Figure 32

- Click on the top front edge of the part. Notice that the whole edge which includes the curves is selected.

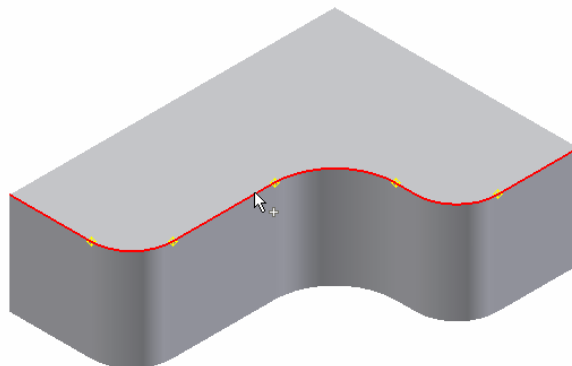


Figure 33

- Now that the edge has been selected, we can add points in which our fillets radius

can be varied. Notice that there are 5 work points along the selected edge. Refer to Chapter 7 Work Features. Select each point in order from 1 - 5 as shown in Figure 34.

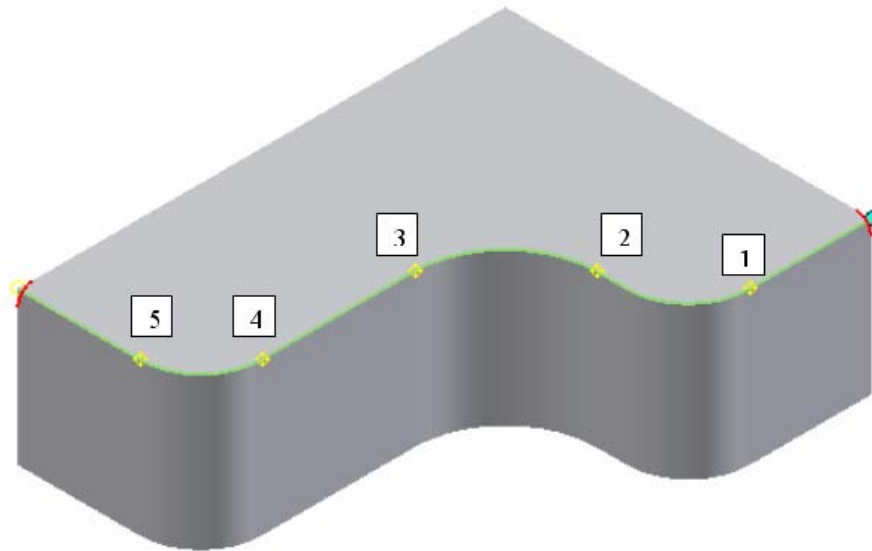


Figure 34

- In the dialog box click on each point and set the radius to the required value as shown in the following figures.

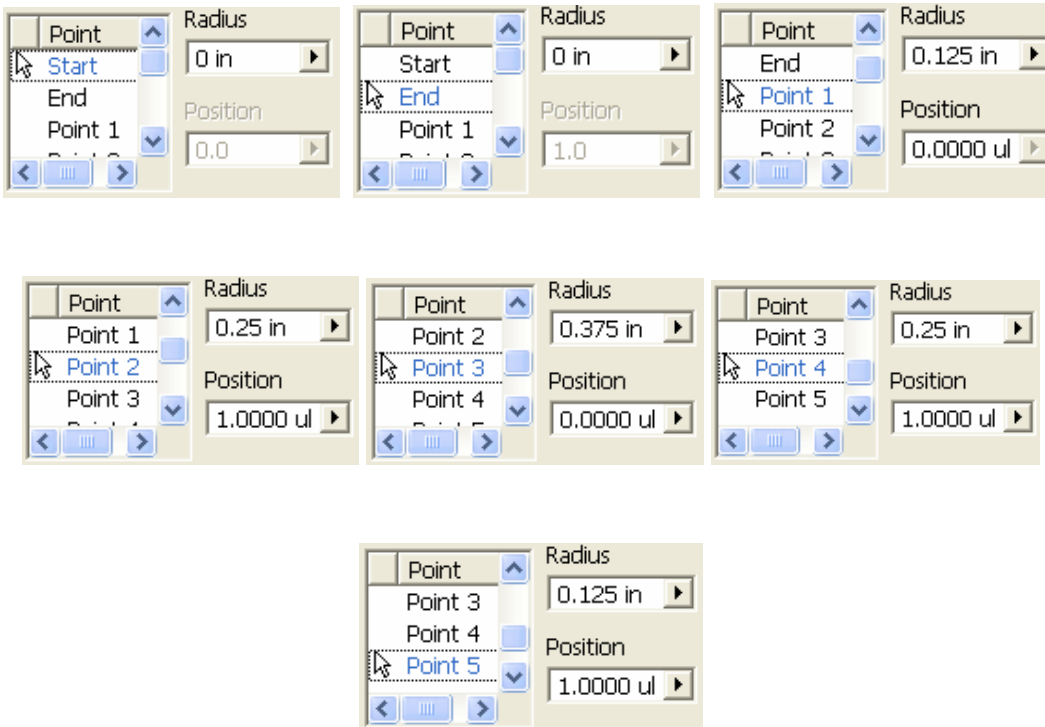
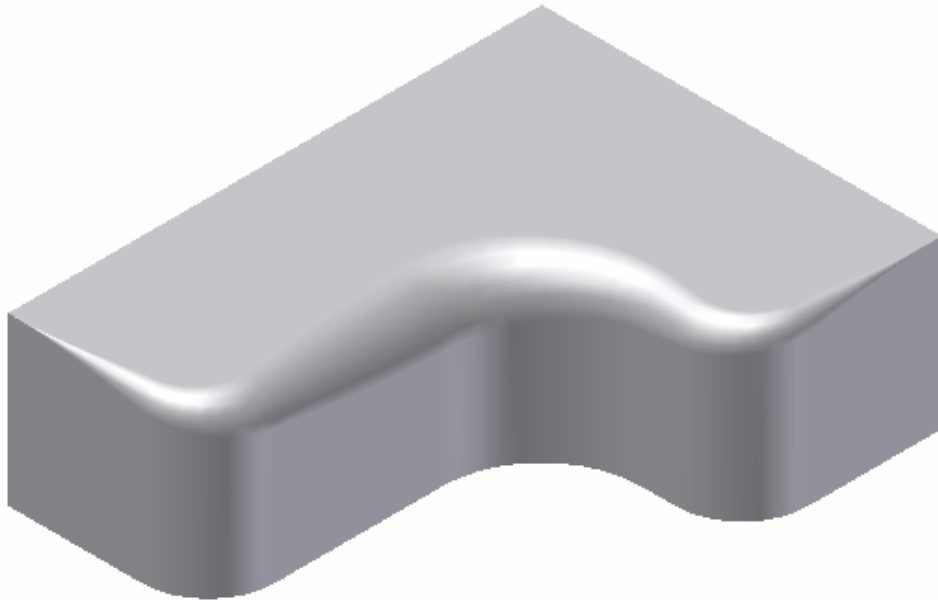


Figure 35

- ❑ Once all radii are set click **OK** or press **Enter** on the keyboard.
- ❑ The finished part should look like Figure 36.



**Figure 36**

## **Creating Chamfers**

Chamfers are created much in the same way as fillets. When creating chamfers the edge between two surfaces is selected and the chamfer will be placed. The three different styles of chamfers to choose from are Distance, Distance and Angle, and Two Distances.

### **Distance**

This allows you to set the distance offset from the edge selected and places a 45° chamfer on the edge.

### **Distance and Angle**

Distance and angle allows you to change the angle of the chamfer as well as setting the distance of offset. Once the two attributes are set, the edge to be chamfered must be selected as well as the surface in which the distance will be measured.

### **Two Distances**

This option allows the offset distances of each surface to be set. Once the edge to be chamfered is selected, Distance 1, which will be offset on the surface highlighted, must be set followed by Distance 2.

#### Example 4

- Open the file “Cover box.ipt”

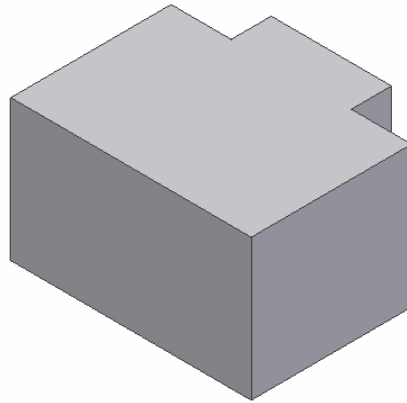
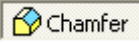


Figure 37

- Select **Chamfer** from the Features Bar. 
- Choose distance and set the value for the distance to .75in.

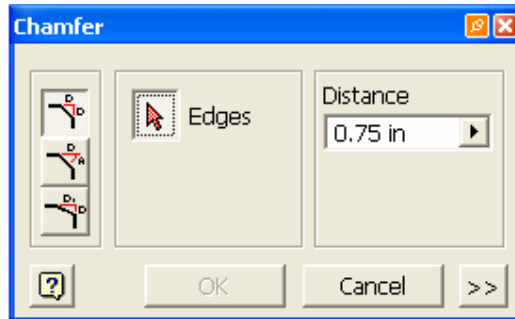


Figure 38

- Click on the 2 vertical edges of the front left surface to place the chamfer.

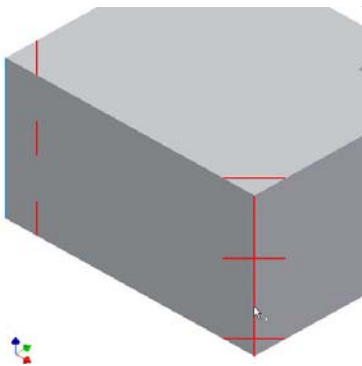


Figure 39

- Click **OK** or press **Enter** on the keyboard.

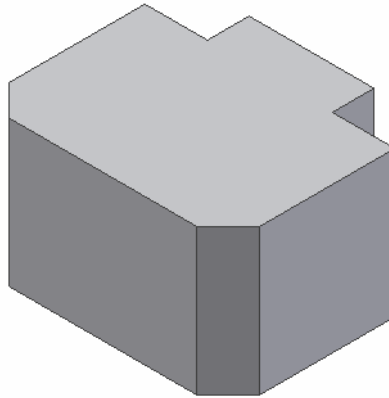


Figure 40

- ❑ Select **Chamfer** from the Features Bar again.
- ❑ Choose **Distance and Angle** and set the value for the distance to .25in and the angle to 60°.

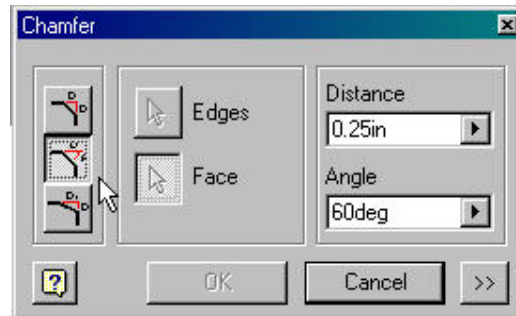


Figure 41

- ❑ The select **Face** tab should already be selected, select the top surface of the part and left click.

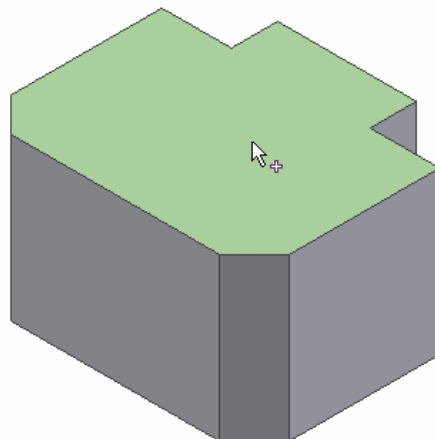


Figure 42

- ❑ This surface will be the direction of offset from each edge chosen that the distance will be measured.
- ❑ Now select the edges that will be chamfered as shown in Figure 43.

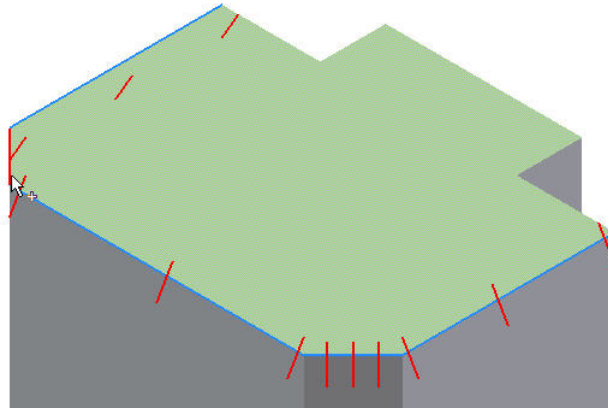


Figure 43

- Click OK or press Enter on the keyboard.

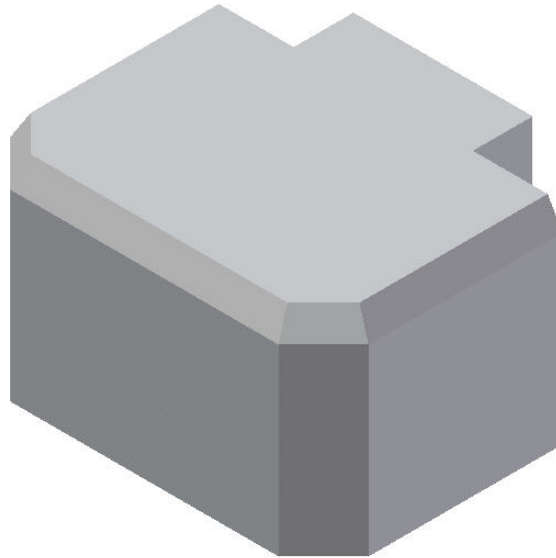


Figure 44

## SHELL

### Creating Shells

Shells are useful tools for creating parts that have thin walls in Inventor. The shell feature has the ability to have a part be hollow or to remove a portion of a face. Some parts may require different wall thicknesses in different sections of the part which can be accommodated with the shell command.



## Steps to creating basic shells

1. Have a base feature created that will have a portion hollowed out or shelled.
2. Select Shell from the Features Panel Bar.

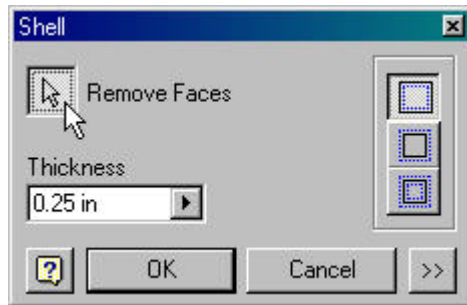


Figure 45

3. Set the wall thickness as desired and choose the faces that will be removed if necessary.
4. Once all surfaces are selected click OK or press enter and the part will be shelled.

### Example 5

- Open the file "Cover box.ipt"

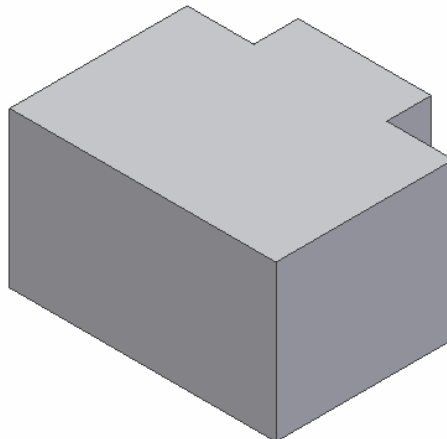


Figure 46

- Select **Shell** from the Part Features panel 
- Make sure that the Remove Faces button in the dialog box is depressed as shown in Figure 47.

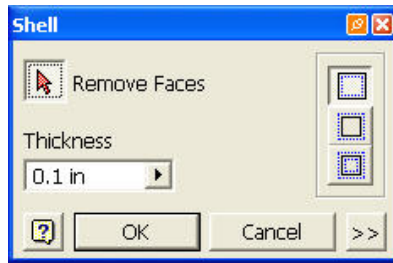


Figure 47

- Move the cursor over the top surface of the part and click to select. This will highlight that face in which material will be removed.

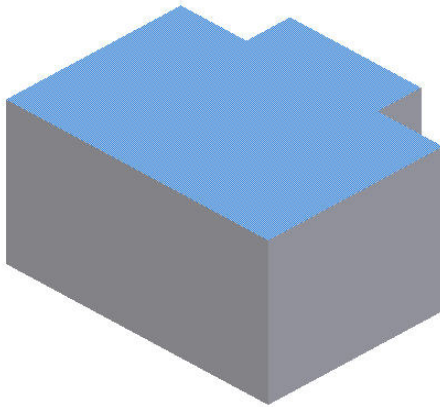


Figure 48

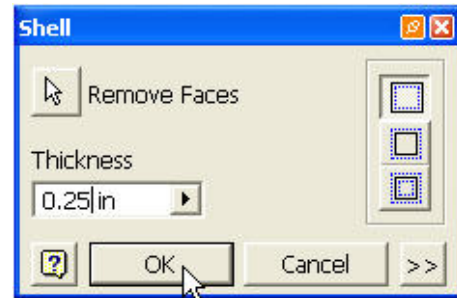


Figure 49

- Set the thickness of the walls to .25in as shown in Figure 49.
- Click **OK** or press **Enter** on the keyboard and the part will appear with walls rather than a solid, with the top face being open as shown in Figure 50.

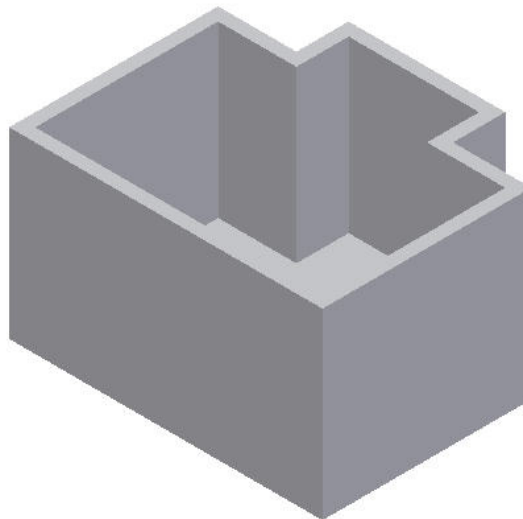


Figure 50

## PATTERNS

### Creating Patterns

The Pattern feature is useful when you have a feature of the part that has multiple instances. By using the pattern command, the selected feature will be copied and placed given the dimensions specified. There are two types of patterns to choose from in Inventor: *circular and rectangular*. Circular patterns will place the features in a pattern around a given axis, while rectangular patterns will place them in rows and or columns.

#### Steps to creating patterns

1. Have a base feature created in which the pattern will be placed on.
2. Choose the feature that will be patterned.
3. Set the direction/axis that the pattern will follow.
4. Set the count and spacing/angle the pattern will follow.

#### Example 6

- Open the file “circular pattern.ipt”

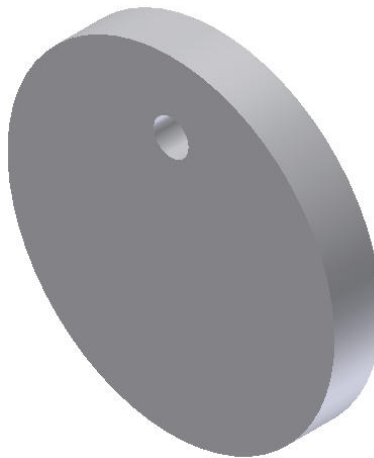


Figure 51

- Select **Circular Pattern** from the Part Features panel, this will open the pattern dialog box.

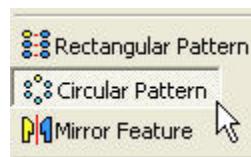


Figure 52

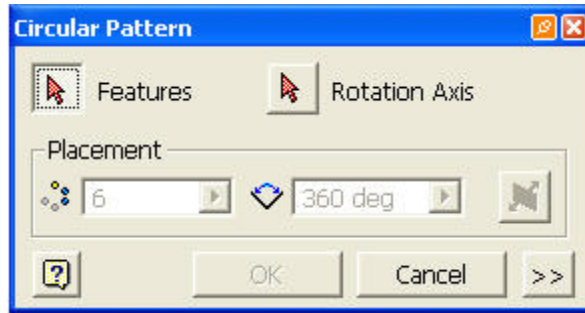


Figure 53

- The Features selection button should be depressed, if not click on it. Now click on the hole that is at the top of extrusion 1 or click on extrusion 2 in the Browser.

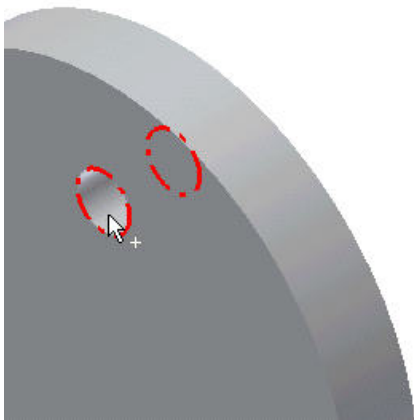


Figure 54

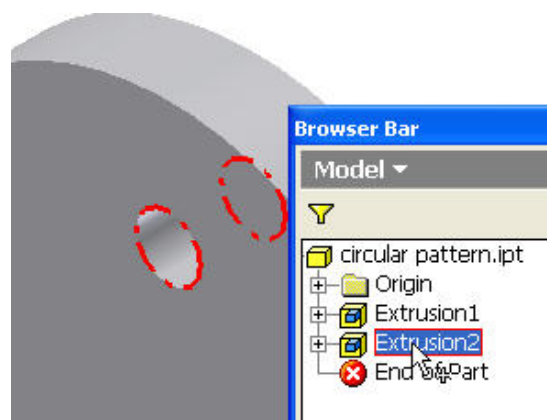


Figure 55

- Click on the rotation axis selection button to choose the axis that the pattern will be revolved around.

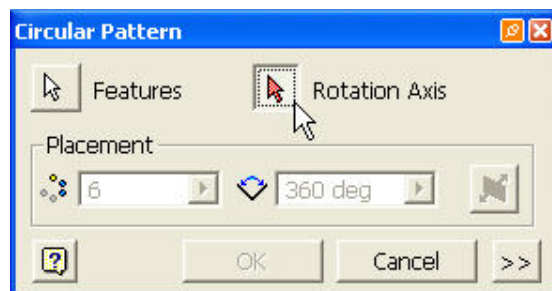


Figure 56

- We will use the given axis of extrusion 1. Select and click on the base feature as shown in Figure 57.

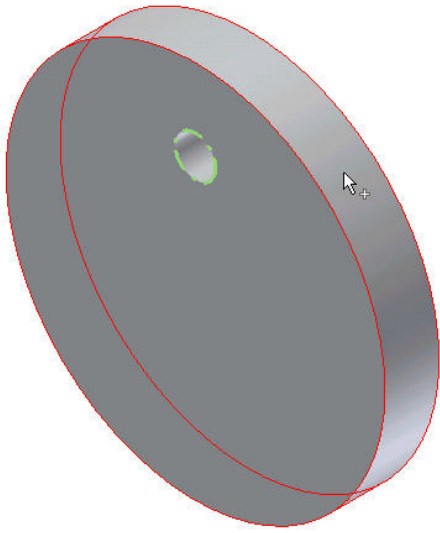


Figure 57

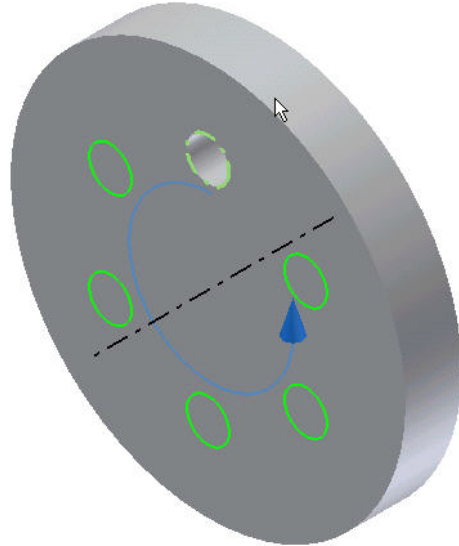


Figure 58

- Set the Count to 6 instances and the Angle to 360°.

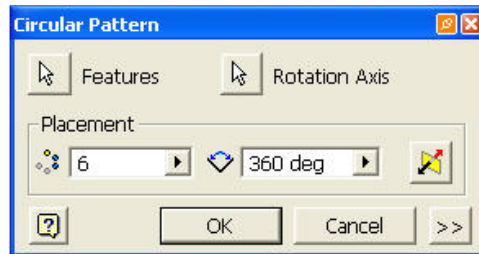


Figure 59

- Click **OK** or press **Enter** on the keyboard.

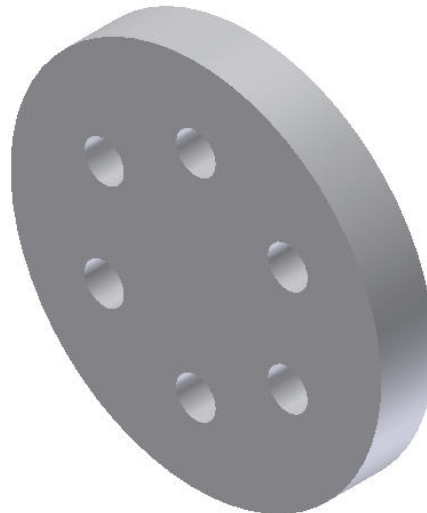


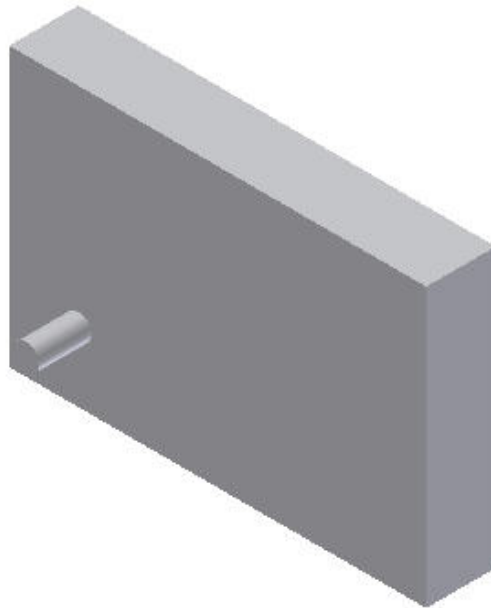
Figure 60

**NOTE**

*Try changing the Instances and Angle values by editing the feature in the Browser and see what happens to the part.*

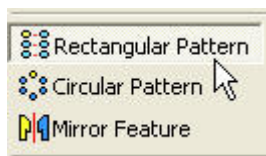
**Example 7**

- Open the file “rectangular pattern.ipt”

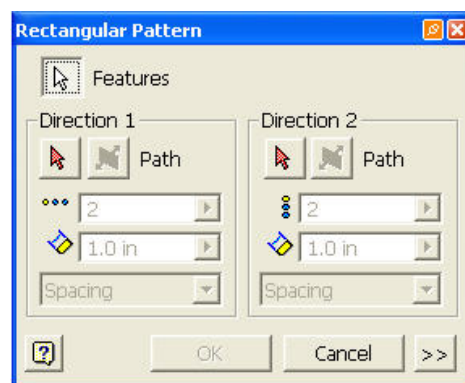


**Figure 61**

- Select **Rectangular Pattern** from the Features Bar, this will open the pattern dialog box.



**Figure 62**



**Figure 63**

- Click on the cylinder that is at the bottom left of extrusion 1 or click on extrusion 2 in the Browser.

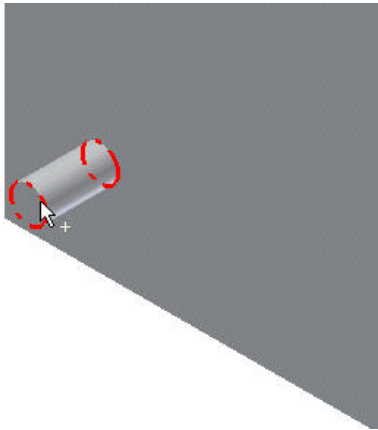


Figure 64

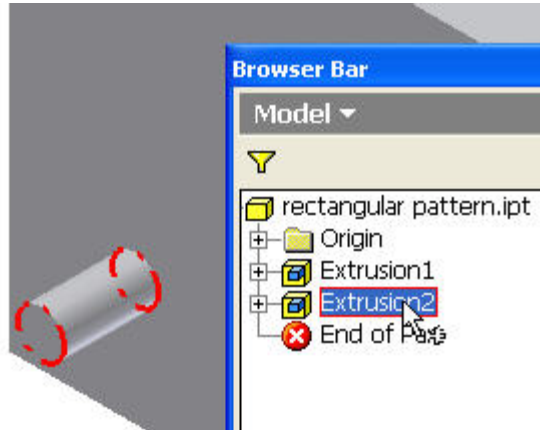


Figure 65

- Now that the feature to be patterned is selected, click on the Direction 1 selection button and choose the bottom edge of the base feature.

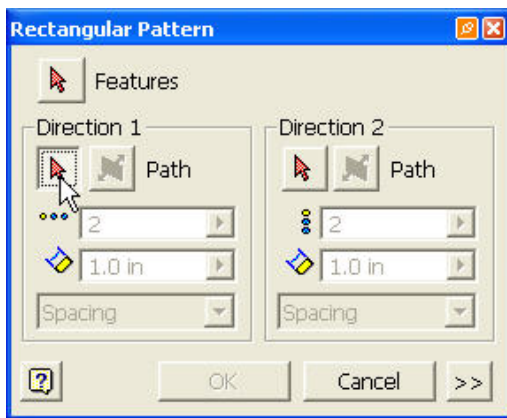


Figure 66

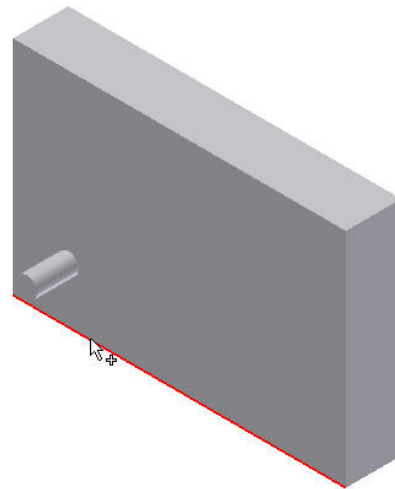


Figure 67

- Notice the direction that the pattern is going to be copied. To change the direction click on the Path button to switch the direction of pattern to the right as shown in Figure 69.

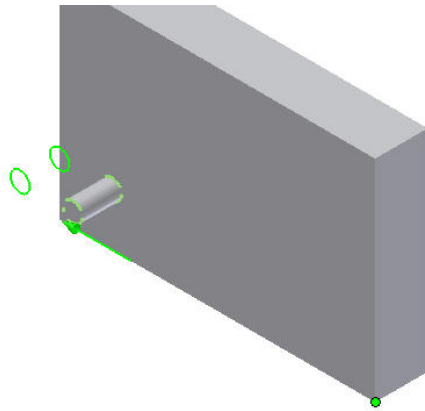


Figure 68

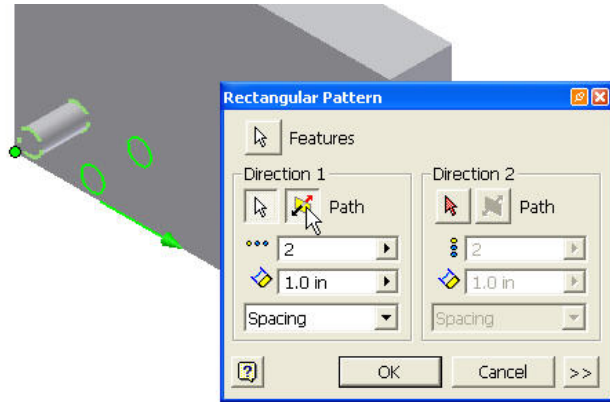


Figure 69

- Set the Count to 5 and the Spacing to 1.0in.

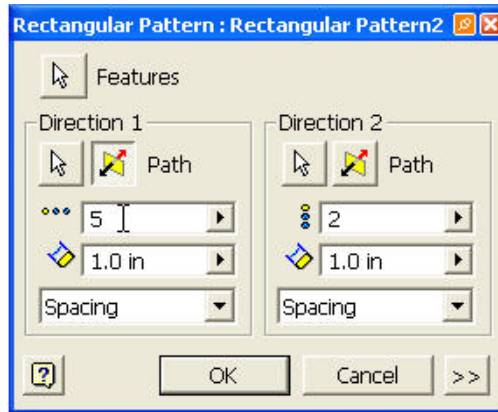


Figure 70

- Click on the Direction 2 selection button and choose the left vertical edge of the base feature.

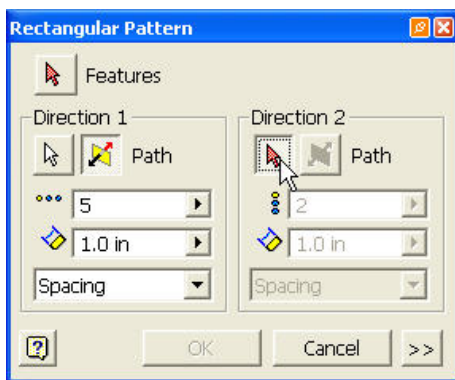


Figure 71

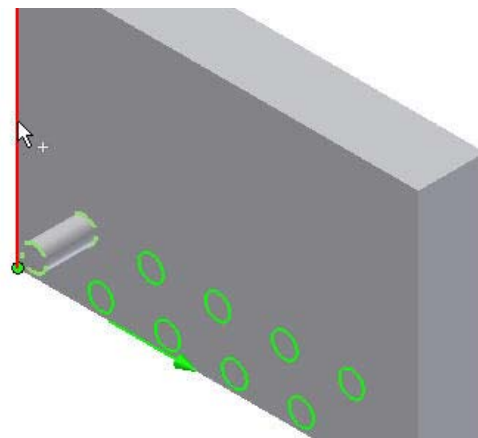


Figure 72



- Set the Count to 3 and the Spacing to 1.0in. Click **OK** or press **Enter** on the keyboard.

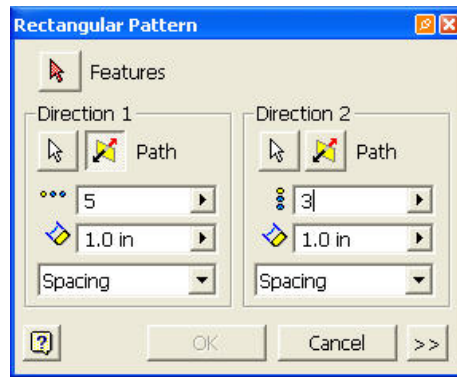


Figure 73

- The finished part with both patterns added should look like Figure 74.

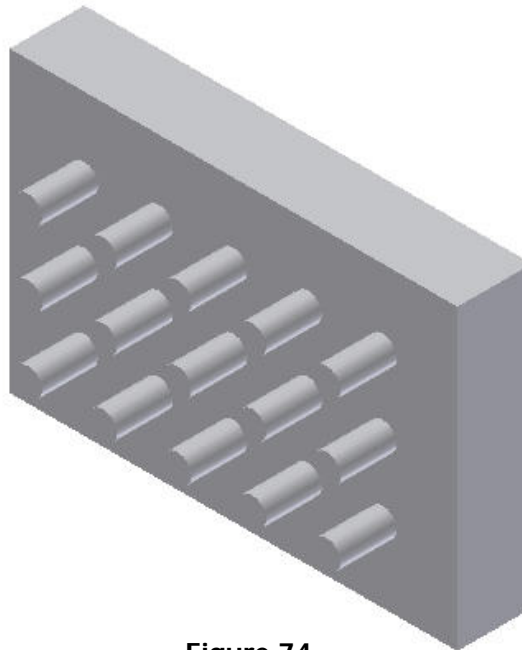


Figure 74

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# CHAPTER 7

## TUTORIAL

# Advanced Part Features

### Learning Objectives

---

*After Completing this chapter, you will be able to:*

- Create the following features:
  - Sweeps
  - Lofts
  - Coils
  - Threads
  - Ribs & Webs
  - Face Draft
  - Split Feature
  - Slice Graphics
- 

Advanced part features will allow the creation of more complex objects or parts that could not be created by basic boolean operations. The use of these techniques will assist in creating complex transitions and shapes.

### SWEEPS

#### Creating Sweeps

Using the sweep feature is a more efficient way of creating piping, tubing and other complex shapes that follow a path.

Steps to creating sweeps

1. Create a sketch of the path that the sketched feature will follow.
2. Create a sketched feature that is a cross section.
3. Use the sweep command to create the part.

#### NOTE

*Make sure that the path forms a continuous line.*

## TUTORIAL 1

- ❑ Open a new Standard. ipt
- ❑ Make the XZ and YZ planes visible.
- ❑ Create the sketch of the paperclip. (Figure 1)

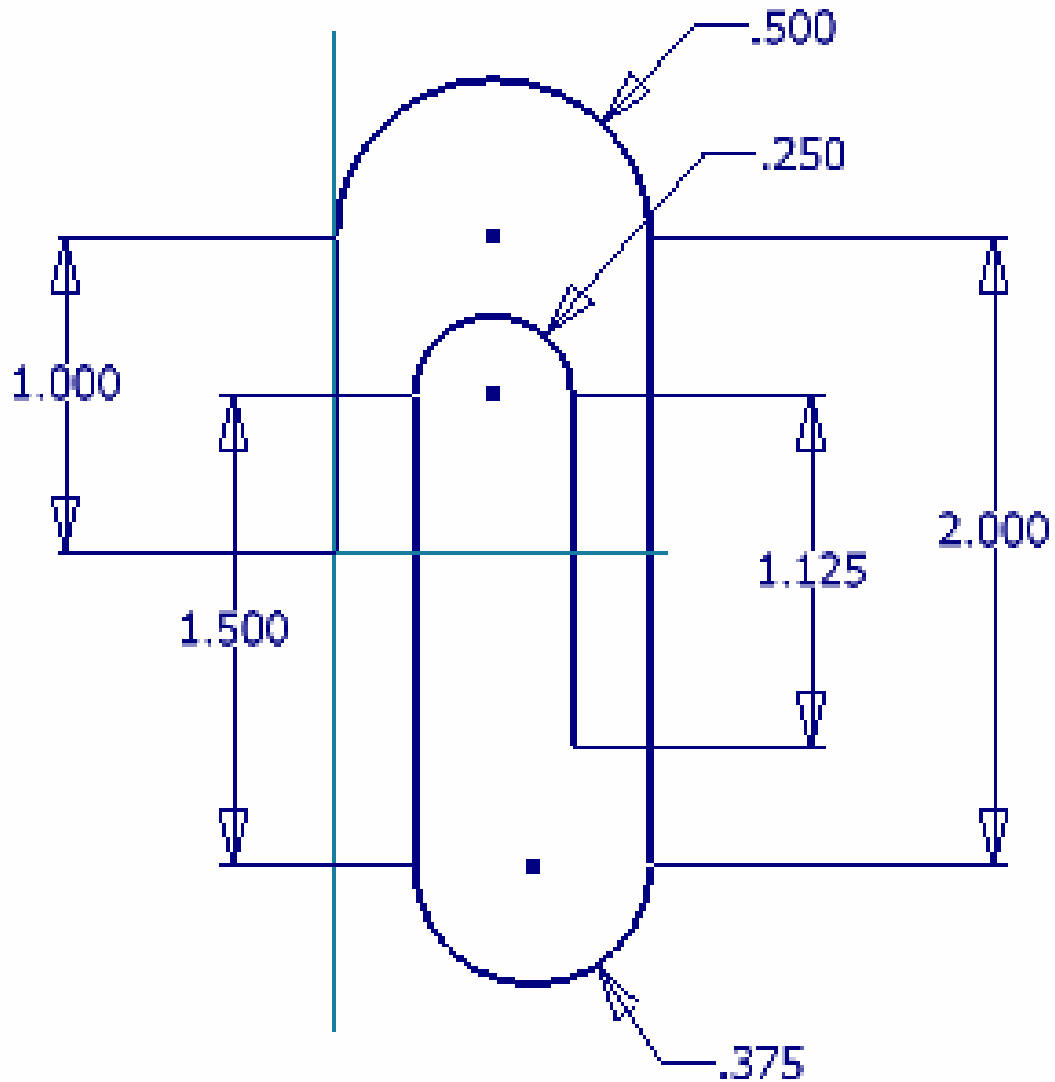


Figure 1

- ❑ Once you have created the sketch click **Return**
- ❑ Change your view to Isometric
- ❑ Select the XZ plane as your new sketch plane as shown in Figure 2.

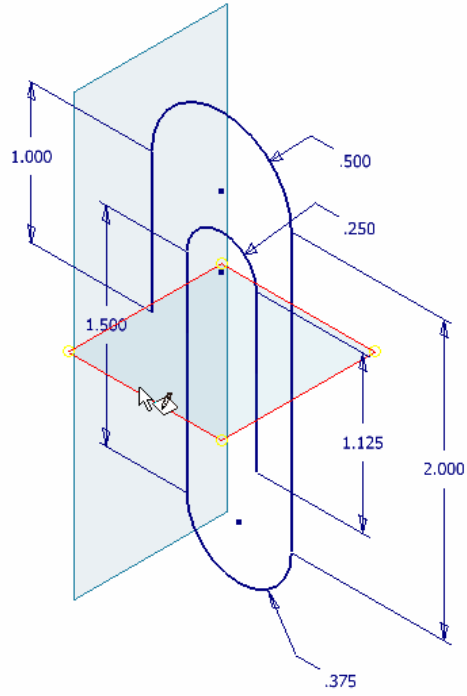


Figure 2

- Click on Project Geometry and select the left vertical line. (Figure 3a)
  - Draw a diameter .125 circle at the projected point. (Figure 3b)

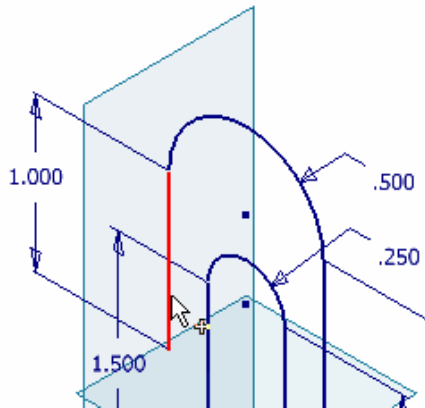


Figure 3a

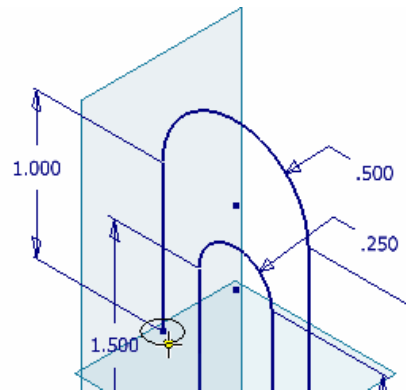


Figure 3b

- Once you have finished the sketch click **Return**
- Click on the Sweep button in the Part Features menu bar

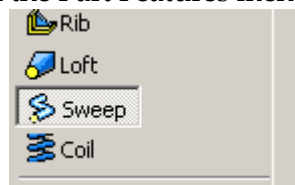


Figure 4

- The profile should already be highlighted and the path button selected

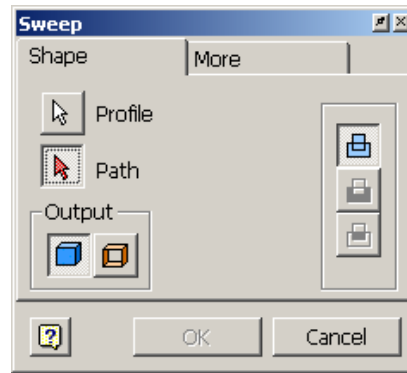


Figure 5

- Click on the line that was created in Sketch1. (Figure 6)

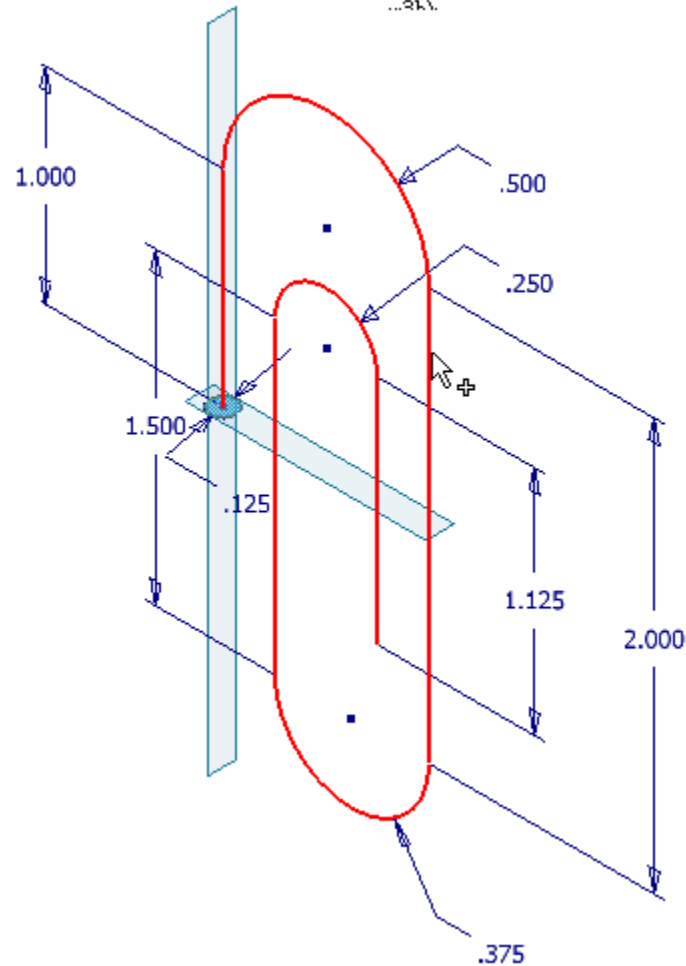


Figure 6

- Click **OK**.



**Figure 7**

## COIL

### Creating Coils

Coils are a useful option for creating such things as springs, coils, and other helical shapes. They can be used to create solids or with the cut option to create cuts in a part. There are three different tabs available in the coil dialog box.

#### Coil Shape

This tab allows you to choose the profile to be coiled as well as the direction and axis the coil will follow.

#### Coil Size

In the coil size tab there are options for adjusting the type of parameters for creating the coil. Depending on what parameters are chosen for creation of the coil, there are places for setting the pitch, height, revolution, and taper.

#### Coil Ends

This tab allows the designer to determine the conditions of the start and end of the coil feature allow for a natural coil or one with flat ends.

### Steps to creating a coil

1. Create an unconsumed sketch, or profile, that is a cross section of the coil feature that will be created.
2. Have an axis in which the coil will be rotated around.
3. Use the coil option to create the part.

## TUTORIAL 2

- ❑ Open a Standard(mm).ipt
- ❑ Make the base planes visible.
- ❑ Create a 5mm circle below the XZ Plane.

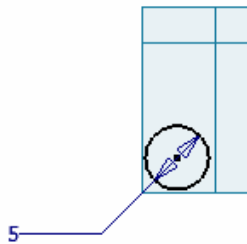


Figure 8

- ❑ Use Project Geometry, and project the XZ plane to be used as the axis.
- ❑ Dimension the center of the circle to projected XZ plane. Give it a dimension of 15.

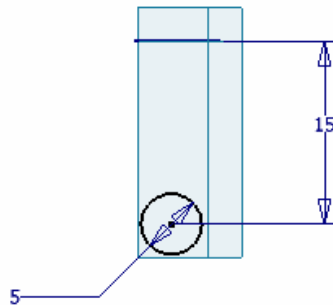
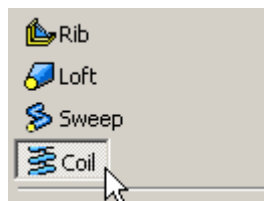


Figure 9

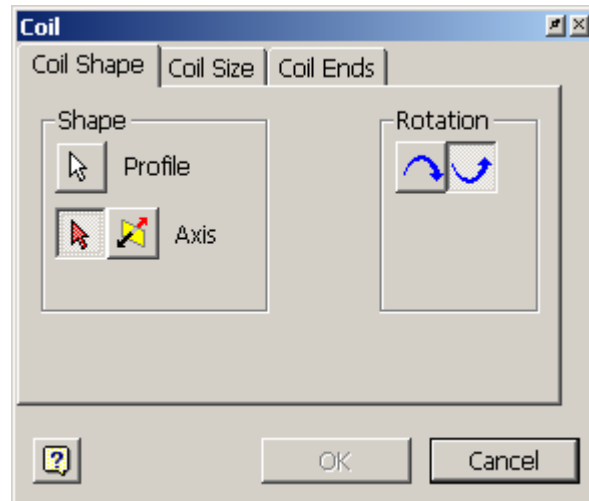
- ❑ Click Return.
- ❑ Select Coil from the Part Feature menu.





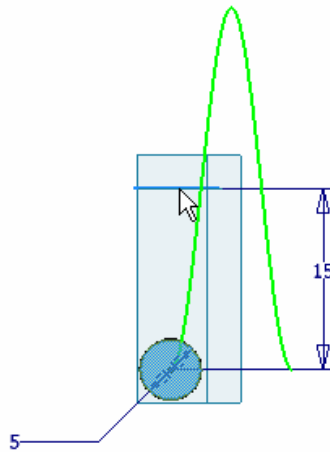
**Figure 10**

- The Coil dialog box will open and the circle should already be selected as the profile. Notice that the Axis button is selected.



**Figure 11**

- Click on the XZ plane as the axis of revolution.



**Figure 12**

- Select the **Coil Size** tab in the dialog box and change the settings as shown in Figure 13.

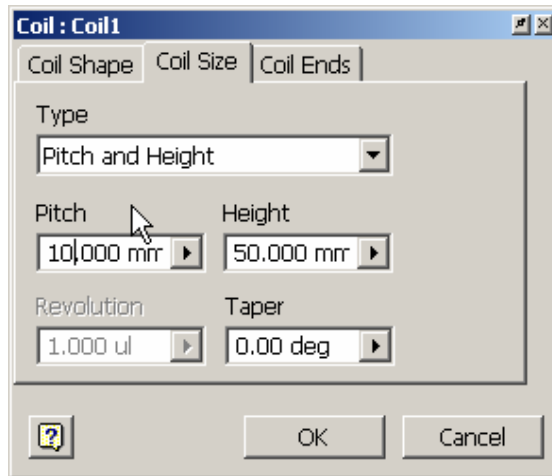


Figure 13

- Next select the **Coil Ends** tab and set the Start to Flat.

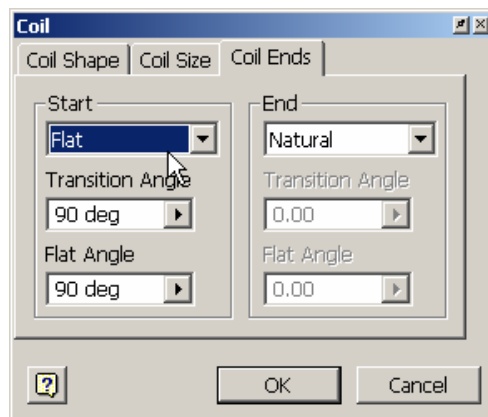


Figure 14

- Click **OK** and the result should look like Figure 15.

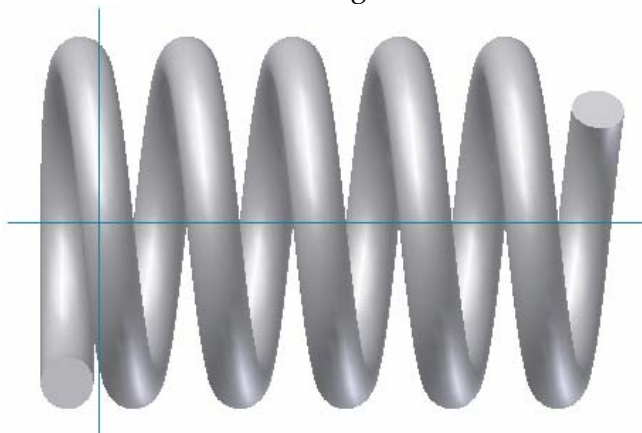


Figure 15

## THREADS

### Creating Threads

The thread feature in Inventor can be used to create a graphical representation of both internal and external threads. It does not actually remove or add material, it simply places a graphical representation to give the appearance of threads as well as allows for thread notes in the drawing mode.

Steps to creating threads

1. Have a base cylindrical feature with the major diameter dimension created that will have the threads placed on it.
2. Select Threads from the Features Panel Bar.
3. Select the surface/edge threads will be placed and set the data necessary for creation.

### TUTORIAL 3

- ❑ Open part "Thread.ipt"
- ❑ Click on Thread in the Feature menu.

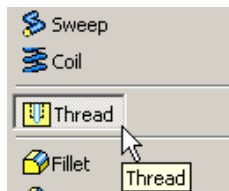


Figure 16

- ❑ This will open Thread Feature dialog box.

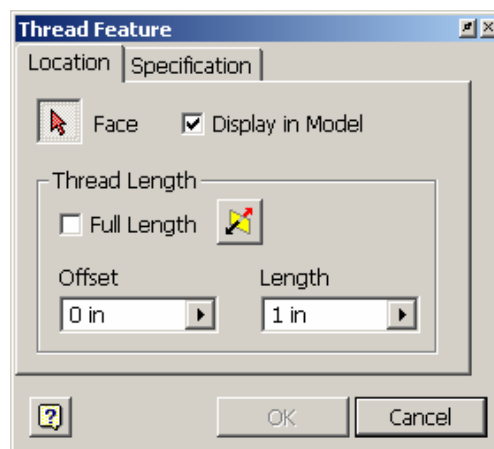


Figure 17

- Click on the cylinder surface at the front right edge of the bolt. Make sure that Full Length is not checked in the dialog box.

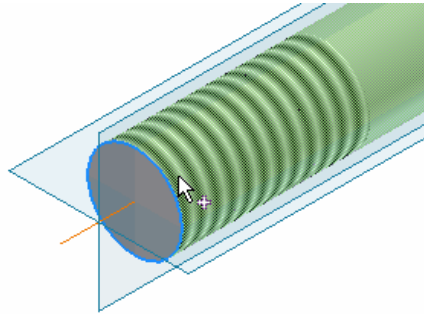


Figure 18

- Set the Length to 2.5 inches.

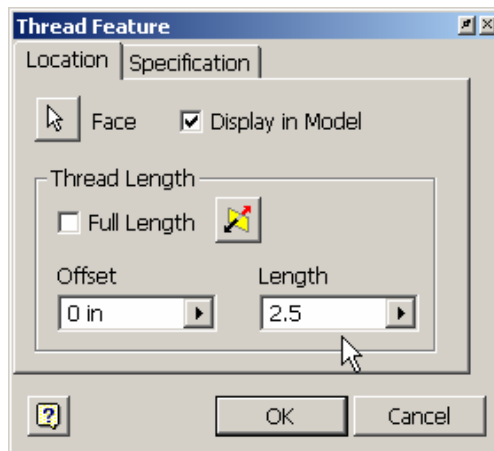


Figure 19

- Click on the **Specifications** tab to set the thread type.

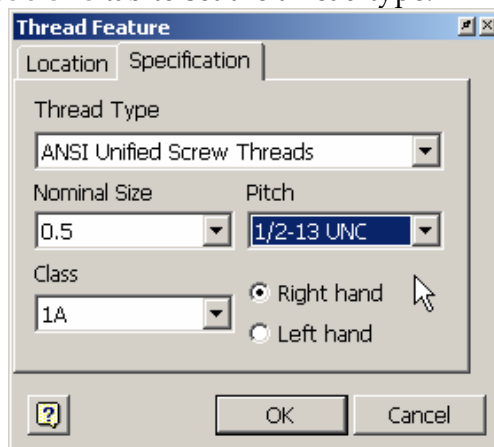


Figure 20

- Set the Pitch to 1/2-13 UNC. Click **OK** and the threads should appear as Figure 21.

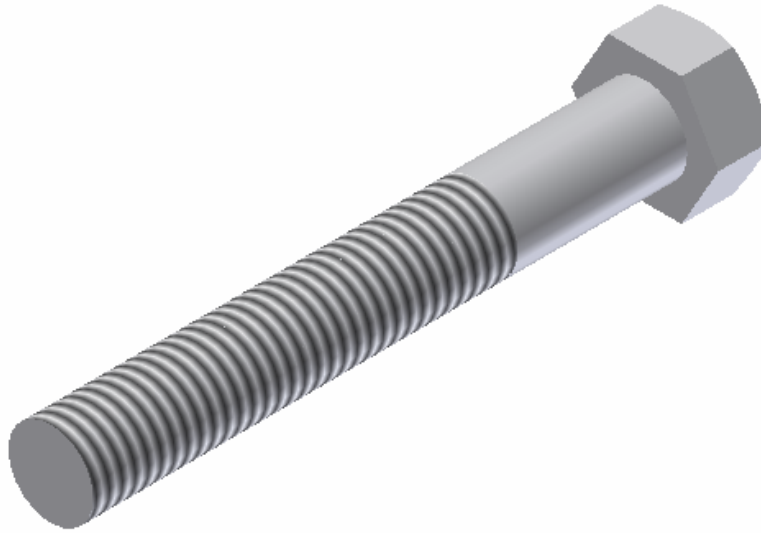


Figure 21

## RIBS & WEBS

### Creating Ribs & Webs

Ribs and webs use an unconsumed sketch to create features that are used as reinforcement in molded parts. By using a single line and the rib feature, Inventor will create a closed ribbed feature or an open webbed feature set to the specifications determined in the Rib Dialog box.

Steps to creating ribs

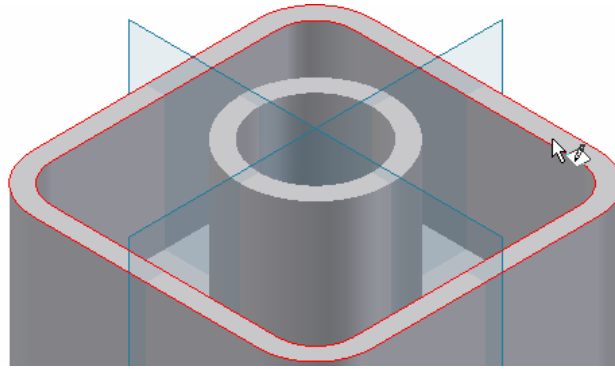
1. Have a base feature created in which the rib will be placed on.
2. Sketch the profile of the rib or web.
3. Select the Rib feature button in the Feature menu bar.

Click on the profile.

4. Set the attributes in the dialog box as desired and click OK.

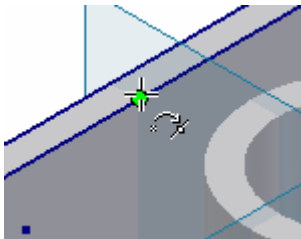
## TUTORIAL 4

- Open the file “base.ipt”
- Select the top surface of the base part as the sketch plane.

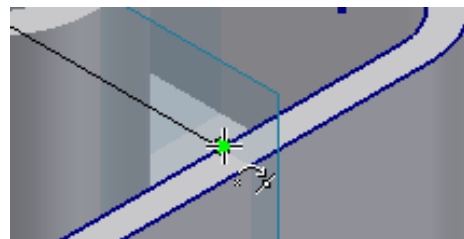


**Figure 22**

- Sketch a line from the mid point of the top left projected edge to the mid point of the bottom right projected edge.

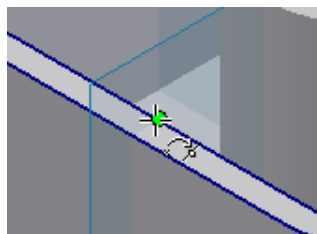


**Figure 23**

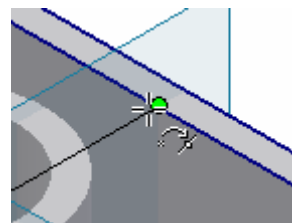


**Figure 24**

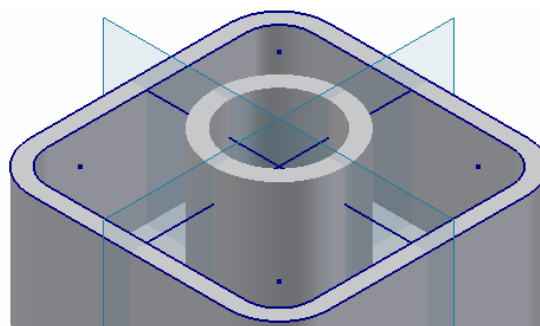
- Create another line from the mid point of the bottom left projected edge to the mid point of the top right projected edge.



**Figure 25**



**Figure 26**



**Figure 27**

- ❑ Click **Return**.
- ❑ Click on the **Rib** button in the Part Feature menu bar.



Figure 28

- ❑ In the Rib dialog box the profile button should already be selected, if not then select the profile button.

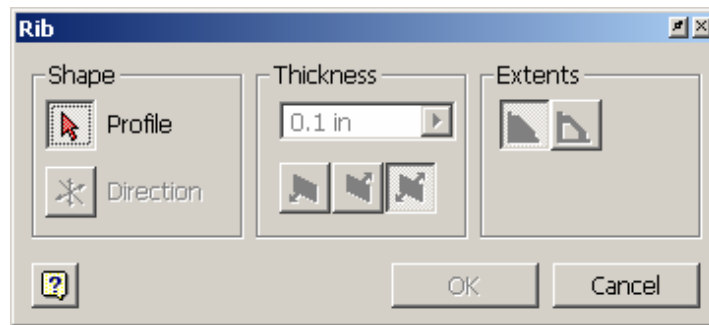


Figure 29

- ❑ Click on the two lines just created as the profile.

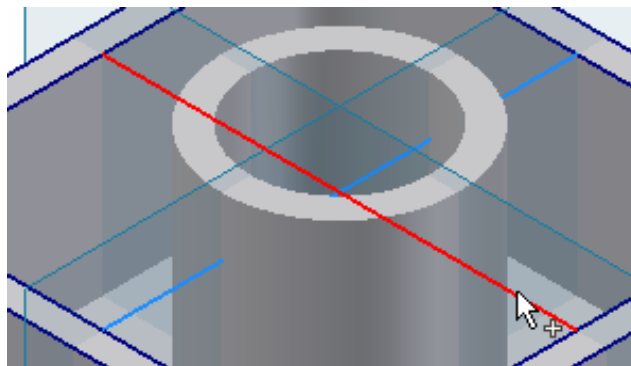


Figure 30

- Set the thickness to .25in. Then click on the Direction button.

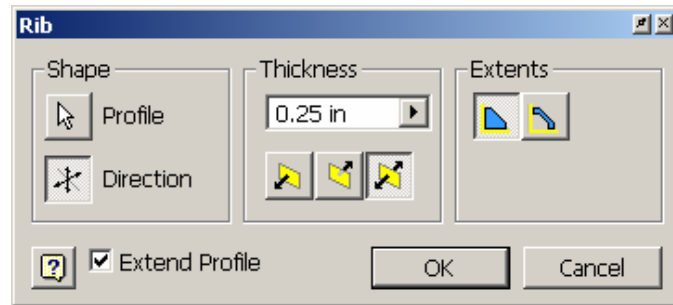


Figure 31

- Set the direction to be facing towards the interior of the part.

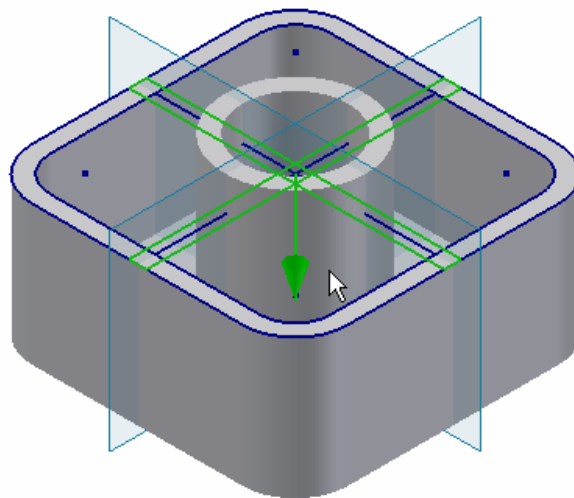


Figure 32

- Click on **OK**. Notice that the rib did not go through the hole portion of the part.

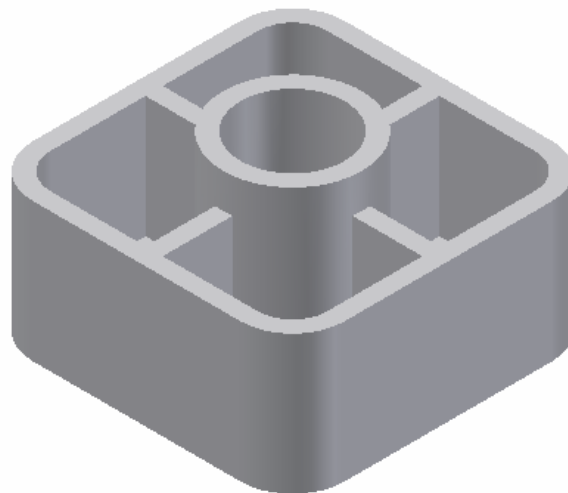


Figure 33



- ❑ Next right click on rib1 in the browser and edit feature.
- ❑ Change the Extents to finite.

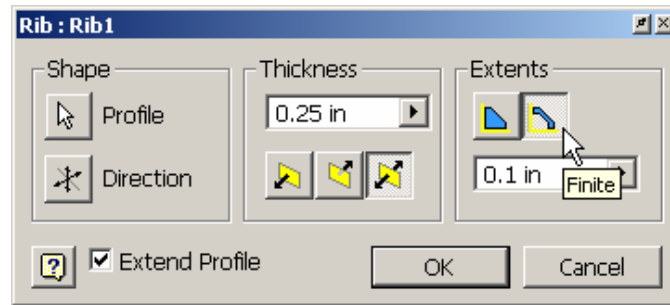


Figure 34

- ❑ A new attribute box opens inside the Extents section of the dialog box. Change this dimension to .25 inches.

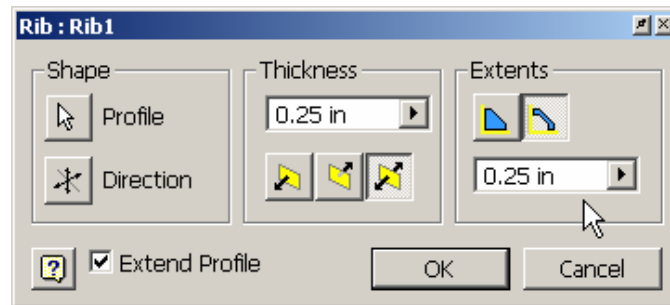


Figure 35

- ❑ Click **OK**. Notice the change from rib to web.

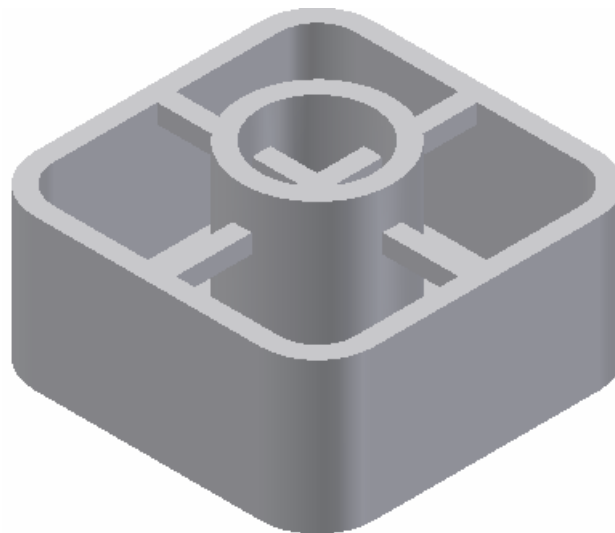


Figure 36

## FACE DRAFT

### Creating Face Drafts

Face draft adds an angled/sloped surface to any face of a part either internal or external. This is commonly used for molded or cast parts.

Steps to creating a face draft

1. Have a base part created in which a face/surface will be sloped.
2. Choose the face draft command in the part feature menu.
3. Select the pull direction that the mold will be removed from the part.
4. Select the face/faces that will have the draft applied to and set the angle of draft.

### TUTORIAL 5

- Open the file “draft.ipt”

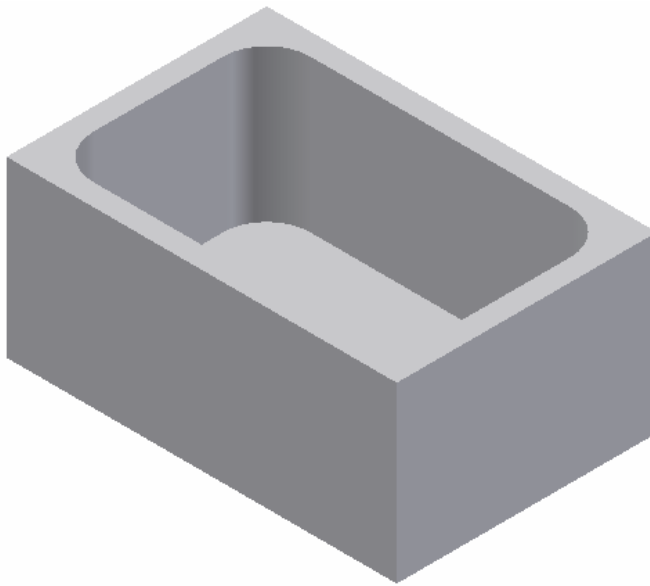


Figure 37

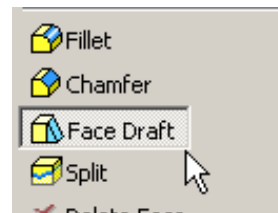


Figure 38

- Select **Face Draft** from the Features Bar.
- The Pull direction button should be selected. (Figure 39)

- Select the interior bottom surface.

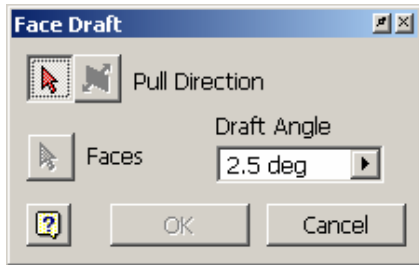


Figure 39

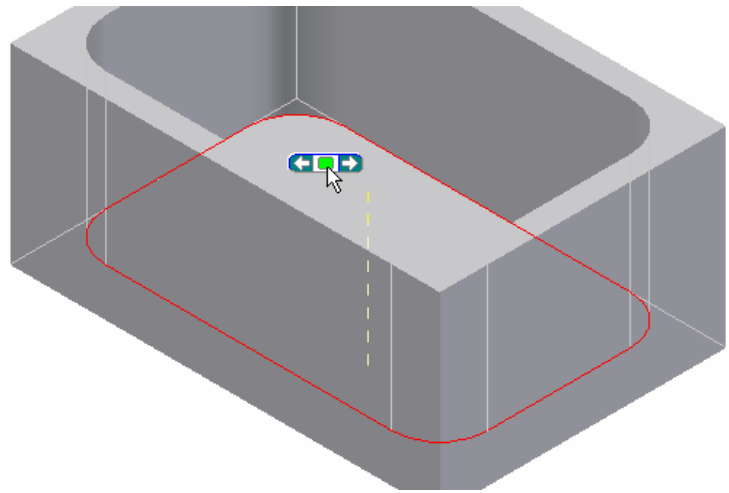


Figure 40

- Next select the interior faces that will be drafted, since there are fillets on the corners it will automatically choose all the faces. Select the top left edge where the interior vertical faces meet the top face.

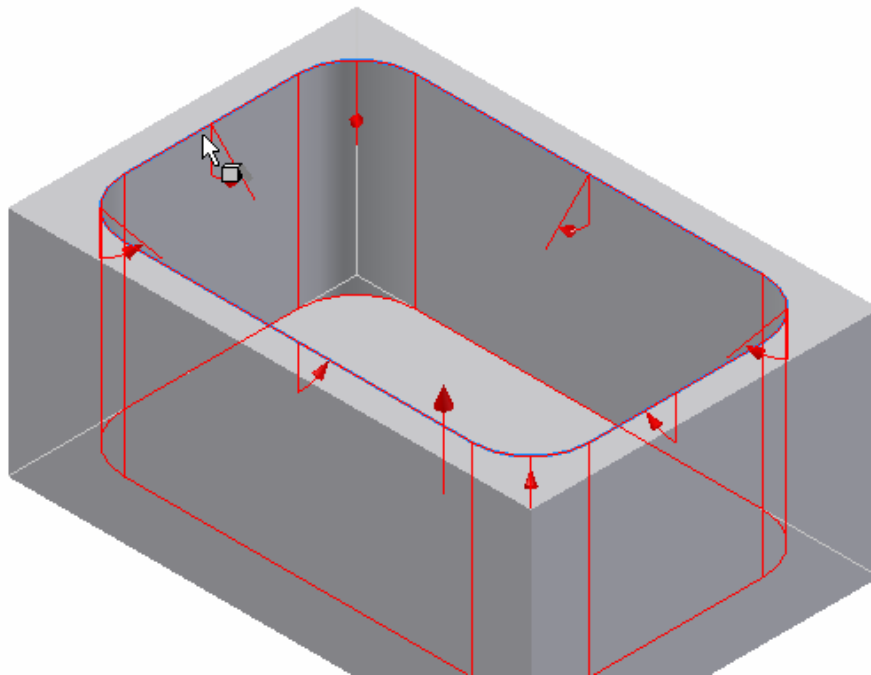


Figure 41

- Set the Draft Angle to 5° and press **Enter** or click **OK**.

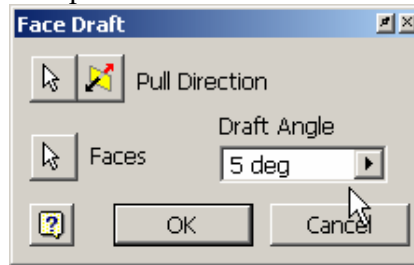


Figure 42

- Change the part display to **Hidden Edge Display**. Notice the slope that has been applied to the faces.

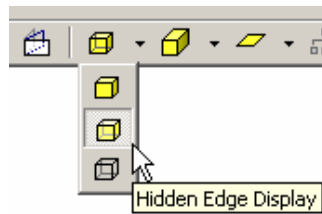


Figure 43

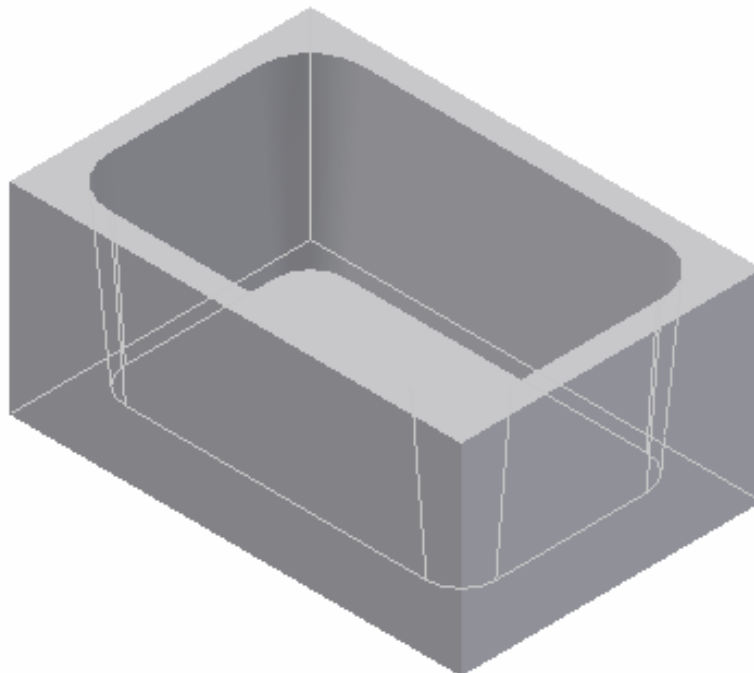


Figure 44

## SPLIT FEATURE

### Creating Split Features

Cutting and removing faces and parts can be performed using the split feature tool. By using a sketched shape, surface, or workplane to use as a cutting edge, the entire part can be split and have a portion removed.

Steps to creating split feature

1. Have a base part created.
2. Either select a created sketch geometry, a workplane, or use an existing surface to be used as the cutting edge.
3. Select whether the part or the face will be split.
4. If the part is to be split select the direction that the material will be removed from.

### TUTORIAL 6

- ❑ Open a new "Standard.ipt"
- ❑ Create the following sketch.

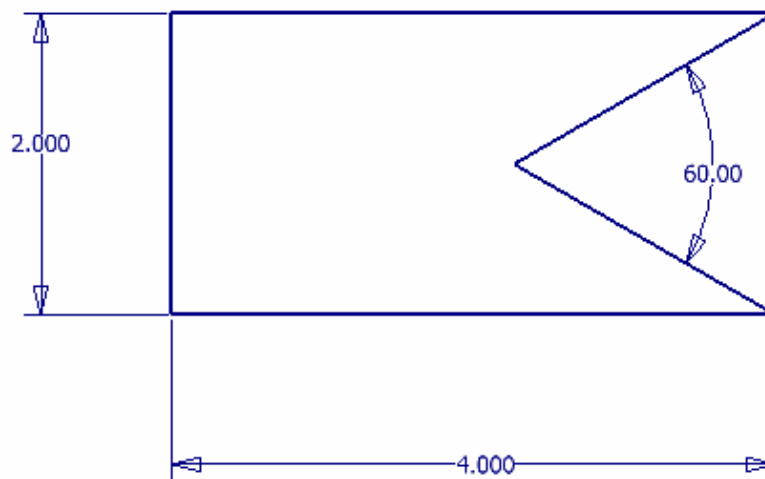


Figure 45

- ❑ Extrude the profile 2 inches..
- ❑ View the part in isometric and select the top surface as the new sketch plane. (Figure 46)

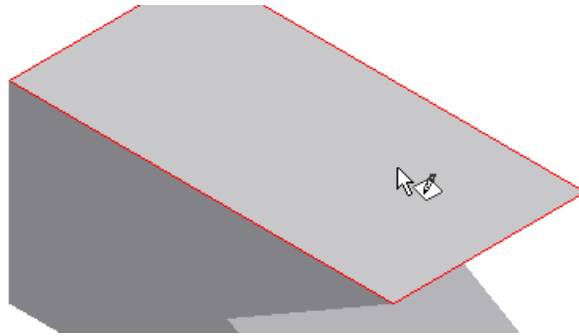


Figure 46

- Sketch a three point arc from corner to corner, giving the arc a dimension of 6.5 inches.

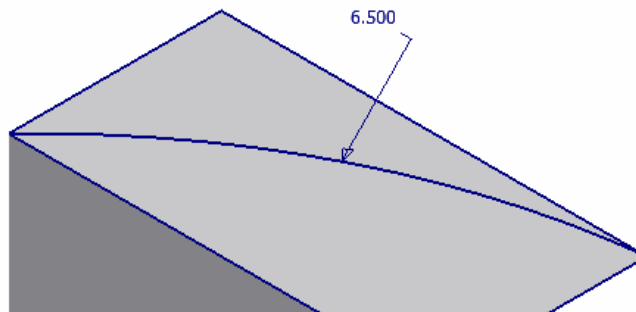


Figure 47

- Select **Split** from the Part Features menu bar.

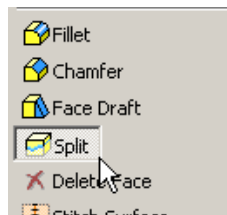


Figure 48

- In the Split dialog box click on Split Part.

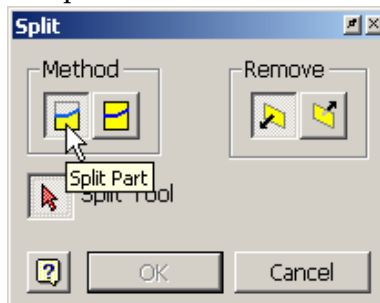


Figure 49

- Select the Split Tool button and click on the arc.

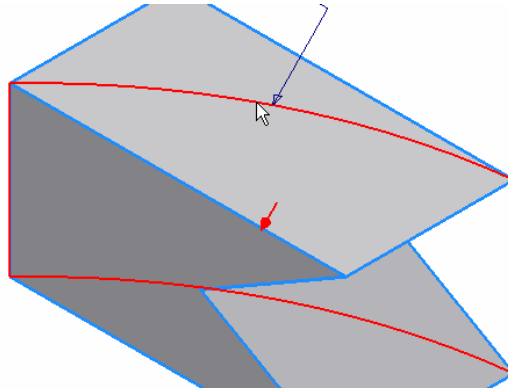


Figure 50

- If the arrow is not pointing toward you, click on the reverse direction button that is in the **Remove** section of the dialog box.

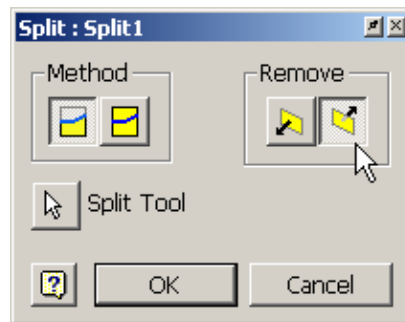


Figure 51

- Click **OK** or press **Enter**.

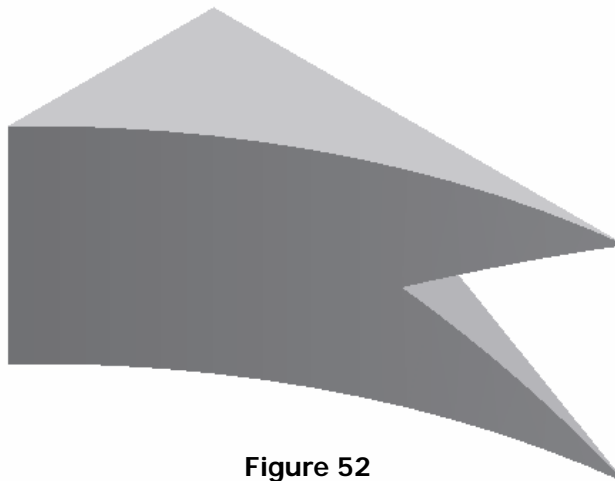


Figure 52

## SLICE GRAPHICS

### Using Slice Graphics

Some parts are rather complex and require sketching on planes or surfaces that are difficult to see. For these problems the use of Slice Graphics can be handy. This will temporarily remove part material so that you can view the sketch plane to work easier.

Steps for slicing graphics

1. Have a base part created in which the sketch plane goes through.
2. Using the View pull down menu or by right clicking the mouse in the work area select Slice Graphics.

### TUTORIAL 7

- ❑ Open the file "swing arm.ipt"
- ❑ Create a work axis through the counterbores in the front and the back of the part.

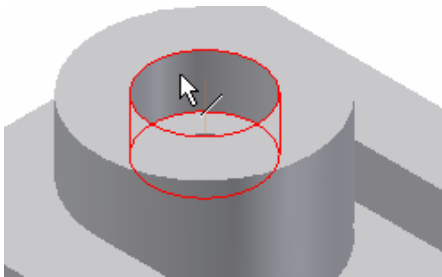


Figure 53

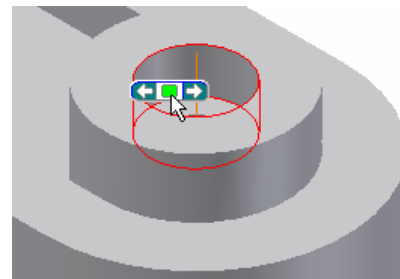


Figure 54

- ❑ Create a new workplane bisecting the part through the new workaxes.

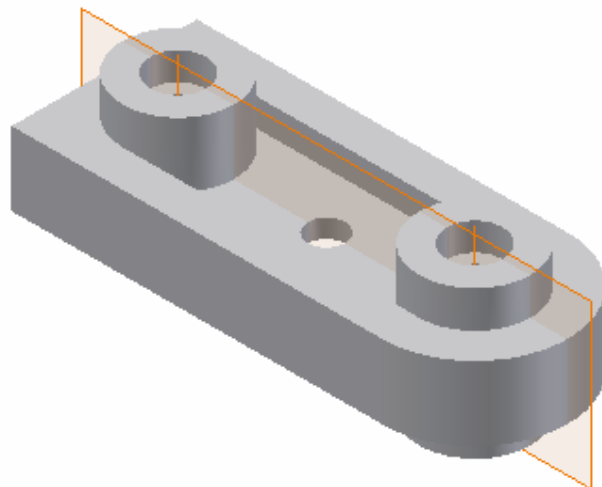


Figure 55



- Choose the new workplane as the sketch.

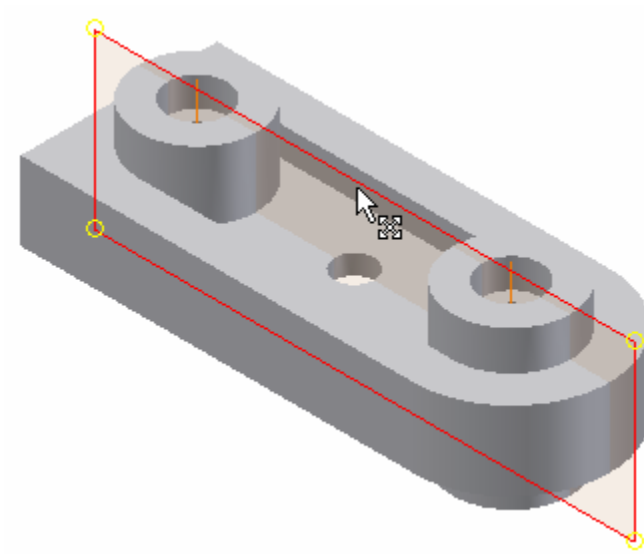


Figure 56

- Click the right mouse button in the work area and select Slice Graphics.

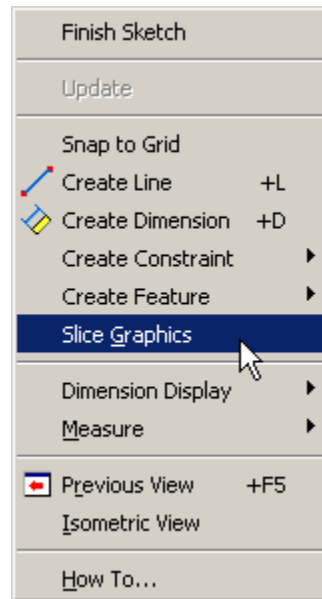
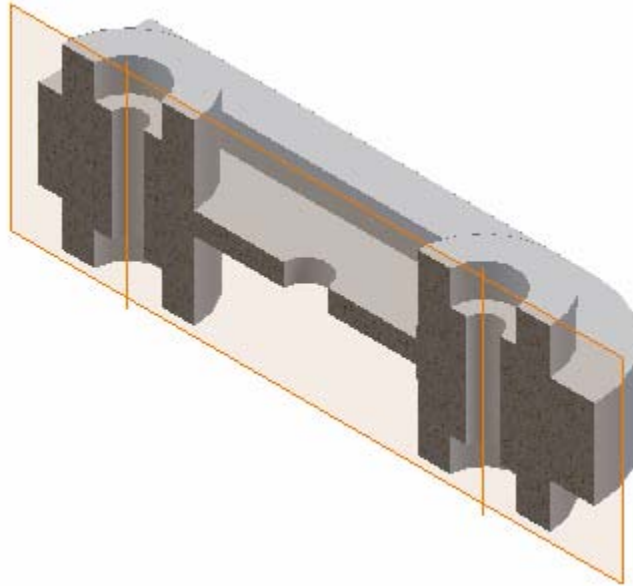


Figure 57

- Notice that we can now view inside the object along the sketch plane we are using as shown in figure 58.



**Figure 58**

- We can now project geometry onto our current sketch plane and work without using wireframe.

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# CHAPTER 7

## TUTORIAL

# Sweep and Multiple Thickness Shell

### Learning Objectives

---

*After completing this tutorial, you will be able to:*

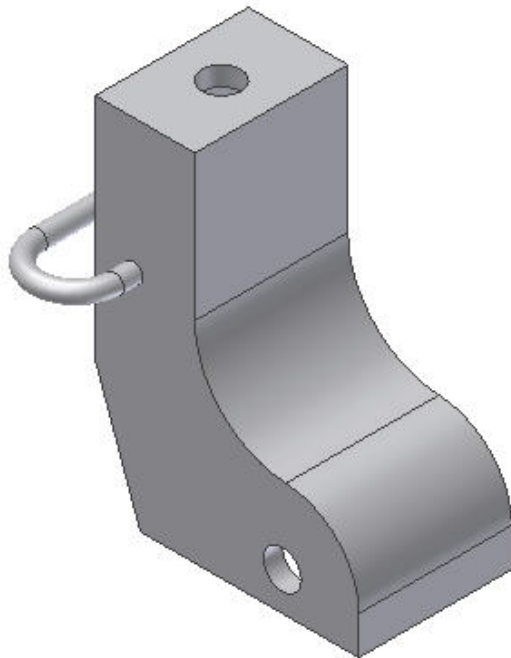
- Create a multiple thickness shell
- Create a sweep

### Required Competencies

---

*Before starting this tutorial, you should have been able to:*

- Construct, constraint and dimension sketches
- Use free rotate
- Create work planes
- Create holes



1. Open the part shellwithsweep.ipt

2. Select **Shell** from the Part Features panel bar as shown in Figure 1. This will open the shell dialog box from which the specifications will be set as shown in Figure 2.



Figure 1

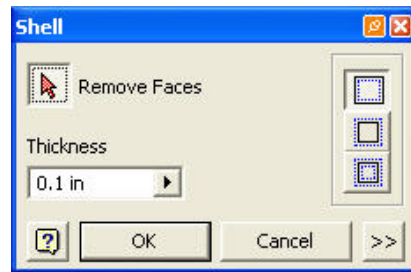


Figure 2

3. Rotate the part to view the bottom surface. This will be the face that will be removed. The button next to **Remove Faces** should be selected already, if not click on the button to select it. Move the cursor to the bottom face and click as shown in Figure 3.

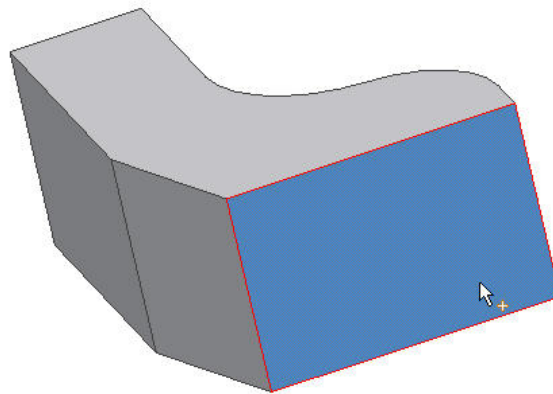


Figure 3

5. Set the wall **Thickness** to .25 in. as shown in Figure 4. For this part the back walls need to be a thickness of .375 due to structural requirements from the addition of a handle to be placed on the back.

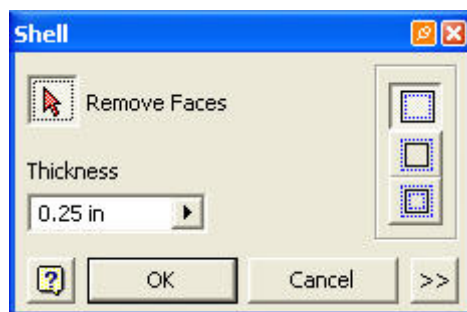


Figure 4

6. Click on the button with the two greater than symbols located at the bottom right corner of the dialog box to open the unique face thickness options dialog as shown in Figure 5.

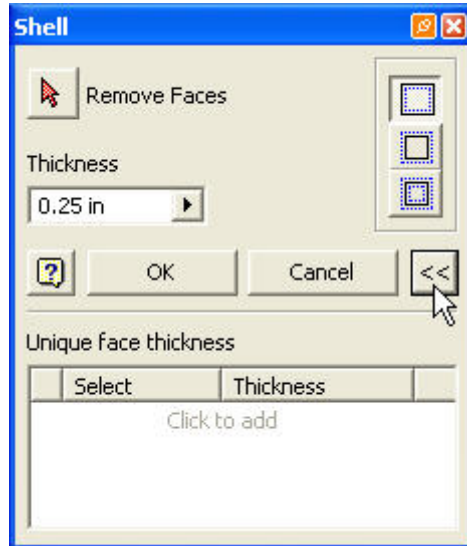


Figure 5

7. Move the cursor in the white area of the dialog box and click where the words **Click to add** are located as shown in Figure 6.

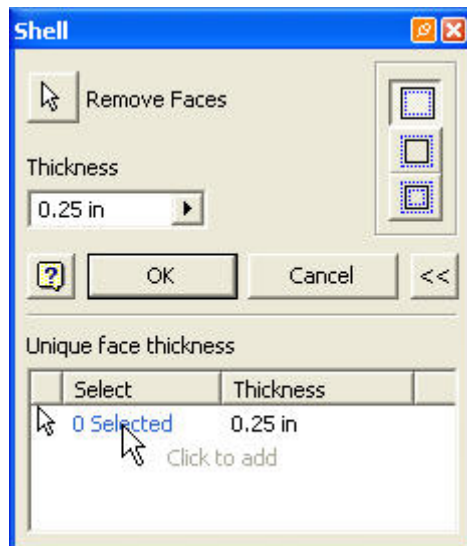


Figure 6

8. The words 0 Selected should be highlighted in blue at this time. This option allows the selection of the faces that will have a different thickness than the rest of the faces. Click on the two back surfaces to have the different thickness as shown in Figures 7 and 8.

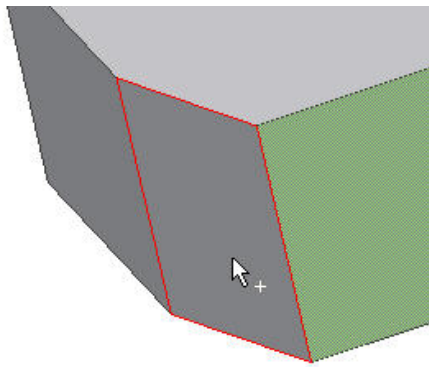


Figure 7

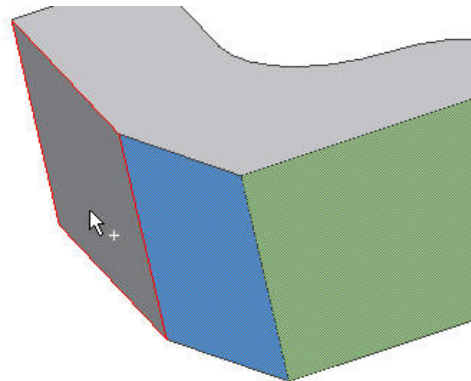


Figure 8

9. After selecting the two faces it will say 2 Selected. Next click on 0.25 under the Thickness tab of the dialog box and type in .375 as the thickness as shown in Figure 9.

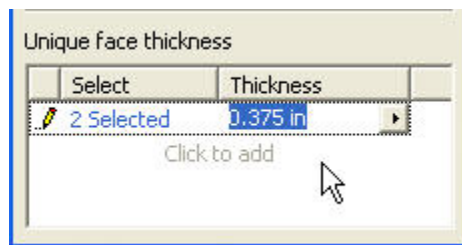


Figure 9

10. Click on the **OK** button and the shelled part should look like Figure 10.

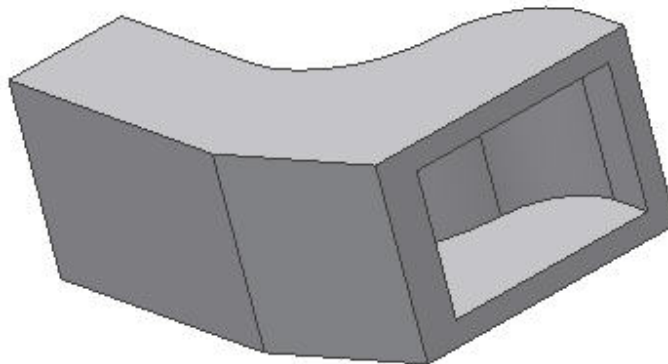


Figure 10

11. Set the part back to the Isometric view to begin placing the holes. Select Sketch and click on the top surface of the part as shown in Figure 11.

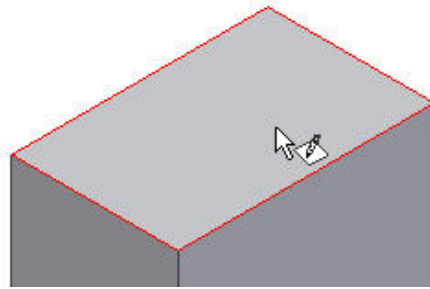


Figure 11

12. Place a **Point, Hole Center** somewhere on the sketch plane and position it in the center of the face using construction lines and Horizontal and Vertical constraints as shown in Figures 12 and 13.

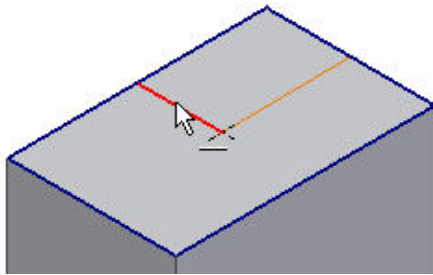


Figure 12

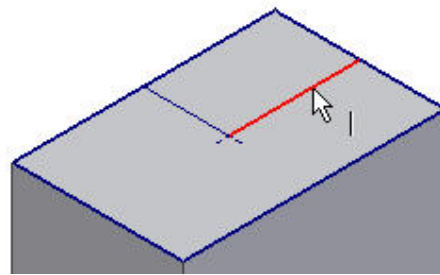


Figure 13

13. Click on the Return button to switch to the Part Features mode.. Select the **Hole** feature to open the hole dialog box as shown in Figure 14. Set the hole depth and diameter to 0.5 in. as shown in Figure15.

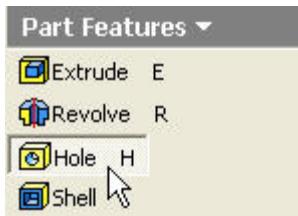


Figure 14

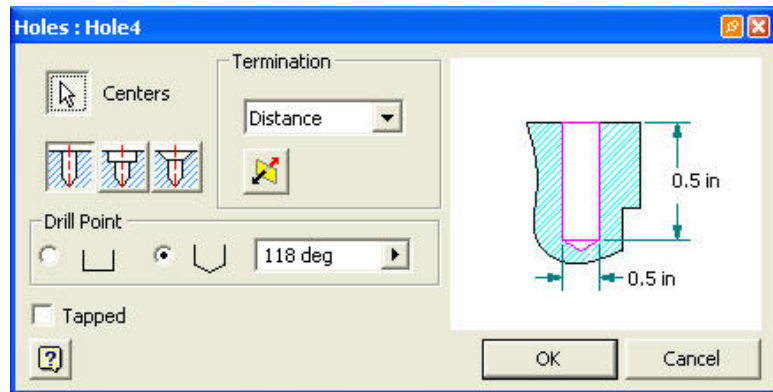


Figure 15

14. After the hole is created on the top of the part select the left surface, as shown in Figure 16, as the sketch plane for creating the second hole. We don't need to place a **Point,Hole Center** because the center point of the bottom right arc will be used as the center of the new hole.

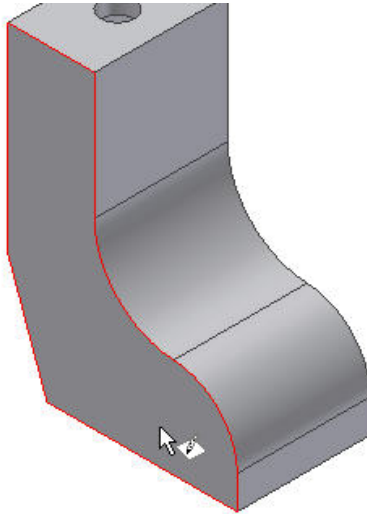


Figure 16

15. Click on the Return button to switch to the Part Features mode. Select the **Hole** feature to open the hole dialog box. Notice that the button next to Centers is selected. Click on the center point of the bottom arc as shown in Figure 17. Set the Termination to **Through All** and the diameter to 0.5 in. as shown in Figure 18.

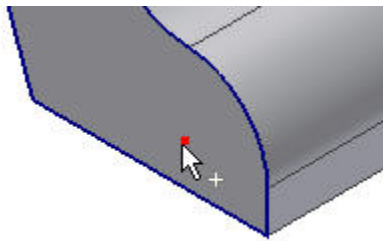


Figure 17

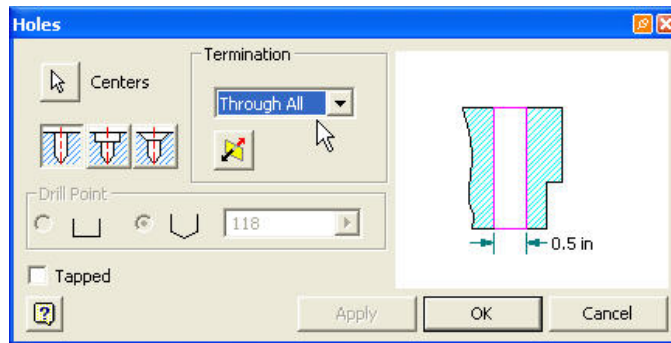


Figure 18



16. In order to begin the sweep a new work plane must be created. Create an offset work plane parallel to the top surface down at a distance of -1.75 in. as shown in Figure 19.

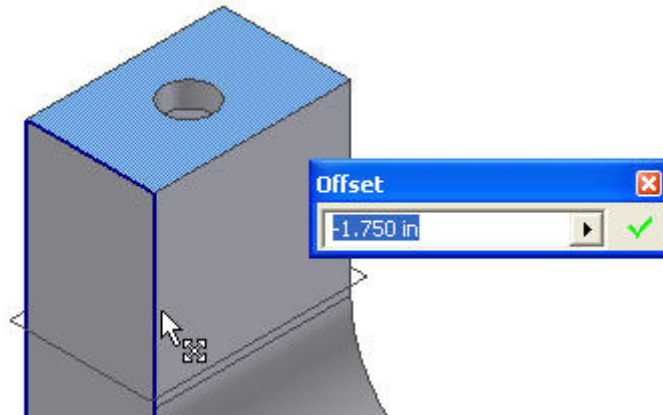


Figure 19

17. Choose **Sketch** from the menu bar and select this new work plane to be the new sketch plane. Once the sketch plane is selected choose the **Look At** command, Figure 20, and click on the work plane to view the part from the top.

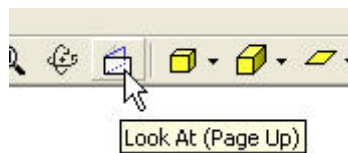


Figure 20

18. Select **Project Geometry** from the 2D Sketch Panel as shown in Figure 21. Select the top and bottom horizontal edges of the part as shown in Figures 22 and 23.

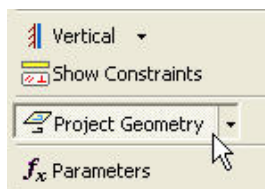


Figure 21

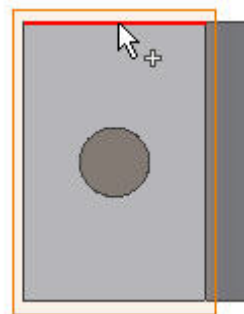


Figure 22

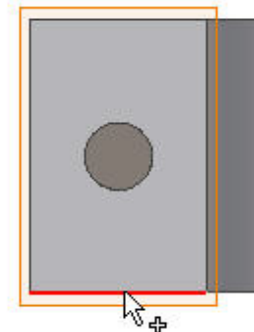


Figure 23

19. Create the sketch, as shown in figure 24, which will be used as the path for the sweep. Make sure that the center points of the arcs are in line with the left edge and equally spaced from the top and bottom edges of the part. Use the **Coincident** and **Equal** constraints to perform this task.

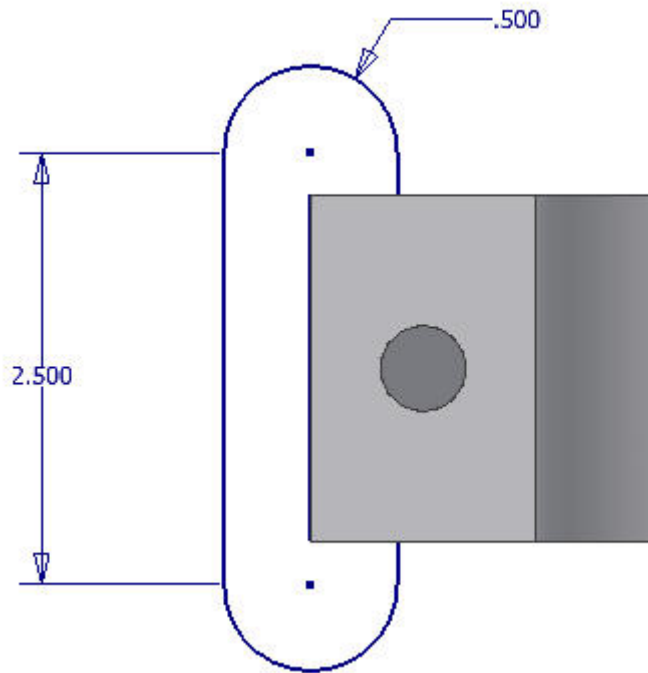


Figure 24

20. Click **Return** to switch to the Part Feature mode. Switch to the Isometric view and select the left surface as the new sketch plane as shown in Figure 25.

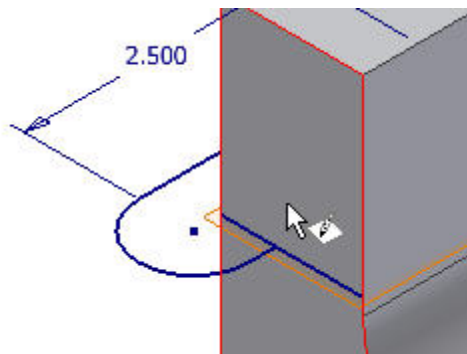


Figure 25

21. Use Project Geometry to project the short line of the previous sketch as a point on the new sketch plane as shown in Figure 26.

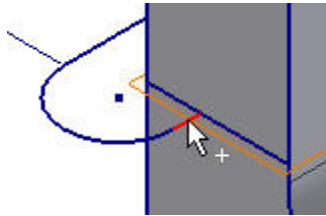


Figure 26

22. Draw a circle using the projected point as the center point as shown in Figure 27. Dimension the circle with a diameter of 0.25 in. as shown in Figure 28.



Figure 27

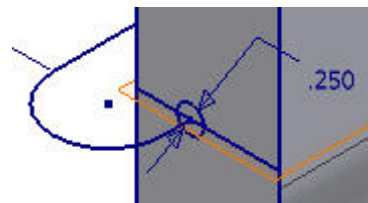


Figure 28

22. Click Return to and select **Sweep** from the Part Features panel to open the Sweep Dialog box as shown in Figures 29 and 30.

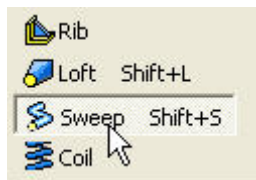


Figure 29

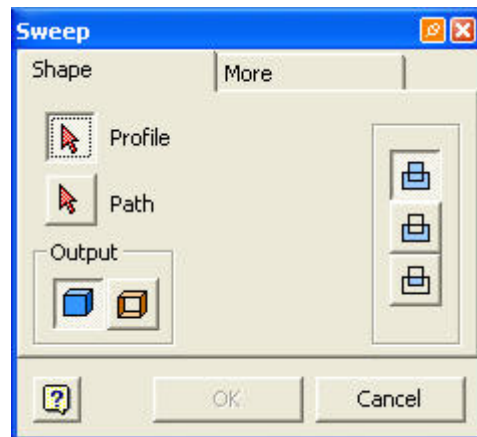


Figure 30

23. Select the circle that was just sketched as the Profile as shown in Figure 31. Next click on the button next to Path in the dialog box and click on the first sketch as shown in Figure 32 as the sweep path.

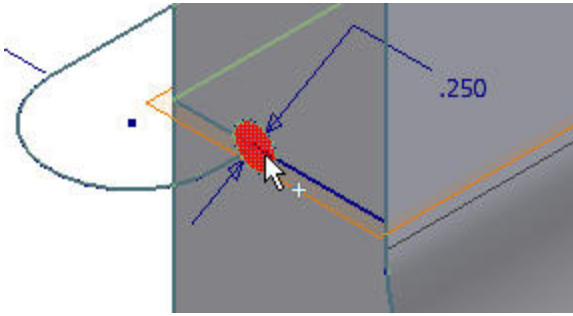


Figure 31

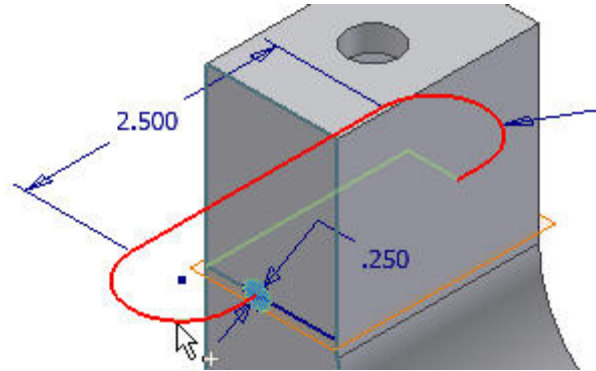


Figure 32

24. Once the Path and Profile of the sweep are selected click **OK**, which should create the finished part shown in Figure 33.

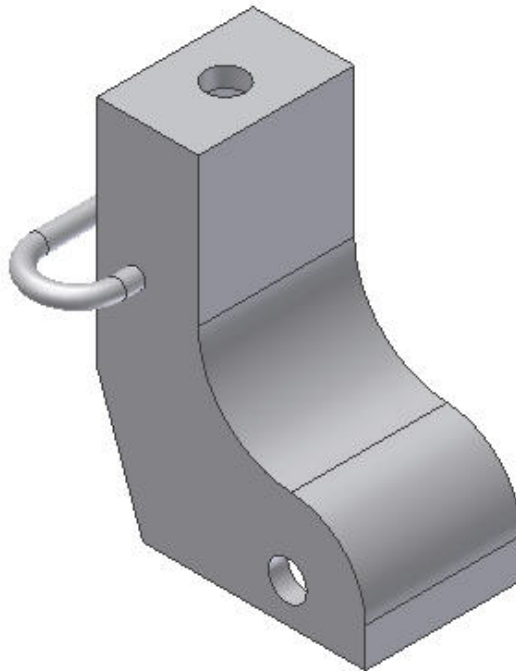


Figure 33

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# CHAPTER TUTORIAL

## Coil Features in Parts

### Learning Objectives

---

*After completing this tutorial, you will be able to:*

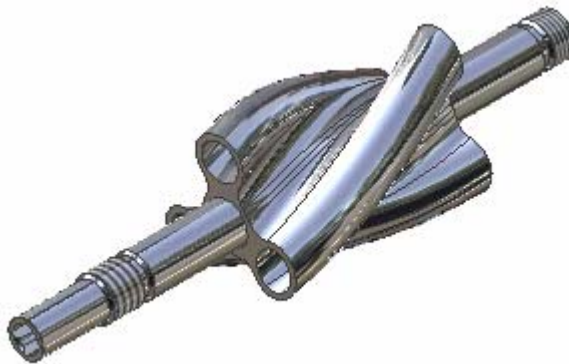
- Construct and use coils

### Required Competencies

---

*Before starting this tutorial, you should have been able to:*

- Construct, constraint and dimension Sketches
- Project Geometry on Sketch Planes
- Construct Work Features



**Figure 1: Impeller**

1. Open file impeller\_shaft.ipt



**Figure 2: Impeller\_shaft**

2. Construct a new **Work Plane** parallel to the default **XY** plane at an offset distance of 4.25”.

a. Click the work plane tool on the part feature toolbar.



Figure 3

b. Expand the Origin in the Browser and click the default **XY** plane.

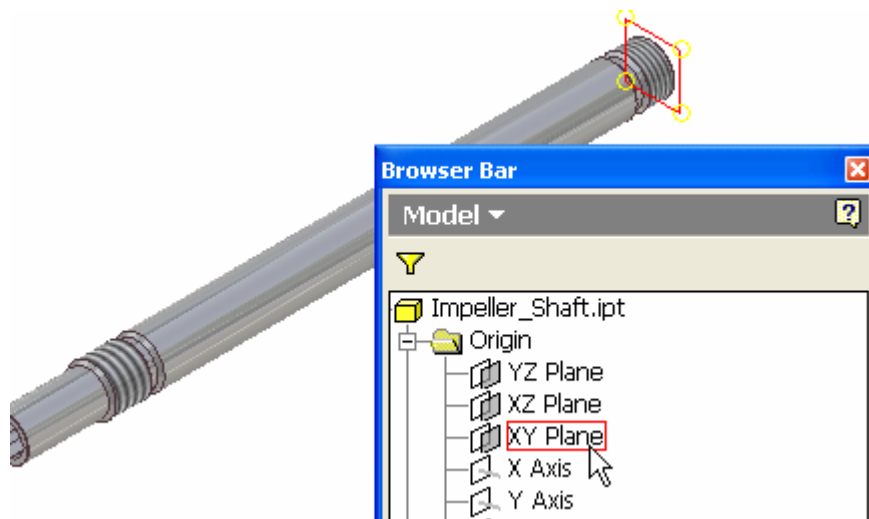
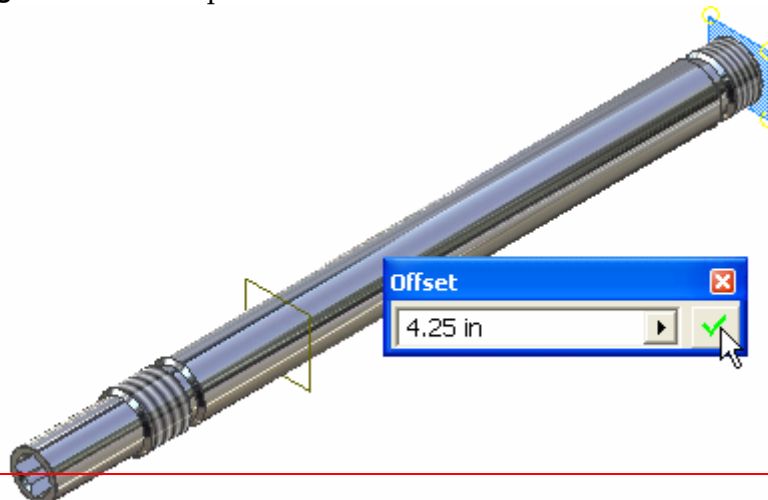


Figure 4

c. **Drag** the new work plane to an offset distance of 4.25”.



## Figure 5



3. Project the default **Y-Axis** on the new work plane.
  - a. Enter **Sketch Mode** and select the newly constructed work plane as a sketch plane.
  - b. Click the **Project Geometry** tool.

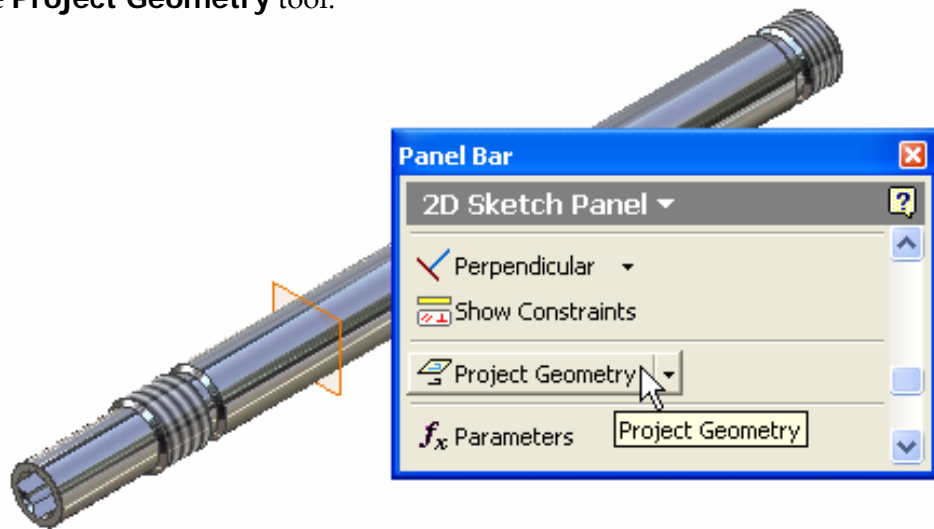


Figure 6

- c. Click the **Y-Axis** in the Browser.

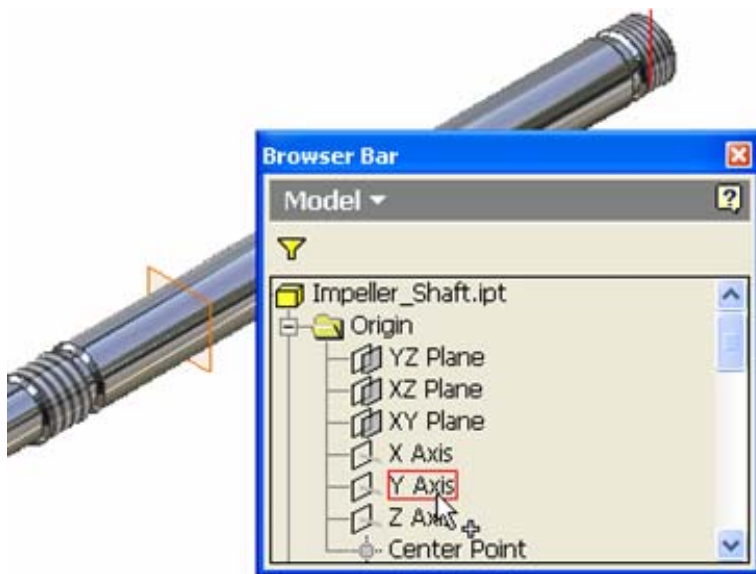


Figure 7.a

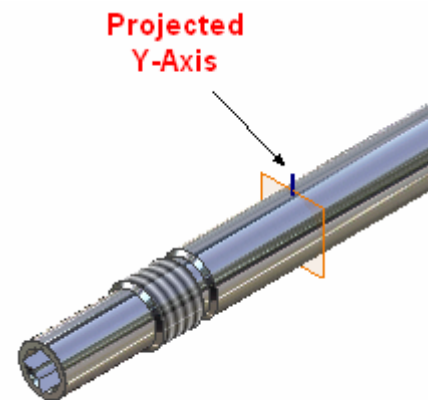


Figure 7.b

d. Exit **Sketch Mode**.

4. Construct a new **Work Plane 2** making a  $24^\circ$  angle with the previously constructed **Work Plane 1** and passing through the **Projected Y-Axis**.

a. Click the work plane tool and select the projected Y-Axis.

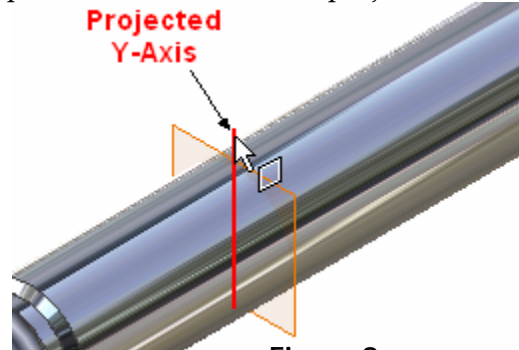


Figure 8

b. Click the previously constructed Work Plane 1 and change the angle to  $24^\circ$  to create Work Plane 2..

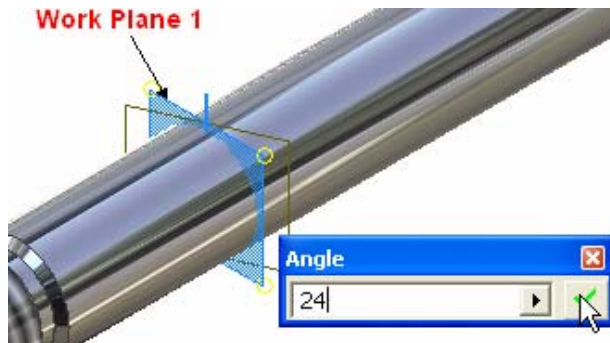


Figure 9.a

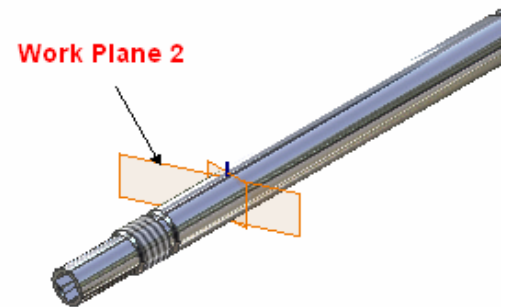


Figure 9.b

5. Construct a new **Work Axis 1** coinciding with the Projected Y-Axis.

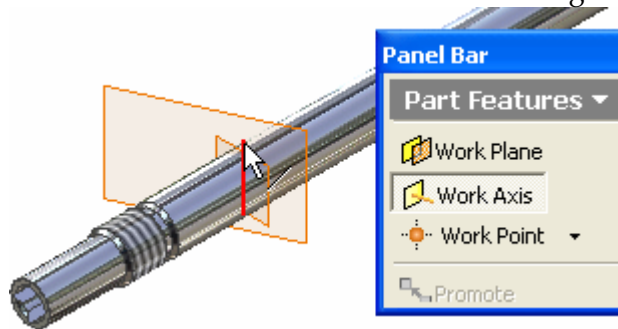


Figure 10.a

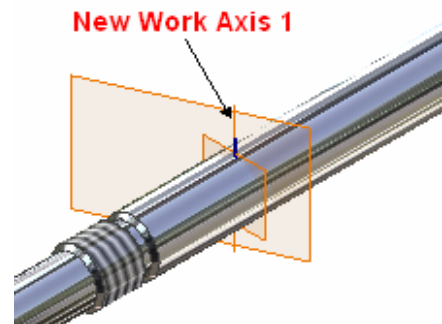


Figure 10.b

6. Turn the visibility of Work plane 1 and the previously constructed Sketch off.

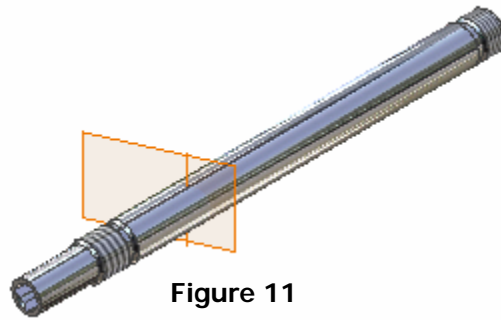


Figure 11

7. Click the **Look At** button and click Work Plane 2.

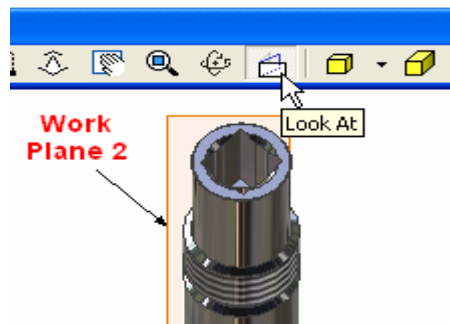


Figure 12

8. Rotate the image 90°.

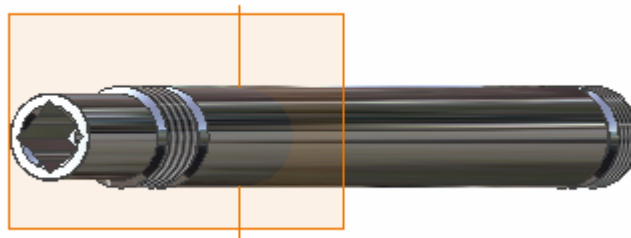


Figure 13

9. Enter **Sketch Mode** and select work plane 2 as a sketch plane.

10. Right-click on the work plane and **Slice Graphics**.

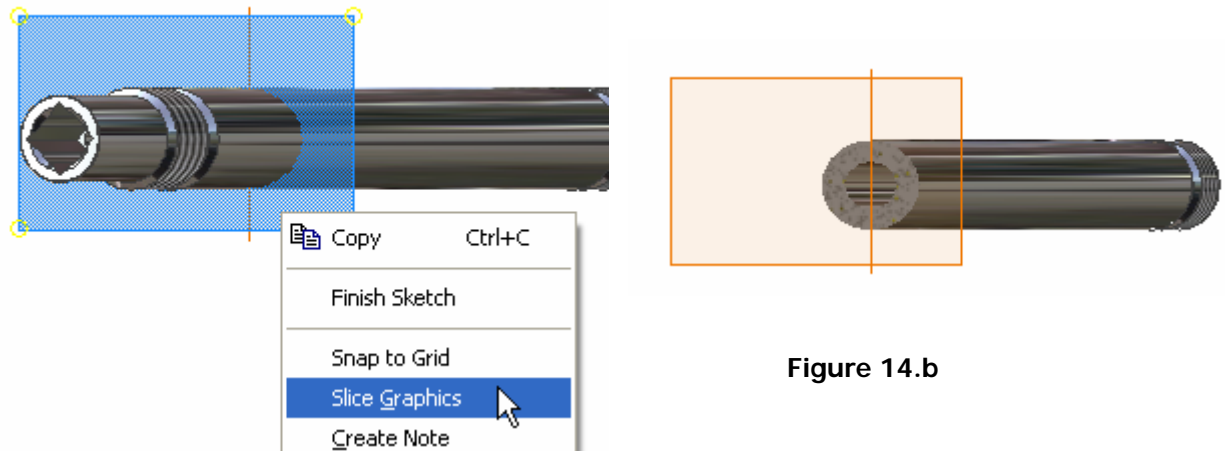


Figure 14.a

Figure 14.b

11. Use the **Project Geometry** tool and project Work Axis 1 and the Z-Axis on the work plane.

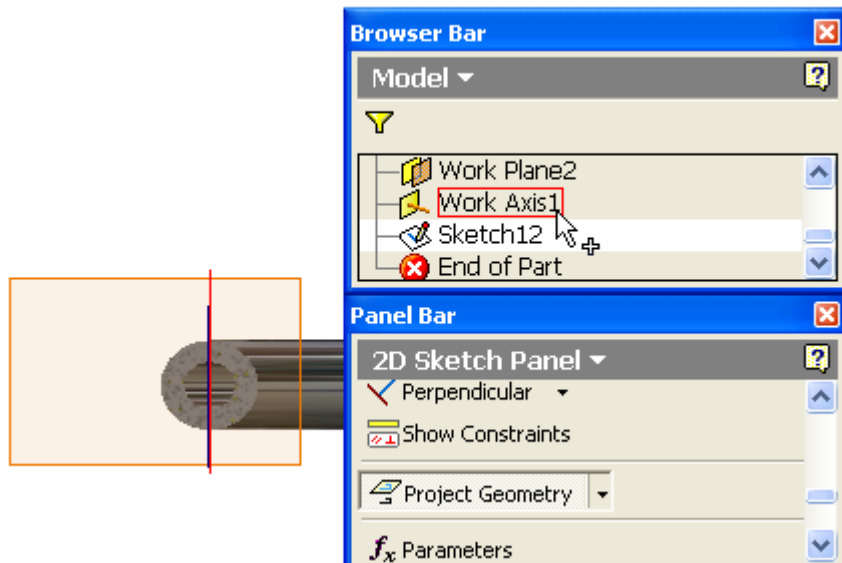


Figure 15.a

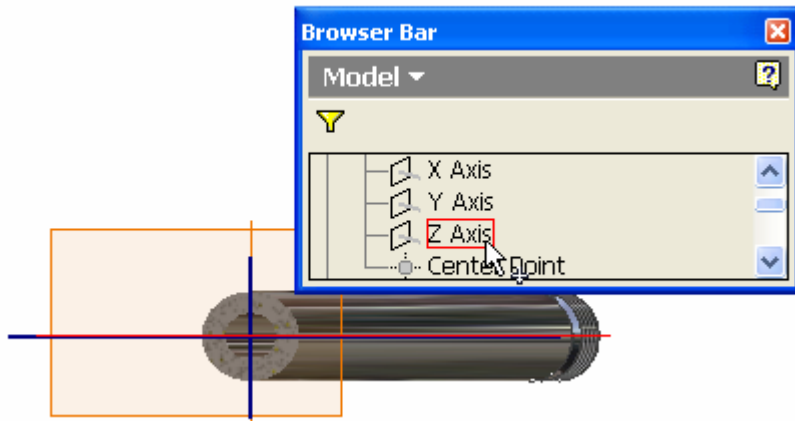


Figure 15.b

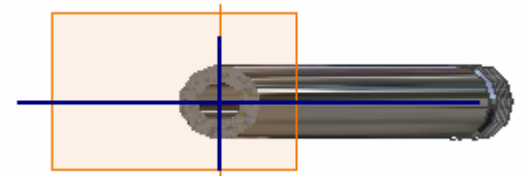


Figure 15.c

12. Construct the following sketch.

*Note: The next three (3) pages outline suggested construction steps.*

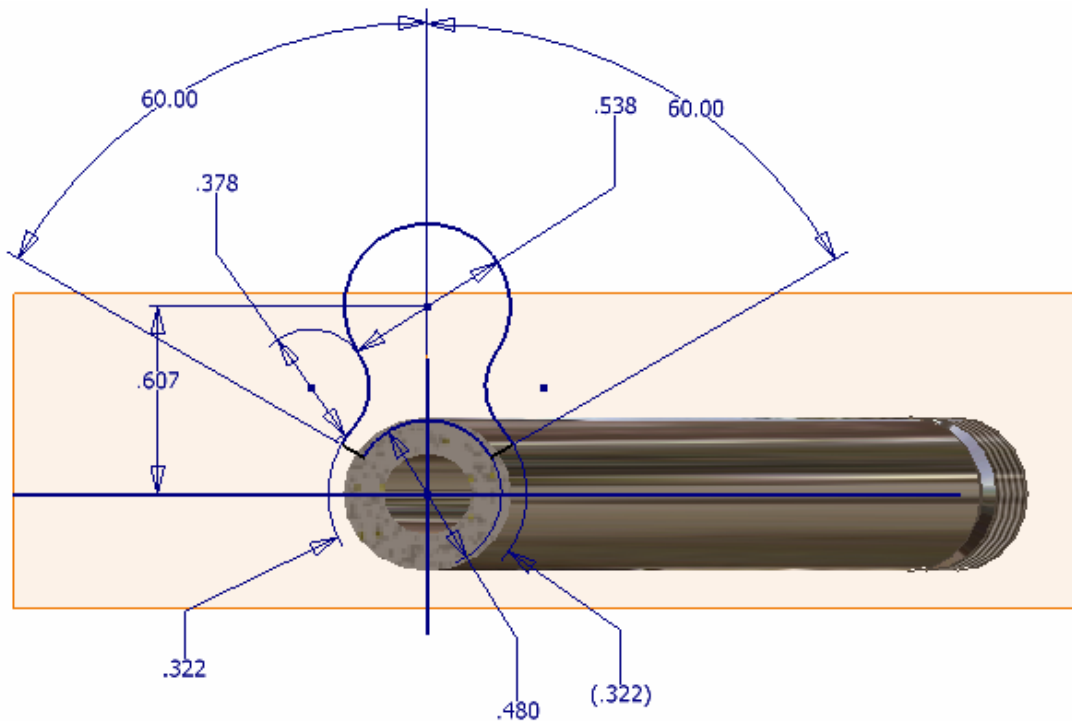


Figure 16

Suggested Construction Steps

a.

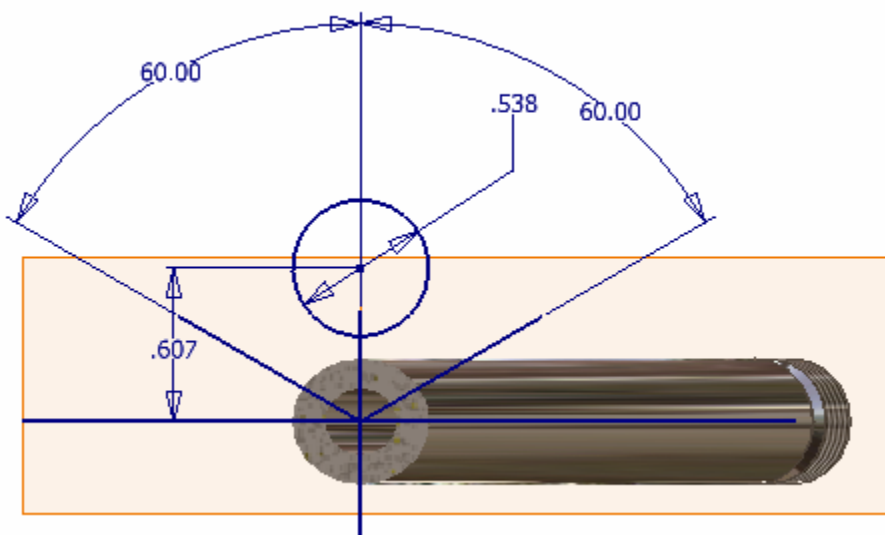


Figure 17.a

b.

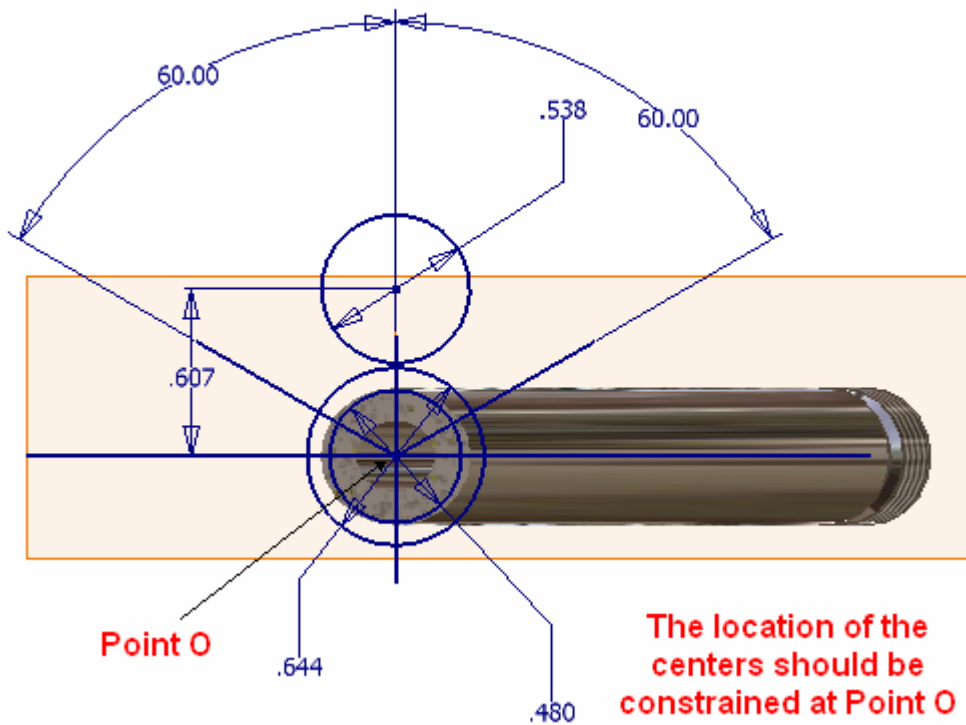


Figure 17.b

c. Trim

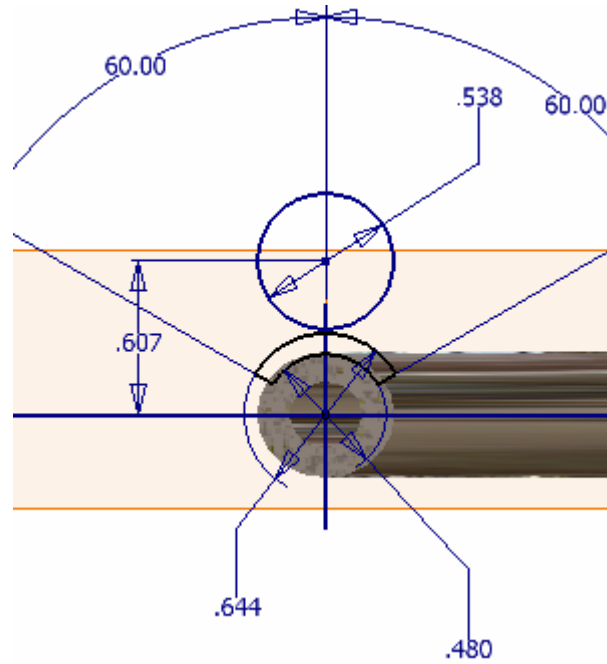


Figure 17.c

d.

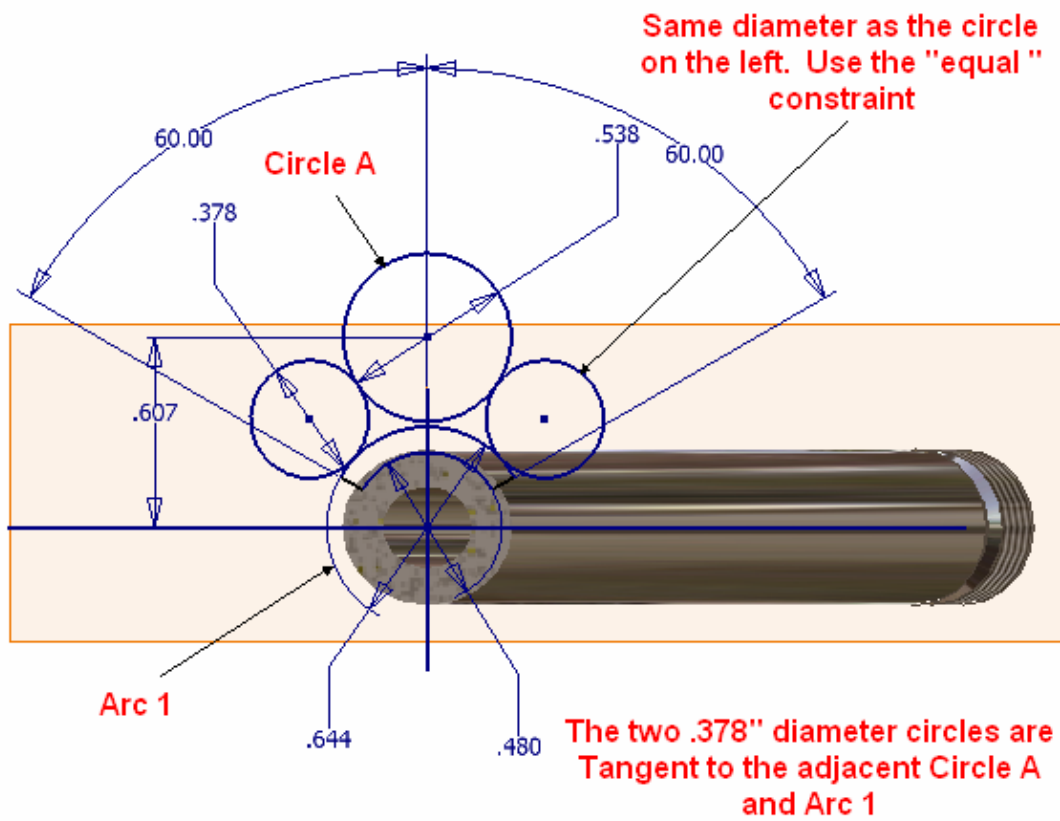


Figure 17.d

e. Trim and verify that the **Tangency Constraints** between adjacent arcs are maintained.

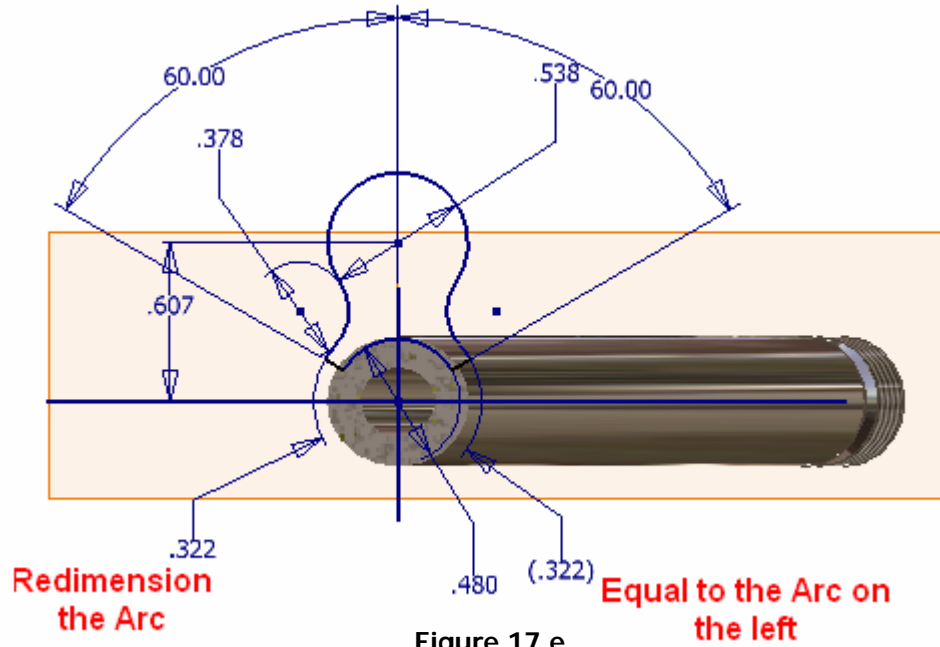


Figure 17.e

f. The following should be Driven Dimensions.

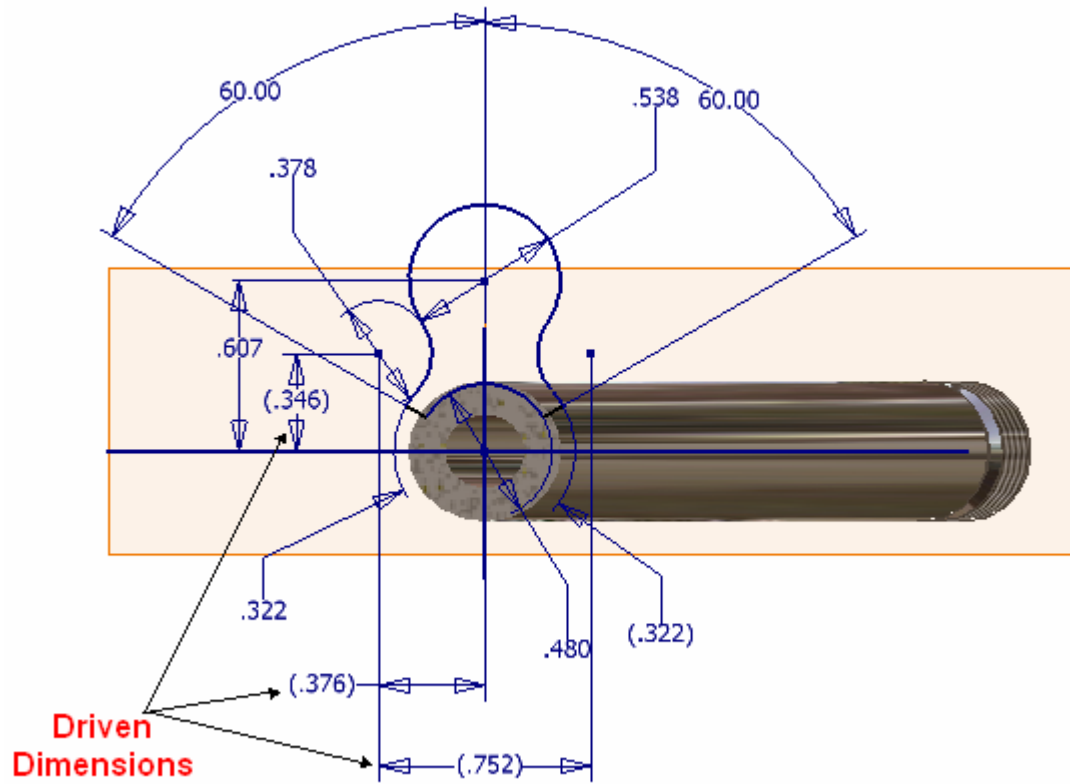


Figure 17.f



13. Exit **Sketch Mode** and **Coil** the drawn profile about the Z-Axis, 1/3 of a revolution to a coil height of 2.923"

a. Click the **Coil** tool on the Feature toolbar.

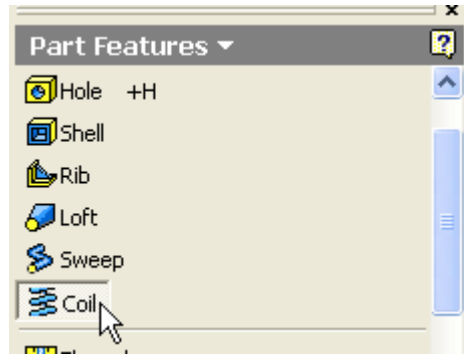


Figure 18

b. Select the drawn profile.

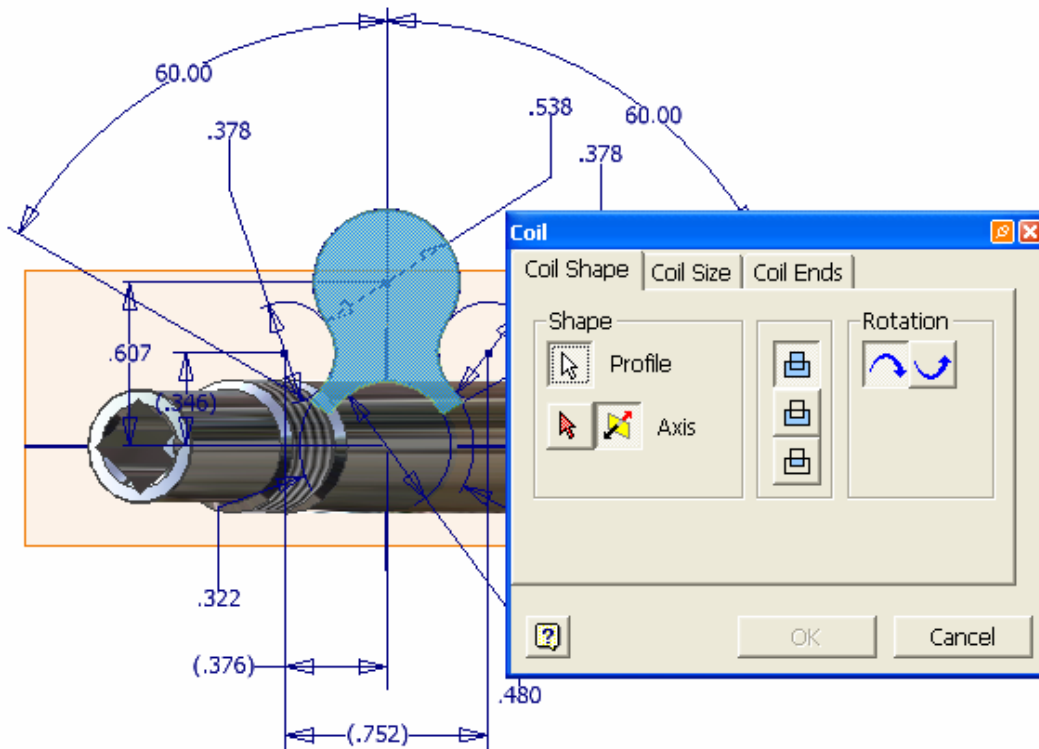


Figure 19

- c. Select the Z-Axis and the proper Direction and Rotation for the required coil shape.

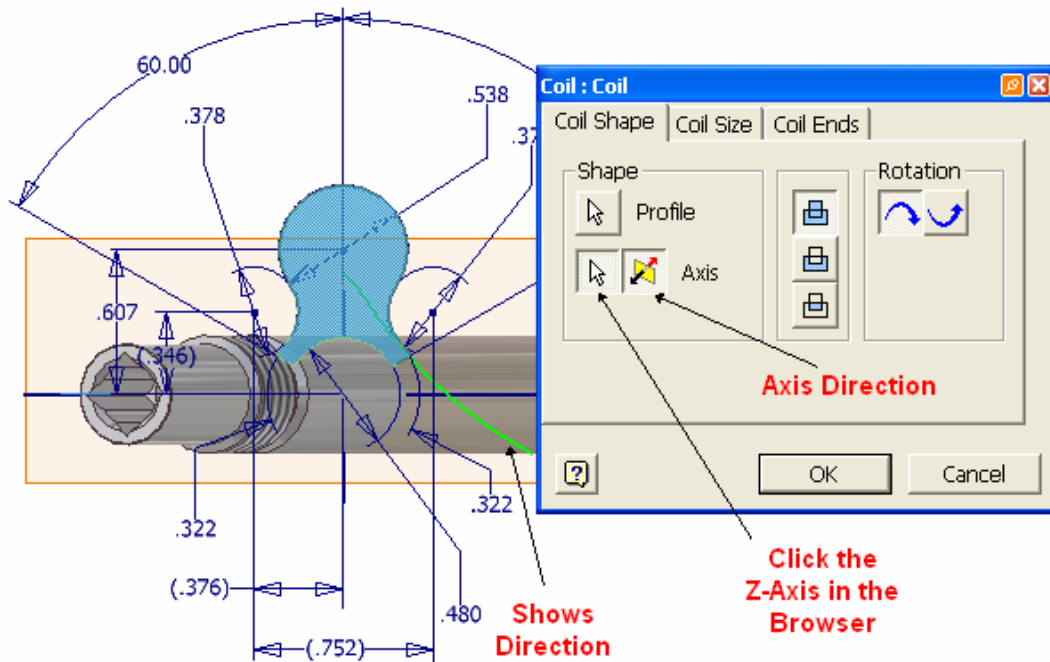


Figure 20

- d. Click the **Coil Size** tab and adjust the parameters as shown below.

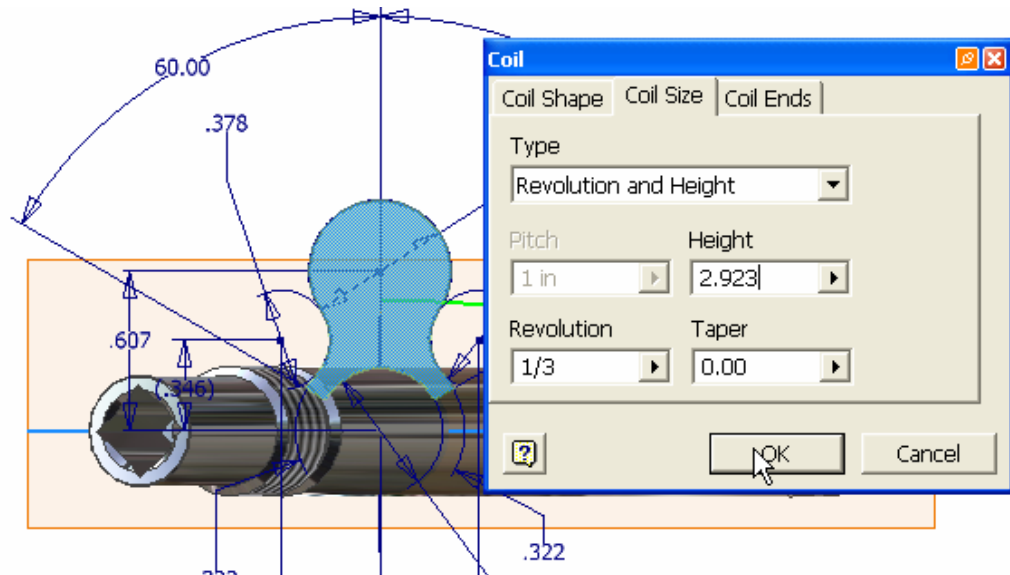


Figure 21

e. Click the **Coil Ends** tab and select **Natural** Start and End and click OK.

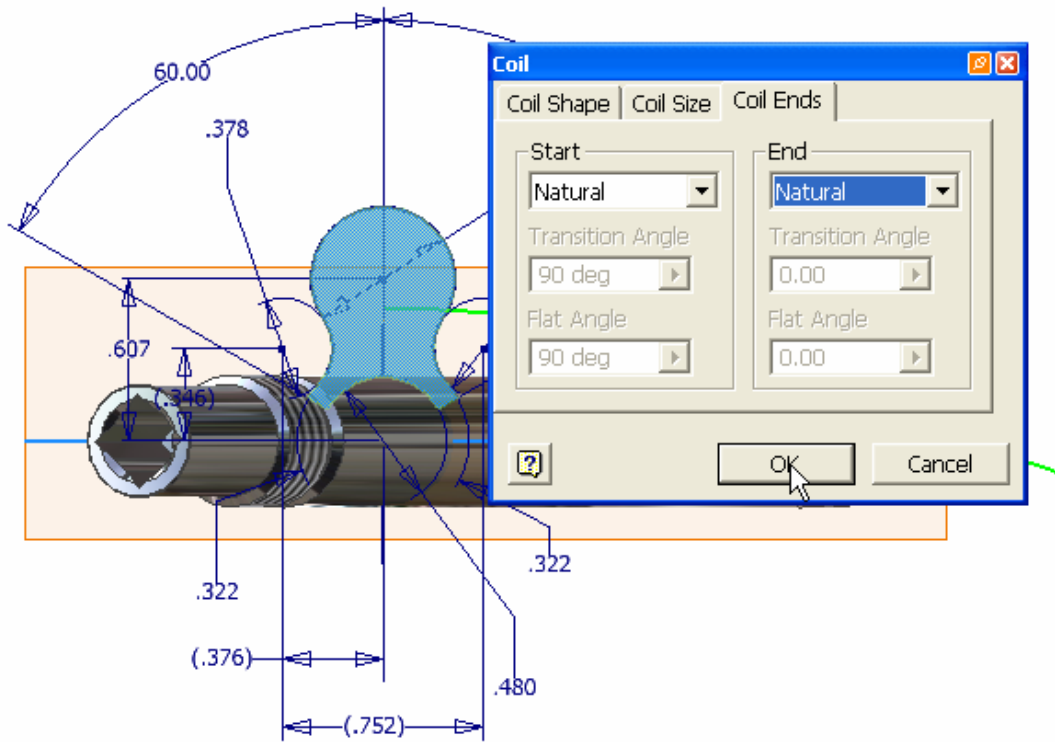


Figure 22.a

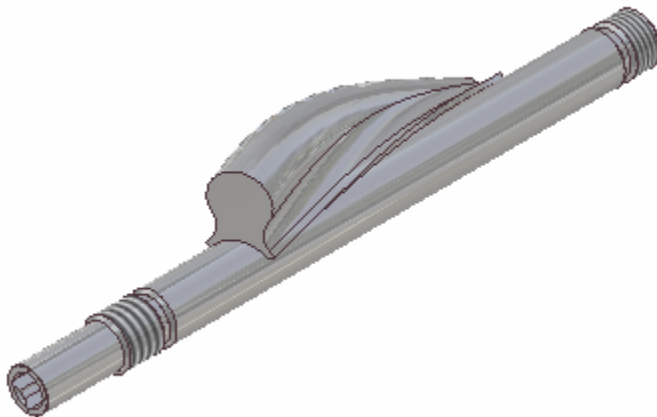


Figure 22.b

14. Enter **Sketch Mode** and select **Surface A** as the sketch plane. Click the Look At button.

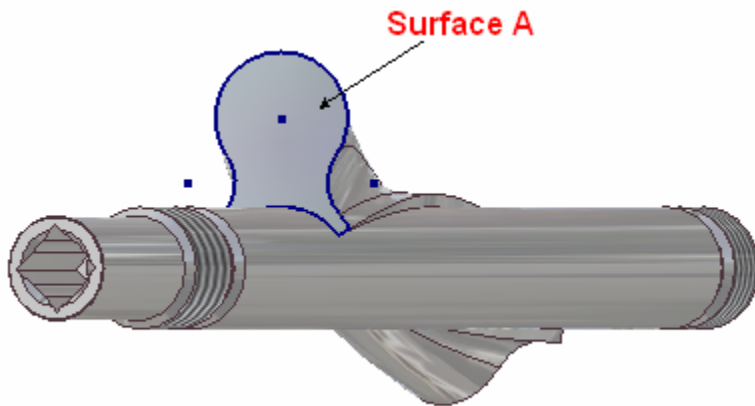


Figure 23

15. Construct the .410" diameter hole as shown below and exit **Sketch Mode**.

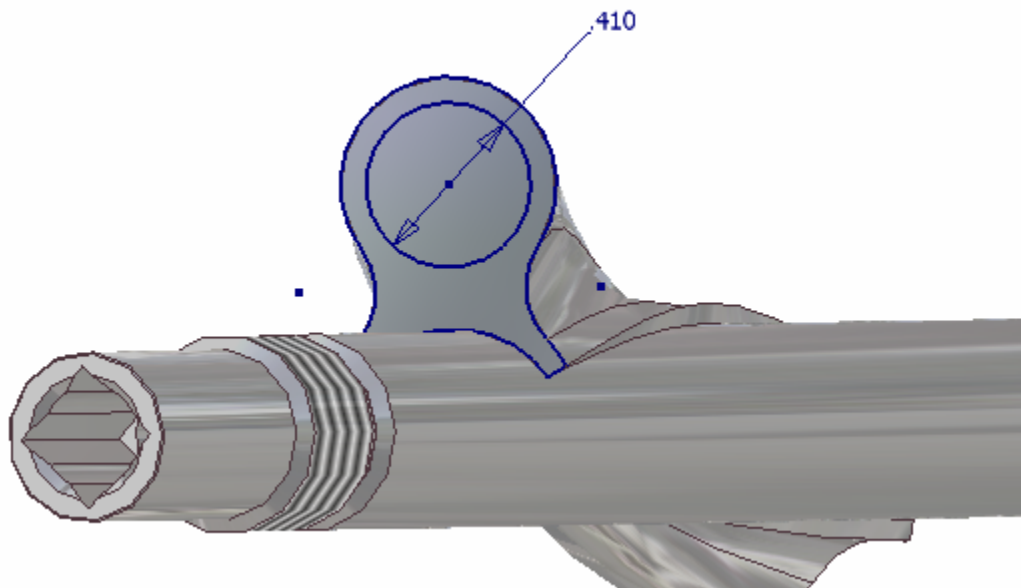


Figure 24

16. **Coil** and **Cut** the drawn profile with the same parameters as in step 13.

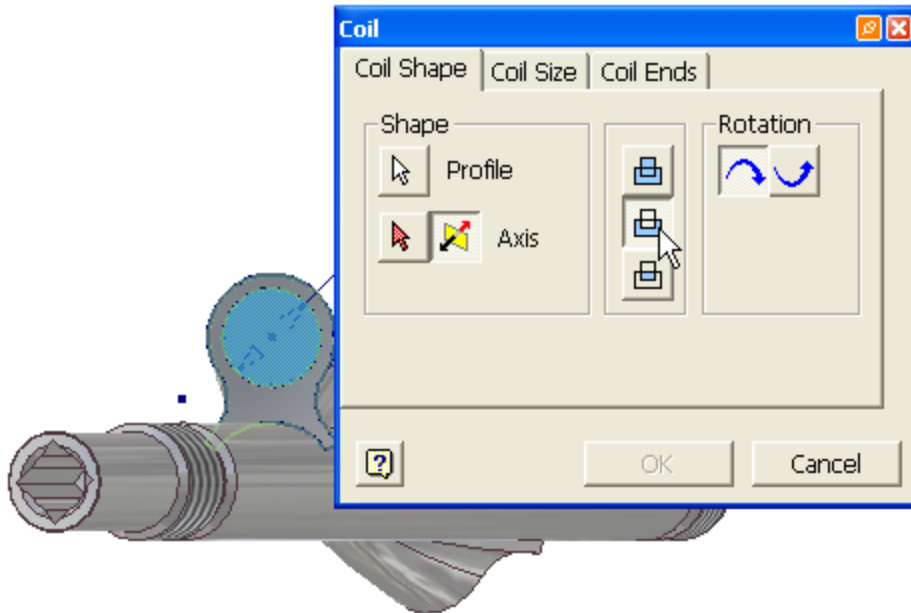


Figure 25.a

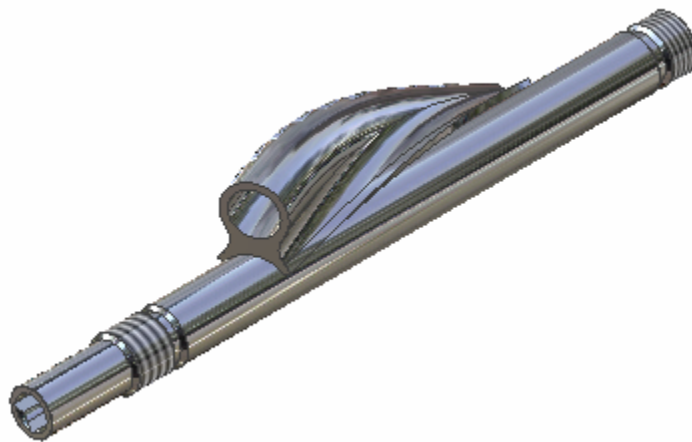


Figure 25.b

17. Create a **Circular Pattern** of the two Coils about the Z-Axis with three (3) occurrences in the pattern.

a. Click the **Circular Pattern** tool on the Feature toolbar and select the two Coils as Features.

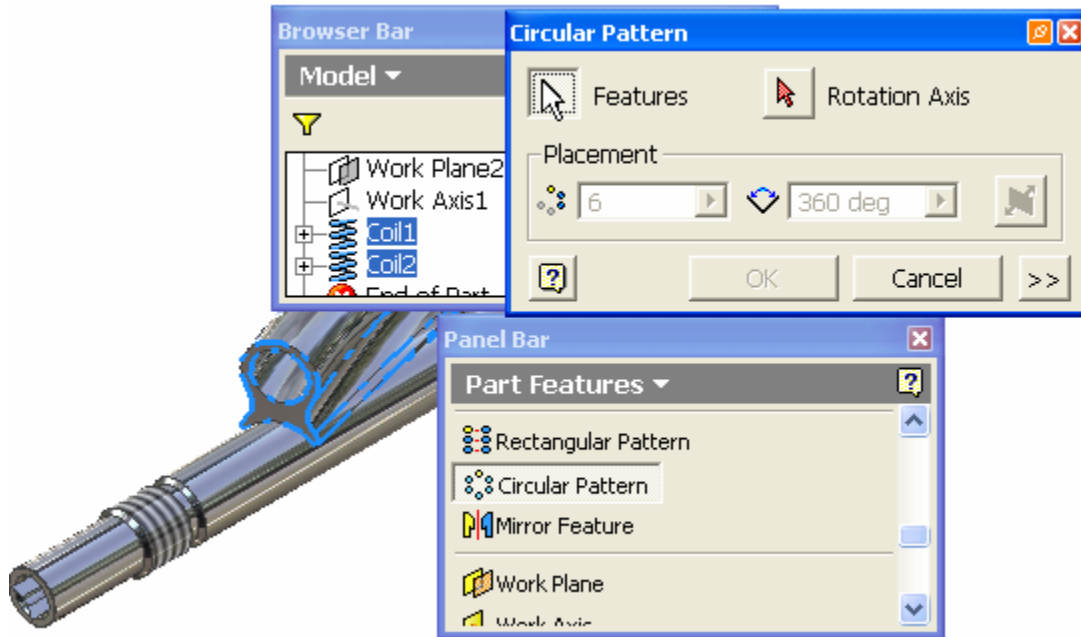


Figure 26

b. Click the Rotation Axis button, select the Z-Axis in the Browser and adjust the required parameters as shown below.

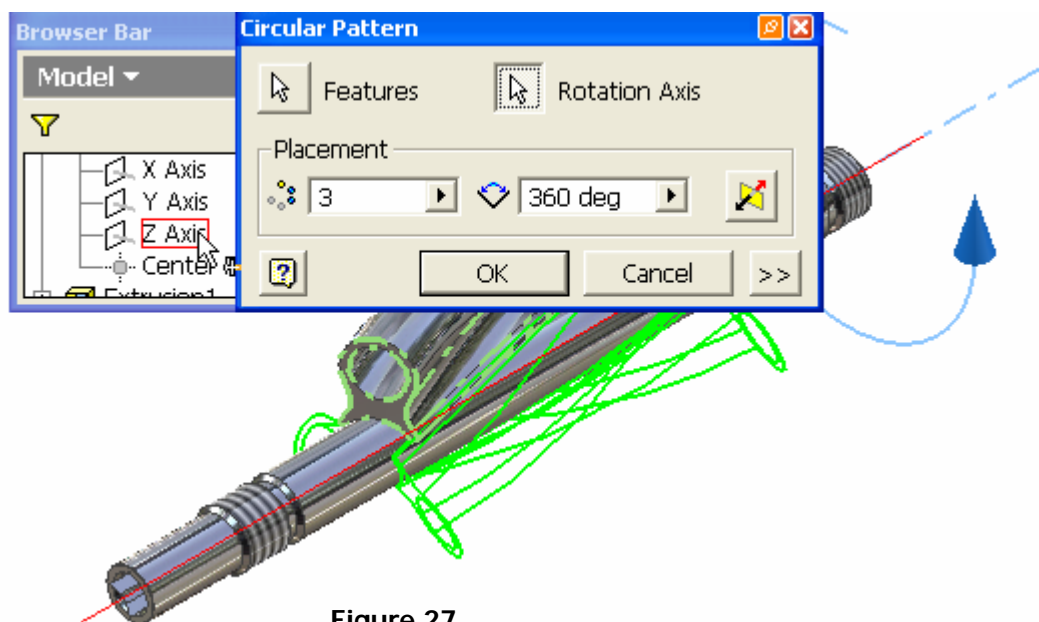
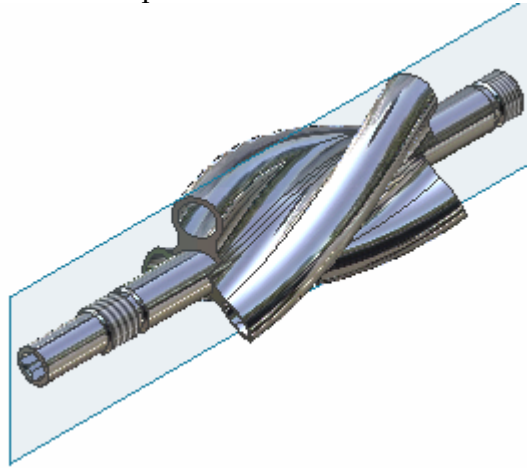


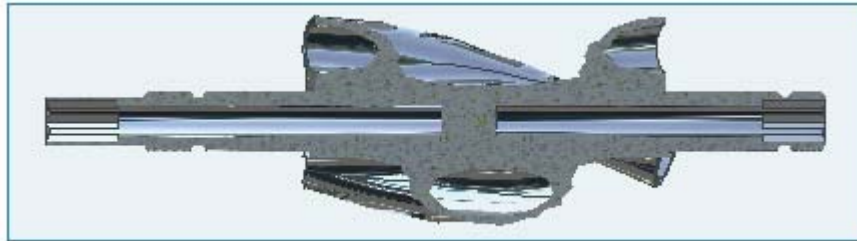
Figure 27

18. Turn the visibility of the **YZ** plane on.



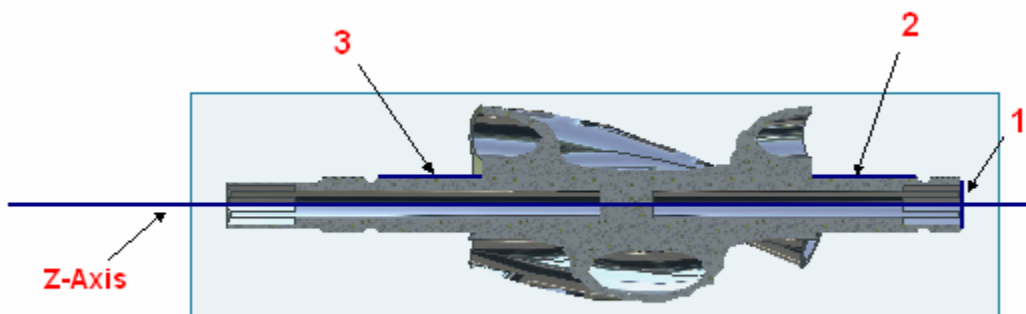
**Figure 28**

19. Enter **Sketch Mode** and select the YZ plane as a sketch plane. Use the **Look At** and **Slice Graphics** functions for easy construction. Rotate the image.



**Figure 29**

20. Use the **Project Geometry** tool and project Edge 1, the larger diameter portion of the shaft 2 and 3, and the Z-Axis on the sketch plane.



**Figure 30**

21. Construct the following sketch.

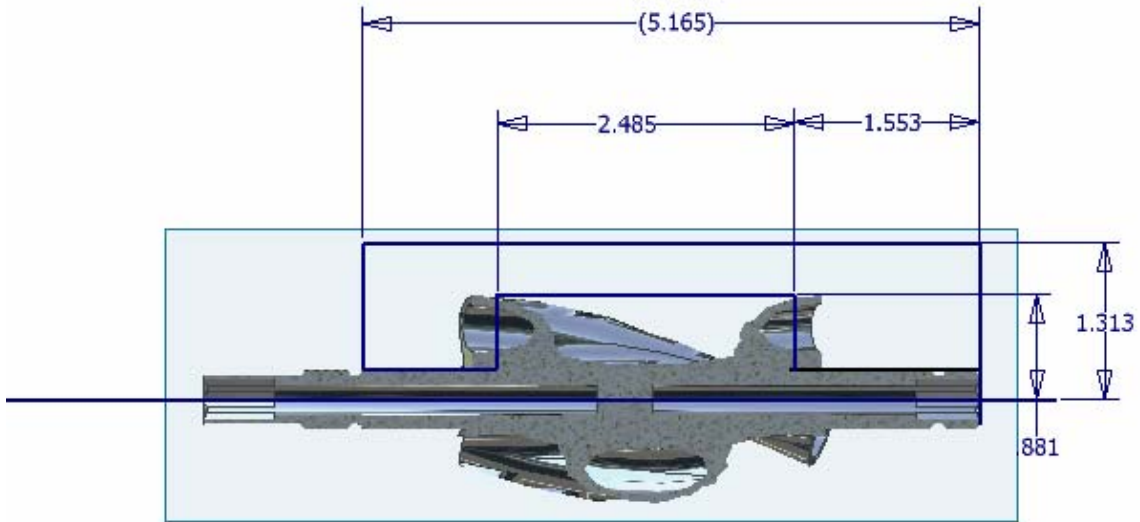


Figure 31

22. Exit **Sketch Mode**, **Revolve** and **Cut** the constructed profile about the Z-Axis.

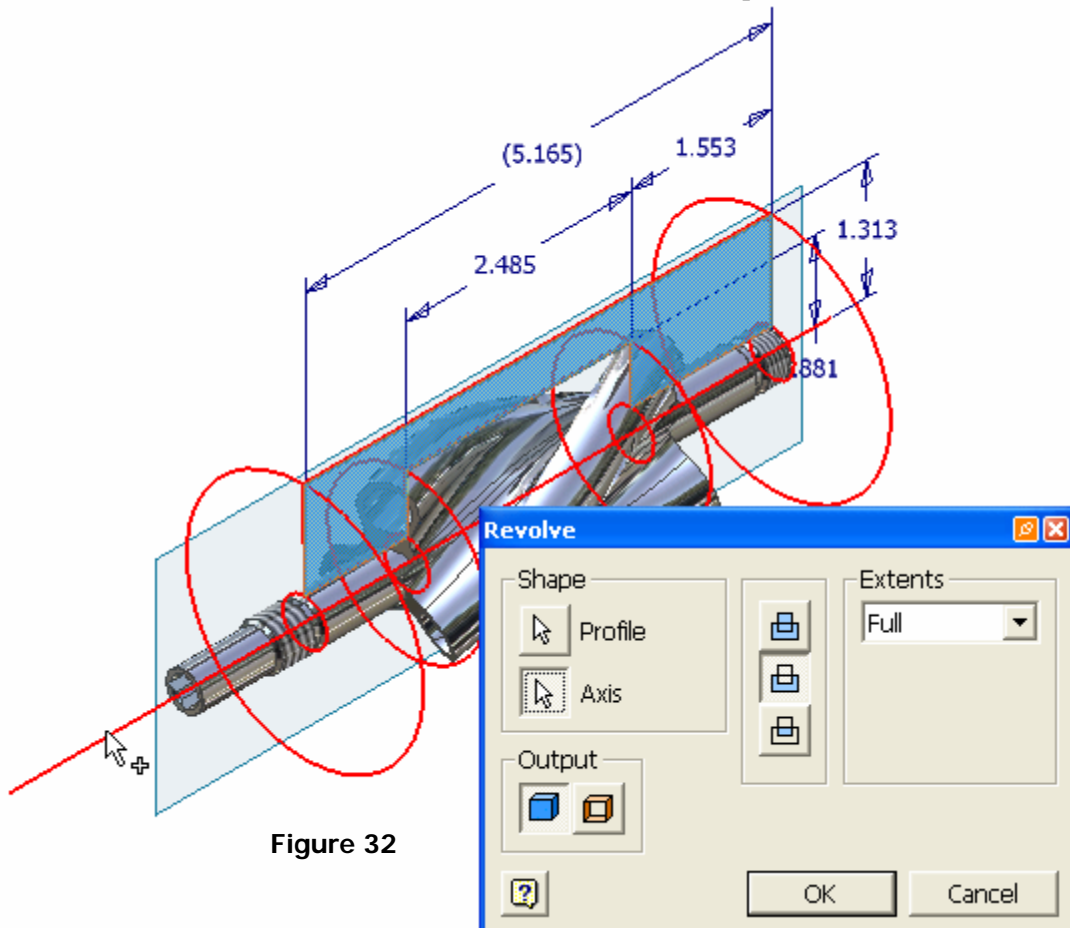


Figure 32



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# Autodesk Inventor Tutorial 1

## Introduction to Autodesk Inventor

*Ron K C Cheng*

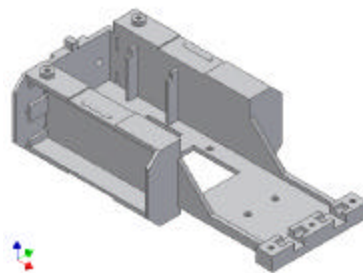
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### AUTODESK INVENTOR FUNCTIONS

A product usually consists of a number of component parts. Using computer as a tool to design and manufacture products, you construct computer models to represent each individual component of the product and put the computer models together to form a virtual assembly to explore and evaluate the integrity of the design. To illustrate how the components of an assembly are related to each other, you explode them apart. Although it is very common to use electronic data of computer models directly in downstream computerized manufacturing operations, there are times when conventional 2D engineering drawings are required. To meet this requirement, you output engineering drawings from the computer models of the individual parts, assemblies, and exploded assemblies. Serving these design requirements, Autodesk Inventor has four basic functions: constructing 3D parametric solid parts, assemblies of solid parts, exploded presentations of assemblies, and engineering drawings.

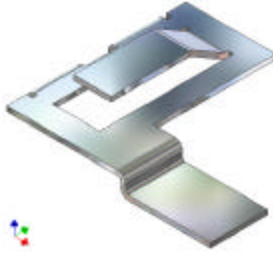
### Constructing Solid Parts and Sheet Metal Parts

The prime function of Autodesk Inventor is to construct computer models in the form of 3D parametric solid parts and 3D sheet metal parts that represent a component part in the computer. Figure 1–1 shows a solid part for the model of a component of a radio controlled toy car.



**Figure 1–1**  
*Solid part of the chassis of a scale model car*

Sheet metal parts are a special kind of solid part. You make a sheet metal component by cutting and folding a sheet of metal of uniform thickness. To meet the manufacturing requirement of providing rounded bends at the joints of faces, relieves at the bends, hems at the edges, and seams at the joints, you need a special kind of solid modeling tool. Figure 1–2 shows the model of a sheet component.



**Figure 1–2**  
*Sheet metal component*

## Constructing Assemblies of Solid Parts

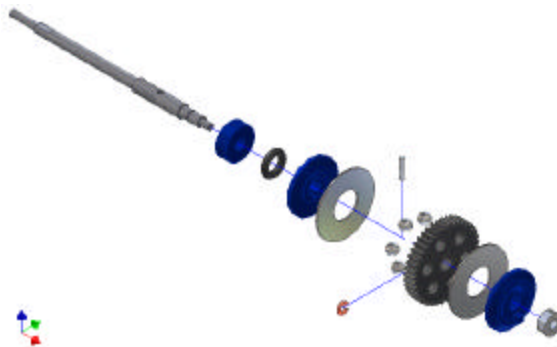
The next function of Autodesk Inventor is to construct a virtual assembly of 3D solid parts. An assembly is a device consisting of a number of component parts. Figure 1–3 shows the assembly of the axle of a toy car.



**Figure 1–3**  
*Assembly of the axle of a toy car*

## Constructing Exploded Presentation of Assemblies

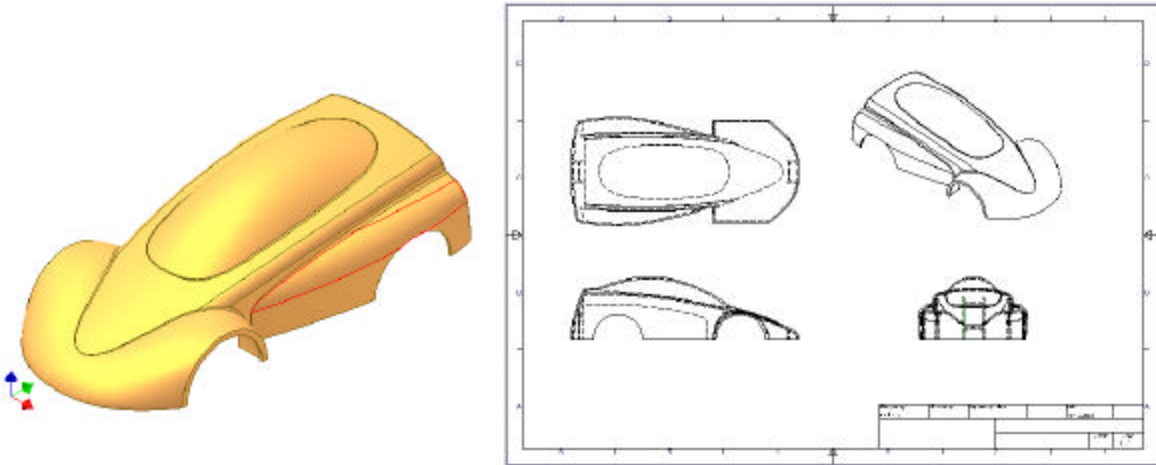
The third function of Autodesk Inventor is to construct an exploded presentation of assemblies. To illustrate how various parts of an assembly are put together, you construct a presentation of an assembly. In the presentation, you explode or tweak the components apart. Figure 1–4 shows that exploded presentation of the assembly shown in Figure 1–3.



**Figure 1–4**  
*Presentation of the assembly of the rear axle*

## Constructing Engineering Drawings

The fourth function of Autodesk Inventor is to construct engineering drawings. Engineering drawing is an engineering communication tool that depicts a 3D design in 2D engineering drawing views. You specify a solid part or an assembly and the computer application can automatically generate 2D orthographic views of 3D solid parts, sheet metal parts, and assemblies, flat patterns of sheet metal, and exploded views of assemblies. Figure 1–5 shows the engineering drawing derived from the solid part of a toy car body.



**Figure 1–5**  
*Toy car body and its engineering drawing*

---

## AUTODESK INVENTOR FILE TYPES

To cope with the four design functions, Autodesk Inventor uses four kinds of files:

<b>Part files</b>	for constructing solid parts
<b>Assembly files</b>	for assemblies of solid parts or sub-assemblies
<b>Presentation files</b>	for exploded views of an assembly
<b>Drawing files</b>	for engineering drawings

### Part Files

You construct a 3D solid part or a sheet metal part in a part file; the file extension is .ipt. A part file stores the definition of the parametric 3D solid part.

### Assembly Files

To construct an assembly or a sub-assembly, you use an assembly file; the file extension is .iam. An assembly file links to a set of parametric 3D solid parts and/or sub-assembly of parametric 3D solid parts. It stores only the information on how the component parts are assembled together and information regarding the details of the parametric 3D solid parts is stored in the corresponding part files. Each time you open an assembly file, information from the part files is retrieved.

## Presentation Files

To construct an exploded presentation of an assembly or animate the exploded presentation, you use a presentation file; the file extension is .ipn. A presentation file links to an assembly file. It stores the information on how the parts of the assembly are tweaked apart. Details regarding how the component parts are assembled are stored in the respective assembly file.

## Drawing Files

To construct a 2D engineering drawing of a parametric 3D solid part, an assembly of 3D solid parts, and an exploded view of an assembly, you use a drawing file; the file extension is .idw. A drawing file links to a part file, an assembly file, or a presentation file. It stores the information about the 2D presentation of 3D objects.

## File Icons

To depict the four kinds of files, different icons are used. See Figure 1–6.



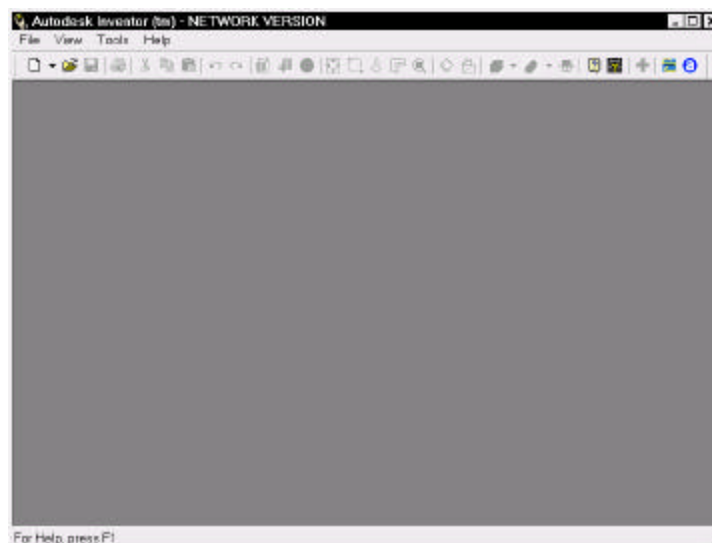
**Figure 1–6**

*Icons (from left to right) for part file, assembly file, presentation file, and drawing file*

---

## APPLICATION WINDOW

Now select the Cancel button of the Open dialog box. This brings you to the Inventor application window. (See Figure 1–7.)



**Figure 1–7**

*Application window*

In the application window, there are five major areas. At the top of the window, there is the standard Windows title bar which displays the name of the application. Below the title bar, is a set of pull-down menus, and below the menus is the Standard toolbar (Figure 1–8). Below the toolbar is a graphics area, and at the bottom of the window, there is a status bar.



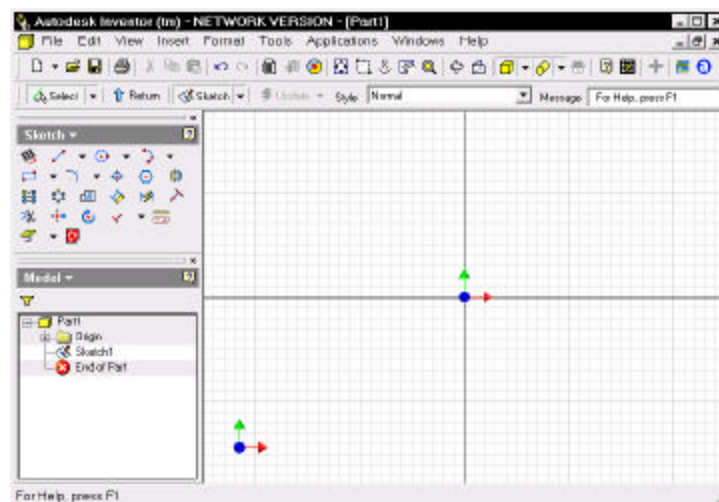
**Figure 1–8**  
*Standard toolbar*

## Starting A New File

Having decided what to do (construct a part, assembly, presentation, or drawing file), you start a new file by selecting New from the File pull-down menu. In the Open dialog box, select New from the What To Do panel. Then you will find three tabs: Default, English, and Metric. In each tab, there are a number of template files. The English tab has English templates; the Metric tab has metric templates (including BSI, DIN, GB, ISO, and JIS standards); and the Default tab has templates configured for the default measurement system you selected when you installed Inventor. Select the Default tab. You will find five template icons: Sheet Metal.ipt, Standard.iam, Standard.idw, Standard.ipn, and Standard.ipt.

### New Part File

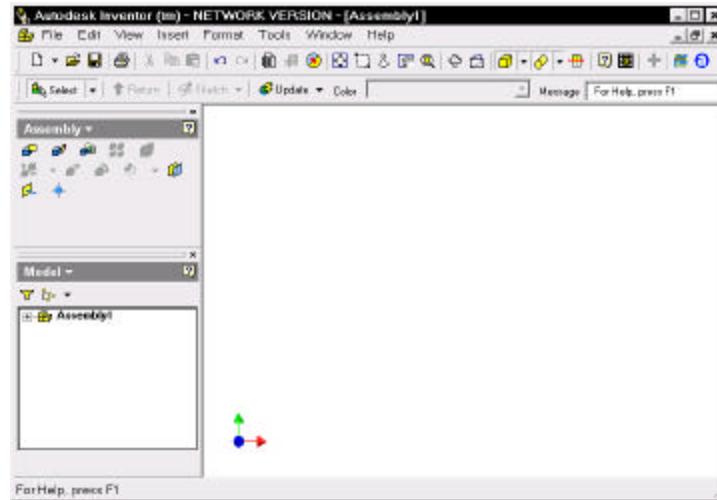
There are two part file templates, Sheet Metal.ipt and Standard.ipt. Figure 1–9 shows the application window for constructing a solid part or a sheet metal part.



**Figure 1–9**  
*Application window for a 3D solid part file*

## New Assembly File

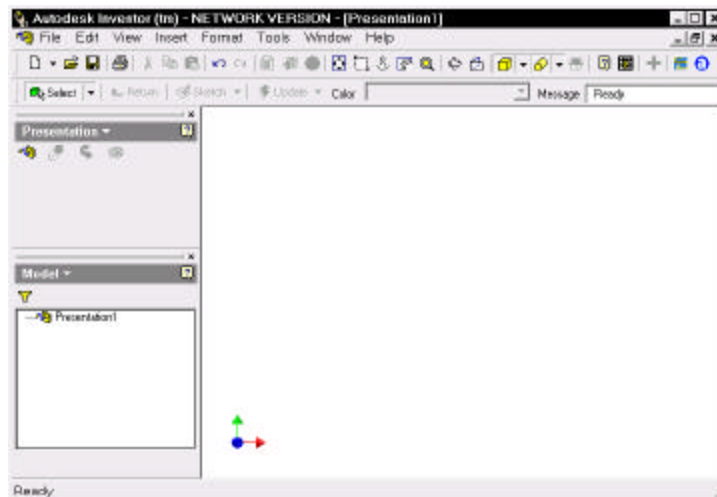
Select the Standard.iam template to construct an assembly of parts. Figure 1–10 shows the application window for constructing an assembly.



**Figure 1–10**  
*Application window for an assembly file*

## New Presentation File

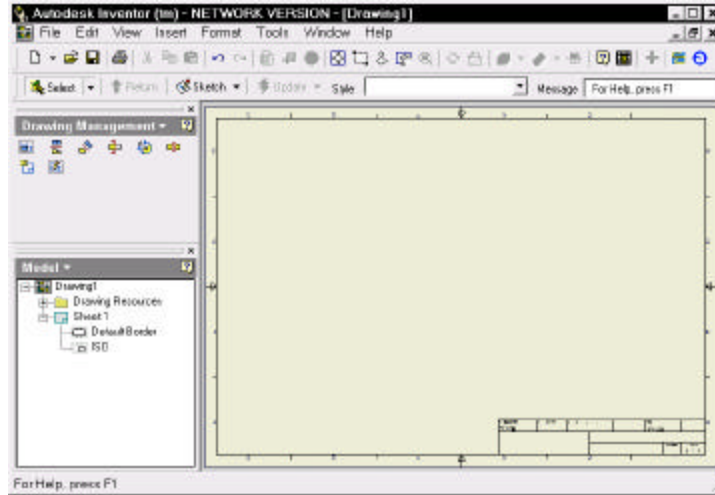
Select the Standard.ipn template to construct a presentation of assemblies. Figure 1–11 shows the application window for constructing a presentation of an assembly.



**Figure 1–11**  
*Application window for a presentation file*

## New Engineering Drawing File

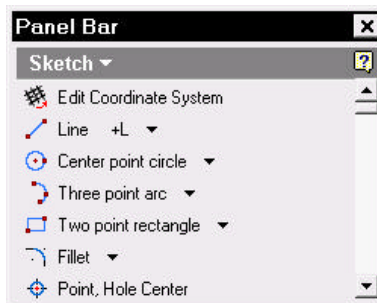
Select the Standard.idw template to construct an engineering drawing. Figure 1–12 shows the application window for constructing an engineering drawing.



**Figure 1–12**  
*Application window for a drawing file*

## Panel Bar and Browser Bar

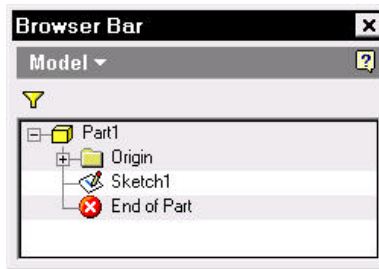
Common to the four application windows, there are three additional window areas: the panel bar and the browser bar. The panel bar has a number of palettes that enable you to access various design tools. The panel bar is context sensitive, so the tools available will vary according to current design context. In each palette of the panel bar, there are two modes available: general mode and expert mode. In general mode, there is text accompanying each icon to depict the command. In expert mode, text is not displayed until you place the cursor on the icon. Figure 1–13 shows the Sketch panel bar in general mode.



**Figure 1–13**  
*Sketch Panel bar*

The browser bar shows a hierarchy of objects in the file. Figure 1–14 shows the browser bar for a part file. To increase the graphics area, you can close the panel bar and the browser bar.

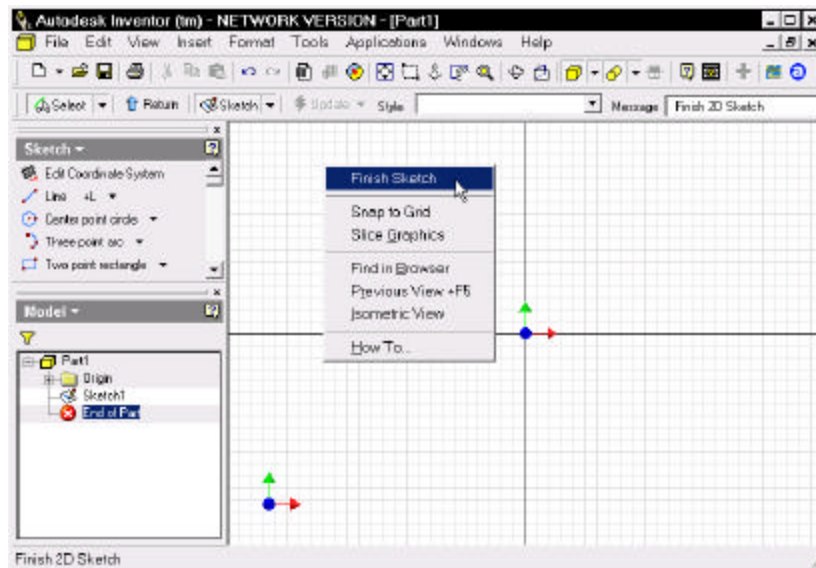




**Figure 1–14**  
*Browser bar*

## Right-Click Menus

Normally, your mouse has two buttons. You use the left button to select an object and use the right mouse button to activate a shortcut menu. Depending on the location of the mouse cursor and the kind of file you are working on, right-clicking the mouse will bring up different kinds of shortcut menus—they are context sensitive. You select an appropriate command from it. You will use the right-click button very often in the following chapters while you work through the tutorials. Figure 1–15 shows a right-click menu when you place your mouse on the graphics area of a part file.



**Figure 1–15**  
*Context sensitive right-click menu*

# Autodesk Inventor Tutorial 2

## Introduction to Part Modeling

Ron K C Cheng

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### PARAMETRIC FEATURE-BASED SOLID MODELING CONCEPTS

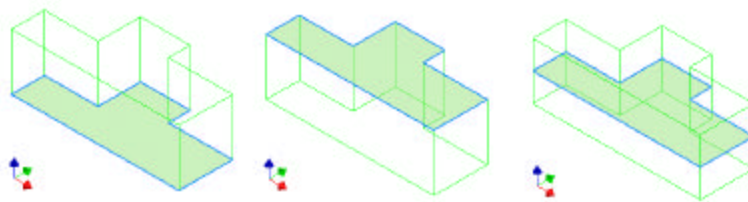
Autodesk Inventor is a parametric feature-based solid modeling application. Parameters of all objects are modifiable. To construct a 3D solid model, you think about how to decompose it into features, construct the features one by one, and combine them as you construct them. To construct the solid features, you either make sketches or select a pre-constructed solid feature from the menu.

#### Sketched Solid Features

Sketched solid features derive from sketches. There are four basic kinds of sketched solid features: extruded, revolved, loft, and sweep. In addition, there are other kinds of sketched features: coil, rib, and split.

##### Extruded Feature

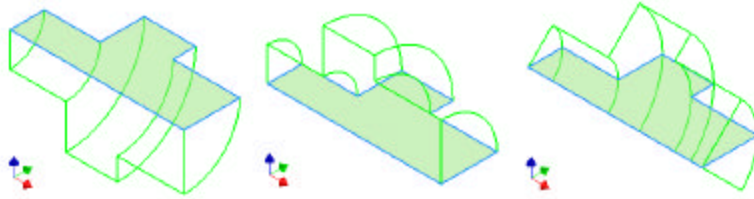
To make an extruded solid feature, you make a sketch and extrude the sketch in a direction perpendicular to the plane of the sketch. (See Figure 2–1.) You extrude the sketch in either direction or from mid-plane.



**Figure 2–1**  
*Sketch extruded in either direction and from mid-plane*

##### Revolved Feature

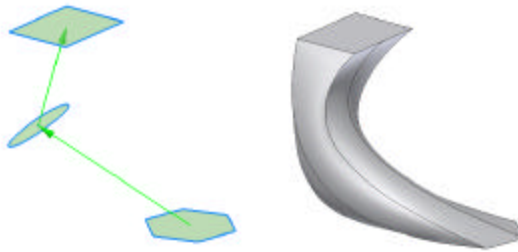
To make a revolved solid feature, you make a sketch and revolve it about an axis in either direction or from mid-plane. (See Figure 2–2.)



**Figure 2–2**  
*Sketch revolved in either direction and from mid-plane*

## Loft Feature

A loft solid feature builds on multiple sketches. It has a variable cross section defined by two or more sketches residing on different sketch planes. To construct a loft solid, you construct two or more sketches on the different sketch planes, and loft from one sketch to another sketch. Figure 2–3 shows a set of sketches and a loft solid constructed from the sketches.



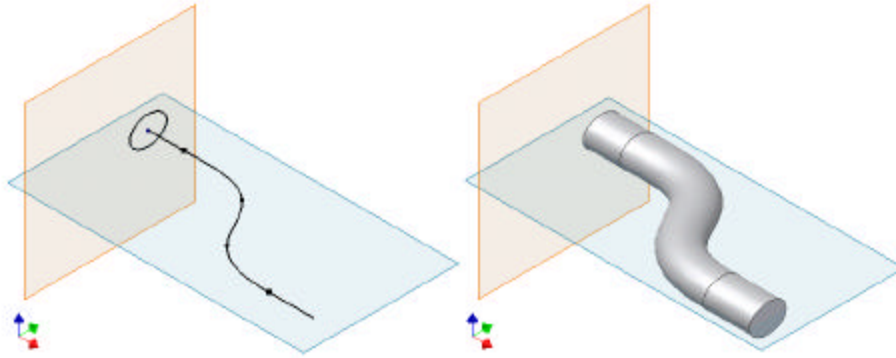
**Figure 2–3**  
*Sketches constructed on three sketch planes and loft solid feature constructed from the sketches*

## Sweep Feature

Construction of a sweep solid feature requires two sketches: a profile sketch and a path sketch. The sweep solid's volume is defined by sweeping the profile sketch along the path sketch. There are two kinds of path sketch: 2D path sketch and 3D path sketch. Sweeping a profile sketch along a 2D path sketch, you construct a 2D sweep solid feature. Sweeping a profile sketch along a 3D path sketch, you get a 3D sweep solid feature.

### 2D Sweeping

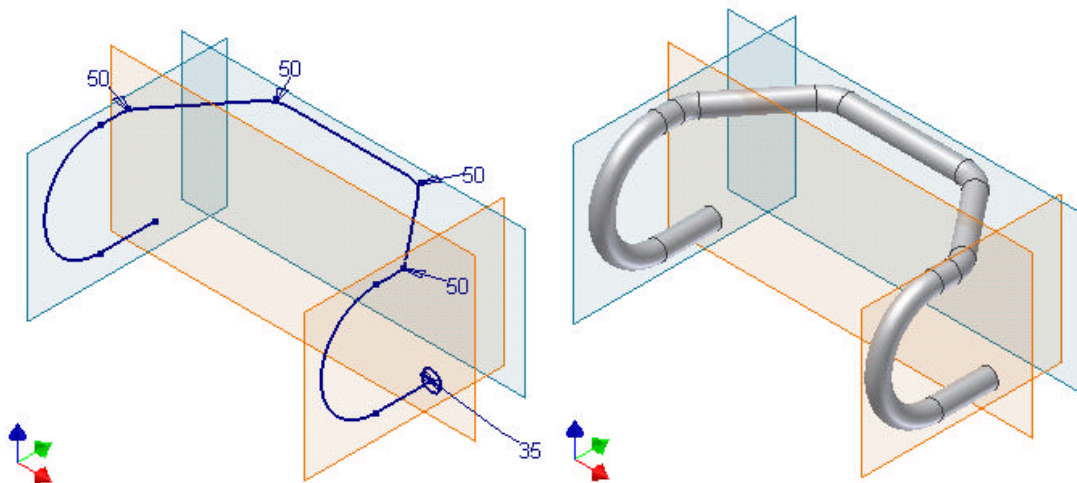
In a 2D sweep feature, both the profile sketch and the path sketch are 2D. They lie on planar sketch planes. Figure 2–4 shows a 2D profile and a 2D path (left) and a 2D sweep solid derived from the sketches (right).



**Figure 2-4**  
*Profile sketch and 2D path sketch (left) and 2D sweep solid (right)*

### 3D Sweeping

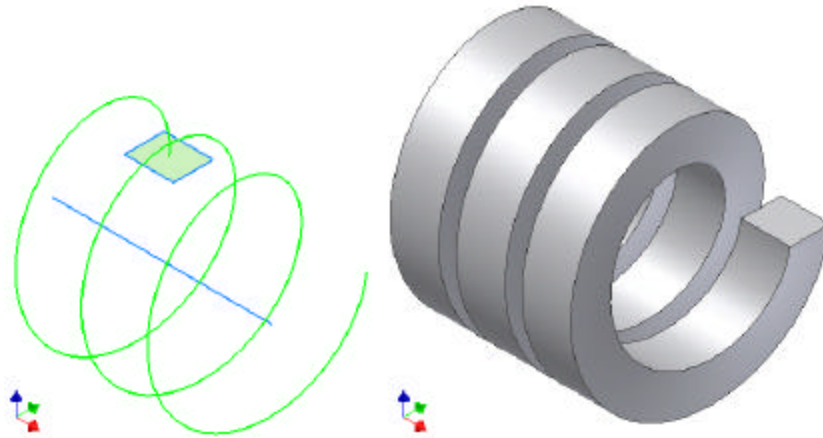
Construction of 2D and 3D sweep solids is similar, both require a profile sketch to depict the cross section of the sweep solid and a path sketch to specify the path of the sweep operation. The difference between them lies on the shape of the sweep path. In a 2D sweep solid feature, the path sketch is planar. In a 3D sweep solid feature, the path is three dimensional. Figure 2-5 shows a profile sketch and 3D path sketch together with a 3D sweep solid.



**Figure 2-5**  
*Profile sketch and 3D path sketch (left) and 3D sweep solid (right)*

### Coil Feature

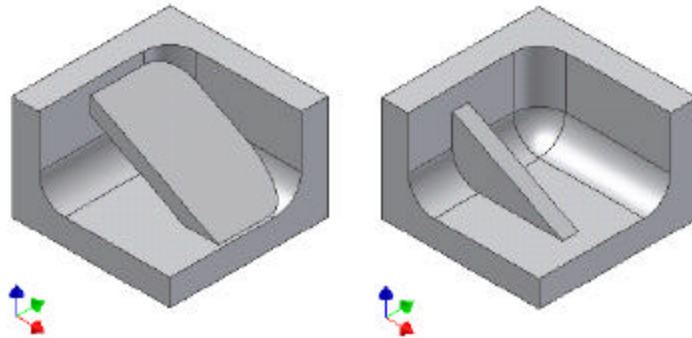
A coil solid is a special kind of 3D sweep solid in which the profile sketch is swept along a helical path. To make a coil solid, you construct a 2D sketch to depict the cross section of the coil, specify an axis, and specify the parameters of the helix. The axis can be a line, an edge, or a work axis. (See Figure 2-6.)



**Figure 2-6**  
*Sketch and coil feature*

### Rib Feature

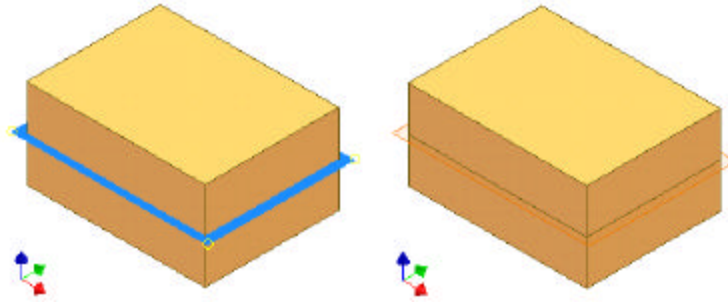
A rib is a triangular or rectangular reinforcing element you add to a component to strengthen it. (See Figure 2-7.) In essence, a rib is a special kind of extruded solid feature. To construct a rib, you construct an open-loop sketch profile and use the body of the solid to form a close loop in extruding.



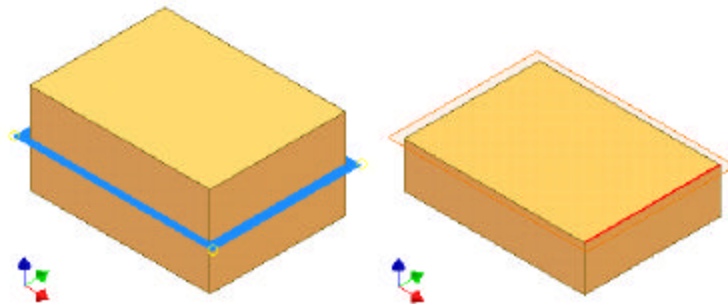
**Figure 2-7**  
*Two kinds of rib*

### Split Features

There are two kinds of split features: face split and part split. Using a split tool, a face split feature splits the faces of a solid part into two pieces, and a part split feature splits a solid part into two solids and removes one of them. You can use a work plane, a sketch, or a surface as split tool. Figure 2-8 shows a face split feature and Figure 2-9 shows a part split feature. Both of them use a sketch as a split tool.



**Figure 2–8**  
*Work plane used as split tool (left) and faces split (right)*



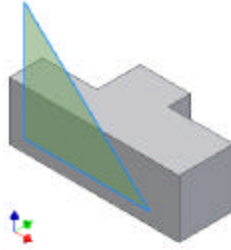
**Figure 2–9**  
*Work plane used as split tool (left) and part split (right)*

### Sketched Surface Features

Using the four basic sketched feature construction methods (extruding a sketch, revolving a sketch, lofting two or more sketches, and sweeping a sketch along another sketch), you can construct surfaces as well as solids. The surfaces that you construct are: extruded surface, revolved surface, loft surface, and sweep surface. To construct surfaces, you select the Surface button from the feature construction dialog box. You can use surfaces as split tool to split a part and as termination objects when constructing an extruded or revolved solid feature.

### Sketch Planes

To construct a sketched solid feature, you start with a sketch. To construct a sketch, you need a sketch plane. There are three default planes where you specify a sketch plane: XY plane, XZ plane, and YZ plane. In addition, you can use any existing planes of a solid feature or use artificial planes established by using work features. Figure 2–10 shows a sketch constructed on a face of a solid part.



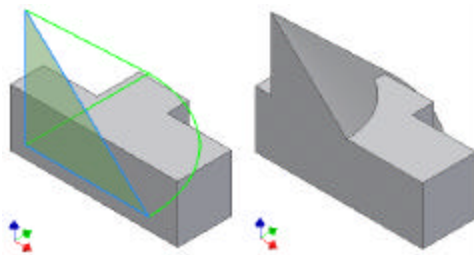
**Figure 2–10**  
*Sketch plane established on the face of a solid*

## Boolean Operations

The first sketched solid feature you construct in a 3D solid part is called the base solid feature. Using feature-based approach to construct a complex solid part, you construct additional features and combine them, one by one, with the existing solid by using one of the three kinds of Boolean operations: join, cut, or intersect.

### Join

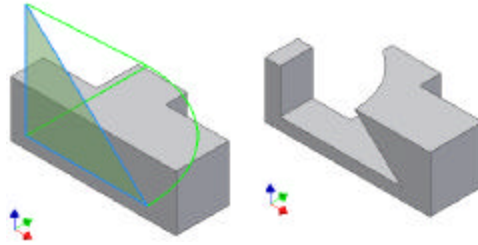
Using the join operation, you join the new sketched solid feature with the existing solid. The resulting solid has the volume enclosed by the new solid feature and the existing solid. Figure 2–11 shows a revolved solid feature joined to an extruded solid.



**Figure 2–11**  
*New sketched solid feature joined to the existing solid*

### Cut

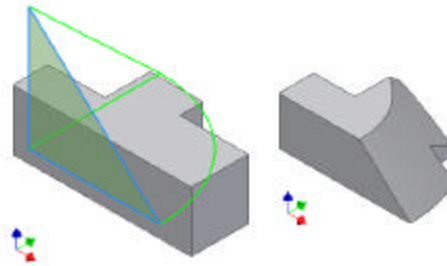
Using the cut operation, you cut the new sketched solid feature from the existing solid. The resulting solid has the volume of the new sketched solid feature removed from the existing solid. Figure 2–12 shows a revolved solid feature cut from an extruded solid.



**Figure 2-12**  
*New sketched solid feature cut from the existing solid*

## Intersect

Using the intersect operation, you intersect the new sketched solid feature with the existing solid. The resulting solid has a volume that contains the portion common to both the new sketched solid feature and the existing solid. Figure 2-13 shows a revolved solid feature intersecting with an extruded solid.



**Figure 2-13**  
*New sketched solid feature intersected with the existing solid*

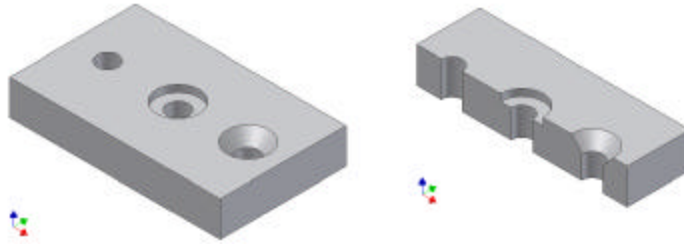
## Placed Solid Features

Apart from constructing features from sketches, you can construct a feature by selecting a shape from the menu and specify the parameters. We call these pre-constructed solid shapes the placed solid features. Autodesk Inventor has nine kinds of placed solid features: hole, thread, shell, fillet, chamfer, rectangular pattern, circular pattern, mirror, and face draft features.

### Hole Feature

A hole is a circular feature cut on a solid. While cutting a hole, you can tap the hole to incorporate an internal thread. To construct a hole, you place a hole feature by specifying type, size, and location. Figure 2-14 shows three hole features placed on a solid.

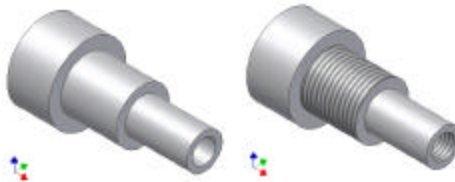




**Figure 2-14**  
An outside view and a cutaway view of a rectangular solid with three hole features

### Thread Feature

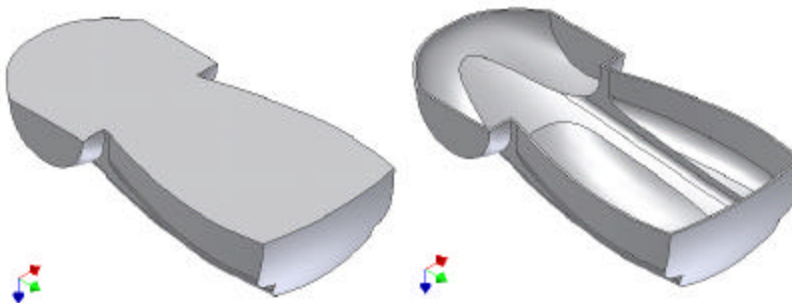
To construct a thread feature, you select a circular feature and specify the pitch and length of the thread. Figure 2-15 shows two thread features placed on the external and an internal circular features of a solid.



**Figure 2-15**  
Thread features placed on circular faces of a solid part

### Shell Feature

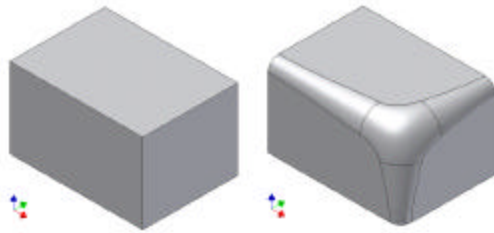
To make a solid hollow, you place a shell feature and state the thickness of the shell. Figure 2-16 shows a shell feature placed on a solid part, making the solid hollow.



**Figure 2-16**  
Solid part made hollow

### Fillet Feature

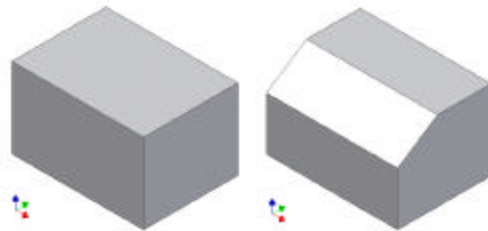
A fillet feature rounds off the edges of a solid; you select edges and specify fillet radii. Figure 2-17 shows a fillet feature placed on a solid, rounding off three edges and a corner.



**Figure 2–17**  
*Fillet feature placed on a solid*

### Chamfer Feature

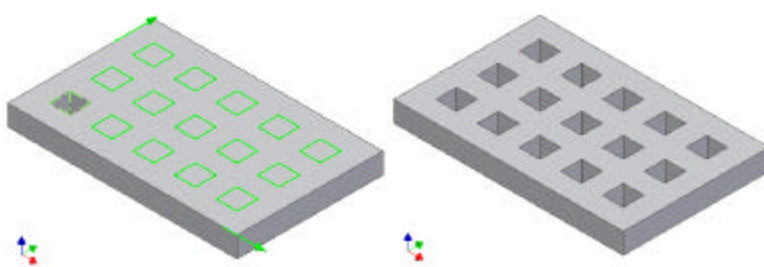
A chamfer feature bevels the edges of a solid; you select edges and specify bevel distances or the bevel distance and bevel angle. Figure 2–18 shows a chamfer feature placed on a solid.



**Figure 2–18**  
*Chamfer features placed on a solid*

### Rectangular Pattern

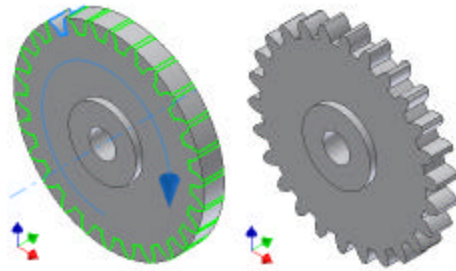
To repeat a solid feature in a rectangular array, you select the feature and specify the directions, distances, and the number of repetitions. Figure 2–19 shows a rectangular slot repeated in a rectangular array.



**Figure 2–19**  
*Rectangular pattern feature*

### Circular Pattern

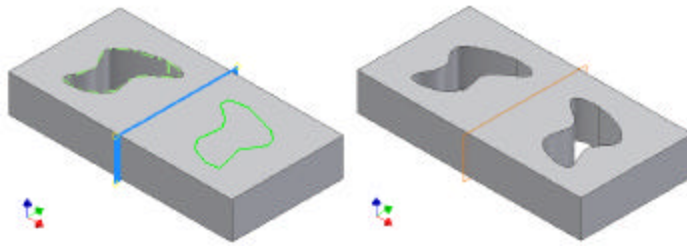
To repeat a solid feature in a circular array, you select the feature and specify an axis, angular distance, and number of repetitions. Figure 2–20 shows a gear tooth repeated in a circular array.



**Figure 2–20**  
*Circular pattern feature*

### Mirror Feature

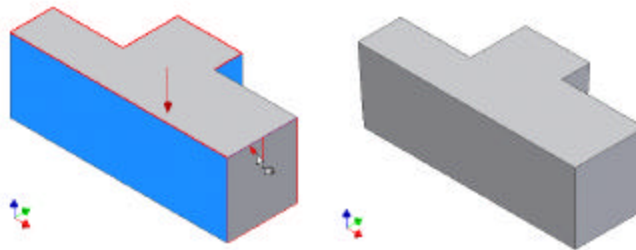
To construct a mirror copy of a solid feature, you select a feature and specify a mirror plane. Figure 2–21 shows a shaped slot mirrored about an artificial work plane.



**Figure 2–21**  
*Mirror feature placed*

### Face Draft Feature

To taper the faces of a solid, you place a face draft feature. You select an edge or a split line and specify a draft angle. Figure 2–22 shows a face draft feature placed along the vertical faces of a solid part.



**Figure 2–22**  
*Face draft feature placed along the edges of a solid*

By now, you should have a general picture of two kinds of sketched solid features (extruded and revolved) and various kinds of placed solid features (hole, thread, shell, fillet, chamfer, rectangular pattern, circular pattern, mirror, and face draft) that you use in making a solid model.

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## **WORK FEATURES**

To help establish sketch plane and references for solid modeling, you construct artificial planes, axes, and points. They are work points, work axes, and work planes. Collectively, we call them work features.

### **Features in a Solid Part**

Together with sketched solid features and placed solid features, there can be three kinds of features in a solid part. They are:

1. Sketched Solid Features
2. Placed Solid Features
3. Work Features

# Autodesk Inventor Tutorial 3

## Assembly Modeling

*Ron K C Cheng*

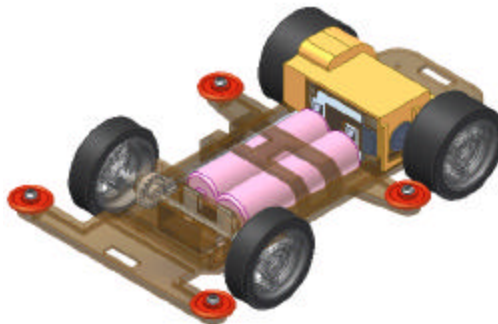
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### Assembly Modeling Concepts

With the exception of very simple objects, such as a ruler, most objects have more than one part put together to form a useful whole. The set of parts put together is called an assembly. When you design the parts for an assembly, the relative dimensions and positions of parts, and how they fit together, are crucial. You need to know whether there is any interference among the mating parts. If there is interference, you need to find out where it occurs; then you can eliminate it. Apart from fitting the parts together, you also need to validate relative motions and check clearances if there are moving parts in the assembly. Moreover, you should critically evaluate the parts and the assembly as a whole to ensure the assembly functions correctly in accordance with the design intent. To shorten the design lead time, you construct virtual assemblies in the computer to validate the integrity of the parts and the assembly.

### Components

For complex devices that have many parts, it is common practice to organize the parts into a number of smaller sub-assemblies such that each sub-assembly has fewer parts. Therefore, an assembly set consists of an assembly file and a number of part files or an assembly file together with a number of sub-assembly files and part files. Collectively calling the individual parts or sub-assemblies as components, you can define an assembly in the computer as a data set containing information about a collection of components linked to the assembly and how the components are assembled together. Figure 3–1 shows an assembly of a model car. It is a set of components.



**Figure 3–1**  
*An assembly of components*

## Part File and Assembly File

To construct an assembly in the computer, you start a new assembly file and connect a set of relevant part files and/or assembly files (sub-assemblies) to the assembly file. In the computer, the part files store the information about the 3D objects and the assembly files store the information about how you assemble the 3D objects together. (See Table 6–1.) Because the part files hold the definitions of the solid parts and link to the assembly file, any change you make to the part files will be incorporated in the assembly automatically.

**Table 6–1 Information**

File	Information
Part File	Definition and information of individual 3D solid part
Assembly File	Information on the locations of the linked components and how the linked components are assembled together

## Construction of an assembly

Construction of an assembly in the computer involves two major tasks: gathering a set of components in an assembly file and assembling the components by applying appropriate assembly constraints. How you gather a set of components in an assembly depends on which design approach you take and how you apply assembly constraints depends on the shapes of the features of the components in the assembly.

## Design Approaches

There are three design approaches: bottom-up, top-down, (bottom being the parts, and top being the assembly) and hybrid. In a bottom-up approach, you construct all the component parts and then assemble them in an assembly file. In a top-down approach, you start an assembly file and construct the individual component parts while you are doing the assembly. The hybrid approach is a combination of the bottom-up and top-down approaches.

### The Bottom-Up Approach

When you already have a good idea on the size and shape of the components of an assembly or you are working as a team on an assembly, you use the bottom-up approach. Through parametric solid modeling methods, you construct all the parts to appropriate sizes and shapes that best describe the components of the assembly. Then you start an assembly file and place the parts in the assembly. In the assembly, you align the components together by applying assembly constraints. After putting all the parts together, you analyze and make necessary changes to the parts.

### The Top-Down Approach

Sometimes you have a concept in your mind, but you do not have any concrete ideas about the component parts. You use the top-down approach—you start an assembly file and design some component parts there. From the preliminary component parts, you improvise. The main

advantages in using this approach are that you see other parts while working on an individual part and you can continuously switch from designing one part to another.

### The Hybrid Approach

In reality, you seldom use one approach alone. You use the bottom-up approach for standard component parts and new parts that you already know about and you use the top-down approach to figure out new component parts with reference to the other component parts. This combined approach is the hybrid approach.

## Part Modeling Mode and Assembly Modeling Mode

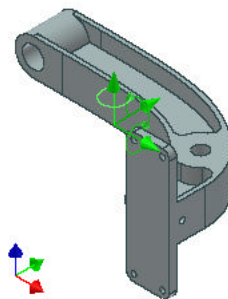
On matter which design approach you use to construct an assembly, you can switch from assembly modeling mode to part modeling mode (or vice versa) any time you feel appropriate. In assembly modeling mode, you use the assembly modeling tools to construct an assembly. (You will learn more about assembly modeling in this and the next chapter.) In part modeling mode while working in an assembly, the assembly model tools are replaced by the set of part modeling tool and the other component parts in the assembly are grayed out.

## Components in an Assembly

In an assembly file, the components are free to translate in three linear directions and three rotational directions. You move and rotate them as if you were manipulating a real object. To impose restriction to the movements and to align a component properly with another component in the assembly, you apply assembly constraints.

### Degrees of Freedom

Initially, the component parts (except the first component) that you place or create in an assembly are free to translate in the 3D space, in three linear directions and three rotational directions. These free translations are called the six degrees of freedom (DOF). The DOF of a component is represented by a DOF symbol (see Figure 3–2). You discover the DOF of the objects in an assembly by selecting Degrees of Freedom from the View menu.



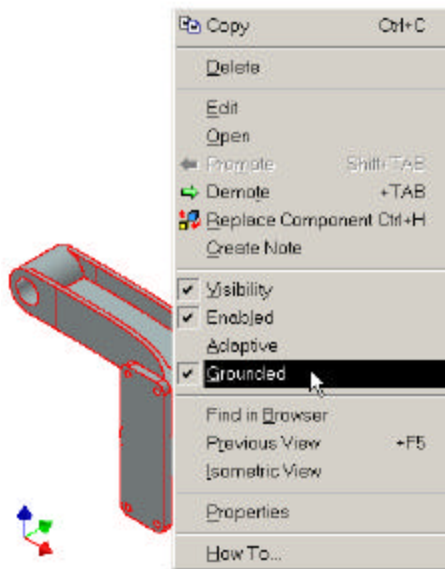
**Figure 3–2**  
*Six degrees of freedom of a free object*

## Grounding

By default, the first component part that you place or create in the assembly is fixed in the 3D space. We call a fixed object a grounded object, it has no degree of freedom—you cannot move it. To free a grounded object, you select the object, right-click, and deselect Grounded. (See Figure 3–3.) On the other hand, if you want to fix an object in 3D space, you ground the object by right-clicking and selecting Grounded.

## Translation of Objects in 3D Space

Unless a component part is grounded, it is free to translate in the 3D space. To translate the component parts in an assembly to their appropriate locations, you move or rotate them. Note that moving or rotating a component does not affect a component's DOF. You simply put it in a new position and new orientation.



**Figure 3–3**  
*Grounded selected*

## Applying Assembly Constraints

To restrict the movement of a component in 3D and to align it with another component in the assembly, you apply assembly constraints to selected faces, edges, and vertices of parts and the origin (with three work axes along the X, Y, and Z directions) of the part file or the sub-assembly file. There are four kinds of assembly constraints: mate, angle, tangent, and insert. The kind of constraint to be applied to a pair of components depends on the design intend of the assembly and the function and shape of the individual components.

### Mate Constraint

A mate constraint causes two selected objects (face, edge, or vertex) to mate or flush at a specified offset distance. You can mate a face to a face, an edge to an edge, a point to a point, an



edge to a face, a vertex to a face, and a vertex to an edge. If you mate a face to a face, you have to decide how the faces face each other, in the same direction or in the opposite directions.

By mating a face of a component with six DOFs to a face of another component, you remove three DOFs (two degrees of rotation freedom and one degree of linear freedom).

By mating an edge of a component with six DOFs to an edge of another component, you remove four DOFs (two degrees of rotation freedom and two degrees of linear freedom).

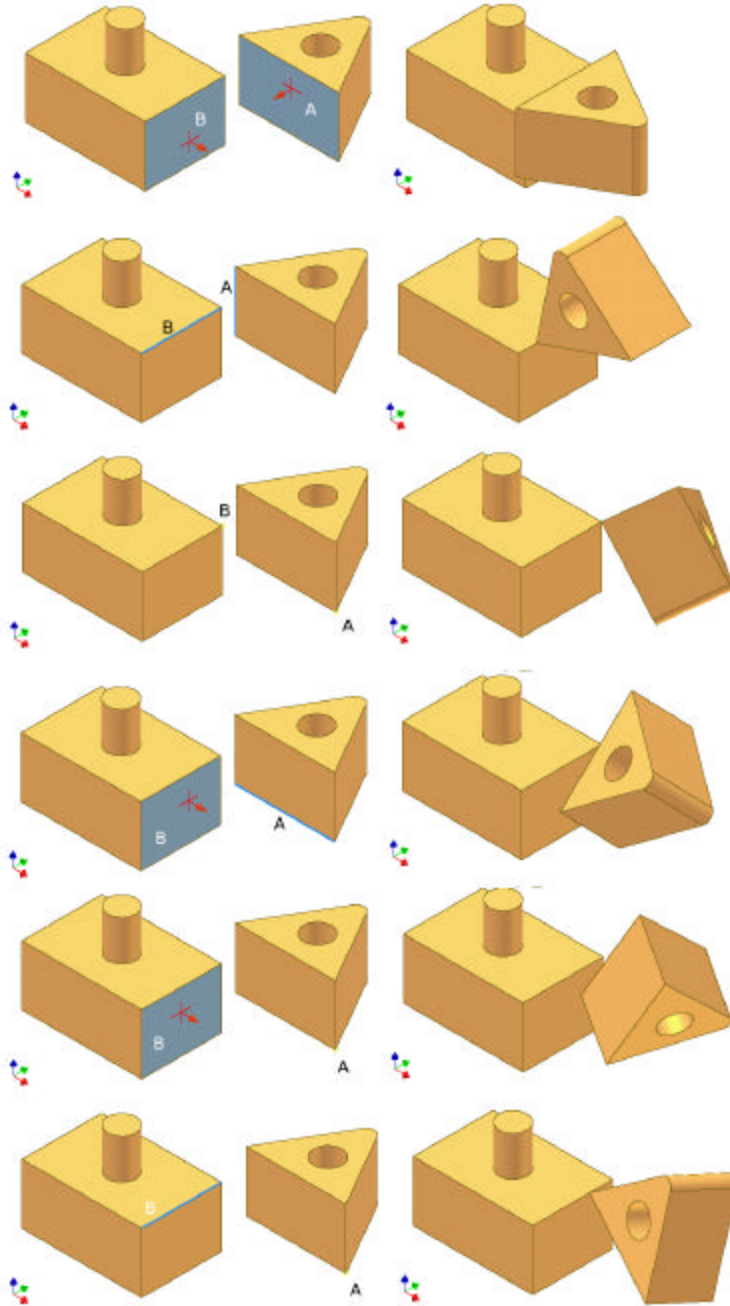
By mating a vertex of a component with six DOFs to a vertex of another component, you remove three DOFs (three degrees of linear freedom).

By mating an edge of a component with six DOFs to a face of another component, you remove two DOFs (one degree of rotation freedom and one degree of linear freedom).

By mating a vertex of a component with six DOFs to a face of another component, you remove one DOF (one degree of linear freedom).

By mating a vertex of a component with six DOFs to an edge of another component, you remove two DOFs (two degrees of linear freedom).

Figure 3–4 shows (from top to bottom) mating face A to face B, mating edge A to edge B, mating vertex A to vertex B, mating edge A to face B, mating vertex A to face B, and mating vertex A to edge B.



**Figure 3–4**  
*Mate constraints*

### **Angle Constraint**

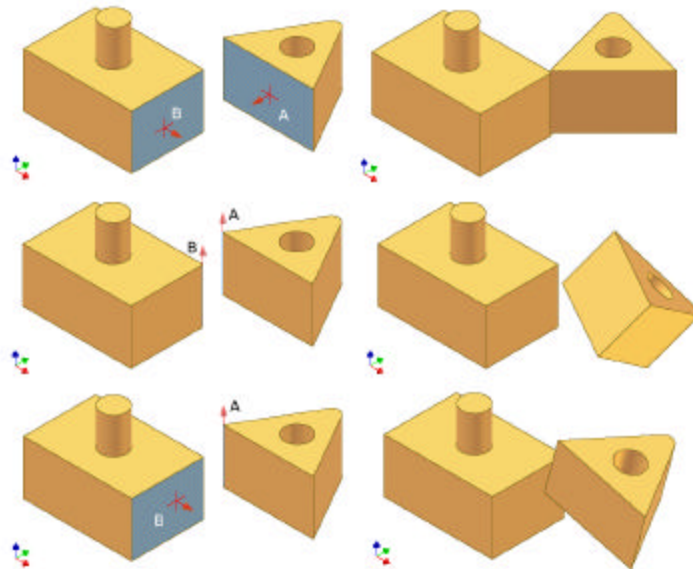
Angle constraint causes two selected objects (face or edge) to align at a specified angle. You can constrain a face at an angle with another face, an edge at an angle with another edge, and a face at an angle with an edge. You can consider an angle constraint a special kind of mate constraint because the angle constraint causes the constrained component to align at an angle.

By constraining a face of a component with six DOFs to a face of another component at an angle, you remove three DOFs (two degrees of rotation freedom and one degree of linear freedom).

By constraining an edge of a component with six DOFs to an edge of another component at an angle, you remove four DOFs (two degrees of rotation freedom and two degrees of linear freedom).

By constraining an edge of a component with six DOFs to a face of another component at an angle, you remove two DOFs (one degree of rotation freedom and one degree of linear freedom).

Figure 3–5 shows (from top to bottom) face A constrained at an angle with face B, edge A constrained at an angle with edge B, and edge A constrained at an angle with face B.



**Figure 3–5**  
*Angle constraints*

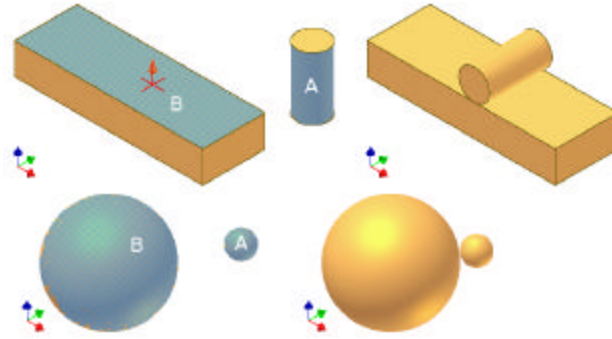
### Tangent Constraint

Tangent causes selected faces, planes, cylinders, spheres, and cones to contact at their tangential point and at a specified offset distance; you select faces.

By applying tangent constraint to a cylindrical face of a component with six DOFs to a face of another component, you remove two DOFs (one degree of rotation freedom and one degree of linear freedom).

By applying tangent constraint to a spherical face of a component with six DOFs to a face of another component, you remove one DOFs (one degree of linear freedom).

Figure 3–6 shows (from top to bottom) cylindrical face constrained to tangent with a planar face and a spherical face constrained to tangent with another spherical face.

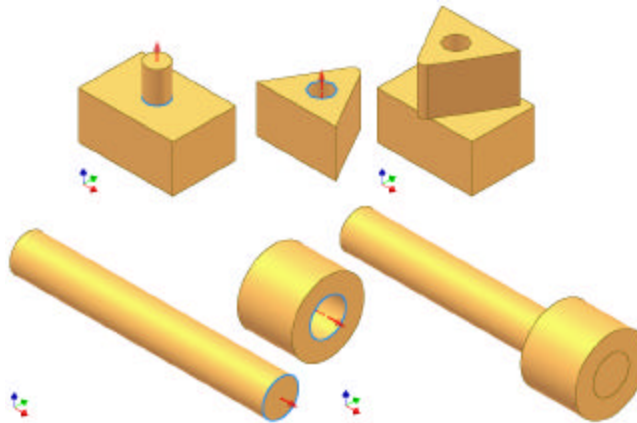


**Figure 3-6**  
*Tangent constraints*

### Insert Constraint

Insert causes selected circular edges to align face to face and concentrically at a specified distance; you select circular edges. An insert constraint is a combination of two mate constraints: mating an axis of a cylindrical object with the axis of another cylindrical object and mating the end face of the cylindrical object with the end face of another cylindrical object.

Figure 3-7 shows (from top to bottom) a cylindrical object inserted to another cylindrical object in an opposed direction and a cylindrical object inserted to another cylindrical object in an aligned direction.



**Figure 3-7**  
*Insert constraints*