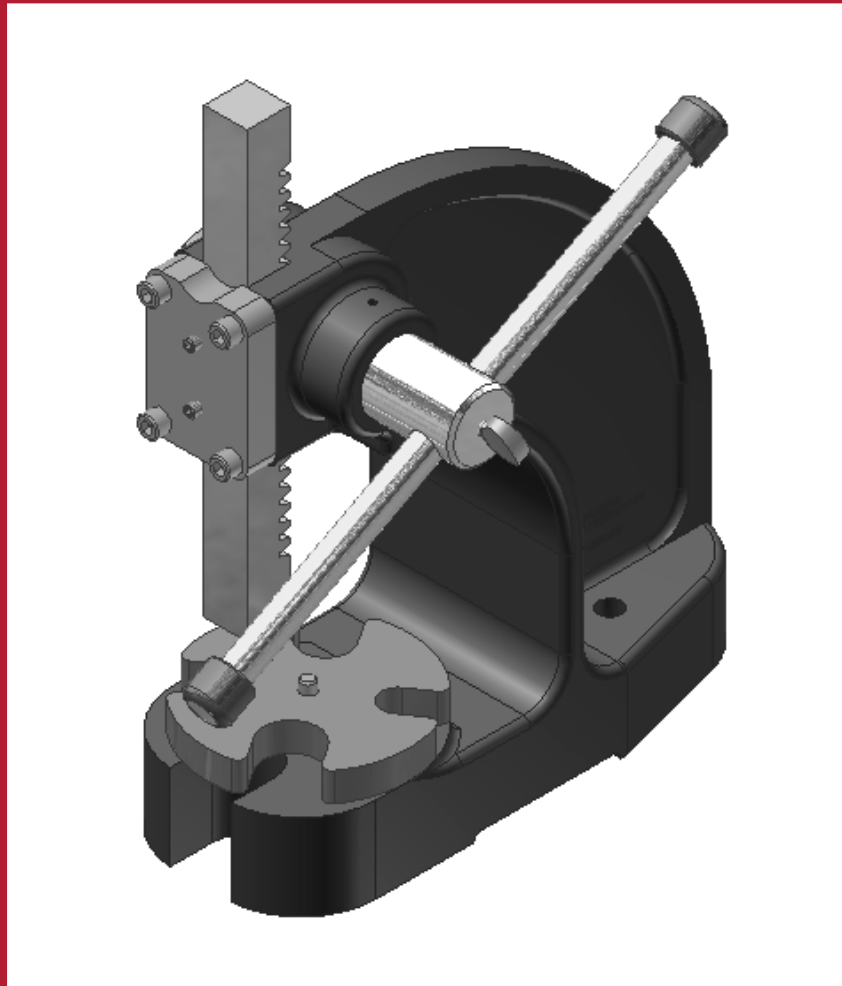


# ***The Power of Design: An Introduction to Autodesk Inventor 2008***



**Elise Moss**

**SDC**  
PUBLICATIONS

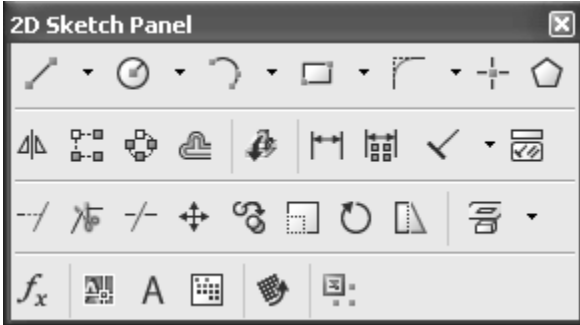
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### Lesson 3

## 2D Sketch Panel Tools



Inventor’s Sketch toolbar contains tools for creating the basic geometry to create features and parts.



On the surface, the Geometry tools look fairly standard: Line, Circle, Arc, Rectangle, Fillet/Chamfer, Point and Polygon.







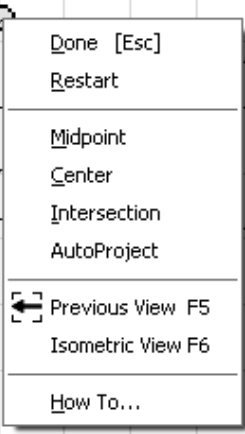
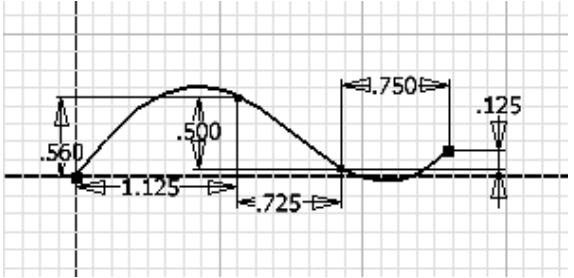
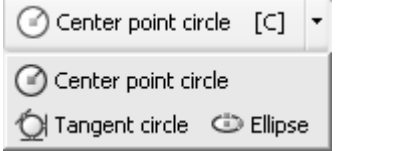

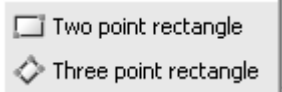

### Line/Spline

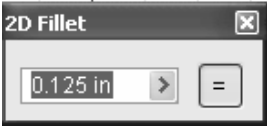
Let’s start with the **Line** tool. Its drop-down has two options: **Line** or **Spline**. Run the mouse over the button and look in the lower left-hand section of the screen; a help description will appear describing the tool function. In this case, the tool creates lines and tangent arcs. This means filleted corners can be created without having to exit the line mode and performing a fillet command.

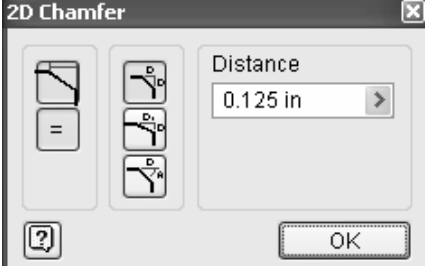



When you are in Sketch mode, these tools will appear on the Standard toolbar.

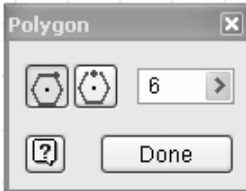









	To create a Construction line instead of an object line, toggle the <b>Construction Line</b> option on the Standard toolbar.
	To create a Center/Axis line, toggle the <b>Center Line</b> option tool.
	To create an <b>Object Line</b> , this tool is toggled.
	To add an <b>Associative/Driven Dimension</b> , toggle this tool.

	<p>Right-click the mouse while in 'Line' mode; this will bring up a submenu to assist in the construction of your sketch.</p> <p>To create a tangent arc while in 'Line' mode, select an endpoint and hold down the left mouse button. When the arc is located properly, release the left mouse button and Inventor will automatically return to 'Line' mode.</p>
	<p>The <b>Spline</b> tool allows you to add and remove points, change the fit, and adjust the shape.</p> <p>You can add dimensions to control the size of the spline and fully constrain it.</p>
	<p><b>Circle/Ellipse</b></p> <p>The default <b>Circle</b> tool creates a circle using <b>Center point</b> and <b>Radius</b>. Access the drop down toolbar to see that there are two other circle options: <b>Ellipse</b> and <b>Tangent, Tangent, Tangent</b>.</p>
	<p><b>Arc</b></p> <p>The default <b>Arc</b> tool creates an arc using three points. The drop down toolbar provides two additional options: <b>Center, Start, End</b> and <b>Start, Tangent</b>. All three methods will draw arcs either clockwise or counter-clockwise.</p>
	<p><b>Rectangle</b></p> <p>The <b>Rectangle</b> tool provides two options:</p> <ul style="list-style-type: none"> <li>• The default is to select the two opposite corners of the rectangle.</li> <li>• The second option has the user define the length of one side and then the length of the adjacent side.</li> </ul>
	<p><b>Fillet/Chamfer</b></p> <p>The <b>Fillet</b> tool is actually a flyout that includes <b>Fillet</b> and <b>Chamfer</b>. Inventor recommends that it is better to add fillets and chamfers as placed features. The reason is that the user can then suppress fillets for faster regens and to conserve memory. It also makes it easier to modify values. However, there are instances where it is preferable to include the fillet or chamfer in the sketch.</p>


	<p>The <b>Fillet</b> tool prompts the user to select the edges of the sketch to be modified and brings up a dialog box where the user can modify the radius value. To modify the value of a fillet you've already placed, just double-click using the left mouse button and a dialog box will pop up allowing you to edit the value. Pressing the equal button allows the user to select an existing fillet and apply that fillet's value to the fillet being defined.</p>
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	<p><b>Chamfer</b></p> <p>Chamfers can be defined in three ways: <b>Equal Distance</b>, <b>2 Distance</b>, and <b>Distance-Angle</b>.</p> <p>The user also has the option of selecting an existing chamfer in the sketch and applying that value to the chamfer being defined.</p>
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	<p><b>Point, Hole Center</b></p> <p>The <b>Point</b> tool is used to determine the location of holes as well as points.</p> <p>To create a <b>Sketch Point</b> (used to constrain geometry), select the Sketch Point under Style.</p>
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<p><b>Polygon</b></p> 	<table border="1"> <tr> <td data-bbox="755 976 885 1081">  </td> <td data-bbox="885 976 1421 1081"> <p><b>Inscribed</b> uses the vertex between two edges to determine the size and orientation of the polygon.</p> </td> </tr> <tr> <td data-bbox="755 1081 885 1186">  </td> <td data-bbox="885 1081 1421 1186"> <p><b>Circumscribed</b> uses the midpoint of an edge segment to determine the size and orientation of the polygon.</p> </td> </tr> <tr> <td data-bbox="755 1186 885 1291">  </td> <td data-bbox="885 1186 1421 1291"> <p>The <b>number dropdown</b> specifies the number of edges used to create the polygon shape. The maximum number allowed is 120.</p> </td> </tr> </table>		<p><b>Inscribed</b> uses the vertex between two edges to determine the size and orientation of the polygon.</p>		<p><b>Circumscribed</b> uses the midpoint of an edge segment to determine the size and orientation of the polygon.</p>		<p>The <b>number dropdown</b> specifies the number of edges used to create the polygon shape. The maximum number allowed is 120.</p>
	<p><b>Inscribed</b> uses the vertex between two edges to determine the size and orientation of the polygon.</p>						
	<p><b>Circumscribed</b> uses the midpoint of an edge segment to determine the size and orientation of the polygon.</p>						
	<p>The <b>number dropdown</b> specifies the number of edges used to create the polygon shape. The maximum number allowed is 120.</p>						

	<p>The next section of the Sketch toolbar contains <b>Pattern</b> tools: <b>Mirror</b>, <b>Rectangular Pattern</b>, <b>Circular Pattern</b> and <b>Offset</b>.</p>
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	<p><b>Mirror</b></p> <p>Use the <b>Mirror</b> tool on the Sketch toolbar to mirror sketch geometry across a centerline.</p> <p>Equal constraints are automatically applied to the mirrored geometry and source geometry. You can delete or edit segments after you mirror them and the remaining segments will retain their symmetry and constraints.</p>
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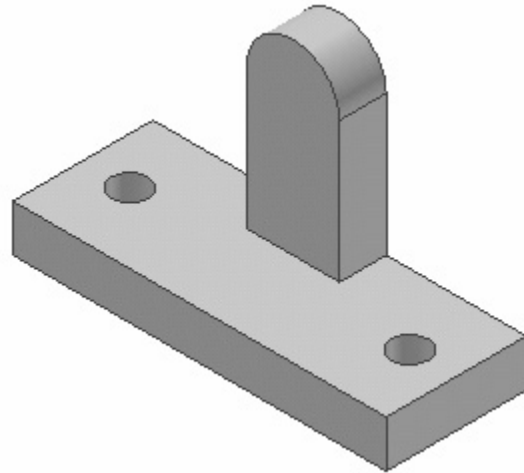
 <p><b>TIP:</b> You must keep the mirror axis selection separate from the geometry selection. If you accidentally include the mirror axis in the selection of geometry to be mirrored, you will get an error message.</p> <p>Use of Mirror when creating symmetric parts will use up less system resources.</p>
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### Exercise 3-1: Mirror

File: New (Standard using Inches)  
Estimated Time: 30 minutes

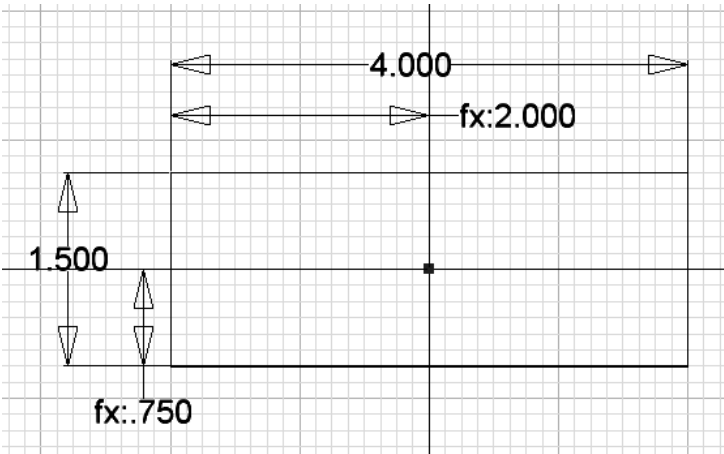
This lesson reinforces the following skills:

- ◆ Rectangle
- ◆ Project Geometry
- ◆ Dimension
- ◆ Extrude
- ◆ Redefine Sketch
- ◆ New Sketch
- ◆ Mirror
- ◆ Close Loop
- ◆ Show Dimensions
- ◆ Update



1.  Start a new file using Standard units.



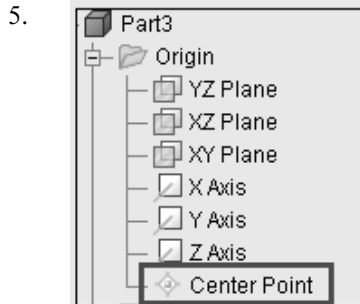
2.  Draw a 4 x 1.5 in. rectangle and center it using a projected center point.

First draw the rectangle using the rectangle tool.

To project the center point, select **Project Geometry** then select the **Origin** point in the Browser.

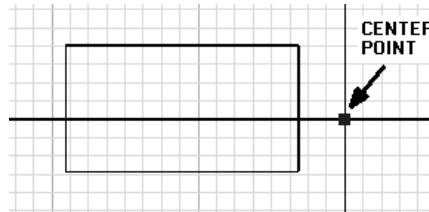
3.  Draw a **Rectangle** in the graphics window.


4.  Select the **Project Geometry** tool.

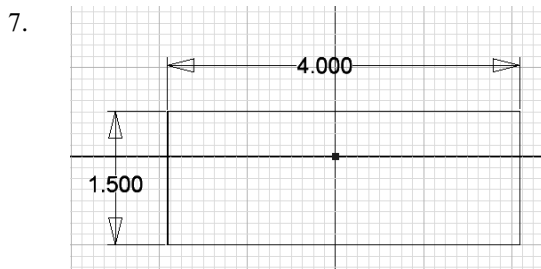


Left pick on the **Center Point** located in the Panel Browser.

This will project or add the center point to the active sketch.




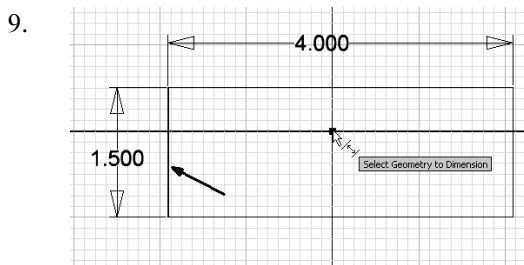
6.  Select the **Dimension** tool.



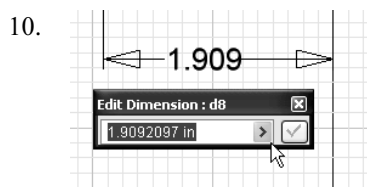
Add the **4.00** and **1.500** dimensions to the rectangle.

The size of the rectangle will automatically adjust to the new dimensions.

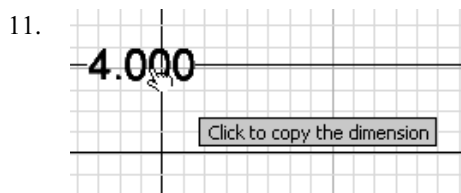
8.  Select the **Dimension** tool.



Select the left vertical line and the center point and place a horizontal dimension.



Erase the dimension shown by highlighting and pressing the backspace or delete button.



Left-click on the **4.00** dimension and that value will be copied into the edit box.



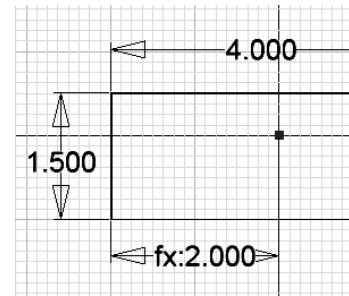
The dimension name will appear in the **Edit Dimension** box. (This is the name assigned by Inventor to keep track of dimensions.)

Add a 'divided by' symbol (/) and the number 2.

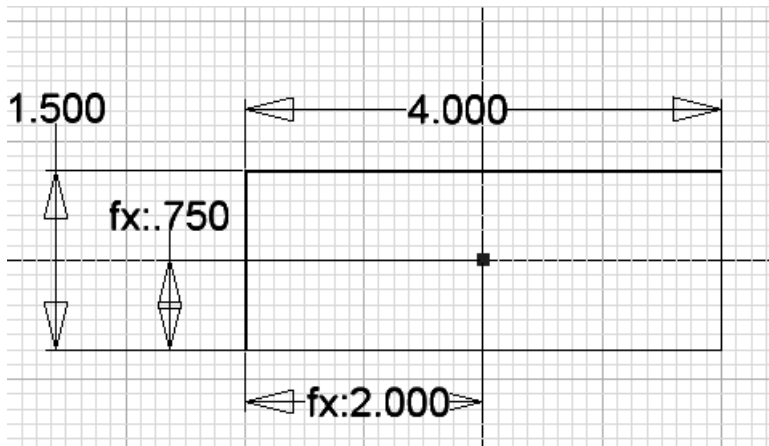
This will automatically center the rectangle in the horizontal direction.

Press the green check symbol on the dimension edit box to end.

13. **NOTE:** The *fx*: displayed on the dimension indicates that a formula was used to define the dimension.



14. Repeat for the horizontal direction to center the rectangle on the origin.



15. Right-click and select **Finish Sketch**.



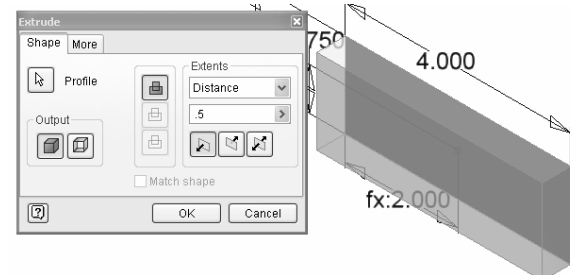
16. Press **F6** to switch to an Isometric View or right-click in the graphics window and select **Isometric View**.



17. Select the **Extrude** tool.



18. Extrude **0.5** inches.  
Set the Extents to **Distance**.  
Set the Distance value to **.5**.

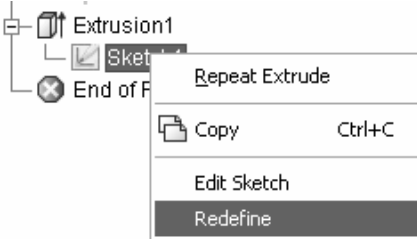


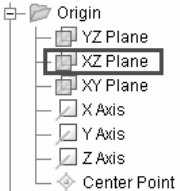
Press **OK**.

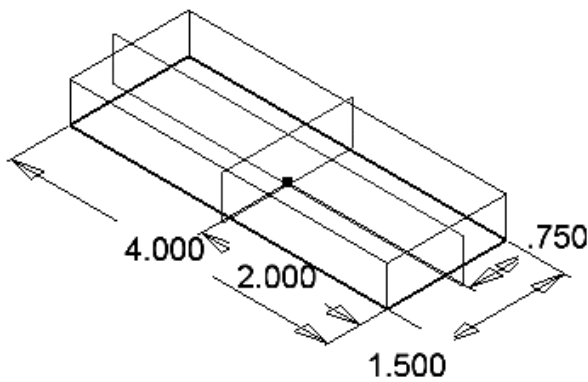
We would like to reorient our isometric view so the block is lying flat.  
We can do this two ways:

- We can redefine the isometric view, or
- We can redefine the sketch to a different workplane.

In this exercise, we will redefine the sketch.

19.  Highlight the sketch in the Browser. Right-click and select **Redefine**.

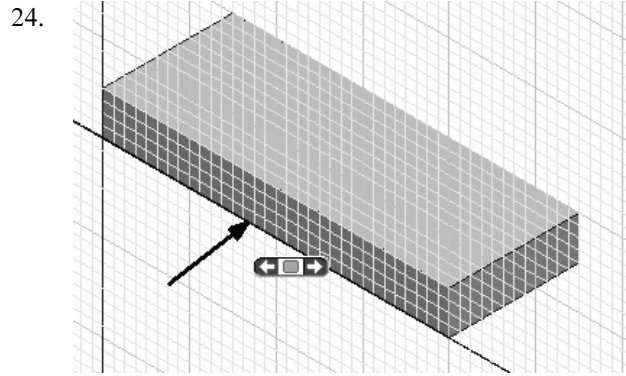
20.  Left pick on the **XZ Plane**.

21.  The box flips to the correct orientation.  
Note how it remains centered on the center point.

22.  Highlight the front face. Right-click and select **New Sketch**.

23.  Select **Project Geometry**.




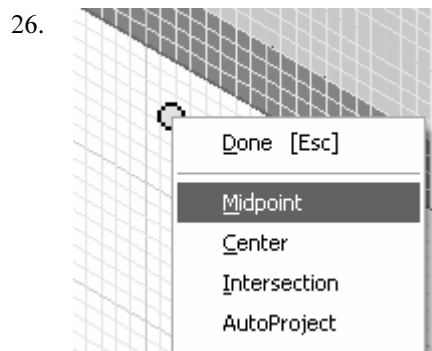


Select the bottom edge of the block.

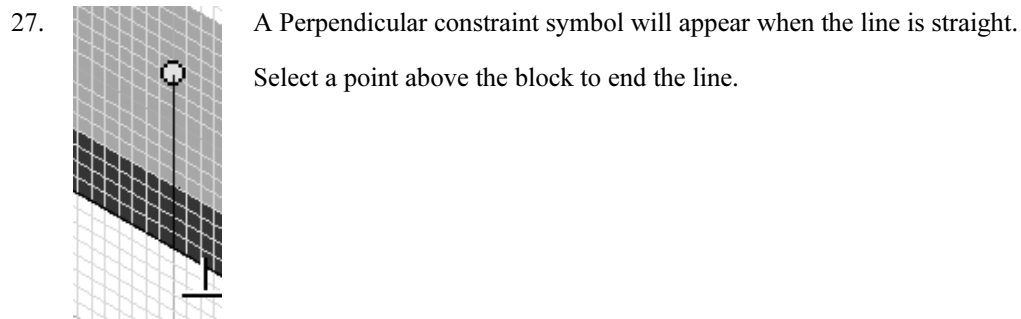
This copies a line using the edge into your active sketch.


In order to locate the midpoint of the bottom edge, you need to project the bottom edge.

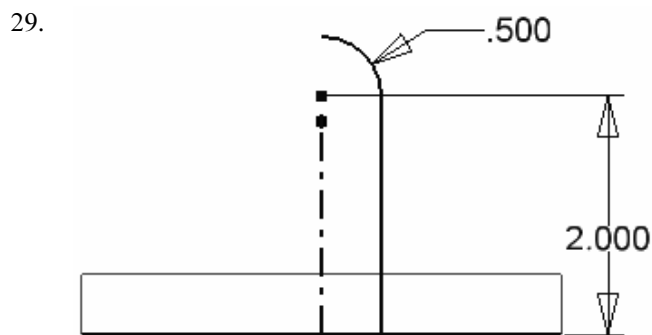
25.  Select the **Line** tool.



Draw a vertical line at the midpoint of the front side. Right-click to select **Midpoint** to have your mouse locate the midpoint. A green point will appear at the end of the cursor to indicate that the midpoint has been selected.



28.  Highlight the line and toggle **Centerline** on the Standard toolbar.



Create the sketch shown.


Draw a vertical line and an arc.



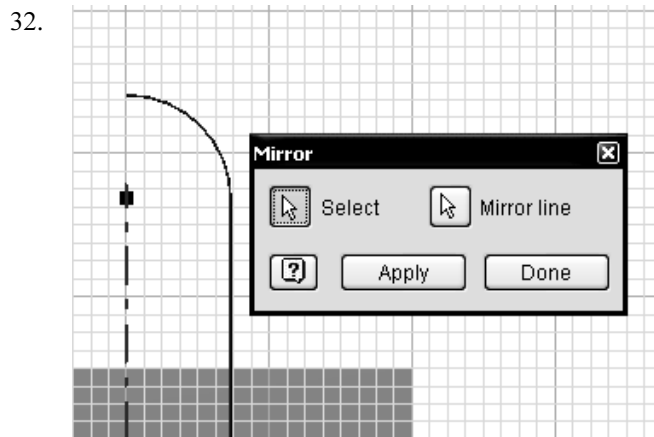
Add a **Coincident** constraint between the arc center and the centerline.



Add a **Vertical** constraint between the open arc end and the centerline.

30.  Select the **Mirror** tool.

31. Select the line and arc. You can do this using a window or by picking each object.



You may need to deselect the centerline to specify it as the **Mirror line**.

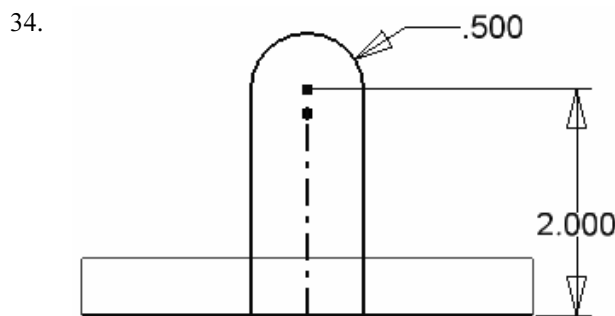
To deselect, press down the Control key then pick the centerline.

Select the **Mirror line** select button.

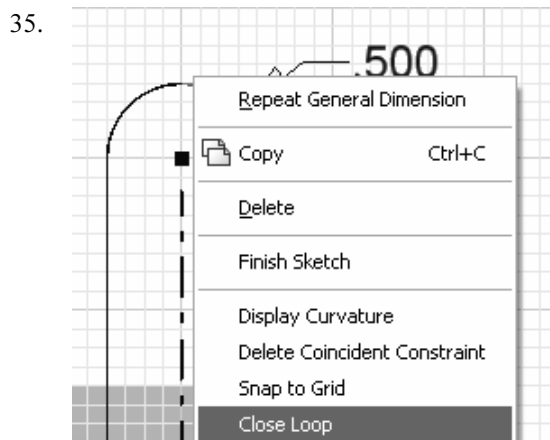
Then select the center line.

Once all the selections are done, press **Apply** and then **Done**.

33. Draw a horizontal line between the two vertical lines to create a closed profile.



Use the **Dimension** tool to add dimensions.

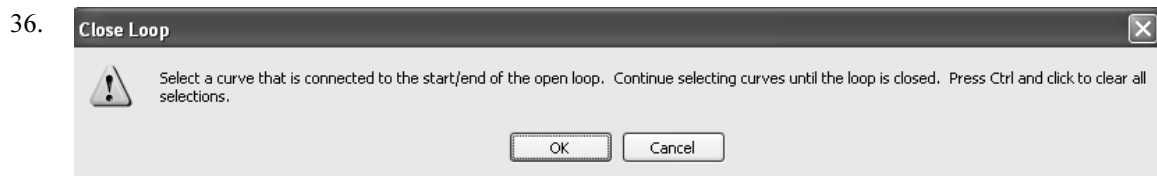


Our mirrored geometry does not define a closed loop.

Select the right arc of the sketch.

Right-click and select **Close Loop**.

Then select the remainder of the sketch.



Press **OK**.

Select the vertical line, horizontal line, left vertical line, and left arc, in order.

37.



As you select geometry, you will see messages to add Coincident constraints or to close gaps. When those messages appear, press **Yes**.

38.

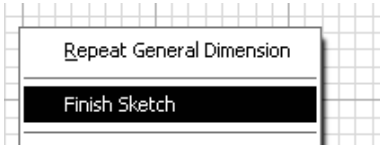


When you have fully defined your closed polygon, this message will appear.

**NOTE:** If you continue to get errors, select the bottom horizontal line and then the remaining geometry to form the closed polygon.

Press **OK**.

39.



Right-click and select **Finish Sketch**.

40.



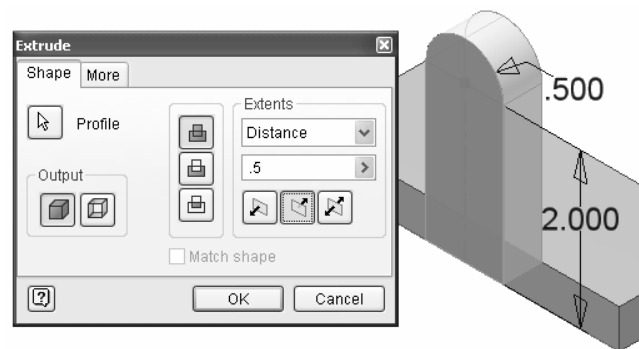
Press **F6** to switch to an Isometric View, or right-click in the graphics window and select **Isometric View**.

41.



Select the **Extrude** tool.

42.

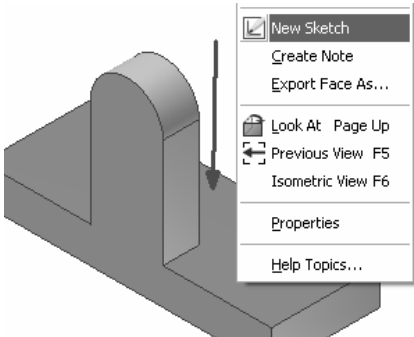


Extrude the geometry into the block a depth of **0.5** units.

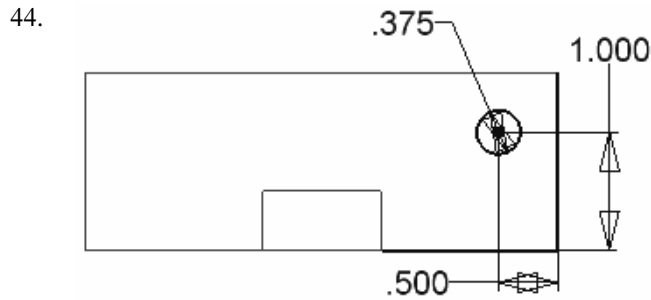
If the sketch is not selected, press the select **Profile** button, then select the sketch.

Use the **Direction** buttons to determine the direction of the extrusion.

43.



Select the top face, right-click, and select **New Sketch**.



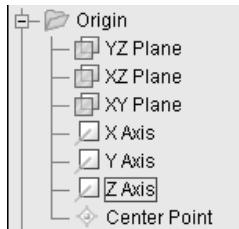
Draw a circle on the right side and dimension as shown.

**NOTE:** It is generally not considered a good idea to extrude a circle to create a hole. The preferred method is to use the Hole tool. The reason is that, when you create your 2D layout, you will be able to use the Hole Note tool to designate the hole rather than a basic dimension.

45. Select the **Project Geometry** tool.



46. Select the **Z-axis** to add it to the sketch.



47. Select the projected axis.

48. Change the style to **Centerline** using the Centerline toggle.

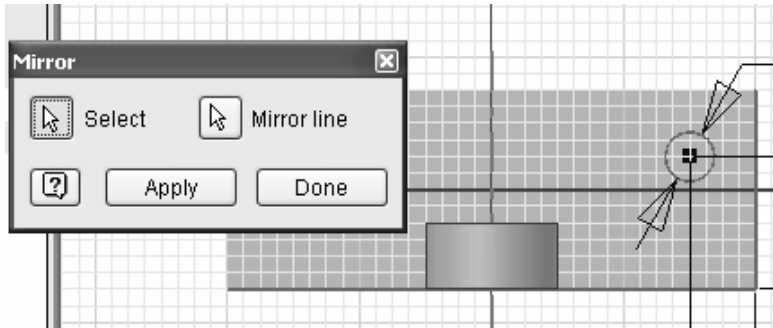


49. Select the **Mirror** tool.



Select the circle to be mirrored and the axis as the **Mirror line**.

50. Select **Apply** and **Done**.



**NOTE:** One advantage to mirroring the hole is that you only need to change the dimensions on one hole and both holes will automatically update.

51. Right-click and select **Finish Sketch**.

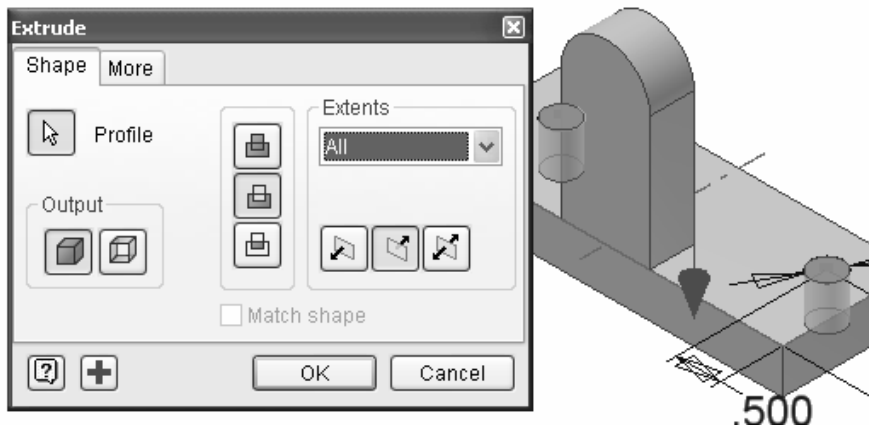


The view will automatically switch to an isometric view.

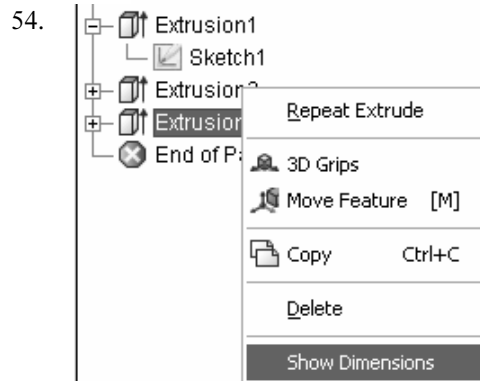
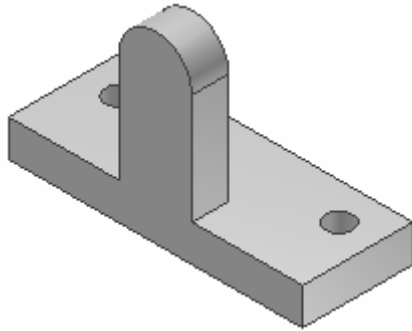
52. Select the **Extrude** tool.



- 53. Select both holes.  
Set to **Cut**.  
Set Extents to **All**.  
Extrude the holes as a cut through all.



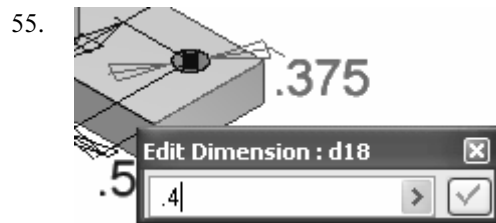
Our model so far.



Select the **hole extrusion** in the Browser.

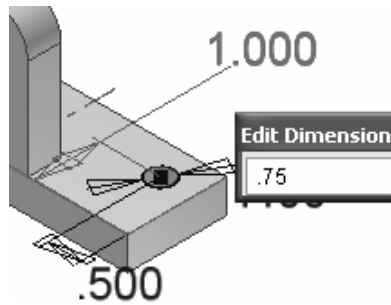
The holes will highlight in the graphics window when selected.

Right-click and select **Show Dimensions**.



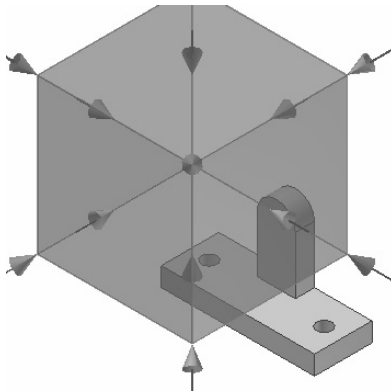
Change the **hole diameter** to **0.400**.

56. Change the **1.00 dimension** to **0.75**.



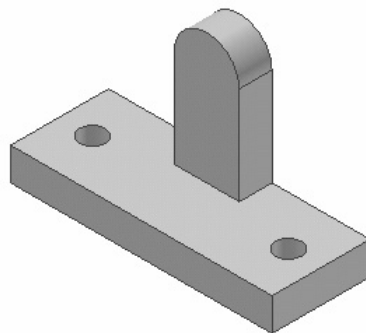
57.  Press **Update**.

58. Use **Common View** to change the view orientation of the part.



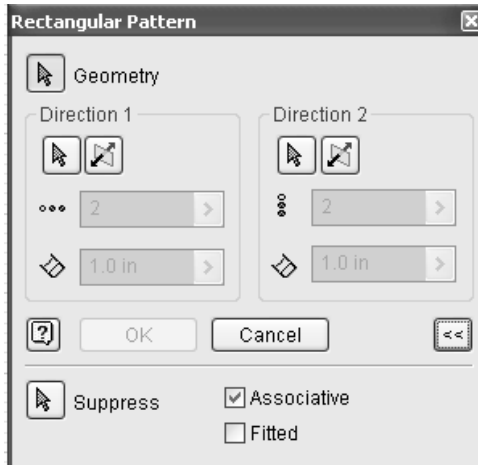
Select the **3D Orbit** tool then press the **SPACE** bar to switch to Common View mode.

Select the green arrows to change the display.



Note that both holes shift and change size. This is because the mirror definition causes them to be linked.

59. Save as **ex3-1.ipt**.  
Close the file.




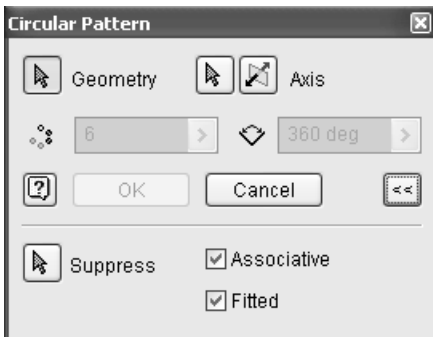
### Rectangular Pattern

This **Rectangular Pattern** tool only works in Sketch mode.

Select the geometry to be patterned.  
Select the edge to be used as the axis for the pattern.

To suppress an instance, pick the **Supress** button and then select the instance to suppress. A dashed line designates suppressed instances.

 **TIP:** When you edit pattern dimensions, you can use parametric equations to drive the position of your sketch patterns.

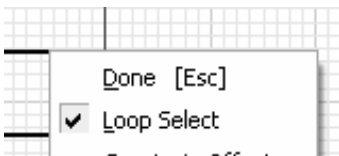


### Circular Pattern

Select the geometry to pattern.

For the axis, select an arc or circle.

To suppress an instance, select the **Supress** button and then the sketch geometry to suppress. A suppressed instance is designated by a dashed line.



This rectangular pattern tool only works in Sketch mode. To suppress an instance, pick the **Supress** button and then select the instance to suppress. Suppressed instances are designated by a dashed line.




### Offset

The **Offset** tool prompts the user to select the object to offset and the user then uses the mouse to drag and drop the offset copy to the approximate location. To constrain the offset object, the user can add dimensions using the **Dimension** tool.

A right mouse click brings up a submenu where the user can determine the constraints used for the offset or change views to facilitate editing.

More than one object can be selected at a time for **Offset**. The selected objects will highlight in green. When we have completed our selections, right-click the mouse and select **Continue** in the submenu. Then drag the offset to the approximate location desired.

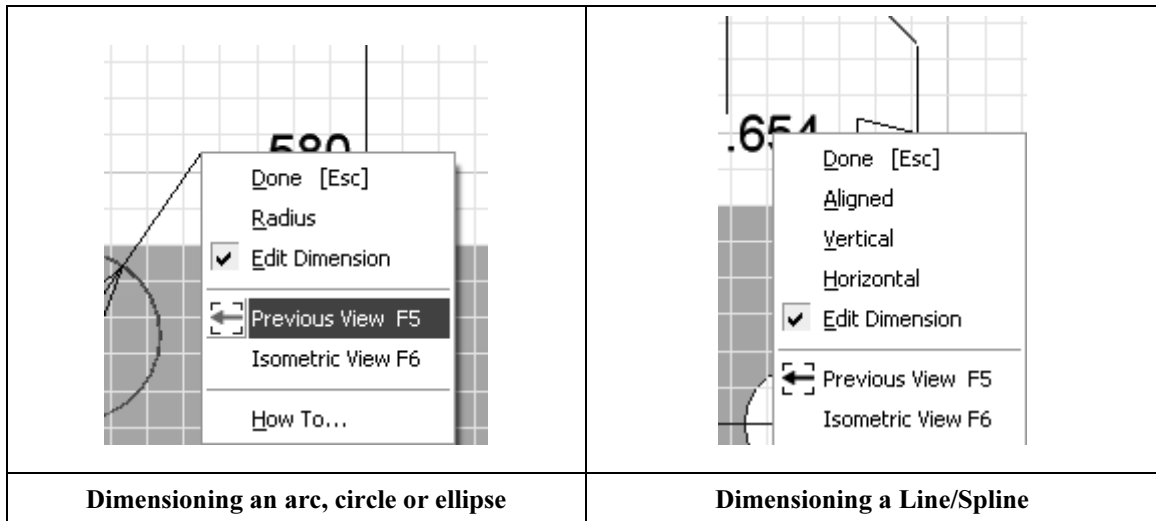
**The default setting automatically selects loops (curves joined at the endpoints) and constrains the offset curve to be equidistant from the original curve. To offset one or more individual curves or omit the Equal constraint, right-click and clear the checkmarks on Loop Select and Constrain Offset in the submenu.**

 General Dimension

**General Dimension**


The first icon, which resembles a paintbrush roller, is used for General Dimensioning. Inventor automatically knows whether the object being dimensioned is a line or an arc.

If an arc is being dimensioned, you can right-click the mouse and bring up a submenu. This submenu allows you to switch from Radius mode to Diameter mode simply by selecting that option.



When dimensioning a line, right-clicking the mouse will bring up a submenu with the options for **Aligned**, **Vertical** or **Horizontal** linear dimensions.

Simply selecting a dimension and then editing the value in the dialog box that appears will modify any dimension.

  Edit Dimension

**TIP:** When **Edit Dimension** is enabled, as displayed in the dimension shortcuts, this means that the Edit Dimension dialog will automatically display whenever you place a dimension. You can toggle this user option off or on from the shortcut menu.



**Auto Dimension**

**Auto Dimension** tells the user how many dimensions are required to fully define a sketch and applies constraints as needed.



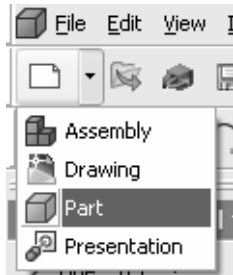
### Exercise 3-2: Auto Dimension

File: New (Standard using Inches)  
Estimated Time: 30 minutes

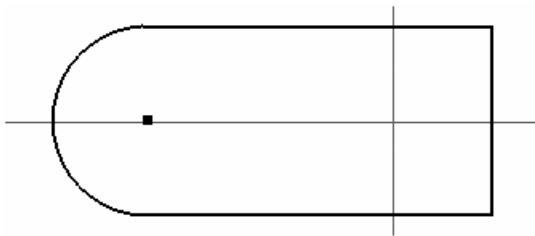
This exercise reinforces the following skills:

- ◆ Sketch
- ◆ Auto Dimension
- ◆ Sketch Constraints

1. Start a new file using Standard units.



2. Draw the sketch shown.

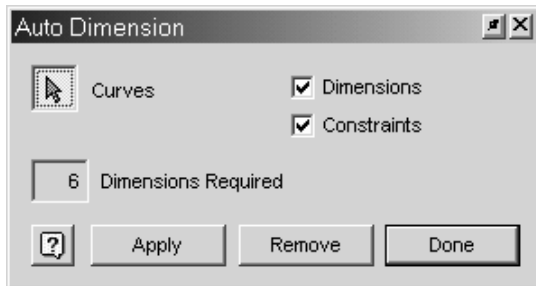


Do not add any dimensions or constraints.

3. Select **Auto Dimension**.

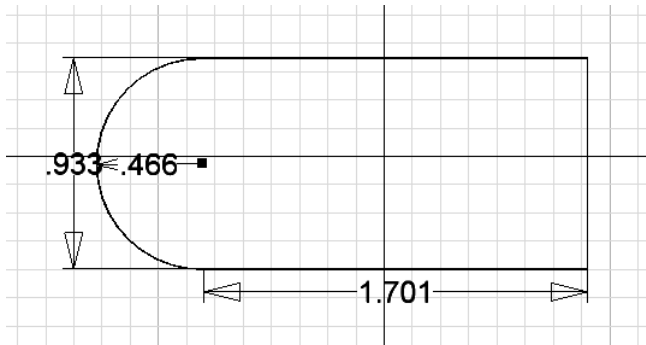


4. A dialog appears indicating how many dimensions are required to fully constrain the sketch.



Press the **Apply** button.

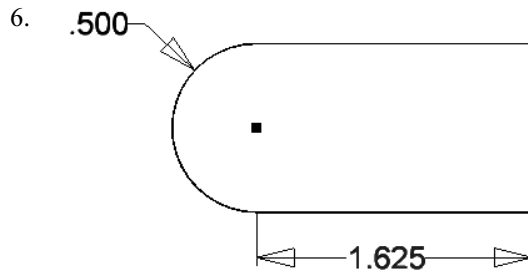
5. The dimensions appear as shown.



***NOTE:** Your dimensions will probably be different depending on how you drew your sketch.*

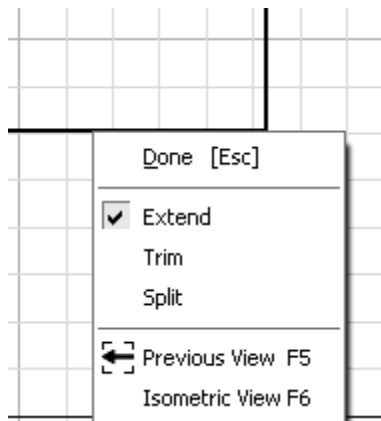
Press **Done**.

You can now select dimensions and edit them as needed.



Modify the dimensions as shown.  
You will have to delete two of the redundant dimensions in order to make the changes.  
Right-click and select **Finish Sketch**.

7. Save the file as *ex3-2.ipt*.  
Close the file.



### Extend

The **Extend** tool works differently than in AutoCAD. The user is prompted for the object to extend. The object then highlights in red and the user moves the mouse to indicate how far to extend the object. Inventor previews the object as modified and the user left-clicks the mouse to accept the modification.

Right-clicking the mouse while in 'Extend' mode will bring up a submenu giving the option to switch to 'Trim' mode or change views.



### Trim

The **Trim** tool prompts the user to select the object to trim and automatically uses any intersecting edges as the cutting tool. Inventor previews the modification in red for the user and the user accepts by left-clicking the mouse. A right mouse click brings up the same submenu as the Extend right mouse click, only with the check mark appearing next to the Trim option. Thus, the user can easily switch from 'Trim' mode to 'Extend' mode.



### Split

The **Split** tool works similarly to the Break @ function in AutoCAD. Select the tool, select the line, and select a break point. The line will then be broken into two separate segments.



**TIP:** Press and hold SHIFT to temporarily enable Trim when in Extend mode, or to enable Extend when in Trim mode.




**TIP:** You can double left-click on a sketch name to activate Edit Sketch mode.

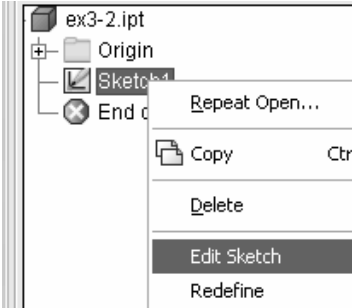
### Exercise 3-3: Move and Copy


File: Ex3-2.ipt  
Estimated Time: 15 minutes

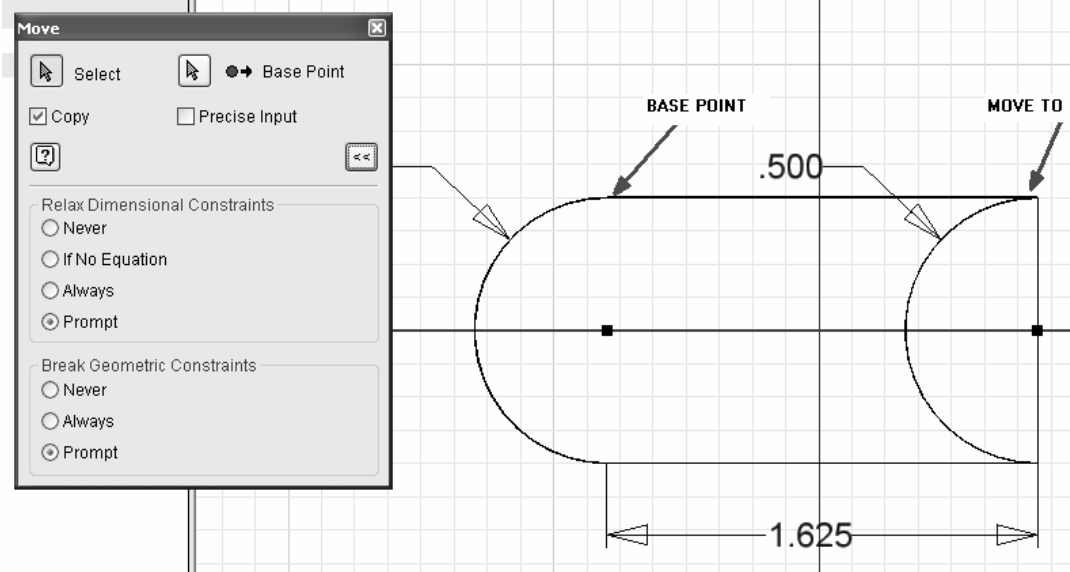
This exercise reinforces the following skills:

- ◆ Using the Move Sketch tool

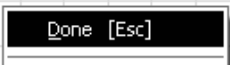
1.  Open *ex3-2.ipt*.

2.  Highlight the **Sketch** in the Browser. Right-click and select **Edit Sketch**.

3.  Select the **Move** tool from the 2D Sketch toolbar.

4. 

Press the **Select** button and select the arc.  
Enable the **Copy** button.  
The arc will highlight to indicate it has been selected.  
Press the **Base Point** button and select the top arc endpoint.  
Select the top endpoint of the rectangle as the destination point.

5.  Right-click and select **Done**.  
The arc is now copied to the new position.

6.  Exit Sketch mode.


7. Save the file as *ex3-3.ipt*.  
Close the file.

**Exercise 3-4:  
Copy**


File: copy.ipt  
Estimated Time: 5 minutes

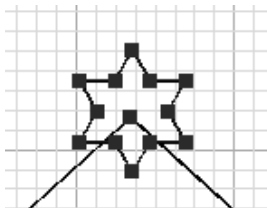
This exercise reinforces the following skills:

- ◆ Using the Copy Sketch tool

1.  Open *copy.ipt*.  
This file must be downloaded from the publisher’s website at [www.schroff.com/resources](http://www.schroff.com/resources).

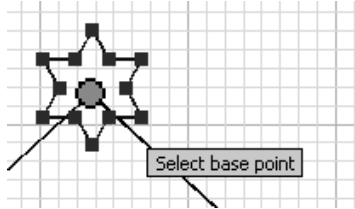
2.  Highlight **Sketch** in the Browser.  
Right-click and select **Edit Sketch**.

3.  Select the **Copy** tool.

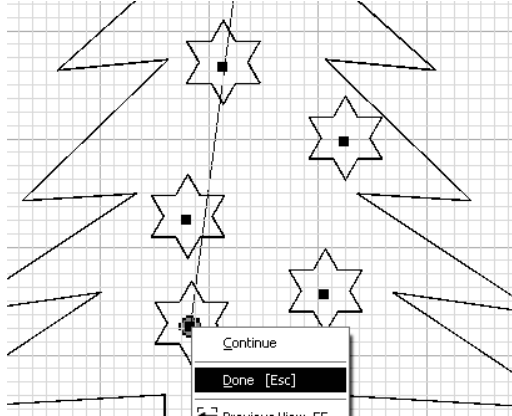
4.  Window around the star to select it.

5.   Base Point Select the **Base Point** button.

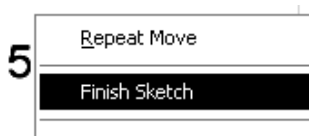
6. Select the tip of the tree as the base point.



7. Place several stars on the tree.  
Right-click and select **Done**.




8. Exit Sketch mode.



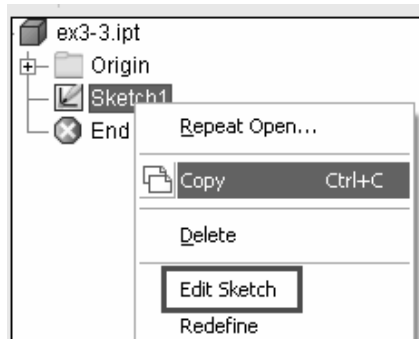
9. Close the file without saving.


### Exercise 3-5: Scale

File: Ex3-3.ipt  
Estimated Time: 10 minutes

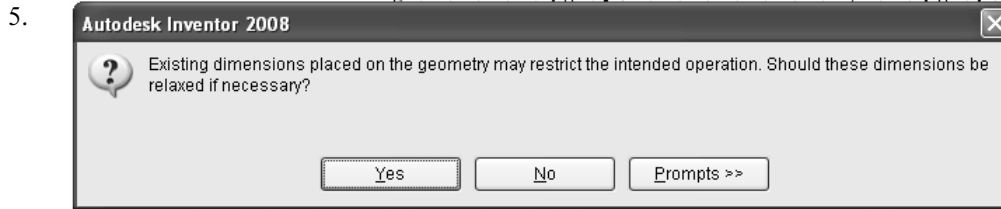
1.  Open *ex3-3.ipt*.

2. Highlight the **Sketch** in the Browser.  
Right-click and select **Edit Sketch**.

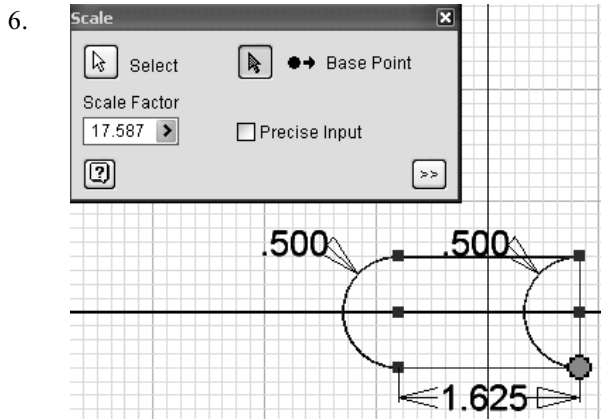


3.  Select the **Scale** tool.

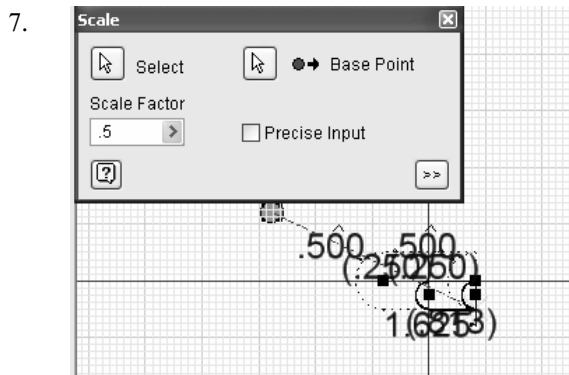
4. Press the **Select** button and window around the sketch to select it.



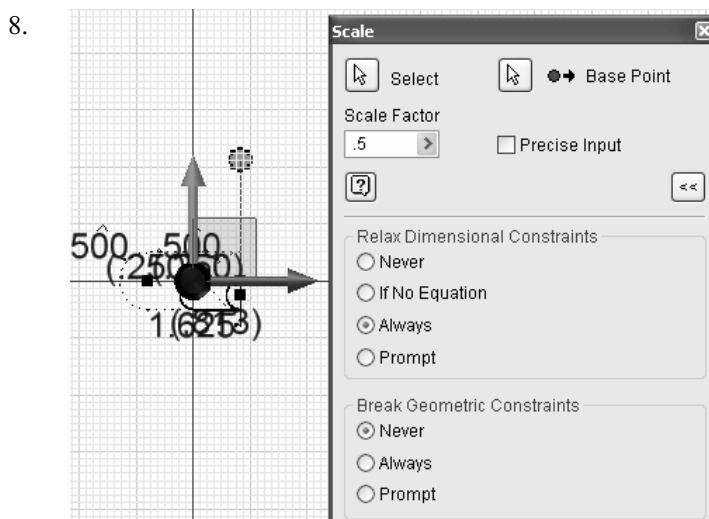
Press **Yes**.



Select the **Base Point** button and select the lower right corner of the sketch as the base point.

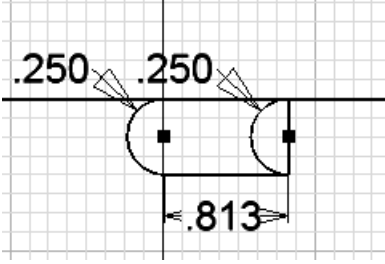


Type **0.5** in the Scale Factor box.



Press the >> **More** button and enable **Always** under Relax Dimensional Constraints and **Never** under Break Geometric Constraints.

9. Press **ENTER**.

10.  The sketch is scaled.


11. Close the file without saving.



**TIP:** The Scale command is particularly useful when you copy geometry from AutoCAD drawings.

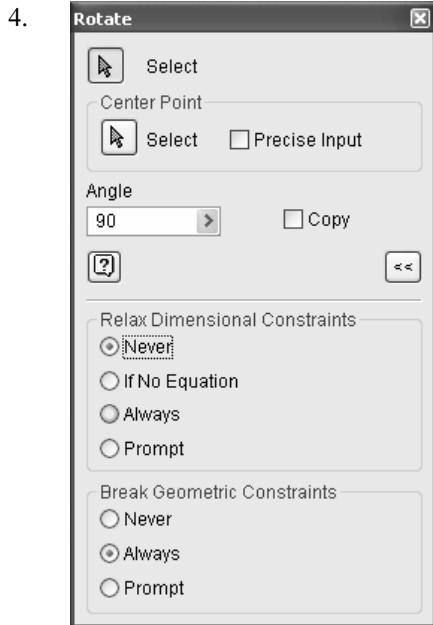
### Exercise 3-6: Rotate

File: Ex3-3.ipt  
Estimated Time: 5 minutes

1.  Open *ex3-3.ipt*.

2.  Highlight the **Sketch** in the Browser.  
Right-click and select **Edit Sketch**.

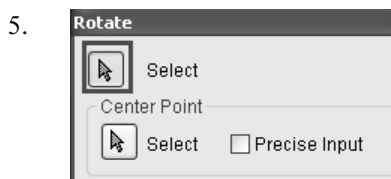
3.  Select the **Rotate** tool.



Press the **>> More** button.

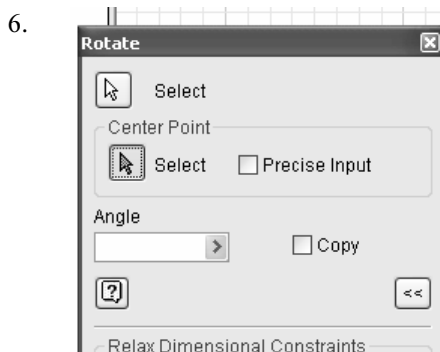
Enable **Never** under Relax Dimensional Constraints and **Always** under Break Geometric Constraints.

This means the size of the objects will be maintained, but the orientation may change.

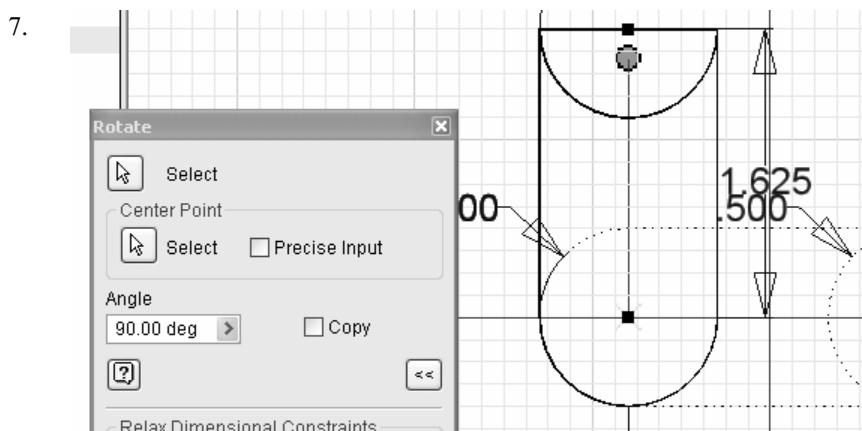


Press the **Select** button and window around the entire sketch.

If you projected the center point into the sketch, you need to make sure you do not select this center point or you will get an error message.



Press the **Center Point** button and select the center point of the left arc.



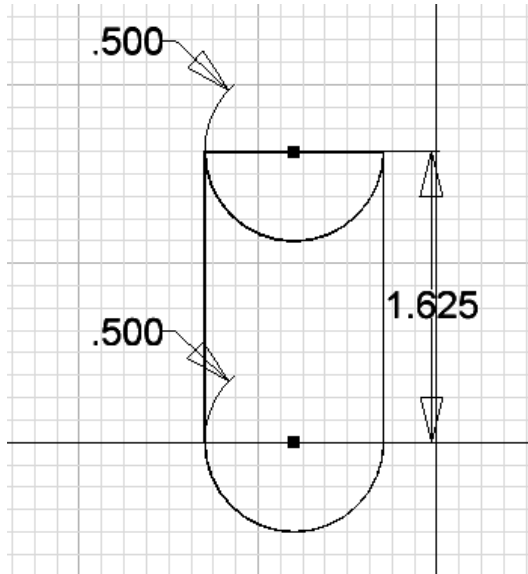
Change the **Angle** value to **90**.

You can do this by moving your cursor until you see the 90 value in the dialog.

Left-click to end.



8.

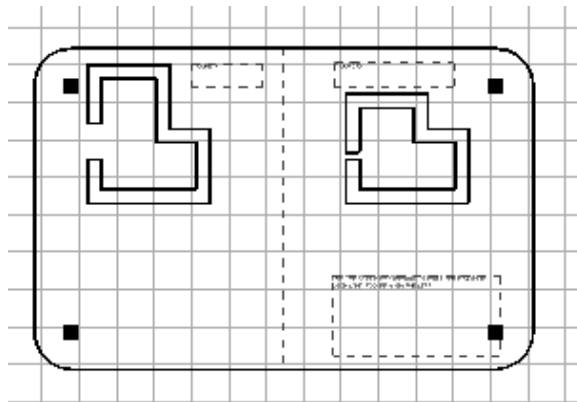


Close the dialog.


Right-click and select **Finish Sketch**.

Close the file without saving.

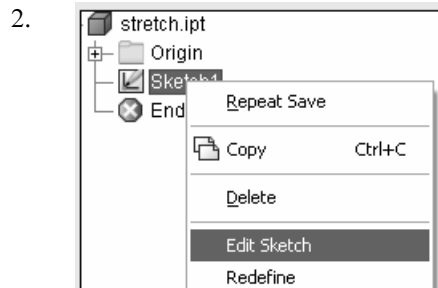
### Exercise 3-7: Stretch



File: stretch.ipt  
Estimated Time: 5 minutes

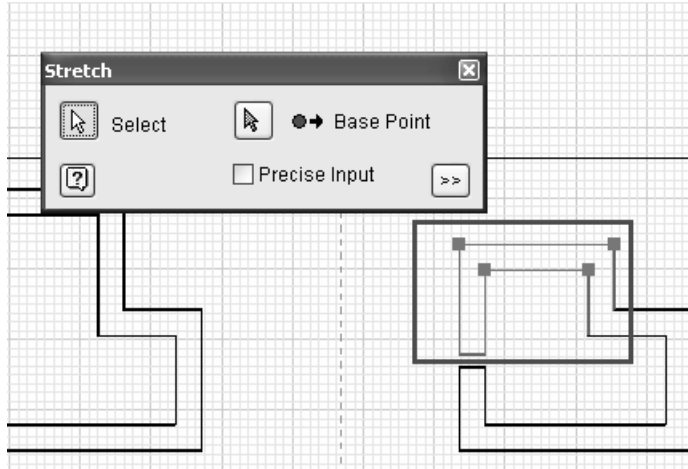
1.  Open *stretch.ipt*.  
*This file can be downloaded from the publisher's website at [www.schroff.com/resources](http://www.schroff.com/resources).*

*Note: The **Snap to Grid** option is enabled to make this exercise easier to perform.*



Highlight the **Sketch** in the Browser.  
Right-click and select **Edit Sketch**.

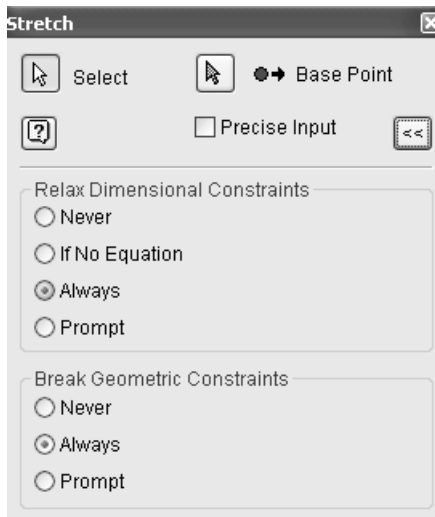
3.



Press the **Select** button and window around a portion of the sketch to select it.

*If you select the entire sketch, the command does not work.*

4.

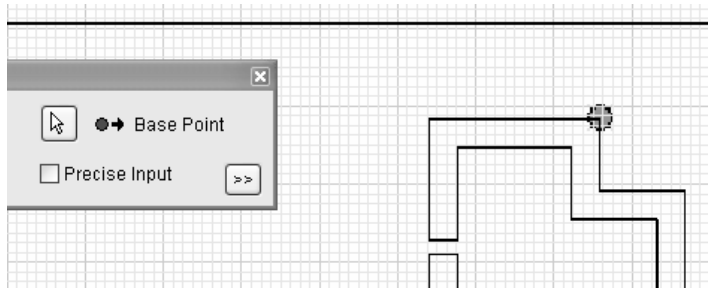


Press the >> **More** button.

Enable **Always** under **Relax Dimensional Constraints** and **Always** under **Break Geometric Constraints**.

This means both the size of the objects and the orientation can be changed.

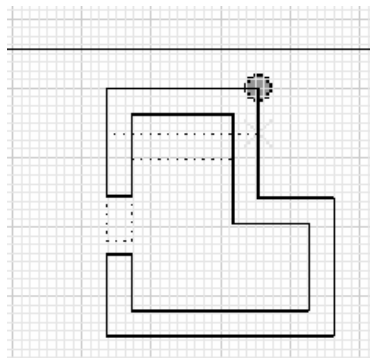
5.



Select the **Base Point** button.

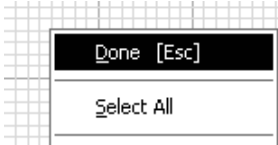
Pick the middle upper corner of the sketch as the base point.

6.

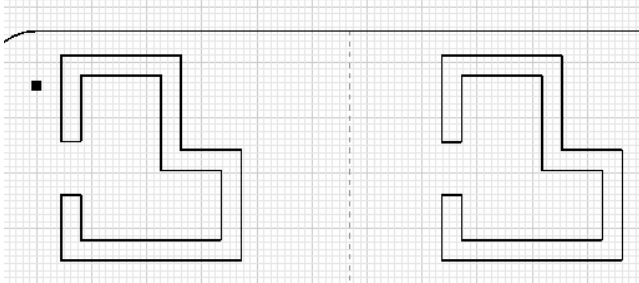


Drag the mouse up two grid squares so it lines up with Figure A and pick to place.

7. Right-click and select **Done**.



8. Your figures should match.



9. Exit sketch mode.

10. Close without saving.

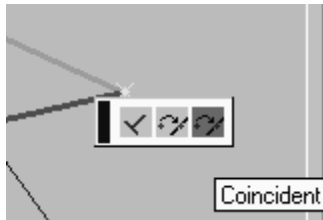


### Constraints

The next tool is used for adding geometric constraints. Pressing on the arrow reveals a fly-out toolbar with all the available constraints. The top row of constraints from left to right are: Perpendicular, Parallel, Tangent, Smooth, Coincident, and Concentric. The bottom row of constraints from left to right are: Collinear, Equal, Horizontal, Vertical, Fixed and Symmetric.

The **Coincident** constraint may be used to ensure that two lines form a closed angle with no overlap. The **Fixed** constraint fixes an object to a location relative to the sketch coordinate system. The other constraints are used in a similar manner to other parametric modeling software.

**TIP:** Press and hold CONTROL to prevent constraints from being added while sketching geometry.



### Show/Delete Constraints

To show constraints, press the **Show/Delete Constraints** tool button. Next, select the object. A small Constraint Bar will appear displaying the constraints for that object. Moving the mouse along the constraint bar will highlight each constraint.

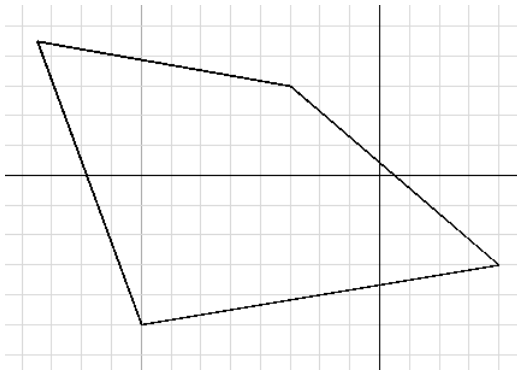
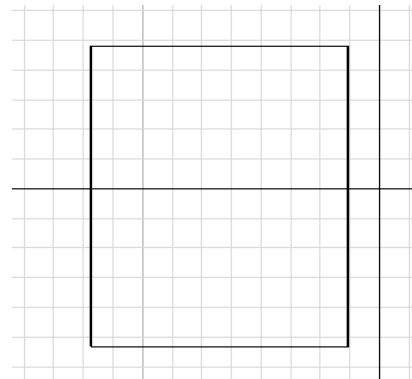


To delete a constraint, enable the Constraint Bar. Move the mouse to the constraint to delete on the Constraint Bar. Note the highlighted objects to ensure that the correct constraint will be deleted. Right-click the mouse and the 'Delete' key will appear. Left-click the mouse to accept. If we don't wish to delete, just move the mouse off of the Constraint Bar and left-click anywhere in the window.

**TIPS:**

- ◆ Use the Zoom Window button on the Standard toolbar to zoom in on the area where you are working.
- ◆ Set the grid to the spacing needed to quickly line up the sketch elements.
- ◆ Check the Snap to Grid setting to more easily place sketch elements.
- ◆ To select a group of sketch elements, activate the Select tool, then click in the graphics window and drag a box around the elements.
- ◆ Use the dimension tools to set the size of sketched geometry or to add dimensions between the geometry in a sketch and elements in the underlying drawing view.
- ◆ When you use dimensions to set the size of elements in a title block or border, the dimensions are hidden when you finish editing.


### Exercise 3-8: Adding Horizontal and Vertical Constraints

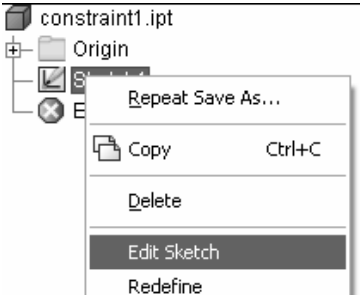
**BEFORE****AFTER**

File: constraint1.ipt  
Estimated Time: 5 minutes

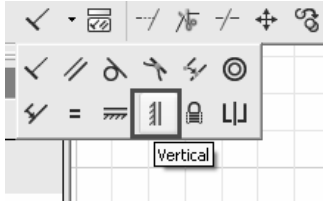
This exercise reinforces the following skills:

- ◆ Add Constraints

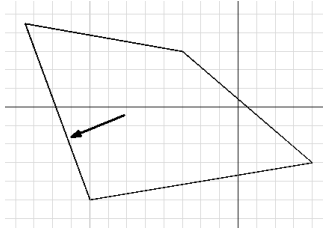
1.  Open **constraint1.ipt**.  
*This file can be downloaded from the publisher's website at [www.schroff.com/resources](http://www.schroff.com/resources).*

2.  Highlight the **Sketch** in the Browser.  
Right-click and select **Edit Sketch**.

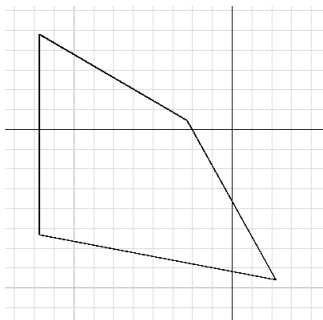
3. Select the **Vertical** constraint tool.



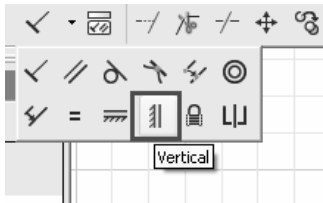
4. Select the line located on the left to apply the constraint.



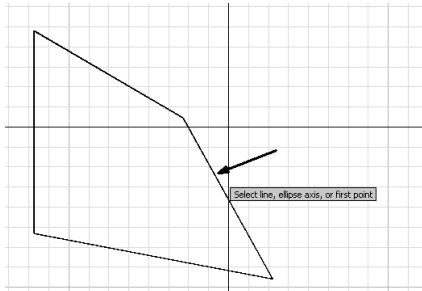
5. The object will shift as the line is constrained.



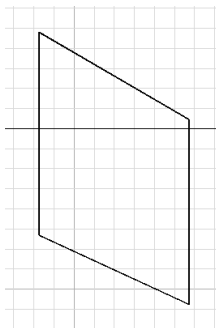
6. Select the **Vertical** constraint tool.



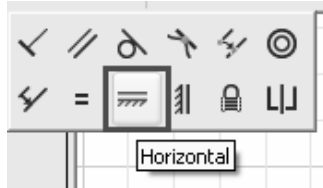
7. Select the line located on the right to apply the constraint.



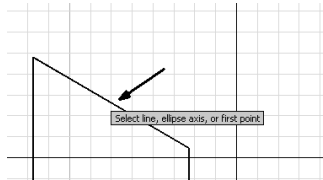
8. The object will shift as the line is constrained.



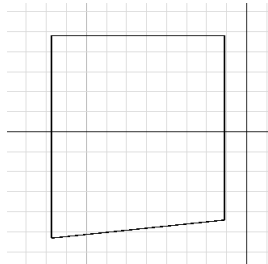
9. Select the **Horizontal** constraint tool.



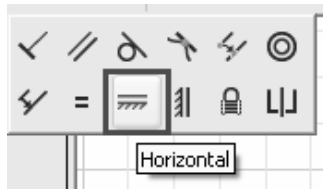
10. Select the angled line.



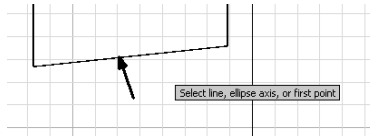
11. The object will shift as the line is constrained.



12. Select the **Horizontal** constraint tool.



13. Select the line located on the bottom to apply the constraint.



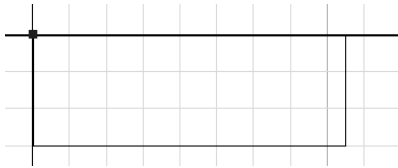
14. Close without saving.

### Exercise 3-9: Deleting Constraints

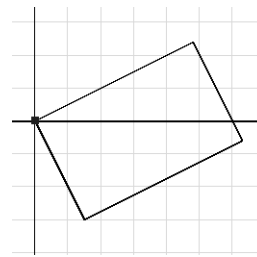
File: constraint2.ipt  
Estimated Time: 5 minutes

This exercise reinforces the following skills:


- ◆ Show Constraints
- ◆ Hide Constraints
- ◆ Delete Constraints
- ◆ Modify Sketch

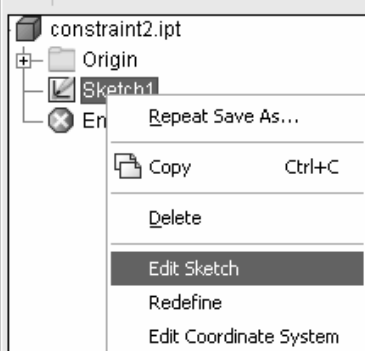



**BEFORE**

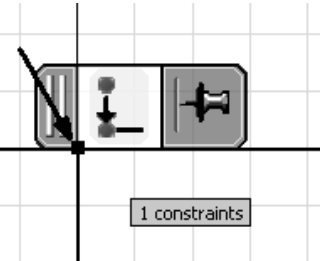


**AFTER**

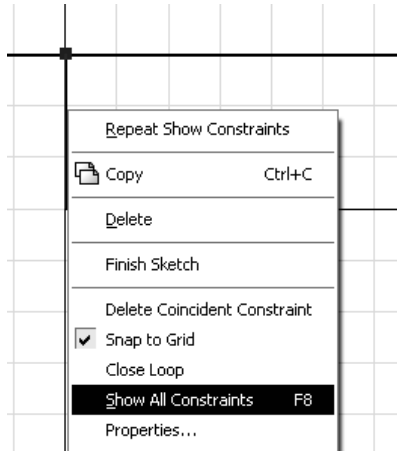
1.  Open **constraint2.ipt**.  
*This file can be downloaded from the publisher's website at [www.schroff.com/resources](http://www.schroff.com/resources).*

2.  Highlight the **Sketch** in the Browser.  
Right-click and select **Edit Sketch**.

3.  Select the **Show Constraints** tool.

4.  Select the upper left corner of the rectangle.  
You will see a Coincident constraint indicated.

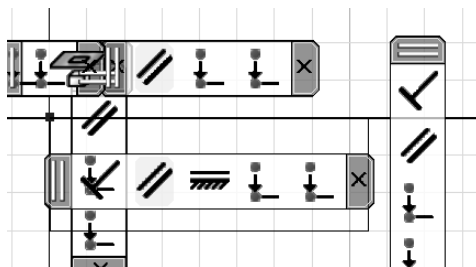
5.



Select the left vertical line.

Right-click and select **Show All Constraints**.

6.



Mouse over each constraint.

Note how the entities highlight to indicate where the constraints are applied.

7.



Locate the horizontal constraint.

Right-click and select **Delete**.

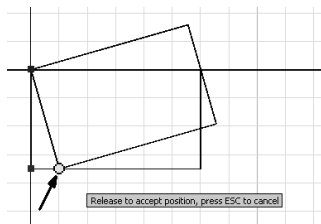
8.



Right-click in the display window.

Select **Hide All Constraints**.

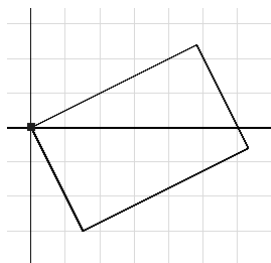
9.



Select the lower left corner of the rectangle.

Use your mouse to rotate the rectangle. Notice that it remains pinned at the coincident constraint you located earlier.

10.










Rotate the rectangle and release the mouse to place.

11. Close without saving.





Inventor features three projection tools: Project Geometry, Project Cut Edges, and Project Flat Pattern.

	<p><b>Project Geometry</b></p> <p>Our next tool button creates reference geometry by projecting model geometry (edges and vertices), work features, or sketch geometry from another sketch onto the active sketch plane. Reference geometry can be used to constrain other sketch geometry or used directly in a profile or path sketch.</p>
	<p><b>Project Cut Edges</b></p> <p>This tool projects edges cut by the sketch plane onto the current sketch plane.</p>
	<p><b>Project Flat Pattern</b></p> <p>This tool is grayed out unless a flat pattern exists. If a flat pattern is available, the user may select a face to project it onto a selected plane.</p>
	<p><b>Parameters</b></p> <p>The Parameters tool is used to create table-driven parts and features.</p>
	<p><b>Insert AutoCAD file</b></p> <p>Inserts an AutoCAD drawing into a sketch.</p>
	<p><b>Insert Text</b></p> <p>Adds text to a sketch. It can then be extruded using the Emboss tool.</p>
	<p><b>Insert Image</b></p> <p>Adds an image to a sketch. It can be converted into a decal.</p>

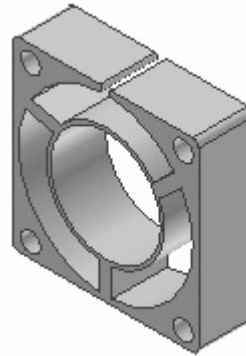
### Exercise 3-10: Inserting an AutoCAD file

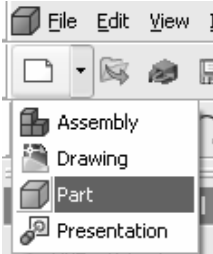
File: Ta100dcd.dwg  
 [can be downloaded from the publisher's website ([www.schroff.com/resources](http://www.schroff.com/resources)) for free]  
 Estimated Time: 30 minutes


To demonstrate how it works, we use a drawing from Nidec's fan catalog, but any AutoCAD drawing will do.

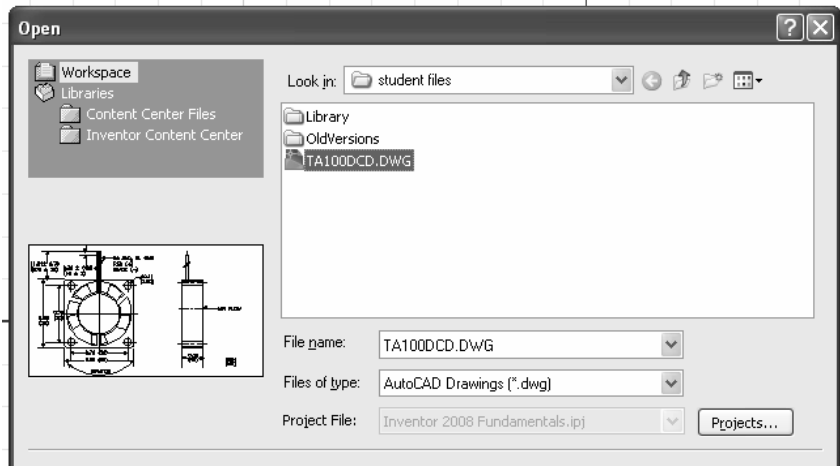
This exercise reinforces the following skills:

- ◆ Insert AutoCAD file
- ◆ Measure Distance
- ◆ Trim
- ◆ Delete
- ◆ Close Loop
- ◆ Extrude

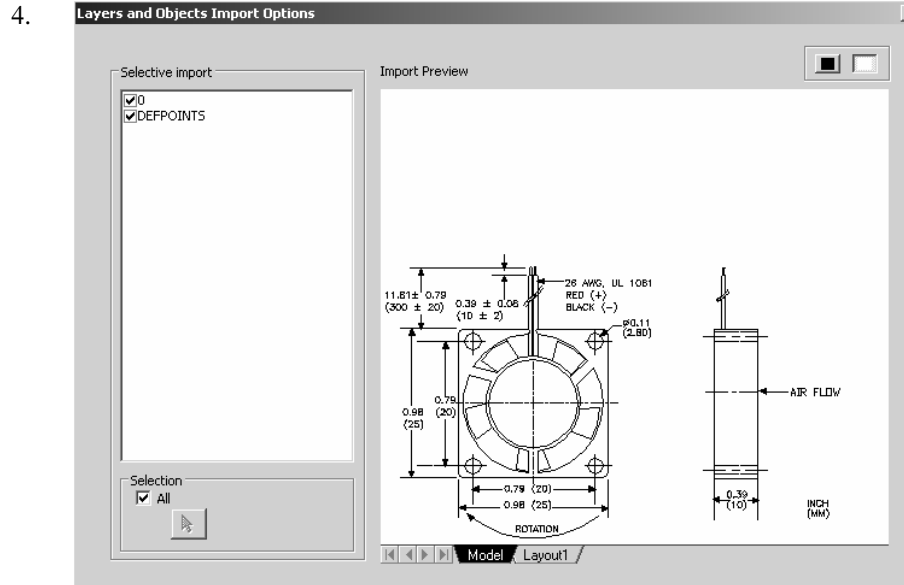


1.  Start a new file using Standard units.

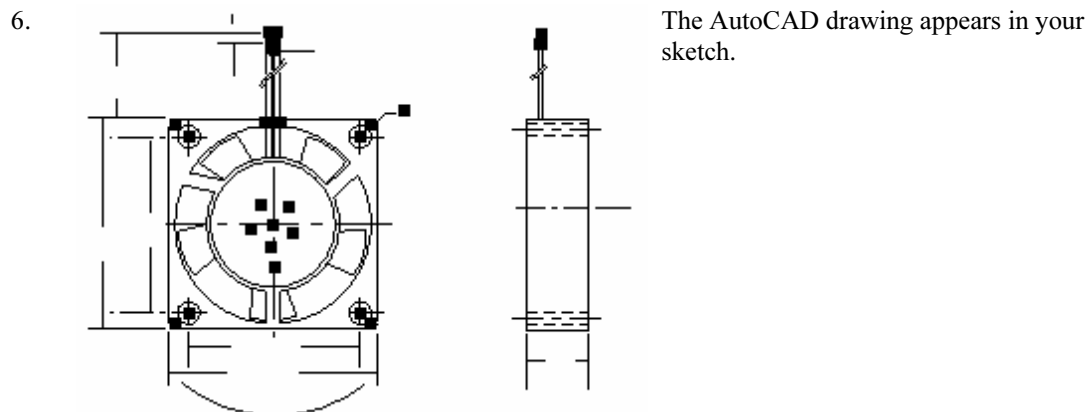
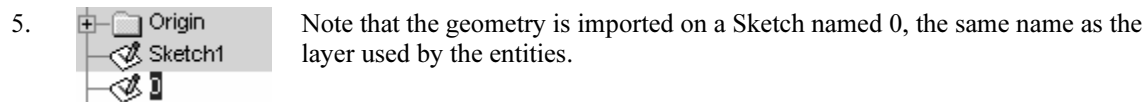
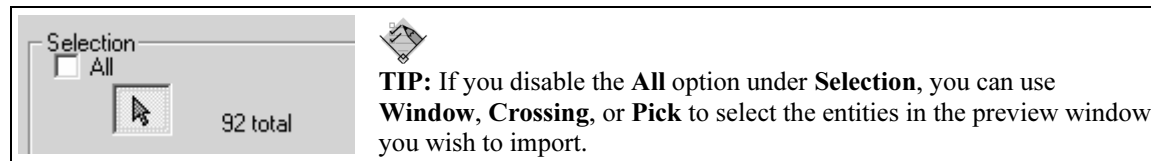
2.  Select the **Insert AutoCAD file** tool.  
The Browser dialog will come up.

3.  Locate the **TA100DCD.dwg** file you downloaded from the publisher's website.

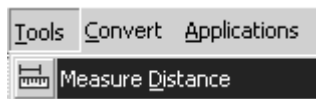
Press **Open**.



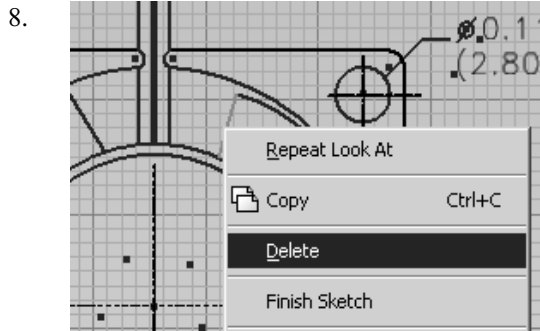
A dialog will appear to preview the drawing.  
 Note you can select which layers you want to import.  
 Press **Finish**.



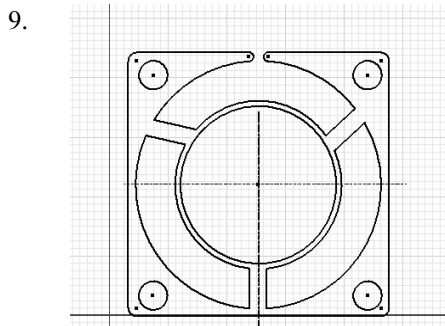
7. Measure the thickness of the fan box using the side view, so you know what dimension to apply for the extrusion.



You can measure by placing a linear dimension or using the **Measure Distance** tool under the Tools menu.

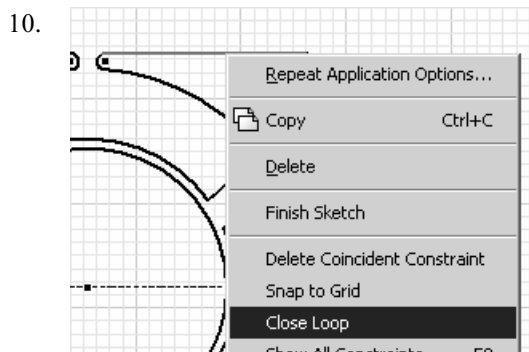


By holding down the Control key and picking with the left mouse, we can select all the dimension lines, then right-click and press **Delete**. We also need to delete the side view.



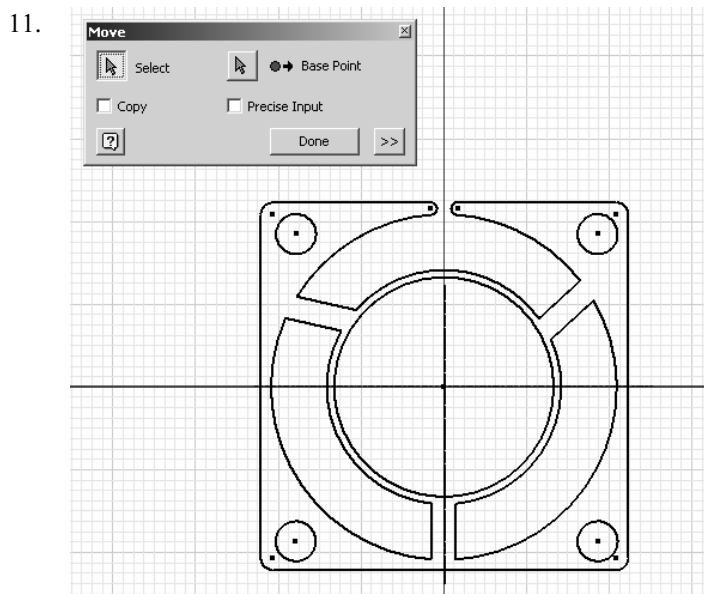
Continue cleaning up the sketch until you have a basic profile.

The sketch should look like this once it has been cleaned up.



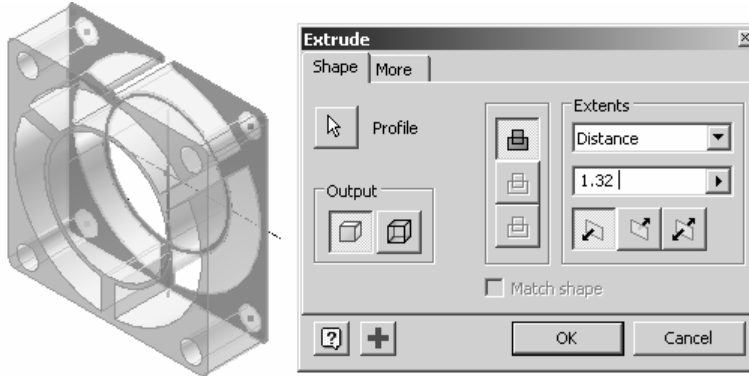
Use the **Sketch Doctor** to assist you in creating a closed loop profile.

You can also use the **Close Loop** option from the shortcut menu to create your profile.



Project the origin into the current sketch and using the center point of the fan body sketch, move the profile so it is centered at the origin point.

12.



Select **Extrude**.

Set the **Distance** to **1.32 mm**.

13. We have successfully transformed an AutoCAD 2D drawing into a 3D Parametric part in minutes.

Save the file as *ex3-10.ipt*.

### Cursor Cues

As we create our sketches, we see cursor cues telling us how Inventor is interpreting what we are drawing. By watching for the visual feedback Inventor provides, we can create sketches faster and with fewer edits required.

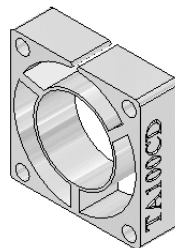
<b>Tangent</b>	<b>Parallel</b>	<b>Coincident</b>	<b>Vertical</b>	<b>Horizontal</b>	<b>Perpendicular</b>

### Exercise 3-11: Creating Etched Text

File: Ex3-10.ipt  
Estimated Time: 15 minutes

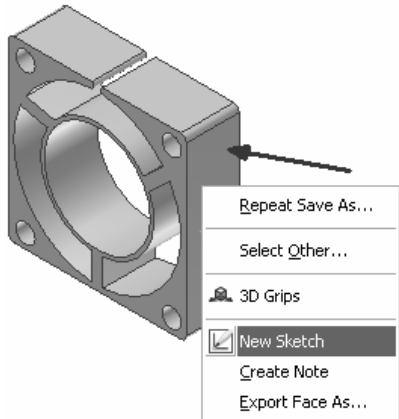
This exercise reinforces the following skills:

- ◆ Create Point
- ◆ Text
- ◆ Rotate Sketch
- ◆ Extrude



1. Open or continue working in file *ex3-10.ipt*.

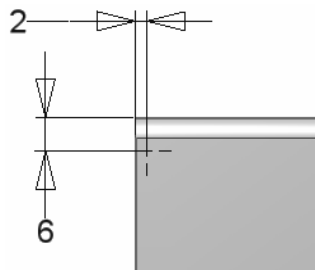
2.



Select the right vertical side of the fan box as shown.

Right-click and select **New Sketch**.

3.



In order to place the text, it is a good idea to place a point to use to align your text.



Select the **Point** tool and place a point as shown.

**Dimensions are in mm.**

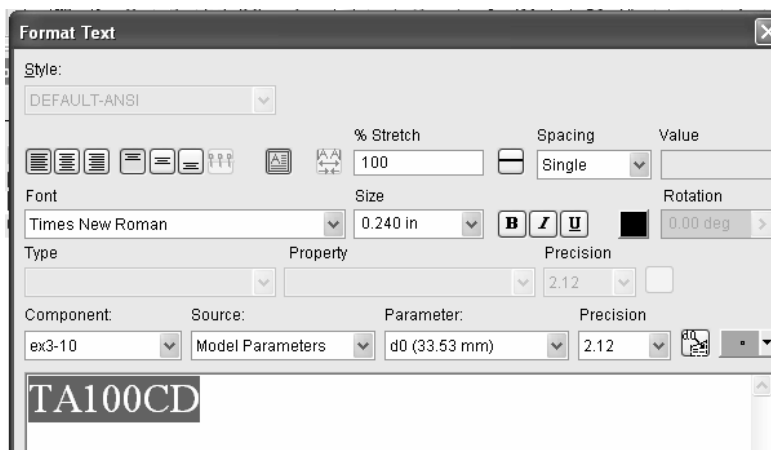
**You can go to Tools→Document Settings to change the units at any time.**

4.

**A** Select the **Text** tool.

Pick the point to bring up the **Format Text** dialog.  
This will act as the insertion point for the text.

5.



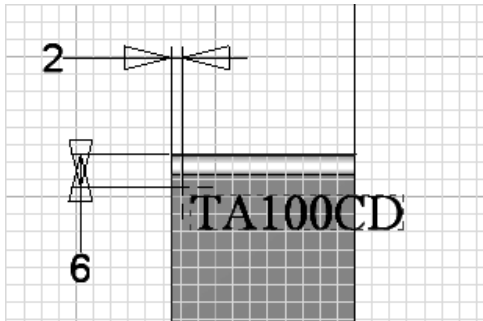
Select **Times New Roman** for the font to be used.

Set the **Font Size** to **0.240 in**.

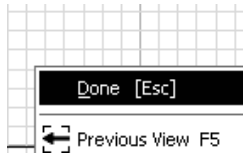
Type **TA100CD** in the text field area.

Press **OK**.

6.



Your text appears with a rectangle around it.




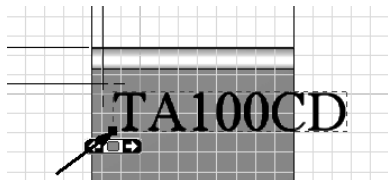
Right-click and select **Done**.

You can use the rectangle to locate your text.

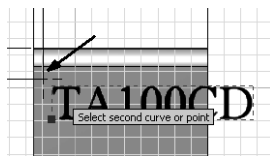
Simply pick the upper left corner of the text rectangle and drag it to the point.

You can also add a Coincident constraint between the point and the upper left corner of the rectangle to locate the text.

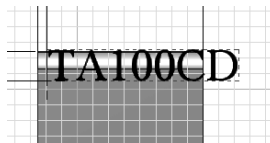
7.  Select the **Coincident** tool.




Select the lower left corner of the text box.

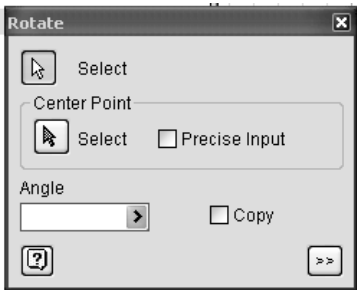
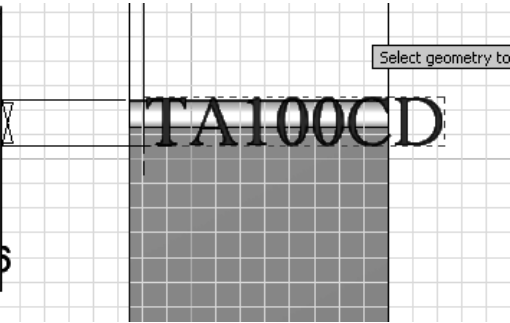


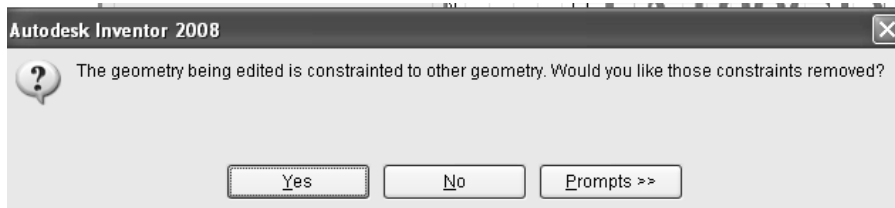
Select the point that was placed.



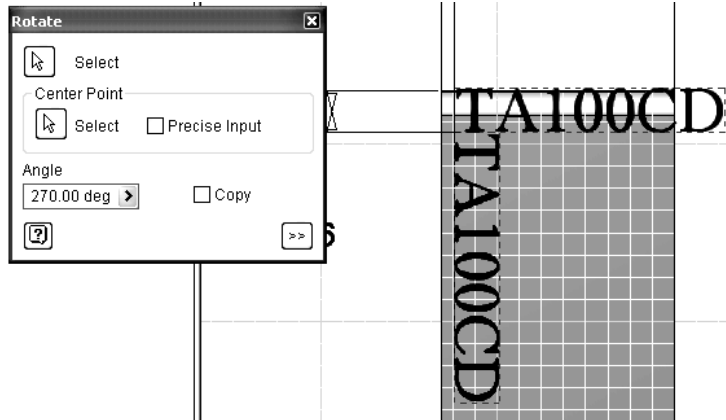
The text shifts position.

8.  Select the **Rotate** tool to rotate the sketch 90 degrees.

9.   Select the text.  
Select the bottom left corner as the center point.



Press **Yes**.

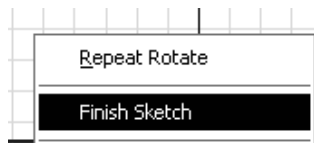


Move your cursor down until you see the angle value at 270 degrees.


Then left-click to place.



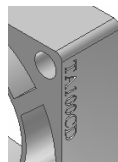
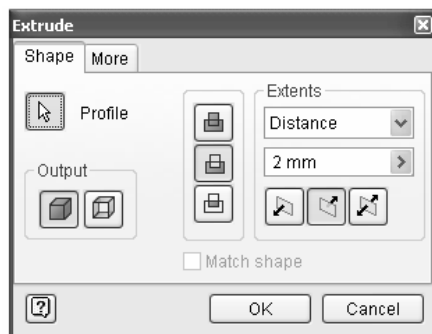
Right-click and select **Done**.



Right-click and select **Finish Sketch**.

10.  Select the **Extrude** tool. You can also use the **Emboss** tool. The Emboss tool is best for curved faces.

11.  Set the **Cut** option and set the Distance to **2 mm**.



The text is etched into the fan body.

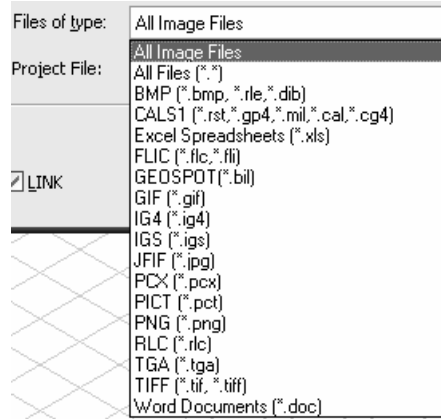
12. Save the file as *ex3-11.ipt*.





## Insert Image

Images can be inserted into a sketch and then applied as a decal to a face using the **Insert Image** tool.



You can use any image file, as well as \*.doc and \*.xls files with the **Image** tool.

### Exercise 3-12: Insert Image

File: Ex3-11.ipt, Nidec-logo.bmp (downloaded from the publisher's website.)  
Estimated Time: 15 minutes

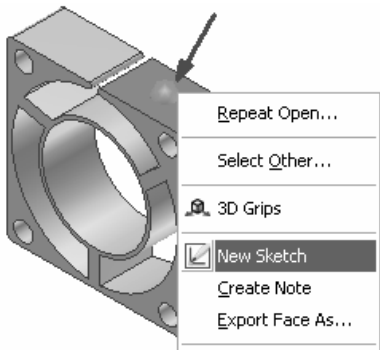
This exercise reinforces the following skills:

- ◆ Insert Image
- ◆ Decal




1. Open or continue working in file *ex3-11.ipt*.

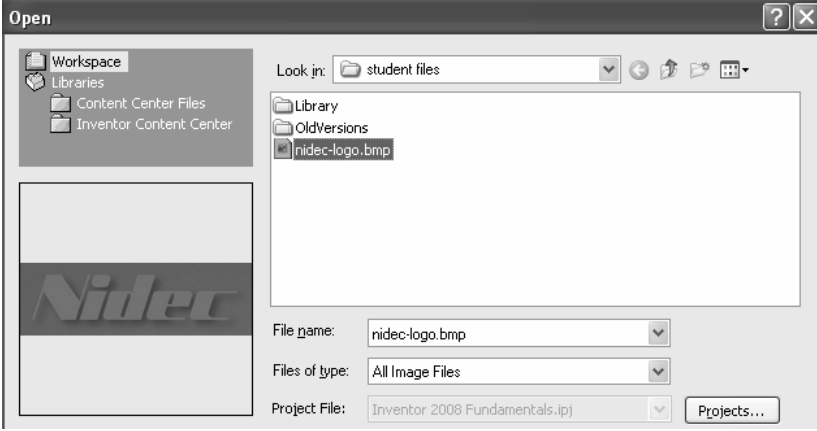
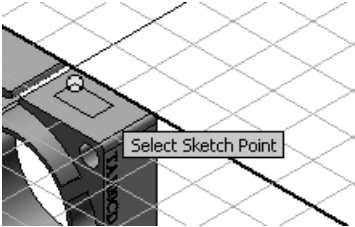
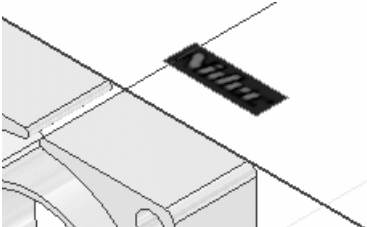
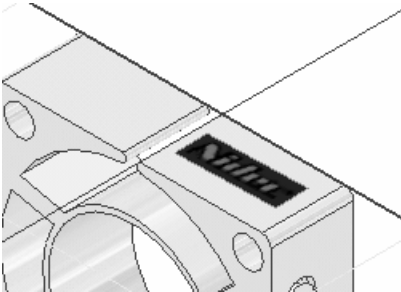


2.



Select the top face of the fan body.  
Right-click and select **New Sketch**.

3. Download the *nidec-logo.bmp* file from the publisher's website, or use any bmp file of your choice.

4.  Select the **Insert Image** tool.

5.  Locate the *nidec-logo.bmp* file you downloaded.
- Note that you can see a preview of the image in the preview window on the left.
- Press **Open**.
6.  Pick to place in your sketch.
- Right-click and select **Done**.
7.  You can scale the image by picking on the corners and holding down the left mouse button until you have the desired size.
8.  To position the image, window around the entire image to select and drag into place.
9.  Select the **Decal** tool.
10.  Select the Image. Then, select the face where you placed the Image.
- Press **OK**.
- Rotate the part around to see how the image appears from different angles.
- Remember you can only see the Decal in Shaded Display Mode.**
11. Save the file as *ex3-12.ipt*.  
Close the file.



## Edit Coordinate System

This tool allows you to redefine the Coordinate System of a sketch. The Coordinate System controls the orientation of features. Modifying the Coordinate System can affect applied constraints and reference geometry. The Edit Coordinate System tool does not work when you are in Sketch mode. This has been a bug for the past few releases. If you delete features, you may get an error message requiring that you redefine the coordinate system. Use this method to repair any existing sketches.

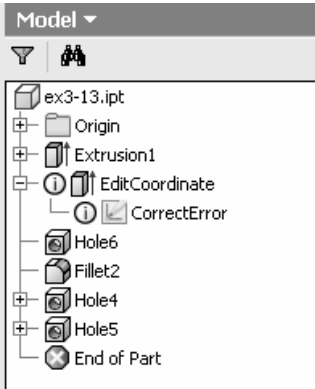
### Exercise 3-13: Edit Coordinate System


File: Ex3-8.ipt  
Estimated Time: 10 minutes

This exercise reinforces the following skills:

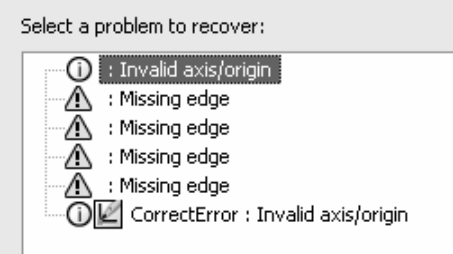
- ◆ Edit Coordinate System
- ◆ Use of the Sketch Doctor

1. Open *ex3-8.ipt*.

2.  If you look in the Browser, you will see an extrusion that has an **i** symbol next to it. The symbol indicates that there is an error that needs to be corrected.

3.  If you look in the Standard toolbar, you will see a Red Cross **+** symbol. This is the symbol for the **Sketch Doctor**, indicating a sketch needs to be repaired.

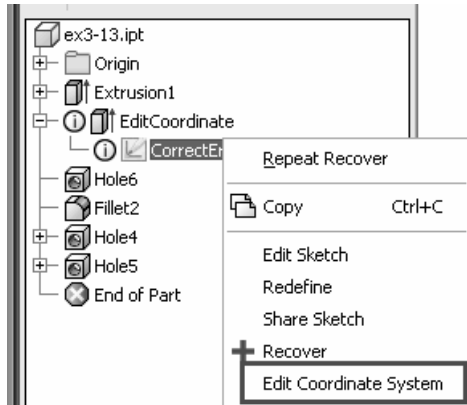
Select the **Red Cross** symbol.

4.  A list of errors is displayed. Note that the primary error is an **Invalid axis or origin**.

This error usually occurs when you delete an existing feature and a dependent sketch is affected. The error can be corrected by editing the coordinate system for that sketch.

Close the Sketch Doctor.

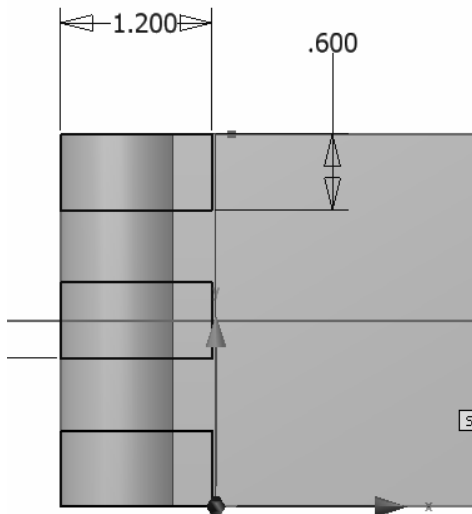
5.



Highlight the sketch called **CorrectError**.

Right-click and select **Edit Coordinate System**.

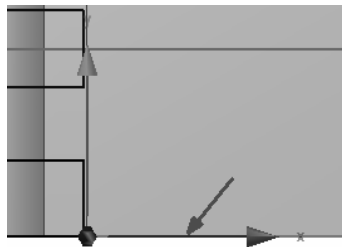
6.



Select the **Red Origin Point**.

Select the lower corner to relocate the coordinate system.

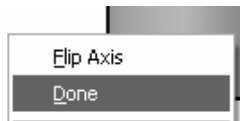
7.



Select the **X-axis**.

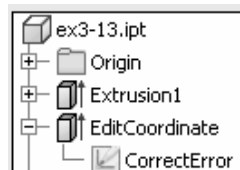
Select the edge of the part as the new X-axis.

8.



Right-click and select **Done**.

9.



Note that the error symbols are now gone.

10.



Note that the Red Cross symbol is now gray, indicating no sketch errors.

11. Close without saving.



### Import Points

This tool allows you to add points into a sketch using an Excel spreadsheet. You may import either 2D or 3D point sets (X,Y) or (X,Y,Z). In order to import 3D point sets, you must be in 3D Sketch mode.

The screenshot shows a Microsoft Excel spreadsheet titled "Microsoft Excel - ex6-9.xls". The spreadsheet has a menu bar with "File", "Edit", "View", "Insert", "Format", "Tools", "Data", and "Win". Below the menu bar is a toolbar with various icons. The spreadsheet grid shows columns A, B, C, and D, and rows 1 through 11. The data is as follows:

	A	B	C	D
1	in			
2	X	Y	Z	
3	0.00	0.00	0.00	
4	4.00	5.00	0.00	
5	4.00	5.00	5.00	
6	0.00	5.00	5.00	
7	4.00	0.00	5.00	
8	0.00	5.00	0.00	
9	4.00	0.00	0.00	
10				
11				

In order to import the points, the Excel spreadsheet must follow a specific format. If your spreadsheet does not meet the format, you will get an error message.

The first row and cell (A1) should state the units (in or mm).

The second row should be the header row.

The remaining rows are the point values.

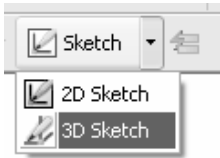
### Exercise 3-14: Import Points


File: Ex3-9.xls  
Estimated Time: 5 minutes

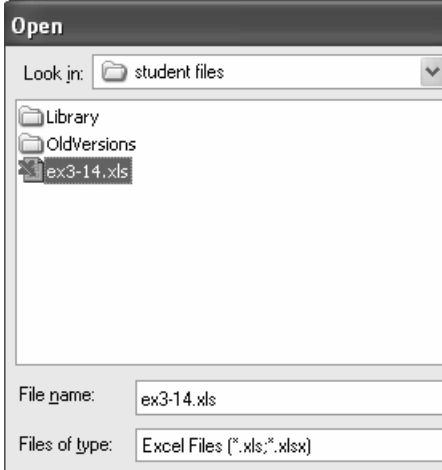
This exercise reinforces the following skills:

◆ Import Points

1. Start a new file using Standard.
2. Close the active sketch.

3.  Select **3D Sketch** from the sketch toolbar.

4.  Select the **Import Points** tool.

5.  You can use the spreadsheet available for download from the publisher's website, or use your own.

Press **Open**.

6. The points are placed.



7. Close without saving.

## Review Questions

- A.  B.  C.  D. 

Identify the geometric constraint

- Vertical
  - Fixed
  - Parallel
  - Coincident
5. The Spline tool is located under this drop-down:
- Line
  - Arc
  - Circle
  - Rectangle
6. The three types of arc options are:
- 3 Point, Tan-Tan-Tan, Start End Radius
  - 3 Point, Start Direction Radius, Start End Radius
  - 3 Point, Center Two Ends, Tangent
  - 3 Point, Center Radius, Start End Radius
7. To draw a construction line or circle:
- Use the Style drop-down and select Construction
  - Use the Construction Line/Circle tool
  - Select the Line/Circle, Right-click and enable 'Construction'
  - While drawing the line, hold down the CONTROL button.
8. To switch to arc mode while using the Line tool:
- Hold down the CONTROL key
  - Left-click and Hold down the left mouse button
  - Hold down the TAB Key
  - Right-click and select ARC from the menu.
9. To switch from TRIM mode to EXTEND mode:
- Hold down the CONTROL key
  - Press and hold SHIFT
  - Right-click and select EXTEND from the menu.
  - Hold down the TAB Key
10. To modify a dimension:
- Double click on top of the dimension
  - Select the Edit Dimension tool
  - Select the Dimension in the Browser, right-click and select Edit
  - Select Edit Text from the Modify menu.

- 
11. In order to see a Decal, your model must be in:
- A. Wireframe Display mode
  - B. Hidden Edges Display mode
  - C. Shaded Display mode
  - D. It makes no difference
12. You cannot move and copy sketch geometry in the same operation.
- A. True
  - B. False
13. When mirroring a sketch, the mirror line can be selected as part of the set to be mirrored.
- A. True
  - B. False
14. When you add text to a sketch, it can be Extruded or Embossed to create a feature.
- A. True
  - B. False
15. Inserted images cannot be scaled once they are placed in a sketch.
- A. True
  - B. False

**ANSWERS:** 1) D; 2) B; 3) A; 4) C; 5) A; 6) C; 7) A; 8) B; 9) B; 10) A, 11) C; 12) B; 13) B; 14) A; 15) B



**NOTES:**