

Lesson 6

Sketch Tools



Inventor's Sketch Tool Bar contains tools for creating the basic geometry to create features and parts.



Edit Coordinate System

We use the Edit Coordinate System to rotate the origin.

We select the Edit Coordinate System tool and then pick on the Origin. Watch the messages shown in the Message box to assist you through tasks. We then select the lower corner of the sketch to reset the origin.

The next section of the Sketch toolbar is the geometry tools, which sketch basic geometric shapes to use for creating features.

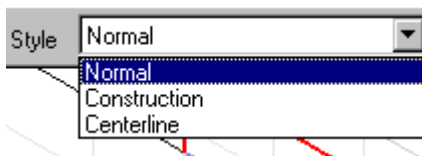


On the surface, the Geometry tools look fairly standard: line, circle, arc, rectangle, and point.

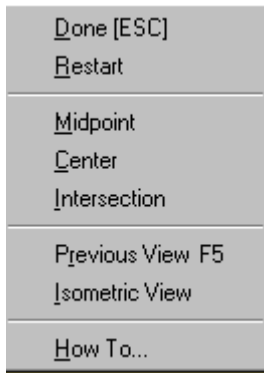


Line/Spline

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 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.

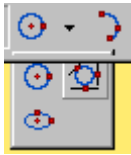


To create a Construction line instead of an object line, select the Construction option under Style.



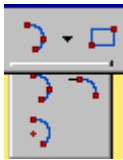
Right click the mouse while in 'Line' mode, this will bring up a submenu to assist in the construction of your sketch.

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.



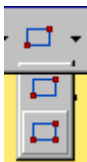
Circle/Ellipse

The default circle tool creates a circle using Center point and radius. Access the drop down toolbar to see that there are two other circle options, ellipse and Tangent, Tangent, Tangent.



Arc

The default arc tool creates an arc using three points. The drop down toolbar provides two additional options: Center, Start, End and Start, Tangent. All three methods will draw arcs either clockwise or counter-clockwise.



Rectangle

The rectangle tool provides two options. The default is to select the two opposite corners of the rectangle. The second option has the user define the length of one side and then the length of the adjacent side.

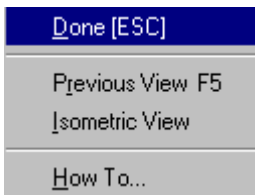


Fillet/Chamfer

The Fillet tool is actually a flyout that includes Fillet and Chamfer. 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.



The fillet 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.



Right-clicking the mouse while in 'Fillet' mode brings up a small dialog box that allows the user to switch views on the fly to allow for easier editing.



Equal Distance

2 Distance

Distance-Angle

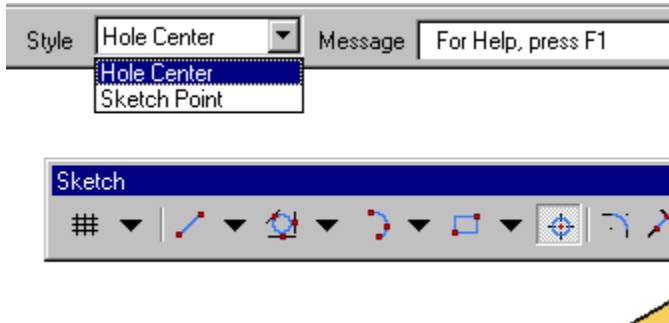
Chamfers can be defined in three ways: Equal Distance, 2 Distance, and Distance-Angle.

The user also has the option of selecting an existing chamfer in the sketch and applying that value to the chamfer being defined.



Point, Hole Center

The point tool is used to determine the location of holes as well as points.



To create a Sketch Point (used to constraint geometry), select the Sketch Point under Style.

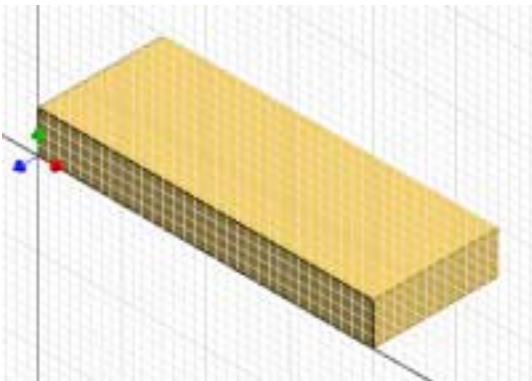


The next section of the Sketch toolbar contains Mirror and Offset.



Mirror

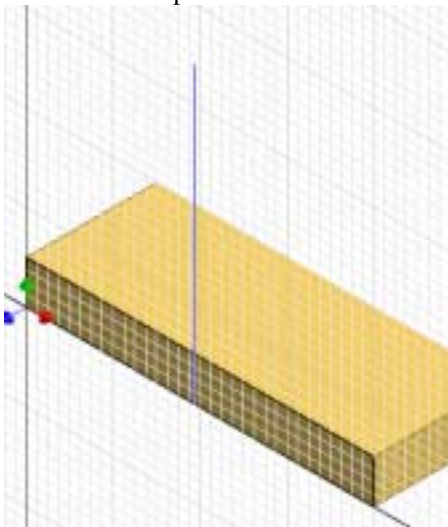
Mirror is a new tool introduced in Release 4.



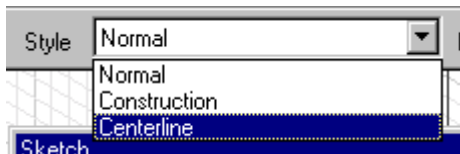
We create a simple block: 4 units by 1.5 units x 0.5 units thick on the top (XZ) plane.



Select the front plane for a New Sketch.



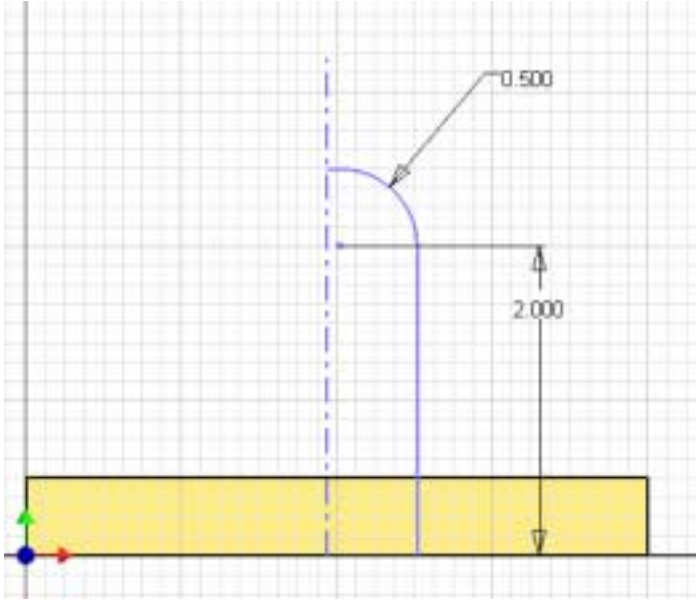
Draw a vertical line roughly in the center of the front side.



Select the line and change it to a centerline.

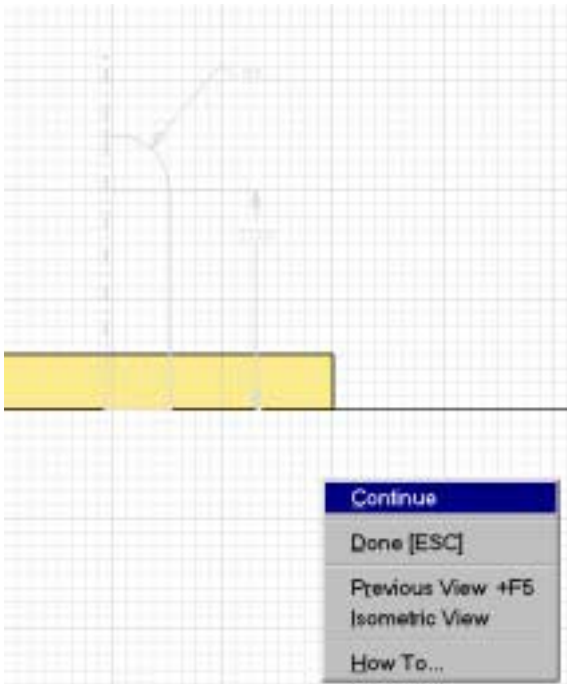


The line now appears as a centerline.

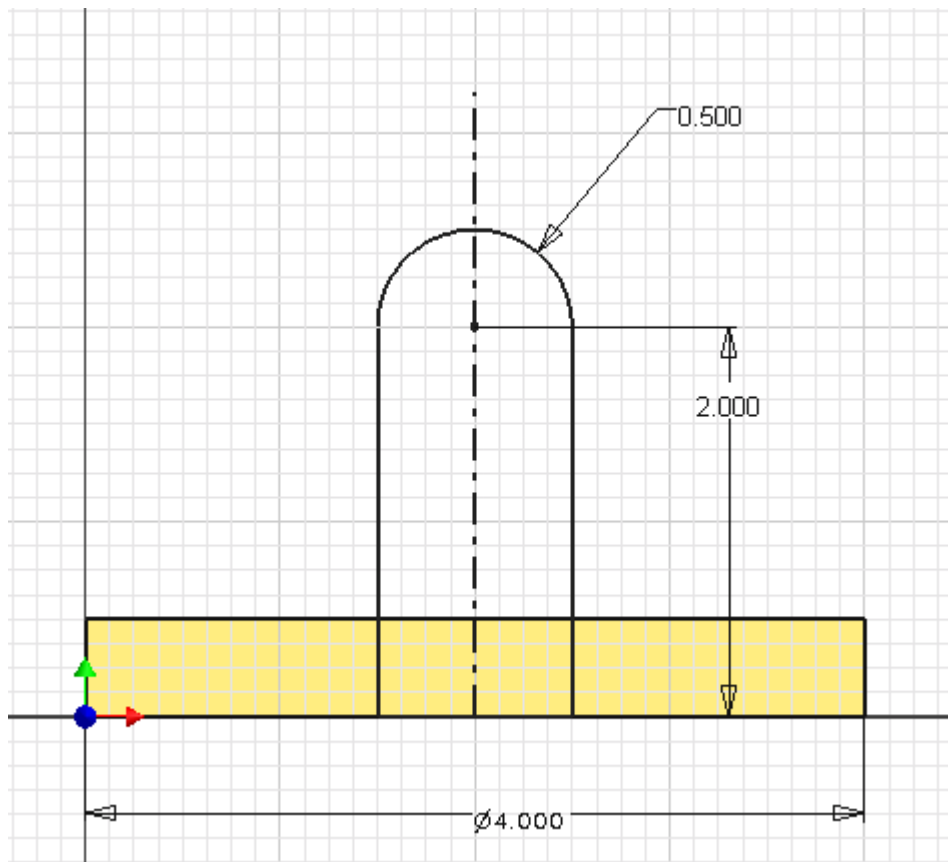


Create the sketch shown. There is a small horizontal line at the bottom of the sketch that will be used to close the profile. Add a coincident constraint between the arc center and the center line.

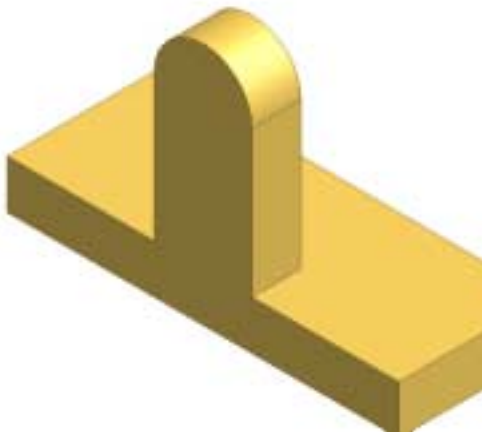
Select the Mirror tool. Then select the entire sketch. You can select the entire sketch by drawing a window around the sketch and centerline.



Right click and select 'Continue'.



Our geometry is now mirrored.

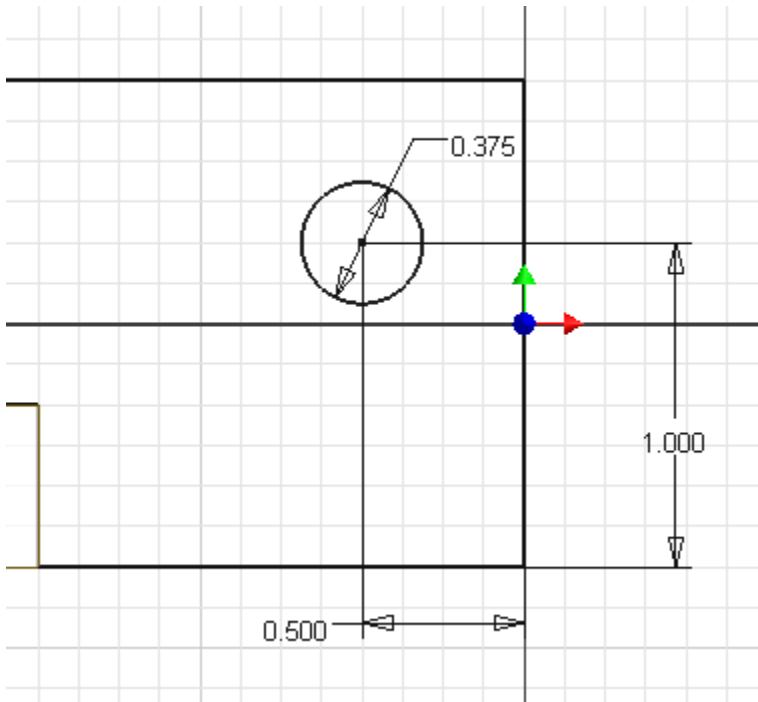


Extrude the geometry into the block a Depth of 0.5 units.

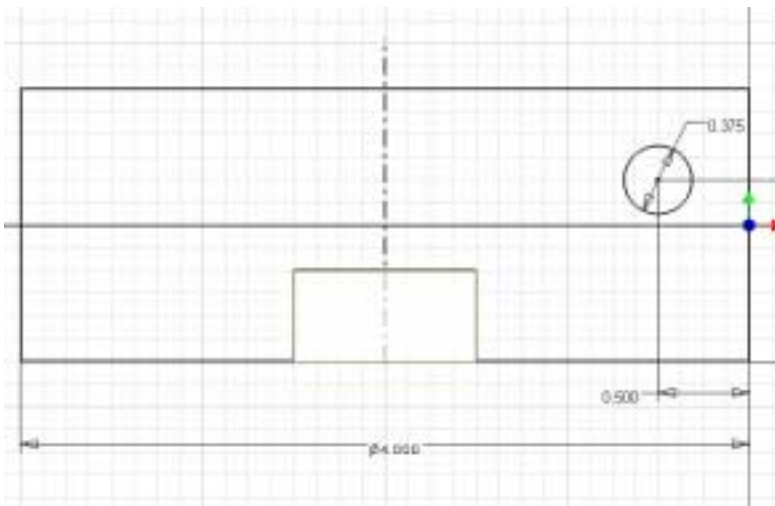
If you have difficulty extruding the profile, there may be some gaps between the arcs and lines. Use the Design Doctor to assist you in creating a closed profile.



Select the top face for a New Sketch.



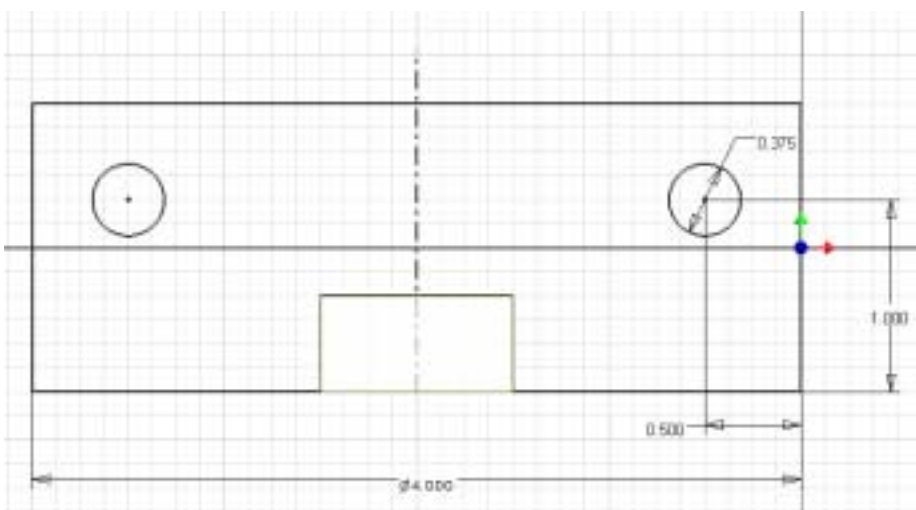
Draw a circle on the right side as shown.



Create a centerline. Draw a vertical line. Select it and set the Style to Centerline. Add a 4.00 dimension to center it on the block.



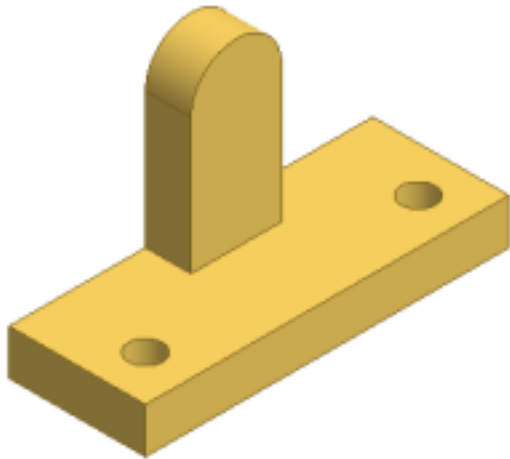
Select the Mirror tool. Select the Centerline and the circle.



Right click and select 'Done'.

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

Extrude the holes through all.



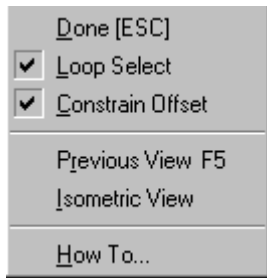
Our completed exercise.
Save this as 'Mirror-Exercise.ipt'.



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.



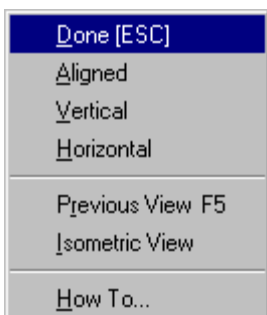
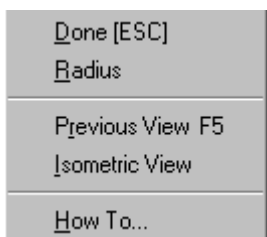
Our next section contains Dimensioning tools.



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 sub-menu. This submenu allows you to switch from Radius mode to Diameter mode simply by selecting that option.

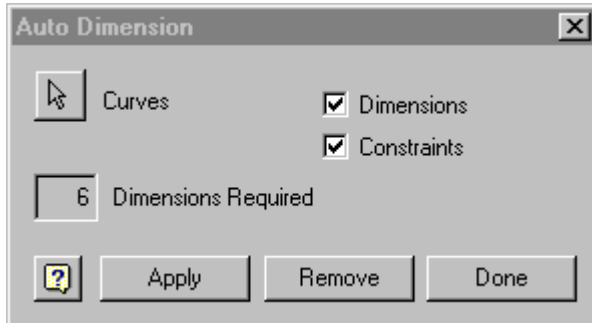


When dimensioning a line, right clicking the mouse will bring up a submenu with the options for aligned, vertical or horizontal linear dimensions.

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



Auto Dimension



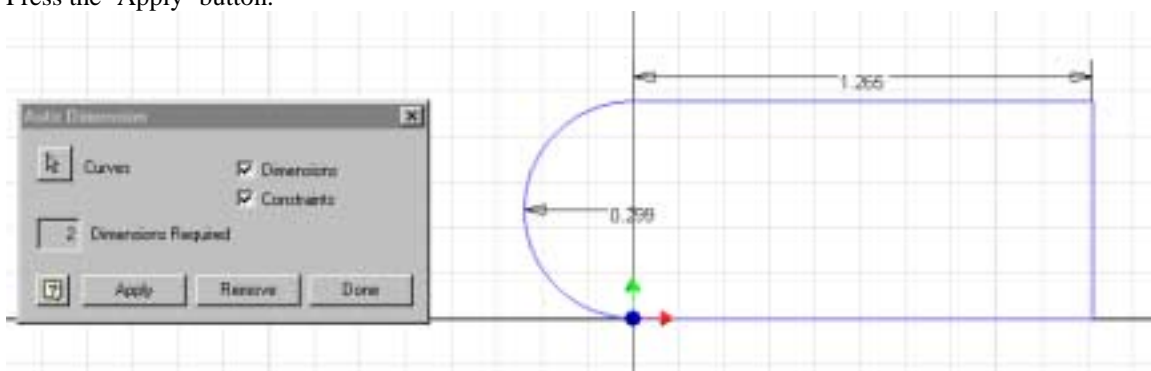
Auto Dimension tells the user how many dimensions are required to fully define a sketch and applies constraints as needed.



Draw the sketch shown.



Select Auto Dimension.
Press the 'Apply' button.



The dimensions appear as shown. (Note: Your dimensions will probably be different depending on how you drew your sketch.)

Press 'Done'.

The dimensions are then placed. You can now select dimensions and edit them as needed.



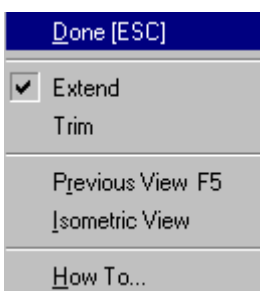
Modify Tools

Our next section contains tools used to modify sketches: Extend, Trim, Move, and Rotate.



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

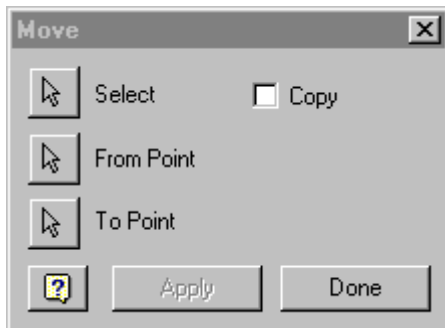
The trim tool prompts the user to select the object to trim and automatically uses any intersecting edges as the cutting tool. Inventor previews the modification in red for the user and the user accepts by left clicking the mouse. A right mouse click brings up the same submenu as the extend right mouse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



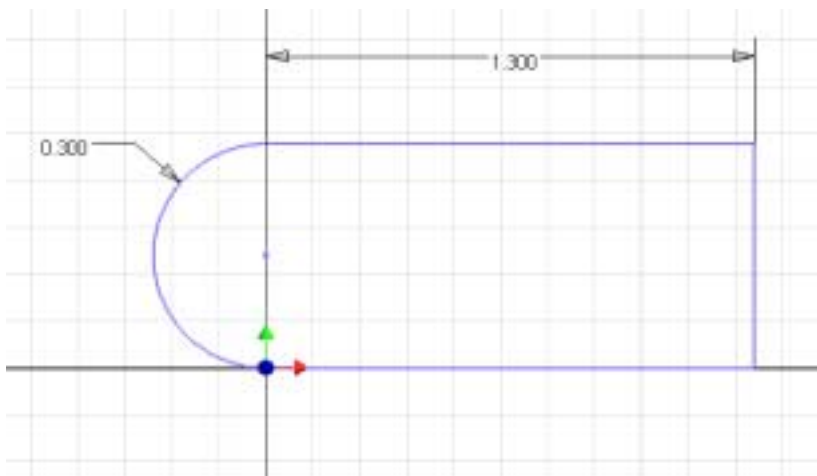
TIP: Press and hold SHIFT to temporarily enable Trim when in Extend mode or to enable Extend when in Trim mode.



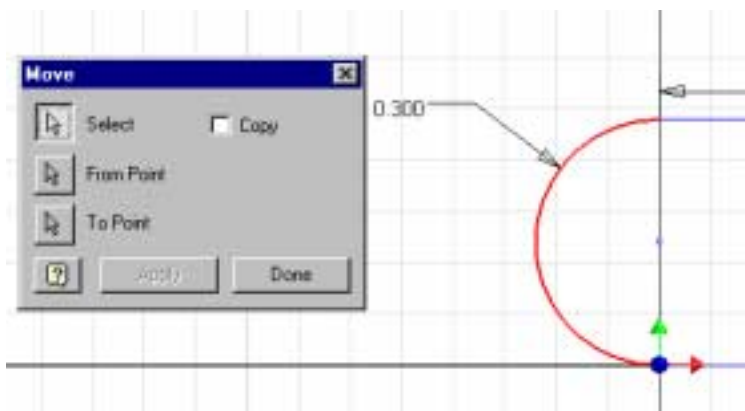
Move



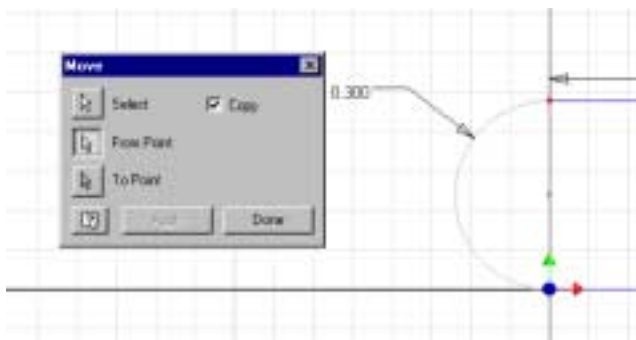
Pressing this tool brings up this dialog box.



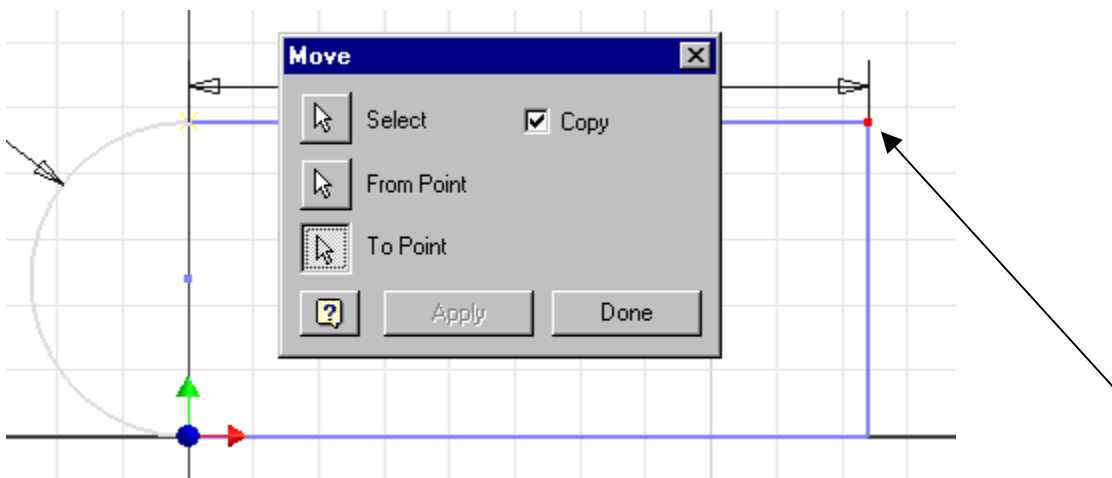
Using the sketch we used for the Auto Dimension exercise, we will use the Move tool to copy the arc.



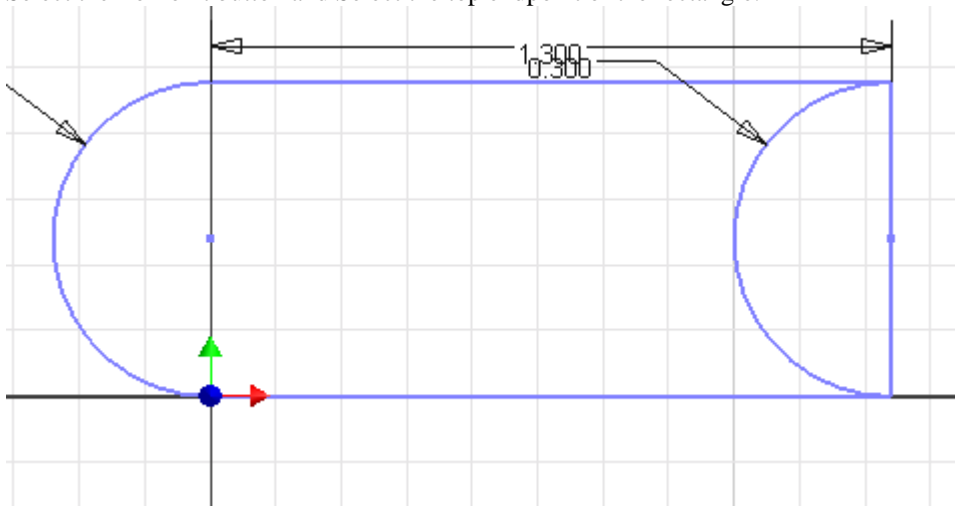
Press the Select button and select the arc. Enable the Copy button.



The arc will grey out to indicate it has been selected. Press the From Point button and select the top arc endpoint.



Select the To Point button and Select the top endpoint of the rectangle.



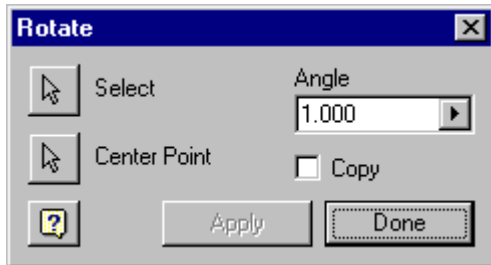
Press 'Apply' and 'Done'.
The arc is now copied to the new position.



Rotate

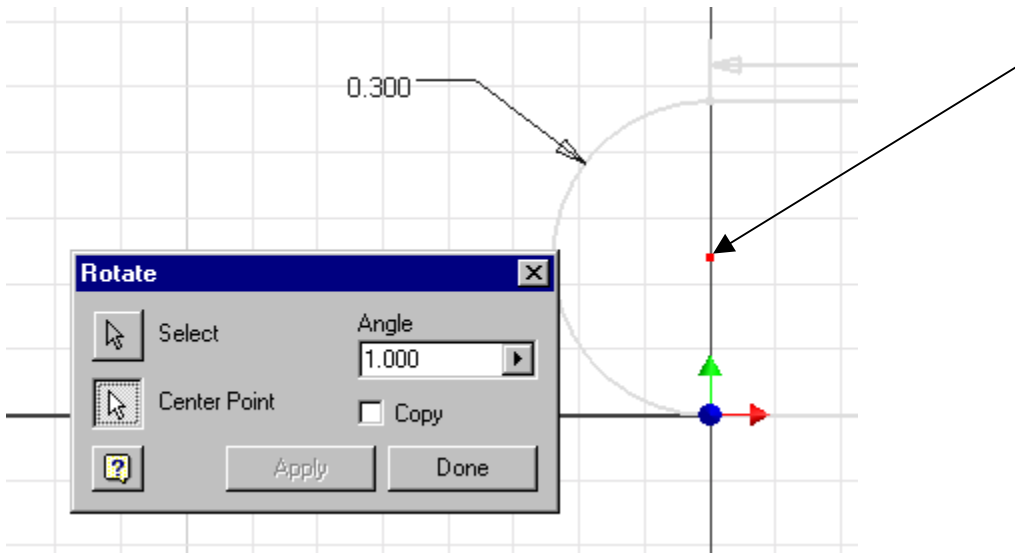
We will now rotate the sketch we just modified.

Select the Rotate tool.

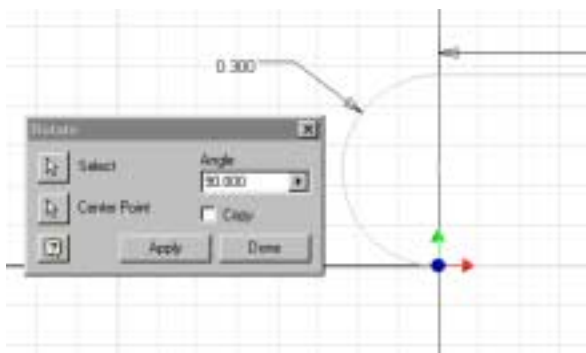


The Rotate dialog box appears.

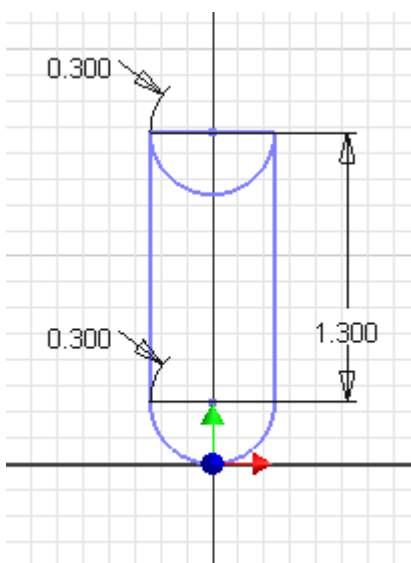
Press the Select button and window around the entire sketch.



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



Change the Angle value to 90.
Press 'Apply' and 'Done'.



Our rotated sketch.



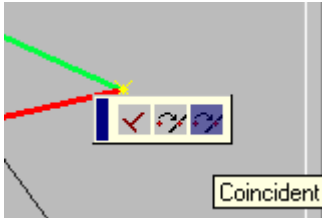
Constraints

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, Coincident, and Concentric. The bottom row of constraints from left to right are Collinear, Horizontal, Vertical, Equal, and Fixed.

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.



Inventor R4 features three projection tools: Project Geometry, Project Cut Edges, and Project Fat Pattern.



Project Geometry

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.



TIP:

- ◆ 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



Project Cut Edges

This tool projects edges cut by the sketch plane onto the current sketch plane.



Project Flat Pattern

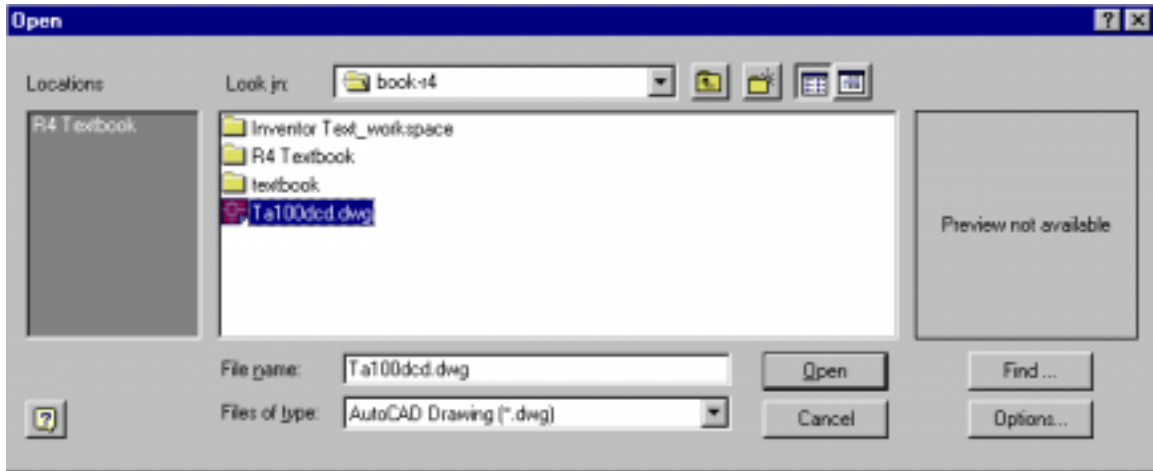
This tool is greyed out unless a flat pattern exists. If a flat pattern is available, the user may select a face to project it onto a selected plane.



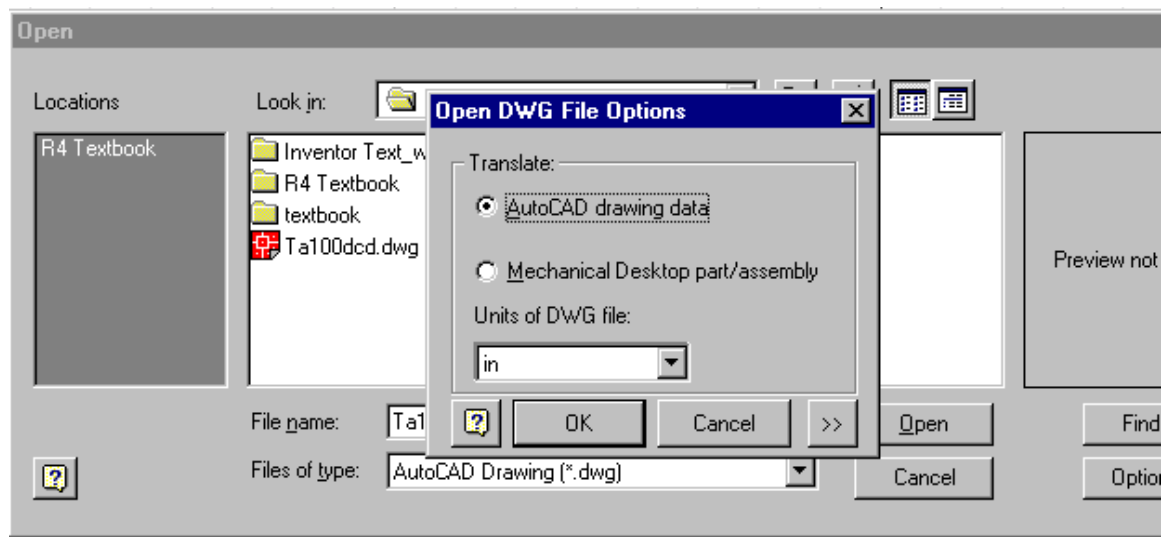
Insert AutoCAD file

AutoCAD remains the Number One 2D drafting software package in the world. Many companies would like to move into the 3D world, but the ability to use existing AutoCAD drawings is a major concern. This new tool in Inventor Release 4 should go a long way in alleviating those concerns.

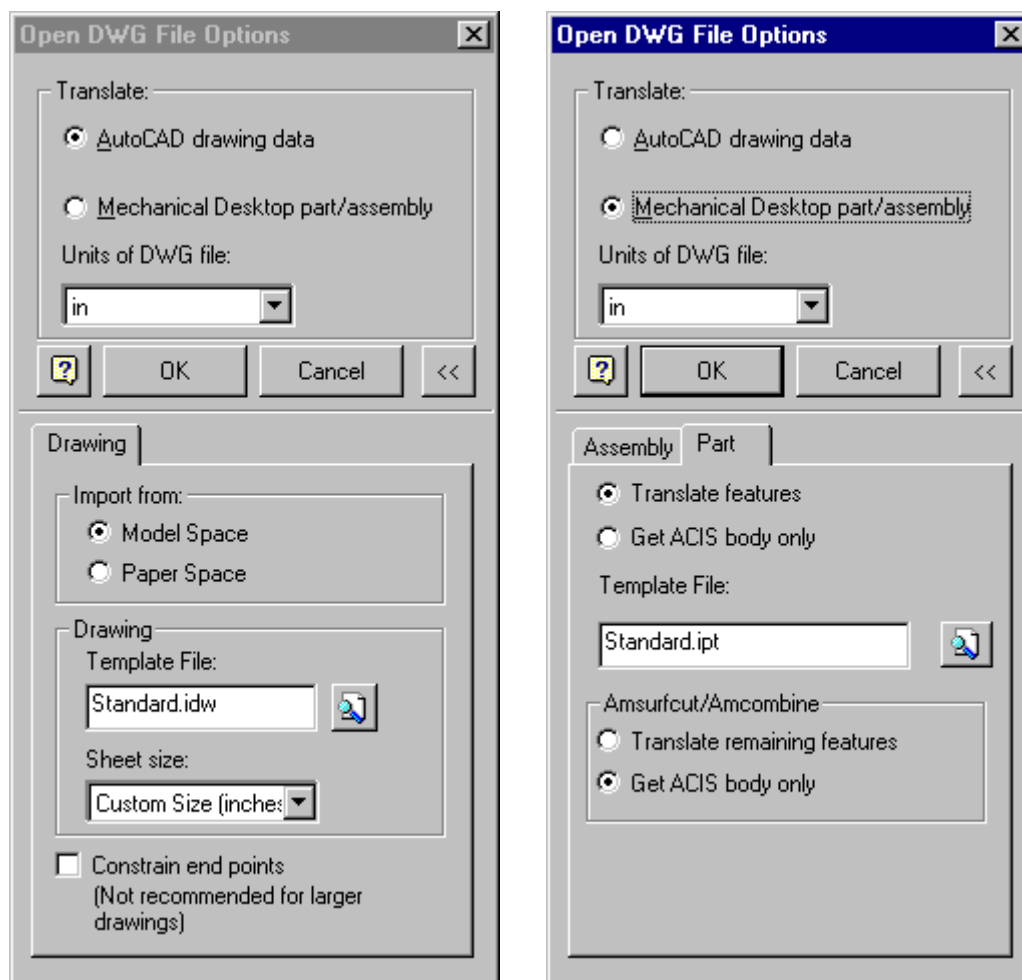
To demonstrate how it works, we use a drawing from Nidec's fan catalog, but any AutoCAD drawing will do.



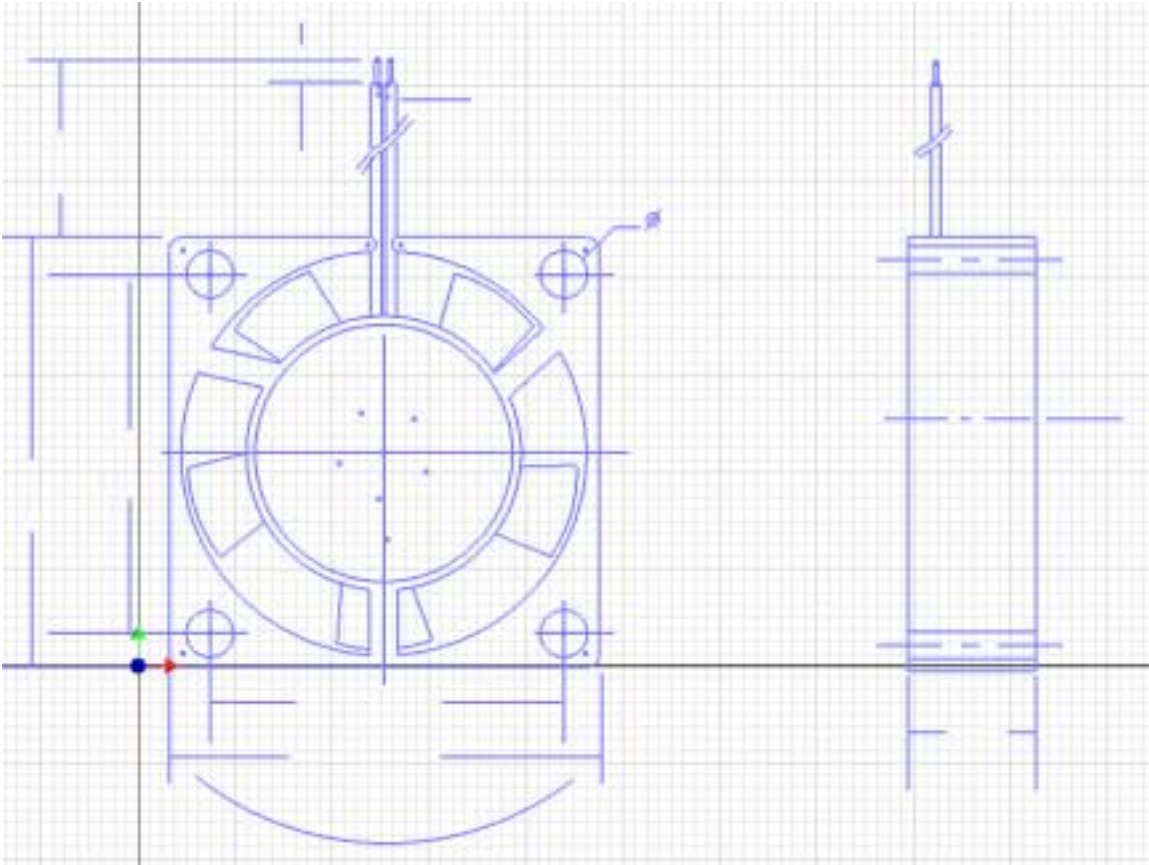
Select the Insert AutoCAD file tool. The browser dialog will come up.



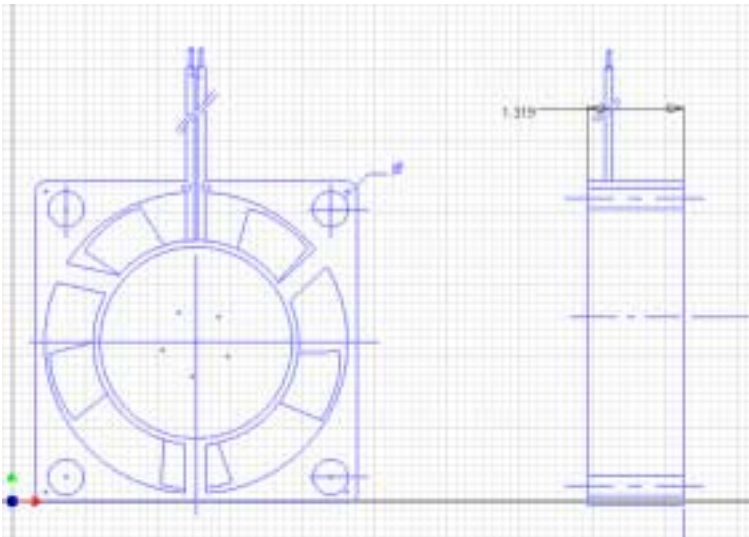
If you select Options, you will get the Open DWG File Options. You have a choice importing a Mechanical Desktop or AutoCAD file.



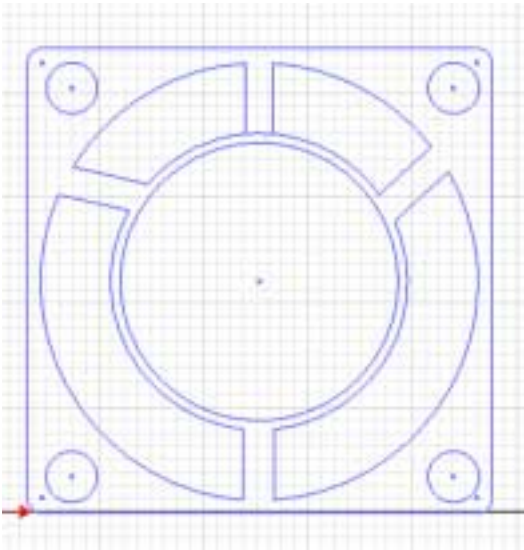
Pressing the More Button provides even more options. Depending on whether the file is an AutoCAD or Mechanical Desktop file, we can determine how Inventor will interpret the data imported.



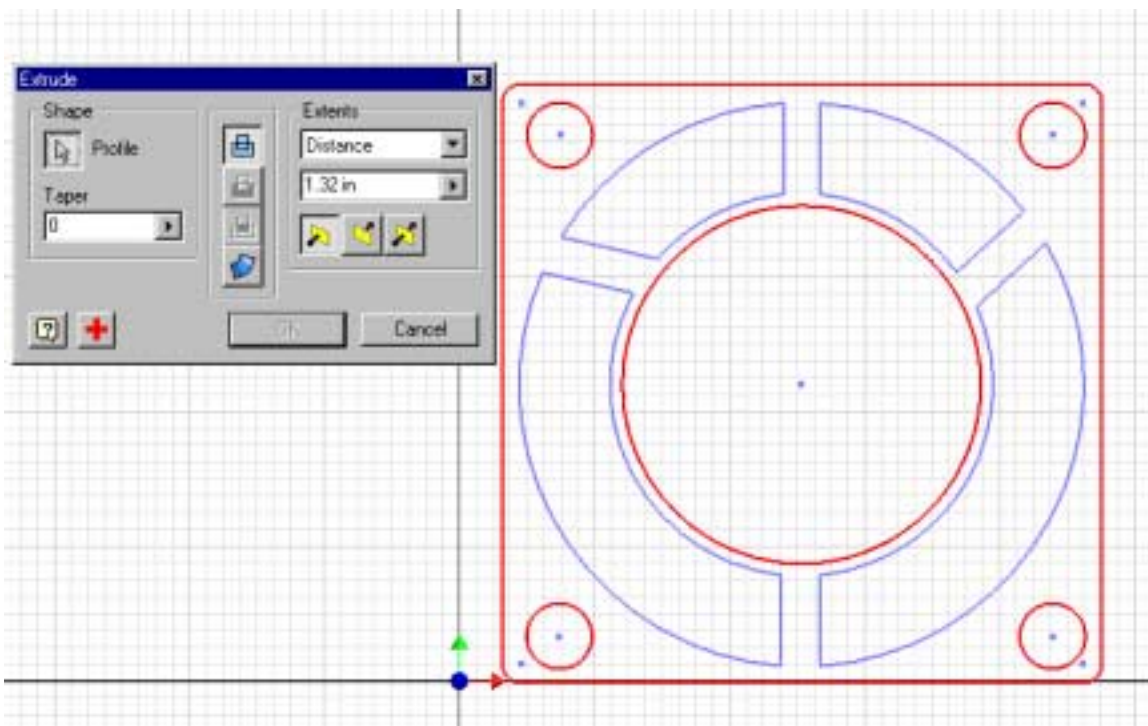
Here is the AutoCAD drawing once the import process is completed.



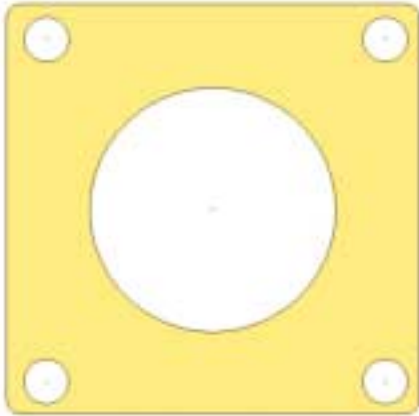
By holding down the Control key and picking with the left mouse, we can select all the dimension lines, then right click and press 'Delete'. We also need to delete the side view.



Continue cleaning up the sketch until you have a basic profile. Use the Design Doctor to assist you in creating a closed loop profile.

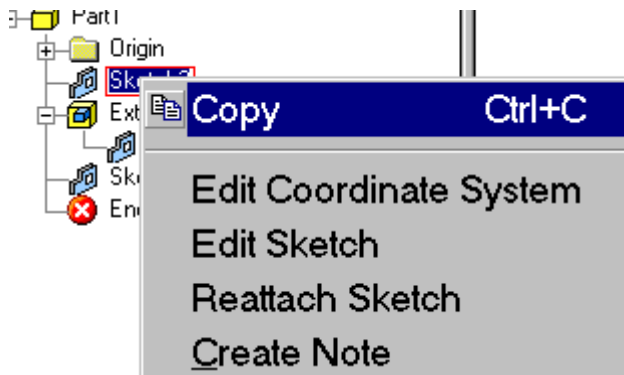


Select 'Extrude'.



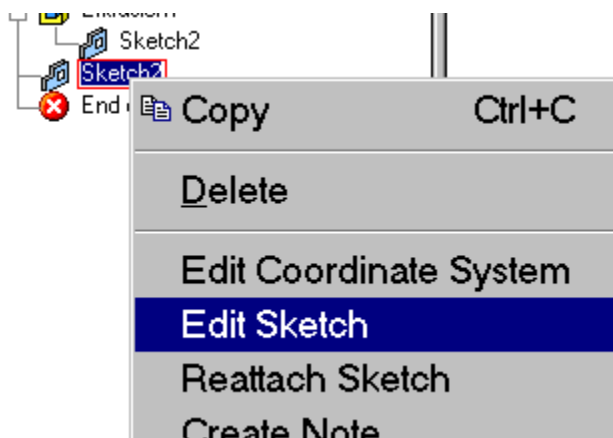
Our base feature.

But what about the cuts for the fan blades?

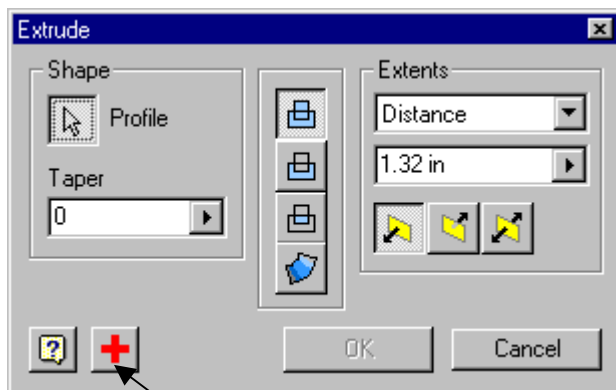


Select the sketch, right click and select 'Copy'.

Select the front face of the extrusion, , right click and select 'Paste'.

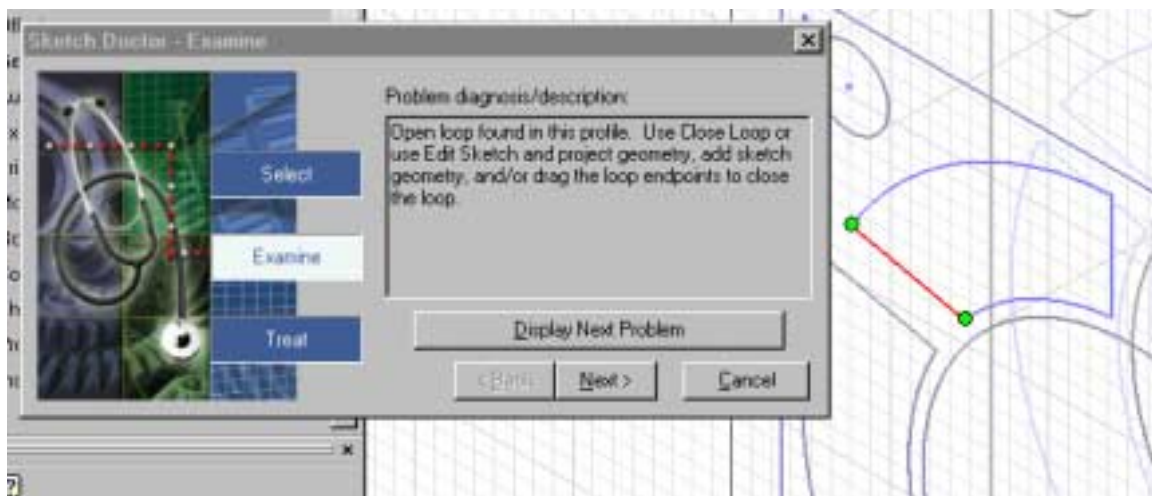


Select the Sketch, right click and select 'Edit Sketch'.



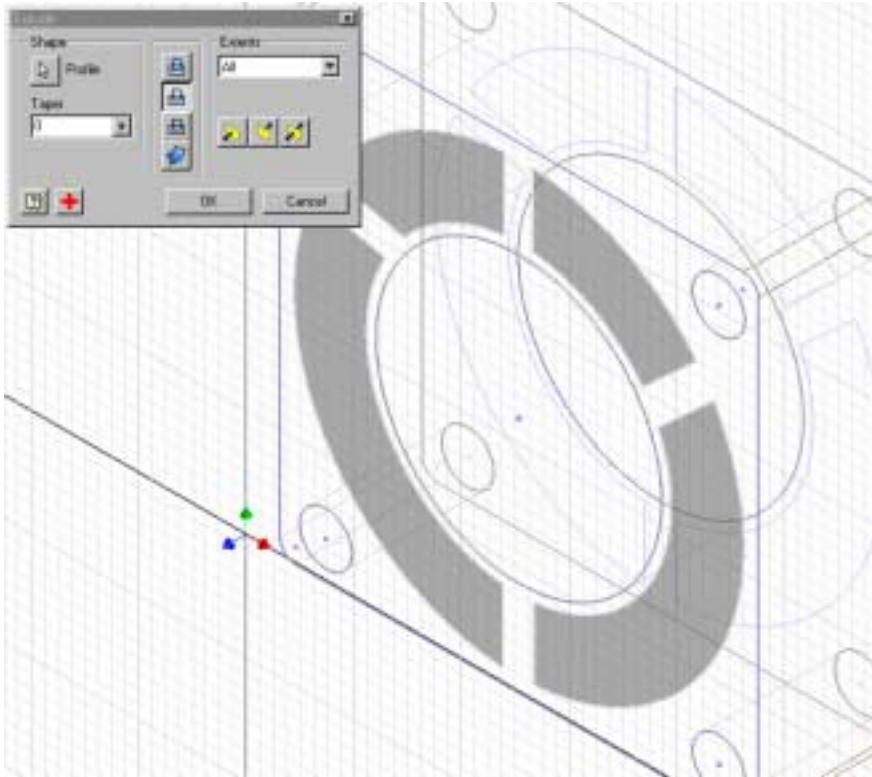
Sketch Doctor

Select the Extrude tool and then select the Sketch Doctor.



The Sketch Doctor will step you through the process of creating closed profiles of the imported geometries.

Once all the profiles have been resolved, you will be able to select the profiles to extrude.



Select the profiles and define as Cuts Through All.



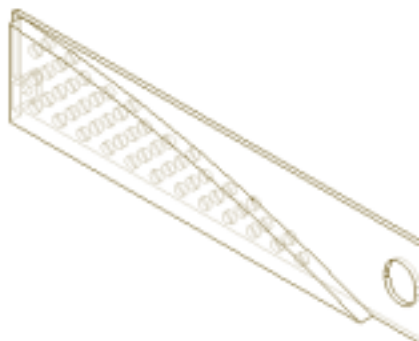
We have successfully transformed an AutoCAD 2D drawing into a 3D Parametric part in minutes.

Reattach Sketch

Use Reattach Sketch to move an existing sketch to a different planar face or work plane than the one on which it originated, then use constraints and dimensions to position it and modify its size as needed.

1. In the browser, select the sketch you want to attach to a face or plane.
2. Right-click and select Reattach Sketch.
3. Click the face or plane on which to attach the sketch.

If the sketch is constrained or dimensioned to the plane on which it originated, the constrained geometry is included when the sketch is moved to a different plane. You can delete extraneous geometry as needed.



In the model shown, the holes are located on the middle plane, so that they do not go all the way through the part. We can move the holes to the top plane using 'Reattach Sketch' and then edit the feature to ensure that the holes cut through the part. Our other choice would be to delete all the holes and start over.



We locate one of the holes to be moved. Right click and select 'Reattach Sketch'.



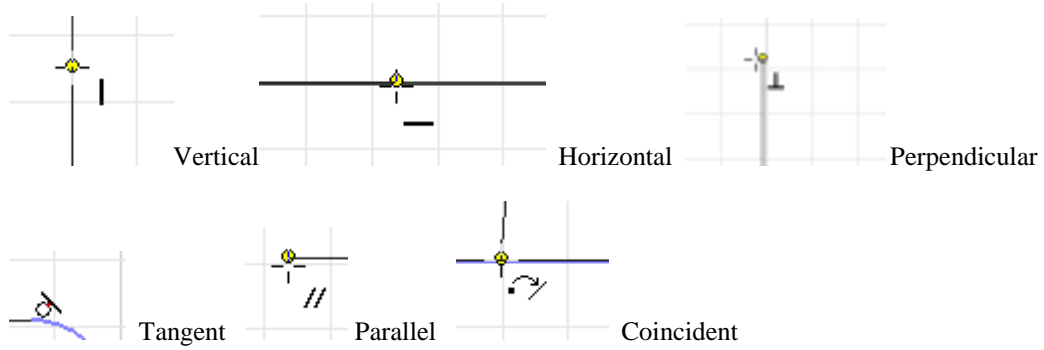
We then select the plane to move the sketch.





















We now see that the holes go through the part.




















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 less edits required.



Sketch Tools

Button	Tool	Function	Special Instructions
	Edit Coordinate System	Use for creating isometric or angled sketches	
	Line	Create line segment	Select NORMAL or CONSTRUCTION from the Style Menu
	Spline	Create spline	
	Circle	Create circle using center point and radius	Select NORMAL or CONSTRUCTION from the Style Menu
	Circle	Create circle tangent to three lines or arcs	
	Circle	Create ellipse	
	Arc	Create 3 point arc	Select NORMAL or CONSTRUCTION from the Style Menu
	Arc	Create arc with center and two endpoints	
	Arc	Create arc tangent to a line	
	Rectangle	Use corner method to create rectangle	
	Rectangle	Create rectangle with three orthogonal points	
	Fillet	Create fillet by entering a radius and selecting two lines or arcs	Radius controlled by dialog box entry
	Chamfer	Create chamfer. Three options available: Equal Distance, Two Distances, and Distance-Angle	
	Point, Hole Center	Position the center point for a hole or sketch point	Select Hole Center (default) or Sketch Point from the Style Menu
	Mirror	Mirrors Geometry about a centerline	Requires Centerline
	Offset	Create parallel lines/curves at a specified distance	
	General Dimension	Apply dimensions to sketches	Use the Right click button to select the type of dimension to apply.
	Auto Dimension	Applies dimensions to selected sketch geometry	

Button	Tool	Function	Special Instructions
	Extend	Extend a line/curve to intersect with the nearest line/curve/point.	Press and hold SHIFT to temporarily enable Trim.
	Trim	Trim a line/curve	Press and hold SHIFT to temporarily enable Extend.
	Move	Moves/Copies Selected Geometry to a new location	
	Rotate	Rotates/Copies Selected Geometry to a new location	
	Add Constraint	Perpendicular	
		Parallel	
		Tangent	
		Coincident	May be applied to lines, points, or arcs.
		Concentric	
		Collinear	May be applied to lines or axes
		Horizontal	
		Vertical	
		Equal	
		Fixed	
	Show/Delete Constraint	Show applied constraints or delete existing constraints	Position the cursor over the constraint and select DELETE. Use the OTHER option to cycle through multiple constraints.
	Project Geometry	Project geometry onto another sketch	
	Project cut edges	Project onto a sketch plane all edges of a selected part that intersect the sketch plane	
	Project Flat Pattern	Project a flat pattern onto a selected plane	
	AutoCAD drawing	Inserts an AutoCAD drawing into a sketch	Use Sketch Doctor to create closed profiles

Review Questions

- A.  B.  C.  D. 

Identify the geometric constraint

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

ANSWERS: 1) D; 2) B; 3) A; 4) C; 5) A; 6) C; 7) A; 8) B; 9) B; 10) A

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