Autodesk Inventor R4 Fundamentals: Conquering the Rubicon

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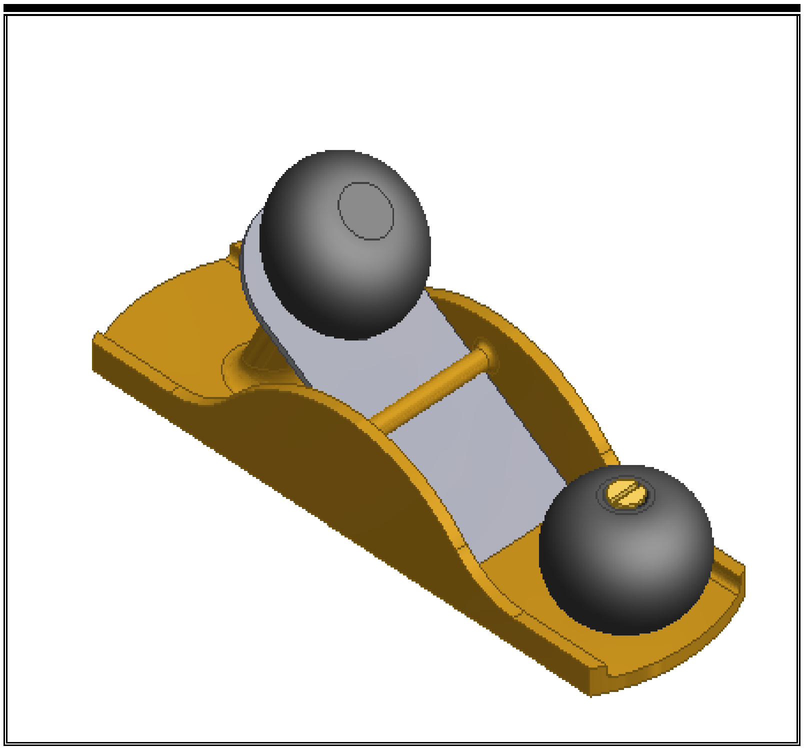
autodesk
authorized author



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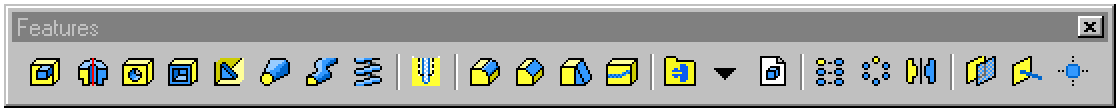
Lesson 5

The Features Toolbar

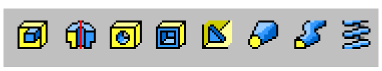


The Features toolbar has six sections. They are Sketched feature tools, Thread, Placed feature tools, Design Element tools, Pattern Feature tools, and Work features.

The Sketched feature tools use sketched geometry to build features.
The Thread tool maps the image of a thread onto a cylindrical face.
The Placed feature tools add finishing touches to existing features.
Design element tools create or add predefined features to a model.
Pattern feature tools add multiple instances of existing features.
Work features create work planes, work axes, or work points that you can used to define other features in a model.

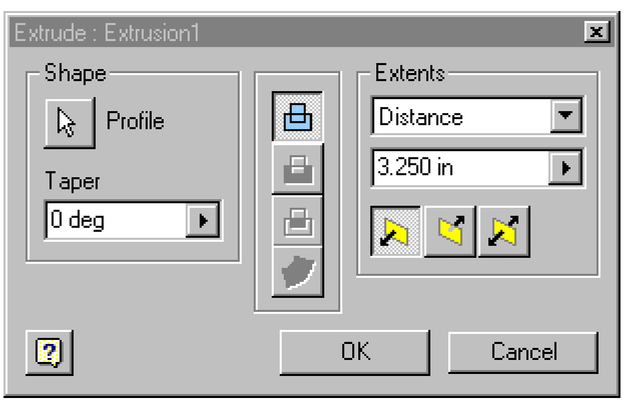
You'll notice that when you start your part most of the Features tool bar is grayed out. That's because Inventor automatically disables any tools you don't need until you need them. Those tools will become available once you have created the appropriate features.

Sketched Feature Tools



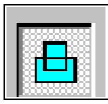
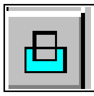
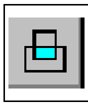
The eight options of this section are Extrude, Revolve, Hole, Shell, Rib, Loft, Sweep and Coil.

Extrude

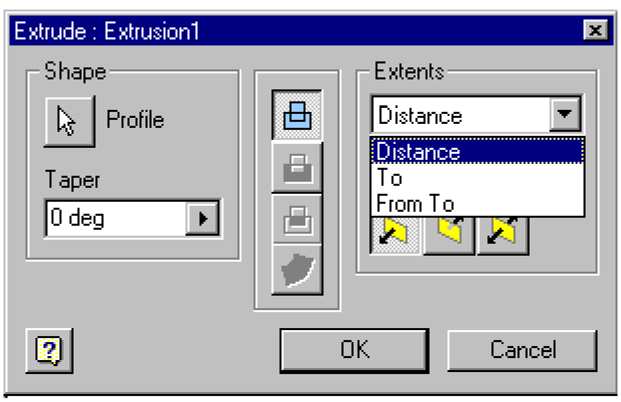


The Shape button selects a profile to extrude. If there are multiple profiles and none are selected, click Profile and then click on one or more profiles in the graphics window.

Inventor allows you to extrude more than one profile at a time. When you select a profile it will highlight to indicate that it is part of the selection set. To remove a profile from the selection set, hold down Ctrl and click a profile. If your selection set is not what you want and you were unable to de-select a profile, just press on the Cancel button and start over.

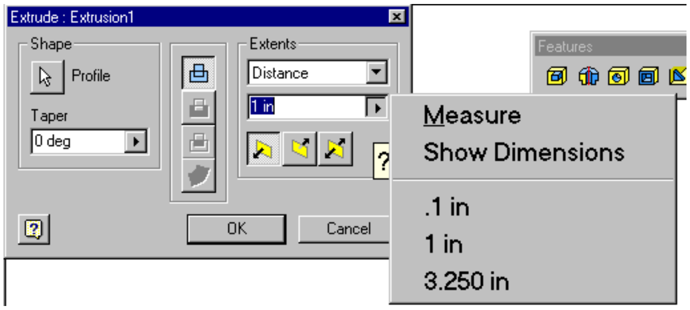
	Join Adds the volume created by the extruded feature to another feature.
	Cut Removes the volume created by the extruded feature from another feature.
	Intersect Creates a new feature from the shared volume of the extruded feature and another feature. Material not included in the shared volume is deleted.
	Surface Creates a 3D surface.

The four buttons in the middle of the Extrude dialog box control the type of extrusion. The top button performs a ‘Join’ operation. The second button performs a ‘Cut’. The third button performs an ’Intersect’. The bottom button creates a surface. You can drag the previewed profile using your mouse to determine the distance you want to extrude or enter a value into the dialog box under extents. The operation is automatically previewed in the graphics window.



Under the Extents section of the dialog box, there is a drop down that allows you to define how far you want to extrude the profile. Your options are Distance, To, and From To.

Distance Default method. Establishes the depth of extrusion between start and end planes. For a base feature, shows negative or positive distance of extruded profile or entered value. The extrusion end face is parallel to the sketch plane.
To Allows you to select an ending face or plane on which to terminate the extrusion. You may terminate the feature on the selected face, or on a face that extends beyond the termination plane. In an assembly, the face or plane may be on another part. After you select the termination plane, choose one of the following: • Click the left button to terminate the feature entirely on the selected face. • Click the right button to terminate the feature on the selected plane and the extended face.
From - To Allows you to select beginning and ending faces or planes on which to terminate the extrusion. In an assembly, the faces or planes may be on other parts. Not available for base features.



Inventor automatically saves your most common values. You can also measure objects on the fly to determine what value you want to use. You can use the Show Dimensions to select an existing dimension on the model to be used for the feature being defined.

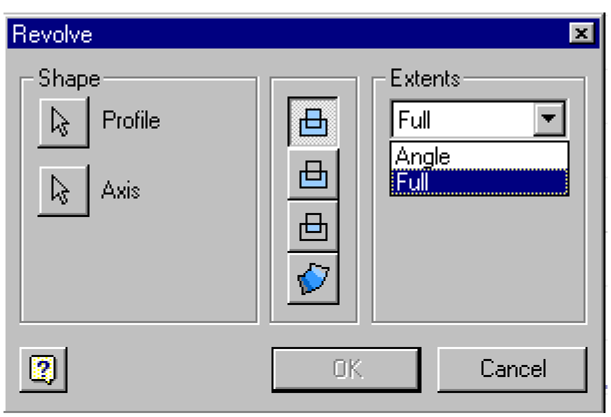
The buttons on the bottom of the Extents section determine the direction for the extrusion. Inventor will automatically preview, so you can see if you have selected the correct direction.

The Taper Angle sets a taper angle of up to 180 degrees for the extrusion (normal to the sketch plane). The taper extends equally in both directions. If a taper angle is specified, a symbol in the graphics window shows the fixed edge and direction of taper.

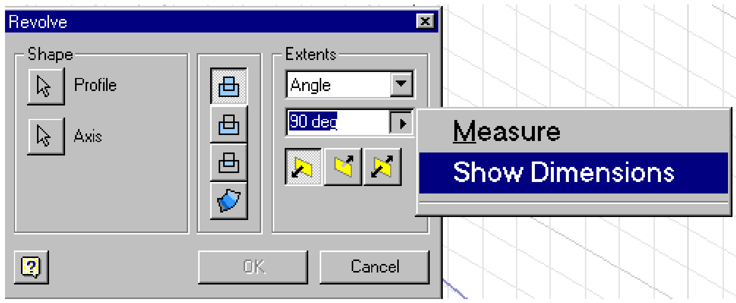
TIP: To taper a feature in only one direction, create an extruded feature with no draft, then use the Face Draft tool to add draft to a specific face.

Revolve

A revolve is done similarly to an Extrude. First you create the sketch using the Sketch toolbar. Then select the Revolve button from the Features toolbar. The Revolve dialog box will appear. Select the profile to revolve. Define the centerline or axis you wish to revolve around by picking the axis button and then selecting the appropriate centerline or axis. The axis may be a work axis, a construction line, or a normal line. The axis of revolution can be part of the profile or offset from it. The profile and axis must be coplanar.



Under Extents, you determine the method for the revolution and set the angular displacement of the profile around a centerline. Click the drop-down arrow to list the extent methods, select one, and enter a value. Revolutions can be a specific distance or can terminate on a work plane or part face.



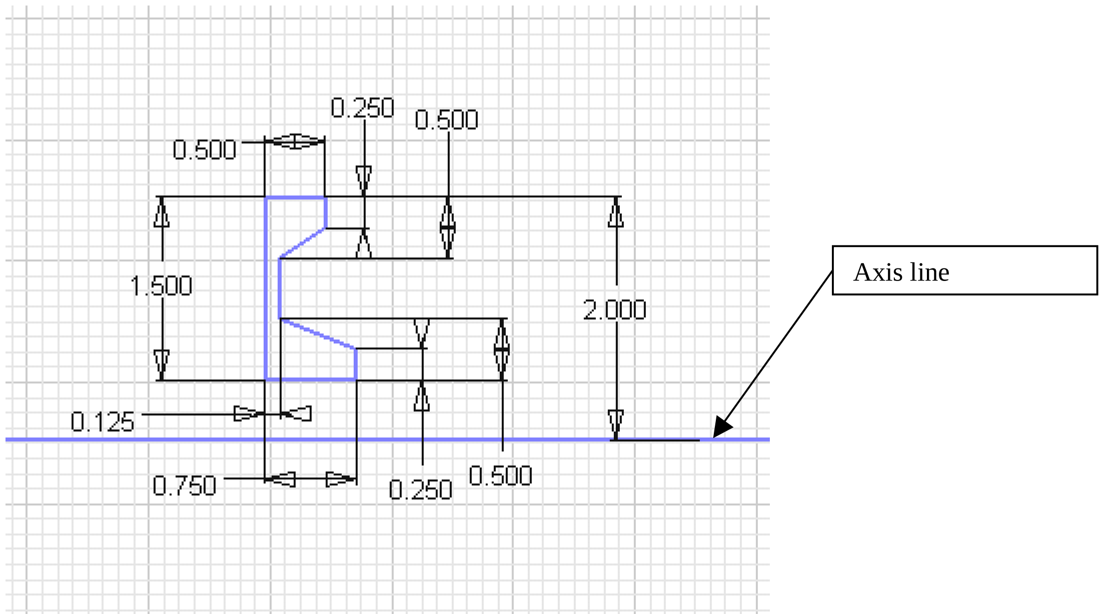
For the angle, you can measure for an angle, show dimensions to help you determine a value or simply type in an angle.

Finally, you can determine the direction for the revolve. Inventor will preview the revolve so you can decide if you have defined it properly before pressing 'OK'.

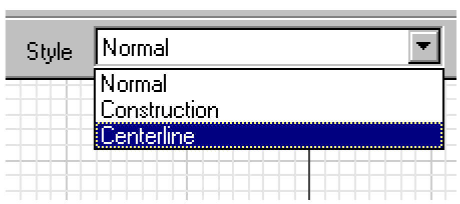
Creating a Revolve

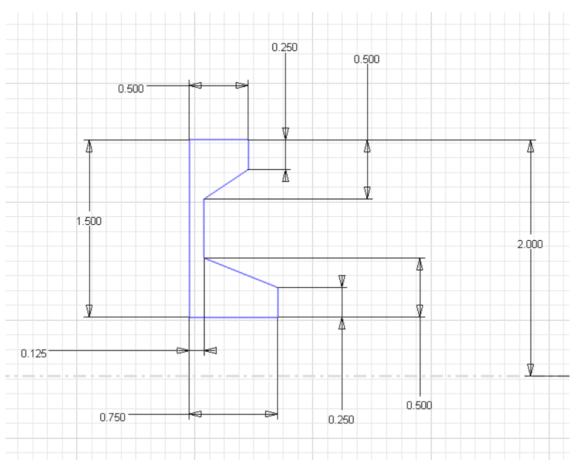
Go to File->New.
We begin our part by opening a new part under the English tab to ensure that our units are in inches.
Left-click on Standard(in).ipt.

Press 'OK'.

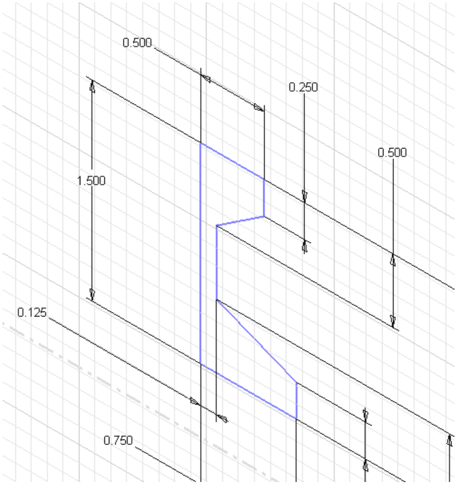


Draw the sketch shown. Note the axis line. This will be used to revolve around.
Inventor R4 comes with three line types: Normal, Construction, and Centerline.
We can identify the axis line as a centerline by selecting it and then selecting Centerline from the drop down.



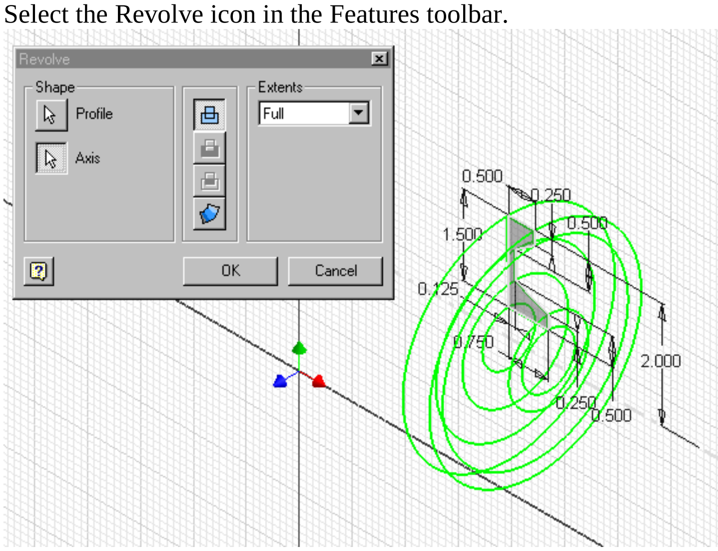


The axis line now changes appearance to a centerline.



Right click and switch to 'Isometric View'.

Select the Revolve icon in the Features toolbar.

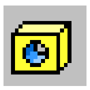


Verify that the correct profile is highlighted. If it is not, left-pick on the profile button in the Revolve dialog box and select the sketch. Because we defined our axis line as a centerline, Inventor automatically knows how to revolve our profile.

In the Extents drop-down, select 'Full' for a complete 360-degree revolution.

Press 'OK'.

Save this part as ‘wheel.ipt’. We will be using this part later in the text when we create patterns.

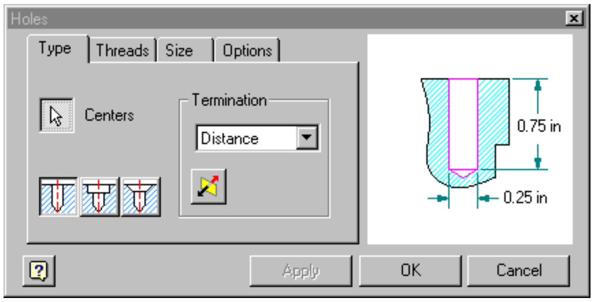


Hole

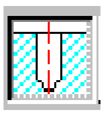
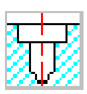
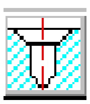
The Hole tool creates parametric drilled; counter bored, or countersunk hole features. A single hole feature can represent multiple holes with identical configurations (diameters and termination methods). Different holes can be created from the same, shared hole-pattern sketch.

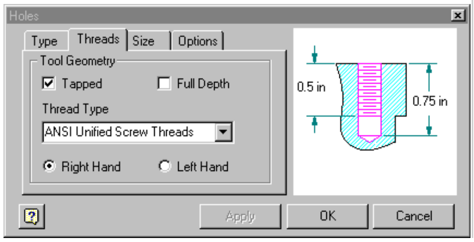
Before you place a hole, you have to place a point or hole center using the Sketch toolbar. Place the point, constrain it’s location and then you can proceed to the Features toolbar.   You can also select endpoints or center points on existing geometry as hole centers.

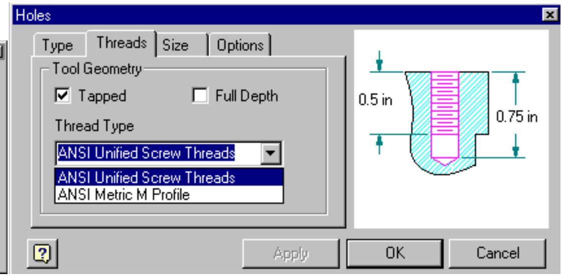
Inventor’s hole dialog box is extremely advanced and has several ‘Gee Whiz’ features. You can actually click inside the dialog box and modify the dimensions there. You can define more than one hole at a time simply by selecting on those hole centers and adding them to the selection set. So, for maximum efficiency, you can place all your hole centers using the sketch tool and then extrude them using the hole feature tool all in one shot.



Hole Types

The Type Tab selects hole centers and specifies hole type.		
Centers	Hole center points are automatically selected. Click to select endpoints or center points of geometry as hole centers.	
Hole type		Drilled holes are flush with the planar face and have a specified diameter. This is the default.
		Counterbored holes have a specified diameter, counterbore diameter, and counterbore depth.
		Countersunk holes have a specified diameter, countersink diameter and countersink depth.
Termination	Distance	Defines the termination method for the hole. Uses a positive value for the hole depth. Depth is measured perpendicular from the planar face.
	Through All	Extends a hole through all faces.
	To	Terminates a hole at the specified planar face.
Flip	Reverses direction of the hole.	

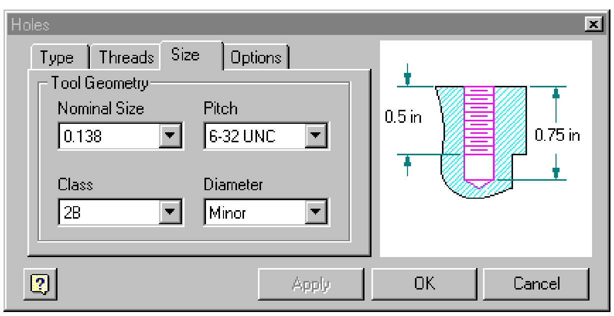




Hole Threads

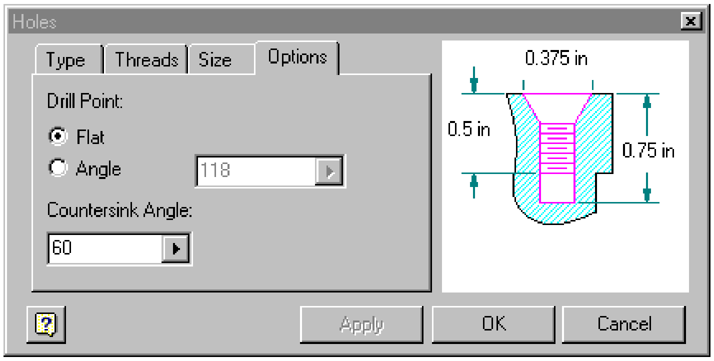
The Threads Tab sets values to define tapped holes. The hole is previewed according to the values you specify.	
Tapped	Select the check box to define hole threads.
Full Depth	Select the check box to specify threads the full depth of the hole.
Thread Type	Either ANSI or Metric. The default depends on which the user selected when Inventor was installed.
Right Hand/Left Hand	Determines the direction of thread.

The threads will not appear in your model, but the information will be used when you move into the drawing stage of your design. It will save you time and frustration to define any tapped holes here rather than relying on memory.



Hole Size

The nominal size and pitch callout pull downs display standard threads depending on whether the user set the drawing as metric or standard. The drafting standard can only be changed in a drawing file. You can decide to switch to a metric thread when you create your drawing file. Use Tools->Drafting Standards to set your options.

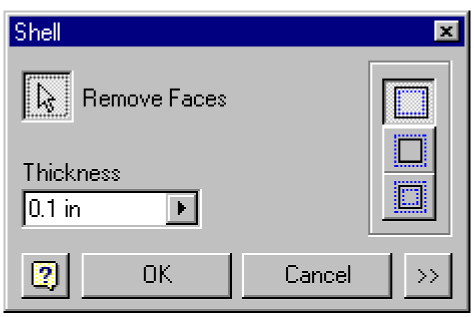


Hole Options

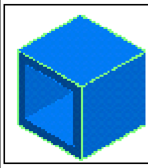
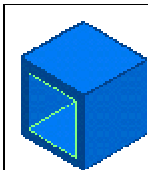
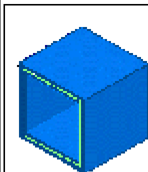
The Options tab defines the angle for countersunk holes and defines angle or flat tip for drill points.	
Countersink Angle	For countersunk holes, click the down arrow to specify the angle of the countersunk head or select geometry on the model to measure a custom angle. The positive direction of the countersink taper is measured counterclockwise from the hole axis, normal to the planar face.
Drill Point	Sets the angle of the drill-point taper to the end of the hole. Select Flat or Angle point. Click the down arrow to specify angle dimension or select geometry on the model to measure a custom angle, if applicable.

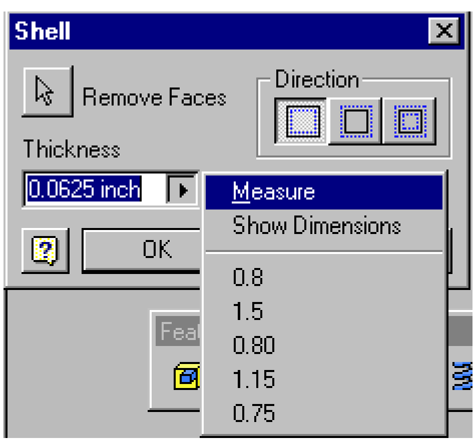
This dialog box grays out the Countersink Angle option unless Countersink is selected under Type.

Shell



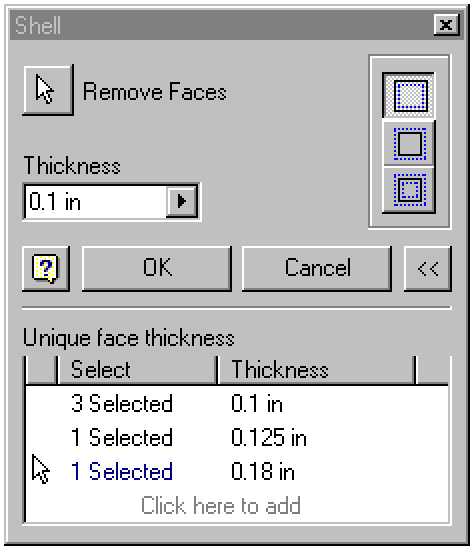
The Shell tool removes material from a part interior, creating a hollow cavity with walls of a specified thickness.

Remove Faces	Selects part faces to remove, leaving the remaining faces as the shell walls. Click to activate the part, then select the faces to remove. To reclaim a face, press and hold Ctrl and select the face. Selected faces are removed. Thickness is applied to remaining faces to create shell walls. If no part faces are selected for removal, the shell cavity is entirely enclosed within the part.
Direction	 Inside Offsets the shell wall to the part interior. The external wall of the original part becomes the external wall of the shell  Outside Offsets the shell wall to the exterior of the part. The external wall of the original part is the internal wall of the shell.  Both Sides Offsets the shell wall equal distances to the inside and outside of the part. Adds half of the shell thickness to the thickness of the part.
Thickness	Specifies the thickness to be applied uniformly to shell walls. Part surfaces not selected for removal become shell walls. If you need to use the thickness value in a parameter table, you can highlight the value in the box, then right-click to cut, copy, paste, or delete it.



The drop down on the Thickness bar allows you to measure a distance to determine the shell thickness or display dimensions to assist you in determining which thickness value to use. Inventor stores the most common values and displays them to aid you in making a choice.

The three direction buttons from left to right are Inside, Outside or Both.



Pressing on the double arrow button located on the lower right of the Shell dialog box enlarges the dialog box, providing the user with the ability to define sides with different thickness values. To remove items from the unique face thickness list, highlight and press the 'Delete' key in your keyboard.

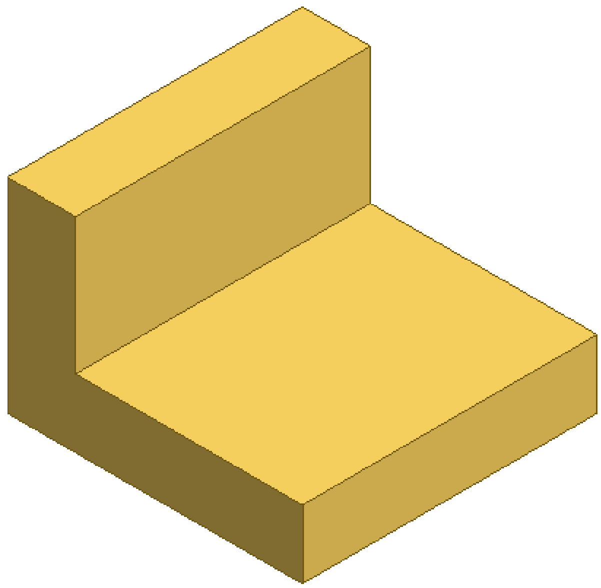
TIP: To reclaim a face, press and hold Ctrl and select the face.

Rib

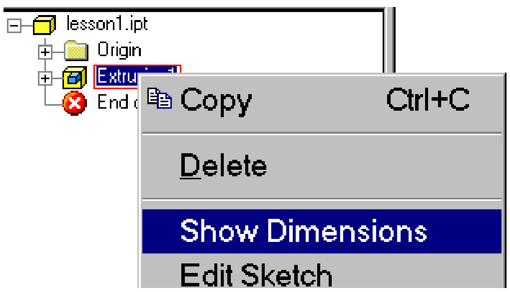
A Rib is a special type of extruded feature created from an OPEN sketched contour. It adds material of a specified thickness in a specified direction between the contour and an existing part.

Adding a Rib

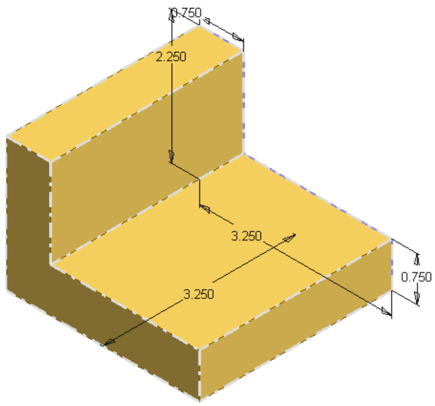
Open the file created in Lesson 1.



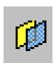
We start by creating an offset work plane in the middle of the bracket.



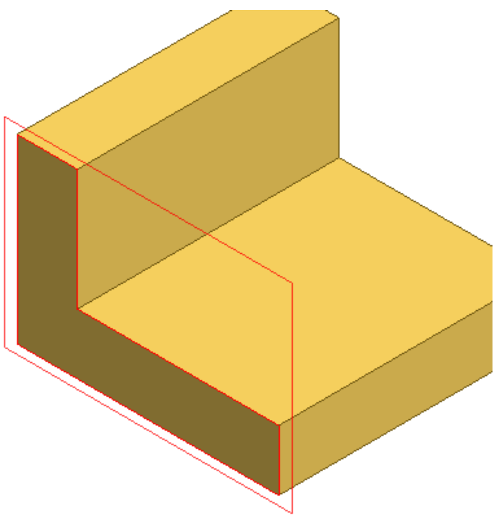
Highlight the extrusion in the browser. Right click and select ‘Show Dimensions’.



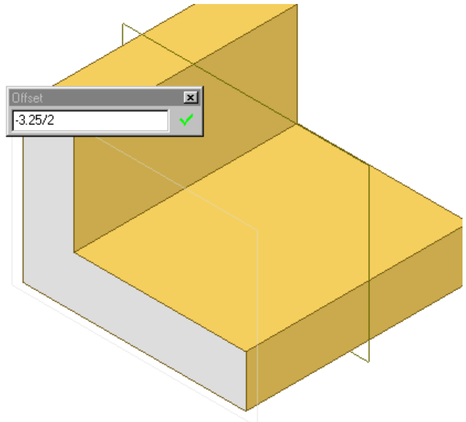
The extrusion’s dimensions are now visible. We see that the width is 3.25 units.



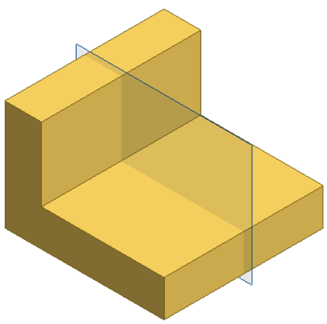
Select the Work Plane tool from the Features toolbar.



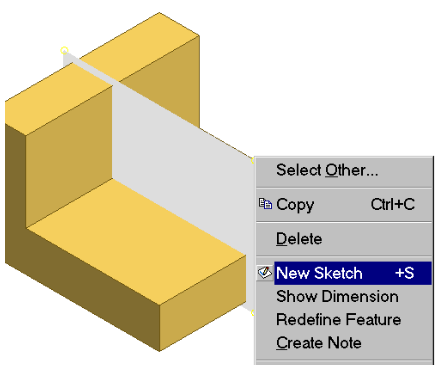
Select the face shown and start dragging the work plane into the part.

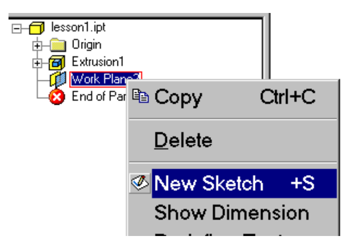


The Offset edit box should appear. Enter the value –3.25/2. The part was extruded 3.25 units.

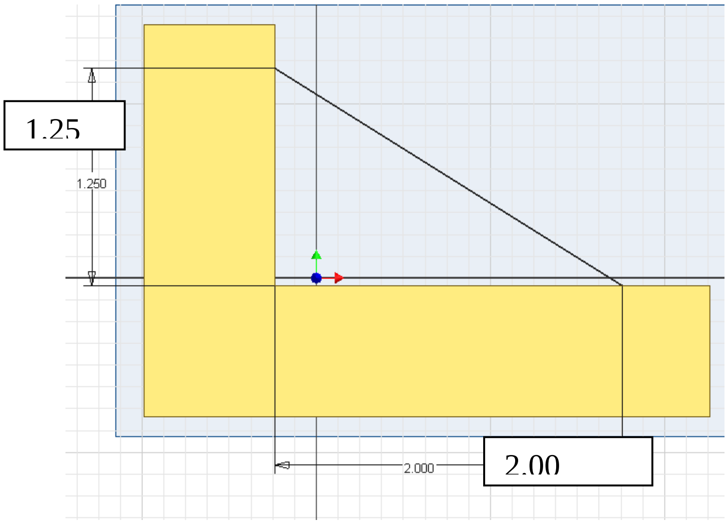


Our work plane now bisects the part. Select the work plane. Right click and select 'New Sketch.'

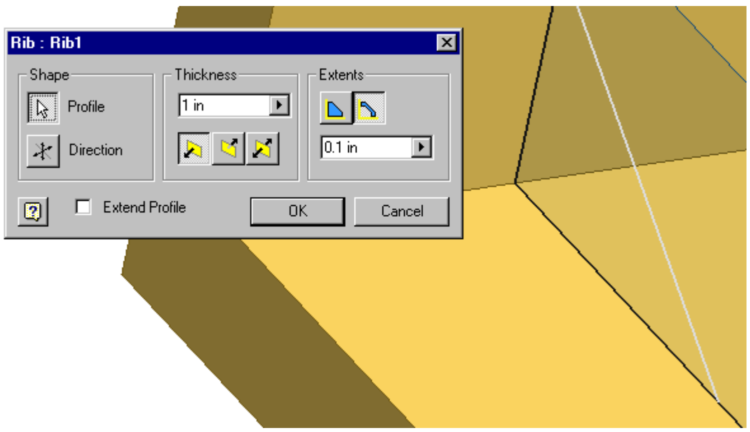




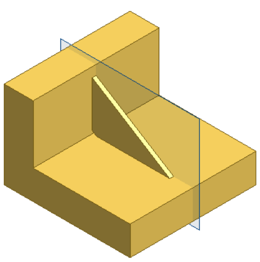
TIP: You can select the work plane for the New Sketch in the browser or in the graphics window.



Draw a single slanted line as shown. Add coincident constraints for the horizontal and vertical endpoints and then dimension.



Select the Rib tool.
Set the thickness to 1 in.
Disable the 'Extend Profile' box.
Set the Extents to Finite and set the value to 0.1 in.
Press 'OK'.

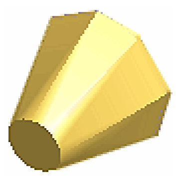


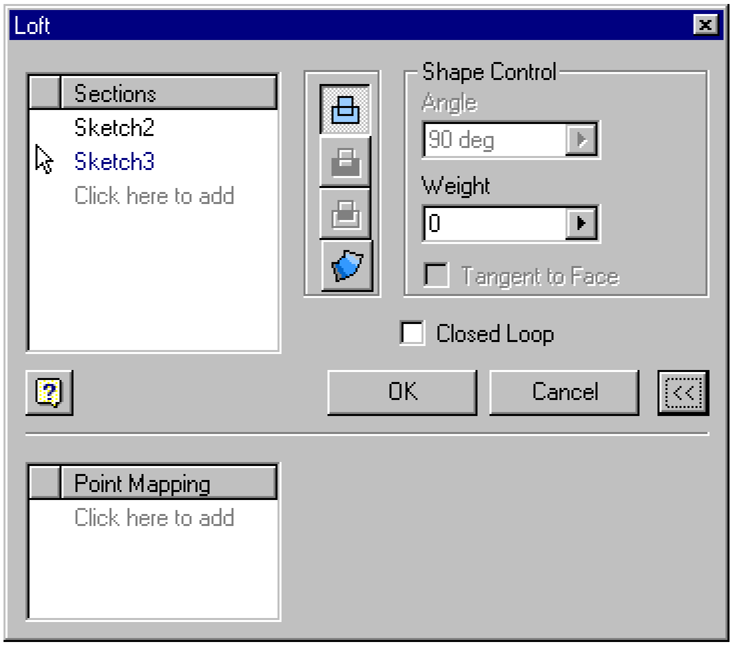
Our bracket with the rib added.
Save the part using 'Save Copy As' 'bracketwrib.ipt'. (Remember this does not save the current file!)

Loft

Loft features are created by blending the shapes of two or more profiles on work planes or planar faces. You can include the following in a loft feature:

- Sketches created on work planes offset from one another by a distance. The planes are usually parallel to one another, but any planes that are not perpendicular can be used.
- An existing planar face, as the beginning or end of a loft.
  To use an existing face as the beginning or end of a loft, create a sketch on that face (inserts a sketch icon in the browser). You do not need to draw anything in the sketch, but creating the sketch makes the edges of the face selectable for the loft.

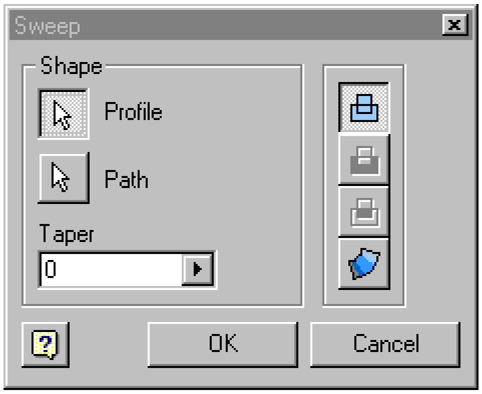




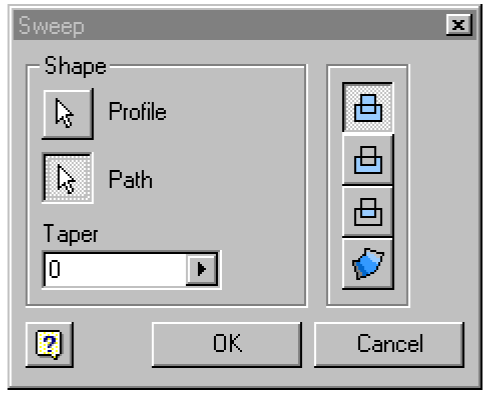
Sections	Specifies the profiles to include in the loft. Click in the row then click two or more profiles. Your selection is identified in the dialog box by sketch number, and a new row is added. To remove a section, highlight and press the 'Delete' key on the keyboard.
Shape Control	Angle Represents the angle between the sketch plane and the faces created by the loft at the sketch plane. The default value is 90 degrees. Weight A unitless value that controls how the angle affects appearance of the loft. A large number creates a gradual transition, while a small number creates an abrupt transition. Large and small values are relative to the size of your model Tangent to Face Constrains profiles created on a planar face to be tangent to the face. (Angle is not selectable if Tangent to Face is selected.)
Point Mapping	Selects a point on a sketch to use as the starting point for the loft surface. Selecting mapping points aligns profiles linearly along the points to minimize

 Sweep

Sweep creates a feature by moving a sketched profile along a planar path. A sweep feature requires two unconsumed sketches, a profile and a path, on intersecting planes.



Base Sweep Feature

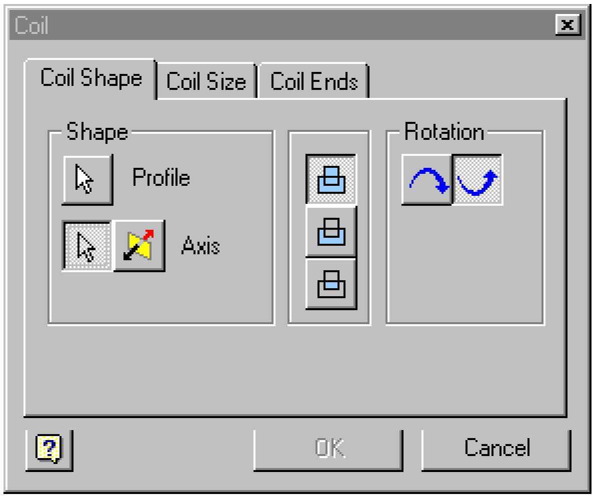


All Other Sweeps

<table>
<tr><td>Profile</td><td>Selects one or more profiles to sweep along the specified path. Profiles may be nested but may not intersect. To select multiple profiles, press Ctrl and continue to select.</td></tr>
<tr><td>Path</td><td>Sets the trajectory and endpoints of the swept feature. The profile remains normal to the path at all points.</td></tr>
<tr><td>Operation</td><td>Specifies whether the sweep joins, cuts, or intersects with another feature. Not available for base features, but required for all other sweep features. The dialog box on the left appears for base features. The dialog box on the right appears for all other sweep features.</td></tr>
<tr><td>Taper</td><td>Sets taper angle for sweeps normal to the sketch plane. If taper angle is specified, a symbol shows the fixed edge and direction of taper. Not available for closed paths.
<table>
<tr><td>Positive Angle</td><td>Positive taper angle increases the section area as the sweep moves away from the start point.</td></tr>
<tr><td>Negative Angle</td><td>Negative taper angle decreases the section area as the sweep moves away from the start point.</td></tr>
<tr><td>Nested Profiles</td><td>The sign (positive or negative) of the taper angle is applied to the outer loop of nested profiles; inner loops have the opposite sign.</td></tr>
</table></td></tr>
</table>

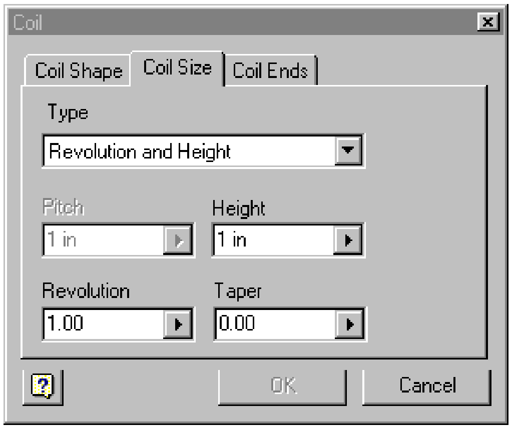
Coil

Our last Sketch feature tool is Coil. Start by creating the profile for the coil, usually a circle or rectangle. Then press the Coil tool button.

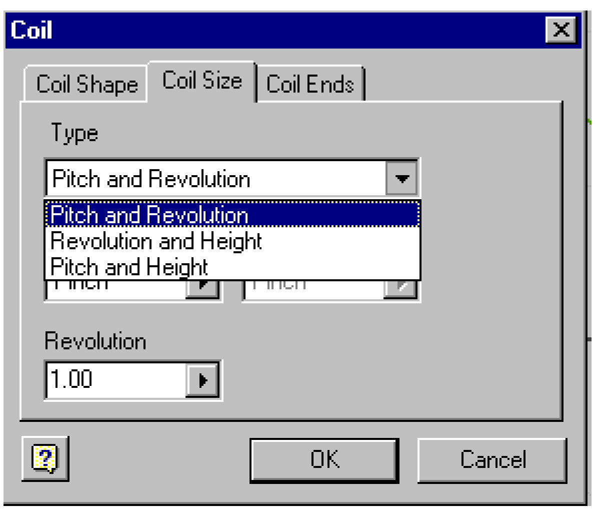


Select the profile we have created. We can adjust whether the coil will be created clockwise or counterclockwise using the Rotation button. Inventor previews our selection so we can decide if the selection is correct. The axis can either be defined using an existing sketch, edge or, if the profile is a circle, selecting the circle profile will automatically set it to the center.

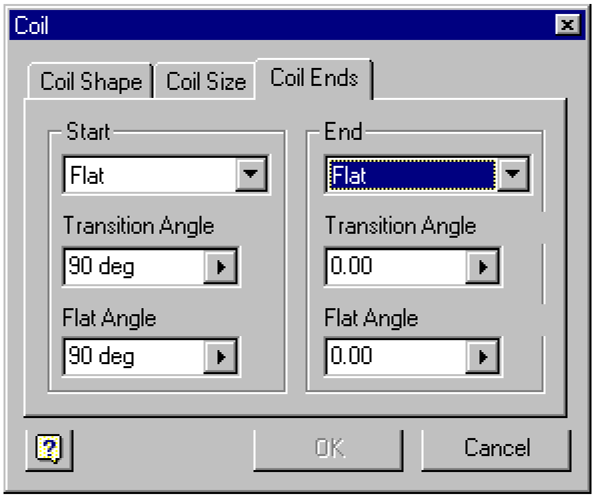
The Coil Shape Tab selects a profile and axis and specifies the direction of coil rotation. Profile and axis must be in the same sketch unless the axis is a work axis.	
Shape	Profile Selects a single profile automatically. If multiple profiles exist, you must specify one. Axis A straight line or work axis that defines the axis of revolution. It cannot intersect the profile.
Operation	Specifies whether the coil Joins, Cuts, or Intersects with the base feature. Not available if the coil is the base feature (first feature in the file). Join Adds the volume created by the coil feature to another feature. Cut Removes the volume created by the coil feature from another feature. Intersect Creates a new feature from the shared volume of the coil feature and another feature. Material not included in the shared volume is deleted.
Rotation	Specifies if the coil rotates clockwise or counterclockwise.

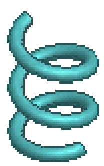
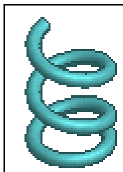


The Coil Size Tab specifies how the coil is created by specifying Pitch, Revolution, and/or Height. Specify two of the three parameters; the third parameter is calculated.	
Type	Selects which pair of parameters to specify: Pitch and Revolution, Revolution and Height, Pitch and Height, or Spiral.
Pitch	Specifies the elevation gain for each revolution of the helix.
Height	Specifies the height of the coil from the center of the profile at the start to the center of the profile at the end.
Revolution	Specifies the number of revolutions for the coil. Must be greater than zero but may include a fraction (for example, 1.5 turns). The number of revolutions includes the end conditions, if specified.
Taper	Specifies the taper angle, if desired, 1 for all coil types except Spiral.

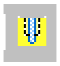


We can define a coil using Pitch and Revolution, Revolution and Height, or Pitch and Height. To modify the values, place the mouse cursor inside the edit box and type the desired value.

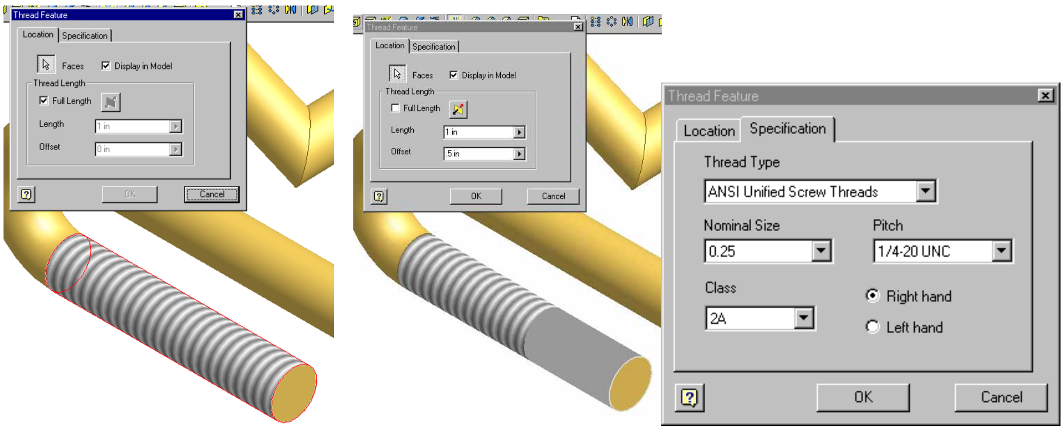


The Coil Ends tab specifies End conditions for the Start and End of the coil. Only the helix is flattened, not the profile of the coil.	
Natural or Flat	Click down arrow to specify Natural or Flat for both ends of the coil. Ends can have dissimilar end conditions.
	Transition angle The distance (in degrees) over which the coil achieves the transition (normally less than one revolution). The example shows the top with a natural end and the bottom end with a one-quarter turn transition (90 degrees) and no flat angle.
	Flat angle The distance (in degrees) the coil extends after transition with no pitch (flat). Provides transition from the end of the revolved coil to a flattened end. The example shows the same coil as above, but with a half-turn (180 degree) flat angle specified.

Thread Tool



This tool maps a bitmap of a thread onto a cylindrical feature.



Location	
Faces	Select the faces to apply the bitmap
Display in Model	When this is enabled, the bitmap is visible.
Full Length	Adds threads to the entire length when enabled. When Full Length is disabled, the user can specify a length and an offset from an end.
Specification	
Thread Type	User can select ANSI or Metric. The default depends on which standard was selected during installation.
Nominal Size	User can select the diameter of the thread from a drop down list.
Pitch	User can select pitch from a drop down list.
Class	User can select from a drop down list.
Right/Left Hand	Determines the direction of the thread.

Placed Feature Tools

Our Placed Feature Tools are FILLET, CHAMFER, FACE DRAFT, and SPLIT.

 **FILLET**

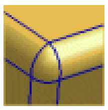
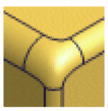
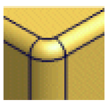
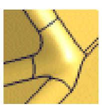
You can include a fillet in your design by adding a 2D fillet when sketching. A 2D sketched fillet and a fillet feature can produce models that are identical in appearance. Although the results look the same, the model with the fillet features has several advantages:

- A fillet feature can be edited, suppressed, or deleted independent of the extrusion feature, without returning to the original sketch.
- If the remaining edges are to be filleted, you have more control over the corners.
- You have more flexibility when performing subsequent operations such as applying face draft.

Because other features may affect fillets and rounds, add the fillets and rounds toward the end of the modeling process. For example:

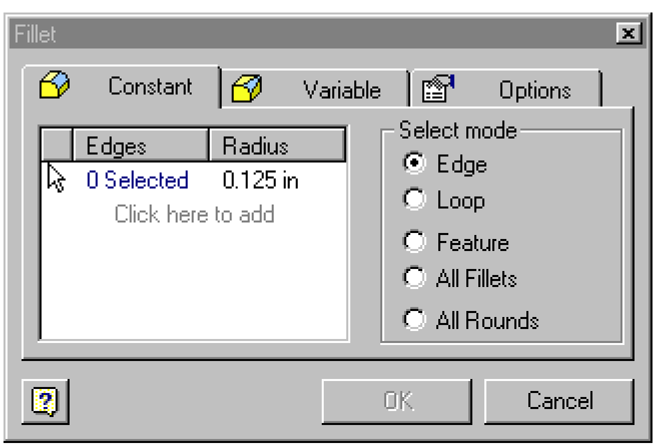
- It is easier to add draft to a face that intersects with other faces at a sharp angle, so it is better to add draft to a face before adding fillets.
- Fillets can increase the time required to update when feature dimensions change. To work most efficiently, add the features of the basic design early and wait until the end of the process to add features such as fillets.
- Reordering features can cause existing fillet features to produce undesired results. Adding features such as fillets and chamfers at the end of the modeling process can minimize problems.

When filleting adjacent edges, you can add the fillets separately or fillet all edges in one operation. Consider the following when deciding whether to use single or multiple operations:

	When adding fillets with the same radius to three adjacent edges, the result is the same whether you add them separately or in one operation. The most efficient method is to add them in one operation.
	If each edge has a different radius, use a single fillet operation, if possible, to ensure a smooth corner. This situation always results in a blended corner.
	When two edges have the same radius and the third edge has a different radius, use a single fillet operation if possible. If you add the fillets as separate operations, the edge with the larger radius must be filleted first.
	When filleting four or more edges, fillet all edges as a single operation

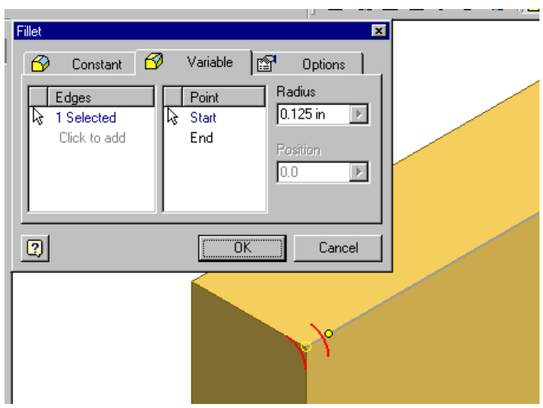
If a fillet is applied to a single edge, any operation that deletes the edge results in an error condition, since the fillet is no longer valid. If a fillet feature is applied to multiple edges, you can delete an edge and the fillet will update to reflect the change, as long as any edge in the set remains.

You can create constant-radius and variable-radius fillets and fillets of different sizes in a single operation. All fillets and rounds created in a single operation become a single feature.



Selecting the Fillet tool brings up the dialog box shown. On the right side, we can set the selection mode of what we want to select. For example, enabling the 'All Fillets' mode, would select all the current fillets defined on the model.

The Constant Tab sets the parameters for adding constant-radius fillets. If necessary, set the options on the Options tab before creating the fillets.	
Edges	Defines a set of edges to fillet. To add edges, select the set from the Edges box, then click the edges in the graphics window. (To remove edges, press Ctrl as you click.) To add another edge set, click the prompt in the last row of the Edges box. Use Select Mode to simplify the selection of edges.
Radius	Specifies the fillet radius for the selected set of edges. To change the radius, click the radius value, then enter the new radius.
Select Mode	Changes the selection method for adding or removing edges from an edge set. Click to select the mode from the list. Edge selects or removes single edges. Loop selects or removes the edges of a closed loop on a face. Feature selects or removes all edges of a feature that do not result from intersections between the feature and other faces. Fillets selects or removes all remaining concave edges and corners. This mode requires a separate edge set. Rounds selects or removes all remaining convex edges and corners. This mode requires a separate edge set.



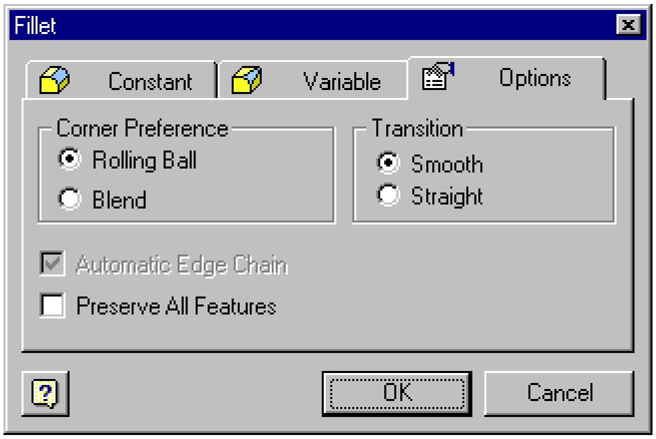
The Variable Tab sets the parameters for adding variable-radius fillets. If necessary, set the options on the Options tab before creating the fillets.	
Edges	Specifies edges to fillet. To add an edge, select the prompt in the Edges box, then click the edge in the graphics window.
Point	Selects the start point, endpoint, or an intermediate point so that you can enter its radius.
Radius	Sets the fillet radius at the selected point. To change the radius, select the point in the point list, then enter the new radius.
Position	Specifies the position of the selected point. To change the position, select the point in the point list, and then enter a value between 0 and 1 as a percentage of the length of the edge.

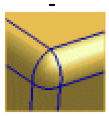
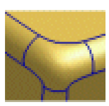
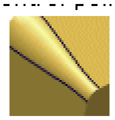
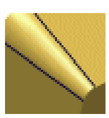
To create a variable fillet, select the Variable tab. Select an edge. The user is then prompted for a series of points along the edge. As the user picks each point, he can assign a value for the radius at that point.

TIP: To select a single edge without selecting edges that are tangent to it, turn off Automatic Edge Chain on the Options tab.

TIP: To select a single edge segment without selecting edges that are tangent to it, turn off Automatic Edge Chain on the Options tab.

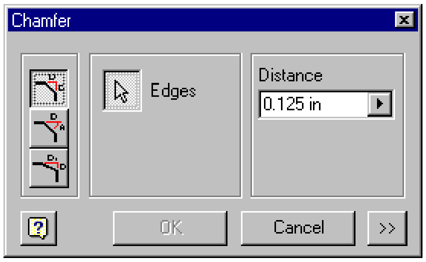
If the selected edge contains multiple tangent edge segments, the position is relative to the start point of individual edge segment on which the point resides.



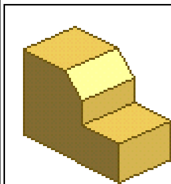
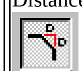
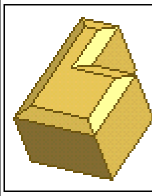
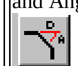
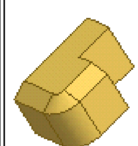
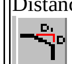
The Options tab sets the corner preference, transition type, and edge chain preference for fillets and rounds. The default settings for these options are correct for most fillet features.	
Corner Preference	Sets the corner style for the fillets. Rolling ball is the default setting.  Rolling Ball creates a fillet defined as if a ball had been rolled along the edge and around the corners.  Blend creates a continuous tangent transition between fillets in sharp corners.
Transition	Defines how variable-radius fillets are created between control points. Smooth is the default setting.  Smooth creates fillets with a gradual blending transition between the points. The transition is tangent (no rate of change between points).  Straight creates fillets with linear transitions between the points.
Automatic Edge Chain	When the check box is selected, selecting an edge to fillet selects all tangent edges automatically. When the check box is cleared, only the indicated edge is selected. The default setting is on.
Preserve All Features	When the check box is selected, all features that intersect with the fillet are checked and their intersections are calculated during the fillet operation. If the check box is cleared, only the edges that are part of the fillet operation are calculated during the operation.

TIP: To select part of an edge that already has filleted corners, clear the check box.

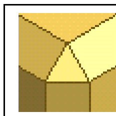
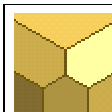
 Chamfer



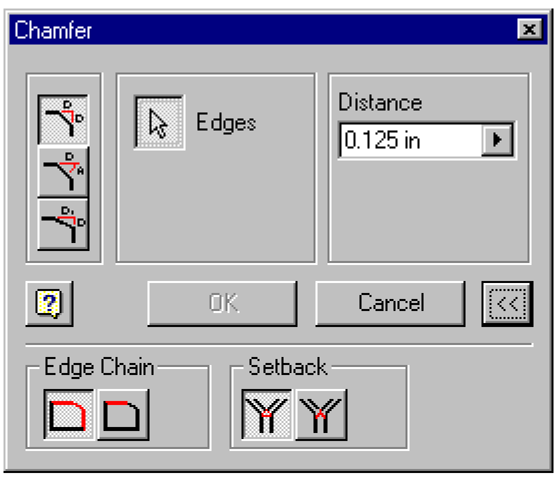
You specify the corner appearance and select edges individually or as part of a chain. All chamfers created in a single operation are one feature.

Method Specifies how the chamfer is constructed		Distance 	Creates a chamfer with the same offset distance from the edge on both faces. Selects a single edge, multiple edges, or chained edges. Corner setback appearance may be specified.
		Distance and Angle 	Creates a chamfer defined by an offset from an edge and an angle from one face to the offset. Any/all edges of a selected face may be chamfered at once.
		Two Distances 	Creates a chamfer on a single edge with a specified distance for each face. Edges may be chained together.

Edges and Faces Selects one or more edges or faces. Pressing Ctrl while clicking removes geometry from the selection.	Edges	Selects individual edges to chamfer and previews default distance.
	Face	For chamfers defined by a distance and angle, selects the affected face.

<table>
<tr><td>Flip</td><td>For chamfers defined by two distances, flips the direction of the chamfer distances.</td></tr>
<tr><td>Distance and Angle</td><td><table>
<tr><td colspan="2">Specifies the extent of selected chamfer method.</td></tr>
<tr><td>Distance</td><td>Specifies offset distance of chamfer from selected edge(s). For chamfer defined by two distances, specifies both offsets.</td></tr>
<tr><td>Angle</td><td>For chamfers defined by a distance and angle, specifies the angle of the chamfer.</td></tr>
</table></td></tr>
<tr><td>If edges intersect at a tangent point, edge chain and setback appearance may be specified.</td><td><table>
<tr><td>Edge Chain</td><td>Selects all edges that share a tangent point.</td></tr>
<tr><td>Setback</td><td>For the Distance chamfer method, defines corner appearance when three chamfered edges meet at a corner.<table>
<tr><td></td><td>Chamfer may be joined at a flat intersection (left button).</td></tr>
<tr><td></td><td>Chamfer may form a corner point at the intersection, as if milled on three edges (right button).</td></tr>
</table></td></tr>
</table></td></tr>
</table>

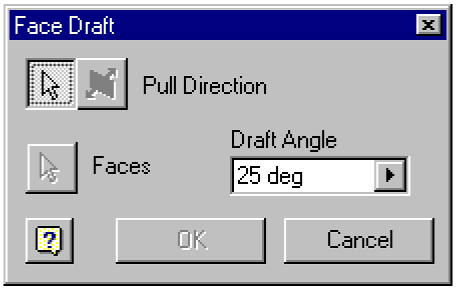
The three modes of chamfers are located on the left side of the dialog box. The top tool is for Equal Distance. The middle tool is for Distance-Angle. The bottom tool is for two different Distance values. Note that on the lower right corner there is a double arrow (More) button.



Pressing the double arrow brings up additional chamfer options. The Edge Chain option controls whether the chamfer affects all tangential edges or only the selected edge. The Setback option controls whether or not a setback is added.

Face Draft

A face draft is a slight angle applied to the walls of a part.



Draft is a taper applied to part faces so that a part can be retrieved from a mold or to cant one or more faces. When designing features for molded or cast parts, you can apply draft by specifying a taper angle for an extrusion or sweep. To add draft to an existing feature or to individual faces, use the Face Draft tool. When applying draft to a face, the relationship between the pull direction and the fixed edge determines the result of the operation.

- If you select a set of tangent continuous faces (like a staircase with fillets at the edges), draft is applied to all faces. In the browser, this draft is labeled TaperShadow.
- If you select a face that is not tangent to another face, draft is applied to that face only. In the browser, this draft is labeled TaperEdge.

<table>
<tr><td>Pull Direction</td><td>Indicates direction in which a mold is pulled from a part. As you move the cursor in the graphics window, a vector displays normal to a highlighted face or along a highlighted edge. When the vector displays as desired, click to select it and display the direction arrow.</td><td><table><tr><td>Flip Direction</td><td>Reverses the pull-direction arrow.</td></tr><tr><td>Accept</td><td>Accepts the displayed pull direction.</td></tr></table></td></tr>
<tr><td>Faces</td><td>Specifies the faces to which draft will be applied. As you move the cursor over a face, a symbol indicates the fixed edge for the draft and how the draft will be applied. Click to select the desired fixed edge.</td><td></td></tr>
<tr><td>Draft Angle</td><td>Sets the angle of the draft. Enter the angle or choose a calculation method from the drop-down list.</td><td></td></tr>
</table>

TIP: The first edge selected determines the curves you can select for draft. For example, if the first edge you select is linear, you cannot select edges with continuous tangency.

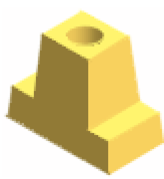
Applying a Face Draft

Click the Face Draft tool.

1. To define the pull direction, move the cursor over a feature until the direction vector is aligned with the desired pull direction, then click to select.
2. If necessary, click the Flip Direction button to change the pull direction.
3. Select the faces to draft. As you move the cursor over a face, a symbol indicates the fixed edge and direction for the draft.
4. Enter the draft angle.
5. Click OK.

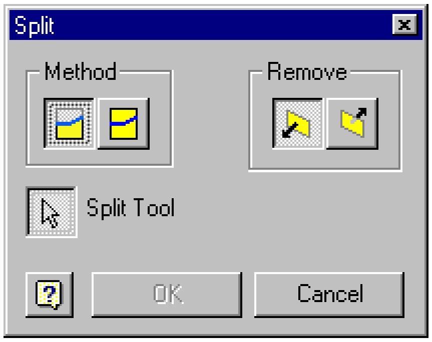
TIP: To select part of an edge that already has filleted corners, clear the check box.

If a face is tangent to other faces, all tangent faces are highlighted.



Split

Part Splitting is a fast and easy way to create a top and bottom for a box or enclosure. It ensures that the parts mate properly. To create a split, the easiest method is to place an offset work plane at the location for the split and then use the work plane for the split operation. We can split part faces or an entire part and remove one of the resulting sides. This tool allows faces on both sides of the split to have draft applied.



Split Part	Selects part to split and discards one side.
Split Face	Selects one or more faces to split into two pieces.
Split tool	Selects workplane or parting line used to split face or part into two pieces.
All	Selects all faces to split. Click OK.
Selected	Selects faces to split. Click Faces to Split tool, select faces to split, then click OK.
When Split Part method is active, click Side to Remove button to preview, then click OK to remove specified side.	

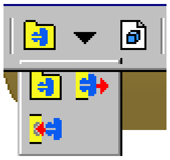
Selecting the Split tool brings up the dialog box shown. The user has two options under Method: Splitting the Part (left icon) and Splitting a Face (right icon).

The Split tool can be a workplane or sketch a parting line on a workplane or part face. The sketched parting line can include lines, arcs, and splines.

By default, Split removes one side of a split part. If you need to split a part into two parts you can use this procedure:

1. Sketch a parting line on a part face.
2. On the File menu, use Save Copy As to save the part with the parting line and both halves intact.
3. Use the Split tool to split the part and remove the selected half.
4. Use Save Copy As to save the first half of the part.
5. Open the original file, then use the Split tool to remove the other half of the part.
6. Use Save Copy As to save the second half of the part.

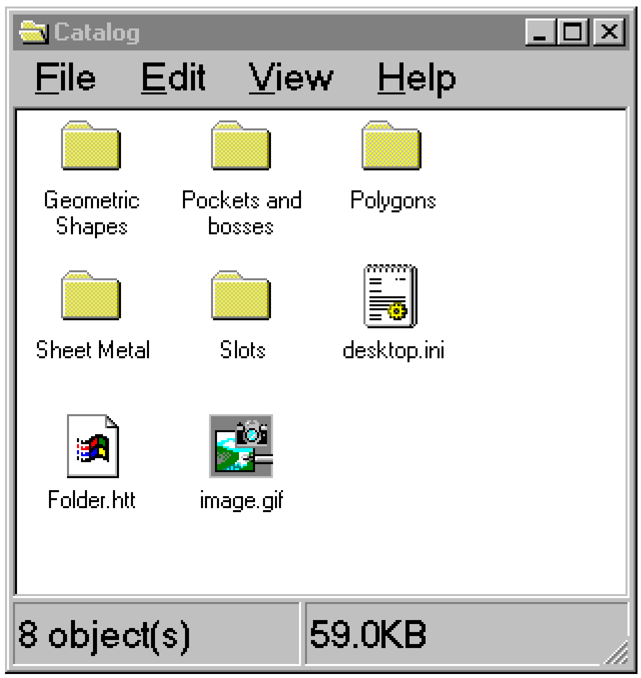
Both halves of the part are now saved in separate files.



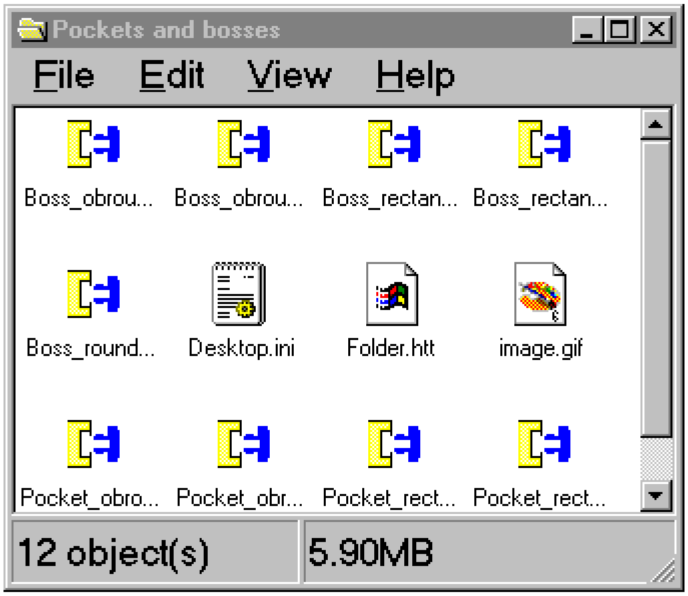
The next section of the Features tool bar contains the Catalog flyout tools and the Derived Component tool.

 View Catalog

The Catalog is similar to AutoCAD Design Center. The user can browse a library of parts and blocks that can be inserted into the current file.

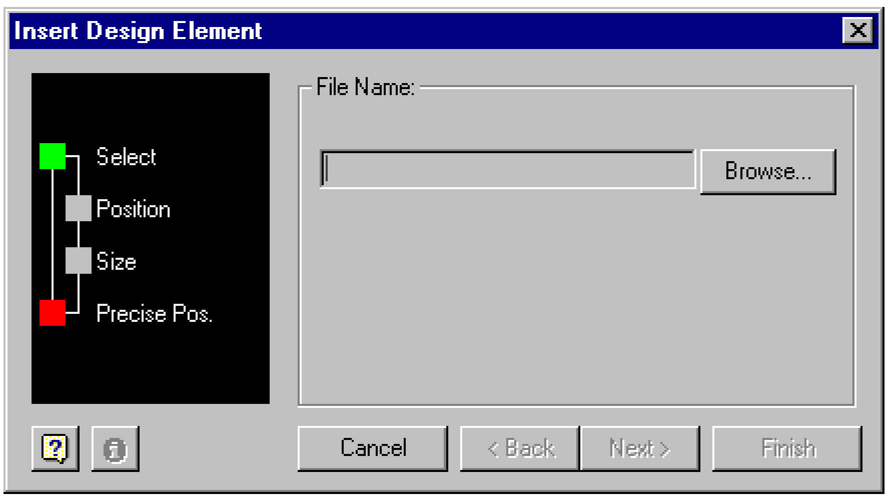


Inventor includes a set of features ready for use. Users can explore the pre-existing library of sketches to save time when constructing their models.

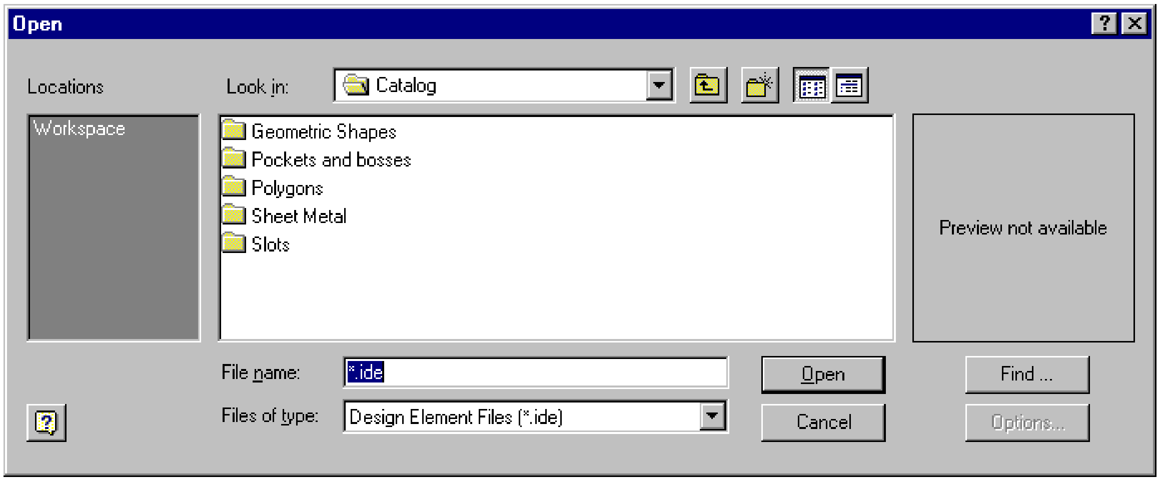


The library features can be previewed using the Insert tool.

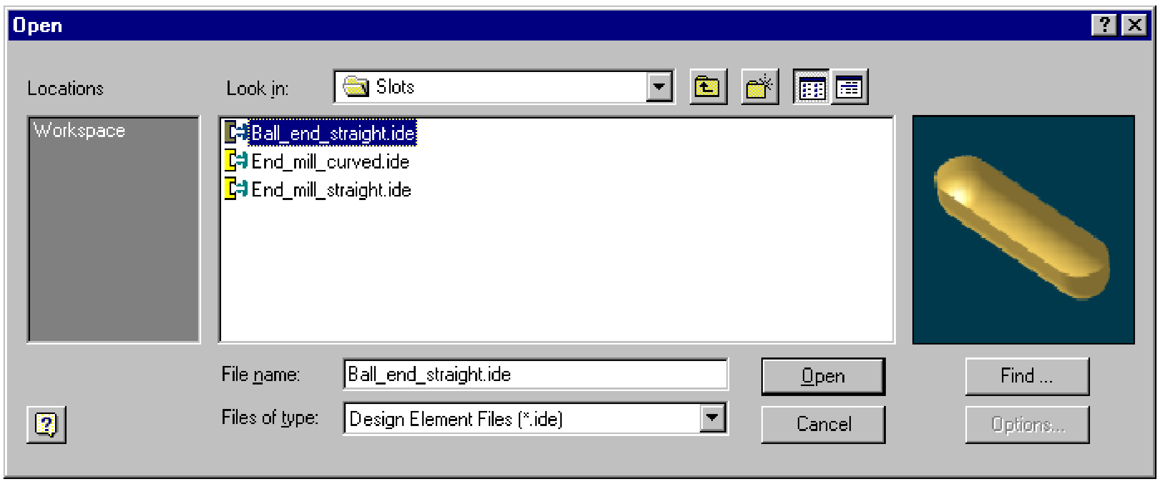
 **Insert Design Element**



Selecting this tool brings up the dialog box shown. Select the ‘Browse’ button to locate the element to be added to the model.

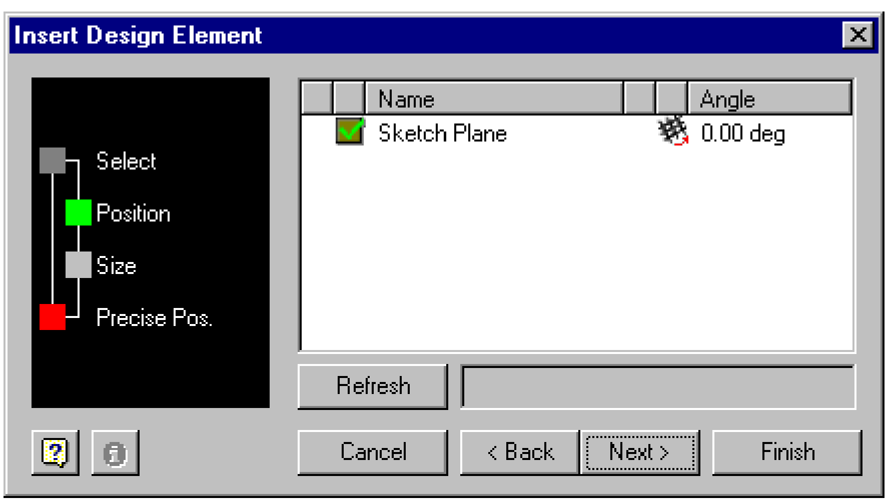


The user can then select the element category.

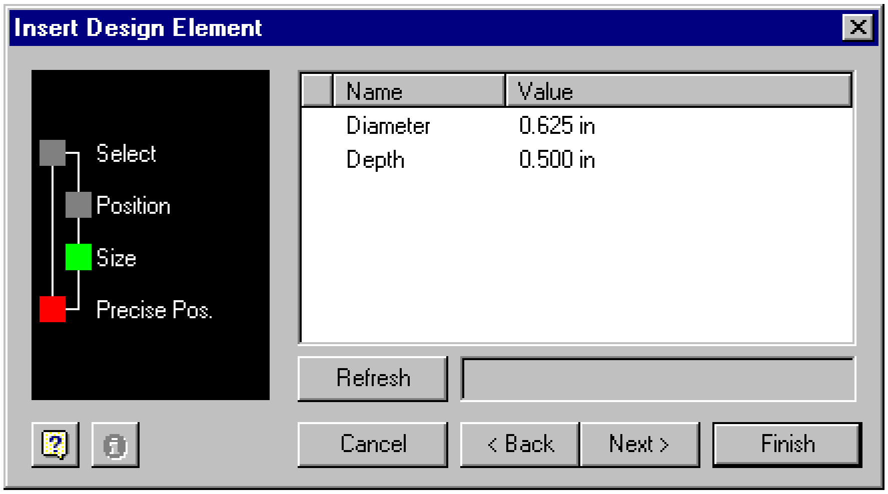


Once we get into the category area, we can preview each element to determine which one we want.

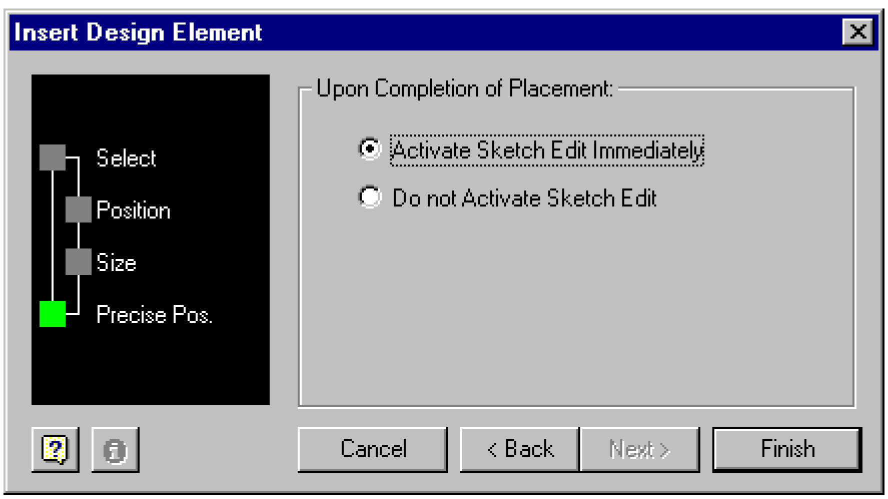
Select the desired element and press 'Open'.



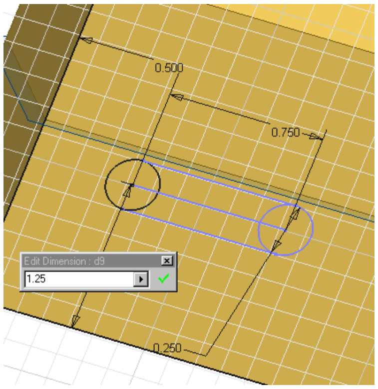
Once the element is selected, the next dialog appears. The user is required to select the plane on the existing model to place the element.



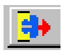
The user is then prompted for the values for the element to be placed. At this time, the user can specify his specifications for the design element.



The user is then allowed to set whether to enter Sketch Edit mode immediately upon placement. Entering Sketch Mode allows the user to reposition the element or make further modifications to the basic element.

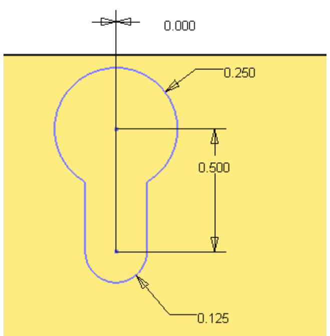


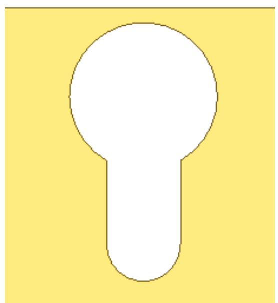
Modifying the inserted element is exactly the same as if we had created the sketch from scratch.

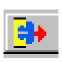


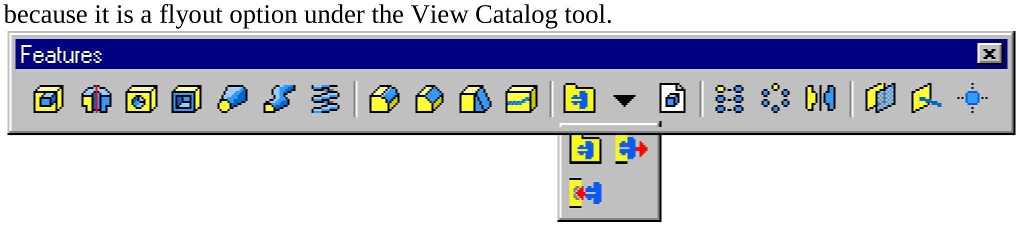
Create Design Element

We start by creating a basic shape. In this case, I've created a basic keyhole slot.

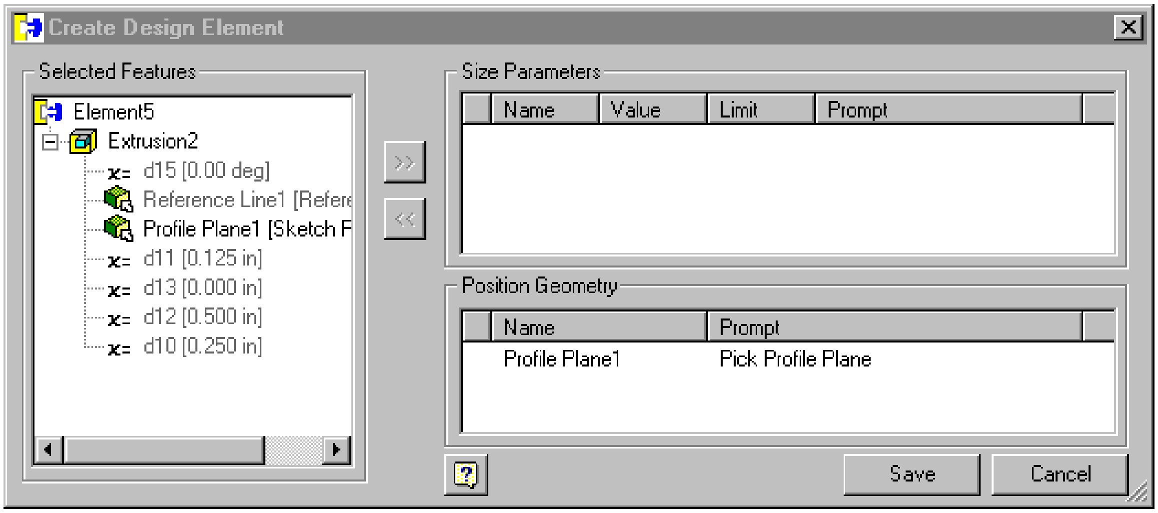




 We use the Create Design Element tool located on the Features toolbar. If you don't see the tool, it is because it is a flyout option under the View Catalog tool.

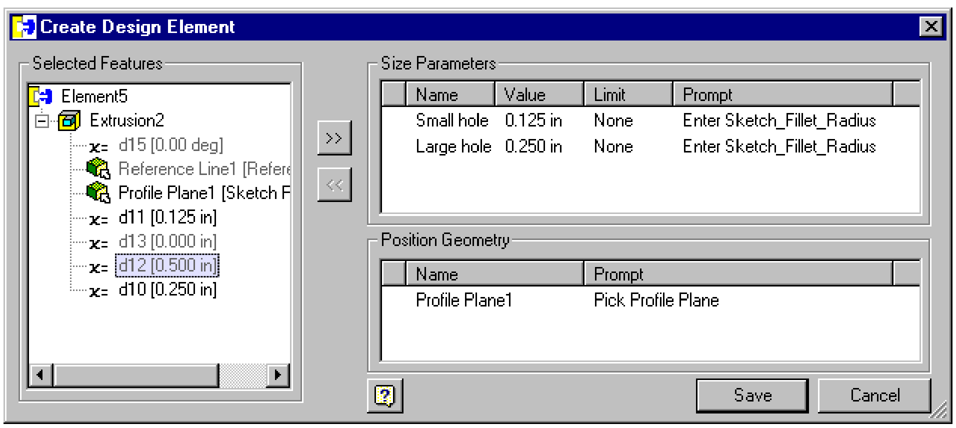


TIP: The sketch must be turned into a feature using Extrude, Revolve, or Sweep BEFORE it can be turned into a design element.

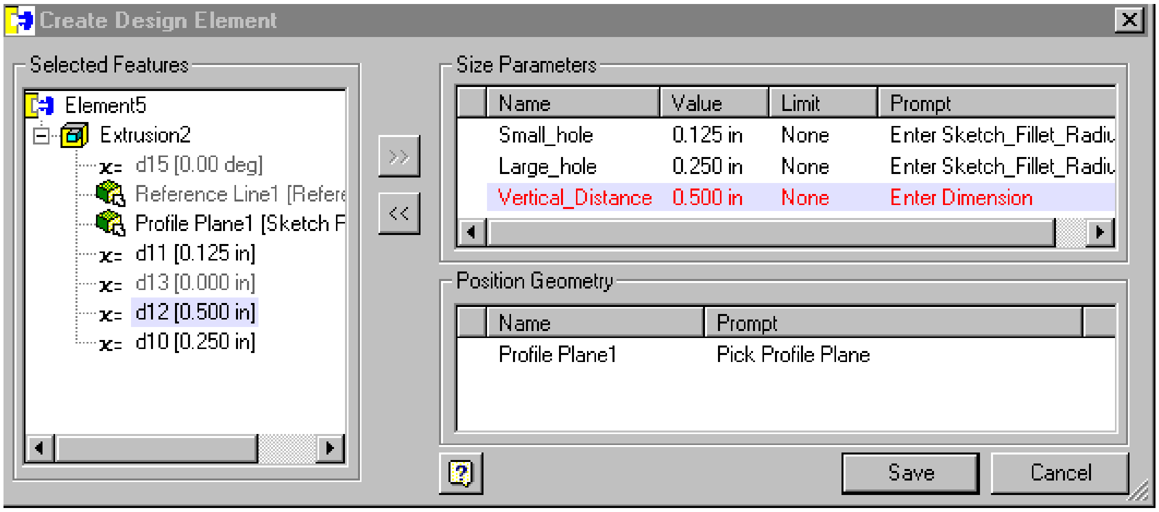


Initiate the Create Design Element command. To select the feature, simply pick it from the Browser. The window with the selected features will fill in as shown.

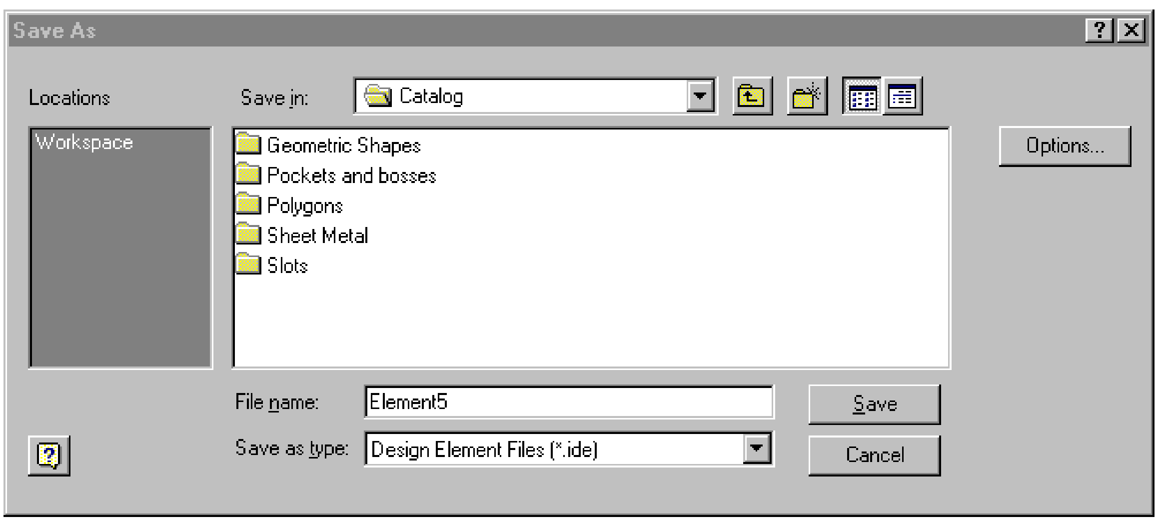
To define the variable parameters for the design, highlight it and then select the top double arrow. We can rename the parameter to allow useful prompting.



Notice that when a parameter is highlighted, the top double arrow becomes available to insert into the Size Parameters.

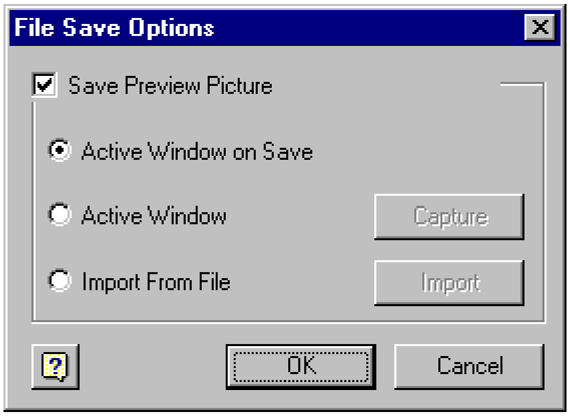


In this example, we have defined the design parameters to be the large hole, the small hole, and the vertical distance between the holes. Notice that the 0.00-inch dimension which forces the holes to line up is omitted at a user-defined parameter. In this way, we can control our design elements and decide which geometries are changeable and which are not.

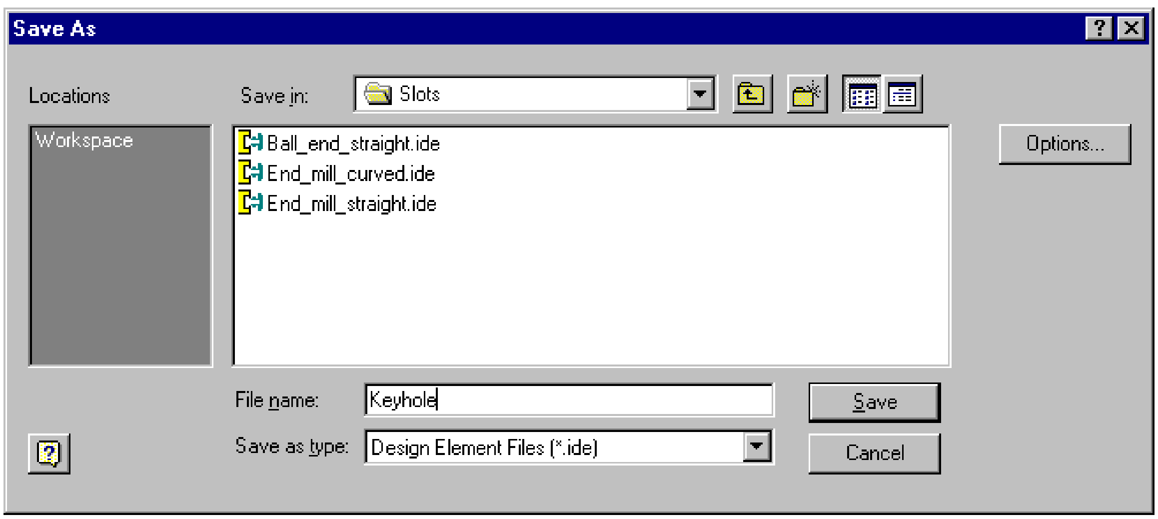


When we press ‘SAVE’, we get the Catalog. Since this is a keyhole slot, we will place this design element in the Slots Category.

TIP: When naming parameters, you can not use spaces, dashes or other punctuation marks. If you use an invalid character, you will get an error message to provide a proper name.



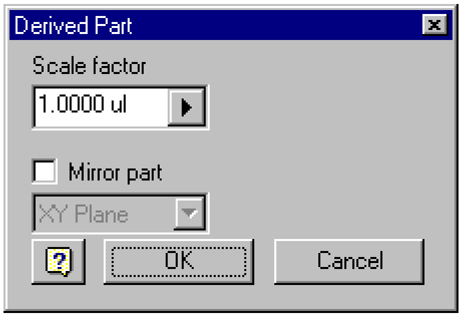
Pressing the Options button on the far right allows the user to set whether or not to create a Preview Picture of the design element and how it should be captured.



We select the Slots category and assign the filename 'Keyhole' to our design element.

Derived Part

Creates a new derived part using an Autodesk Inventor part as the base part. The derived part can be scaled larger or smaller than the original part or mirrored using any of the origin workplanes of the base part. The location and orientation of the derived body is the same as the base part.	
Access:	Insert>Derived part
Scale factor	Default value is 1.0. Scale factor may be expressed in whole numbers of percentages (expressed with a decimal). Click arrow to select from recently used values.
Mirror body	Mirrors part when check box is selected. Specifies X, Y, or Z origin workplanes as the mirror plane. Click arrow to select plane.



To capture design intent, you can use an Autodesk Inventor part as the basis for a derived part. A derived part is a new part that uses an existing part as its base feature. You can add features to the derived part feature (the base feature), and update it to incorporate changes made to the original part.

You can use the derived part to explore design alternatives and manufacturing processes. You can add features to the derived part feature and create multiple derived parts from one original design. For example:

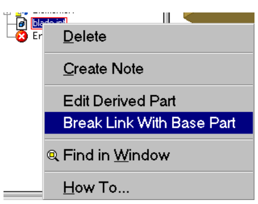
- A casting blank can be machined several different ways.
- A standard length of tubing can be machined in different configurations.

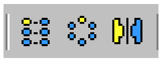
Changes to the original part do not affect the derived part unless you specifically request an update. For example, you can insert a part into a file as the base feature of the derived part. You can add features to the derived part feature (base feature) to modify it. When you click the Update button on the Command bar, the derived part feature does not update, but all other features update.

To update the derived part feature, right-click on the part in the browser and select Update. Changes made to the original part are reflected in the updated derived part feature.

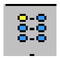
A derived part is linked to its original part so changes to the original part can be incorporated. You can modify the derived part feature and save the file, but changes to the original part will not affect it unless you specifically update the derived part feature.

You can break the link to the original part if you no longer wish to update the derived part feature. Right-click on the derived part feature in the browser and select Break Link. The derived part feature becomes a regular feature and its changes are saved only in the current file.



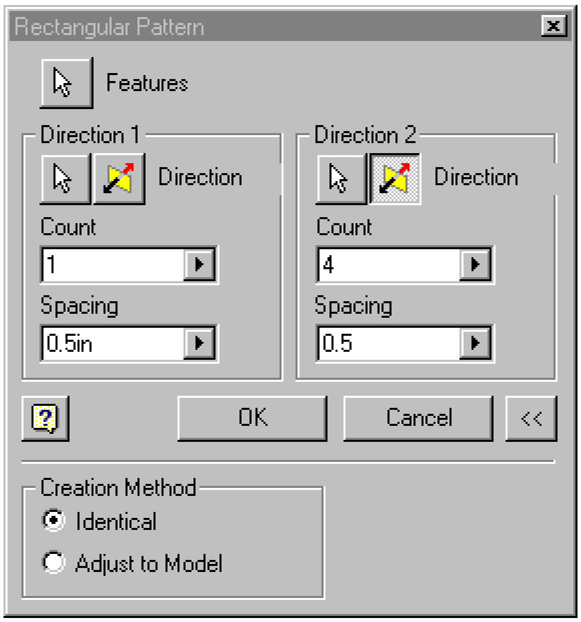


Our next section allows us to pattern or mirror features.



Rectangular Pattern

Use the Rectangular Pattern tool on the Feature toolbar to duplicate a feature and arrange the resulting features in a rectangular pattern. You can arrange the features in rows and columns by a specific count and spacing, suppressing individual features if desired.



1. Click the Rectangular Pattern tool.
2. Click in the graphics window to select the feature to duplicate and arrange in a pattern.
3. Click Column Placement, then click an edge or work axis to indicate the direction of the column. Click Flip to change the column direction, if desired.
4. Enter the count (number of features) for the column and the spacing between features.
5. If the pattern has multiple rows, Click Row Placement and set the row direction, count, and spacing.
6. If desired, click the More button to set a Creation Method:
    - Choose Identical to create identical features, regardless of termination.

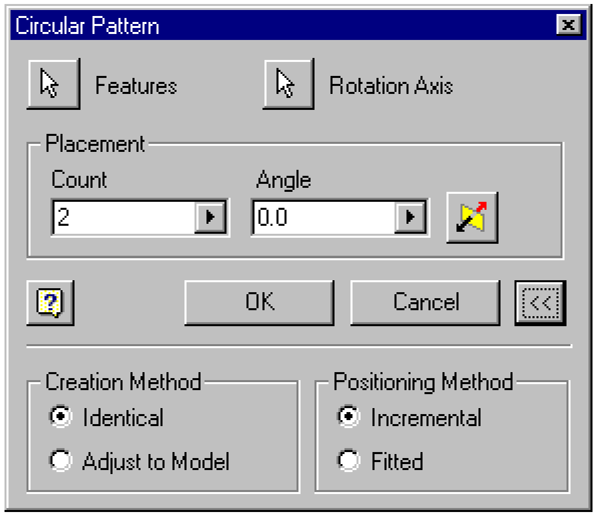
Choose Adjust to Model to terminate features when it encounters a face.

TIP: Patterns created with the Identical method calculate faster than the Adjust to Model method. Using Adjust to Model, the pattern terminates if it encounters a planar face, and may result in a feature whose size and shape differs from the original.

You can suppress individual occurrences in a pattern (except the original feature) to allow the pattern to flow around another feature, an irregular shape, or create a missing-tooth pattern. In the browser, right-click the occurrence and click Suppress.

Circular Pattern

Use the Circular Pattern tool on the Feature toolbar to duplicate a feature and arrange the resulting features in an arc or a circle



To begin, create the one or more features to include in the pattern.

1. Click the Circular Pattern tool.
2. Select the feature to arrange in a pattern.
3. Click the axis (pivot point of angle) about which occurrences are repeated. The axis can be on a different plane from the feature being patterned.
4. In the Count box, enter the number of occurrences in the pattern.
5. In the Angle box, enter the angle, as follows:
   - For Incremental positioning, the angle specifies the spacing between occurrences.
   - For Fitted positioning, the angle specifies the total area the pattern feature occupies.
6. Click the Flip button to reverse the direction of the pattern, if desired.
7. If desired, click the More button to specify the following options:
   - Under Creation Method, click Identical to create identical features or click Adjust to Model to terminate features when it encounters a face.
   - Under Positioning Method, click Incremental to space occurrences at the angle specified or click Fitted to arrange occurrences within the specified angle.

Click OK.

TIP: Patterns created with the Identical method calculate faster than the Adjust to Model method. Using Adjust to Model, the pattern terminates if it encounters a planar face, and may result in a feature whose size and shape differs from the original.

You can suppress individual occurrences in a pattern (except the original feature) to allow the pattern to flow around another feature, an irregular shape, or create a missing-tooth pattern. In the browser, right-click the occurrence and click Suppress.

Mirror Feature

Use the Mirror tool on the Features toolbar to mirror one or more features at equal distances across a plane.

To begin, create one or more features to mirror. Create a work plane to serve as the mirror plane or, if you prefer, identify a planar face to use.

1. Click the Mirror tool.
2. Click one or more features to mirror.
3. Click Mirror Plane and select a work plane or planar face.
4. If desired, click the More button to choose one of the following Creation Methods:
    - Click Identical to create identical mirrored features, regardless of termination.

Click Adjust to Model to terminate features on model planes.

TIP: Features mirrored with the Identical method calculate faster than the Adjust to Model method. Using Adjust to Model, the mirrored feature terminates if it encounters a planar face, and may result in a feature whose size and shape differs from the original.

Create Work Plane

Use the Work Plane tool on the Feature toolbar to define a work plane using unconsumed sketch geometry, feature vertices, edges, faces, or other work features. Work planes can also be created in-line when a work feature command requires you to select a plane.

Use one or more of the following relationships to define a work plane:

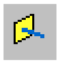
- On geometry (on three points, for example)
- Normal to geometry
- Parallel to geometry
- At an angle to geometry (on a plane and an axis)

1. Click the Work Plane tool.
2. Select appropriate vertices, edges, or faces to define a work plane.

   For offset work planes, drag the work plane to desired location and enter a distance or angle in the Offset dialog box. Click the check mark in the dialog box to accept the preview and create the offset work plane.

If desired, you can create multiple work planes offset from one another at a specific distance or angle. Follow the steps above, selecting the last created work plane as the sketch plane, then drag the new work plane to the desired offset distance.

TIP: If more than one solution is possible, a selection box appears. Click the forward or reverse arrows in the selection box, then click the check mark when the correct solution is previewed.

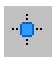


Work Axis

Use the Work Axis tool on the Feature toolbar to designate unconsumed sketch geometry, points, or a part edge as a work axis. Work axes can also be created in-line as input to other work feature commands.

1. Click the Work Axis tool.
2. Select one of four methods:
    - Select a revolved feature to create a work axis along its axis of revolution.
    - Select two valid points to create a work axis through them.
    - Select a work point and a plane (or face) to create a work axis normal to the plane (or face) and through the point.

Select any two non-parallel planes to create a work axis at their intersection.

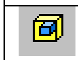
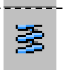
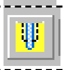
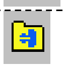
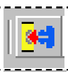
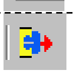
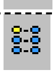
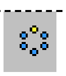
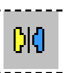
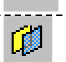
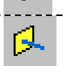
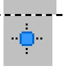


Work Point

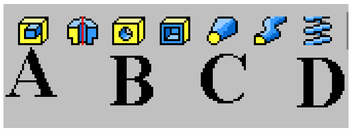
Use the Work Point tool on the Feature toolbar to select model vertices, edge and axis intersections, intersections of three non-parallel faces or planes, and other work features as work points.

Work points can also be created as input to other work feature commands that require you to select a point. Work points are partially constrained in place relative to vertices, edges, and other topological characteristics of the parent part or feature.

Feature Tools

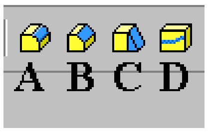
Button	Tool	Function	Special Instructions
	Extrude	Extrude a profile normal to the sketch	Can be base feature
	Revolve	Revolve a profile around an axis	Can be base feature
	Hole	Create a hole in a part	Use hole points or line endpoints as hole centers
	Shell	Create a hollow part	Placed feature
	Rib	Creates a rib	Uses an open contour
	Loft	Construct a feature with varying cross sections; can follow a curved path	Requires multiple work planes
	Sweep	Extrude a profile along a curved path	Can be base feature
	Coil	Extrude a profile along a helical path	Can be base feature
	Thread	Maps a bitmap of a thread to a cylindrical face.	
	Fillet	Create a fillet or round edges	Placed feature
	Chamfer	Create a chamfer on selected edges	Placed feature
	Face Draft	Create a draft on selected faces	Placed feature
	Split	Part Split using parting line or spline	
	View Catalog	Open a Catalog of Design Elements	
	Insert Design Element	Add a Design Element	
	Create a Design Element	Create a Design Element from an Existing Feature	
	Derived Part	Create a new derived part from a base part	
	Rectangular Pattern	Creates a rectangular pattern of features	Pattern can be suppressed, items in the pattern can be individualized
	Circular Pattern	Creates a circular pattern around a center	Pattern can be suppressed, items in the pattern can be individualized
	Mirror Feature	Create a mirror image using a plane, line or axis as mirror line	
	Work Plane	Create a work plane	
	Work Axis	Create a work axis	
	Work Point	Create a work point	Can be used to place holes on curved surfaces

Review Questions



Identify the icon.

1. Hole
2. Extrude
3. Coil
4. Loft



Identify the icon.

5. Part Splitting
6. Face Draft
7. Fillet
8. Chamfer

9. A Derived Part is:

   A. Used to explore different design ideas
   B. A copy of a base part
   C. Can be scaled or mirrored
   D. All of the above

10. The three options for creating a chamfer are:

   A. Equal distance, distance-angle, two distances
   B. Variable, constant, radial
   C. Equal Distance, Variable Distance, Distance-Angle
   D. Two angles, Two Distances, Equal Distance

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