

Cross-Part Relationships

By Neil Munro

In previous versions of Autodesk Inventor™ software, geometry projected from other parts in an assembly does not automatically update if the underlying geometry is changed. To create a relationship between parts, you can make underconstrained geometry adaptive, and then change position and size to match assembly constraints placed between assembly components. To capture complex edge relationships between parts can require a large number of assembly constraints. To capture topology changes in underlying geometry, you can derive multiple parts from a common layout part that controls the geometry relationships in place of assembly constraints—but that is a topic for another day.

Autodesk Inventor 5 expands the use of adaptive technology to enable cross-part sketch associativity. In this tutorial, we'll examine how and where to take advantage of this new tool and throw in a few tips and tricks along the way. As this tutorial is aimed at advanced Autodesk Inventor users, some of the instructions assume you have a good knowledge of the part and assembly environments.

Download and Open File

First download, unzip, and save the zipped file which contains the following:

- Cross Part.iam (assembly file)
- Cross Part.idv (design views file)
- Ring.ipt



[Download Files](#) (zip - 259Kb)



Figure 1: Cross Part.iam on opening.

To start the tutorial:

1. Extract the files to a folder listed in your current project file.
2. Start the Autodesk Inventor software.
3. Open Cross Part.iam. The assembly should match Figure 1.



Figure 2: Start new part on ring face.

Cross-Part Associativity Option

The Assembly tab in the Application Options dialog box contains a toggle to turn on the behavior of geometry projected across parts. The Enable Associative Edge/Loop Geometry Projection During In-Place Modeling option is on by default.

Tip: You can toggle on the opposite behavior of this setting by holding down the Control key when projecting geometry across parts. With the option checked, the Control key would force projected edges to be nonassociative. (Projected loops are not available during non-associative projection.)

Starting a New Part

To simplify the tutorial we'll create a single new part, a sheet metal cover for the ring, and contrive a few scenarios to examine the new cross-part associativity in Autodesk Inventor 5.

1. In the assembly, create a new part named Cover.ipt, based on the Sheet Metal(in).ipt template. Ensure that the check box, Constrain sketch plane to selected face or plane, is checked, and then start the new part on the flat face of the ring as shown in Figure 2.

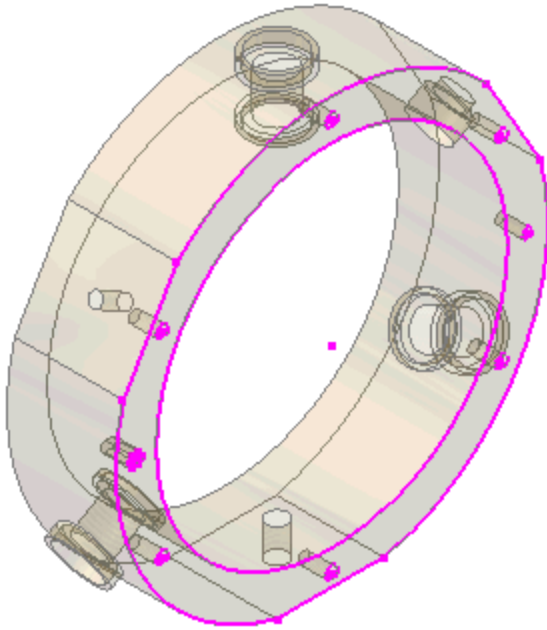


Figure 3: Projected loops from ring.

2. From the Sketch panel bar, click the Project Geometry tool.
3. Move the cursor over the sketch face. Note that when the cursor is over an edge, only the edge is highlighted. When the cursor is over a face, all boundary and internal loops on that face are highlighted. Click on the ring face to project all loops to the current sketch (see Figure 3).

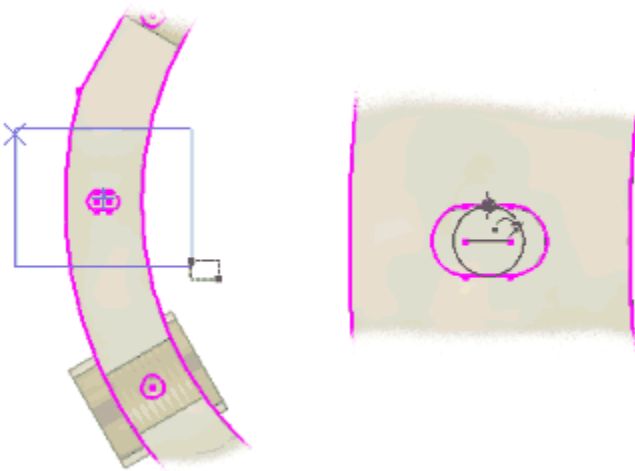


Figure 4: Zoom area and additional sketch geometry.

It is important to differentiate between projecting loops and projecting edges. Each time you project geometry from another part, a reference object is created that contains the associative link. The references are listed under the sketch in the browser. If you project two individual edges (from the same or different parts), two separate references are created.

The part and sketch are marked as adaptive in the browser.

4. Expand Sketch1 in the browser. An adaptive link, Reference1, is listed below Sketch1.

Note: You can manage these adaptive relationships using the same methods as with other adaptive relationships. You can toggle on the adaptive state of a Reference or unconsumed sketch from the shortcut menu.

5. Look at the sketch face and zoom in on the slot shown in the left-side image in Figure 4.

6. Add a construction line between the two arc centers and a normal style circle centered on the construction line, with a radius tangent to the slot edge as shown in the right-side image in Figure 4.

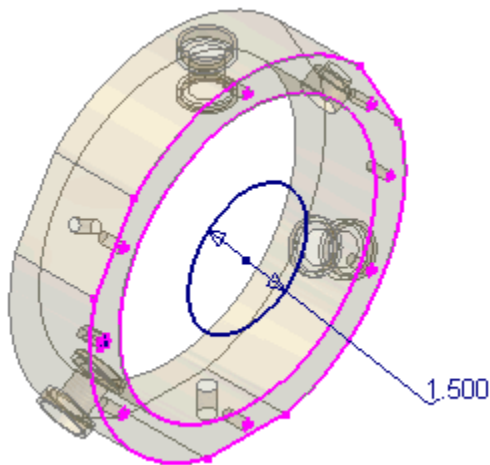


Figure 5: Sketch circle.

7. Finally, add a 1.5-inch diameter sketch circle centered on the ring as shown in Figure 5.

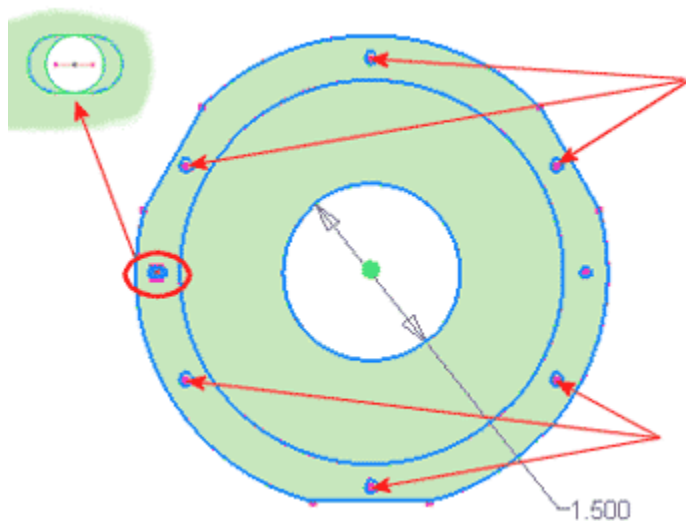


Figure 6: Profiles for sheet metal face.

8. Exit the sketch. Edit the default Sheet Metal Style as follows:

- Thickness = 0.04 in
- Material = Aluminum-6061

Creating a Face

Before we create a face from the current sketch, let's look at what we want to accomplish. The requirements are:

- The cover must match the outer profile of the ring.
- The two dowel holes in the cover must match the dowel hole and slot in the ring.
- Mounting holes in the cover must match the tapped holes in the ring. The number of tapped holes in the ring is controlled by a circular pattern, and the number of tapped holes is subject to change.

The first two requirements are met with our current sketch. The variable number of tapped holes presents a challenge. You can offset the projected circle of each tapped hole to create six matching clearance holes in the cover. If the number of tapped holes were later increased to eight, you would have two tapped holes without matching clearance holes in the cover. For now, let's ignore the loops from the projected holes.

1. Create a sheet metal face, selecting the two outer ring areas, the six tapped hole circles, and the outer profile of the projected slot as profiles (see Figure 6).

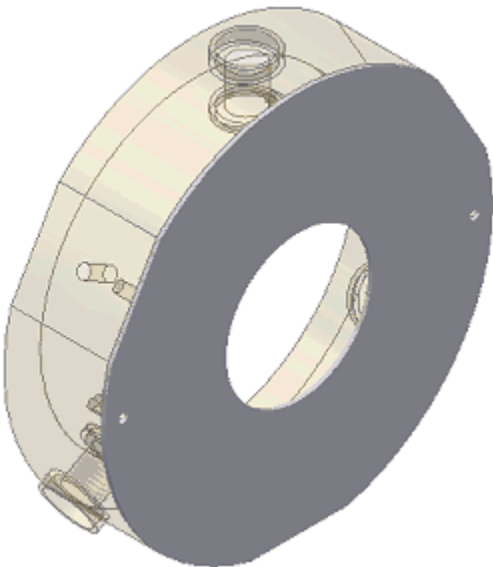


Figure 7: Resulting face.

The cover should match the one shown in Figure 7.

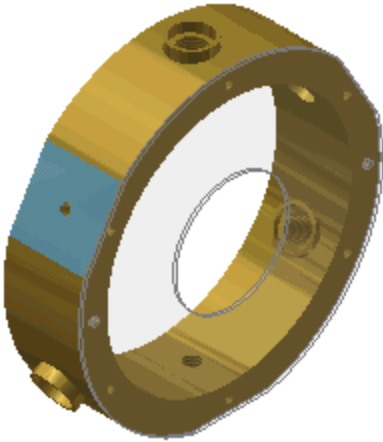


Figure 8: Sketch face on ring.

Adding Additional Geometry to the Ring

Next we will add some additional geometry to the ring and examine how it affects the cover.

1. Return to the assembly, and then in-place edit Ring.ipt. Start a sketch on the face shown in Figure 8.

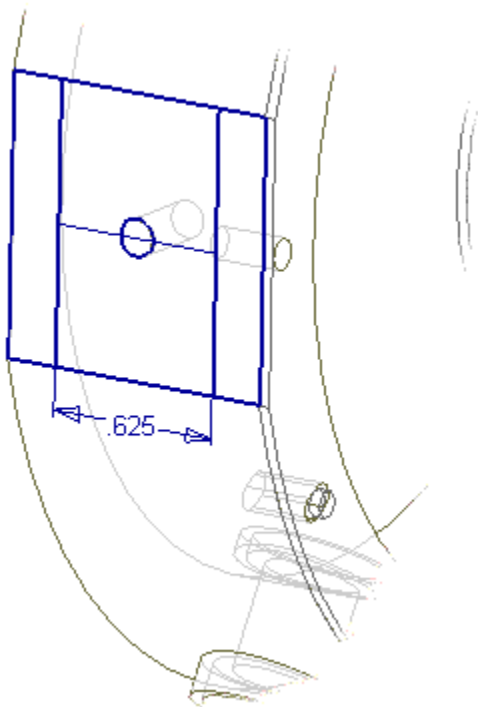


Figure 9: Sketch geometry.

2. Complete the sketch shown in Figure 9. Two horizontal construction lines of equal length center the vertical lines about the hole.



Figure 10: Extruded rectangles.

3. Extrude the two outer rectangles a distance of 0.375 inches. Your ring should match the one shown in Figure 10.

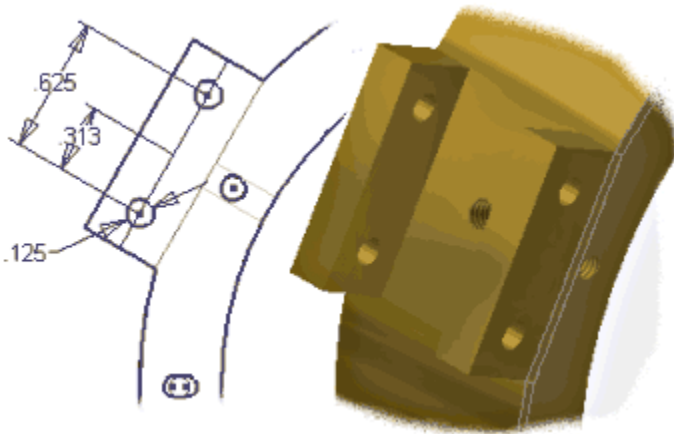


Figure 11: Sketch and cut extrusion result.

4. On the front face of the just completed extrusion, sketch two 0.125-inch diameter circles as shown in the left-side image in Figure 11. Extrude the circles with a Through All to create the feature shown in the right-side image in Figure 11.

Tip: When creating a sketch while in-place editing a component, select the sketch face on the component before activating the sketch (noun-verb). In Autodesk Inventor 5 you can select planar surfaces on other parts to define a new sketch. If you click the Sketch tool and then select a face (verb-noun), you will need to cycle through a number of faces if other parts are in front of the desired face on the active part.

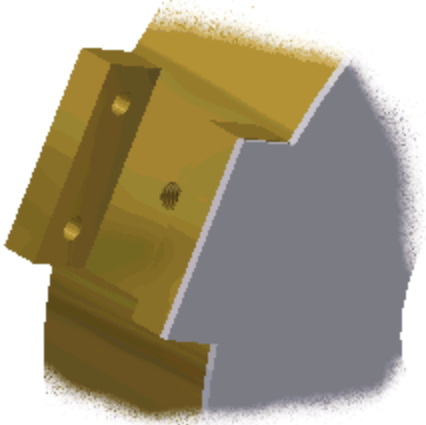


Figure 12: Cover adapts to outer loop change.

5. Now, return to the assembly environment. The cover adapts to match the change to the outer loop of the ring face, but the two additional inner loops (circles) are not considered in the cover extrusion (see Figure 12).

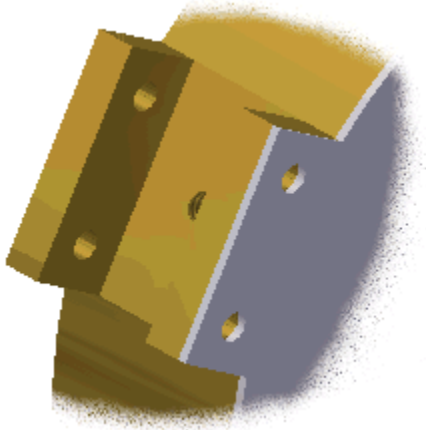


Figure 13: Circle profiles removed from extrusion.

6. In-place edit the cover, and edit the Extrusion1 feature. Remove the two circle profiles for the through holes, and then return to the assembly environment. The cover should match the one shown in Figure 13.

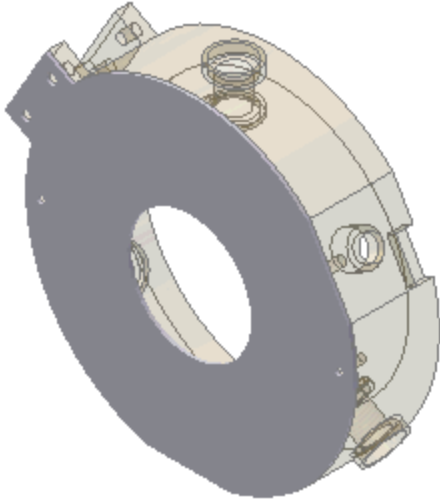


Figure 14: Reoriented view.

Changes that modify boundary loops of the underlying geometry are automatically reflected in the associated sketch. If the Reference is a projection of all loops on a face, interior loops added as a result of additional modeling are automatically added to the projection, but do not automatically modify previously created geometry in the associated part. (Try saying that fast, three times.)

Adaptive Behavior

Because the cross-part associativity is an extension of Autodesk Inventor software's adaptive technology, you can treat the cover like other adaptive parts. In the assembly, turn off adaptivity for Cover.ipt. You can drag the cover about the assembly, constrained only by the Flush constraint generated during part creation. You can easily create a gap between the parts, simply by changing the offset value of the Flush constraint. Toggle Cover.ipt to be adaptive again, and it snaps back into position.

Adding a Face

Next, we'll add a face to the cover using a direct reference to a face on the ring. We'll examine cross-part projected edges in the sketch for the face.

1. In-place edit the cover. Reorient your view to match Figure 14.

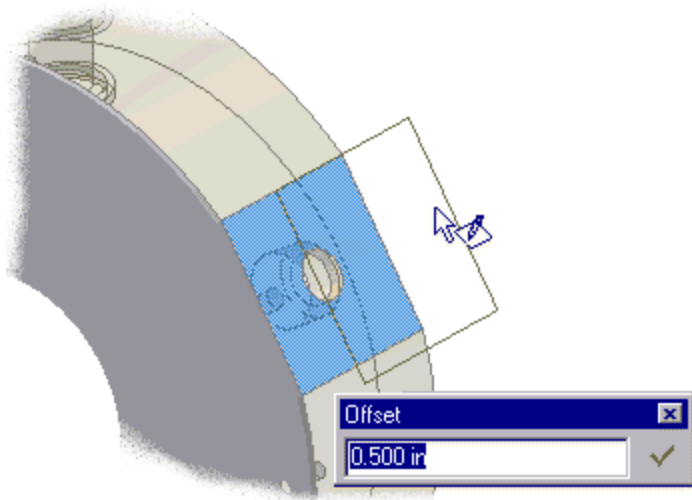


Figure 15: Offset plane creation.

2. Click the Sketch tool and then click and drag off the ring face as shown in Figure 15. Offset the plane 0.500 inches from the ring face.

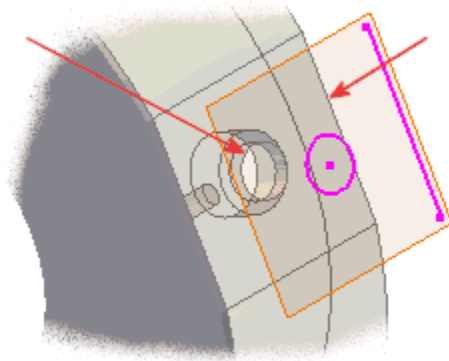


Figure 16: Projected edges.

The same technique can be used in Autodesk Inventor 4. In Autodesk Inventor 5, a sketch is automatically created, and is active, on the adaptive work plane created above.

3. Click the Project Geometry tool and project the circle and edge shown in Figure 16. Expand the sketch in the browser and note there are two References listed under the sketch, one for each projected edge.

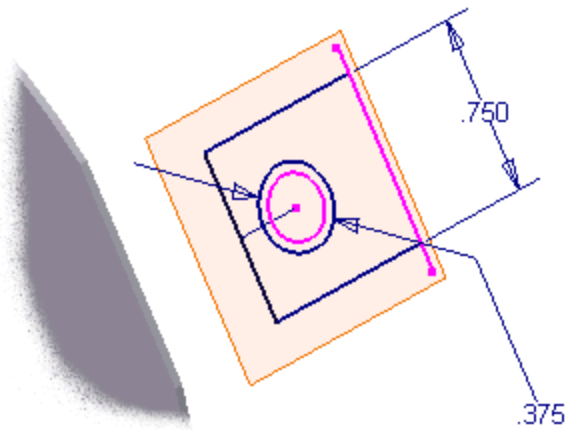


Figure 17: Sketch for sheet metal face.

4. Sketch three perpendicular lines as shown in Figure 17. Two lines are coincident to the projected edge. The construction line centers the sketch about the hole. For clarity, the ring part is not shown in Figure 17.

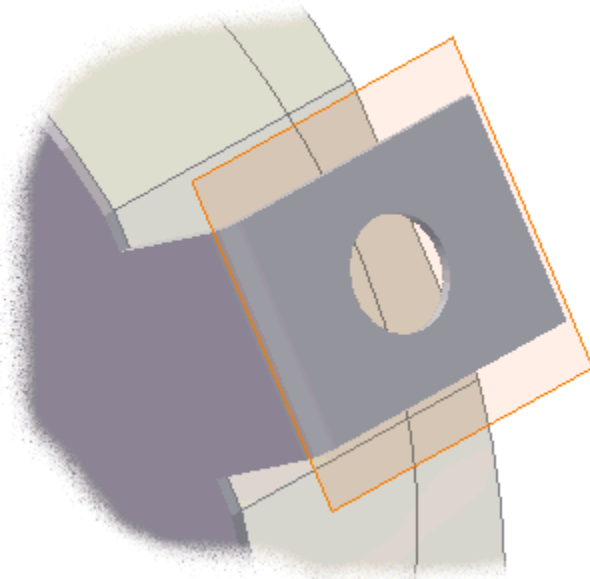


Figure 18: Completed sheet metal face.

5. Exit the sketch and create a sheet metal face. Click the Select Edge tool in the Face dialog box, and then click the adjacent parallel edge on the cover to add a bend between the two faces. Your part should match Figure 18.

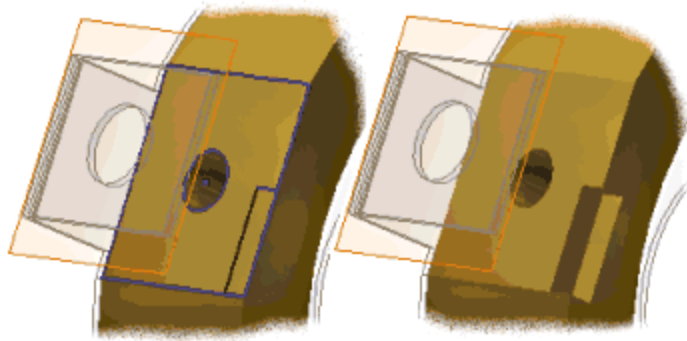


Figure 19: Sketch and extruded cut on ring.

6. Return to the assembly and then in-place edit the ring. Sketch two lines on the face under the sheet metal tab just created to define the rectangle shown in the left-side image in Figure 19. Extrude a cut similar to the one shown in the right-side image in Figure 19.

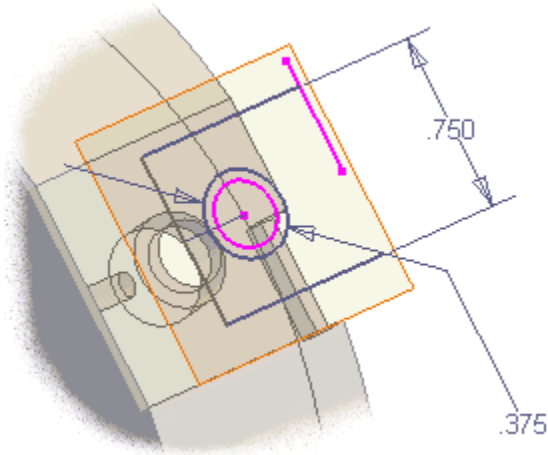


Figure 20: Sketch failure due to modified projected edge.

7. Now, return to the assembly environment. Accept the reported error. Note that the sheet metal face has disappeared.

8. In-place edit the cover. Edit the sketch under Face2. Notice that the projected edge has been shortened by the extruded cut added to the ring (see Figure 20).

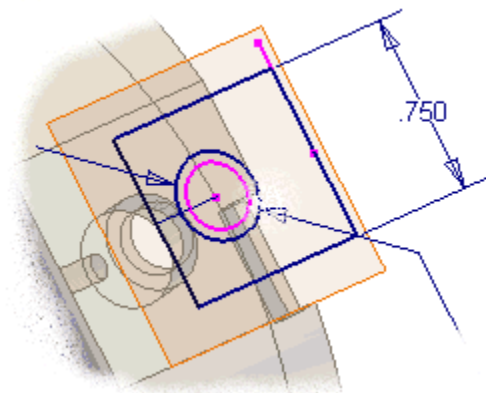


Figure 21: Modified sketch.

9. Fix the sketch by adding a line between the two sketch lines as shown in Figure 21.

| | | | | |
|------------|-----|---------|------------|-------------------------------------|
| d57 | in | 0.25 in | | |
| d58 | deg | 118 deg | 118.000000 | <input type="checkbox"/> |
| Hole_Count | ul | 6 ul | 6.000000 | <input checked="" type="checkbox"/> |
| d61 | deg | 360 deg | 360.000000 | <input type="checkbox"/> |

Figure 22: Parameter renamed and exported.

In general, avoid coincident constraints between sketch points and projected geometry, especially if the underlying edge may be modified during the design process. Depending on the circumstances, it may be a better choice to project the face (all loops), and then add appropriate sketch geometry to get the desired profile(s). In this exercise, a better method would be:

- Sketch a rectangle.
- Add a colinear constraint between the projected edge and the adjacent edge of the rectangle.

Note: This behavior is not unique to Autodesk Inventor. Other modelers that use cross-part associativity display similar behavior when underlying edges are modified.

Note: Unexpected problems can also arise when using projected loops. If the underlying face is split, the projected loop will reflect the loops of one side of the split face.

Now, What About Those Mounting Holes

Let's examine a new feature of derived parts that enables us to capture the number of tapped holes in the ring.

1. In-Place edit the ring. In the browser, right-click Tapped Hole Pattern and select Show Dimensions.
2. Double click the #6.000 dimension, and note the parameter name (d59) in the edit box. Exit the edit box without modifying the value.
3. Open the Parameters dialog box, and then change the name of parameter d59 to Hole_Count. Also check the Export Parameter option for this parameter (see Figure 22).

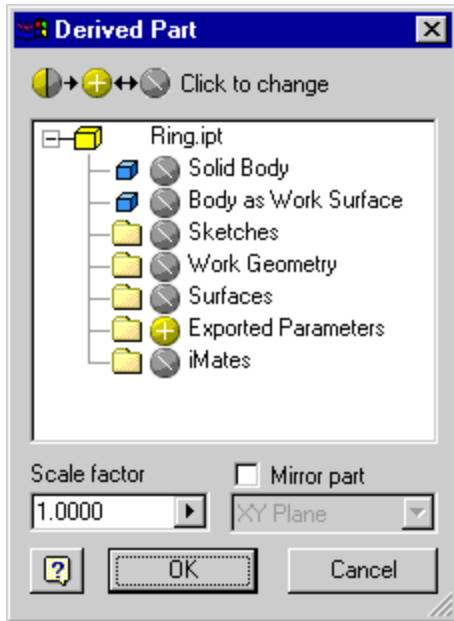


Figure 23: Derived Part dialog box.

4. Return to the assembly environment and in-place edit the cover.
5. Change to the Features panel bar and click the Derived Component tool. Select Ring.ipt and click Open. You can derive exported parameters from the selected part. Click OK to accept the default setting, derive Exported Parameters only (see Figure 23).

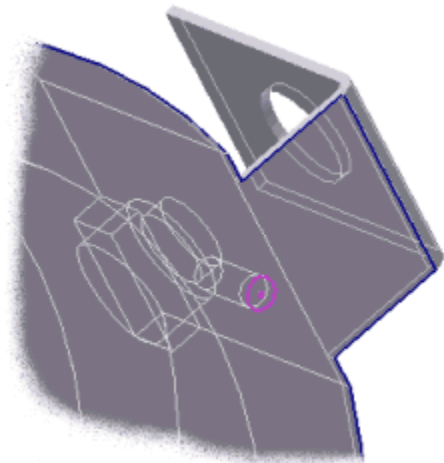


Figure 24: Projected edge from tapped hole.

6. Start a sketch on the front face of the cover.
7. Project the tapped hole edge from the ring as shown in Figure 24.

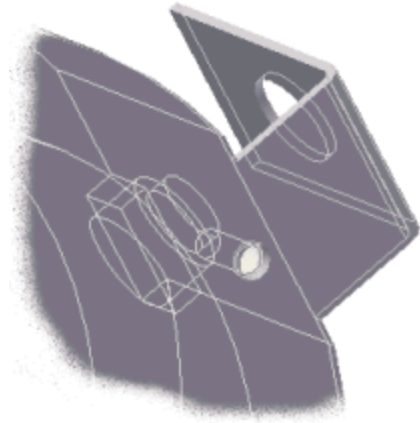


Figure 25: Clearance hole in cover.

8. Use the center point of the projected edge as the center for a 0.136-inch diameter clearance hole in the cover (use Through All for the hole termination). The cover should match Figure 25.

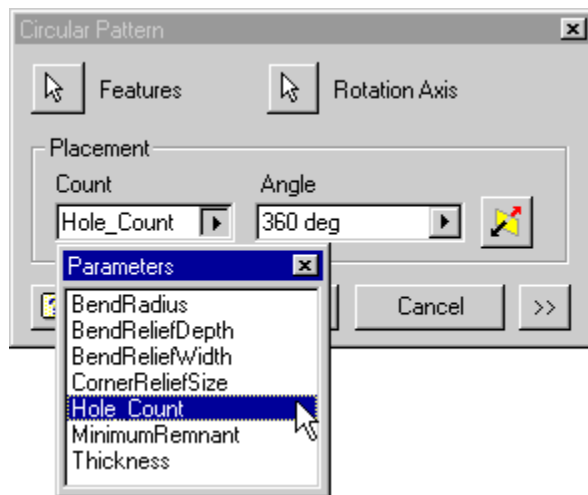


Figure 26: Selecting derived parameter for circular pattern count.

9. Click the Circular Pattern tool. Select the clearance hole as the feature to pattern, and click the surface or edge of the large inner hole in the cover to define the rotation axis.

10. Click the arrow to the right of the Count edit box. Select List Parameters from the shortcut menu. Click Hole_Count in the Parameters list (see Figure 26). Click OK to complete.

11. Now, return to the assembly environment and in-place edit the ring. Make the following changes to Ring.ipt:

- Edit the sketch under Extrusion1. Change the outer diameter of the ring from 4.000 inches to 4.125 inches.
- Edit the sketch under the Tapped Hole feature. Change the angle dimension from 30 degrees to 18 degrees. Change the 1.813-inch dimension to 1.875 inches.
- Edit the Tapped Hole Pattern feature and change the count from 6 to 5.

12. Return to the assembly environment. The cover updates to match the changes to the ring.

It is good practice to turn off adaptivity (at the part level) when editing is complete. You can turn it back on if you need to edit the parts at a later date. You can add additional assembly constraints to ensure parts with cross part projections remain assembled with adaptivity toggled off.

Summary

Autodesk Inventor 5 enables you to generate designs using top-down design methods. Cross-part associative sketches can be a help in capturing complex topology, but you must be aware of how changes in the underlying geometry will affect the associated sketch. You can turn off the cross part associativity setting until you feel comfortable with the consequences. In addition, the Derived Component tool enables you to easily share parameters between parts, without the use of an external spreadsheet.