

# Using Projected Geometry in Assemblies

By Neil Munro

In this Autodesk Inventor™ tutorial we'll focus on the use of projected geometry to include geometry from existing parts when creating a new part in an assembly. You can also take advantage of the software's adaptive technology and other techniques for building relationships between parts in an assembly; we'll cover these topics in future articles.

When would you use projected geometry between parts in an assembly? When the reference part is a "fixed" or released design in which the projected geometry will not change. You can also project geometry from one part into a new part and then change the line style of the projected entities from a fixed "Reference" style to "Normal" or "Construction." This projected geometry can then be constrained and driven by dimensions similar to those of other sketch entities in the new part.

As we use these projected-geometry techniques to help build a part in an assembly, we'll also examine some helpful Autodesk Inventor viewing tools.

## Download and Open File

First, download, unzip, and save to a folder on your hard drive the ZIP file containing the source files for this tutorial.

It contains the following:

- Cut\_Project.iam (assembly file)
- Cut\_Project.idv (assembly design view file)
- Glass.ipt (glass insert part file)
- Ring.ipt (chrome ring part file)

Now, let's get started.

1. Extract the files to a folder on your hard drive.
2. Start Autodesk Inventor.
3. Click the Open tool on the Standard toolbar.
4. Browse to the folder containing the extracted files.
5. Select Cut\_Project.iam.
6. Click OK.

 [Source Files](#) (zip - 206Kb)

## The Current Assembly

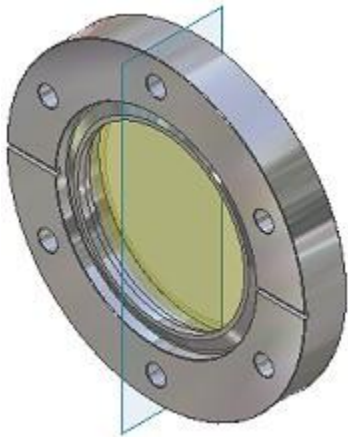


Figure 1: Assembly on Opening.

The assembly we're going to be working on currently consists of a chrome mounting ring and a glass insert. A visible work plane bisects the assembly (see Figure 1). Let's assume the ring is a standard part in many designs and can accept inserts of varying thickness and diameter.

### Let the Games Begin

In this tutorial, you will be designing an intermediate support using projected geometry from both the ring and the insert. Here's how.

### Create Part In-Place

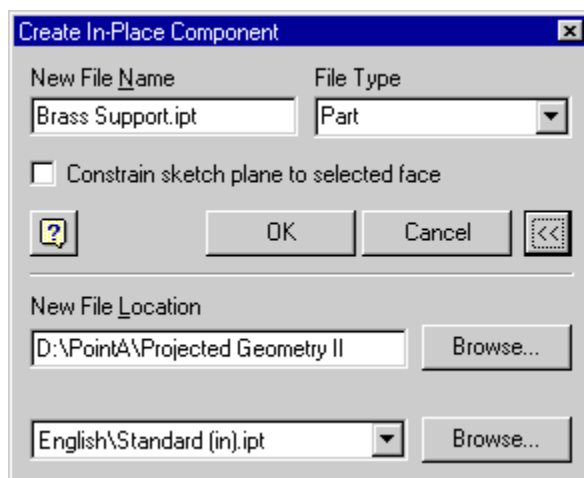


Figure 2: Create In-Place Component Dialog Box.

1. From the Assembly Panel Bar, click the Create Component tool.

2. In the Create In-Place Component dialog box (see Figure 2):

- Enter Brass Support as the name for the new part.
- Click the More button.
- In the New File Location area of the Component dialog, click the bottom Browse button, which takes you to the Open Template dialog box. There you choose a template for the new assembly.
- In the Open Template dialog box:
  - Click the English tab.
  - Select Standard (in).ipt as the template for the new part.
  - Click OK to select the template and to return to the Create In-Place Component dialog.
- Clear the check mark next to "Constrain sketch plane to selected face."
- Click OK.

3. Click the visible work plane that bisects the assembly to define the sketch plane for the base feature of your new part.

**Note:** The outer ring contains a number of edges that you want to reference as the basis for the first feature of the support. It is difficult to see the required edges because the ring is so complex. We'll take care of that problem shortly.

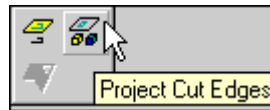


Figure 3: Project Cut Edges Tool.

4. From the Sketch Panel bar, click the down arrow next to the Project Geometry tool and click the Project Cut Edges tool (see Figure 3).

5. Click the ring in the graphics window.

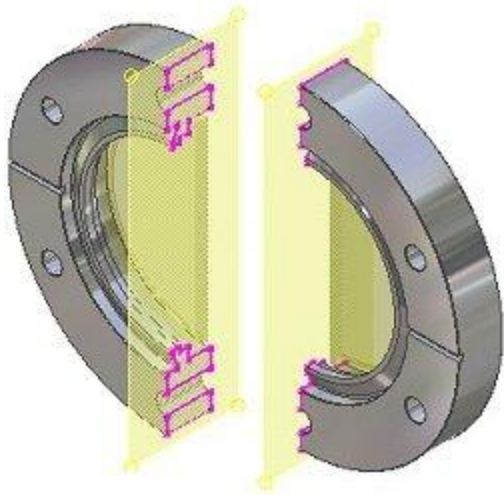


Figure 5: Half-Section View and Flipped Section.

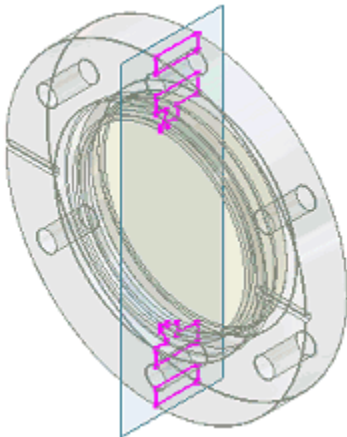


Figure 4: Projected Cut Edges.

A cross section of the ring cut by the sketch plane is projected onto the sketch. The sketch remains a bit confusing because the ring geometry still somewhat obscures the sketch area.

### Assembly Sections

You can get a better view of the assembly by cutting it into one-quarter, half, or three-quarter sections. The one-quarter and three-quarter sections allow you to select any two intersecting planes to create virtually any assembly slice. Better yet, an assembly section can be saved in a Design view and recalled when editing the assembly. Unfortunately, assembly sections don't carry through the Design views to presentations or drawings.

1. Double-click the top-level assembly or click the Return tool twice to return to the assembly level.

**Tip:** The Return tool is new in Autodesk Inventor 4. A history of your editing locations is retained and clicking the Return tool moves you back through this

history, not just to the level above your current environment.

2. Click the Down arrow next to the End Section View and select Half-Section View.
3. Click the visible work plane. The half of the assembly closest to you is removed.
4. Right-click in the graphics window and select Flip Section from the pop-up menu. The original section and the flipped section are shown in Figure 5.

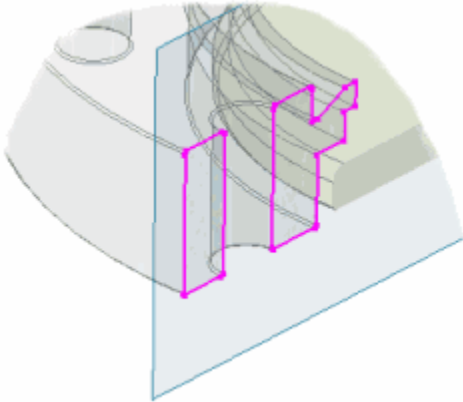


Figure 7: Reoriented View.

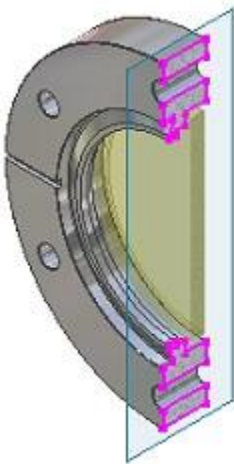


Figure 6: Sketch Exposed by Section View.

5. Right-click and select Flip Section again to return to the original half. Right-click and select Done. You now have a much better view of the sketch (Figure 6).
6. Right-click Sketch1 under Brass Support.ipt in the browser and select Edit Sketch from the main menu. The Brass Support part is activated and you are ready to sketch the profile defining the shape of the part.
7. Reorient your view to match the one shown in Figure 7.

## Project Geometry

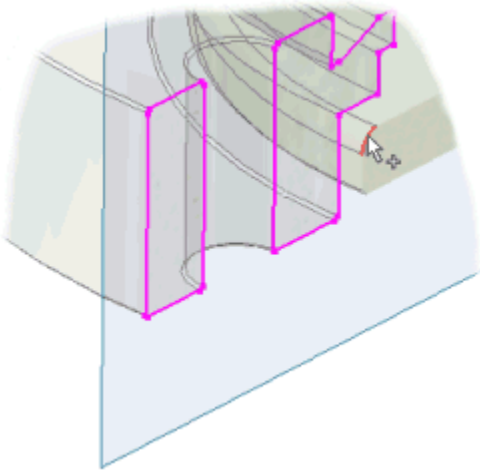
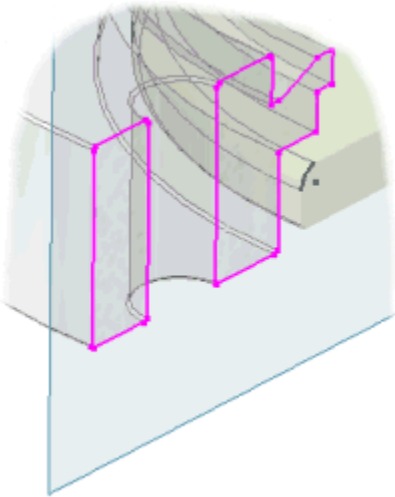


Figure 8: Single Projected Edge.



1. From the Sketch Panel bar, click the Project Geometry tool.
2. Click the filleted edge of the glass insert as shown in Figure 8.

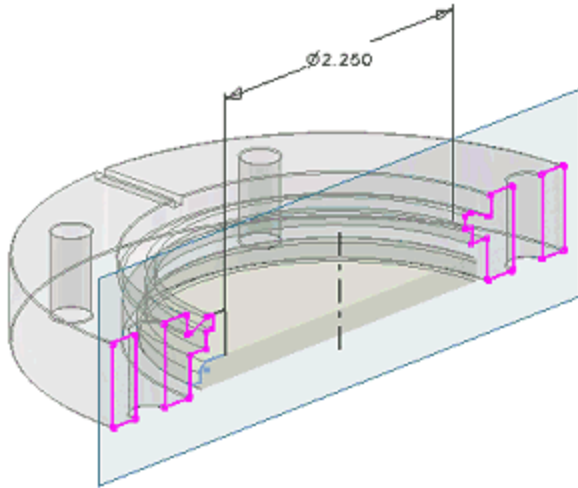


Figure 10: Projected Axis and Dimension.

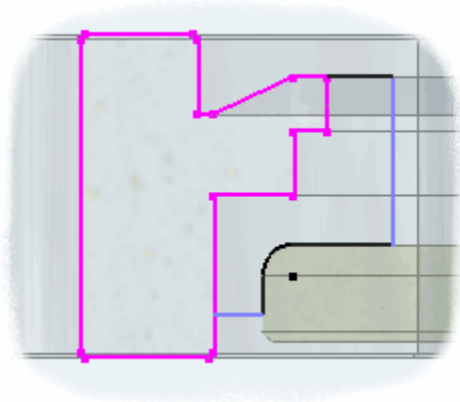


Figure 9: Completed Sketch.

3. Use the Line tool to complete the sketch shown in Figure 9.

Note the lines (black) tangent to the projected arc are positionally constrained, as they are connected to the "fixed" reference geometry.

4. Right-click in the graphics window and select Done.

5. In the graphics window, click the projected arc and select Normal from the Style list on the Command bar. The arc and the lines tangent to the arc are now underconstrained.

6. From the Sketch Panel bar, click the Project Geometry tool again.

7. Expand the Origin under Brass Support.ipt and click Y-Axis. The reference axis is projected onto the sketch.

8. In the graphics window, click the projected axis and select Centerline from the Style list.

9. Place a dimension between the projected axis and the vertical line as shown in Figure 10.
10. Press the R key (R is a keyboard shortcut for the revolve tool). Select the profile highlighted in Figure 11 and select the centerline as the axis. Click OK to complete the revolved feature.

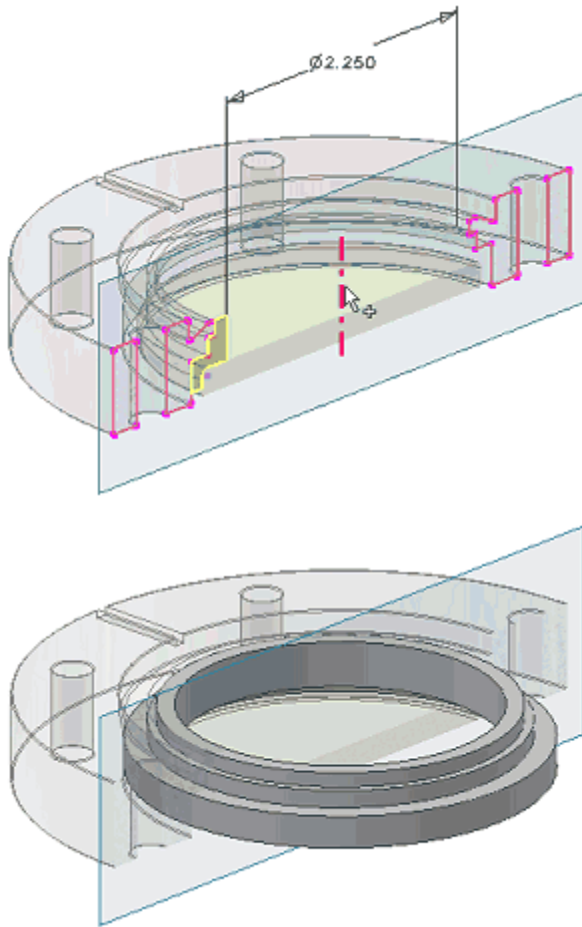


Figure 11: Revolved Feature.

11. Click the Return tool to activate the top-level assembly.
12. In the graphics window, click and drag the Brass Support part away from the other components. There is no inter-part relationship that locks the position of the support.

### What's Next

At this stage, you need to add assembly constraints to locate the support in the assembly. In addition, you can make the revolved feature adaptive and the underconstrained geometry of the feature can change size (adapt) to match other assembly constraints. For example, the Brass Support part could adapt to match a change in the diameter of the insert.

For now, you should have a good grasp of how and when to project geometry in your Autodesk Inventor models. We will discuss adaptive parts and assemblies in a future article. We will also look at using derived parts and assemblies to create relationships between components.