iFeatures and Punches

By Neil Munro

This month's tutorial introduces the new sheet metal punch tool available in Autodesk Inventor™ 5. Because a punch is a special type of iFeature (iFeature is the new name for Design Element), we'll also examine the creation of a moderately complex iFeature. Some of the techniques covered in the tutorial can be used in previous versions of Autodesk Inventor, but to complete the tutorial you will need Autodesk Inventor 5 installed on your computer.

The Punch Tool

Sheet metal part production is often highly automated. Flat stock is cut to shape and internal cutouts, reliefs, and deformations such as dimples, lances, and louvers are created in CNC turret punch machines prior to forming the folded part. Each punch "hit" is programmed with a tool number and tool center location on the flat sheet. Some machines can also rotate either the tool, or the stock, to add an angle variable to the mix.

You can now emulate this process to some extent in Autodesk Inventor 5 using the new sheet metal punch tool. And you can save enhanced iFeature shapes and forms in a folder reserved for punches and placed in a manner similar to a hole feature. The obround cutouts in Figure 1 are a single punch feature located by an array of Hole Center sketch entities.

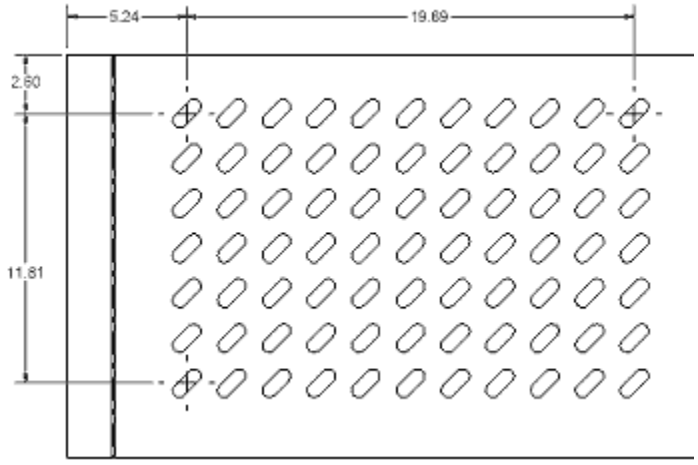

Figure 1: Punched sheet metal cutouts.

iFeature Creation

Saving and placing punches is a straightforward procedure in Autodesk Inventor software. However, creating an easy-to-use iFeature that captures your design intent can be a different

story. Simple cutout shapes are not difficult to capture. The only wrinkle when creating a cutout to be used as a punch is to ensure that the tool center remains in the correct position when the punch changes size. You can constrain the tool center to a fixed position such as the midpoint of an edge, or you can keep it centered with equations (see Figure 2).

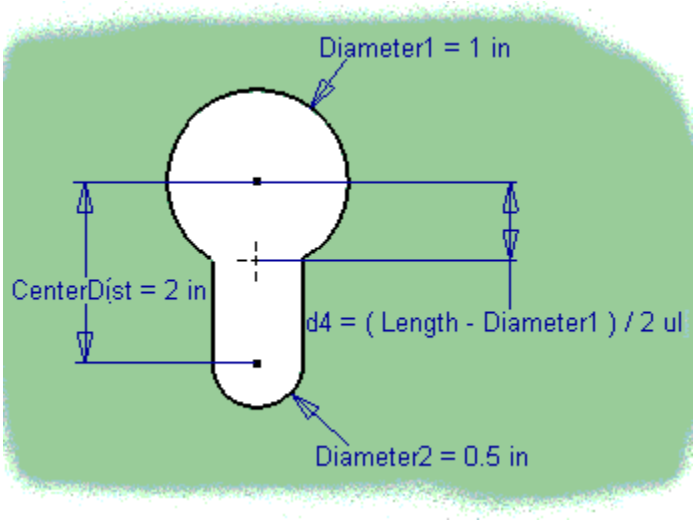

Figure 2: Tool center controlled by equation.

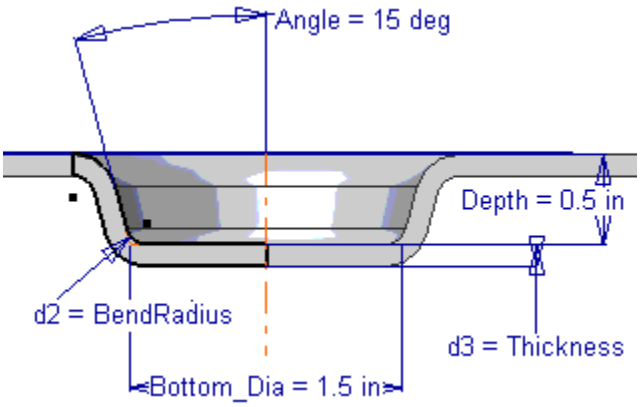

Figure 3: iFeature shape.

iFeatures that represent deformations, such as dimples and louvers, present a greater challenge. The definition of these shapes often requires multiple features, which can include cuts, extrusions, revolves, or sweeps. Although iFeatures can include multiple placement references (planes, edges, and even sketch geometry in Autodesk Inventor 5), an iFeature that has a single placement reference (placement plane) will usually be more reliable when placed as either a standard iFeature or punch.

The iFeature we'll create in this tutorial arose from a request from Kent Keller in the Autodesk Inventor discussion group. The request was for a dimple iFeature that was filleted to match the BendRadius setting for the sheet metal style and whose size was controlled by the diameter at the

bottom of the dimple (see Figure 3).

The obvious approach when creating a dimple is to cut out the shape in the sheet metal part and then add the deformed shape to "fill-in" the cutout. That won't work in this case because the bottom diameter and angle variable control the size of the cutout. This means you must define the deformation before the cutout. The solution is to add the revolved shape shown in Figure 3, and then cut out inside the dimple using projected sketch geometry from the initial revolved feature. The last challenge is to create the features so that the saved iFeature can be placed with a single reference to a point on a sheet metal face. Let's get started.

Create a Base Face

The key to creating an iFeature that has a single reference feature is to ensure that all sketch plane references and projected geometry come from a single plane. In addition, a punch iFeature must include at least one hole-center sketch entity in the first sketch included in the iFeature.

1. Start a new sheet- metal part based on the Sheet Metal(in).ipt template.
2. Sketch a rectangle approximately 10-inches square. Use the Face tool to create a sheet metal face.
3. Click the top surface of the face and press the S key to start a new sketch. Note that the outer edges of the face are automatically projected into the sketch (see Figure 4).
4. Select (use a window select) and delete all projected edges before proceeding.

Make a habit of deleting all unnecessary sketch geometry when creating iFeatures. All sketch geometry in the interface sketch (first sketch in the saved iFeature) is displayed during iFeature placement.

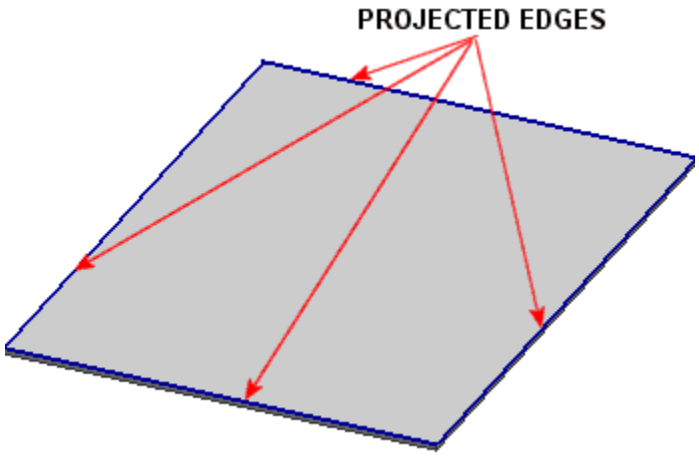

Figure 4: Projected edges.

Create an Interface Sketch

The first sketch must contain a hole center to define the punch tool center. So we'll use a construction line to define a perpendicular work plane that references only the interface sketch. Then we'll model the dimple and the interior cutout as revolved features on the work plane.

1. Sketch a horizontal construction line similar to the one shown in Figure 5.

2. Delete the horizontal constraint on the line.

> **Tip:** Avoid horizontal or vertical sketch constraints in the interface sketch for an iFeature.

3. Place a hole center at the midpoint of the construction line as shown in Figure 5.

4. Right-click and select Finish Sketch.

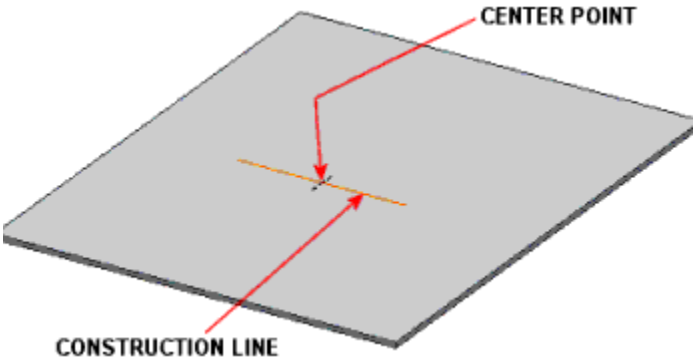

Figure 5: Interface sketch.

Create a Work Plane

You now create a work plane that only references the plane containing the interface sketch.

1. From the Sheet Metal panel bar, click the Work Plane tool  .

2. Click the construction line, and then click the face containing the construction line.

3. Click the green checkmark in the Angle edit box. The work plane should match the one shown in Figure 6.

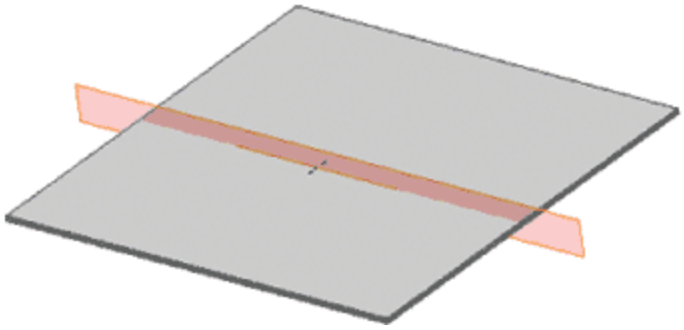

Figure 6: Work plane perpendicular to interface sketch.

Sketch the Dimple Shape

Now we'll sketch a profile that will be revolved to create the "as-formed" dimple shape.

1. Right-click the work plane in the browser or graphics window and select New Sketch from the shortcut menu.

2. From the Sketch panel bar, click the Project Geometry tool  .

3. Click the construction line from the previous sketch. Note that the projected geometry only references the interface sketch.

4. Right-click in the graphics window and select Slice Graphics. Your view should match the one shown in Figure 7.

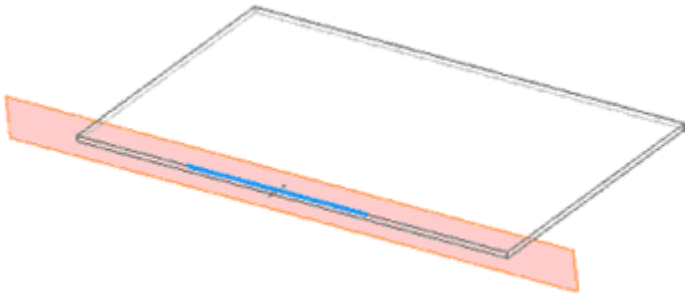

Figure 7: New sketch with Slice Graphics turned on.

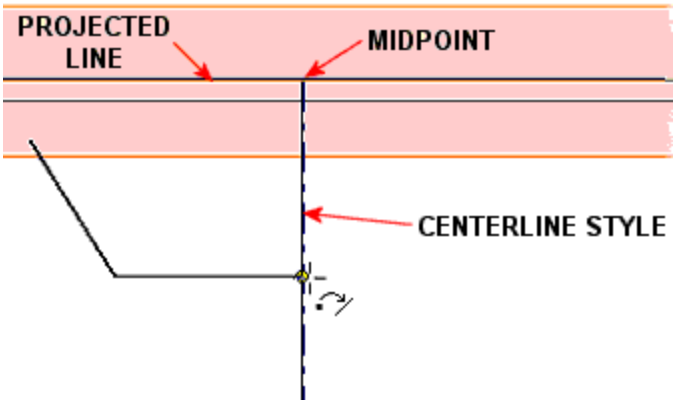

Figure 8: Dimple sketch entities.

5. Draw a vertical line downward from the midpoint of the projected construction line. Select the line and select Centerline from the Style list.

6. Draw two connected lines as shown in Figure 8. The endpoint of the horizontal line is coincident to the centerline.

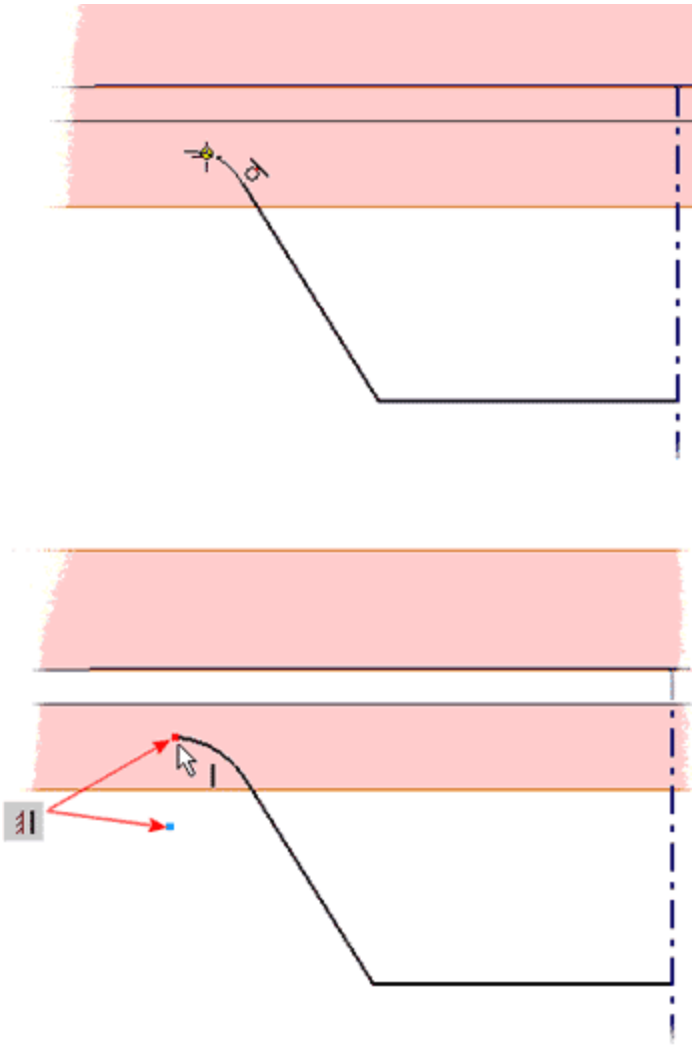

Figure 9: Dimple radius and constraint.

7. Add an arc to the end of the angled line as shown in the upper image of Figure 9. Add a vertical constraint between the free arc endpoint and the arc centerpoint as shown in the lower image of Figure 9.

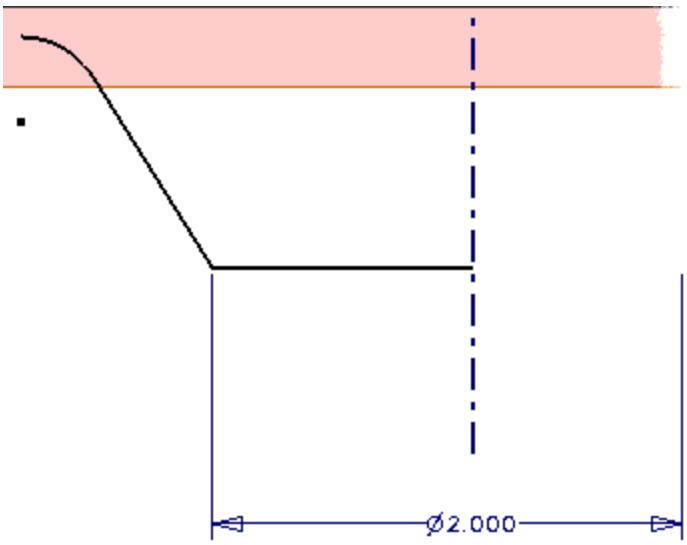

Figure 10: Bottom diameter dimension.

One of the driving dimensions for the punch is the diameter at the bottom of the dimple. We'll add this dimension before adding the radius between the wall and floor of the dimple.

8. From the Sketch panel bar, click the Dimension tool. Click the endpoint of the horizontal line and then click the centerline. Place the dimension as shown in Figure 10.

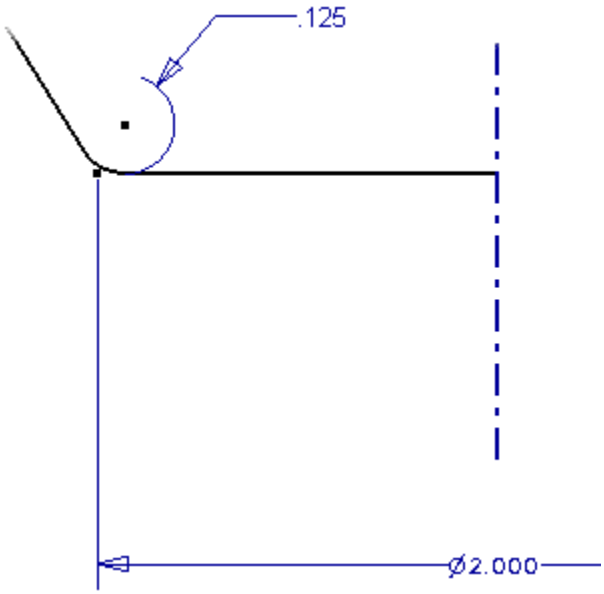

Figure 11: Bottom fillet.

9. From the Sketch panel bar, click the Fillet tool  . Click the angled and horizontal lines to add the fillet shown in Figure 11.

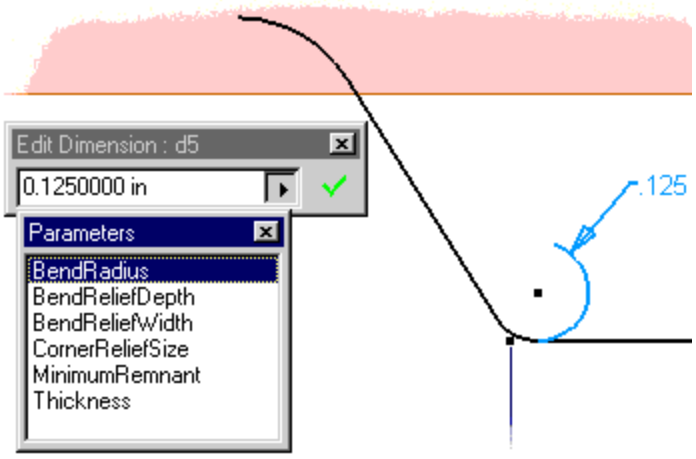

Figure 12: Parameters list.

10. Double-click the .125 fillet dimension. Click the arrow on the right side of the edit box and select List Parameter from the pop-up menu. All named parameters in the part are displayed (see Figure 12). Select BendRadius from the list of parameters. Click the green checkmark in the edit box to complete the dimension.

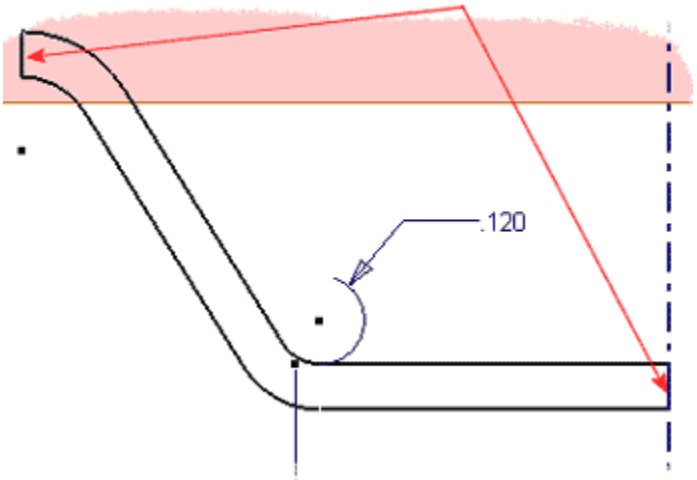

Figure 13: Completed sketch geometry.

11. Use the Offset tool to copy the current sketch entities, and then add vertical lines to close the open ends of the sketch as shown in Figure 13.

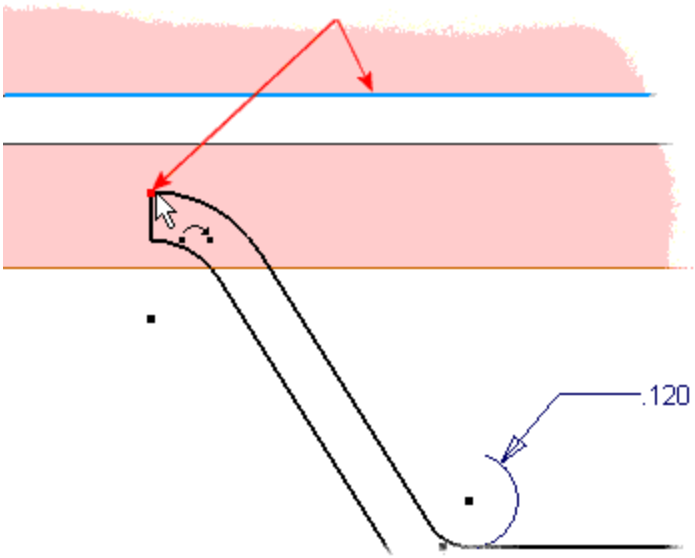

Figure 14: Coincident constraint selections.

The final sketch constraint ties the sketch to the projected construction line.

12. From the Sketch panel bar, click the Coincident constraint tool  . Click the endpoint of the vertical line as shown in Figure 14. Click the projected construction line to complete the constraint. The upper diameter of the dimple can now adjust to match changes to the driving dimensions of the sketch.

Note: Ensure that the projected construction geometry is selected. Use the Select Other tool to cycle the selection if the complete edge is initially highlighted.

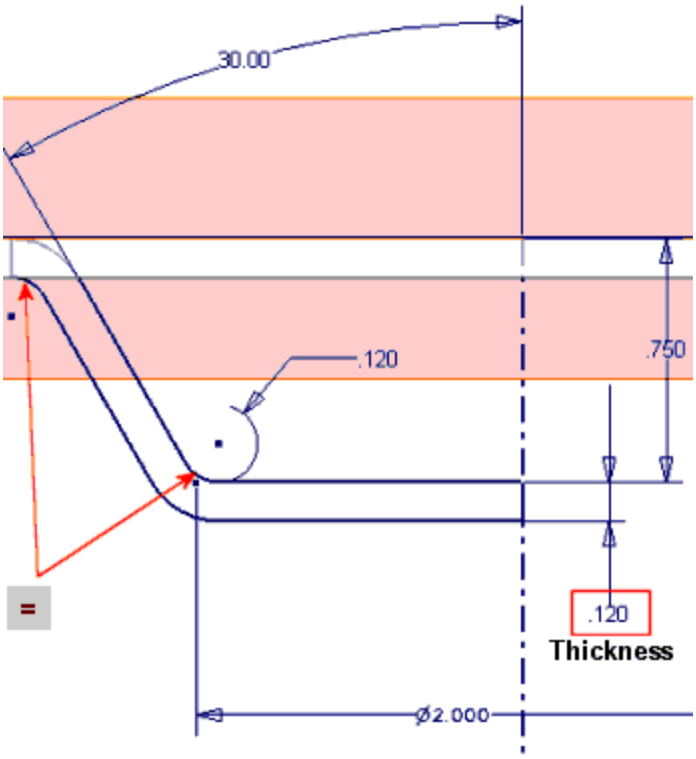

Figure 15: Sketch constraints and dimensions.

13. Complete the sketch by adding the constraints and dimensions shown in Figure 15. Use the List Parameters technique to select the Thickness parameter for the highlighted dimension.

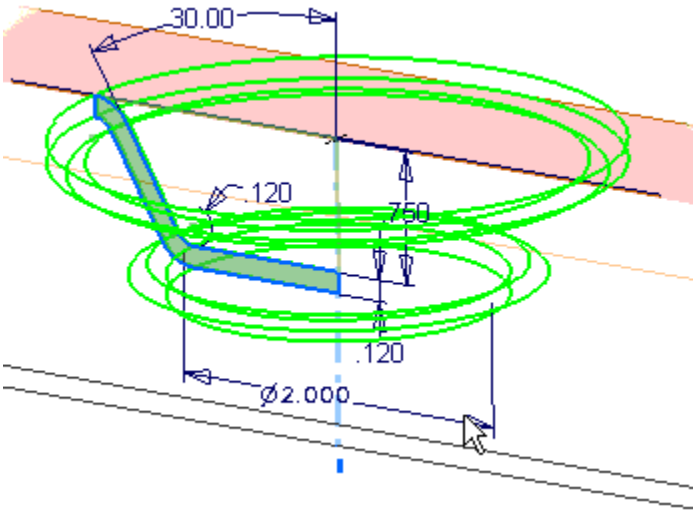

Figure 16: Revolved-feature preview.

14. Finally, press the R key to start the Revolved feature tool. Select the profile as shown in Figure 16. Select the centerline as the feature axis. Click OK to complete the feature.

Complete the iFeature Rotation

You complete the iFeature by adding a second revolved feature that cuts out the interior of the dimple.

1. Start a new sketch on the work plane. Use the Slice Graphics tool to expose the sketch plane.

2. In the browser, expand Revolution1, Right-click Sketch3 and select Visibility from the pop-up menu. Your view should match the one shown in Figure 17.

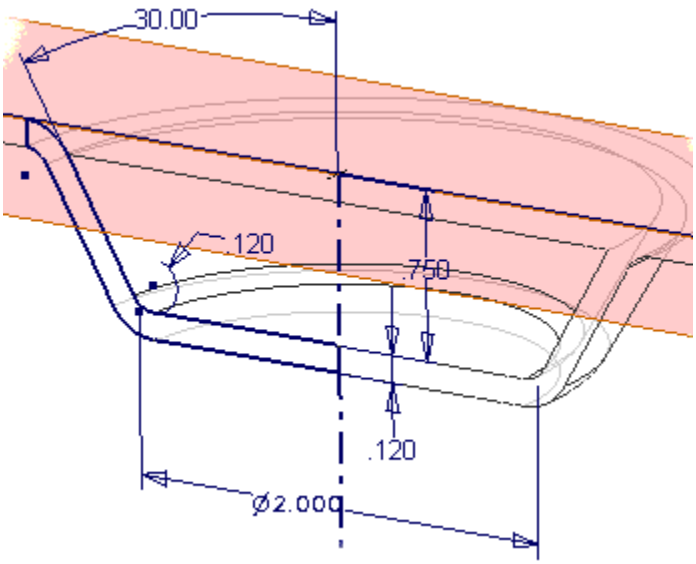

Figure 17: New sketch and visible Sketch3.

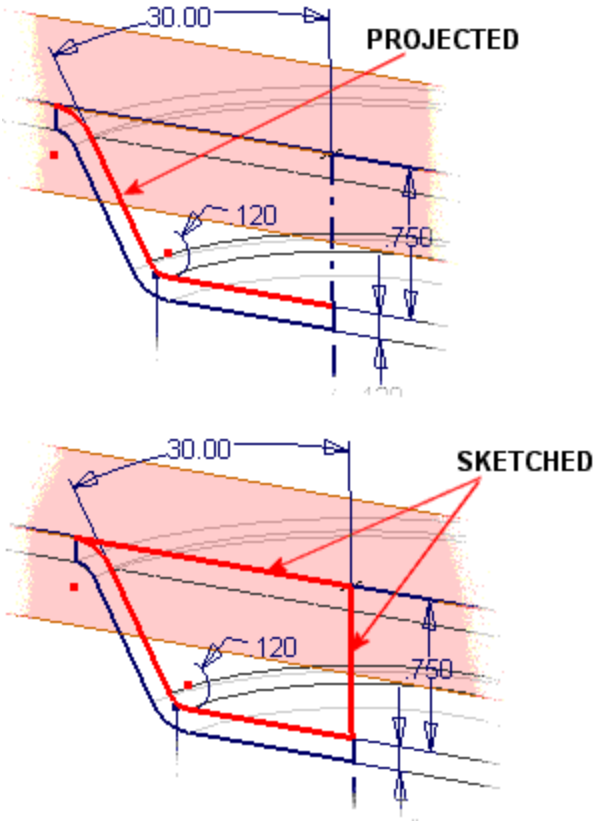

Figure 18: Projected and sketched geometry.

3. From the Sketch panel bar, click the Project Geometry tool . Project the sketch geometry from Sketch3 shown in the upper image of Figure 18. Sketch the two lines shown in the lower image of Figure 18. The two lines are coincident to the endpoints of the projected geometry.

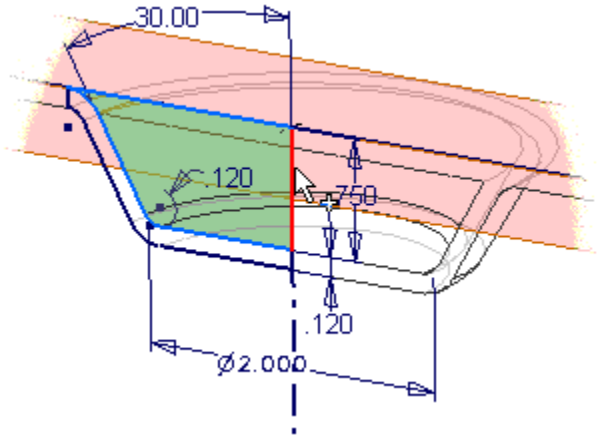

Figure 19: Revolved cut selections.

4. Press the R key to start the Revolved feature tool. Click inside the profile if it is not automatically selected. Select the vertical sketched line as the revolve axis, as shown in Figure 19. Select the Subtract option in the Revolve dialog box and click OK.

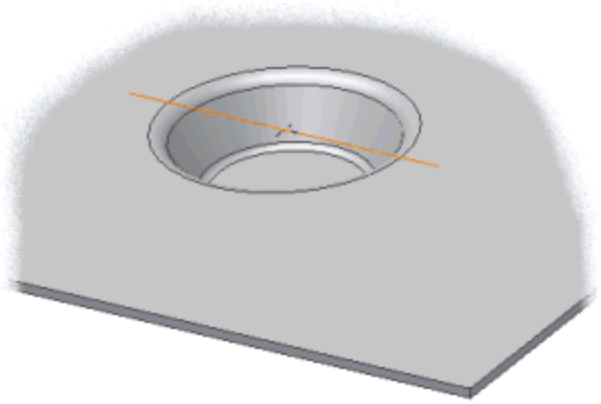

Figure 20: Completed dimple definition.

5. Turn off the visibility of Sketch3 and the work plane. The part should match Figure 20.

Save as a Punch

Next, the appropriate features are saved as an iFeature in the Punches folder.

Note: You can specify the Punches folder on the iFeature tab in the Application Options dialog box.

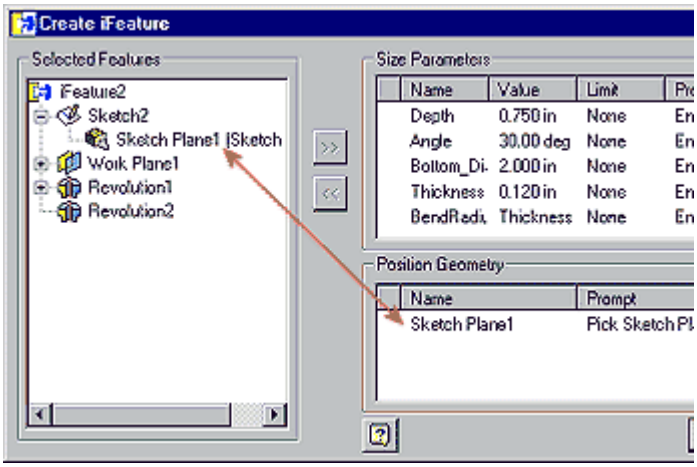

Figure 22: Create iFeature with single-position reference.

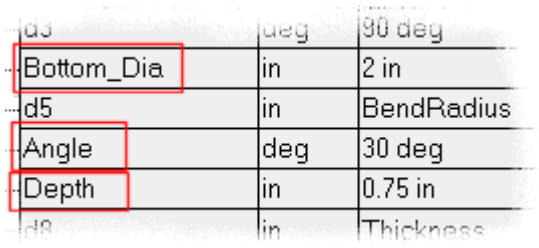

Figure 21: Parameter rename.

1. From the Standard toolbar, click the Parameters tool  .

2. Modify the three parameter names corresponding to the bottom diameter of the dimple, the wall angle, and the depth of the dimple to match those shown in Figure 21.

3. Click Done in the Parameters dialog box.

4. From the Sheet Metal panel bar, click the Create iFeature tool  .

5. In the browser, click Sketch2.

The work plane and the two revolved features are automatically selected. The Size Parameters list contains the renamed parameters, and the sheet metal Thickness and BendRadius parameters. Because all references refer back to the sketch plane for the first feature in the iFeature, the Position Geometry list contains a single entry for the placement plane (see Figure 22).

Tip: If the Position Geometry list contains a Reference Line entry, exit the Create iFeature dialog box and examine the three sketches in the iFeature. One of the sketches will contain an unnecessary projected line. Edit the sketch, delete the line, and repeat the process to create the iFeature.

6. In the Create iFeature dialog box, click Save.

7. Browse to the …Catalog\Punches folder (or your Punches folder if you have changed from the default location).

8. Enter *Dimple01* as the File Name and click Save.

Placing Punch Features

Now that we have a punch defined, we'll place a few dimples on another sheet metal part. Unlike standard iFeatures, Punch features require a sketch for placement. The tool center saved in the iFeature is used to place the punch on hole centers or other points in the sketch.

Note: Unfortunately, this iFeature turned up a defect in the shipping version of Autodesk Inventor 5 that prevents some iFeatures from being placed on faces that are at a different orientation than the positioning plane for the saved iFeature. Look for a fix in a service pack. The following example does not expose the problem.

1. Start a new sheet metal part using the Sheet Metal(in).ipt template.

2. Sketch a rectangle approximately 15-inches square. Create a face from the sketch.

3. Click the near surface of the face and press the S key to start a new sketch.

4. From the Sketch panel bar, click the Offset tool  . Click the outer edge of the face and offset the loop to the inside. Add a 2-inch dimension between the two loops as shown in Figure 23.

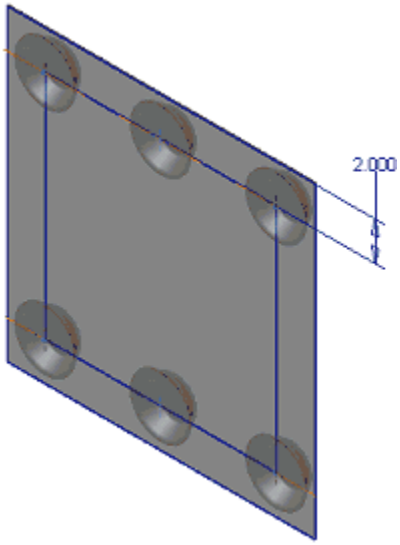

Figure 24: Punch feature preview.

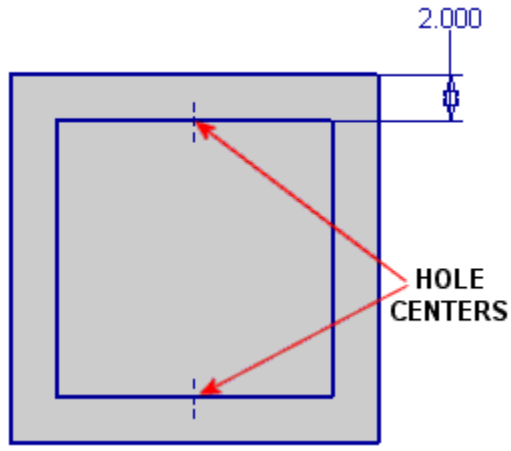

Figure 23: Punch placement sketch.

5. From the Sketch panel bar, click the Point, Hole Center tool  .

Place hole centers at the midpoints of the horizontal lines as shown in Figure 23.

6. Right-click and select Finish Sketch.

7. From the Sheet Metal panel bar, click the PunchTool tool  .

8. In the PunchTool dialog box, select Dimple01.ide from the File Name list and then click Next.

9. The two hole centers are automatically selected and the Centers Select tool remains active.

10. Click the four endpoints of the offset rectangle. The preview should match the one shown in Figure 24.

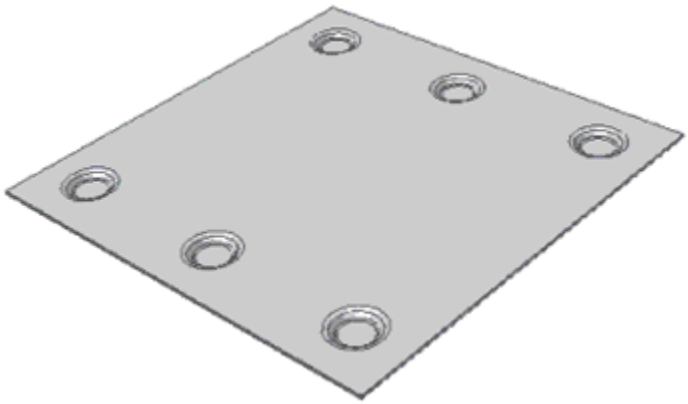

Figure 25: Completed tutorial.

Tip: You can hold down the Shift key and select points to remove them as tool centers. To prevent the automatic selection of all hole centers, make another sketch visible before using the PunchTool tool. When multiple sketches are visible, you must manually select the points (on a single plane) to use as tool centers.

11. Click Next.

12. Enter the following values for the size parameters:

- Depth, 0.25 in
- Angle, 45 deg
- Bottom_Dia, 1.25 in

13. Click OK. The part should match the one shown in Figure 25.

Conclusion

Creating complex iFeatures requires planning to capture design intent. Create work features from common reference geometry to help build multifeature iFeatures and to reduce the reference geometry required when placing a Punch or iFeature. Use the new PunchTool tool in Autodesk Inventor 5 to streamline the process of placing multiple instances of an iFeature.

If you have a specific technique or question that you would like to see covered in a future tutorial, send me an email.