

Exercise - Create Loft Features

In this exercise, you create work planes for loft section profiles. You then use loft options to define the shape of a razor handle.

The completed exercise is shown in the following image:




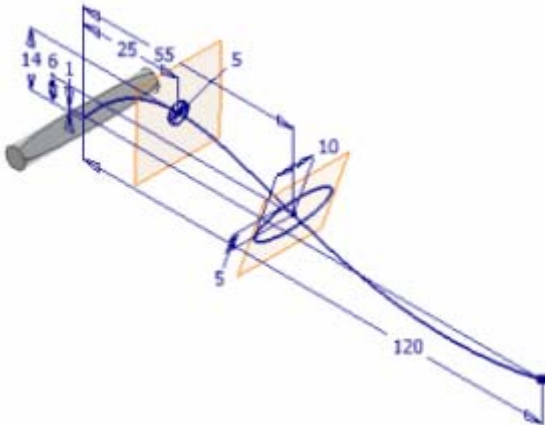
Note: Refer to [Change the Active Project](#) before you start this exercise.




[Try It](#)

To create a sketch point


1.  Open *inv_unit4_04.ipt*.




2. In the browser, double-click Sketch1 to edit the sketch, and then use the Look At tool to set up a view normal to the sketch.

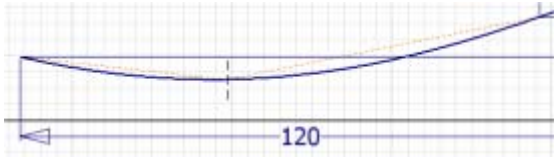
3.  Click the Point, Hole Center tool, and then create a point coincident with the spline near the bottom of the curve (make sure the coincident glyph is displayed as you place the point).



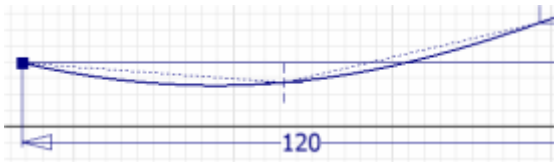
4.  Click the Line tool.

5.  On the Standard toolbar, click Construction.

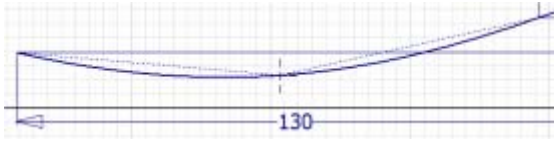
6. Draw construction lines coincident with the sketched point and the spline points on both sides.



7. Place an equal constraint between the construction lines to parametrically position the sketched point midway between the spline points.



8. Change the 120 mm dimension to **130 mm** and verify that the sketched point moves along the spline to maintain its position midway between the spline points.



9. Click Return to finish editing the sketch.

10. Restore the Isometric view.




To create a work plane from a sketched point

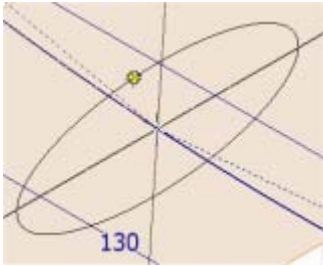
1.  Click the Work Plane tool.

2. Select the sketched point you just created, and then select the spline (at any point that is not close to either construction line) to create the work plane perpendicular to the spline.



To create an ellipse

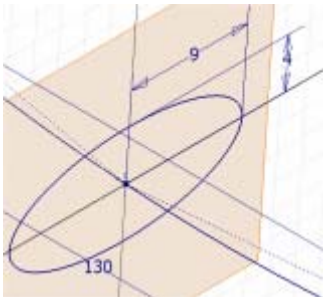
1.  On the Standard toolbar, click Sketch, and then click the work plane.
2.  Click Project Geometry, and then select the sketched point.
3.  Click the down arrow on the Circle tool, click the Ellipse tool, and then draw an ellipse with its center coincident with the projected point and the second point horizontally constrained with the center.




4.  Click General Dimension, and then add a dimension of **9 mm** to the major axis, and **4 mm** to the minor axis.

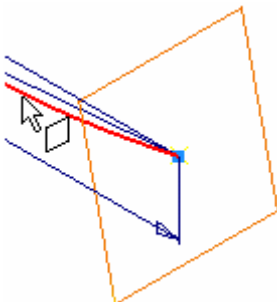
Note: To dimension an axis, click the ellipse and drag the dimension.

5. Click Return to finish the sketch.







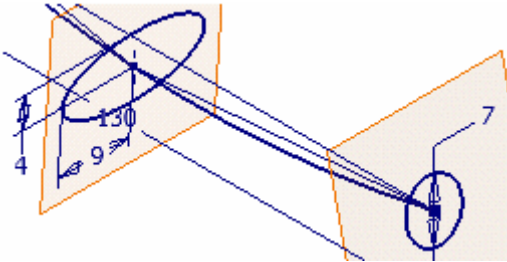
To create a work plane on a spline point

1.  Click the Work Plane tool.
2. Select the spline's right endpoint, and then select the spline to create the work plane perpendicular to the spline curve.




To create a circular loft profile section

1.  On the Standard toolbar, click Sketch, and then click the work plane.
2.  Click Project Geometry, and then select the sketched point.
3.  Click Center Point Circle, and then draw a circle coincident with the projected point.
4.  Click General Dimension, and then create a **7 mm** dimension to control the diameter.

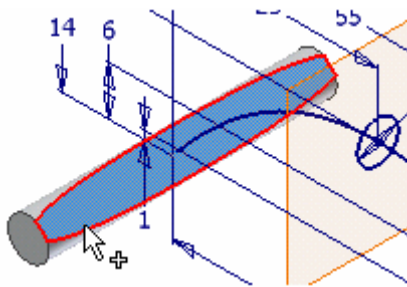


5. Click Return to finish the sketch.

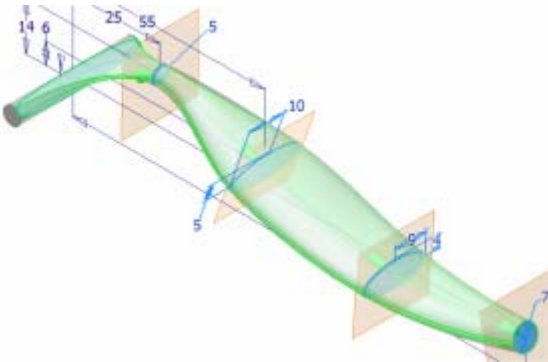
To create a loft

1.  Click the Loft tool.

- For the first section, select the concave 3D face as shown in the following image:




- Select the other four profile sections in order, and then click OK when the preview looks like the following image:



- In the browser, turn off the visibility of Sketch1 and all four work planes, and then use the Rotate and other view tools to examine the loft feature.



5.  Save the file using the class file-naming convention.

You created a loft feature through section profiles.

Exercise - Create Emboss and Engrave Features

In this exercise, you emboss and engrave sketched profile objects on faces of a razor handle model. You then create a sketch text object, and engrave it on the handle.

The completed exercise is shown in the following figure:




Note: Refer to [Change the Active Project](#) before you start this exercise.



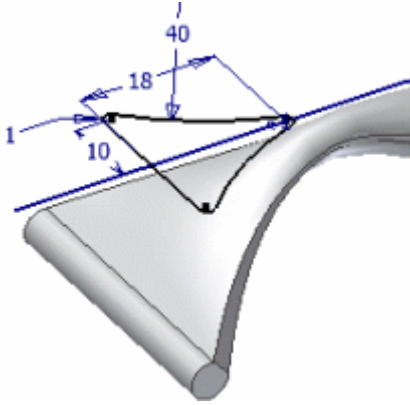
[Try It](#)


To engrave a profile

1.  Open *inv_unit4_05.ipt*.

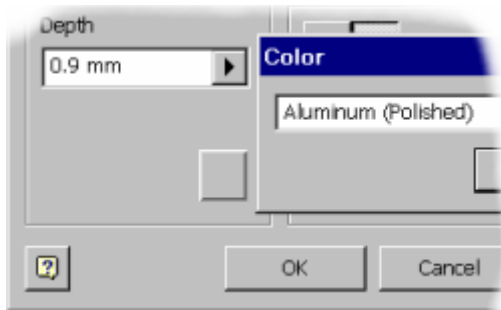


2. Use the Rotate and Zoom tools to examine the part, and then turn on visibility of Sketch11 and rotate the view to display the top triangular face.



3.  Click the Emboss tool.

- The profile is automatically selected.
- In the Emboss dialog box, select the Engrave from Face option.
- In Depth enter **0.9 mm**.
- Click the Top Face Color button and choose Aluminum (Polished) from the Color dialog box drop-down list.

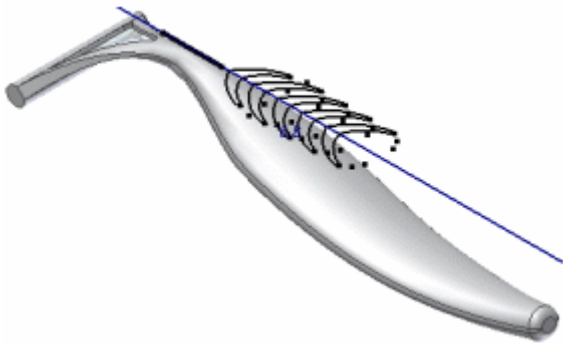


4. Click OK twice to close both dialog boxes and create the engraved feature.



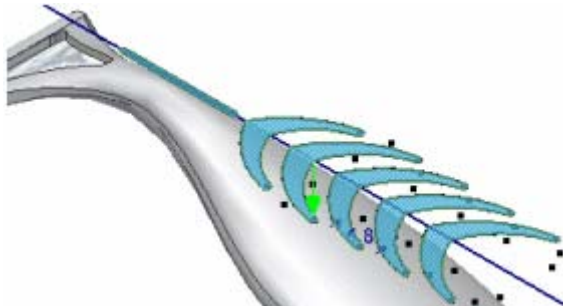
To emboss a profile

1. Restore the Isometric View and turn on the visibility of Sketch10.



2.  Click the Emboss tool.

- Select all six closed profiles. Be sure to select the top and bottom half of the first two herringbone profiles (on the left).

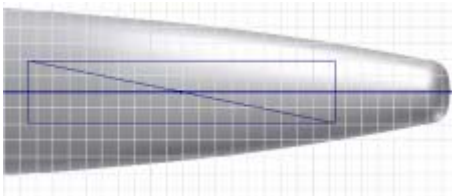


- In the Emboss dialog box, make sure the Emboss from Face option is selected, and then change the Depth to **0.2 mm**.
- Click the Top Face Color button and choose Black from the Color dialog box drop-down list.
- Click OK twice to close both dialog boxes and create the embossed feature.
- Use the Rotate tool to examine the engraved and embossed features.



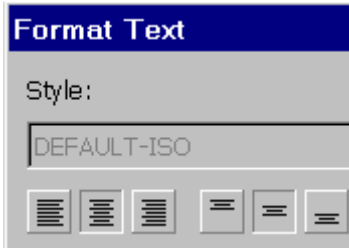
To create sketch text

1. Turn on the visibility of Sketch9, and then double-click Sketch9 to edit.
2. Use the Look At tool to set up a view normal to the sketch.



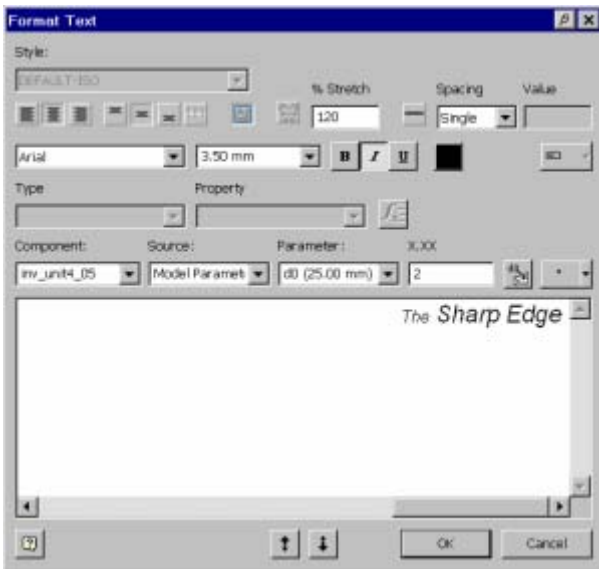
3.  Click the Create Text tool.

- Click in an open area below the part and under the left edge of the construction rectangle to specify the insertion point and display the Format Text dialog box.
- Select the Center and Middle Justification buttons., and then select the Italic option. Change the % Stretch value to **120**.

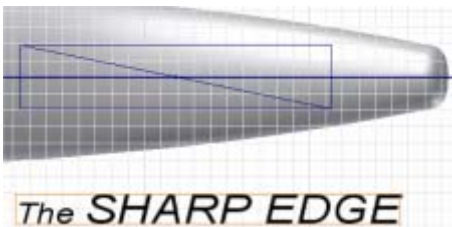


- Select the Italic option.
- Change the % Stretch value to **120**.
- Click in the text field and enter **The SHARP EDGE**.

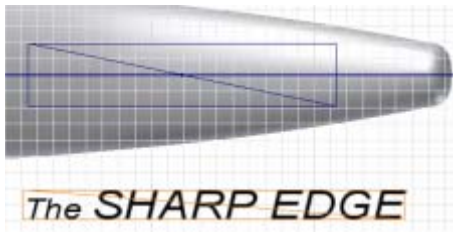
4. Double-click *The* in the text field and change the text size to **2.5 mm**. Press TAB to update the preview.



5. Click OK to place the text.




6. Draw a diagonal construction line coincident with the top left corner and bottom right corner of the text bounding-box. Place a coincident constraint between the midpoint of diagonal construction lines on the text and the razor body.




7. Click the Return button to finish editing the sketch and then restore the Isometric View.

To engrave a sketch object

1.  Click the Emboss tool.
 - For the profile, select the text object.
 - In the Emboss dialog box, select the Engrave from Face option, and then change the Depth to **0.5 mm**.
 - If necessary, click Flip to set the direction arrow downward.
 - Click the Top Face Color button and choose Black from the Color dialog box drop-down list.
 - Click OK twice to close both dialog boxes and create the engraved feature.
2. Use the Rotate tool to examine the engraved and embossed features.



3.  Save the file using the class file-naming convention.

You embossed and engraved features onto a razor.