

TUTORIAL 7

3D Sketch Intersections

Learning Objectives

After completing this tutorial, you will be able to:

- Construct and use surface features in solid modeling
- Incorporate surface features in part design when appropriate
- Construct complex Sweep paths
- Find 3D Sketch intersections

Required Competencies

Before starting this tutorial, you should have been able to:

- Construct, constraint and dimension sketches
- Project geometry on sketch planes
- Sweep sketched profiles
- Understand the concepts of work and placed features
- Understand how to manipulate the history tree in the browser

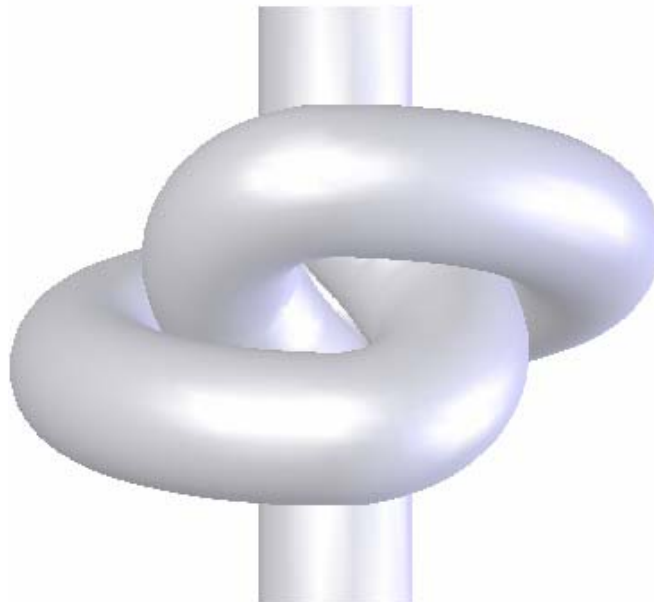


Figure 1: Knot

The model in **Figure 1** is created by finding a 3D sketch path intersection between a surface and a sketch.

The **Design Intent** is to create a complex path – in this case a knot – to demonstrate that we should be able to create any desired path for a length of tubing. There are many ways to create tubing paths, using intersecting surfaces is just one method. The method presented in this tutorial is applicable to complex sweep problems other than tubing.

1. Open the file named **Knot.SLDPRT** and roll up to Sketch1. Extrude Sketch1 as a surface mid-plane a distance of 60.

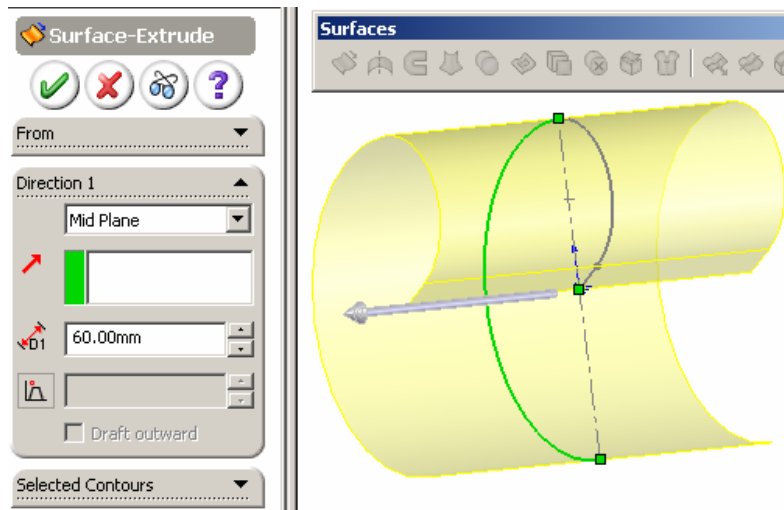


Figure 2

2. Show Sketch2 and from the Curve drop-down on the Surfaces toolbar select Project Curve.

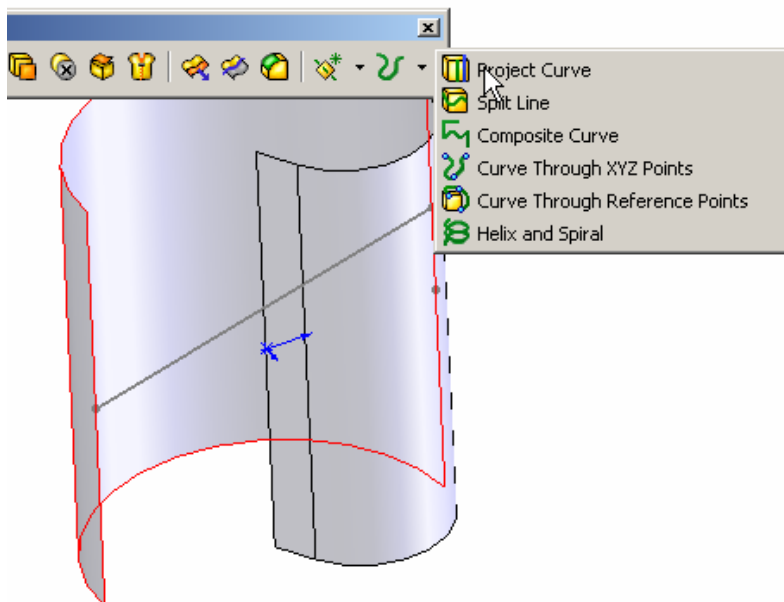


Figure 3

3. Select Sketch onto Face(s).

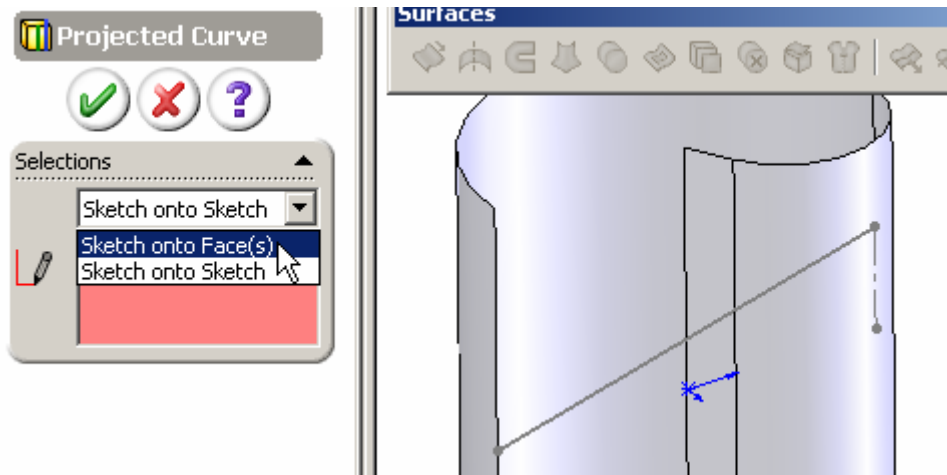


Figure 4

4. Select Sketch2 and the face shown and enter a checkmark in the Reverse Projection window. You should see a preview of the projection.

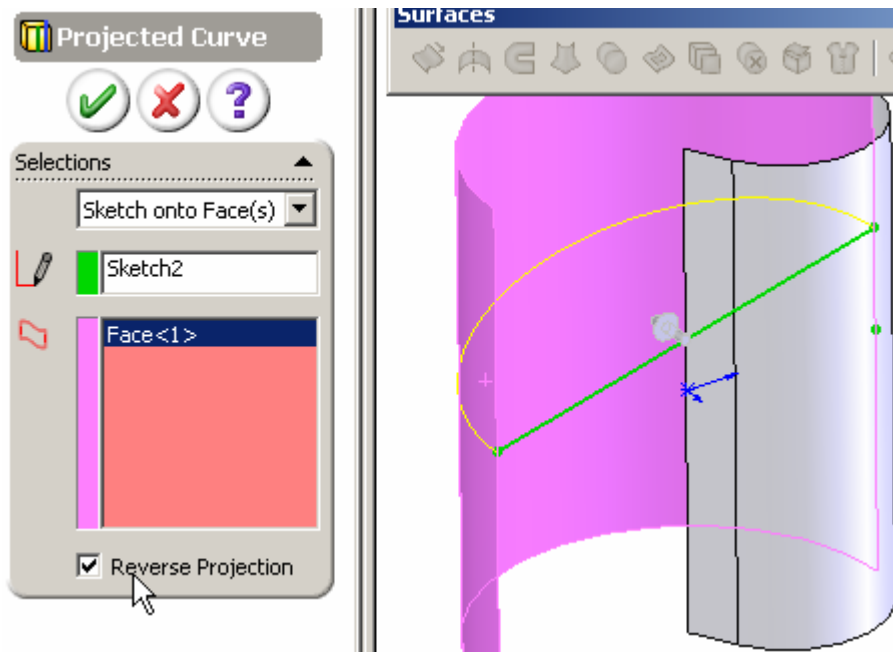


Figure 5

5. Show Sketch3 and repeat the Project Curve command selecting two faces as shown. Hide the surface.

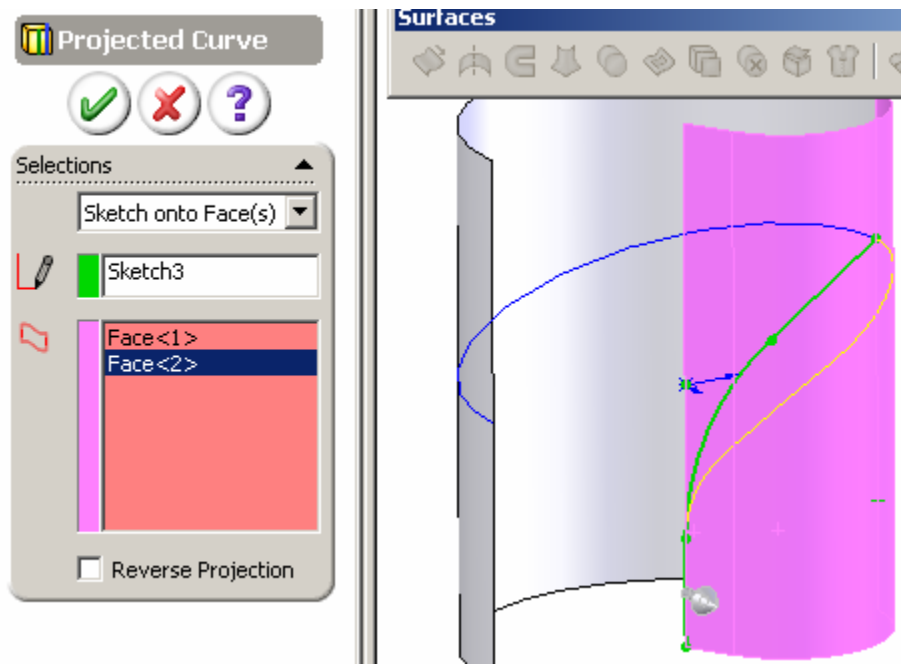


Figure 6

6. Select the Composite Curve command from the Surfaces toolbar and click on the two 3D curves to join.

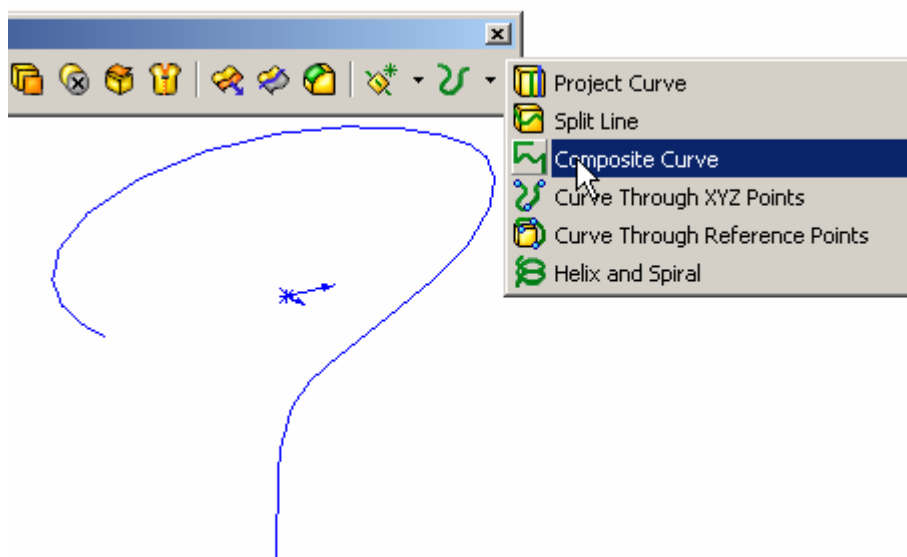


Figure 7

7. Show Sketch4 and Sweep the sketch along the composite curve as the path.

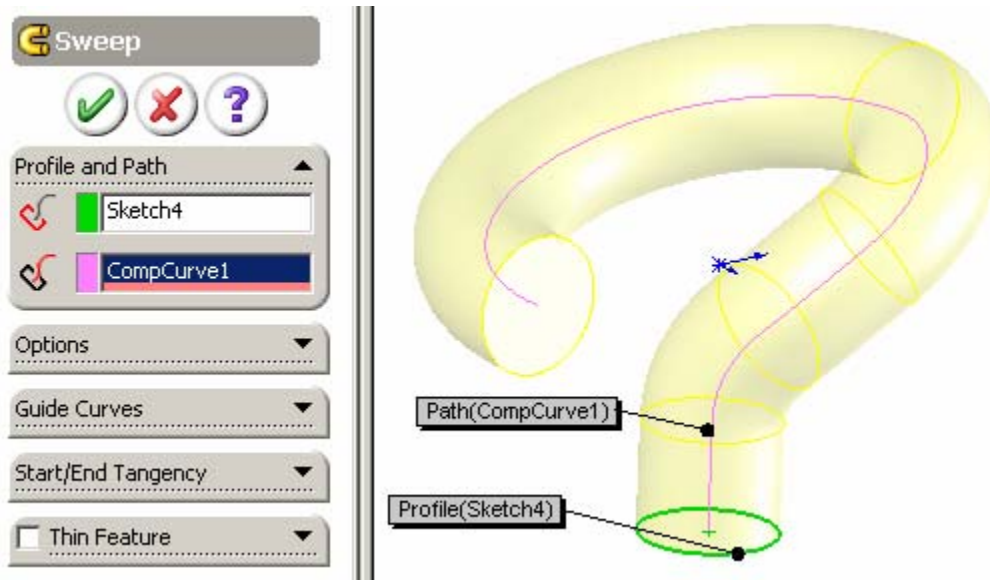


Figure 8

8. Show Sketch1 again and create an Axis on the centerline. Hide the sketch.

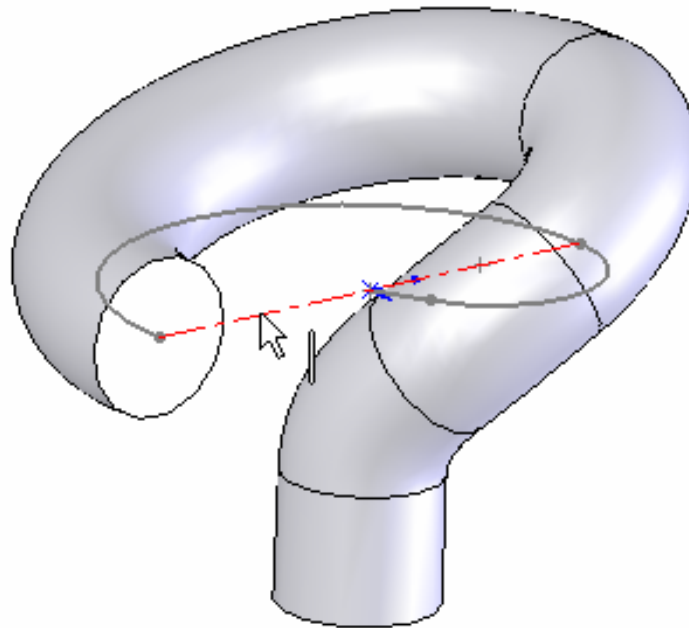


Figure 9

9. Start the Circular Pattern command and select the Axis and Feature to Pattern with 2 copies in 360deg.

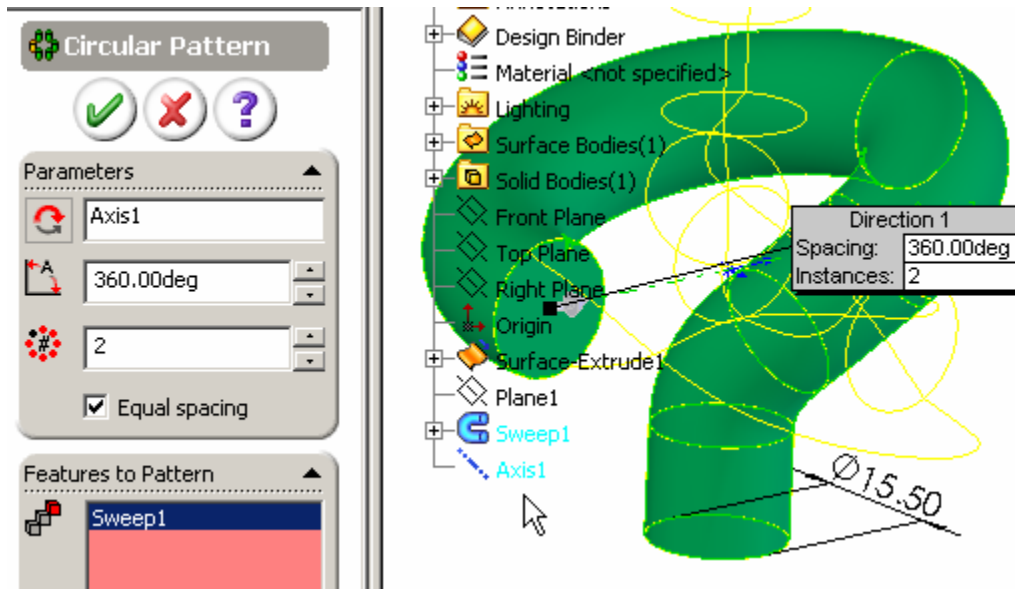


Figure 10

10. If you wanted a tube you could Shell the part.

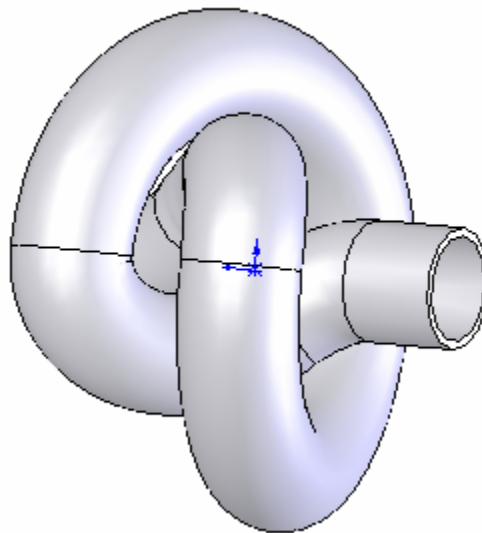


Figure11

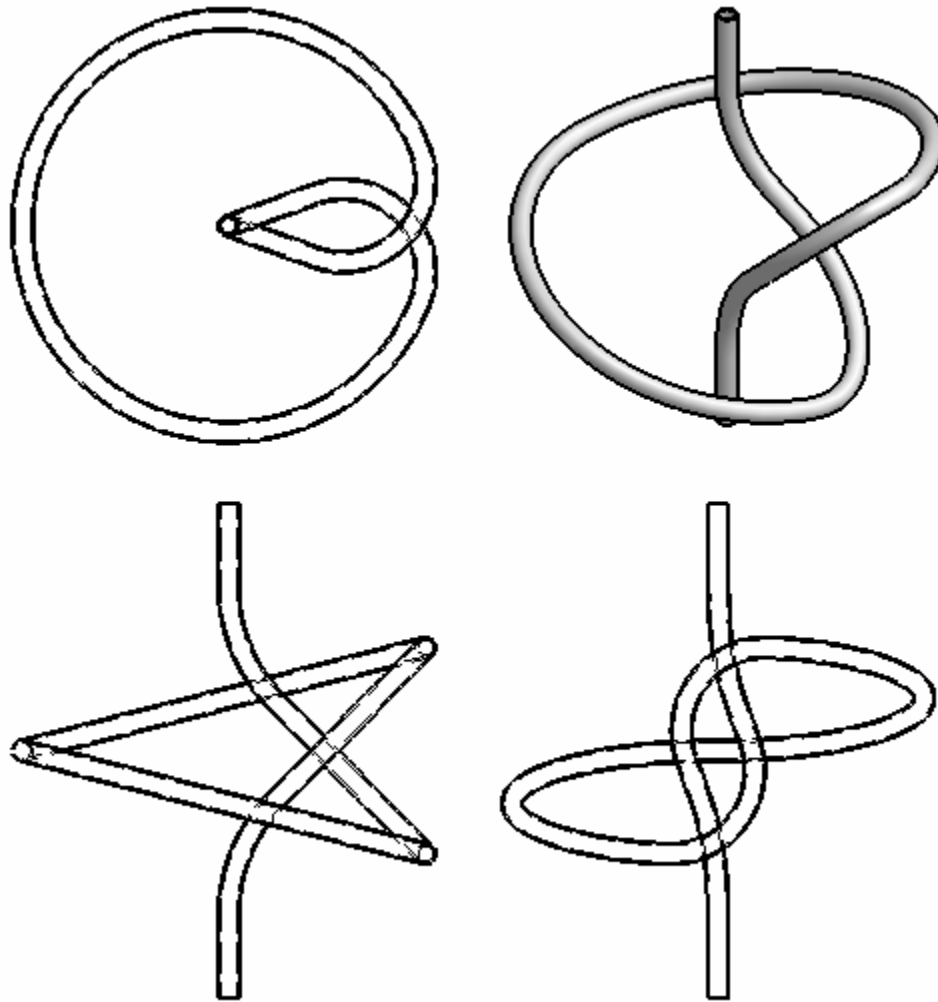


Figure 12

In this tutorial we learned how to create a complex 3D sweep path using the intersection of multiple surfaces or 3D sketch intersections. Compare the sketches used to create these surfaces with the top and front orthographic views in Figure 12. The surfaces were simply the centerlines (from half of the view as they are symmetrical) of these projected views. By analyzing the path in this manner we see that the problem is not so complex after all.