

6 - Create Surfaces from Curves

One way of working in 3-D is to draw curves that represent edges, profiles, cross-sections, or other surface features and then to use surfacing commands to create surfaces from those curves. Curves can be free-standing curves or the edges of existing surfaces.

Surface from edge curves

You can create a surface from three or four curves that define the edges of the surface.



Create a surface from edge curves

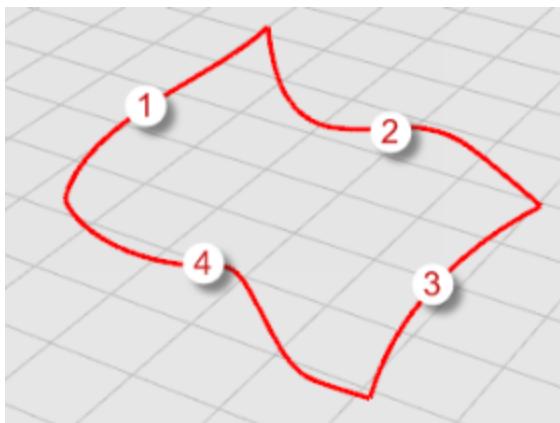
1. Open the tutorial model **EdgeSrf.3dm**.

(Help menu > Learn Rhino > Tutorials and Samples > User's Guide > EdgeSrf)

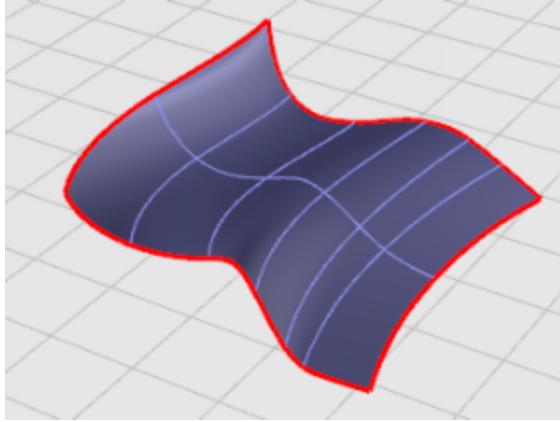
2. On the **Surface** menu, click **Edge Curves**.

Optional: Press **F1** to review the Help topic for the EdgeSrf command.

3. At the **Select 2, 3, or 4 open curves** prompt, select the four curves as shown in the illustration below.



To select the curves, you can click each individual curve using the **Shift** key, or you can use a window or crossing selection to select all of the curves at once. To review selecting objects, see [Chapter 3, Selecting Objects](#). A surface is created from the curves, which define its edges.



Extrude curves

Surfaces can be created by extruding curves in a straight line perpendicular to the construction plane.



Create an extruded surface

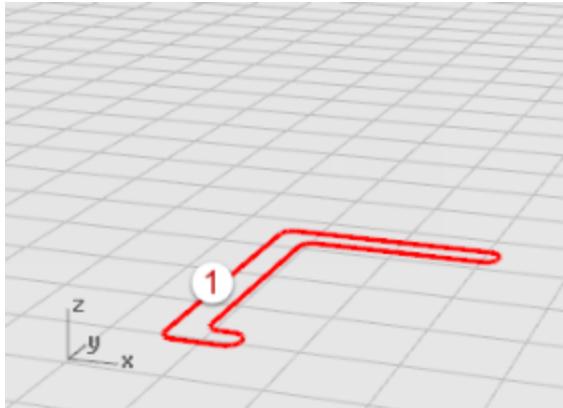
1. Open the tutorial model **ExtrudeCrv.3dm**.

(Help menu > Learn Rhino > Tutorials and Samples > User's Guide > Extrude)

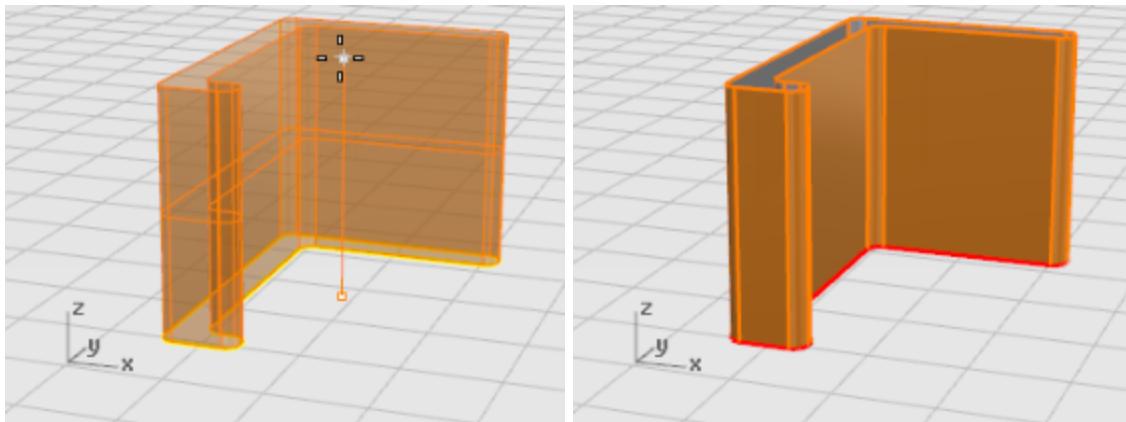
2. On the **Surface** menu, click **Extrude Curve**, and then click **Straight**.

Optional: Press **F1** to review the Help topic for the ExtrudeCrv command.

3. At the **Select curves to extrude** prompt, select the curve (1) as shown in the illustration below.



4. At the **Extrusion distance** prompt, drag a distance with your mouse and click.



Loft curves

Lofting creates a smooth surface that blends between selected curves. The curves define "ribs" for the surface you are creating.



Create a lofted surface

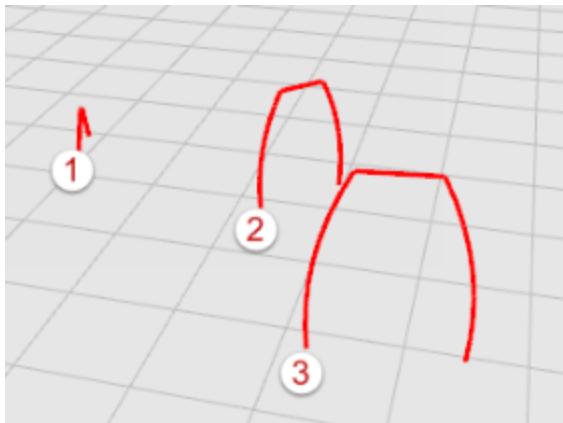
1. Open the tutorial model **Loft.3dm**.

(Help menu > Learn Rhino > Tutorials and Samples > User's Guide > Loft)

2. On the **Surface** menu, click **Loft**.

Optional: Press **F1** to review the Help topic for the Loft command.

3. At the **Select curves to loft** prompt, select the three curves (1), (2), and (3) as shown in the illustration below, and press **Enter**.

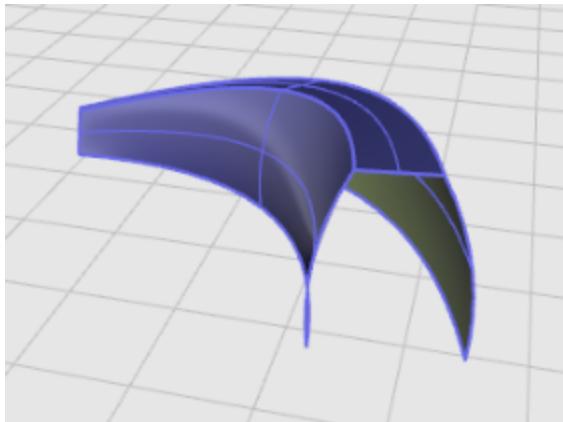


Tip: Select the curves by picking near the same end where the numbers are. This avoids twisting the loft surface. In this model, you can use any selection method. The Loft command will sort out the order of the curves. In other more complex cases, picking the curves near the same end may be important, and you may have to make adjustments for the loft to work properly.

4. in the **Loft** dialog box, click the drop-down menu under **Style** to try some of the options.

The default style is **Normal**. As you change the style, the preview will update in real time to show you what the results will be.

5. In the **Loft Options** dialog box, click **OK**.



Revolve curves

Revolving creates a surface by revolving curves about a *revolve axis*. The curve defines the silhouette of the revolved surface. This action is sometimes called *lathing* in other programs.

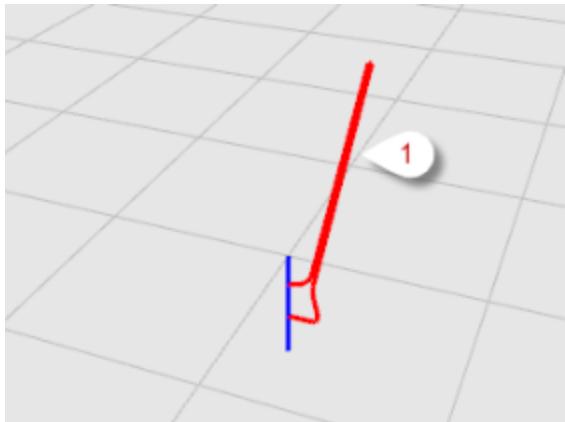


Create a revolved surface

1. Open the tutorial model **Revolve.3dm**.
(Help menu > Learn Rhino > Tutorials and Samples > User's Guide > Revolve)
2. In the **Status bar**, click the **Osnap** pane to turn on the Osnap control.
3. In the **Osnap** control, click **End**.
4. On the **Surface** menu, click **Revolve**.

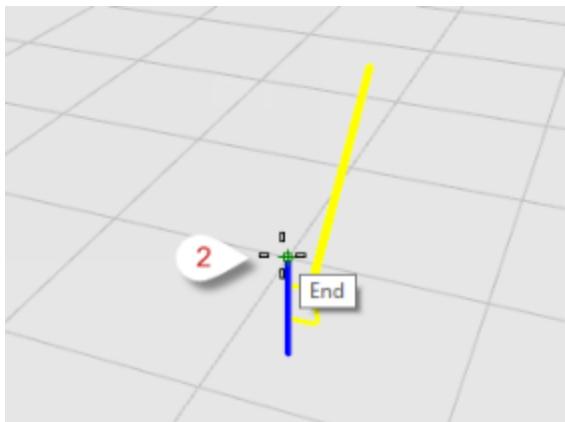
Optional: Press **F1** to review the Help topic for the Revolve command.

5. At the **Select curves to revolve** prompt, select the curve (1) and press **Enter**.

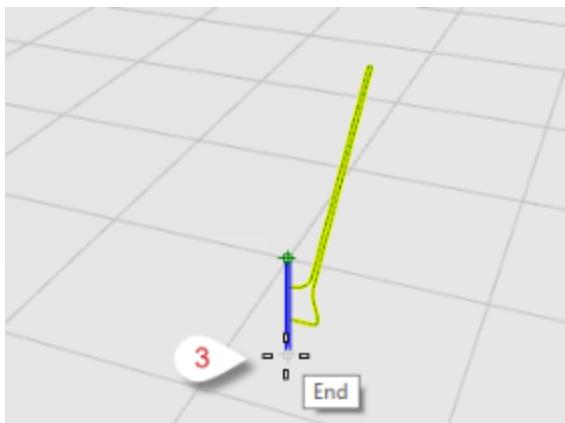


6. At the **Start of revolve axis** prompt, snap to one end of the line (2) that defines the revolve axis.

Note: A reference line is not strictly necessary; you can pick any two points to define the axis.

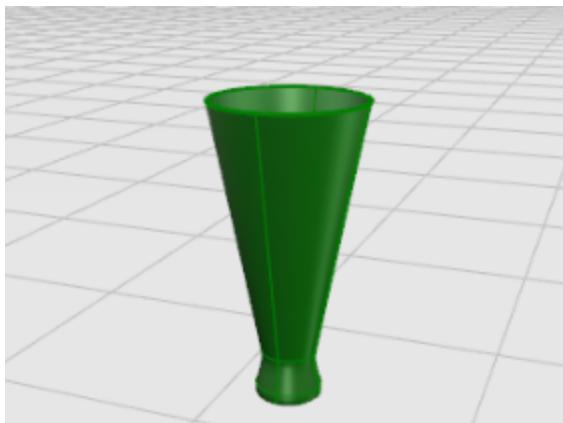


7. At the **End of revolve axis** prompt, snap to the other end of the line (3) that defines the revolve axis.



8. At the **Start angle...** prompt, click the **FullCircle** option.

The FullCircle option creates a complete 360-degree surface.



Revolve curves with a rail

Rail revolve creates a surface by revolving a *profile curve* around a reference axis while at the same time following a *rail curve* as closely as possible. The *profile curve* defines the silhouette of the surface, and the *rail curve* defines a path the profile curve will try to follow.



Create a revolved surface with a rail curve

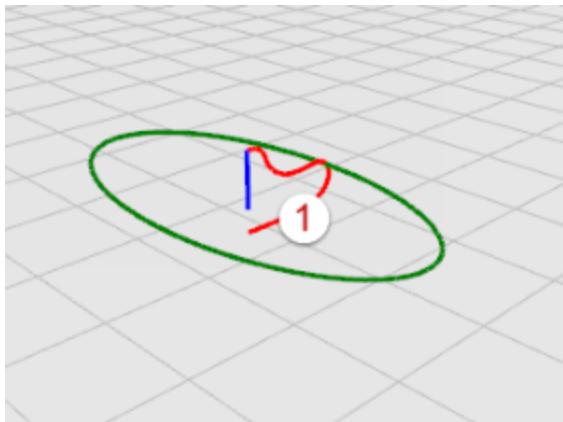
1. Open the tutorial model **RailRevolve.3dm**.

(Help menu > Learn Rhino > Tutorials and Samples > User's Guide > RailRevolve)

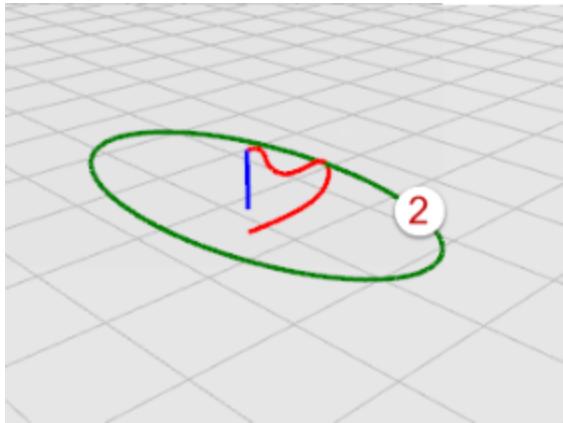
2. On the **Surface** menu, click **Rail Revolve**.

Optional: Press **F1** to review the Help topic for the RailRevolve command.

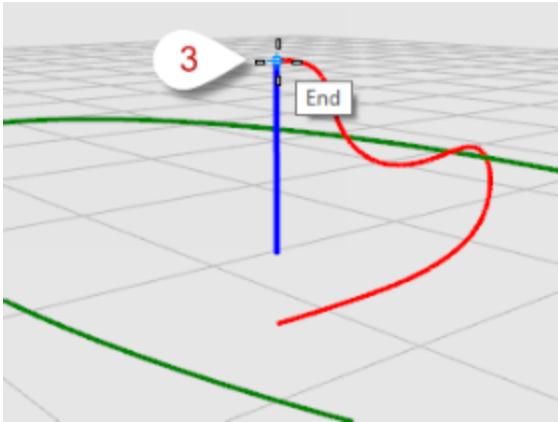
3. At the **Select profile curve** prompt, select the profile curve (1).



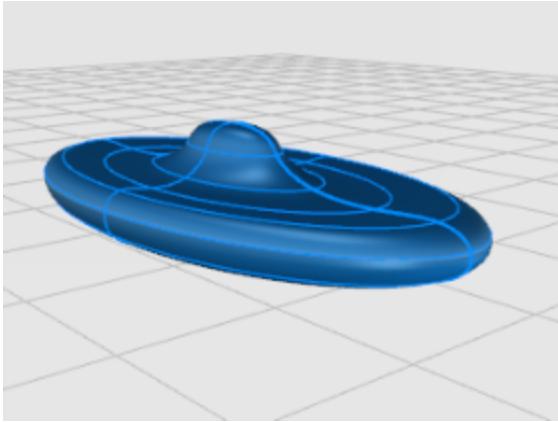
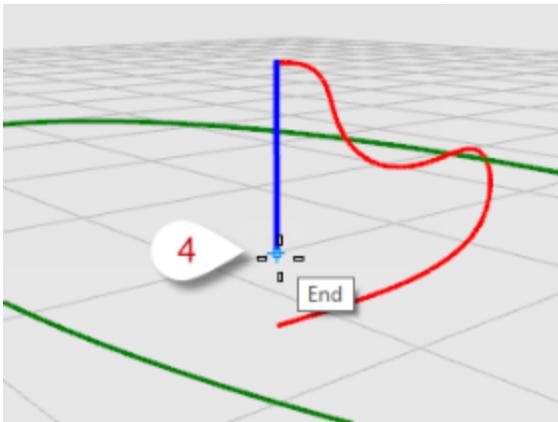
4. At the **Select rail curve...** prompt, select the rail curve the revolve will follow (2).



5. At the **Start of RailRevolve axis** prompt, snap to an endpoint of the axis line (3).



6. At the **End of RailRevolve axis** prompt, snap to the other end of the axis line (4).



Sweep along one rail curve

Sweeping with one rail creates a smooth surface with from multiple *cross-section curves* and a single *rail* curve. The *cross-section curves* define the surface shape, and the *rail* curve defines a path the surface will follow.



Create a sweep surface with one rail

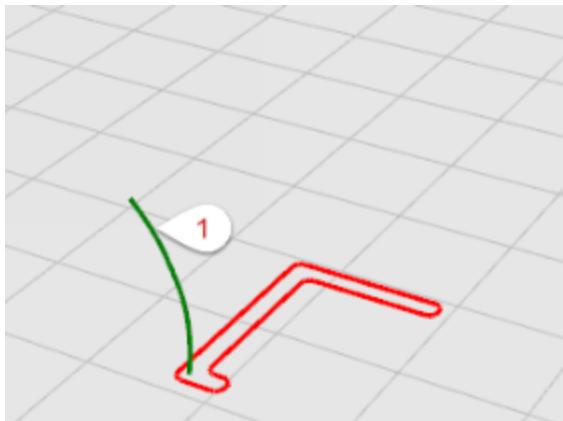
1. Open the tutorial model **Sweep1.3dm**.

(Help menu > Learn Rhino > Tutorials and Samples > User's Guide > Sweep1)

2. On the **Surface** menu, click **Sweep 1 Rail**.

Optional: Press **F1** to review the Help topic for the Sweep1 command.

3. At the **Select rail** prompt, select the rail curve (1).

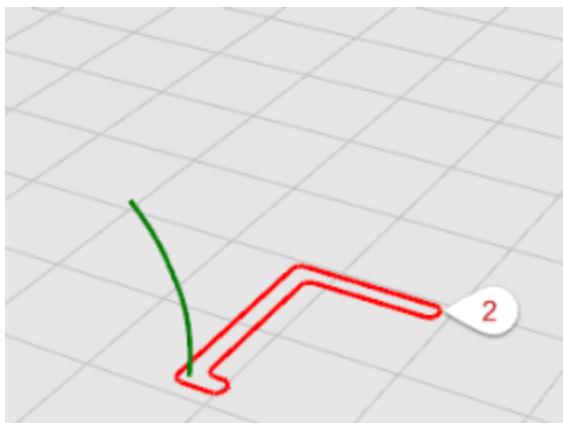


4. At the **Select cross section curves...** prompt, select the cross-section curve (2), and press **Enter**.

Only one cross-section curve is used in this example.

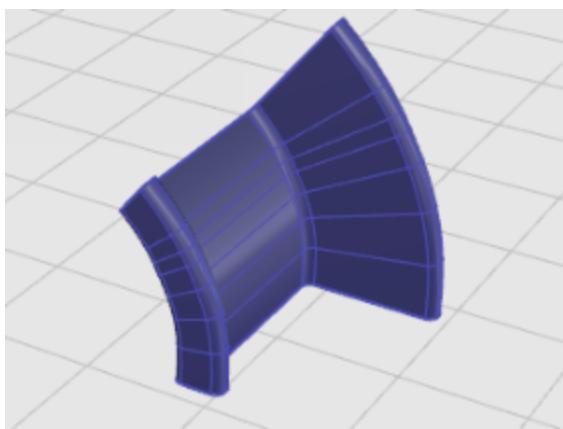
5. At the **Drag seam point to adjust...** prompt, press **Enter**.

No adjustment is necessary in this case.



6. In the **Sweep 1 Rail Options** dialog box, click **OK**.

The default Freeform option will be used.



Sweep along two rail curves

Sweeping with two rails creates a smooth surface from multiple *cross-section curves* and two *rail curves*. The *cross-section curves* define the surface shape, and the *rail curves* define the path the two edges of the surface will follow as closely as possible. Use the Sweep2 command when you want to control the location of the edges of the surface; for example, when using the edges of other surfaces as the rail curves.



Create a sweep surface with two rail curves

1. Open the tutorial model **Sweep2.3dm**.

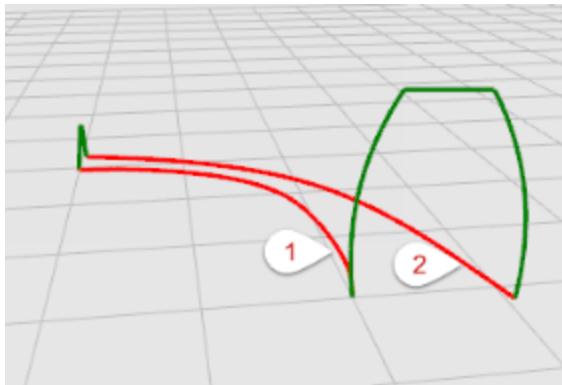
(Help menu > Learn Rhino > Tutorials and Samples > User's Guide > Sweep2)

2. On the **Surface** menu, click **Sweep 2 Rails**.

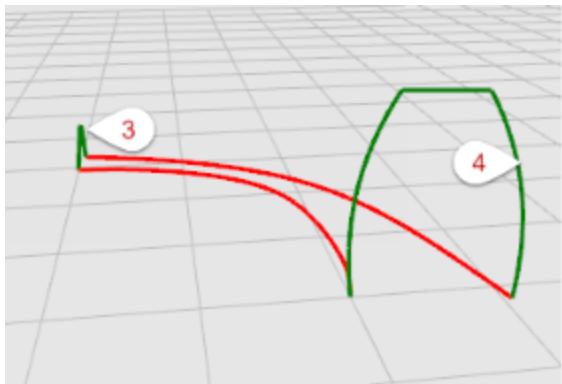
Optional: Press **F1** to review the Help topic for the Sweep2 command.

3. At the **Select first rail...** prompt, select the first rail curve (1).

4. At the **Select second rail...** prompt, select the second rail curve (2).



5. At the **Select cross section curves** prompt, select the two cross-section curves (3) and (4), and press **Enter**.



6. In the **Sweep 2 Rails Options** dialog box, click **OK**.

No further adjustments are needed in this case.

