

7 - Edit Curves and Surfaces

The editing operations in this section break objects apart, cut holes in them, and put them back together. Some of these commands join curves to curves, or surfaces to surfaces or polysurfaces, and break a composite curve or polysurface into its components.

The commands: **Join**, **Explode**, **Trim**, and **Split** apply to curves, surfaces, and polysurfaces.



Join

The **Join** command connects curves or surfaces together into one object.

Join turns lines that touch end to end into polylines, curves that touch end to end into polycurves, and surfaces and polysurfaces that touch on an edge into polysurfaces or solids.

When a curve is selected first, surface edges or mesh edges can be joined with the curve to create a polyline or polycurve.



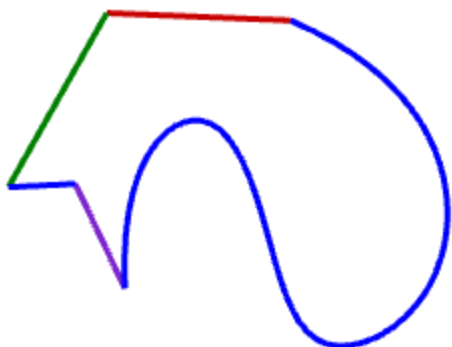
Join curves into a polycurve

1. Open the tutorial model **Join Curves.3dm**
(*Help menu > Learn Rhino > Tutorials and Samples > User's Guide > Join Curves*)

2. **Select** all of the curves using a window or crossing.

The command history displays **5 curves added to selection**.

The **Object Type** in the **Properties** panel shows **5 open curves**.



3. On the **Edit** menu, click **Join**.

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

The command history displays **5 curves joined into one closed curve**.

The **Object Type** in the **Properties** panel shows **closed curve**.





Join surfaces into a polysurface

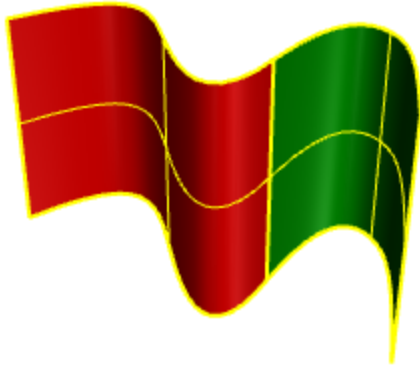
1. Open the tutorial model **Join Surfaces.3dm**.

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

2. **Select** both of the surfaces using a window or crossing.

The command history displays **2 surfaces added to selection**.

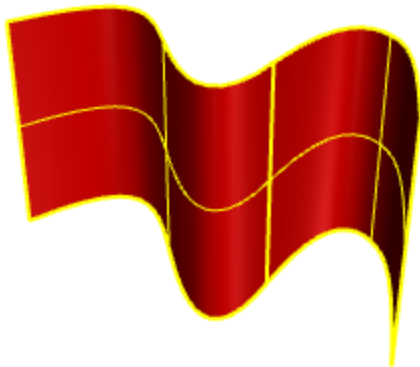
The **Object Type** in the **Properties** panel shows **2 surfaces**.



3. On the **Edit** menu, click **Join**.

The command history displays **2 surfaces or polysurfaces joined into one open polysurface**.

The **Object Type** in the **Properties** panel shows **open polysurface**.



Join surfaces into a solid polysurface

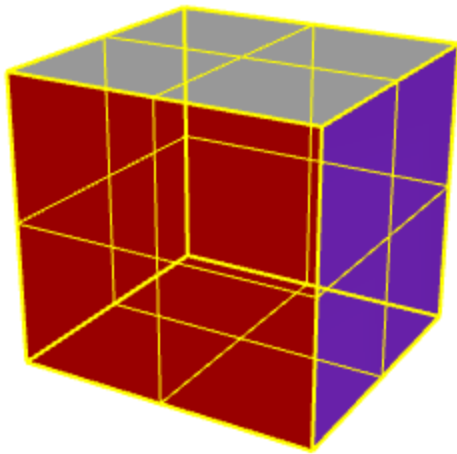
1. Open the tutorial model **Join Solid.3dm**.

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

2. **Select** all of the surfaces using a window or crossing.

The command history displays **6 surfaces added to selection**

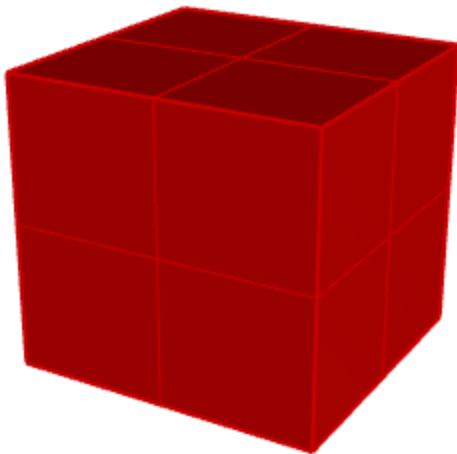
The **Object Type** in the **Properties** panel shows **6 surfaces**.



3. On the **Edit** menu, click **Join**.

The command history displays **6 surfaces or polysurfaces joined into one closed polysurface**.

The **Object Type** in the **Properties** panel shows **closed solid polysurface**.



Explode

The **Explode** command removes the connection between joined curves and polysurfaces. Explode works on any object that can be reduced to its component parts.



Explode a polycurve into individual curves

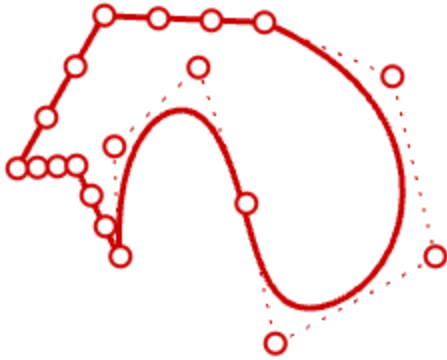
1. Open the tutorial model **Explode Curves.3dm**.

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

2. **Select** the curve.

The command history displays **1 curve added to selection**.

The **Object Type** in the **Properties** panel shows **closed curve**.



3. On the **Edit** menu, click **Explode**.

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

The command history displays **Exploded a curve into 5 segments**.

The **Object Type** in the **Properties** panel shows **5 open curves**.



Note: For the illustration, the **ScalePositions** command was used to separate the individual curves. The **Explode** command does not move the curves.



Explode a polysurface into individual surfaces

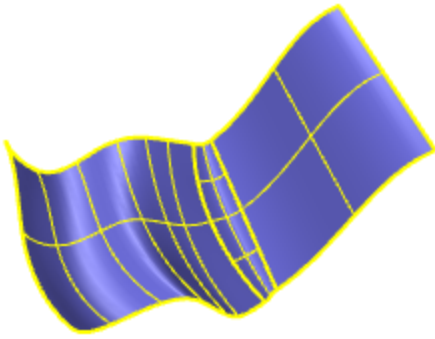
1. Open the tutorial model **Explode Polysurface.3dm**.

(*Help menu > Learn Rhino > Tutorials and Samples > User's Guide > Explode Polysurface*)

2. **Select** the polysurface.

The command history displays **1 polysurface added to selection**.

The **Object Type** in the **Properties** panel shows **open polysurface**.

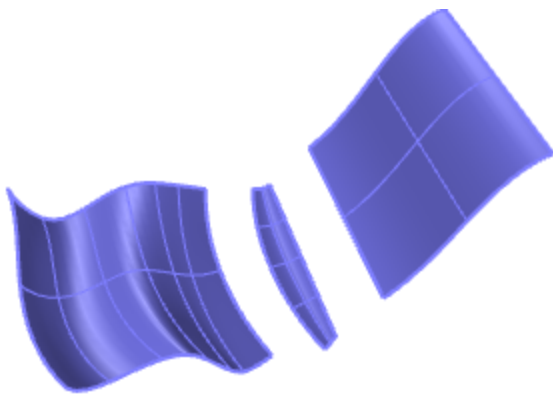


3. On the **Edit** menu, click **Explode**.

The command history displays **Exploded a polysurface into 3 surfaces..**

The **Object Type** in the **Properties** panel shows **varies**.

Three surfaces were created, two surfaces and a trimmed surface. Select each surface individually to check the object type.



Note: For the illustration, the **ScalePositions** command was used to separate the individual surfaces. The **Explode** command does not move the exploded surfaces.

Trimming and Splitting

The **Trim** and **Split** commands are similar. They both cut objects with other objects. When you trim an object, you select the parts to remove and they are deleted. When you split an object, no parts are deleted.



Trim

The Trim command cuts and deletes selected portions of an object at the intersection with another object used as a cutter.



Trim curves

1. Open the tutorial model **Trim Curves.3dm**.

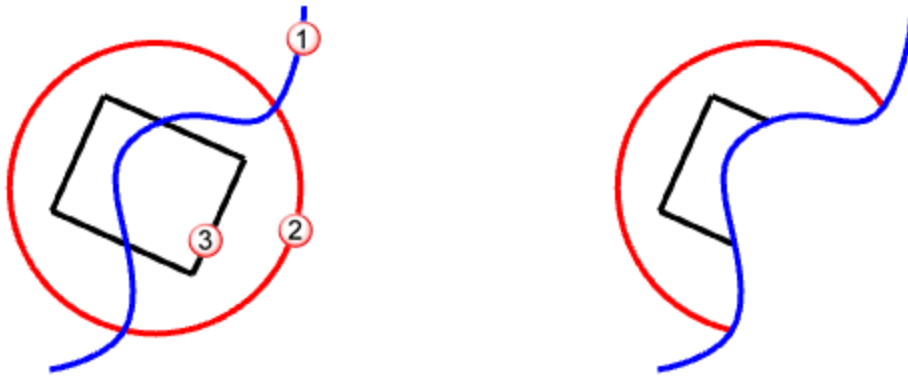
(Help menu > Learn Rhino > Tutorials and Samples > User's Guide > Trim Curves)

2. On the **Edit** menu, click **Trim**.

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

3. At the **Select cutting objects** prompt, select the blue curve (1) as illustrated below, and press **Enter**.

4. At the **Select object to trim** prompt, select the right side of the red circle (2) and the black rectangle (3), and press **Enter**.

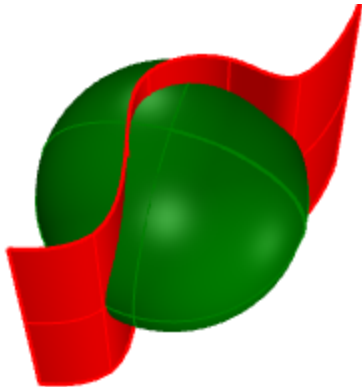


The circle and the rectangle are trimmed by the blue curve.

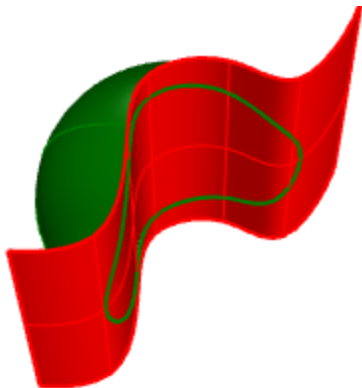


Trim surfaces

1. Open the tutorial model **Trim Surfaces.3dm**.
(Help menu > Learn Rhino > Tutorials and Samples > User's Guide > Trim Surfaces)
2. On the **Edit** menu, click **Trim**.
3. At the **Select cutting objects** prompt, select the green sphere as illustrated below, and press **Enter**.



4. At the **Select object to trim** prompt, select the right side of the red circle and the black rectangle, and press **Enter**.



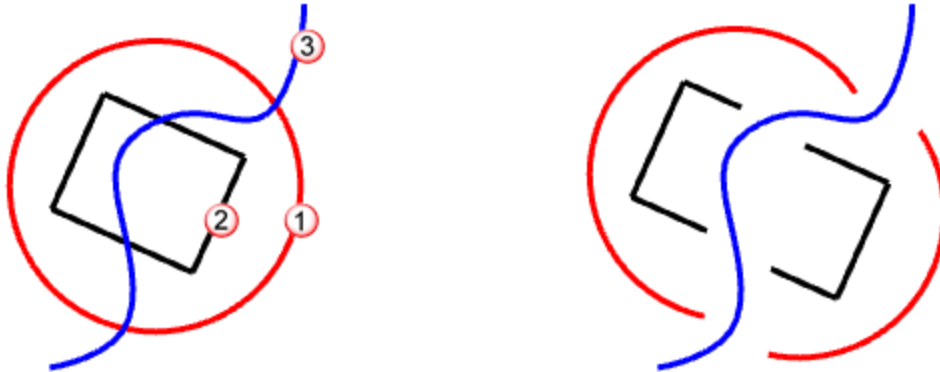
Split

The Split command divides objects into parts using other objects as cutters.



Split curves

1. Open the tutorial model **Split Curves.3dm**.
(Help menu > Learn Rhino > Tutorials and Samples > User's Guide > Split Curves)
2. On the **Edit** menu, click **Split**.
Optional: Press **F1** to review the Help topic for the Split command.
3. At the **Select object to split...** prompt, select the red circle (1) and the black rectangle (2), and press **Enter**.
4. At the **Select cutting objects** prompt, select the blue curve (3) as illustrated below, and press **Enter**.



The red circle and the black rectangle are split by the blue curve.

Note: For the illustration, the individual curves were moved. The Split command does not move the split objects.



Split surfaces

1. Open the tutorial model **Split Surfaces.3dm**.
(Help menu > Learn Rhino > Tutorials and Samples > User's Guide > Split Surfaces)
2. **Select** the green sphere.



Note: This is an example of pre-picking input for a command. You can select the input for the first prompt of most commands.

3. On the **Edit** menu, click **Split**.

- At the **Select cutting objects** prompt, select the red surface, and press **Enter**.



The sphere is split by the red surface.

Note: For the illustration, the split surface was moved. The *Split* command does not move the split objects.



Untrim

The Untrim command removes trim curves from a surface restoring the underlying untrimmed object.



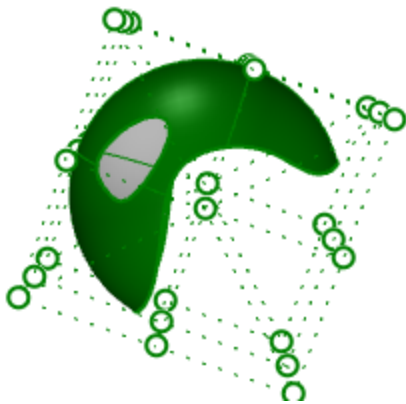
Untrim surfaces

- Open the tutorial model **Untrim Surfaces.3dm**.

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

- Press **F10** to turn on the control points for the green surface.

The control point array shows that this surface is actually a trimmed sphere. When the trim curves are removed, the surface fills in to reform the original sphere.



- On the **Surface** menu, click **Surface Edit Tools > Untrim**.

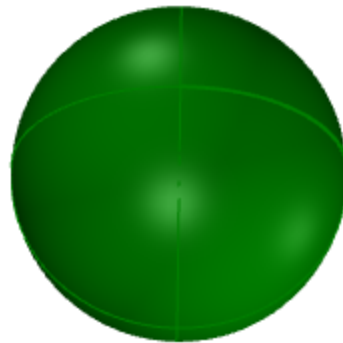
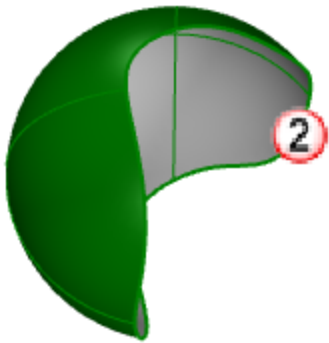
Optional: Press **F1** to review the Help topic for the Untrim command.

4. At the **Select edge to untrim...** prompt, select the edge of the hole (1) as illustrated below.



The hole edge is deleted causing the hole to be removed.

5. At the next **Select edge to untrim...** prompt, select the edge of the surface (2) as illustrated below.



The edge is deleted causing the sphere to return to its full untrimmed state.