

From this point forward, our goal is to create SOLID objects. Generally, will do that by creating surfaces and then giving them thickness. Some of the commands listed here can only make open surfaces, while others can make solids if you are careful to set up your input objects so that any gaps or openings will be closed when you complete the command.

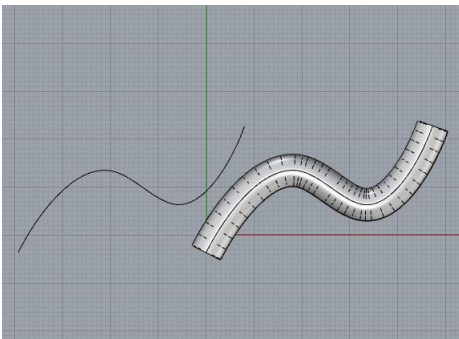
Terms:

Surfaces: 2D objects that have no thickness. They are often defined by curves that you have created, and surfaces can be joined together to become solids.

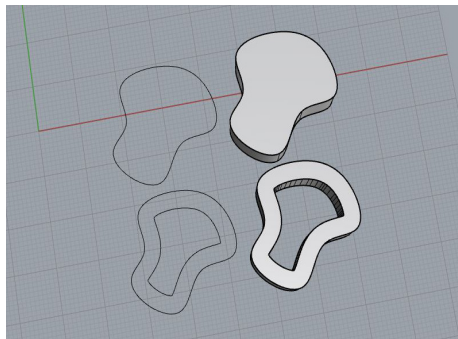
Solids: Closed objects made up of surfaces. Just like “closed” and “open” curves, a solid is only closed when all of its edges meet perfectly and it is “watertight.”

Polysurface: Multiple surfaces that have been joined together into one object. Polysurfaces can be open or closed. A closed polysurface is a solid.

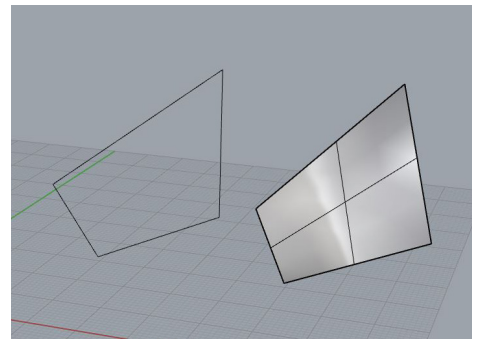
Commands that Create Surfaces:



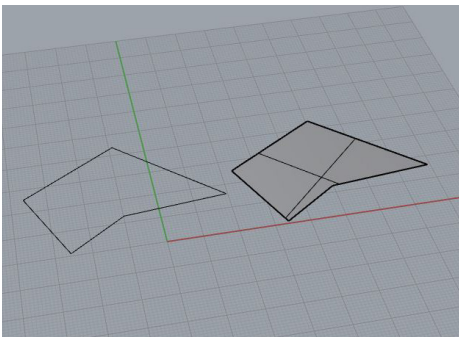
Pipe: Creates a tubular surface around a curve.



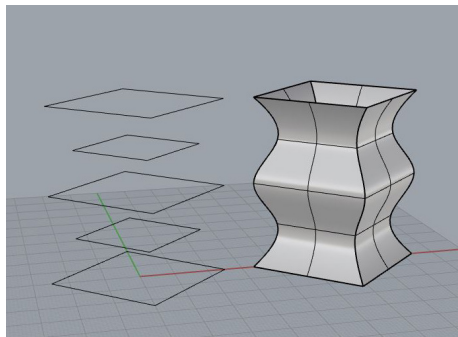
ExtrudeCrv: Creates a surface or solid by push/pulling your curve straight up or down.



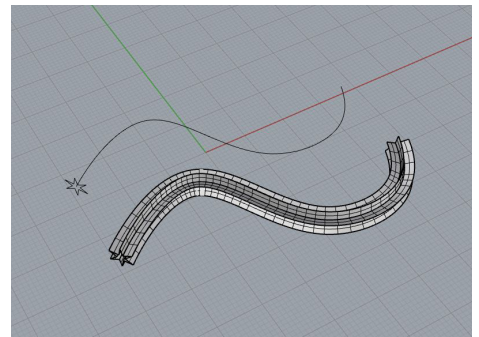
EdgeSrf: Creates a surface from 2, 3, or 4 curves.



PlanarSrf: Creates a surface between curves that are on the same plane.

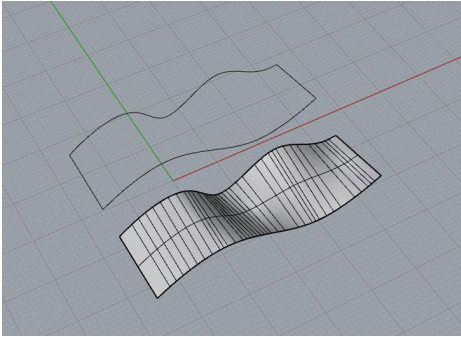


Loft: Fits a surface through selected profile curves that define the surface shape.

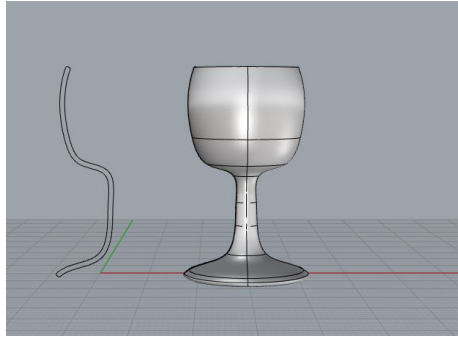


Sweep1: Fits a surface through a series of profile curves that define the surface shape and one curve that defines the path of the extrusion

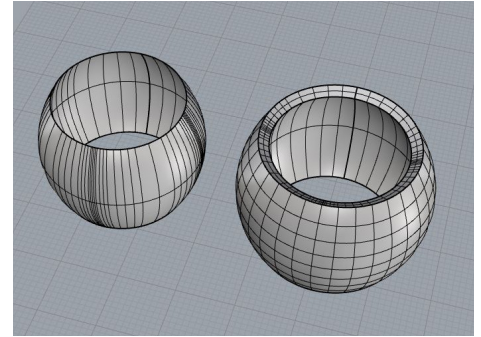
Commands that Create Surfaces:



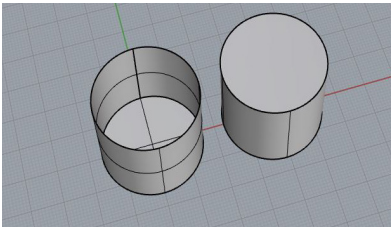
Sweep2: Fits a surface through a series of profile curves that define the surface shape and two curves that define the path of the extrusion



Revolve: Creates a surface by revolving a profile curve that defines the surface shape around an axis

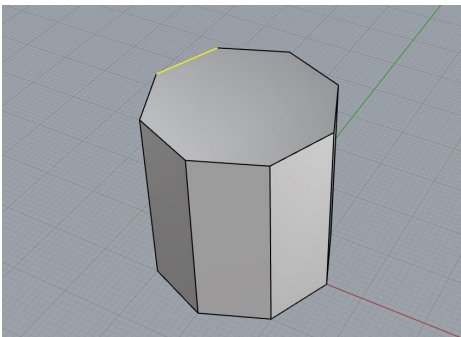


OffsetSrf: Creates a larger or smaller offset of your surface with the option to have your input surface and new surface become a solid.

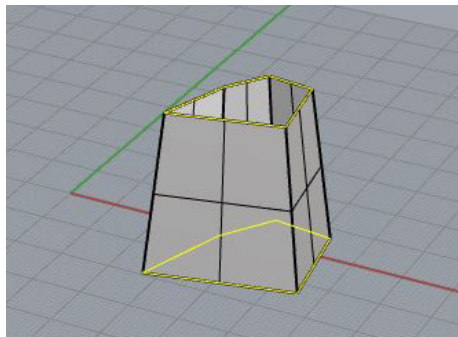


Cap: Fits a surface anywhere on your object that has a planar opening and joins it on. A shortcut for doing PlanarSrf then Join.

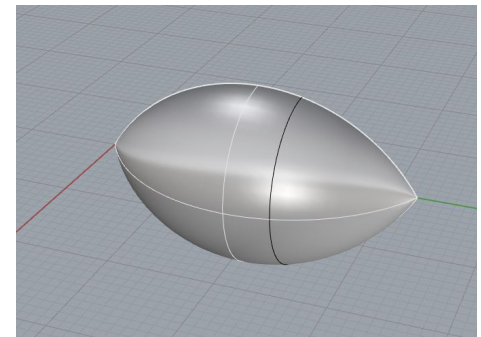
Commands that Pull Curves from Surfaces



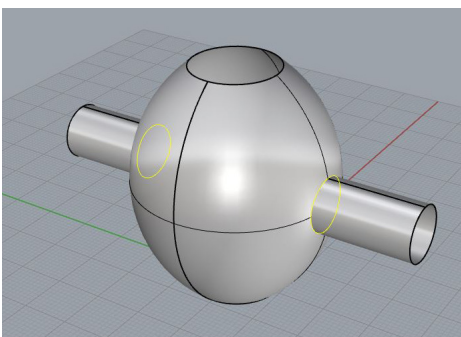
DupEdge: Duplicates a curve from any edge of a surface.



DupBorder: Duplicates all open edges of your object and joins them together automatically.



ExtractIsoCrv: Creates a curve along the U or V direction of your object. Useful to use with OSnaps on.



Intersect: Places a curve anywhere two objects intersect.

TO COMBINE *SOLID* OBJECTS (SEE NEXT WEEK'S VIDEOS) - Save before trying these commands, they can't be undone later!

BOOLEANUNION: Combines two solid objects into one

BOOLEANDIFFERENCE: Removes one solid object from another

BOOLEANSPLIT: Splits one solid by another. All remaining objects are solids.