

Useful Rhino 7 commands to work with engineering models

These commands are often used alongside *Griddle* tools to build/clean/modify model geometry in preparation for meshing

<i>Command</i>	<i>Brief description</i>
<i>General</i>	
<i>Ctrl+Z, Ctrl+Y</i>	Undo, Redo
<i>Zoom</i>	Zoom within mouse selection. If clicked on <i>Selected</i> – zoom to the selected objects, <i>Extents</i> - zoom to the extents of visible objects (also: CTRL+ALT+E)
<i>SelAll</i>	Select all visible objects
<i>SelPt</i>	Select (visible) points only
<i>SelCrv</i>	Select (visible) curves only
<i>SelSrf</i>	Select (visible) surfaces only
<i>SelPolysrf</i>	Select (visible) polysurfaces only
<i>SelMesh</i>	Select (visible) meshes only
<i>SelShortCrv</i>	Select all curves with length less than the specified length
<i>SelSmall</i>	Select objects by the diagonal of the bounding box
<i>SelVolumeObject</i>	Select objects enclosed by an extrusion object, surface, polysurface, or mesh
<i>Show</i>	Show all object
<i>Hide</i>	Hide selected objects
<i>Related to operations with Points</i>	
<i>Point</i>	Create a point
<i>Contour</i>	Create a spaced series of planar curves and points resulting from the intersection of a defined cutting planes through objects
<i>DrapePt</i>	Create a grid of point objects at the intersections of objects and points projected toward the construction plane
<i>PointsOn</i>	Displays control points (useful for displaying polylines or mesh vertices)
<i>PointsOff</i>	Turn off displaying control points (also: use ESC button)
<i>ExtractPt</i>	Create points at the position of curve/surface control points or mesh vertices
<i>Related to operations with Curves/Lines</i>	
<i>Curve</i>	Create a curve with specified polynomial form
<i>Line</i>	Create a straight line
<i>Polyline</i>	Create a polyline (with linear segments)
<i>CurveThroughPt</i>	Create a curve/polyline through selected points
<i>Divide</i>	Create points along a curve or split a curve into equal length segments
<i>SubCrv</i>	Extract a curve segment from a curve/polyline
<i>DeleteSubCrv</i>	Delete a curve segment from a curve/polyline
<i>ExtrudeCrv</i>	Extrude a curve/polyline in specified direction
<i>ExtrudeCrvAlongCrv</i>	Extrude a curve/polyline along another curve
<i>Sweep1</i>	Fit a surface through a series of profile curves that define the surface cross-sections and one curve that defines a surface edge
<i>Related to operations with Surfaces/Polysurfaces/Solids</i>	
<i>Box</i>	Create a box (extrusion of a polygon)
<i>Cylinder</i>	Create a cylinder (extrusion of a circle)
<i>Plane</i>	Create a rectangular planar surface
<i>PlaneThroughPt</i>	Create a rectangular planar surface through points

<i>Patch</i>	Fit a surface through selected curves, meshes, point objects, and point clouds
<i>Trim</i>	Cut and delete selected portions of an object at the intersection with another object
<i>Split</i>	Divide objects into parts using other objects as cutters
<i>Explode</i>	Break objects down into components
<i>SolidPtOn</i>	Turn on grips (solid edit points) at the ends of joined polysurface edges, including closed solids
<i>Cap</i>	Fill (cap) planar holes in objects with planar faces
<i>Drape</i>	Create a surface through points defined at the intersection of objects and points projected toward the construction plane

Related to operations with Meshes

<i>Mesh</i>	Create triangular mesh fitting selected object (triangulate an object)
<i>MeshBox</i>	Create a mesh box
<i>MeshPlane</i>	Create a rectangular mesh plane
<i>MeshPatch</i>	Create a mesh from curves and points
<i>MeshFromPoints</i>	Create a mesh from selected points or point clouds
<i>MeshFromLines</i>	Create a mesh from three or more line segments
<i>MeshToNURB</i>	Duplicate each polygon in a mesh with a (NURBS) surface
<i>3DFace</i>	Create a polygon mesh face in 3D
<i>ExtractMeshFaces</i>	Separate selected mesh faces from the parent mesh
<i>ExtractMeshFacesByArea</i>	Separate mesh faces that are within a specified area range from the parent mesh
<i>TriangulateMesh</i>	Split all planar quadrilateral mesh faces into two triangular mesh faces
<i>TriangulateNonPlanarQuads</i>	Split all non-planar quadrilateral mesh faces into two triangular mesh faces
<i>FillMeshHole</i>	Fill hole in a mesh by selecting its outline
<i>MeshSplit</i>	Divide meshes into parts with another object (cutter)
<i>MeshIntersect</i>	Create a polyline at the intersection of meshes
<i>ReduceMesh</i>	Decrease mesh face count while minimizing geometric and texture distortion

Other

<i>BoundingBox</i>	Create a box that encloses all selected objects, or each object in selection
<i>ShowEdges</i>	Show naked, non-manifold or all edges of the selected surface, polysurface, or mesh
<i>BoxEdit</i>	Edit object based on the bounding box parameters (switches to EditBox pane)
<i>Rotate</i>	Rotate an object (can use Gumball to rotate)
<i>Move</i>	Move an object
<i>Join</i>	Connect similar objects together to form a single object (e.g., 2 surfaces -> 1 polysurface)
<i>Smooth</i>	Average the positions of object points or vertices
<i>DupBorder</i>	Create curves from the boundaries of the selected objects
<i>Distance</i>	Report (measure) distance from one location to another
<i>Check</i>	Report errors in the selected object's data structure
<i>PolygonCount</i>	Report the number of mesh polygons in a selected object
<i>Area</i>	Report the area of the supported object types
<i>AreaCentroid</i>	Report the coordinates and place a point at the area centroid of selected objects
<i>Volume</i>	Report the volume of the supported object type
<i>VolumeCentroid</i>	Report the coordinates of and place a point at the volume centroid of selected objects
<i>Section</i>	Create a planar curve or points resulting from the intersection of a defined cutting plane through objects (show sections of objects)
<i>ClippingPlane</i>	Create a clipping plane object that represents a plane for visibly clipping away geometry
<i>RunPythonScript</i>	Run Python script from a file (also in Menu → Tools)

*This list is not extensive or complete. If Rhino is set in a language other than English, type “_” in front of the commands. See Rhino help for more information.