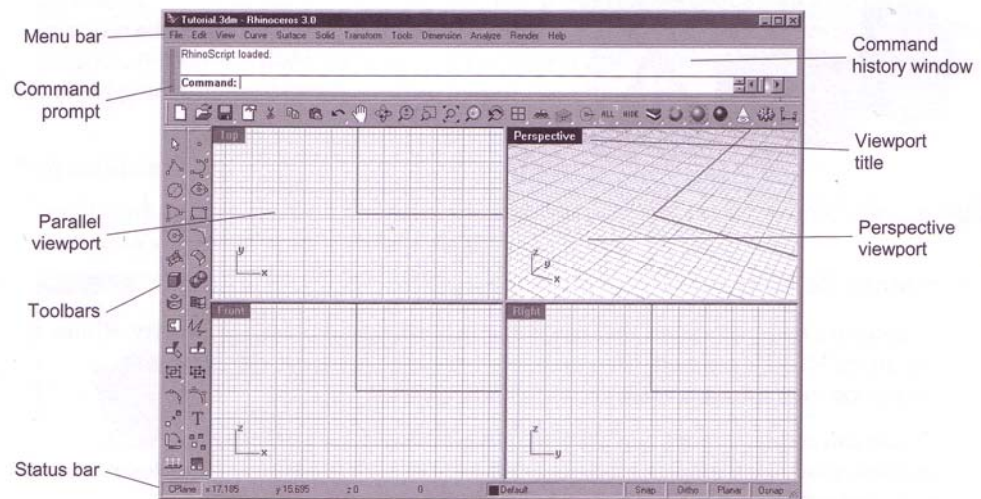


## RHINO 3D – NURBS Modeling for Windows

- **INTRODUCTION**
- Surface modeling vs. Solid modeling
  - Rhino uses a hybrid modeling environment, allowing you to model surfaces and closed polysurfaces (a “solid” when converted to STL, VRML, etc.) with precision and ease. The hybrid model allows Rhino to be used for virtually any application.
  - Surfaces are individual, zero thickness sheet bodies that are 2D or 3D.
  - Polysurfaces are multiple surfaces joined together to create zero thickness sheet bodies that are either 2D or 3D.
  - Solids are single surfaces or polysurfaces that form a closed body (3D only). (A closed polysurface object forms “watertight” geometry that can be used for rapid prototyping, i.e. 3D printing.)
- **GETTING STARTED**
  - File > New > (Select template file for units desired - for architectural models feet is usually best)
  - Rhino Interface



*Three parallel viewports and one perspective viewport.*

- Viewports can be maximized by double clicking on the viewport title (or right click > maximize). Double click again to restore the 4 viewports.
  - The command line functions much like AutoCAD. Simply begin typing the desired command to access the function. Rhino will automatically start to list commands that match the letters you have typed.
  - Holding the **right mouse button** (RMB) down in the perspective view will allow you to **rotate** the view. Holding **SHIFT+ RMB** will **pan** and **CTRL+RMB** will **zoom**.
  - Holding the **RMB** in an parallel view will **pan**.
  - The **middle wheel** will allow interactive **zooming**.
- **WORKING WITH OSNAPS AND UNITS**
    - At the bottom of the screen you will see the button for Osnaps, which is similar to AutoCAD’s snaps. Clicking on Osnap will reveal or hide the Osnap palette. By using a combination of different snaps, you can have great precision when locating and drawing objects in Rhino. (Note: Knot is Rhino’s terminology for the value of the curve parameter where the polynomial definition of the b-spline changes, simply meaning a break in the curve by another control point).

- The **Project** option projects the object snap you are using to the construction plane.
- Unit options can be accessed by going to File > Properties > Units. In this dialog you can change units, the way distances are displayed and the precision of the modeling windows.
- Also in the Properties dialog you can select the snap spacing of the grid by going to Grid > Grid Snap > Snap spacing.
- **Ortho** Mode: Ortho constraint can be turned on and off at the Ortho pane on the status bar. Press and hold **Shift** to temporarily toggle the ortho mode.
- To constrain to a distance, select the first point, then at the next prompt, type a distance, drag the cursor to the angle you want and click.
- Angle constraint allows you to set any angle and is a one time setting. To use angle constraint when drawing a line, for example, click for the start of the line, enter < (for angle) followed by the degree. <30 would allow you to snap at 30, 60, 90, etc. degrees. You can combine distance and angle by typing the distance desired, press **Enter**, then type the angle desired and **Enter** again.
- Elevator mode allows you to constrain to the Z-axis. When drawing, hold the **CTRL** key and click on the construction plane and drag up or down to move the point in the Z-axis.
- Use the **Tab** key to constrain the marker movement along the line between the first point and the marker's location.
- **SELECTING OBJECTS**
  - There are multiple ways to select objects in Rhino. Dragging a box around the objects from left to right will select only objects completely contained in the box. Dragging from right to left will select everything that is inside and that overlaps.
  - Holding **ALT** will allow you to draw a selection box without accidentally selecting a single object. **Shift** clicking will add objects to the selection. **Ctrl** clicking will remove objects from the selection.
  - When Rhino cannot tell which object you want to select, a menu will pop up with potential objects to select on the list. As you move the cursor through the list, the object and its name highlight. With some commands, you can select surface edges as curves.
- **POINTS AND CURVES**
  - Points mark a single point in 3D space. They are most often used as placeholders, or objects to snap to. The **Point** function (command: point) places a point object.
  - **Divide a curve with points** (command: divide): Marks curves into a specified number of segments or of a specified length. Useful for creating points to snap to along the curve.
  - **Project** (command: project): A curve can be projected straight onto the surface perpendicular to the current construction plane.
  - Creating lines is similar to AutoCAD, formZ, etc. For help on creating curves see the help menu.
  - **Contour from surface** (command: contour): Creates a series of curves at a specified distance or number of a surface.
  - **Extract Isocurve** (command: extractisocurve): Creates an isocurve on a surface at the point desired. Can be extracted in either the U or V direction.
  -
- **SURFACE MODELING TECHNIQUES**
  - Rhino offers a multitude of ways to create accurate, complex NURBS surfaces.
  - **Surface from Points** (command: SrfPt): Creates a surface by defining three or four corner points.
  - **Surface from Planar Curves** (command: PlanarSrf): Creates a surface from closed planar curves. If a closed curve is wholly within another closed curve, it will be treated as a hole boundary.

- **Surface from Edge Curves** (command: EdgeSrf): Creates a surface from curves that define the surface edges.
- **Plane** (command: plane): Creates a rectangular planar surface.
- **Cutting Plane** (command: cutplane): Creates a cutting plane, which is a rectangle that is automatically drawn through selected objects. You define the direction of the plane, and it will cut through the selected objects.
- **Extrude** (command: extrudecrv): Creates a surface from any curve, in a straight line perpendicular to the construction plane. Setting mode to straight will extrude the curve perpendicular to the plane of the curve. Setting Cap to yes will create a closed polysurface (solid).
- **Loft** (command: loft): Creates a surface through a series of closed or open lines. At the **Select curves to loft** prompts, select the curves in the order that the surface should pass through them. When selecting open curves, select near the same ends. After selecting all curves, hit enter or right click (Note: right clicking = enter). A dialog window will appear with options how to construct the surface.
  - **Normal** uses chord-length parameterization in the loft direction.
  - **Loose** allows the surface to move away from the original curves to make a smoother surface.
  - **Tight** forces the surface to stick closely to the original curves.
  - **Straight sections** creates straight sections between the curves (a ruled surface).
  - **Developable** creates a separate developable surface or polysurface (developable surfaces are those that can be formed by rolling a flat sheet of material such that the material does not stretch, tear, or wrinkle).
- **Sweep** (command: sweep1 for one-rail sweeps, sweep2 for two rail sweeps): Creates a surface by sweeping a shape along a path. First select the rail curve (the path of the object) and then the object you wish to sweep. A two rail sweep provides more control over the path, allowing you to select two rails instead of one.
- **Revolve** (command: revolve): Creates a surface by revolving a line around a central axis.
- **Revolve with rail** (command: railrevolve): Creates a surface by sweeping one end of a profile curve along a shape curve, while keeping the other end fixed.
- **Fillet between two surfaces** (command: filletsrf): Creates an arc-shaped surface between two starting surfaces.
- **Chamfer between two surfaces** (command: chamfersrf): Inserts a straight surface between two surfaces.
- **Blend between two surfaces** (command: blendsrf): Creates a smooth blend between surfaces rather than the arc-shaped fillet. The edges of the starting surface are maintained.
- **Offset a surface** (command: offsetsrf): Creates a new surface at a specified distance from the original. Can be used to create a solid surface by selecting the Solid option.
- **Surface from Curve Network** (command: networksrf): Creates a surface from a network of smooth curves. All curves in one direction must cross all curves in the other direction and cannot cross each other.
- **SURFACE EDITING TECHNIQUES**
  - Once you have objects drawn, you will want to manipulate them by trimming, joining, moving, rotating, copying, deforming, etc.
  - When editing surfaces there are multiple ways to control them. By turning on a surface's control points (**F10**), you can manipulate the position, scale, rotation, etc. of each individual control point.
  - **Join** (command: join): Connects curves or surfaces together to form one object.
  - **Explode** (command: explode): Removes the connection between joined curves or surfaces. Note: for polysurfaces this command is very useful, as you cannot edit control points when it is a polysurface.

- **Trim and Split** (commands: trim, split): When you trim an object, you select the parts to remove, when you split an object the parts are left, but separated.
- **Untrim** (command: untrim): Allows you to untrim a previously trimmed object.
- **Split at Isocurve** (Surface>Surface edit tools> Split at isocurve): Allows you to split a surface at a designated isocurve.
- **Smooth** (command: smooth): Can reduce the amount of bumps in a surface or curve.
- **Match** (command: match (for curves), matchsurf (for surfaces)): Changes the position, tangency, or curvature of a curve or surface to match another at the edge.
- **Merge two surfaces into one** (command: mergesrf): Changes untrimmed surfaces into a single surface. The seam can be smoothed. The resulting surface can be edited.
- **Rebuild** (command: rebuild): Re-spaces the control points on a curve or surface evenly and sets the number of control points to be the same for a group of curves.
- **Control Points on/off** (key: F10): Turns on or off the display of editable control points.
- **Polysurfaces cannot be edited with control points. For most editing functions you must extract the surfaces you want to work on from the polysurface and rejoin them.**
- **Extract a surface from a polysurface** (command: extractsrf): Disconnects a surface from a polysurface. Similar to Explode, but only removes one surface, allowing the others to remain joined.
- **Booleans:** These work best on closed solids, but can be used on other surfaces.
  - Command: **BooleanUnion**: Combines objects into one object.
  - Command: **BooleanDifference**: Subtracts one object from another.
  - Command: **BooleanIntersection**: Creates the volume that was enclosed by both objects.
- To make a NURBS surface a polygon mesh, use the **Mesh** command.
- **ANALYSIS**
  - To change the direction of a curve or the normals of a surface, use the **Direction** function (command: dir). These are important for rapid prototyping.
  - To measure an object, use the **Distance** function (command: distance).
  - Before exporting for 3D printing, use the **What** command to check the object's properties. To 3D print, the object **MUST** be a **closed polysurface**.
  - If the polysurface is not closed, use the **Show Edges** function (command: ShowEdges) to display unjoined edges.
- **MORE HELP**
  - For more assistance, use the Rhino User's Guide in the Help Menu.
  - If you still cannot figure something out, go to Help > Help on the Web > Rhino Newsgroup. There you can post questions to other Rhino users who are extremely helpful in solving your problems.