



RHINO 3D 1 CHEAT SHEET

This guide orients new Rhino users to the software interface and key commands. Rhinoceros is a commercial 3D computer graphics and computer-aided design application, compatible with a number of applications at Open Works.

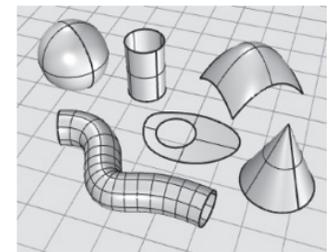
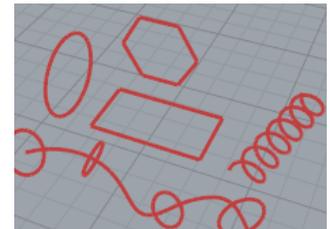
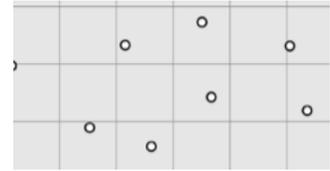
NURBS MODELING

NURBS (non-uniform rational B-splines) are mathematical representations that can accurately model any shape from a simple 2-D line, a simple shape, or a complex 3-D form. Because of their flexibility and accuracy, NURBS models can be used in any process from illustration and animation to manufacturing. The advantage of using NURBS over using polygon modelers (such as Sketchup) is that NURBS surfaces have no facets. The models can be rendered at any resolution.

A point is the simplest object in Rhino. Points can be placed anywhere in space, and are often used as placeholders.

A curve is similar to a piece of wire. It can be straight or wiggled, and can be open or closed. A polycurve has several curve segments joined together end to end. Curves can be edited by changing their control-point structure. Curves encompass straight lines, polylines made of connected line segments, arcs, circles, polygons, ellipses, helices, and spirals. You can also draw curves using curve control points and draw curves that pass through selected points.

A surface is like a rectangular stretchy sheet. The NURBS form can represent simple shapes, such as planes and cylinders, as well as free-form, sculptured surfaces. Rhino has many tools for constructing surfaces directly or from existing curves.



NAVIGATING RHINO

Coordinate Systems follow what is called the right-hand rule. The right-hand rule can help you determine the direction of the z-axis in a coordinate system. As the L shaped hand shows, the thumb represents the +x-direction, the index finger is the "+y" direction, and the palm faces in the "+z"-direction.

Viewports are windows in the graphics area that display views of your model.

- To toggle between a small viewport and one that fills the graphics area, **double-click** the viewport title.
- Rhino also allows you to render the work in different styles. **Right-click** the viewport title, or click the small black triangle to display the viewport title menu.

To zoom in and out, hold down the Ctrl key and drag up and down with the right mouse button held down.

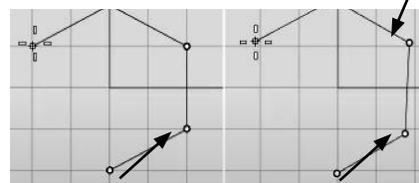
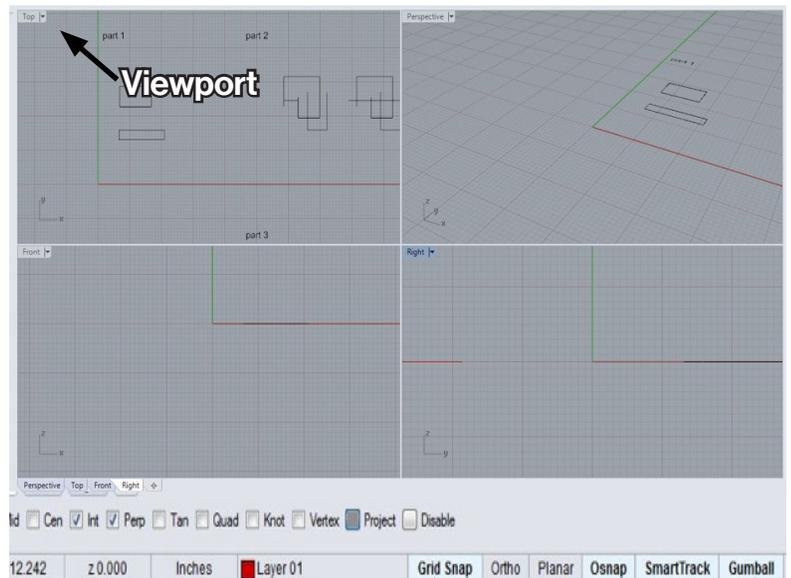
- If you have a mouse with a wheel, you can also scroll the wheel to zoom in and out.
- To zoom to a specific area, hover cursor over the object.

The Pan command shifts the location of the view camera and target parallel to the view plane.

- In a viewport with a parallel projection, like "Top" view, drag the mouse with the right mouse button held down.
- In "Perspective" view, hold the Shift key and drag while holding the right mouse button.

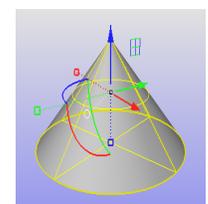
OBJECT SNAPS

When Rhino asks you to pick a point, you can use object snaps to pick a precise location on an existing object to ensure accurate connections and improved workflow. When you move the cursor near a location on an object that corresponds to an object snap, the marker jumps to that location. You can select the object snap by checking or unchecking the box next to the snap type (End, near, point etc).



Grid Snap ON vs Grid Snap OFF

Grid snap constrains the marker to the crossing points of the grid. Click the Grid Snap pane on the status bar to turn grid snap on and off. Right-click the Grid Snap to set the snap parameters.



Gumball displays the gumball widet on a selected object to move, scale, and rotate transformations around the gumball origin. Drag gumball arrows to Move the object. Drag scale handles (squares) to Scale the object in one direction.



COMMANDS

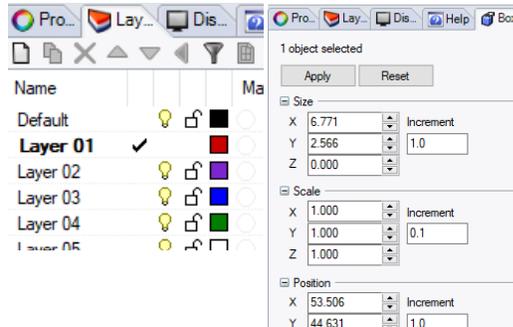
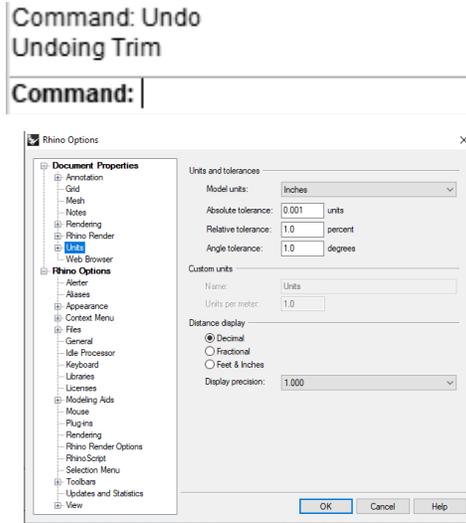
Rhino is a command-driven program. Commands can be utilized with icons from the toolbar, in the menu bar, or by simply typing the command. At the top of the screen there is a window that shows the command line. Using the command line is helpful as it walks you through complex commands and allows you to access and edit your history.

FILE SETUP

Rhino can work in any unit system and with any tolerance. The default unit system is millimeters, and the default tolerance is 0.01 millimeters. You can change the default unit system or tolerance by setting up a template. It is best to select a tolerance when you start modeling and stick with it. The small window to the right of your workspace contains useful tabs.

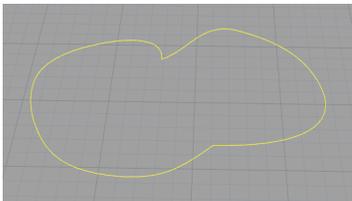
- **Layers** tab will help you keep track of objects, especially when creating laser and CNC cut files.

- The **“Box Edit”** command will open a tab that allows you to manually edit and determine aspects of objects in your file, including coordinates and dimensions.

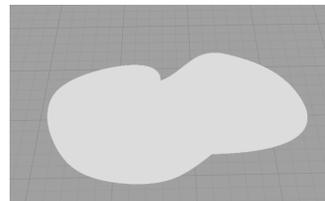


Extruding forms

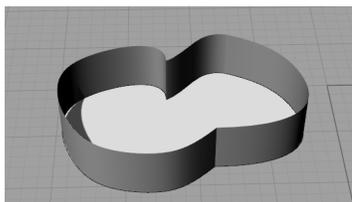
A very essential function of 3D Modeling, and most useful methods to design furniture and components for CNC routing is turning a curve into a 3D surface. There are several different approaches to doing this in Rhino. Here's how to turn an organic curve into a 3D solid.



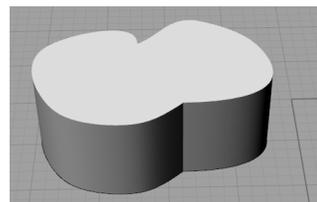
1. Select curve



2. Type command *Planarsrf*



3. Command *ExtrudePlanarsrf*, type desired height of extrusion.

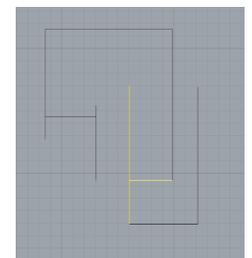
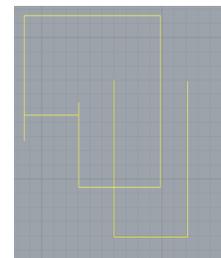


4. Close the shape by selecting the curves of open edge and command *Planarsrf* or *Cap*.

TOOL BARS

Depending on what mode you are working in, you can select custom tool bar arrangements. The image here is the default tool bar, consisting mostly of Curve tools. Icons with a small grey triangle indicate that you can right click it to reveal a set of controls related to that command. Every icon has its own key command.

- Select** tool will allow you to pick objects and groups. Selected objects
- Polyline** creates joined line or arc segments.
- Curve tools** use control point locations to create organic lines and trace images. A wide range of Curve tools exist for making Circle, Ellipse, Arc, Rectangle, and Polygons.
- Surface tools** are for creating NURBs surfaces made up of facets. A surface may be open or closed and made directly, from points or curves and may be used to create
- Join** connects curves, surface edges, or surfaces to form a single object.
- Group** creates a single named unit from selected objects. **Ungroup** removes the group status from the selected objects.
- TextObject** creates text-shaped curves, surfaces, or polysurfaces based on TrueType fonts.
- Copy** makes duplicates of the selected objects. The user picks the start and end points to copy.
- Array** is used to copy and uniformly arrange multiples of objects. **Align** lines up the bounding boxes of objects.
- Deformation tools**, including **FlowAlongSrf**, allow you to morph objects from a source to a target
- Blocks** manage repeated objects in a file by replacing the geometry you select with a **“block instance”**. It save the geometry you selected to define the block, in the file.
- Point** allows you to create a single point object.
- HandleCurve** creates chained Bézier curves with editing handles.
- A **Bézier Curve** is a curved line or path connected to and manipulated by fixed points, and defined by mathematical equations.
- Fillet** creates arc-shaped transitions between curves or surfaces.
- Boolean** commands are like joining and trimming for solids. Work best on closed surfaces and polysurfaces that intersect each other. New forms can be created from the subtracted/added
- Explode** breaks up an object into sections of its composing curves and surfaces
- Both Trim and Split** cut objects with other objects. To trim an object, the selected parts are deleted. Splitting an object “breaks” it into parts.
- PointsOn, EditPoints** To display or edit a curve
- Move** an object a certain distance or if you want to use object snaps to place an object accurately.
- Transformation tools** allow you to scale, rotate, invert, and flip the objects
- Check command** reports errors in the selected object's data structure.
- History** stores the connection between a command's input geometry and the result, and is one of the great strengths of Rhino. When the input geometry changes, the result updates accordingly.



Curves can be broken up into segments for editing, or joined together to create solid objects or unified shapes. The left image is a “joined” object. The right image is an “exploded” and “trimmed” object.