Lesson 1 – Rhino. Basics (rhino3d.com)

NURBS Modeler (vs Mesh modeler)
Points , Curves ,Surfaces , Solids (surface that encloses a volume), SubD, Mesh

Template - Sets Units & Tolerance

Interface

Commands – Three ways to enter commands:

1. Toolbars - Default (Standard, Main1, Main2 - right-click to see list)

   Left/right button commands
   Carets - Additional Toolbars

2. Command Prompt (F2) - Type in command or alias of command

   Enter/Right-Click/Spacebar - "Enter" or Repeats
   Can also click on option text in command prompt

3. Pull-Down – (i.e. “Solid/Box/Corner to Corner, Height”)

Viewports - Four, 3 ortho, 1 perspective - Double-Click on name does Min/Max

Each view defines an X-Y “Construction Plane” passing through 0,0,0

Ortho Views
Wheel = Zoom
Right button & Drag = Pan

Perspective Views
Wheel = Zoom
Right button & Drag = Rotate/Orbit View (+ Shift = Pan)

Shaded Modes – Wireframe, Shaded, Ghosted/X-Ray, etc…

Settings – “Tools/Options…”. Controls settings and options.

Status Bar
Grid snap (F9), Ortho (F8 – also “Shift”) – Right-click for settings

OSnap - END, etc... (also type "end", etc... on the command prompt)
Right-click selects and clears other osnaps
Project – Forces snapped point to construction plane (grid)
Disable - De-activates any active osnap settings

Lesson 1 – Rhino, Basics, Curves (cont...)

Status Bar (cont…)

Planar
ON Keeps in current XY plane
OFF Snaps to viewport's construction plane (grid)

Smart Track – Offers logical points.

Gumball – Cover later
Record History – Cover later

Select - "Edit>Select Objects"
Window – Right (up or down). Crossing – Left (up or down)
Shift – Adds. Ctrl - Removes

Drag - "easy move". Not accurate.
Nudge - ALT+Arrow key (+CTRL=less, +Shift=more).
0, 45, 90 – Shift (set under "tools/options/modeling aids")
Lock angle - Tab
"ALT" - copies while dragging (click, then alt)
Elevator Mode ("Z") - Ctrl, best in perspective view

Transforms ("Transform") - For accuracy. Check command prompt

Gumball – "Gizmo" with all three transforms. Click on icon for keyboard input.
Move – "Arrow"
Rotate – "Arc"
Scale – "Box"

Move
Copy
Rotate

Mirror

Orient (aligns edge) – Can scale and/or copy
2 Points – First p1 goes to the second p1 and first p2 to second p2
3 Points – Orients in 3-axes.

Lesson 1 – Rhino, Basics, Curves (cont...)

Layers ("Edit")

Set (double-click on layer to set)
Lightbulb = on/off. Padlock = locked/unlocked
Color Swatch – Change layer color. Geometry assumes color of layer.
New/Delete

"Right-Click" – Options (new, rename, etc…), change object's layer, etc...

Properties ("Edit/Object Properties" or F3)

1. Object (if selected)
2. Viewport (if nothing selected)

Creating Geometry - Points, Curves, Surfaces, Solids

1. Points - Placeholders
   Multiple Points ("Curve/Point Objects/...")

2. Curves (Polycurve - An object with more than one curve)
   Fit to Points ("Curve/Free-Form/...") - Must go through point objects.

3. Surfaces (Polysurface - An object with more than one surface)
   Extrude Curve/Straight ("Surface/...") - Extrudes line to surfaces

   Selection Menu – Displays when multiple objects coincide.
   Objects independent of each other.

4. Solid - Surface that encloses a volume
   Extrude Planar Curve/Straight ("Solid/...") - Extrudes line to solids

5. SubD – Subdivision surfaces
   Extrude Curve/Straight ("SubD/...") - Extrudes line to SubD

6. Meshes – Triangles
   From NURBS Object (“Mesh”) – Converts a nurb object into a mesh

1. Points - Placeholders. ("Curve/Point Objects/...")

Single Point/Multiple Points

Divide curve By - Creates points on a curve

Lesson 1 – Rhino, Basics, Curves (cont...)

2. Curves - Define surfaces, paths/rails.
    Common settings/concepts
        Degree of curve is "Degree+1" = Min Control Points
            1 degree [x^1] - straight (2 control points)
            2 degree [x^2] - simple shape
            3 degree [x^3] - Default

        Autoclose – curve closes automatically if near. "ALT" suspends.

        Control Points – Determines shape of curve
            By default, Activated/Deactivated when curve is selected/deselcted

            Editing control points - “Edit/Control Points/..”
                ON (also "F10")
                OFF (also "F11" or ESC)

                Insert/Remove (delete key) Control Point – Will effect curve
                Edit Weight – Strength of control point on curve

                Insert Knot/Remove Knot – Adds CP without effecting curve

                Insert Kink – Changes curve’s direction

        Have direction – Effects surface orientation
            To change - ("Analyze/Direction") or "Dir", Flip or click to flip
            Many commands allow “flipping” internally

        Record History –Links geometry (i.e. extruded curve)

    Curves (Stand-alone)

        Primitives ("Curve") – Not parametric
            Circle
            Rectangle
            Arc, etc...
            Parabola, Conic, Hyperbola

        Lines
            Single Line/ Line segments ("Curve/Line/...")

            Polyline ("Curve/Line/...") - Continuous Sequence, polycurve

Lesson 1 – Rhino, Basics, Curves (cont...)

2. Curves (cont…)

Curves (Stand-alone)

Control Points ("Curve/Free-Form/...")
Clicked points become “Control Points”

Interpolate Points ("Curve/Free-Form/...")
Curve goes through clicked points
Control points are derived from curve, not clicks

Handle Curve ("Curve/Free-Form/...")
By handles. ALT makes kinks

Sketch Curve ("Curve/Free-Form/...")
Draw with mouse button depressed
Check command prompt for options (closing)

Fit to Points ("Curve/Free-Form/...")
Creates “Interpolate Point” curve from existing point objects.

Curves from other curves

Offset Curve ("Curve") - Offsets perpendicular. Choose side.
Distance – Type in command prompt
Cap closes

Tween Curves ("Curve") - Averages 2 or more Curves. Click ends.
Number = Number of averaged (new) curves
Flip – Reverses endpoints

Curve from 2 views ("Curve") - Choose curves in different planes
Curves MUST be in different construction planes (i.e. TOP & RIGHT)

Cross Section Profiles ("Curve") - Makes cross-section curves
Choose 3 or more profiles in 2 views (i.e. FRONT & RIGHT)
MUST be selected in order (i.e. clockwise)
Cut one or more section lines.
Closed curves are created in section plane and touch profiles

Lesson 1 – Rhino, Basics, Curves (cont...)

Editing Tools

Curves **AND/OR** Surfaces (“Edit”)

Trim ("Edit") - Cutting edge, then curve/surface + portion to trim off
Untrim ("Surface/Surface Edit Tools/..") - Surface then edge of trim

Split ("Edit") - Breaks with split curve/surface. To split, then cutting edge.

Curves **OR** Surfaces (“Edit” – If commands works on either)

Join ("Edit") - Joins Curves to Polycurves, Surfaces to Polysurfaces, etc…

Explode ("Edit") - Breaks apart Polycurves/Polysurfaces

Rebuild ("Edit") - Redistributes control points

Extend ("Curve/surface") - Boundary, then extend

Fillet ("Curve/Surface") - Joins two curves (surfaces), set "Radius"
Click part to keep

Blend ("Curve/Surface") - Works existing curves/surfaces.

Transforms

Array(s) – Rectangular, Polar, Linear

Array Along Curve/Surface – Arrays object along curve or surface
Maintains orientation/distance between selected objects and curve

Flow Along Curve/Surface - Aligns objects from a curve/surface to another.
Select Object(s) to flow
First curve/surface, then
Second curve/surface

Perpendicular to Curve – Aligns curve perpendicular to curve.
Select object to move and object’s base point.
Select Curve and then point on curve.