2 - Rhino Objects

The fundamental geometric objects in Rhino are points, curves, surfaces, polysurfaces, solids, light-weight extrusion objects, and polygon mesh objects.

This chapter goes into some detail to explain the mathematical basis for Rhino modeling. It is not necessary to understand everything in this chapter to start using Rhino.

Why NURBS modeling

NURBS (non-uniform rational B-splines) are mathematical representations that can accurately model any shape from a simple 2-D line, circle, arc, or box to the most complex 3-D free-form organic surface or solid. Because of their flexibility and accuracy, NURBS models can be used in any process from illustration and animation to manufacturing.

NURBS geometry is an industry standard for designers who work in 3-D where forms are free and flowing; where both form and function is important. Rhino is used in marine, aerospace, and automobile interior and exterior design. Makers of household and office appliances, furniture, medical and sports equipment, footwear, and jewelry use Rhino to create free-form shapes.

NURBS modeling is also widely used by professional animators and graphic artists. The advantage of using NURBS over using polygon modelers is that NURBS surfaces have no facets. The models can be rendered at any resolution. A mesh can be created from the model at any resolution.
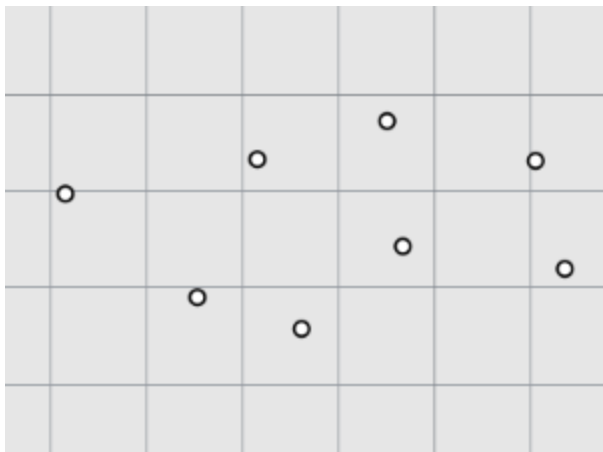
See also

For more information about the mathematics of NURBS, see *What are NURBS?*.

For perhaps more than you want to know, see the Wikipedia article, *Non-uniform Rational B-Spline*.

Points

Point objects mark a single point in 3-D space. They are the simplest objects in Rhino. Points can be placed anywhere in space. Points are most often used as placeholders.
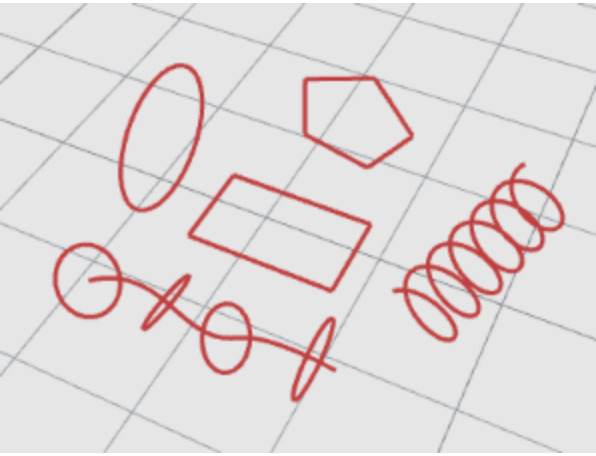

Curves

A Rhino 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.

Rhino provides many tools for drawing curves. You can draw straight lines, polylines that consist of connected line segments, arcs, circles, polygons, ellipses, helices, and spirals.

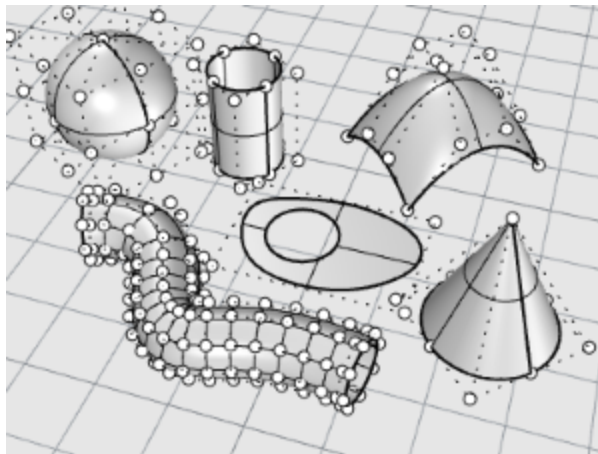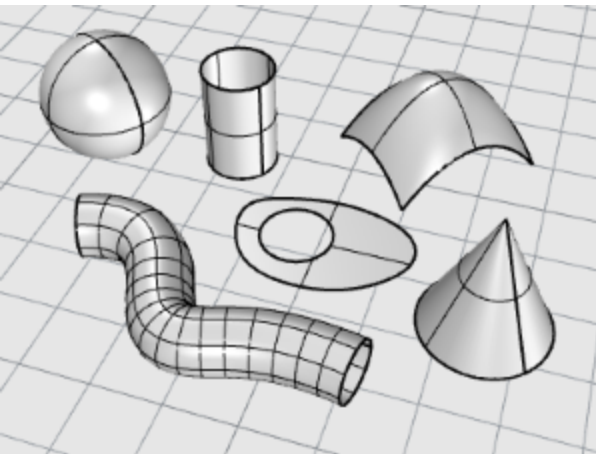You can also draw curves using curve *control points* and draw curves that pass through selected points.

Curves in Rhino include lines, arcs, circles, free-form curves, and combinations of these. Curves can be open or closed, planar, or non-planar.

Surfaces

A surface is like a rectangular stretchy rubber sheet. The NURBS form can represent simple shapes, such as planes and cylinders, as well as free-form, sculptured surfaces.

All surface creation commands in Rhino result in the same object: a NURBS surface. Rhino has many tools for constructing surfaces directly or from existing curves.
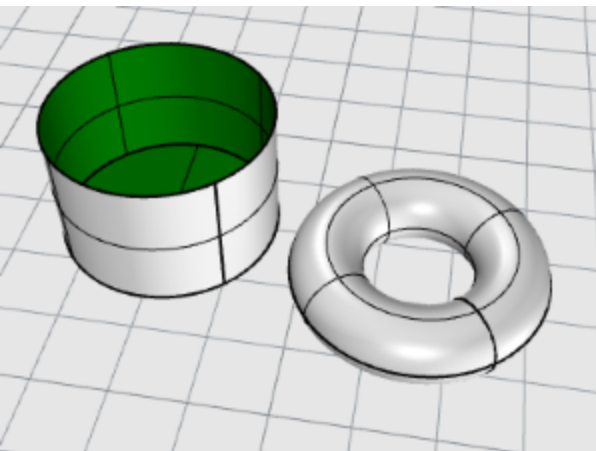

All NURBS surfaces have an inherently rectangular organization.

Even a closed surface such as a cylinder is like a rectangular piece of paper that has been rolled up so two opposite edges are touching. The place where the edges come together is called the *seam*.

Closed and open surfaces

A surface can be open or closed. An open cylinder is closed in one direction.
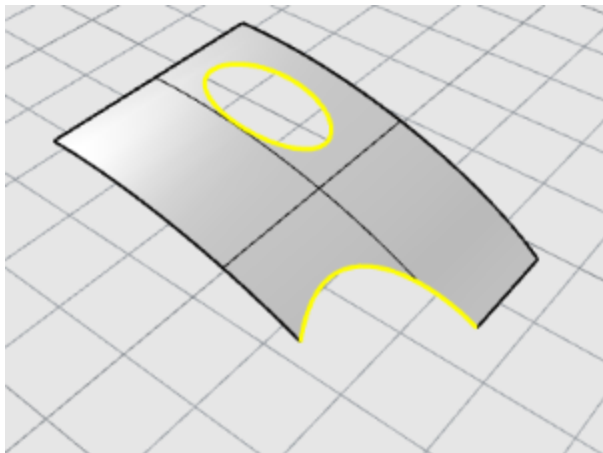
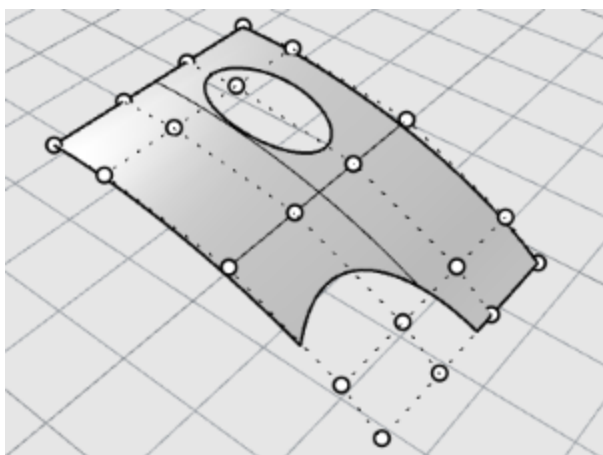A torus (donut shape) is closed in two directions.

Trimmed and untrimmed surfaces

Surfaces can be trimmed or untrimmed. A trimmed surface has two parts: a surface that underlies everything and defines the geometric shape, and trimming curves that mark sections of the underlying surface that are trimmed away.

Trimmed surfaces are created with commands that trim or split surfaces with curves and other surfaces. Some commands create trimmed surfaces directly.
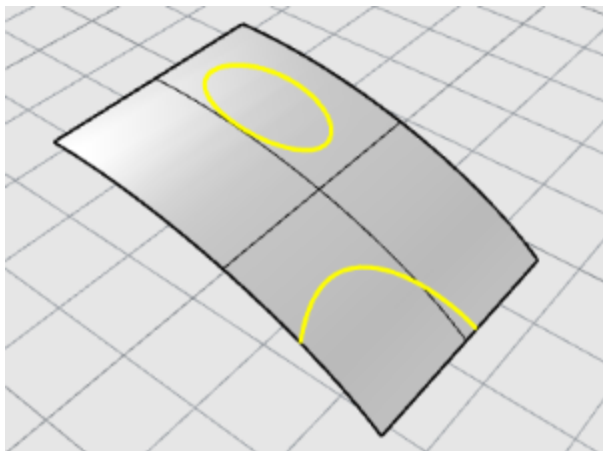

The shape of a surface is still defined by a set of control points arranged in a rectangular pattern.
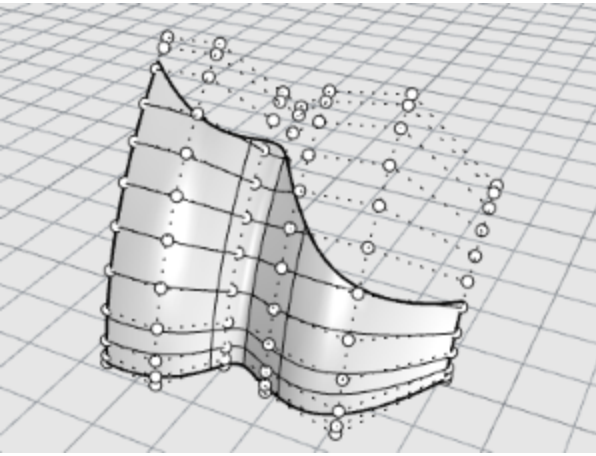

Since it can be important for you to know if a surface is trimmed, the **Properties** command lists the trimmed or untrimmed state of the surface. Some Rhino commands work only with untrimmed surfaces and some software does not import trimmed NURBS surfaces.
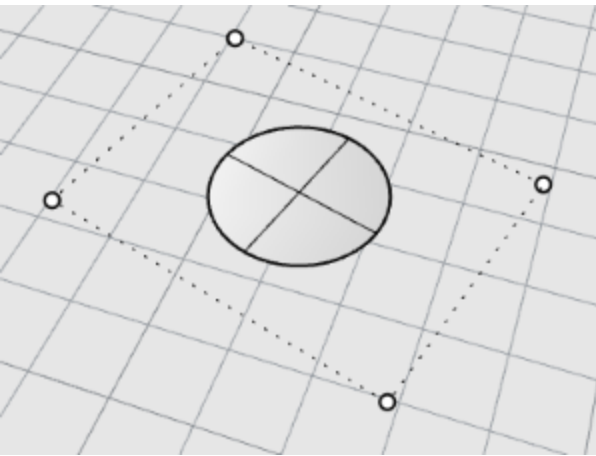
Trimming curves lie on the underlying surface. This surface may be larger than the trimming curves, but you will not see the underlying surface because Rhino does not draw the part of the surface that is outside the trimming curves. Every trimmed surface retains information about its underlying surface geometry. You can remove the trimming curve boundaries to make the surface untrimmed with the **Untrim** command.
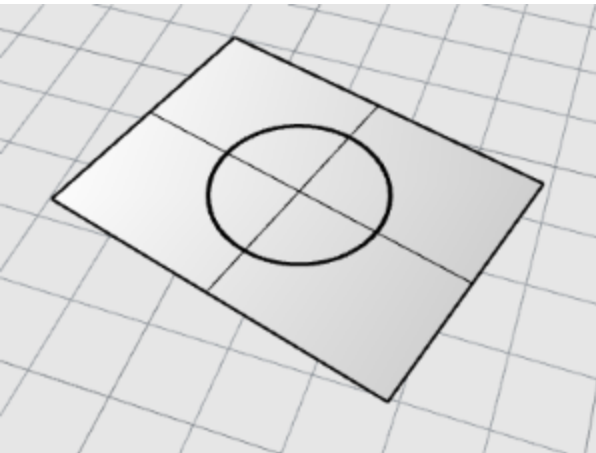

If you have a trimming curve that runs across a surface, the trimming curve itself does not have any real relationship to the control point structure of the surface. You can see this if you select such a trimmed surface and turn its control points on. You will see the control points for the whole underlying surface.

If you create a surface from a planar curve, it can be a trimmed surface. The illustrated surface was created from a circle. The control points display shows the rectangular structure of the surface.
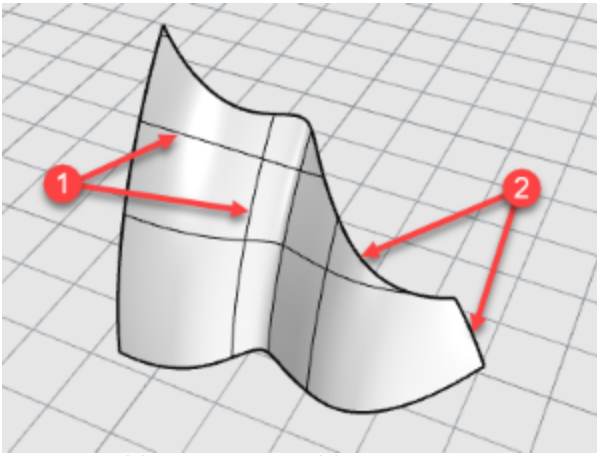

The **Untrim** command removes the trimming curve from the surface to get back to the underlying untrimmed rectangular surface.

Surface isoparametric and edge curves

In wireframe view, surfaces look like a set of crossing curves. These curves are called *isoparametric curves* or *isocurves*. These curves help you visualize the shape of the surface. Isoparametric curves do not define the surface; they are merely a visual aid that allows you to see the surface on the screen. When a surface is selected, all of its isoparametric curves highlight.

Edge curves bound the surface. Surface edge curves can be used as input to other commands.

Isocurves (1), Edge curves (2).

Curve and surface degree

A NURBS curve is defined by four things: degree, control points, knots, and an evaluation rule.

NURBS functions are rational polynomials and the degree of the NURBS is the degree of the polynomial.

A polynomial is a function like $y = 3x^3 - 2x + 1$. The "degree" of the polynomial is the largest power of the variable. For example, the degree of $3x^3 - 2x + 1$ is 3; the degree of $-x^5 + x^2$ is 5, and so on.
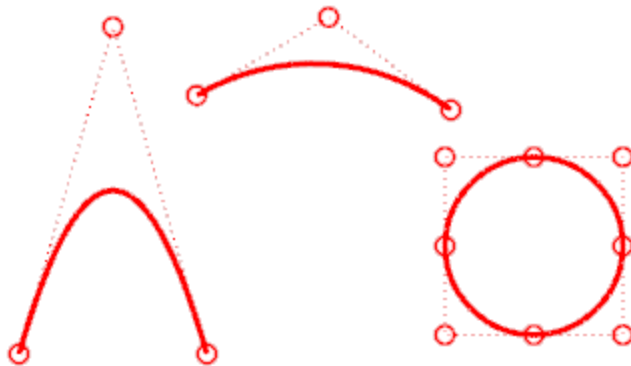
From a NURBS modeling point of view, the (degree –1) is the maximum number of "bends" you can get in each span. This determines how much you can affect the shape of the curve.

For example:

- A degree-1 curve must have at least two control points.
  A line has degree 1. It has zero bends.

- A degree-2 curve must have at least three control points.
  A parabola, hyperbola, arc, and circle (conic section curves) have degree 2. They have one bend.

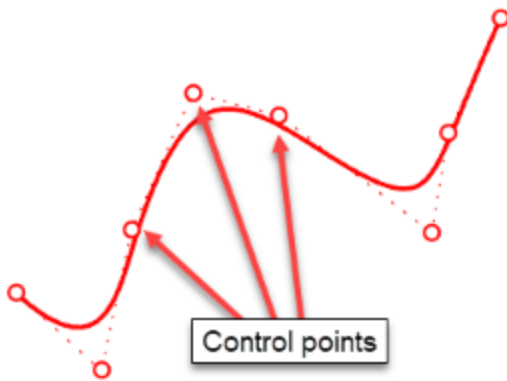- A degree-3 curve must have at least four control points.
  A cubic Bézier has degree 3. If you arrange its control points in a zig-zag shape, you can get two bends.

For more information about curve geometry, see the Wikipedia article, *Gallery of Curves*.

Control points

Control points influence the shape of a curve or surface. The control point holds such information as location, direction, and weight. You can make subtle changes in the shape of a curve or surface by moving the location of its control points. Rhino offers many tools for editing control points. Some of the later tutorials show control point manipulation.

Control point visibility

To edit curves and surfaces by manipulating control points, use the **PointsOn (F10)** command to turn the control points on.

When you are finished with control-point editing, use the **PointsOff (F11)** command or press **Esc** to turn them off.

Control points of polysurfaces cannot be turned on for editing with PointsOn. Editing the control points of polysurfaces could separate the edges of the joined surfaces creating "leaks" in the polysurface. A special **SolidPtOn** command turns on grip points for polysurfaces, which act like control points.

Change control point locations

When you move control points, the curve or surface changes, and Rhino smoothly redraws it. Rhino's transform commands, such as Move, Copy, Rotate, and Scale, can manipulate individual or multiple points.

Add, delete, or redistribute control points

Adding control points to a curve gives you more control over the shape of the curve. Commands such as **InsertControlPoint**, **InsertEditPoint**, and **InsertKink** add control points to curves.

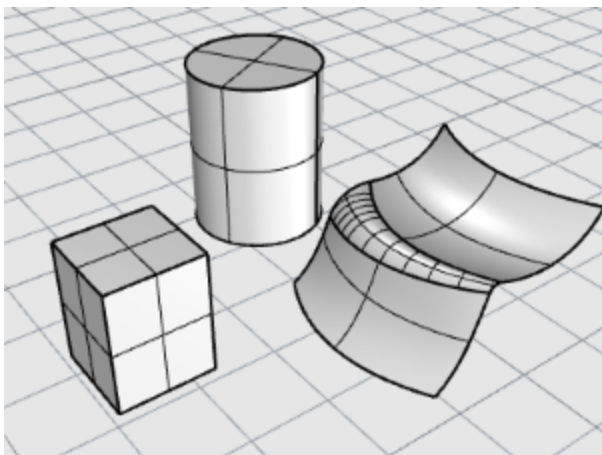Manipulating control points also lets you remove kinks, make curves uniform, and add or subtract detail.

Some commands such as **Rebuild**, **Fair**, **ChangeDegree**, and **Smooth** offer some automated solutions for redistributing control points over a curve or surface.

Other commands, such as control point dragging and nudging, **MoveUVN**, and **HBar** let you manually control the location of individual or groups of control points.

The **Delete** key removes selected curve control points. This changes the shape of the curve.
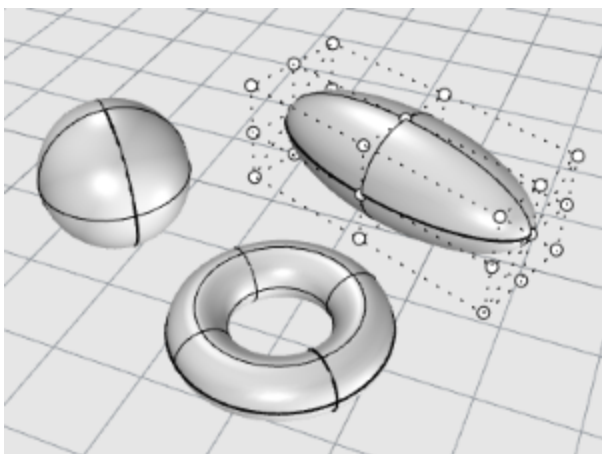
Polysurfaces

A *polysurface* consists of two or more surfaces that are joined together. A polysurface that encloses a volume of space defines a solid.
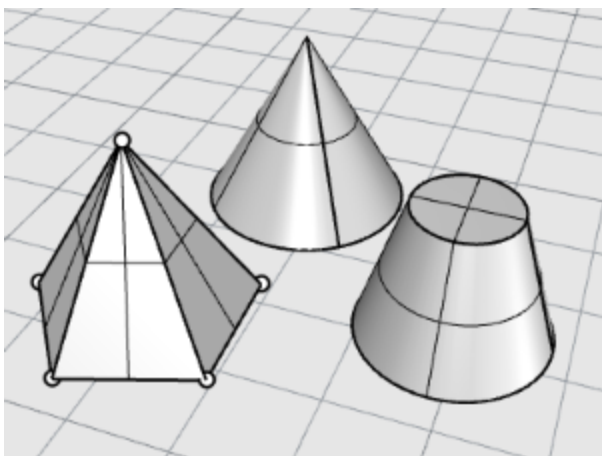
Solids

A solid is a surface or polysurface that encloses a volume. Solids are created anytime a surface or polysurface is completely closed. Rhino creates single-surface solids, polysurface solids, extrusion solids and mesh solids.

A single surface can wrap around and join itself. Example commands include **Sphere**, **Torus**, and **Ellipsoid**. Control points can be displayed on single-surface solids and moved to change the surface.
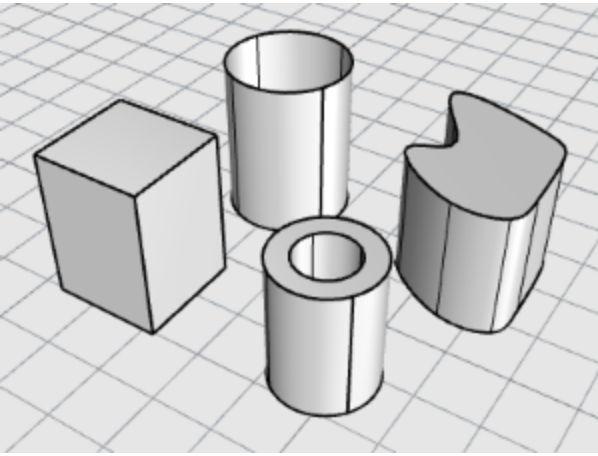

Some Rhino commands create polysurface solids. **Pyramid**, **Cone**, and **TruncatedCone** are examples of commands that create polysurface solids.

The **SolidPtOn** command turns on grip points for polysurfaces, which act like control points.
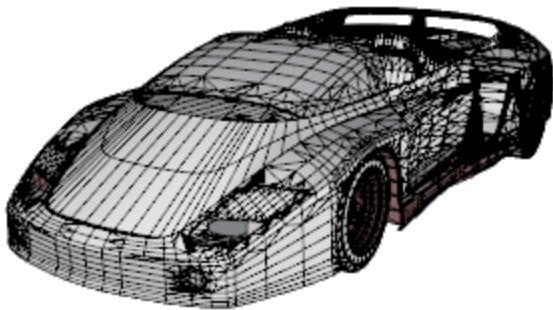

Light-weight extrusion objects

Light-weight extrusion objects use only a profile curve and a length as input instead of the network of isocurves normally needed for NURBS objects. The **Box**, **Cylinder**, **Tube**, and **ExtrudeCrv** commands create extrusion objects. Extrusion objects can be closed with a planar cap or open. These objects will be converted to polysurfaces by some commands if necessary to add more information for editing.

Polygon mesh objects

Because there are many modelers that use polygon meshes to represent geometry for rendering, animation, stereolithography, visualization, and finite element analysis, the **Mesh** command translates NURBS geometry into polygon meshes for export. Also, the Mesh creation commands **MeshSphere**, **MeshBox**, **MeshCylinder**, etc., draw mesh objects.

Note: There is no easy way to convert a mesh model into a NURBS model. The information that defines the objects is completely different.

However, Rhino has a few commands for drawing curves on meshes and extracting vertex points and other information from mesh objects to assist in using mesh information to create NURBS models.