

# Rhinoceros<sup>®</sup>

NURBS modeling for Windows



Introduction to Rhinoceros 4.0

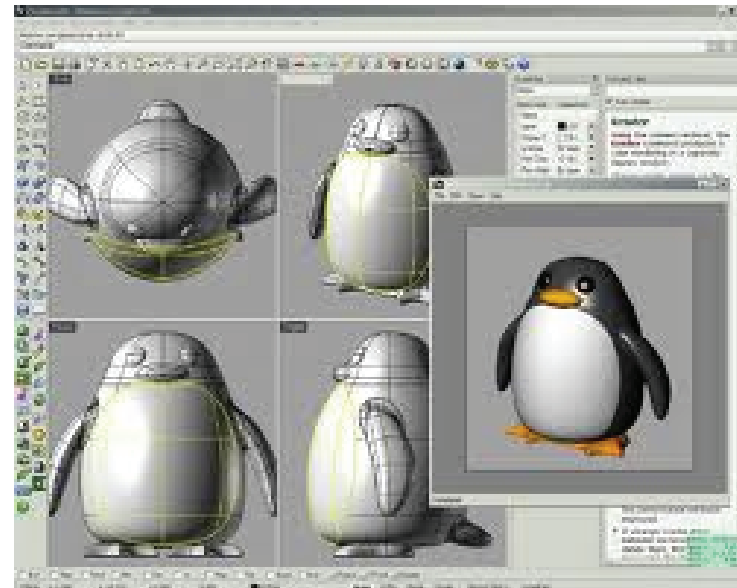
The ARC.

# What is Rhino 4.0?

- It is a stand-alone, commercial NURBS-based 3-D modeling software.  
NURBS- Non-uniform rational basis spline
- Many versions of rhino can be downloaded from their website,  
<http://www.rhino3d.com/download.htm#new>
- The software is commonly used for industrial design, architecture, marine design, jewelry design, automotive design, CAD / CAM, rapid prototyping, reverse engineering, product design as well as the multimedia and graphic design industries
- Can be used to import and export drawings from or to other softwares, such as AutoCAD, Revit, 3DsMax, etc.

## Capabilities

- 2D line drawings
- 3D construction drawings
- Rendering (with sunlight and materiality)
- Parts Assemblies



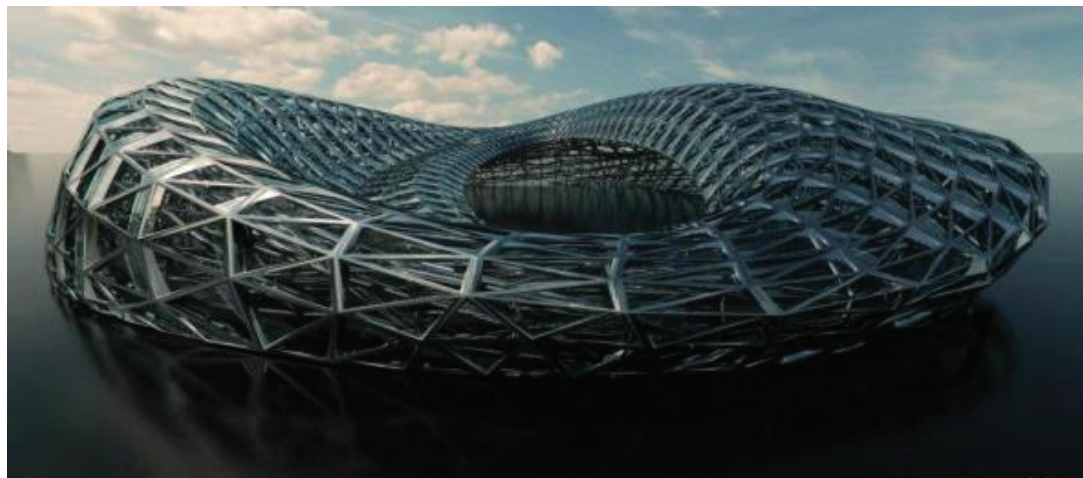
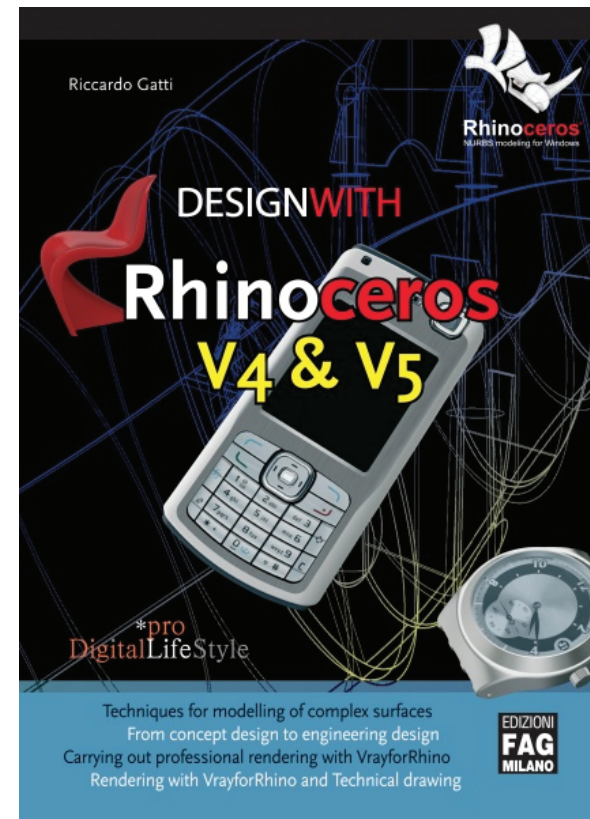
# Workshop Goals

## What We'll Cover:

- Setting up the Workspace
- Basic/Common Commands
- Manipulating Properties
- Viewports
- Printing/Exporting

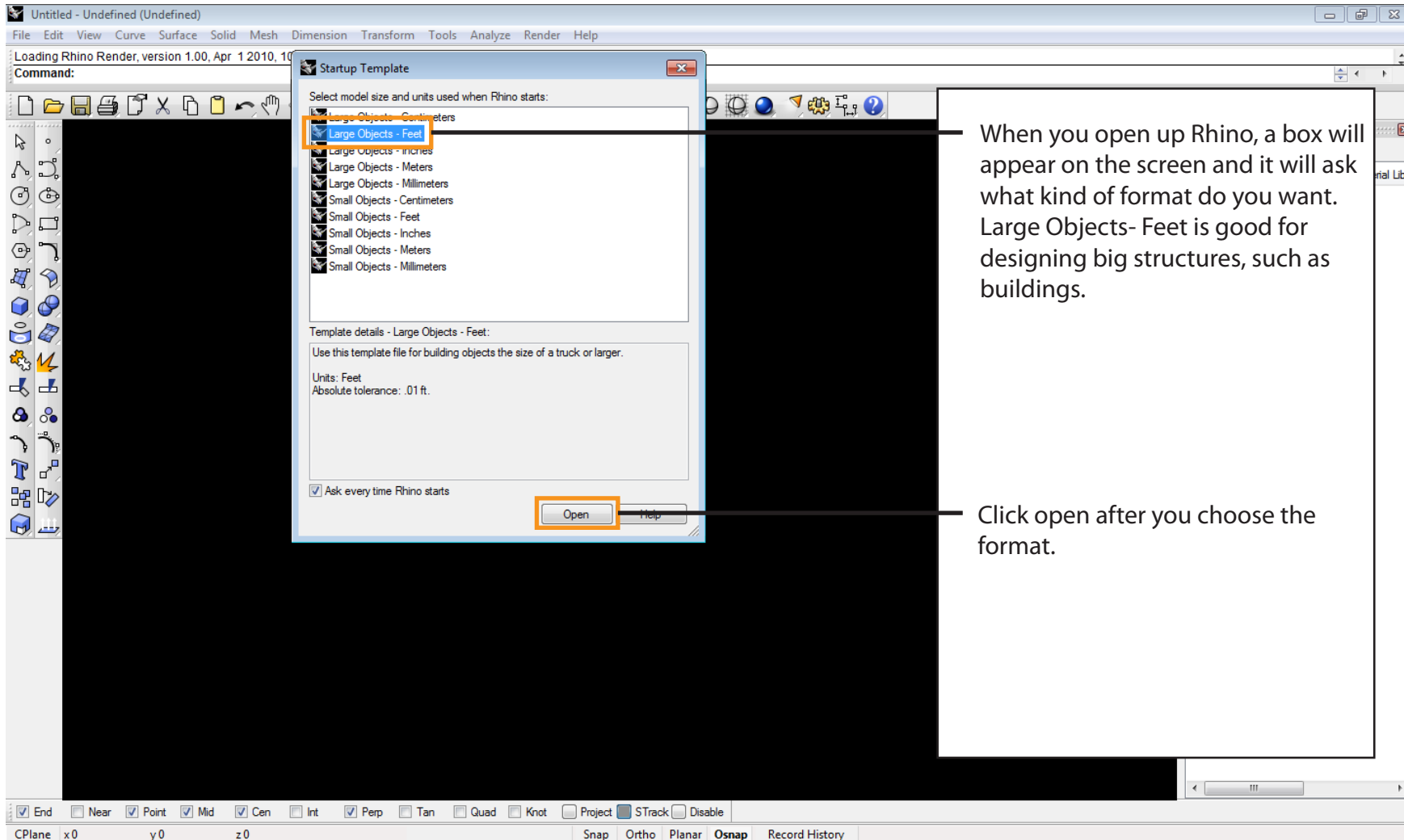
## What We Won't Cover:

- Rendering
- Parts/Assemblies
- Rendering Properties



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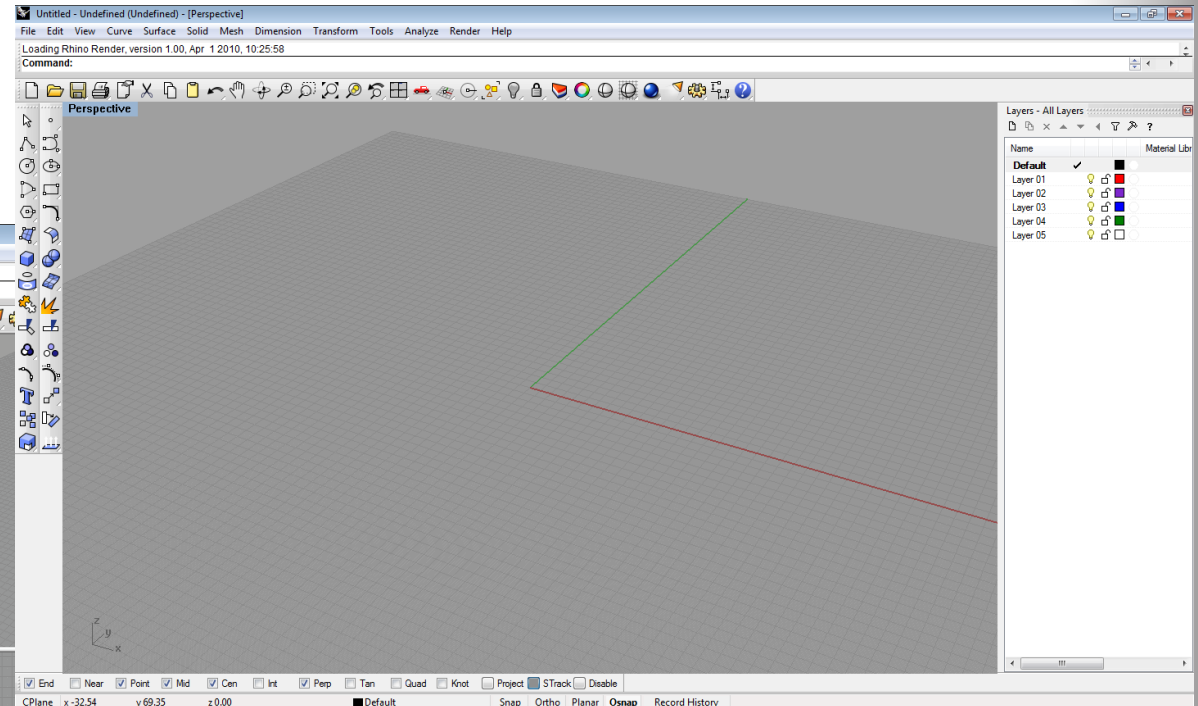
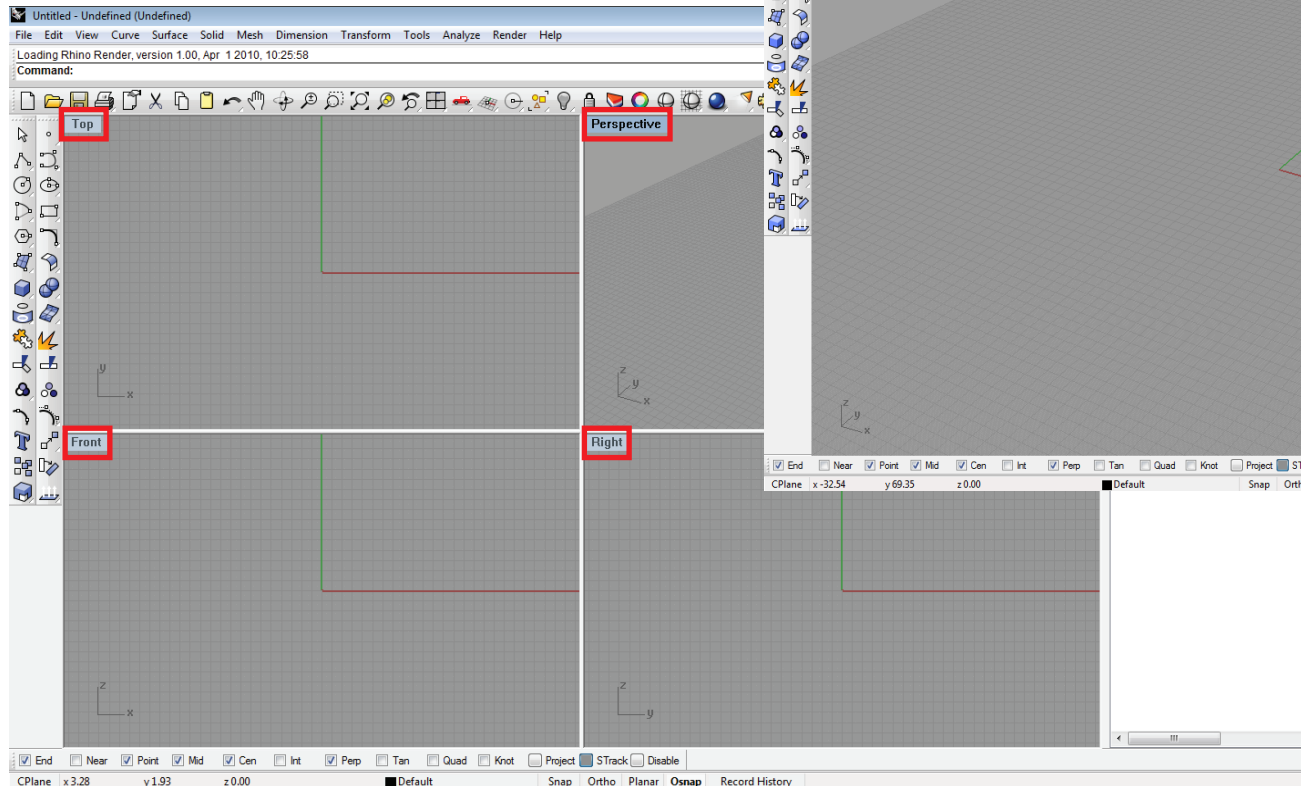
# Starting a New Drawing



# Looking at the Workspace

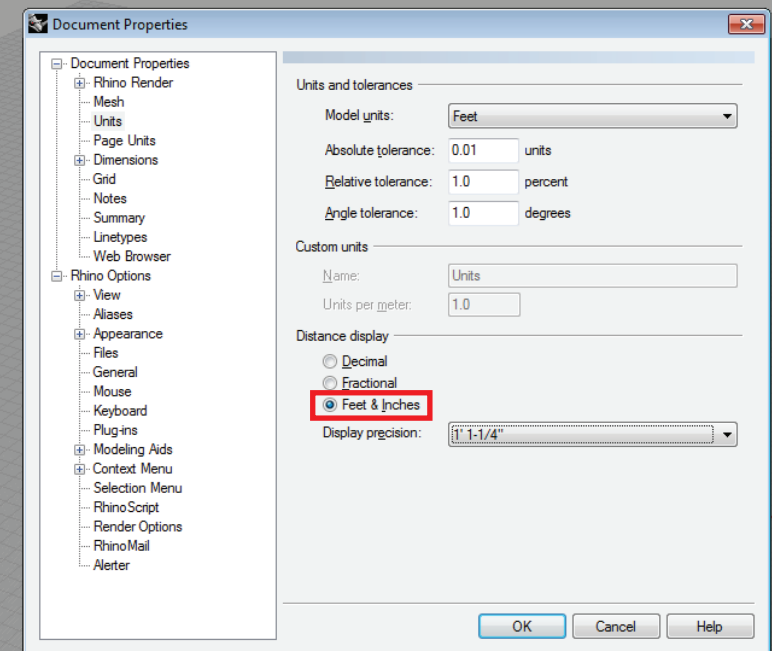
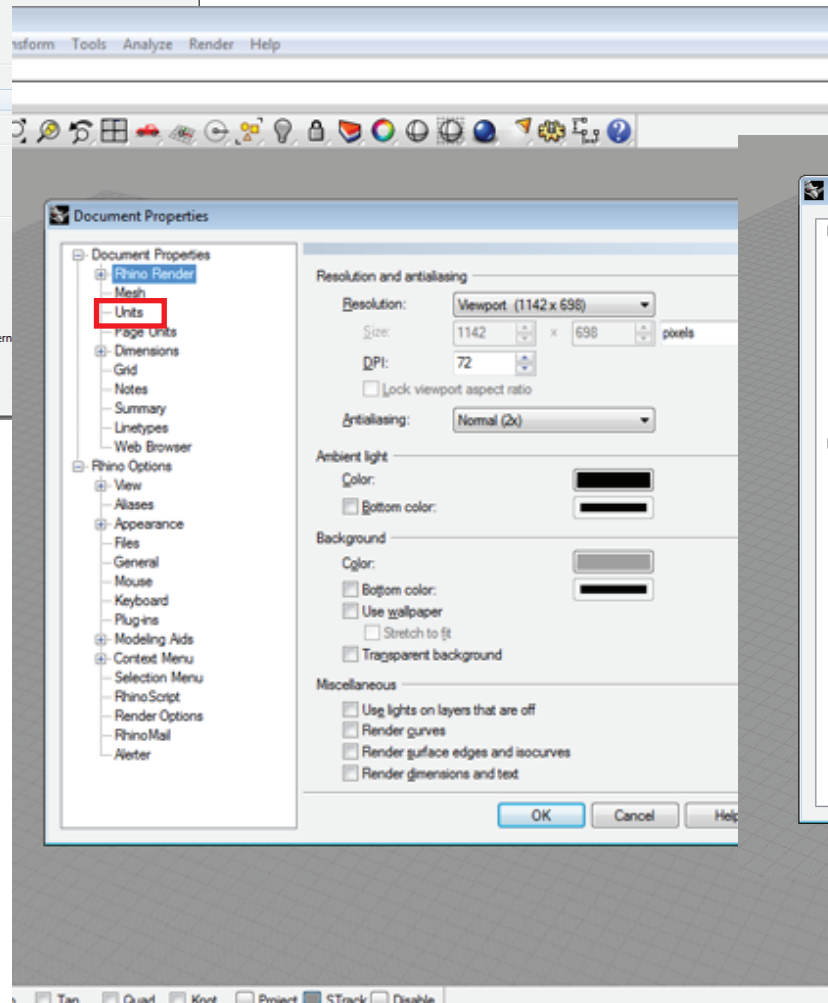
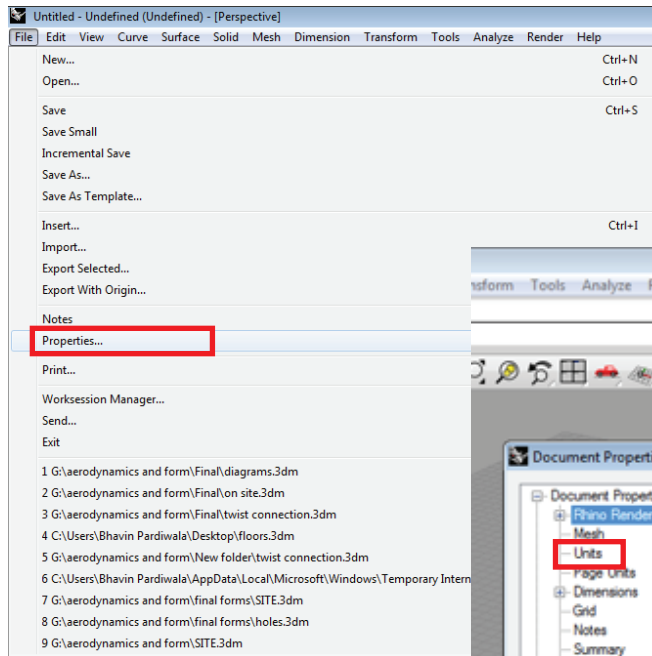
As you can see, you have 4 viewports: The Top View, Perspective View, Front View, and Right Side View.

By double-clicking on the word, it will maximize that viewport, which is useful for zooming in and detailing the drawing.



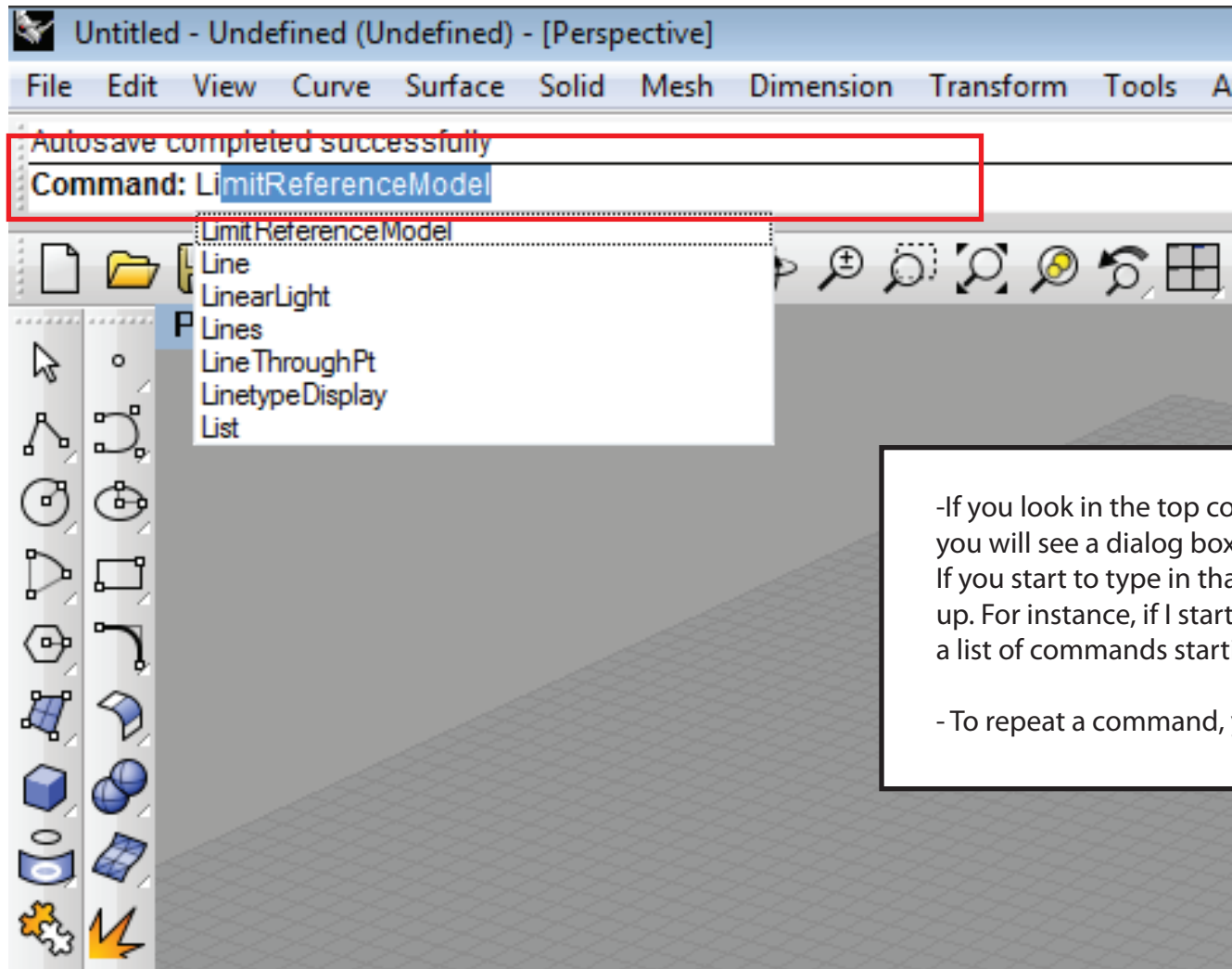
# Setting up the Workspace

If you go to File ---> Properties, a Document Properties box will show up, and from here, you can modify anything you want. The units, the meshing, resolution, dimensions, background color, keyboard shortcuts, and many more options. If we go under units, you can change the model units to whatever you like, however we are in Feet, so Feet should be already displayed. If not, you can change it. Change the distance display to feet and inches, and keep the display precision the same as the default.





# The Command Line

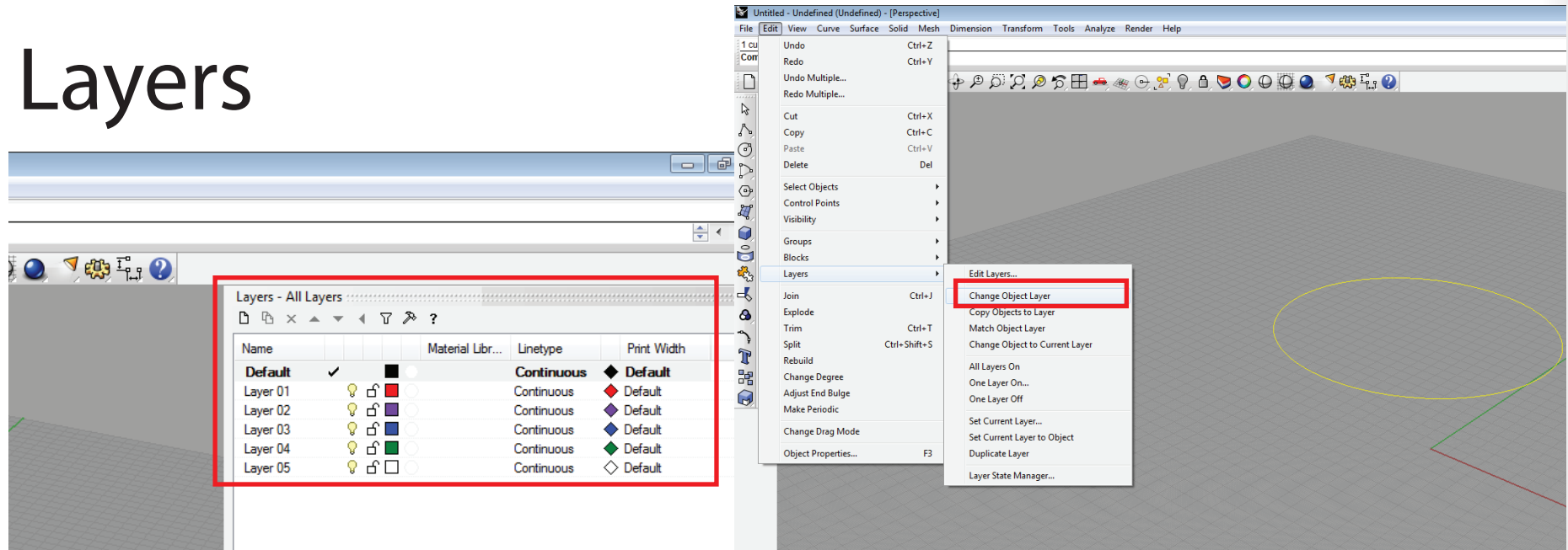


-If you look in the top corner, under the File and Edit, you will see a dialog box and the word COMMAND: written. If you start to type in that box, a list of command will show up. For instance, if I start to type in the word LINE, a list of commands starting with L-I- will show up.

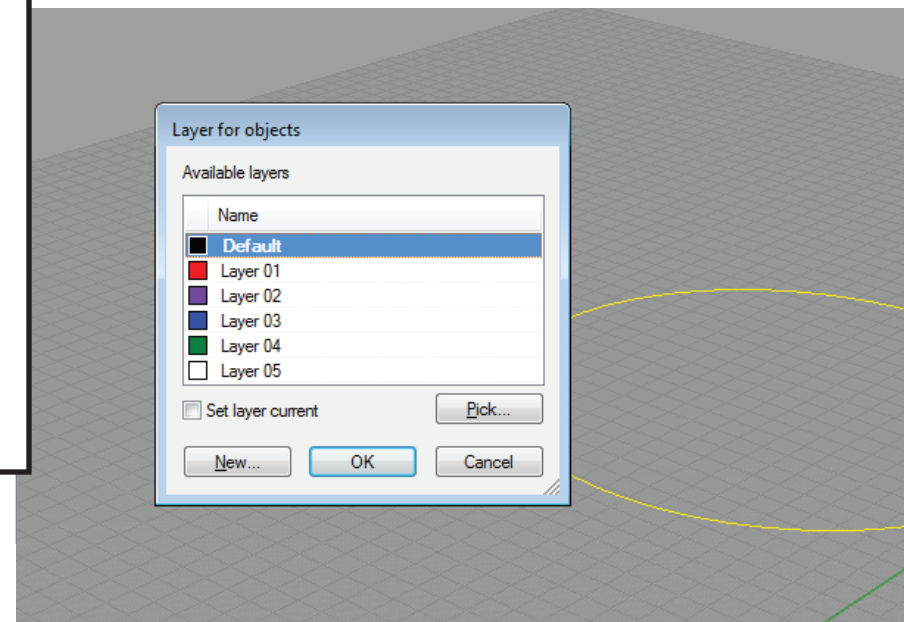
- To repeat a command, you can press ENTER.



# Layers

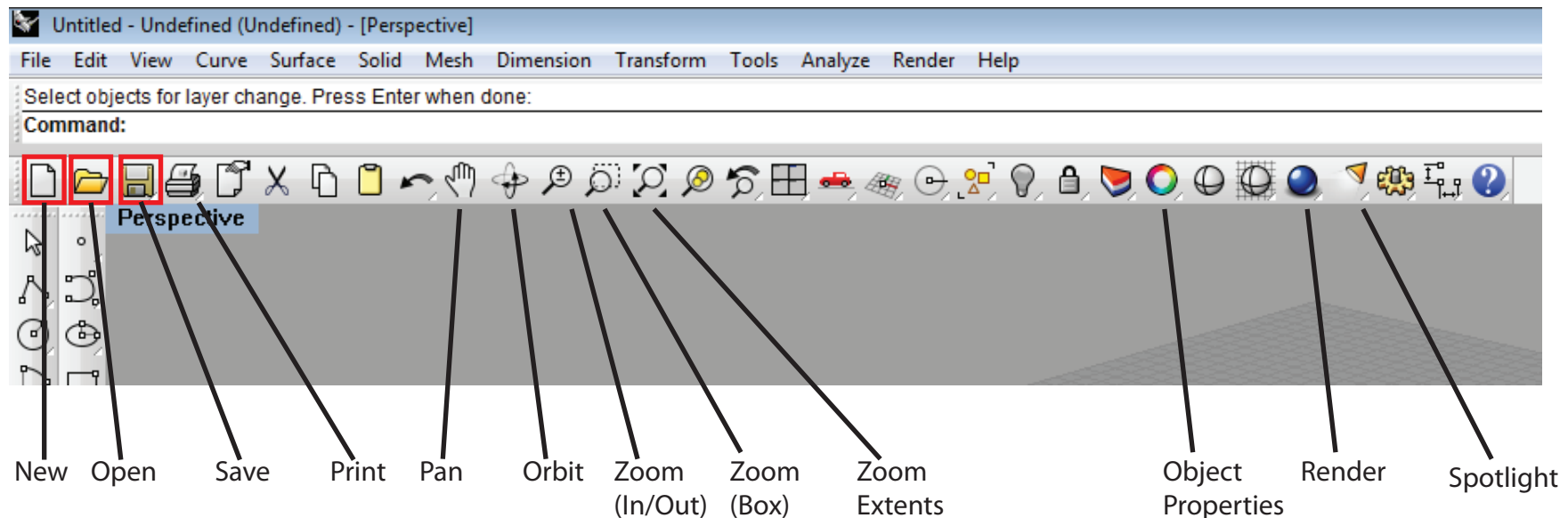


- Similar to AutoCAD, Rhino has layers that makes it easier to group objects and lines. However, these layers do not distinguish different lineweights.
- You can add or delete layers.
- If you want to change the layer of an object, highlight the object, then go to Edit ---> Layers ---> Change Object Layer
- A box listing the layers will show up, and you can change the layer of the object by clicking on the desired layer.





# Shortcuts



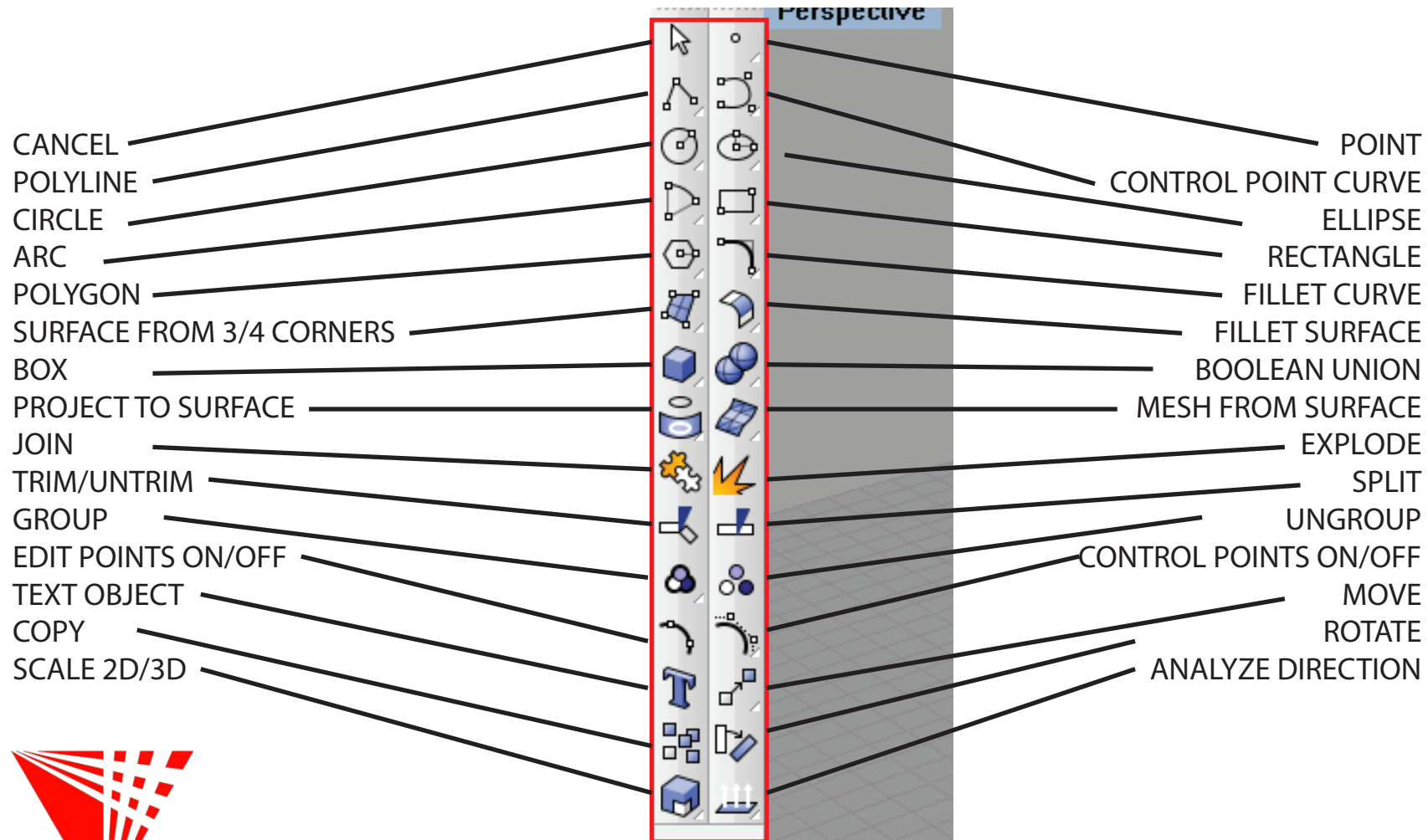
This simple shortcuts, located below the command line, can become useful when trying to open or save a document.

You can also use zoom, pan and orbit to modify the views of your project, and after becoming familiarized with the program, you can render you project, (generate an image using a computer program).

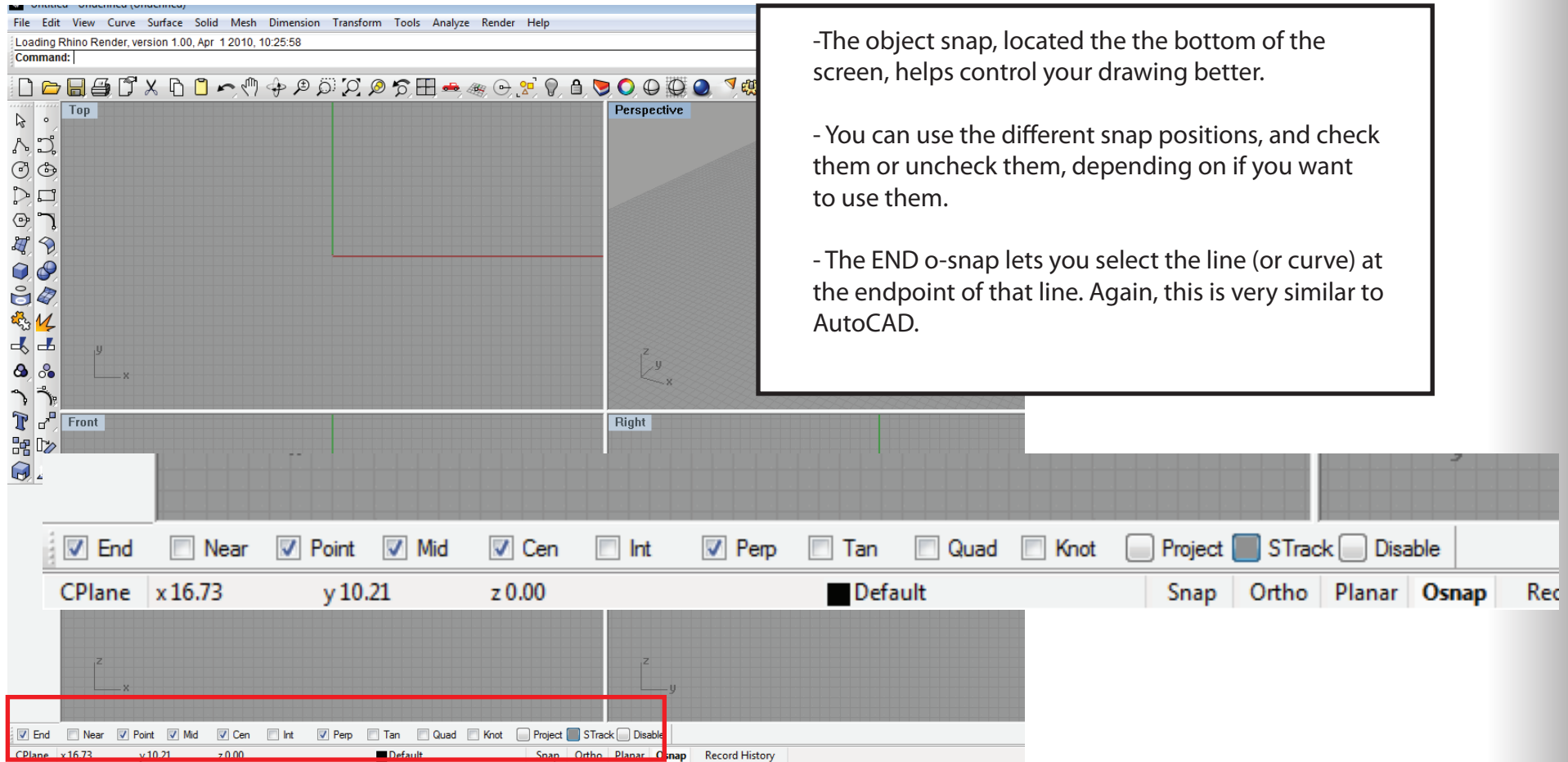


# Generic Commands

The general commands are located on the left side of the screen. It is very similar to AutoCAD in terms of commands available, including a few extra commands that are more useful for renderings and 3D objects.



# Object Snaps

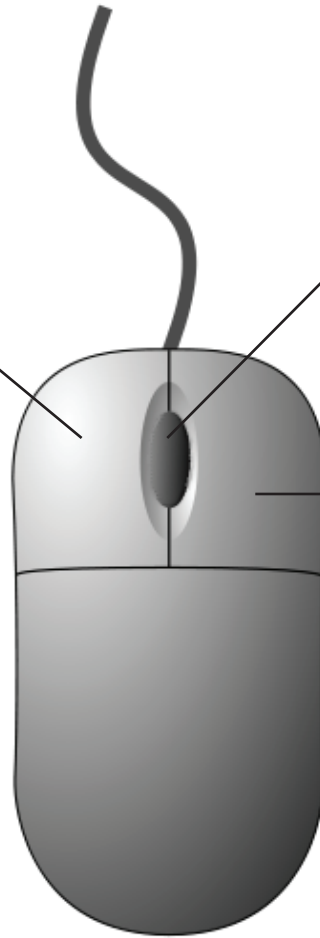


# Using the Mouse

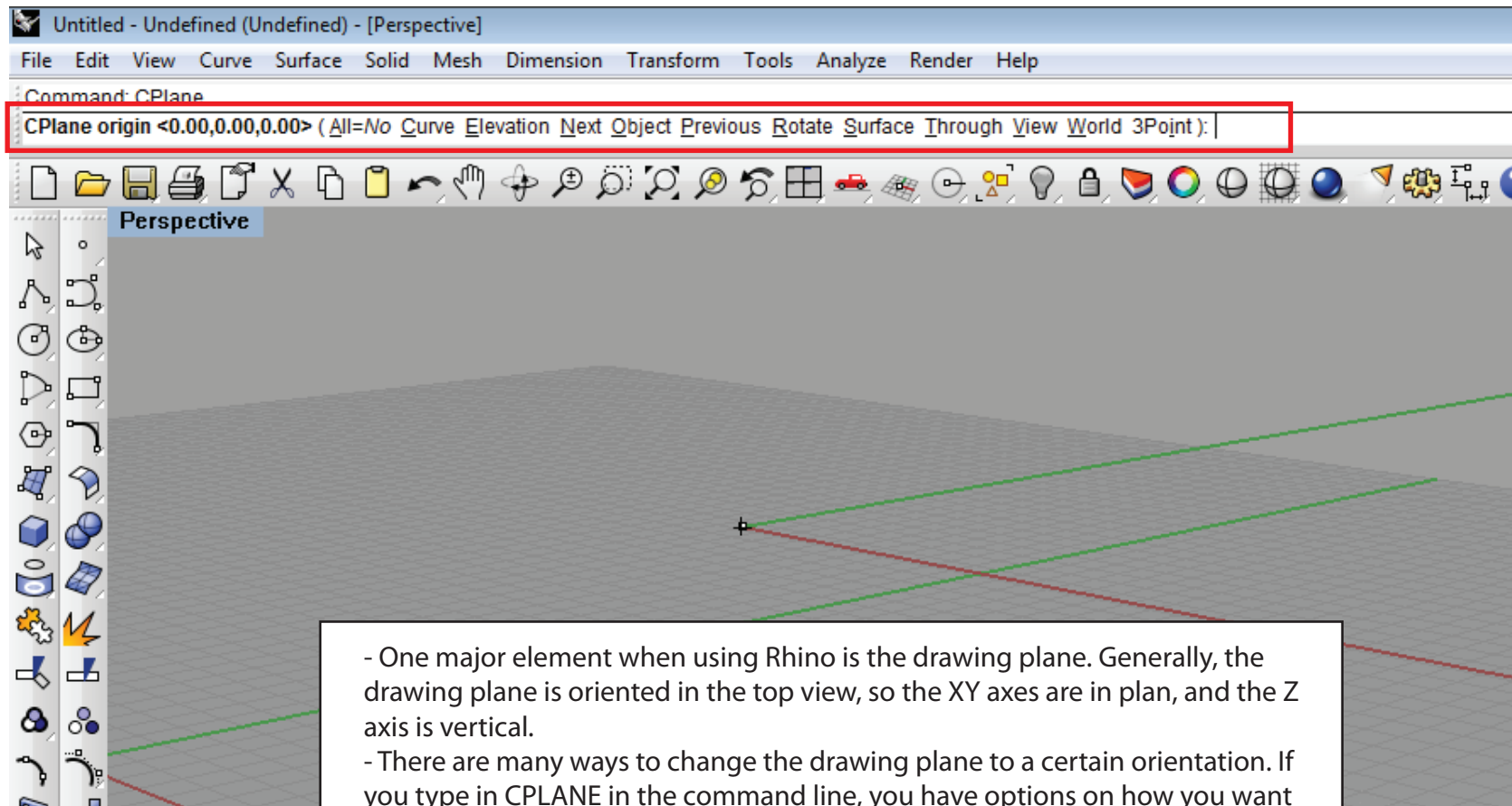
Just clicking on the left side of the mouse, it will create a selection box to highlight your drawing.

The middle scroll zooms in and out of the drawing, like in AutoCAD. Clicking on the scroll with create a popup box that gives you a list of some useful commands.

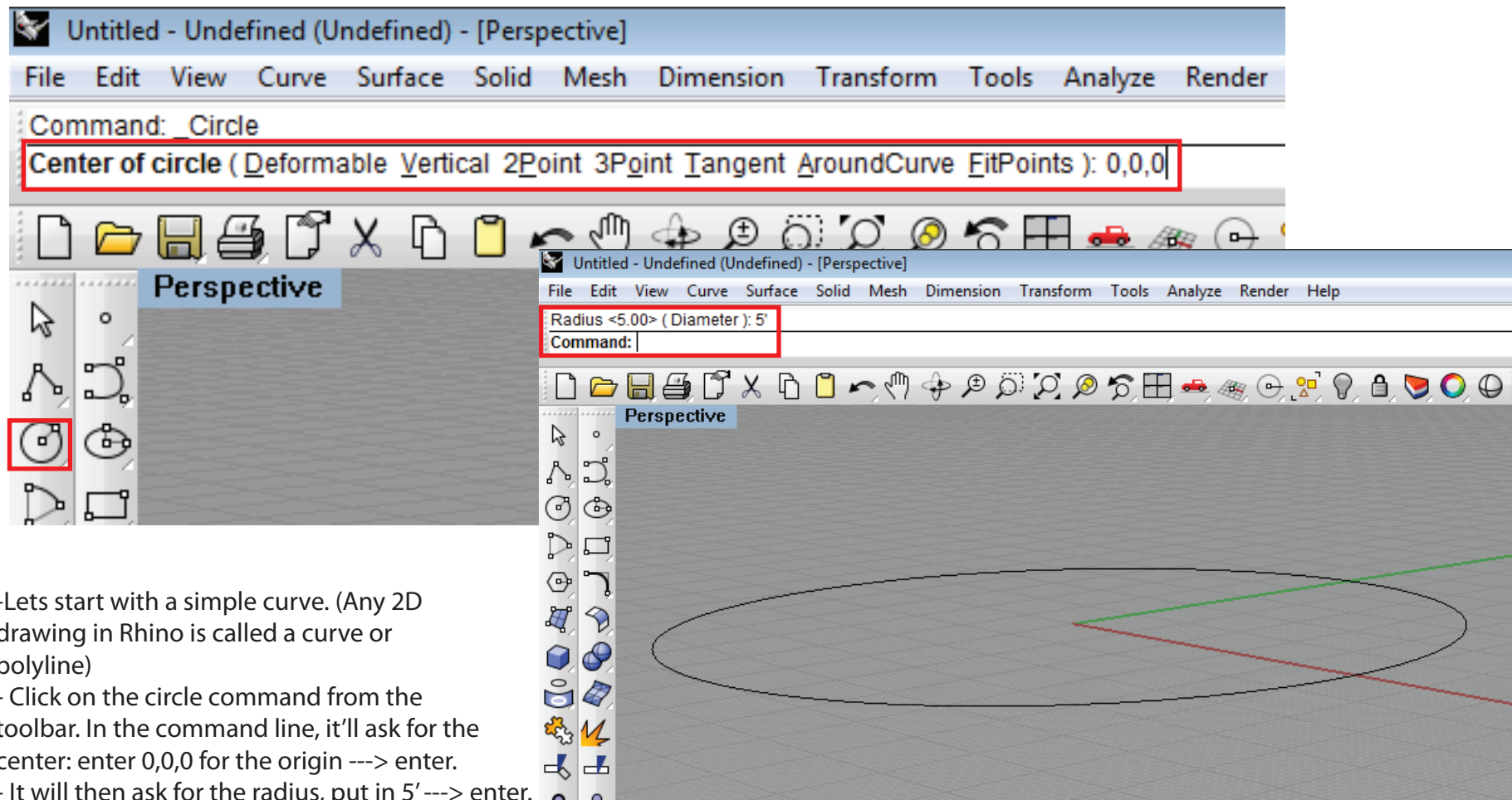
The right click lets you orbit the model space in the Perspective viewport, and Pan in the other viewports. Shift+Right Click allows you to Pan in all the viewports. CTRL+Right Click allows you to zoom in and out of the model space.



# Planes



# Starting a 2D Drawing

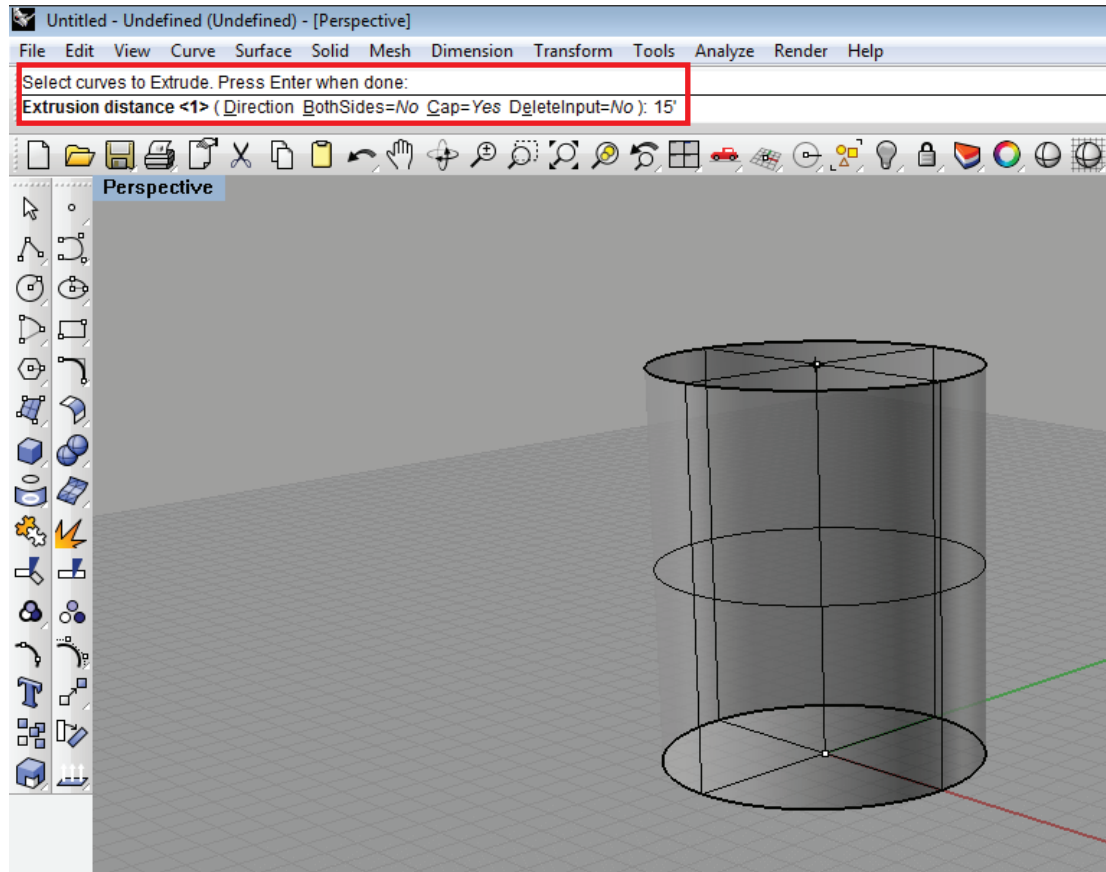


- Lets start with a simple curve. (Any 2D drawing in Rhino is called a curve or polyline)
- Click on the circle command from the toolbar. In the command line, it'll ask for the center: enter 0,0,0 for the origin ---> enter.
- It will then ask for the radius, put in 5' ---> enter.





# 3D Construction



There are many ways create a cylindrical shape. The loft and extrude commands are very common commands in 3D construction. The loft command requires 2 or more closed curves, while the extrude command only requires one.

**LOFT:** If you want to loft a 3D object, which essentially means combining to shapes and creating a surface between them, aligning the corners to your preference, you will need 2 curves. You can copy the circle, move it vertically and type loft in the command bar. Then you select the curves, enter for all the defaults, and you should end up with a 3D object.

**EXTRUDE:** The extrude command is fairly simpler. Type in ExtrudeCRV in the command bar ---> select the circle ---> Enter ---> Type C (for a cap on the extrusion) ---> Enter 20' for distance ---> Enter

You should end up with a cylindrical shape, similar to the one on the left.



# 2D and 3D Commands

## 2D Commands

- o Line
- o Rectangle
- o Circle
- o Polyline
- o Trim
- o Extend
- o Offset
- o Move
- o Mirror
- o Scale
- o Rotate

## • 3D Commands

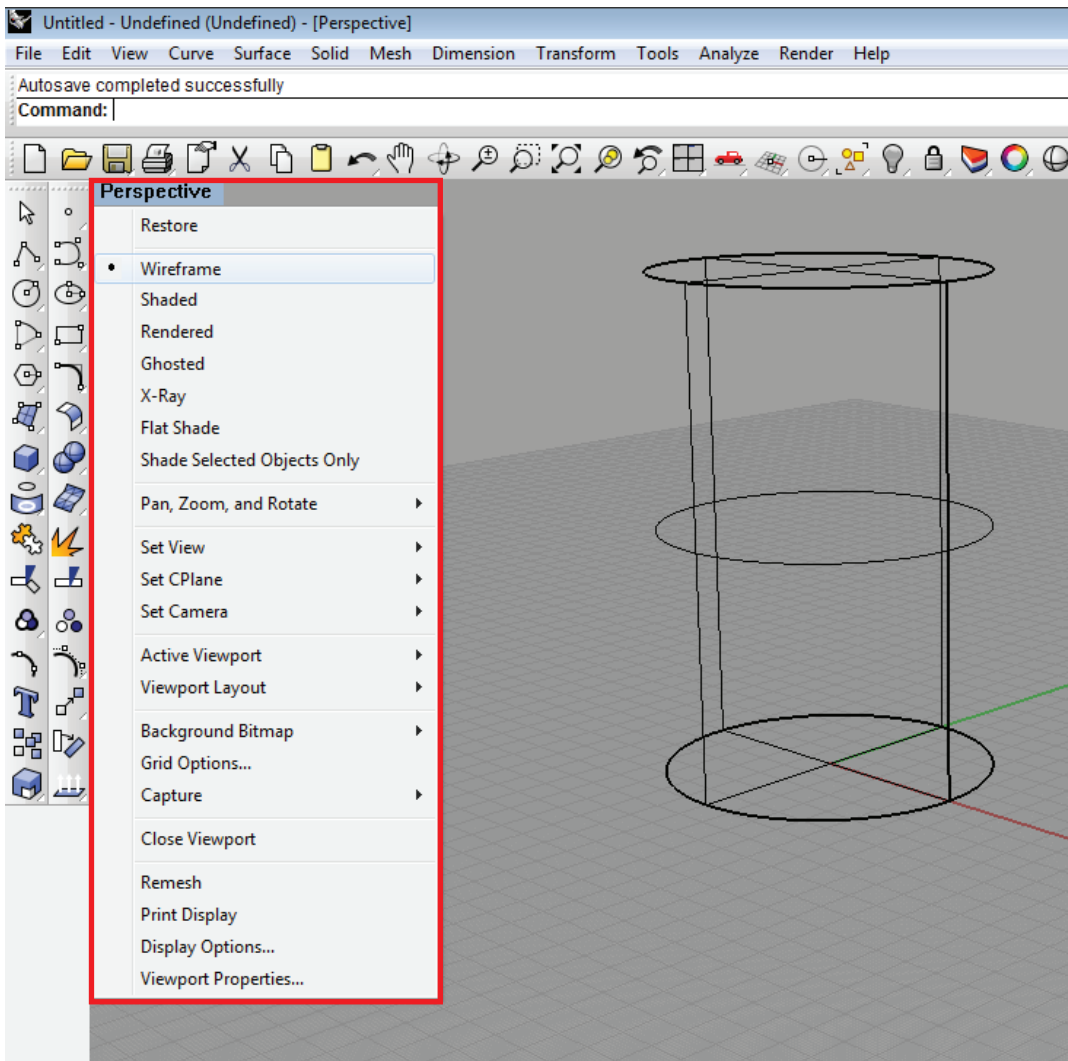
- o Extrude
- o Union
- o Subtract
- o 3D Rotate
- o Boolean

## • Helpful Commands

- o Units
- o Properties
- o Measure
- o Dimension



# Views: Face Style



You can also look at your 3D image in different views of face-styles.

If you right-click on the Perspective Box, a popup window will show up, and list the different face-styles you want to present your image as.

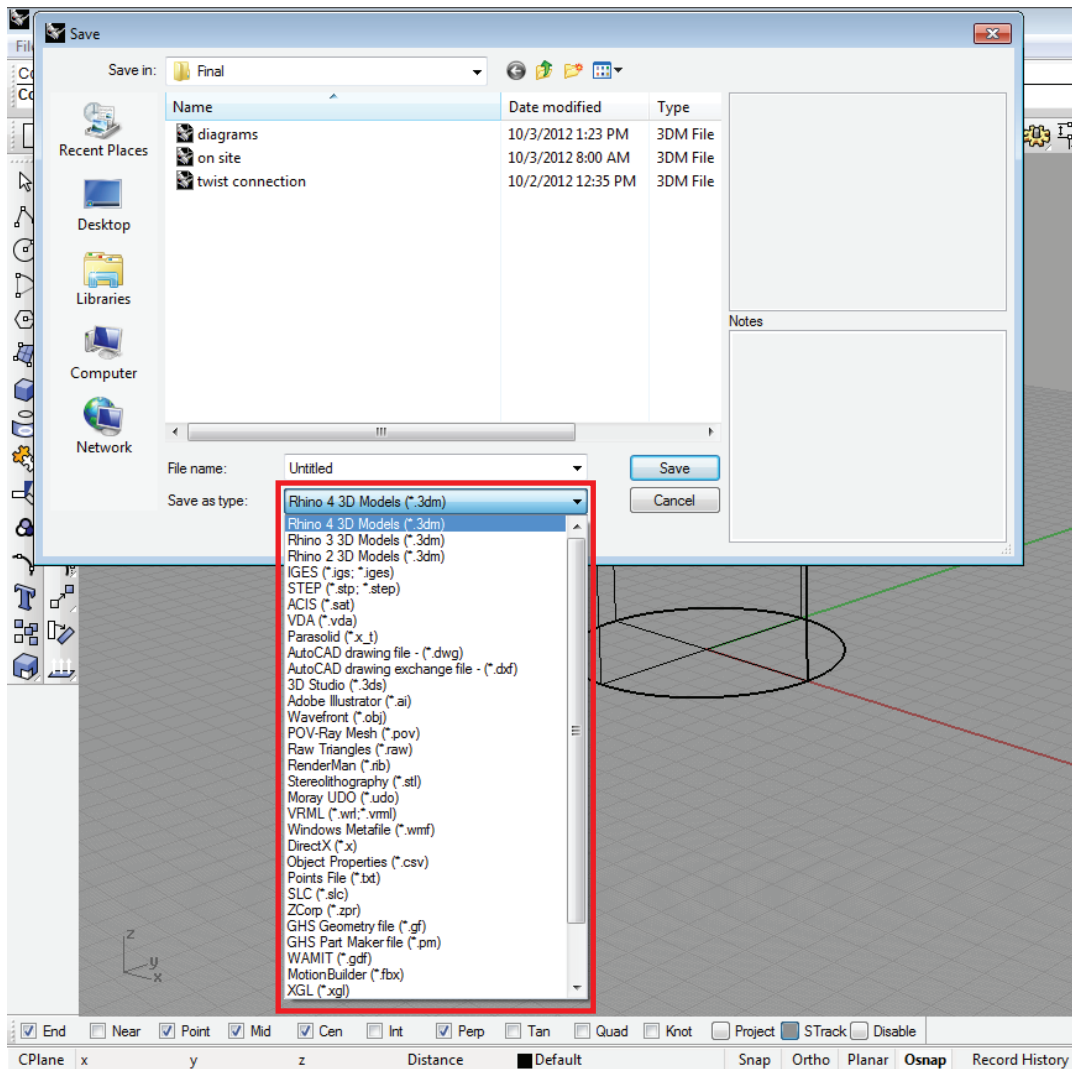
Different Styles:

- Wireframe
- Shaded
- Rendered
- Ghosted
- X-Ray
- Flat Shade

You can also change the CPlane, Views, and Camera from this pop-up window.



# Exporting

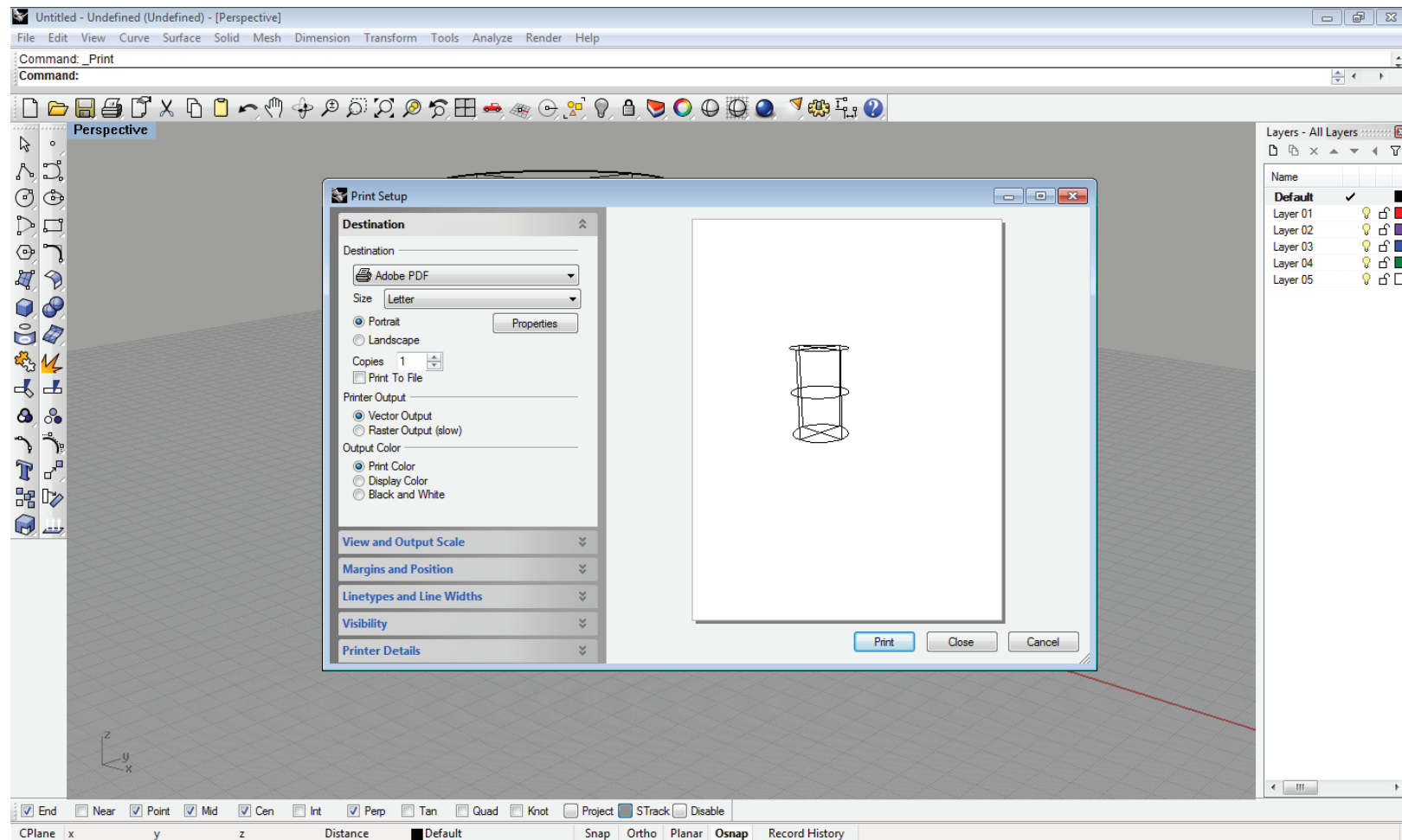


You can export your drawing, or save it as a specific file and then import this file into other programs, such as Revit, AutoCAD, 3DsMax, Falcon, Blender, and many more.

If you go to File ---> Save As ---> Scroll down to save file type as ---> Choose the correct format you want to save it as in order to open it in that certain program.



# Printing



When printing, go to File ---> Print ---> Adjust all the properties and destination of printing, as you would do in AutoCAD or any other program. There are no lineweights in Rhino, so you would need to adjust that in AutoCAD or photoshop.



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# More Topics to Investigate

Rhino has lots of options and possibilities that you should explore on your own

- o Properties (Object and Rendering)
- o Interface options
- o Complex shapes
- o 3D modeling
- o Rendering
- o External References



- To download student versions or trials, go <http://www.rhino3d.com/download.htm>
- You can also find tutorials and help on their website.
- Questions?

