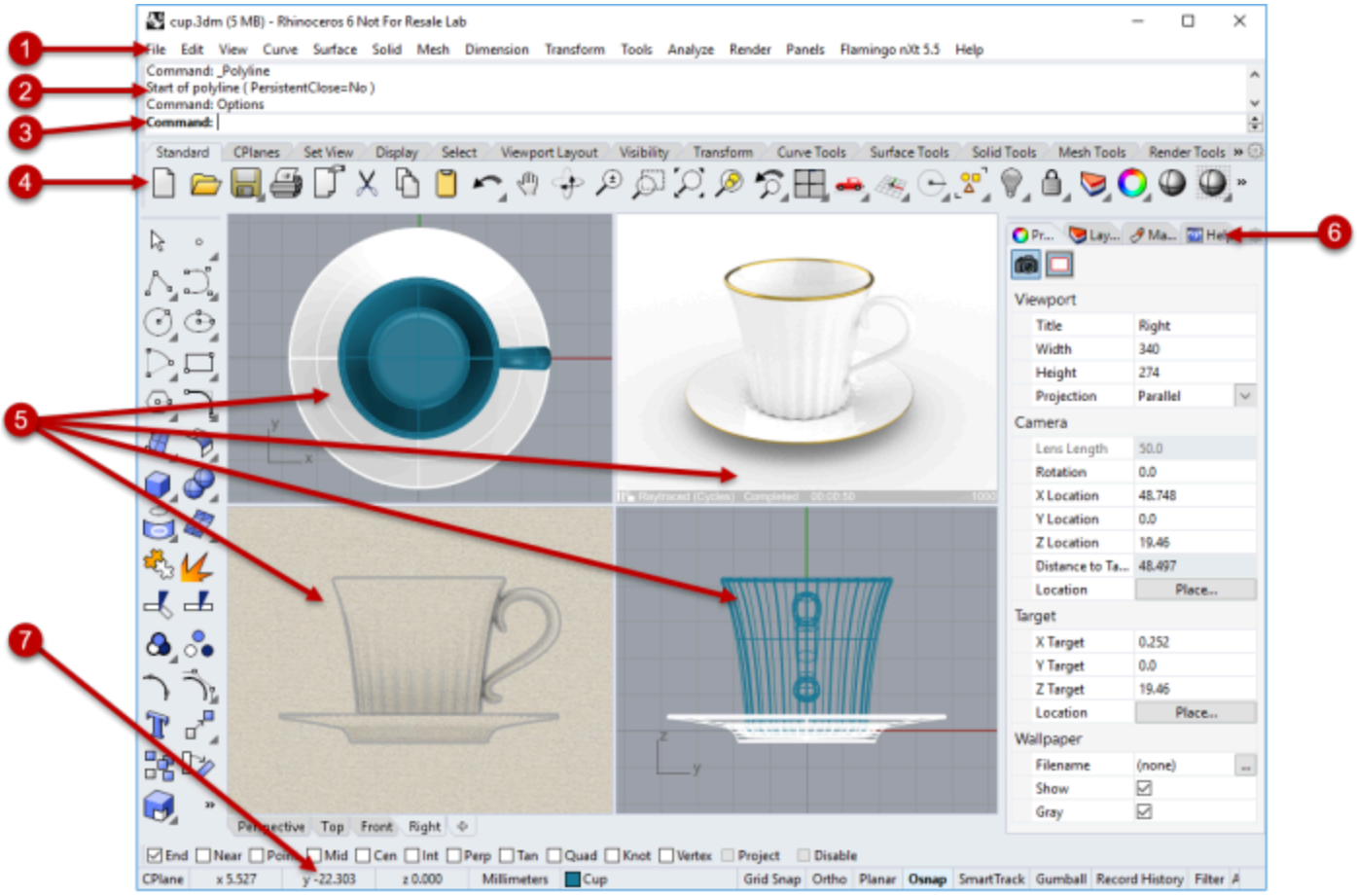
Rhino User's Guide for Windows

1 - Introduction

Modeling in 3-D is the process of creating a mathematical representation of an object's surfaces. The resulting model is displayed on your screen as a two-dimensional image. Rhino provides tools for creating, displaying, and manipulating these surfaces.

The Rhino interface

The image below illustrates some of the major features of the Rhino window.



Menu (1)

The menus group Rhino commands by function.

History window (2)

The command history window displays the previous commands and prompts.

Command prompt (3)

The command prompt displays prompts for the current command actions, options for the command that you can click, and allows typing command names and options.

Toolbars (4)

Toolbars contain graphical icons for initiating commands. Many toolbar icons have a second command that you can access by right-clicking the icon. The tooltip that appears when you hover over the icon tells you what the left and right mouse button do.

To access the command on the first line

- Click the icon with the **left mouse button**.

To access the command on the second line

- Click the icon with the **right mouse button**.

Viewports (5)

Viewports display the Rhino working environment.

Panels (6)

Tabbed panels contains layers, properties, and other settings.

Status bar (7)

The status bar is located at the bottom of the Rhino window. It displays the current coordinate system, the cursor location, and system unit. It also provides quick access to layers and toggles of modeling aids.

Rhino commands

Rhino is a *command-driven* program. In other words, all actions are activated by named commands such as **Line**, **Box**, or **CurvatureAnalysis**.

Tip: To learn more about a command, use the Command Help panel. Or, run the command and press **F1** to open its help topic.

Commands are accessed through the *menus*, or the *toolbars*, or by *typing* the command name. In the next sections, you will explore using these methods. You may find one method easier than another. The choice is yours, and there is no preference for one method over another.

In the exercises, you will use Rhino's commands, navigation tools, shaded modes, render, and use some basic object manipulation.

Tip: To cancel a command any time, press the **Esc** key.

In this session you will:

- Start a command by choosing from the menu.
- Start a command by choosing from a toolbar icon.
- Start a command by typing.

To start your first Rhino model

1. Start **Rhino**.
2. On the **File** menu, click **New**.
3. In the **Open Template File** dialog box, select **Small Objects - Centimeters.3dm**, and click **Open**.

Using templates

A *template* is a file that serves as a starting point for a new Rhino model. When starting a new model, Rhino provides default templates that set units appropriate for the model size, provide a few layers, and display a standard four-viewport layout.

Custom templates can include any kind of document information such as units, viewport layouts, layers, and even geometry. If you find yourself creating similar models over and over, set up and save a model as a template (see the **SaveAsTemplate** command in the Help). That way you will not have to spend time formatting your models each time you make a new one. You can start a new model from your own template, and start from there.

Start from the menu

Most Rhino commands are located in the menus.

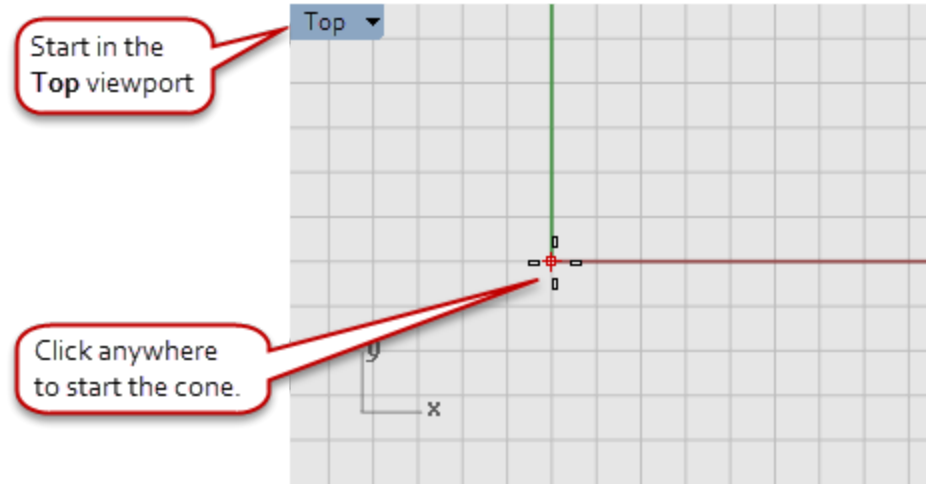
Start the Cone command

- On the **Solid** menu, click **Cone**.

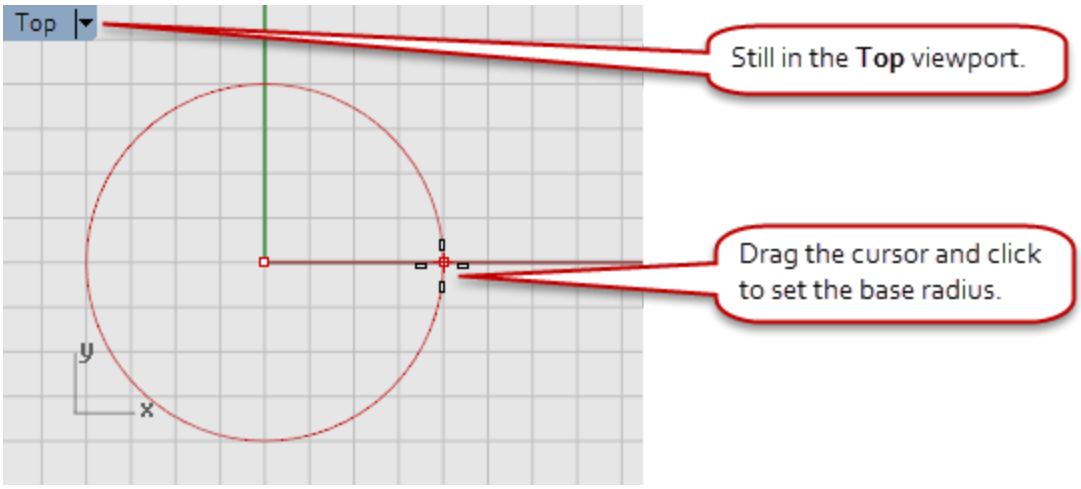
  For Help with the Cone command, press **F1** now or at any time during the command sequence.

Draw the cone

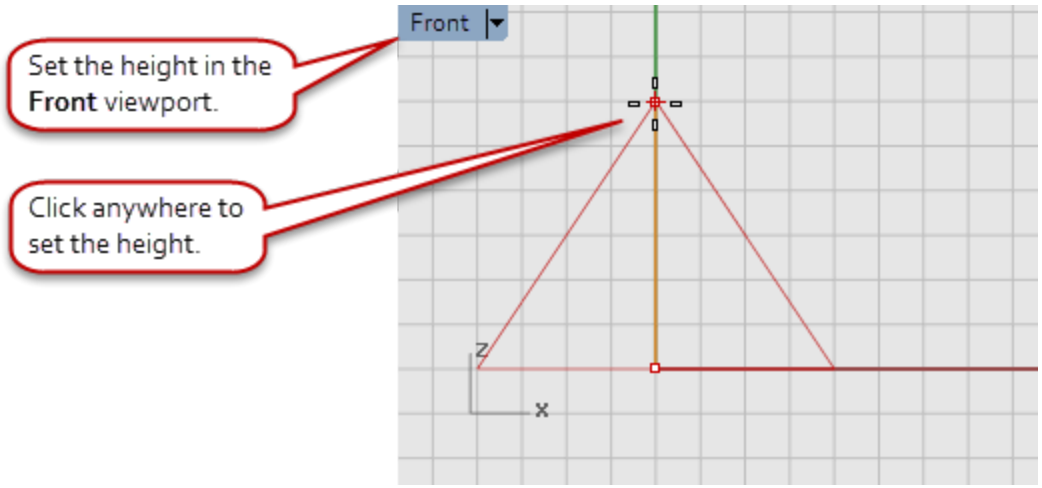
1. At the **Base of cone…** prompt, in the **Top** viewport, click the left mouse button to pick the center point for the base of the cone.



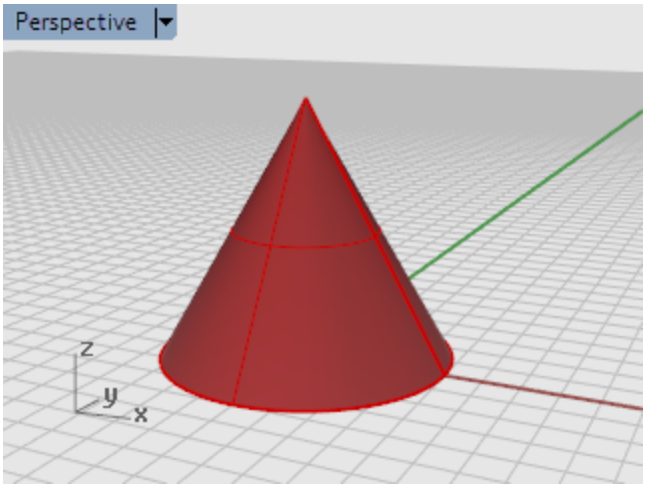
2. At the **Radius…** prompt, in the **Top** viewport, drag the mouse, and click to draw the cone's base.



3. At the **End of cone** prompt, in the **Front** viewport, drag the mouse up and down. Watch what happens in the **Perspective** viewport.



4. Click to place the end of the cone.

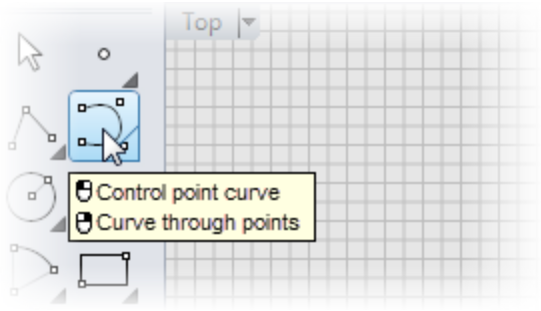


Start from a toolbar icon

Toolbars provide a graphic interface to the commands.

To display a button tooltip, hover your mouse over the icon

- The command names that are activated with a mouse click appear.
  The tooltip shows the commands assigned to the left and right mouse buttons .



Start the Curve command

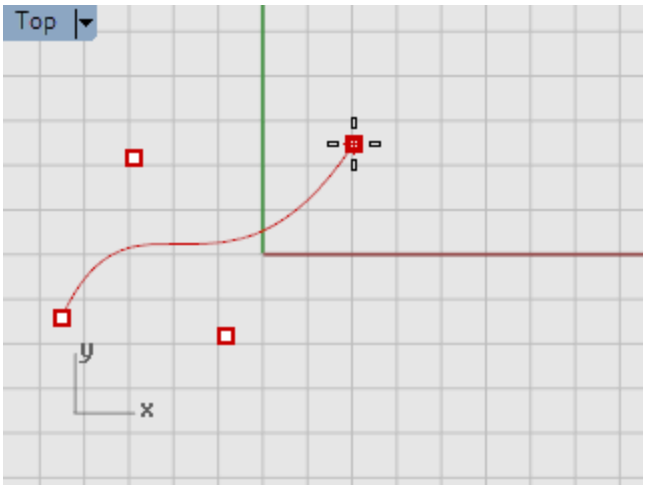
- On the toolbar docked on the left side of the Rhino window, click the icon for **Control point curve**.

  For Help with the Curve command, press **F1** now or at any time during the command sequence.

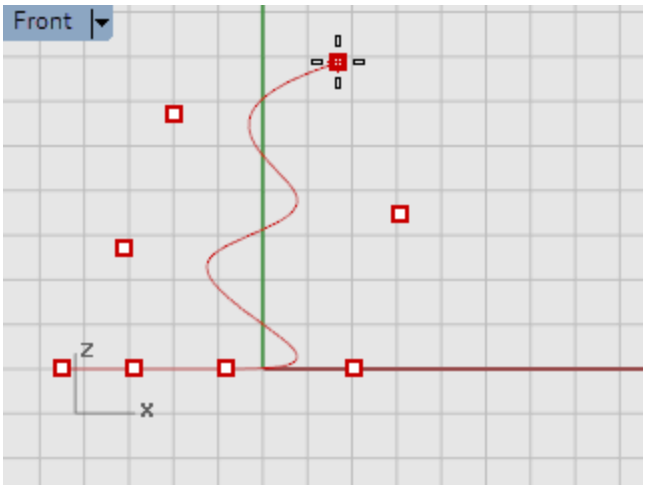
Draw the curve

1. At the **Start of curve…** prompt, in the **Top** viewport, click the mouse to start the curve.

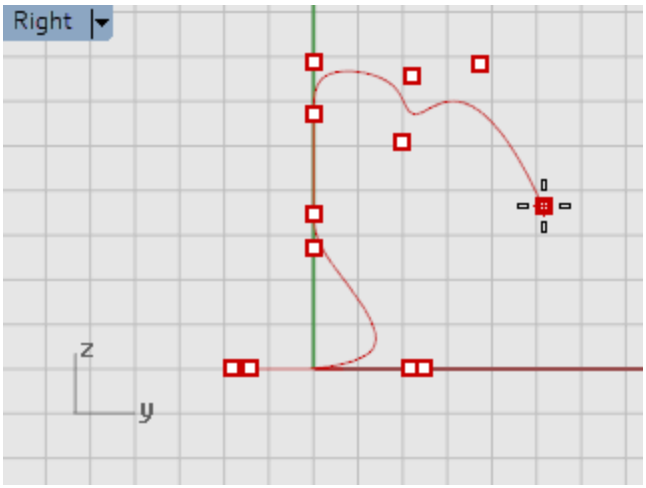
2. At the **Next point...** prompts, in the **Top** viewport, click a few more points.



3. At the **Next point...** prompts, move the mouse into the **Front** viewport and click a few more points.

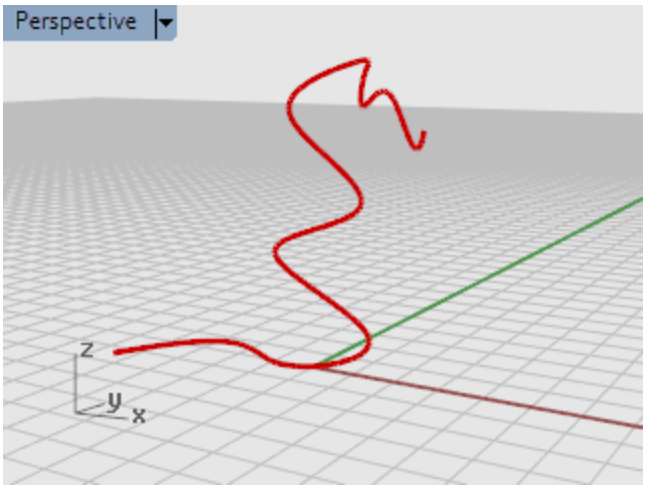


4. At the **Next point...** prompts, move the mouse into the **Right** viewport and click a few more points.



5. Right-click, press **Enter**, or press the **Spacebar** to complete the curve.

6. Look at the curve in the **Perspective** viewport.



Admire your work

- In the **Perspective** view, drag with the right mouse button to rotate the view.

Start from the command line

You can start a command by typing the command name.

Start the Sphere command by typing

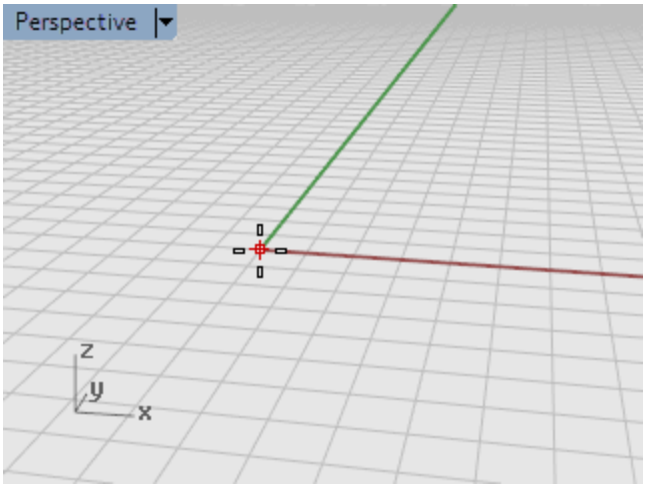
1. With the command prompt clear (if there is text at the command prompt, press the **Esc** key), just start typing **Sphere**.
   When you type the first letter of a command, a list of possible commands appears. The command most used in the past automatically completes on the command line.
2. When the command name **Sphere** appears, press **Enter**, or choose **Sphere** from the list.

   For Help with the Sphere command, press **F1** now or at any time during the command sequence.

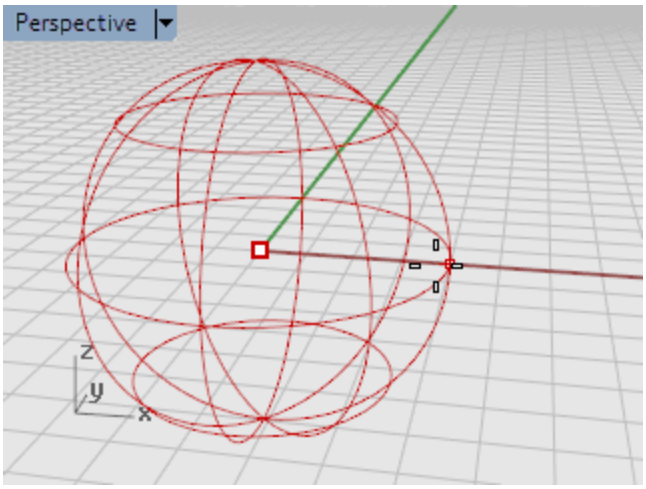
Draw the sphere

The default option for the **Sphere** command is **Center, Radius**, so you can simply start drawing the center of the sphere.

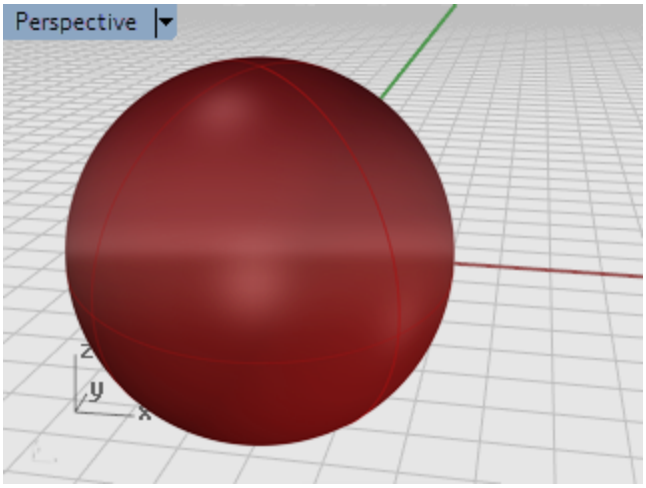
2. At the **Center of sphere…** prompt, in the **Perspective** viewport, click the mouse anywhere in the viewport to pick the center point for the sphere.



3. At the **Radius…** prompt, in the **Perspective** viewport, move the mouse away from the center point, and click to draw the sphere.



4. In the **Perspective** viewport, right-click the **viewport title**, and on the menu, click **Shaded**.
   You can also use the **View** menu to set the viewport to **Shaded** mode.
   Shaded mode lets you see surfaces as opaque objects rather than wires.



For more information on display modes

1. On the Rhino **Tools** menu, click **Options**.
2. Under **Rhino Options**, click **View** and then click **Display modes**.
3. In the **Display Modes** dialog box, click the **Help** button.

Undo a mistake

If you did something you did not want to do, you can undo your actions.

Undo a command

- On the **Edit** menu, click **Undo**.
  Or press the **Ctrl** and **Z** keys.

  You can undo a series of commands, or **Redo** the **Undo** if you go too far.

Redo commands

- On the **Edit** menu, click **Redo**.
  Or press the **Ctrl** and **Y** keys.

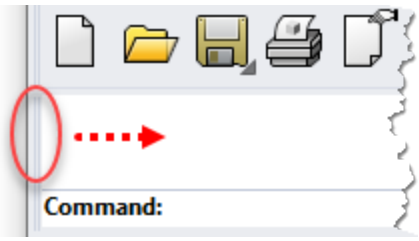
The command window and command prompt

The command window contains the command history and the command prompt.

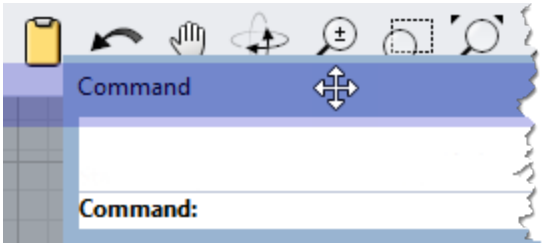
The window is normally docked at the top of the Rhino window, but you can float the command window just like any toolbar. You can dock it at the top or bottom of the Rhino window, above, below, or next to the main toolbar or the object snap toolbar.

To float or dock the command window

1. Drag the gripper located at the left leaving the Rhino window edge to make the command window floating.



2. Drag the command window title bar to where you would like to dock it.
   As you drag the window, a blue shaded area appears at locations where the window can be docked.



It is a good idea to size the window so it displays at least three lines of text.

To resize the command window

1. Move your mouse to the edge of the command window until the up and down arrow appears.
2. Click the mouse and drag the window edge until at least three lines of text are visible.

The *command prompt* is where you can type command names, set options, enter distances and angles for drawing, and read the prompts for the commands.

The command prompt displays *options* for the command in parentheses. To activate an option, click the option with the mouse or type the option name or the underlined letter in the option.

Options give you alternate methods for using the command. Prompts are messages to you to select objects to act on, to enter information, or to enter a point on the screen. As you start to build objects, prompts become increasingly important, since they tell you what to do next.

Command options

Command options change how a command acts. For example, when you draw a circle, the circle is normally drawn on the active construction plane. The **Circle** command has several options including **Vertical** and **AroundCurve**.

To choose a command option

1. Start typing **Circle**.
   As soon as you have typed enough letters to identify the command, the **Circle** command automatically completes at the prompt.
2. Press **Enter**.

   The options for the **Circle** command appear:

   Center of circle (Deformable Vertical 2Point 3Point...)

3. Click **Vertical**, or type **V** to draw a circle vertical to the active construction plane.

Repeat the last command

Many tasks in Rhino are repetitive. You might want to move or copy several objects, for example. Methods for repeating commands are provided.

To repeat the last command

These all perform the same function:

- Press the **Enter** key when no command is active.
- Press the **Spacebar**.
- Click the **right mouse button** in a viewport.

Note: Some commands, such as **Undo** and **Delete** do not repeat. Instead, the command prior to these commands is repeated. This prevents you from accidentally undoing too many commands or deleting objects accidentally.

Also, you often want to repeat the command you were using before undoing a mistake. To suit your own way of working, you can define the list of commands that do not repeat in **Tools > Options > General > Never repeat these commands**.

Cancel a command

Whenever you run the wrong command or do something wrong in the command procedure, you can cancel the command several ways.

To cancel the current command

When a command is running:

- Press the **Esc** key.
  The command line is cleared and returns to the **Command** prompt.
- Click a toolbar button or click a command on a menu.
  The current command is canceled immediately and the chosen command runs.

Get help any time

The *Rhino Help* file is the major resource for detailed information on specific commands.

To get help on a specific command

- To get **Help** for a command, press **F1** while the command is running.
  The Help window appears with the specific command topic visible.
- On the **Help** menu, click **Command Help**.
  The Rhino **Help** will display in a dockable panel.
  Click **Auto-Update** to dynamically display the Help topic for the current command.
- Find the answers to frequently asked questions at: *Rhino Support*.

To download a PDF version of this guide

1. On the Rhino **Help** menu, click **Learn Rhino**, and then click **Tutorials and Samples**.
2. In the **Tutorials** dialog box, under **User's Guide**, select **Rhino User's Guide for Windows.pdf**.