Chapter 12: Pull Toy - Solids and Transforms

This tutorial demonstrates using solid primitives and simple transforms. You will learn how to:
- Enter coordinates to place points exactly.
- Draw a free-form curve and polygon.
- Create a pipe along a curve.
- Use a polar array to copy objects in a circular pattern.
- Extrude a curve to create a surface.
- Use planar mode.

Enter coordinates

When you pick a point with the mouse, the point lies on the construction plane of the active viewport unless you use a modeling aid such as object snap or elevator mode. When Rhino prompts for a point, you can enter x-, y-, and z-coordinates instead of picking a point. Each viewport has its own construction plane on which its x- and y-coordinates lie. The z-coordinate for the active viewport is perpendicular to the x-y plane. The grid is a visual representation of the construction plane. The intersection of the dark red and green lines shows the location of the origin point (x=0, y=0, z=0) of the coordinate system.

Draw the pull toy body

This exercise uses x-, y-, and z-coordinates to accurately place points. When you are to type coordinates, type them just as they are shown in the manual. The format is x,y,z. For example, type 1,1,4. You must type the commas. This sets the point at x=1, y=1, and z=4 in the active viewport. Whenever you type points, look in all viewports at where the point is placed so you can start getting an idea of how coordinate entry works.

Note: Pay close attention to the viewport required in each instruction.
Start the model

1. Begin a **New** model.
2. In the **Open Template File** dialog box, select **Small Objects - Centimeters.3dm**, and click **Open**.

Draw an ellipsoid

1. Turn on **Ortho**.
2. On the **Solid** menu, click **Ellipsoid > From Center**.
3. With the **Top** viewport active, at the **Ellipsoid center...** prompt, type **0,0,11**, and press **Enter**.
   This places the center point of the ellipsoid at x=0, y=0, and z=11. Look at the point in the **Perspective** viewport.
4. At the **End of first axis...** prompt, type **15**, and press **Enter**.
5. Move the cursor to the right to show the direction and click.
6. At the **End of second axis** prompt, type **8**, and press **Enter**.
7. Move the cursor up to show the direction and click.
   This sets the width of the ellipsoid.
8. At the **End of third axis** prompt, type 9, and press **Enter**.
   You now have an egg shape that has different dimensions in all three directions.

![3D egg shape with different dimensions in all three directions.](image)

9. Rotate the perspective viewport so you are looking along the x-axis as illustrated.
   Turn on **Shaded** display mode in the **Perspective** viewport.

![Perspective viewport with shaded display mode enabled.](image)

**Draw the axles and wheel hubs**

The axles and wheel hubs are cylinders. The axles are long, thin cylinders, and the wheel hubs are short, fat cylinders. You are going to make one axle and one complete wheel. You will then mirror the complete wheel to the other side. You can then either mirror or copy the complete axle and wheel set to the front of the toy.
Create the axle

1. On the Solid menu, click Cylinder.

2. With the Front viewport active, at the Base of cylinder... prompt, for the location of the cylinder's center, type 9,6.5,10, and press Enter.

3. At the Radius... prompt, type .5, and press Enter.

4. At the End of cylinder prompt, type -20, and press Enter.
Create a wheel hub

1. On the Solid menu, click Cylinder.
2. With the Front viewport active, at the Base of cylinder... prompt, type 9, 6.5, 10, and press Enter.
3. At the Radius... prompt, type 4, and press Enter.
4. At the End of cylinder prompt, type 2, and press Enter.
**Draw the lug nuts**

You will make the lug nuts by extruding a hexagonal polygon curve.

---

### Create a hexagon

1. On the **Curve** menu, click **Polygon > Center, Radius**.
2. At the **Center of inscribed polygon ( NumSides=4... )** prompt, type **6**, and press **Enter**.
3. In the **Front** viewport, at the **Center of inscribed polygon...** prompt, type **9,8,12**, and press **Enter**.
   
   This will place the polygon right on the surface of the wheel hub.
4. At the **Corner of polygon...** prompt, type **.5**, and press **Enter**.
5. In the **Front** viewport drag the cursor as illustrated, and click to position the hexagon.

---

### Make a solid from the polygon

1. In any viewport, **select** the hexagon you just created.
2. On the **Solid** menu, click **Extrude Planar Curve > Straight**.
3. At the **Extrusion distance** prompt, notice the command-line options. Set the options as follows:
   
   - **Direction** - use default
   - **BothSides** = **No**
   - **Solid** = **Yes**
   - **DeleteInput** = **Yes**
   - **ToBoundary** - use default
   - **SplitAtTangents** = **No**
   - **SetBasePoint** - use default

   If the option is not set as listed above, click the option to change it.
4. At the **Extrusion Distance**... prompt, type -.5 (Notice the negative number. If you type a positive number at this point, the nuts will be buried in the wheel hub. You want them to stick out.), and press **Enter**.

---

**Array the lug nuts**

To create the lug nuts on the first wheel, you are going to use a polar (circular) array. An array is a set of copies of an object. You control how the copies are made. A polar array copies the objects around a central point. The objects are rotated as they are copied.

**Array the nuts**

1. **Select** the lug nut.
2. On the **Transform** menu, click **Array > Polar**.
3. With the **Front** viewport active, at the **Center of polar array** prompt, use the **Cen** object snap to snap to the center of the hub.

---

4. At the **Number of elements**... prompt, type 5, and press **Enter**.
5. At the **Angle to fill <360>** prompt, press **Enter**.
6. At the Press Enter to accept prompt, check the preview, and press Enter.

Draw the tires

The tires are a solid form called a torus, which looks like a donut. When you are drawing a torus, the first radius is the radius of a circle around which the “tube” is drawn. The second radius is the radius of the tube itself.

To draw the tires, you will draw the center of the torus tube a bit larger than the diameter of the wheel hub. The tube itself is slightly larger than the hub. This makes it dip into the hub.

Create a torus for the tires

1. On the Solid menu, click Torus.
2. In the Front viewport, at the Center of torus... prompt, type 9,6,5,11, and press Enter.
   This places the center of the torus at the same point as the center of the wheel hub.

3. At the Radius... prompt, type 5, and press Enter.
   This makes the radius of the torus tube one unit bigger than the wheel hub.
4. At the **Second radius...** prompt, type **1.5**, and press **Enter**.
   This makes the inner dimension of the torus tube .5 units smaller than the wheel hub.

**Mirror the wheels**

Now that you have a whole wheel created, you can use the **Mirror** command to create the other three.

1. **Mirror the wheel to the other side**
   1. In the **Top** viewport, use a window to select the wheel as illustrated.
   2. On the **Transform** menu, click **Mirror**.
   3. At the **Start of mirror plane...** prompt, type **0,0,0**, and press **Enter**.
4. At the **End of mirror plane...** prompt, with **Ortho** on, drag to the right in the **Top** viewport as illustrated and click.

**Mirror the front wheels and axle**

1. In the **Top** viewport, use a window to select the wheels and axle as illustrated.
2. On the **Transform** menu, click **Mirror**.
3. At the **Start of mirror plane...** prompt, type **0,0,0**, and press **Enter**.
4. At the **End of mirror plane...** prompt, with **Ortho** on, drag down in the **Top** viewport as illustrated and click.

---

**Draw the eyes**

You are going to draw a sphere for an eye and a smaller sphere for the pupil.

---

**Create an eye using a sphere**

1. On the **Solid** menu, click **Sphere > Center, Radius**.
2. At the **Center of sphere...** prompt, in the **Top** viewport, type **-12,-3,14**, and press **Enter**.

---

[Image of a 3D model with instructions for creating an eye using a sphere]
3. At the **Radius...** prompt, type **3** and press **Enter**.

![Image of a toy with a sphere at the center](image1)

**Create the eye pupil**

1. Repeat the **Sphere** command.
2. At the **Center of sphere...** prompt, in the **Top** viewport, type **-13,-4,15**, and press **Enter**.

![Image of a toy with a smaller sphere at the center](image2)

3. At the **Radius...** prompt, type **2** and press **Enter**.

![Image of a toy with a smaller sphere at the center](image3)
Mirror the eye

1. In the Top viewport, use a window to select the eye as illustrated.
2. On the Transform menu, click Mirror.

3. At the Start of mirror plane... prompt, type 0 (this is a shortcut for typing 0,0,0), and press Enter.
4. At the End of mirror plane... prompt, with Ortho on, drag to the left in the Top viewport as illustrated and click.
Chapter 12: Pull Toy - Solids and Transforms
Make the pull cord

To make the cord, you are going to draw a freehand curve using elevator and planar mode. When the curve is complete, use the Pipe command to make it a thick solid.

Set up the view

1. **Zoom** out in all the viewports; you are going to need some space to work.
2. On the status bar, turn **Planar** mode on, and turn **Ortho** off.
3. In the Osnap dialog box, click **Disable** to turn off all object snaps.

Create the pull cord at the front of the toy

1. On the Curve menu, click **Free-form > Control Points**.
2. At the **Start of curve...** prompt, in the Top viewport, hold the **Ctrl** key to activate elevator mode and click near the front end of the body ellipsoid.
3. Move the cursor to the Front viewport, drag the marker near the end of the ellipsoid, and click.
4. At the **Next point**... prompt, click to the left of the ellipsoid in the **Top** viewport.

Planar mode keeps successive points at the same construction plane elevation. Planar mode can be overridden with elevator mode or object snaps. Watch the curve in the **Top** and **Front** viewports.

5. At the **Next point**... prompt, use elevator mode to add another point in the **Top** viewport.

6. At the **Next point**... prompts, turn off Planar mode and click several more points in the **Top** viewport to create a curved line.
Notice that the points are projected to the **Top** construction plane.

**Make the cord handle**

1. Draw an **Ellipsoid** with the **Diameter** option to represent a handle at the end of the curve.
2. At the **Start of first axis** prompt, use the **End** object snap to pick the end of the cord curve.
3. At the **End of first axis** prompt, type **10** to set the length, and press **Enter**.
4. Drag the direction so it lines up with the cord curve and click to set the direction. This does not have to be very accurate.
5. At the **End of second axis** prompt, type 4, press **Enter**, and drag to set the direction.

6. At the **End of third axis** prompt, type 2, and press **Enter**.

---

**Thicken the curve with a pipe**

1. **Select** the curve you just made at the front of the pull toy.
2. On the **Solid** menu, click **Pipe**.
3. At the **Start radius...** prompt, type .2, and press **Enter**.
4. At the **End radius...** prompt, press **Enter**.
5. At the **Point for next radius** prompt, press **Enter**.
   The pipe will be the same diameter for the full length of the curve.