Chapter 19: Mechanical Part - Blocks

This example shows how to use Rhino to model a simple mechanical part. You will learn to

- Create extrusion objects.
- Boolean difference shapes.
- Drill holes.
- Create a 2-D line drawing from the 3-D shapes.
- Dimension the 2-D line drawing and modify dimension text.

Open the tutorial model
1. From the Rhino File menu, click Open.
2. Browse to the Tutorial Models folder that you downloaded with the User's Guide.
3. Click Toolblock.3dm and then click the Open button.

Create solid shapes
Start by creating two basic solid shapes from the profile curves on layers Profile-01 and Profile-02.

Set up the layers
- In the Layers panel, confirm that the Profile-01 layer is current.

Extrude the profile curve
1. On the Solid menu, click Extrude Planar Curve > Straight.
2. Turn on the End object snap.
3. At the Select curves to Extrude prompt, select the blue profile curve, and press Enter.

4. At the Extrusion distance prompt, set the command-line Solid and the DeleteInput options to Yes.

5. Click the end of the magenta construction line.

The extruded shape is a solid because it forms a closed volume in space.

Hide the solid

- Select the solid, on the Edit menu click Visibility > Hide.

Set up the layers

- In the Layers panel, make layer Profile-02 current.
**Extrude the profile curve**

1. **Select** the red profile curve.

2. On the **Solid** menu, click **Extrude Planar Curve > Straight**.

3. At the **Select curves to Extrude** prompt, set the command-line **Solid** and the **DeleteInput** options to **Yes**.

4. At the **Extrusion distance** prompt, in the **Front** viewport, drag the extrusion above the height of the blue curve and click.

The solid appears on the current red layer Profile-02.
Show the solid

- On the Edit menu click Visibility > Show.

Boolean the two solids

1. On the Solid menu, click Difference.
2. At the Select surfaces or polysurfaces to subtract from prompt, select the blue solid, and press Enter.
3. At the Select surfaces or polysurfaces to subtract with prompt, set the command-line DeleteInput option to Yes.
4. Select the red solid, and press Enter.

   The result will be a new solid or polysurface. A polysurface is a collection of surfaces that can be closed or open. A solid is a collection of surfaces that is closed.
Drill the holes

A construction circle is already in place for creating the first hole.

**Make holes in the solid**

1. **Select** the green circle as shown.

2. On the **Solid** menu click **Solid Edit Tools > Holes > Make Hole**.
3. At the **Select a surface or polysurface** prompt, select the blue polysurface.
4. At the **Cut depth point** prompt, drag the hole through the upper portion of the object.

5. Pick a point in **Front** view.
Copy the holes

After one hole is drilled, you can copy the others.

**Copy the holes**

Copy the three remaining holes that are aligned with the previous hole with this command.

1. In the Osnap toolbar, turn on the **Point** object snap.
2. On the **Solid** menu, click **Solid Edit Tools > Holes > Copy Holes**.

*Note: Copy Holes is actually the MoveHole command with the command-line Copy option set to Yes.*

3. At the **Select holes in one planar surface** prompt, select the first hole, and press **Enter**.

4. At the **Point to copy from** prompt, pick the point object in the center of the first circle.
5. At the **Point to copy to (Copy=Yes)** prompt, pick the point that makes the center of the next hole.
6. Repeat this for the two holes that are on the other side of the part.

![Diagram of a part with two holes on each side]

**Note:** Do not use the point in the center of the part.

---

**Create the round hole**

The center hole is different in that it does not pass entirely through the upper part of the blue solid. There is no reference circle to start from.

1. On the **Solid** menu, click **Solid Edit Tools > Holes > Round Hole**.
2. At the **Select target surface** prompt, select the top surface of the blue solid.
3. At the **Center point** prompt, set the command-line options as follows:
   - **Depth** = 0.5
   - **Diameter** = 0.312
   - **DrillPointAngle** = 180
   - **Through** = No
   - **Direction** = CPlaneNormal
4. Click the point object in the middle of the blue solid to finish creating the hole.

![Image of point object in the middle of the blue solid]

**Test the solid**

The resulting polysurface is a closed solid. A solid defines a closed volume in space. The Properties command will report if this part is a closed solid. The Properties command will give you information about the open/closed status of the object.

1. **Select** the part.
2. On the Edit menu, click **Object Properties (F3)**.
3. In the Properties panel, click the **Details** button.
   
   In the Object Description window, you will find the listing to confirm that the object is valid and closed.
   
   Geometry:
   
   Valid polysurface.
   
   Closed solid polysurface with 23 surfaces.
Make a 2-D drawing

The Make2D command generates 2-D lines from the 3-D solid.

Create a 2-D line drawing

1. Select the part.
2. On the Dimension menu, click Make 2D.
3. In the 2-D Drawing Options dialog box, under Drawing layout click 4 view (USA).
   Under Options, check the Show tangent edges and Show hidden lines boxes.
4. Click OK.

Dimension the 2-D drawing

Using the 2-D drawing, add dimensions for the part.

Set up the layers

1. In the Layers panel, make Dimensions layer current.
2. Turn off all layers except Dimensions and the Make2D layers.
3. In the Linetype column for the Make2D > hidden > lines layer, click Continuous.
4. In the Select Linetype window, select Dashed.

Set up the viewport

- Double-click the viewport title to maximize the Top viewport.

Dimension the part

1. On the Dimension menu, click Linear Dimension.
2. In the Osnap toolbar, turn on the End object snap; turn off the Point object snap.
3. At the First dimension point prompt, pick the upper left corner of the part.
4. At the Second dimension point prompt, pick the upper right corner of the part.
5. At the Dimension location prompt, pick a location for the dimension line.
6. Repeat to generate a vertical dimension on the right side of the part.

---

**Chain dimension the part horizontally**

1. On the Dimension click **Linear Dimension**.
2. At the **First dimension point** set the command-line **Continue** option to **Yes**. This will generate a chain of dimensions.
3. At the **First dimension point** prompt, pick the lower left corner of the part.
4. At the **Second dimension point**, prompt turn on the **Cen** object snap and pick the center of the first circle.
5. At the **Dimension location** prompt, pick below the part.
6. At the next **Dimension location** prompts, continue picking the centers of the circles.
7. Finish by picking the lower right corner of the part, press **Enter**.
Chain dimension the holes vertically

- Repeat the chain dimensions to create vertical dimensions.

Add radial dimensions

2. At the Select curve for radius dimension prompt, select the hole on the far right.
3. At the Dimension location prompt, pick above the part.

4. Double click the radial dimension text, and in the text edit box, add the text Typ. 5 Places.