

SolidWorks Part 3

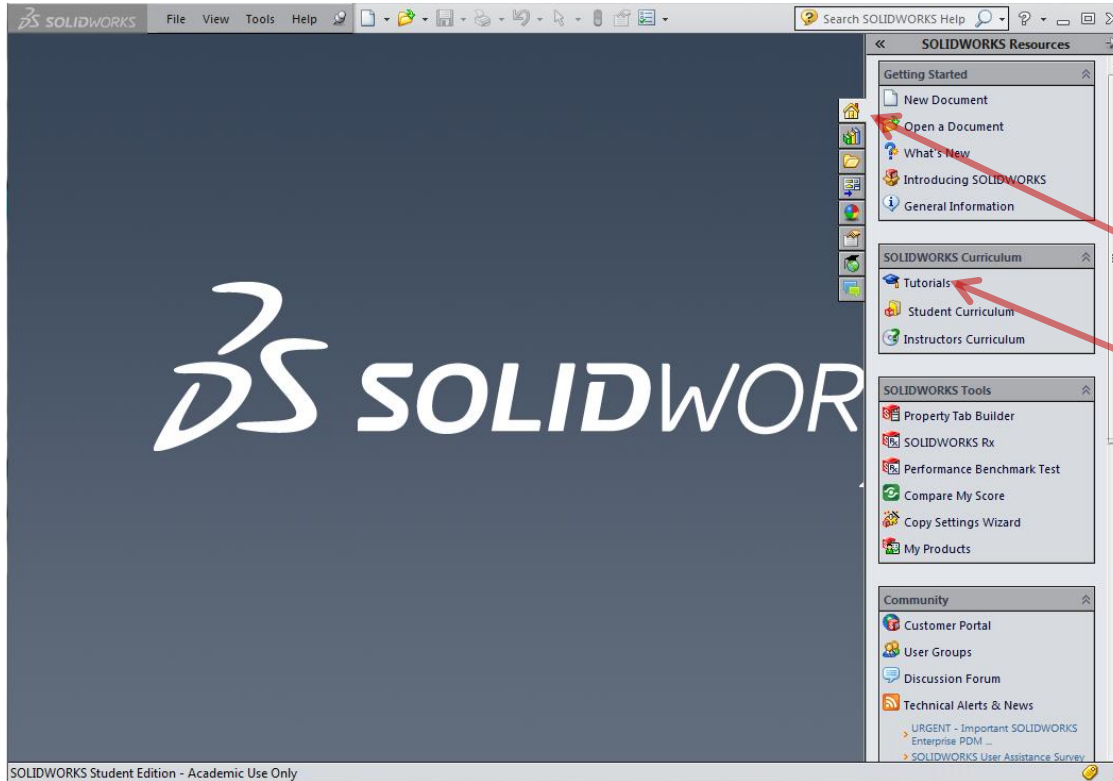
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Starting The Tutorials



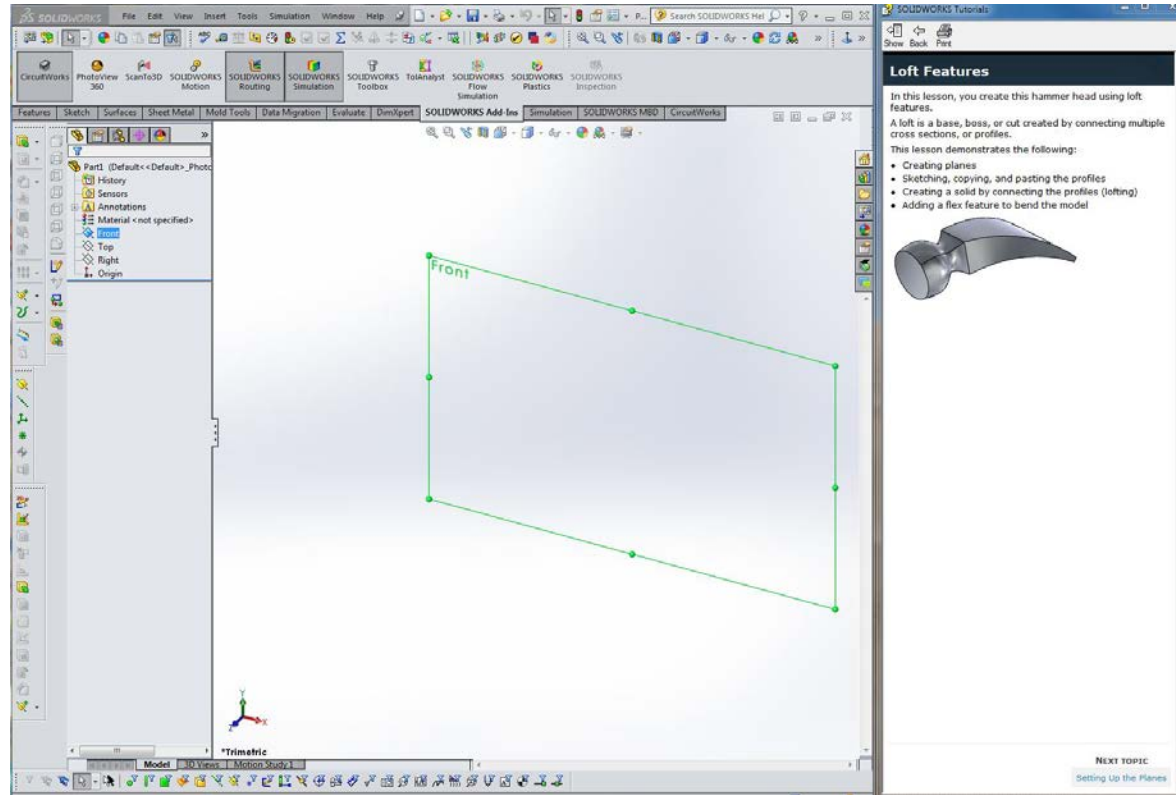
- Launch SolidWorks
- Select Resources to open the Task Pane.
- Select Tutorials

Select Basic Techniques

Open Lofts



Hammer Head



Loft Features

In this lesson, you create this hammer head using loft features.

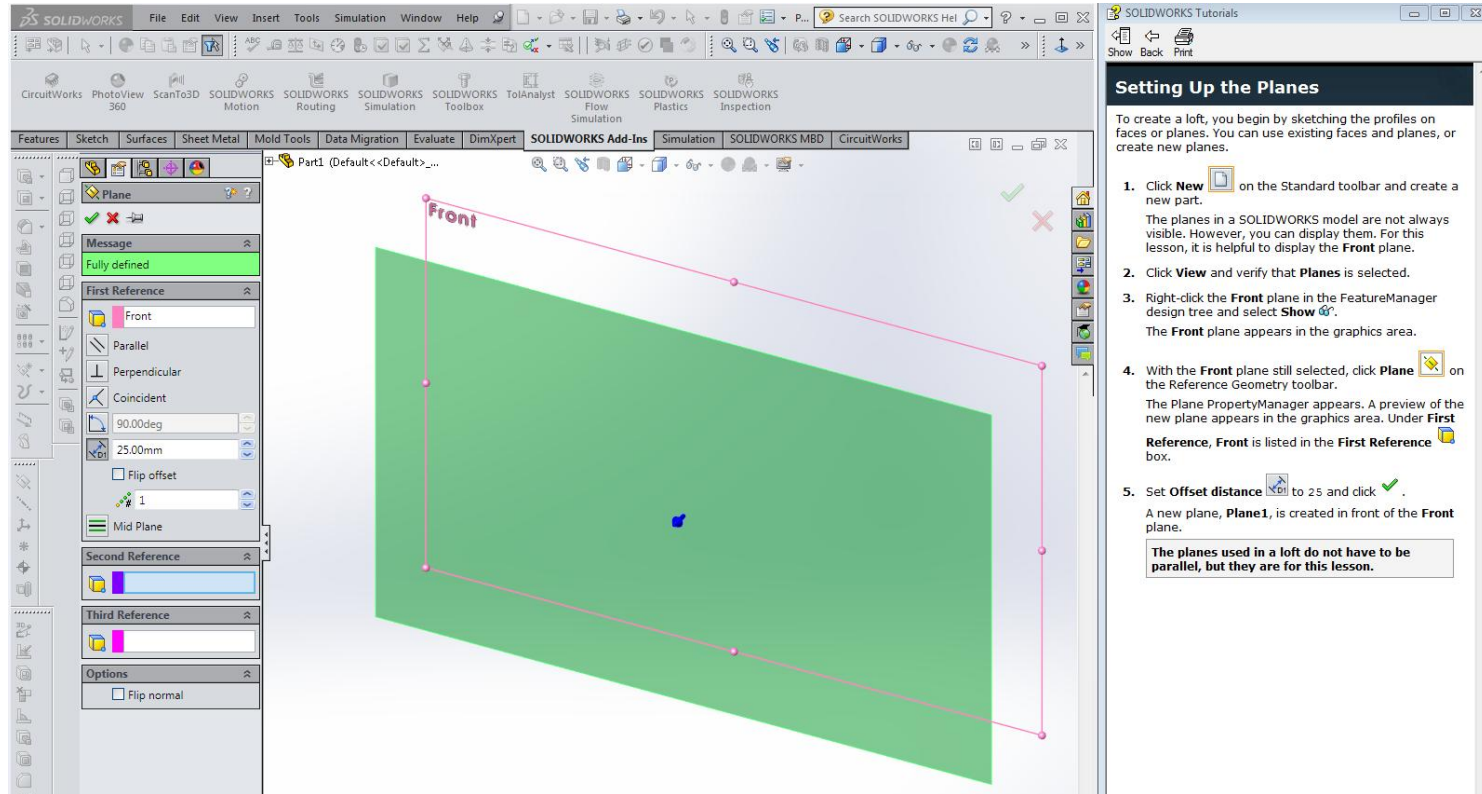
A loft is a base, boss, or cut created by connecting multiple cross sections, or profiles.

This lesson demonstrates the following:

- Creating planes
- Sketching, copying, and pasting the profiles
- Creating a solid by connecting the profiles (lofting)
- Adding a flex feature to bend the model

NEXT TOPIC
Setting Up the Planes

Creating a Parallel Plane



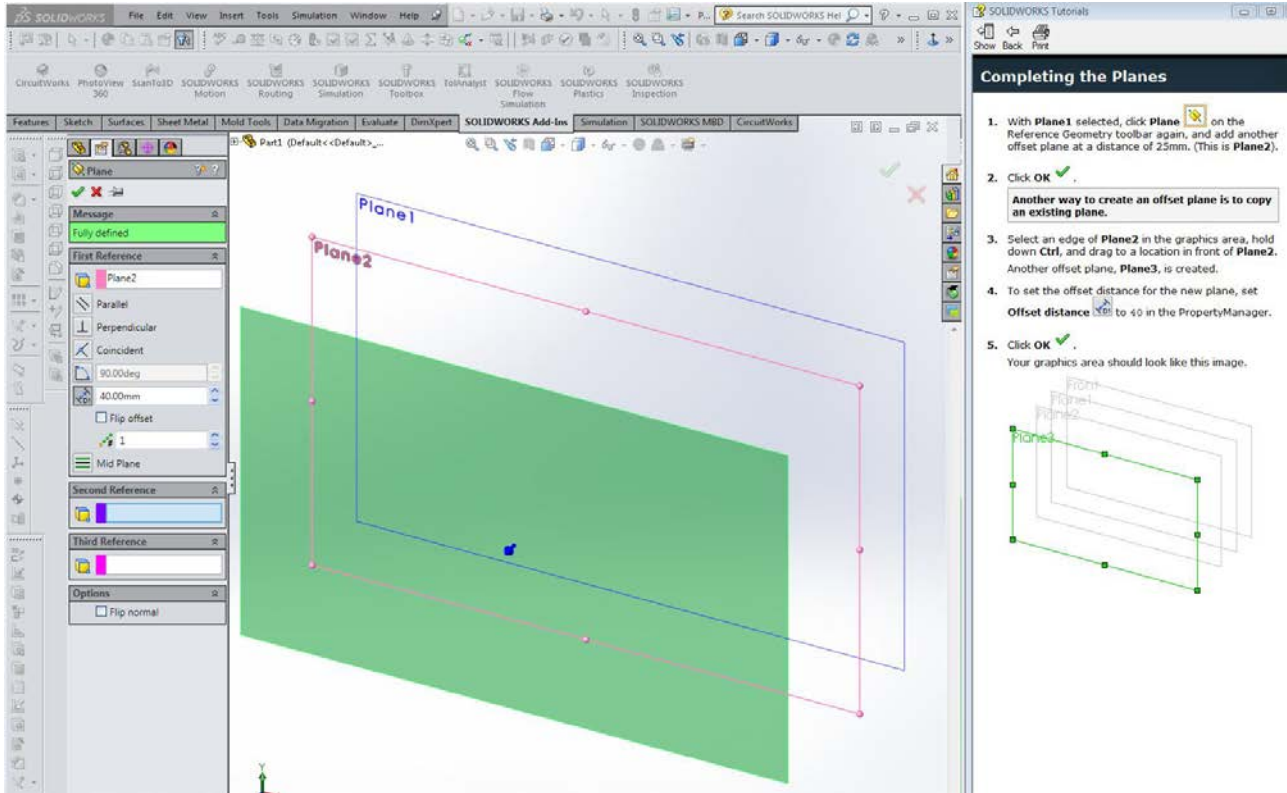
Setting Up the Planes

To create a loft, you begin by sketching the profiles on faces or planes. You can use existing faces and planes, or create new planes.

1. Click **New** on the Standard toolbar and create a new part.
The planes in a SOLIDWORKS model are not always visible. However, you can display them. For this lesson, it is helpful to display the **Front** plane.
2. Click **View** and verify that **Planes** is selected.
3. Right-click the **Front** plane in the FeatureManager design tree and select **Show**.
The **Front** plane appears in the graphics area.
4. With the **Front** plane still selected, click **Plane** on the Reference Geometry toolbar.
The Plane PropertyManager appears. A preview of the new plane appears in the graphics area. Under **First Reference**, **Front** is listed in the **First Reference** box.
5. Set **Offset distance** to 25 and click **OK**.
A new plane, **Plane1**, is created in front of the **Front** plane.

The planes used in a loft do not have to be parallel, but they are for this lesson.

Adding Two More Planes



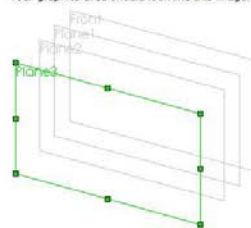
Completing the Planes

1. With **Plane1** selected, click **Plane** on the Reference Geometry toolbar again, and add another offset plane at a distance of 25mm. (This is **Plane2**).
2. Click **OK**.

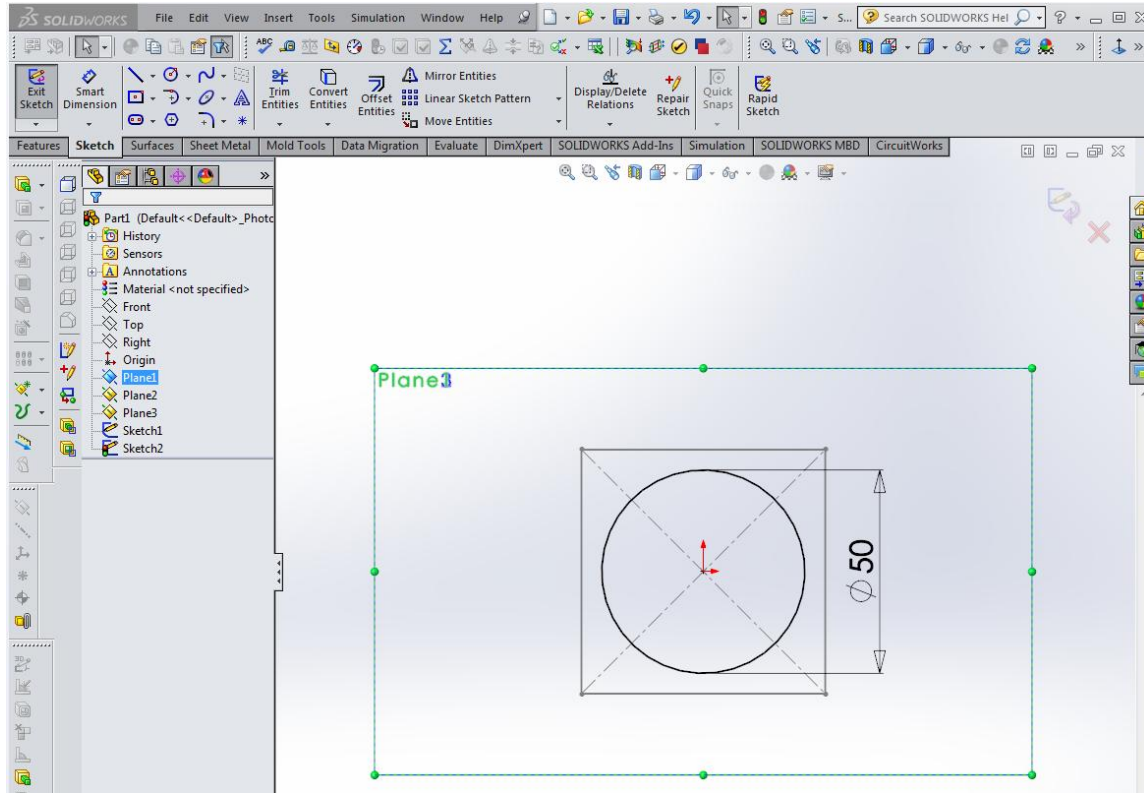
Another way to create an offset plane is to copy an existing plane.

3. Select an edge of **Plane2** in the graphics area, hold down **Ctrl**, and drag to a location in front of **Plane2**. Another offset plane, **Plane3**, is created.
4. To set the offset distance for the new plane, set **Offset distance** to 40 in the PropertyManager.
5. Click **OK**.

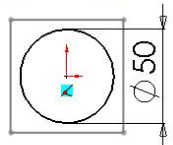

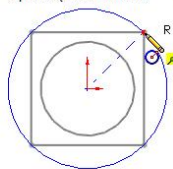
Your graphics area should look like this image.



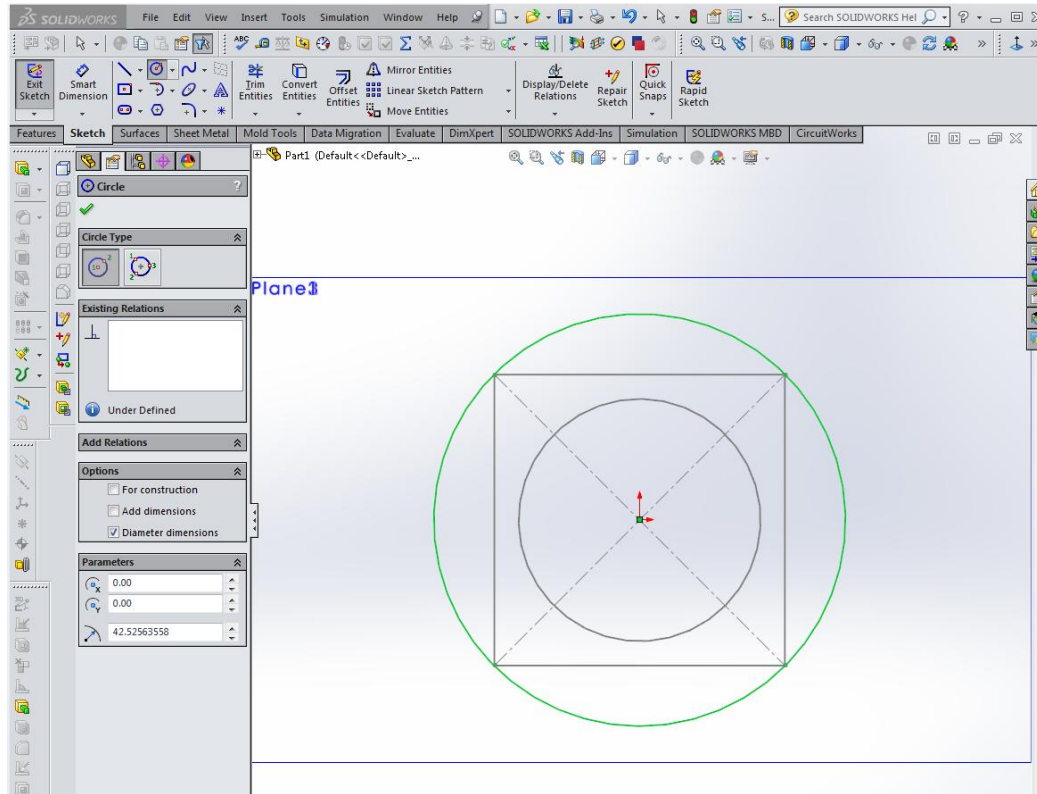
Sketch on Plane 2



Completing the Profiles

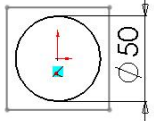

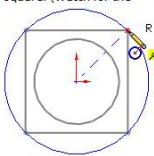
1. Open a sketch on **Plane1**, and sketch a circle, centered on the origin.
It appears as though you are sketching on top of the first sketch. However, the first sketch is on the **Front** plane, and it is not affected by sketching on **Plane1**, a parallel plane in front of it.
2. Dimension the circle to 50mm in diameter.

3. Exit the sketch.
4. Open a sketch on **Plane2**, and sketch a circle, centered on the origin. As you drag, make the diameter of the circle coincident with the vertex of the square. (Watch for the  pointer.)

5. Exit the sketch.

Circle Coincident with Vertices

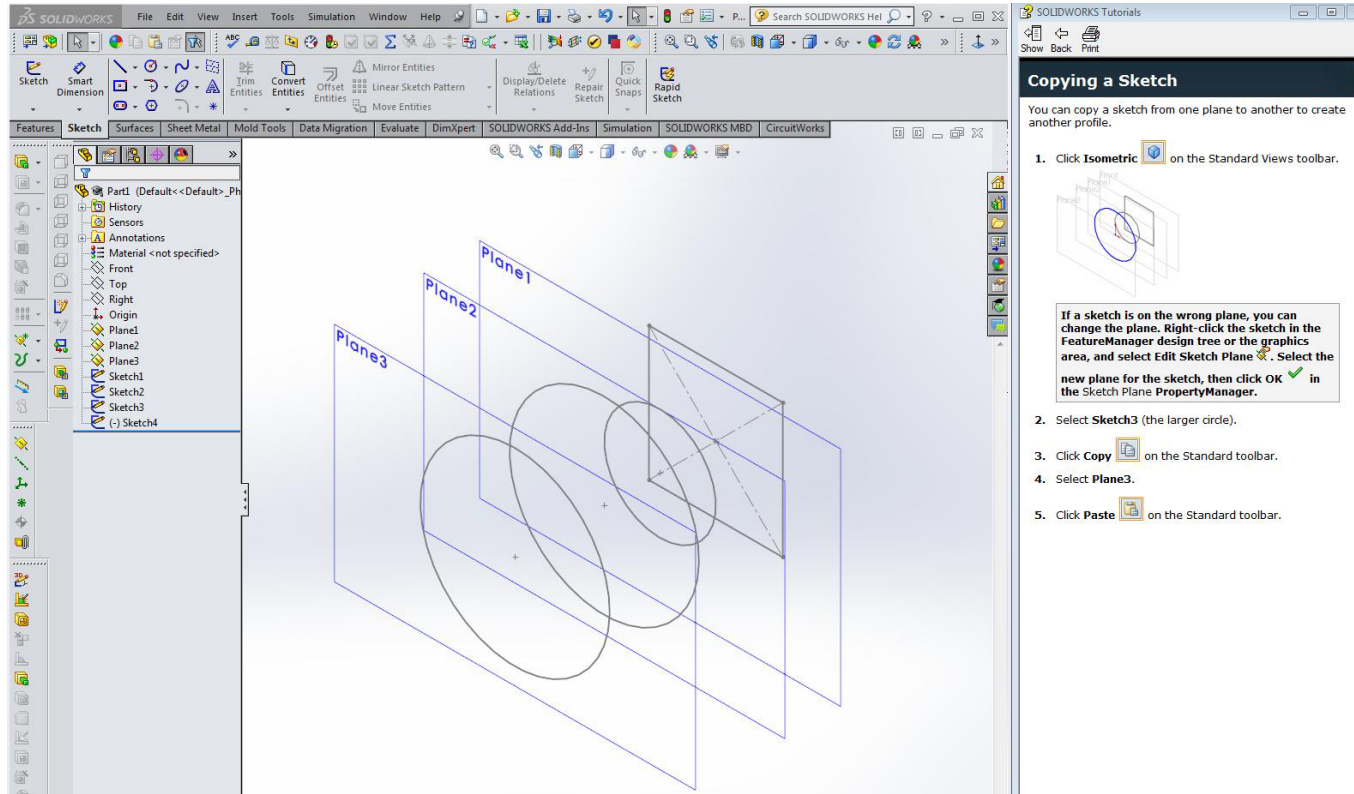


The screenshot shows the SolidWorks interface with a sketch on a plane. A square is sketched, and a circle is sketched such that its center is at the origin of the coordinate system and its diameter is equal to the side length of the square. The circle is highlighted in green. The software interface includes a menu bar, a toolbar, and a left-hand panel with various tool options like 'Circle', 'Circle Type', and 'Add Relations'.

Completing the Profiles

1. Open a sketch on **Plane1**, and sketch a circle, centered on the origin.
It appears as though you are sketching on top of the first sketch. However, the first sketch is on the **Front** plane, and it is not affected by sketching on **Plane1**, a parallel plane in front of it.
2. Dimension the circle to 50mm in diameter.

3. Exit the sketch.
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5. Exit the sketch.

Copy-Paste Sketch from Plane 2 to 3



Copying a Sketch

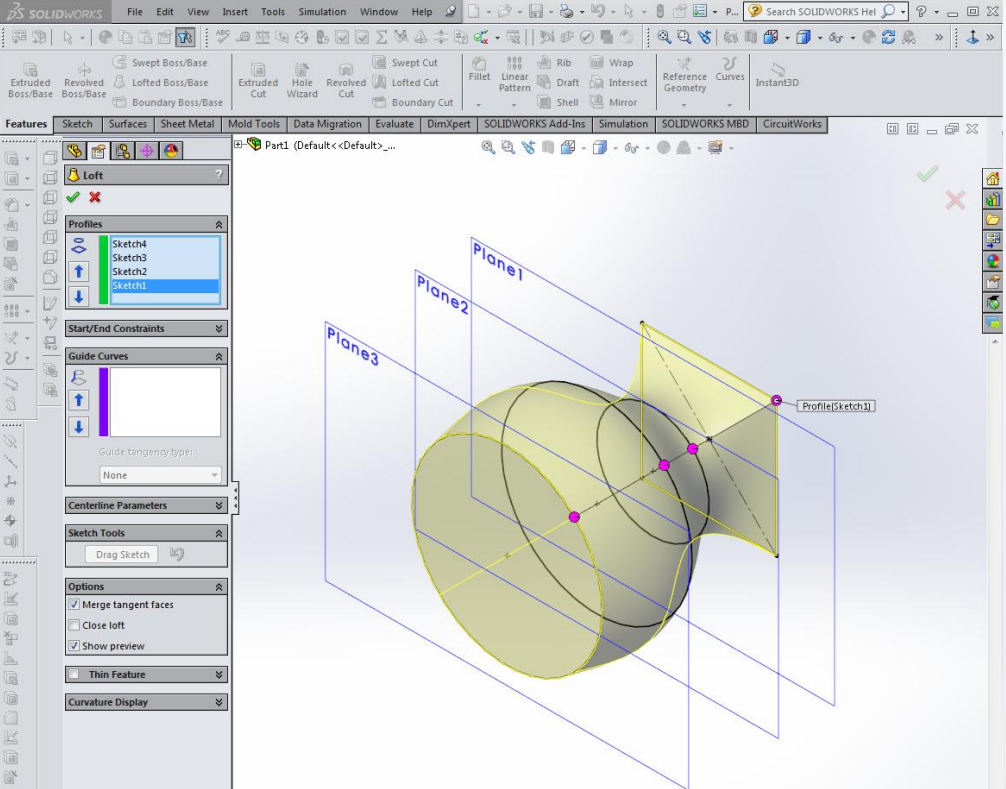
You can copy a sketch from one plane to another to create another profile.

1. Click **Isometric** on the Standard Views toolbar.

If a sketch is on the wrong plane, you can change the plane. Right-click the sketch in the FeatureManager design tree or the graphics area, and select **Edit Sketch Plane**. Select the new plane for the sketch, then click **OK** in the Sketch Plane PropertyManager.

2. Select **Sketch3** (the larger circle).
3. Click **Copy** on the Standard toolbar.
4. Select **Plane3**.
5. Click **Paste** on the Standard toolbar.

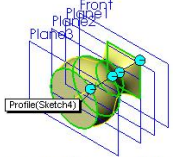
Lofted Boss-Base Feature






Creating the Loft

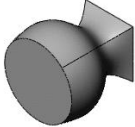
Now use the **Lofted Boss/Base** feature to create a solid model based on the profiles.

1. Click **Lofted Boss/Base** on the Features toolbar.
2. In the graphics area, click near the same place on each profile (for example, the upper-right side), so the loft path travels in a straight line and does not get twisted. Select the sketches in the order you want to connect them.

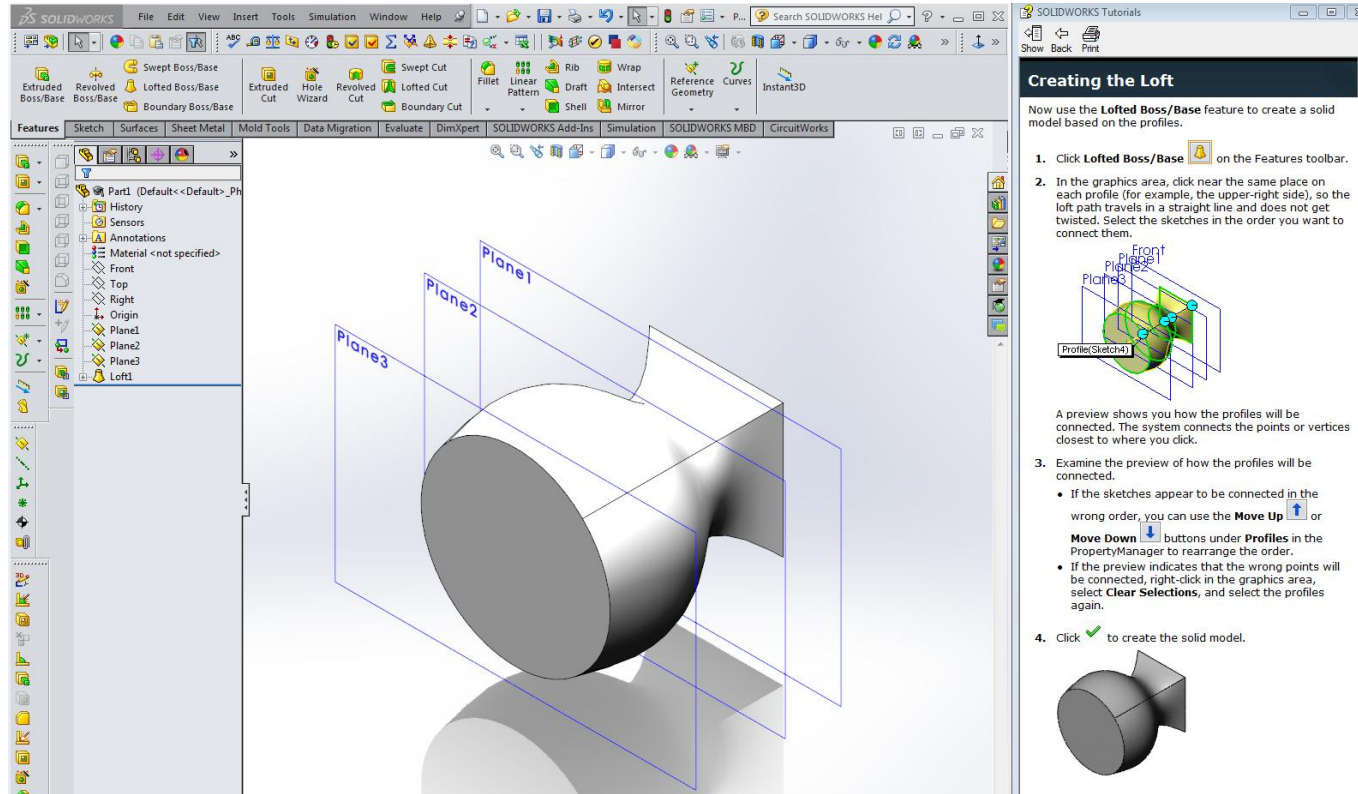


A preview shows you how the profiles will be connected. The system connects the points or vertices closest to where you click.

3. Examine the preview of how the profiles will be connected.
 - If the sketches appear to be connected in the wrong order, you can use the **Move Up**  or **Move Down**  buttons under **Profiles** in the PropertyManager to rearrange the order.
 - If the preview indicates that the wrong points will be connected, right-click in the graphics area, select **Clear Selections**, and select the profiles again.
4. Click  to create the solid model.



Solid Model

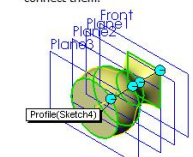


The screenshot displays the SolidWorks interface with the Loft feature active. The main graphics area shows a 3D model of a curved, tapered part with three blue planes (Plane1, Plane2, Plane3) intersecting it. The left-hand side shows the Feature Tree with 'Loft1' selected. The top ribbon contains various modeling tools, and the right-hand side shows a tutorial window titled 'Creating the Loft'.

Creating the Loft

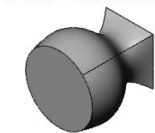
Now use the **Lofted Boss/Base** feature to create a solid model based on the profiles.

1. Click **Lofted Boss/Base** on the Features toolbar.
2. In the graphics area, click near the same place on each profile (for example, the upper-right side), so the loft path travels in a straight line and does not get twisted. Select the sketches in the order you want to connect them.

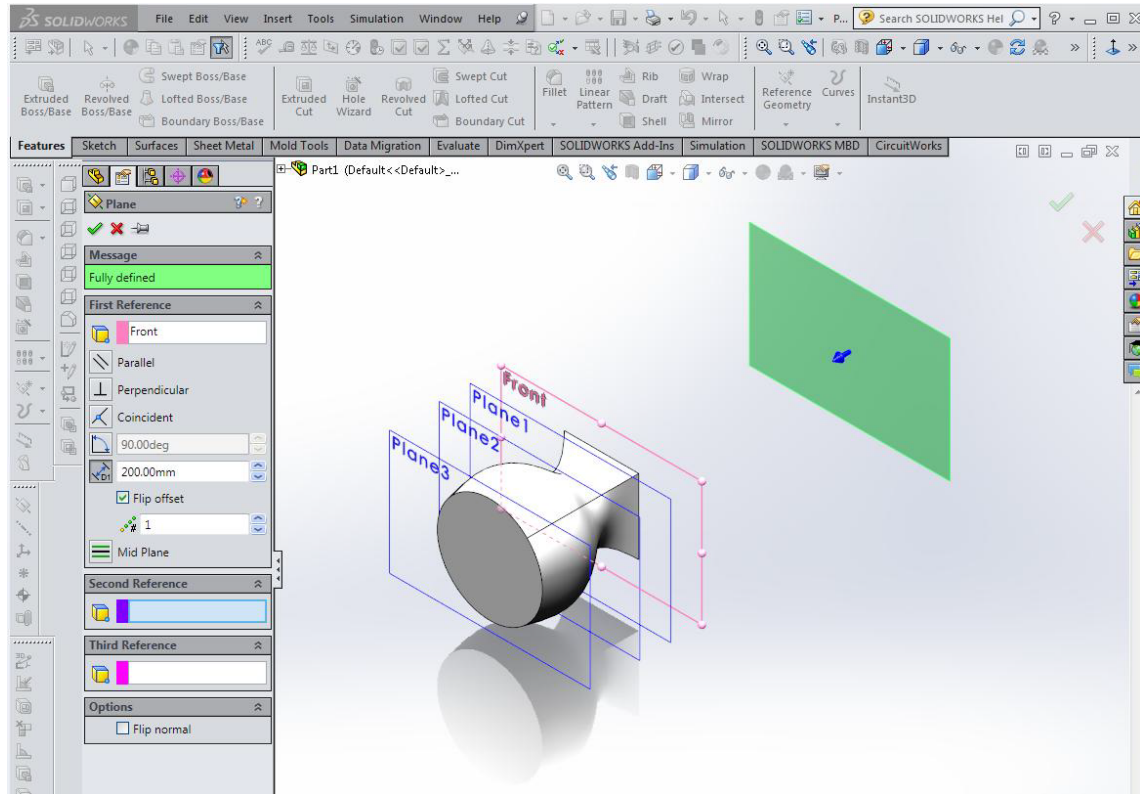


A preview shows you how the profiles will be connected. The system connects the points or vertices closest to where you click.

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Creating Plane 4

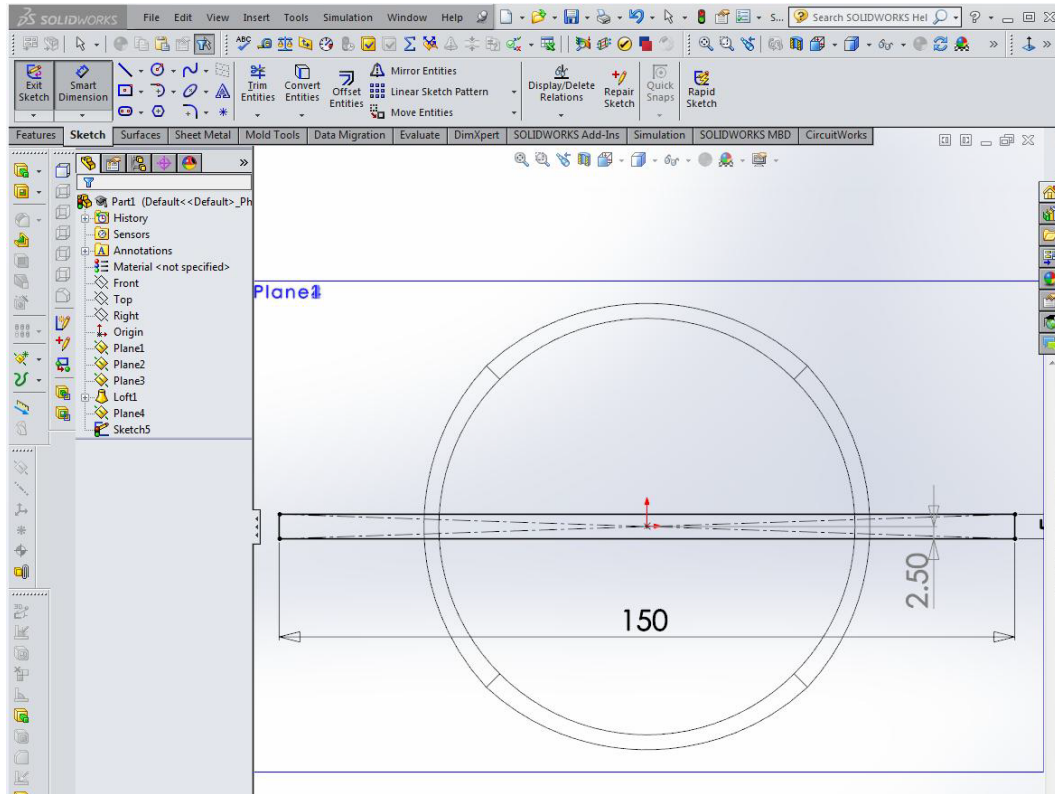


Creating a Boss Loft

For the pointed end of the hammer head, you create another loft.

1. Hold down **Ctrl**, and drag an edge of the **Front** plane to create an offset plane behind the original **Front** plane.
The Plane PropertyManager appears.
2. Set **Offset distance** to 200 .
3. Make sure that **Flip offset** is selected so the new plane is created behind the **Front** plane, then click **OK** to create the new **Plane4**.
4. Click **Hidden Lines Removed** on the View toolbar.
5. Click **Normal To** on the Standard Views toolbar.
6. Open a sketch on **Plane4**, then sketch and dimension a narrow rectangle as shown, which is the profile you use to create the next loft.
7. Exit the sketch.

Sketch on Plane 4

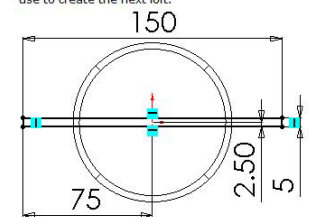


SOLIDWORKS Tutorials

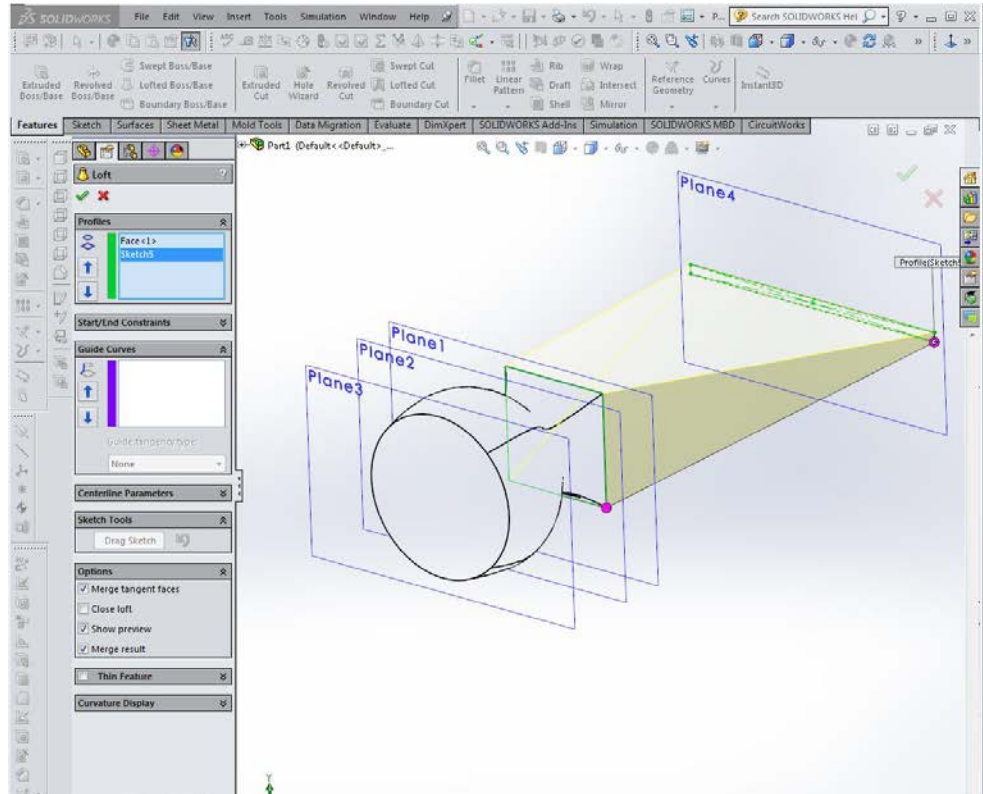
Creating a Boss Loft

For the pointed end of the hammer head, you create another loft.

1. Hold down **Ctrl**, and drag an edge of the **Front** plane to create an offset plane behind the original **Front** plane.
The **Plane PropertyManager** appears.
2. Set **Offset distance** to 200.
3. Make sure that **Flip offset** is selected so the new plane is created behind the **Front** plane, then click **OK** to create the new **Plane4**.
4. Click **Hidden Lines Removed** on the View toolbar.
5. Click **Normal To** on the Standard Views toolbar.
6. Open a sketch on **Plane4**, then sketch and dimension a narrow rectangle as shown, which is the profile you use to create the next loft.
7. Exit the sketch.





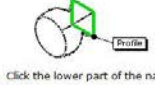
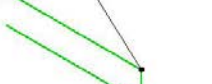


Lofted Boss-Base Connected Profiles



Completing the Second Lofted Boss

Now you complete the second lofted boss.

1. Click **Isometric**  on the Standard Views toolbar.
2. Click **Lofted Boss/Base**  on the Features toolbar.
3. Select the square profile:
 - a. Rotate the model as shown and select the face in the lower corner closest to you.
- b. Click **Isometric**  again.
4. Click the lower part of the narrow rectangular sketch.
5. Examine the preview of how the two profiles will be connected.

Flex Feature and Trim Planes

Bending the Part with the Flex Feature

The flex feature deforms a model. You can use the flex feature to bend, twist, taper, or stretch a model. Here you use the flex feature to bend the hammer head.

1. Click **Flex** on the Features toolbar.
2. In the PropertyManager, under **Flex Input**, select:
 - a. The part in the graphics area for **Bodies for Flex**.
 - b. **Bending**.
3. Under **Trim Plane 2**, click in **Select a reference entity for Trim Plane 2**.
4. In the graphics area, select the vertex as shown.



Trim Plane 2 aligns to the selected vertex.

5. Right-click the triad's center sphere as shown, and select **Align to**.

If you do not see this option, click in **Bodies for Flex** and try again.

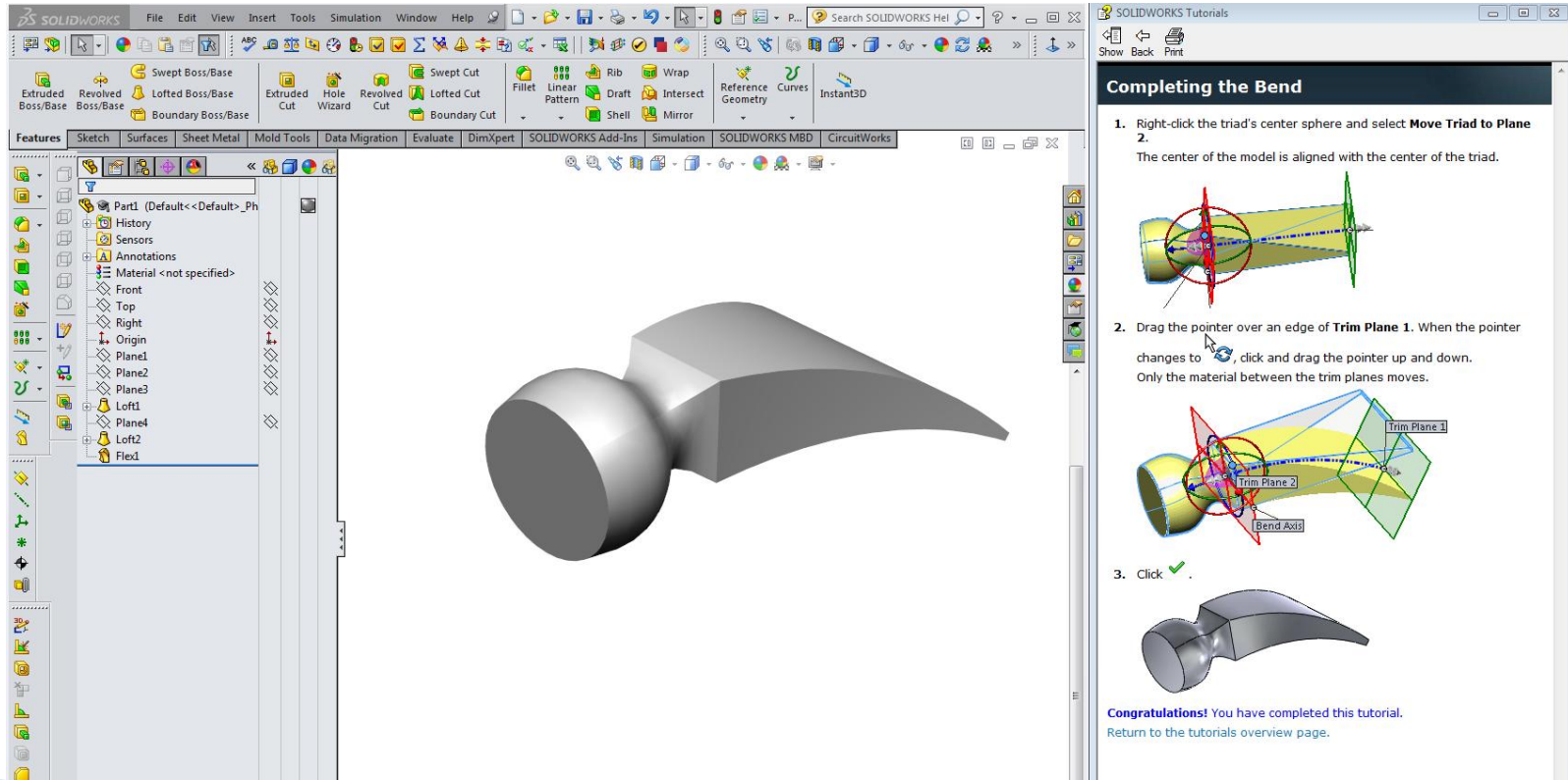
Moving Material Between Trim Planes

Completing the Bend



1. Right-click the triad's center sphere and select **Move Triad to Plane 2**.
The center of the model is aligned with the center of the triad.
2. Drag the pointer over an edge of **Trim Plane 1**. When the pointer changes to , click and drag the pointer up and down. Only the material between the trim planes moves.
3. Click .

Congratulations! You have completed this tutorial.
Return to the tutorials overview page.

Finished Hammer Head



The image displays the SolidWorks CAD environment. The main window shows a 3D model of a hammer head. The left-hand side contains the Feature Tree, listing features such as Part1, History, Sensors, Annotations, Material, and various planes and lofts. The top ribbon includes tabs for Sketch, Surfaces, Sheet Metal, Mold Tools, Data Migration, Evaluate, DimXpert, and others. The right-hand side shows a tutorial window titled "Completing the Bend" with the following instructions:

1. Right-click the triad's center sphere and select **Move Triad to Plane 2**.
The center of the model is aligned with the center of the triad.
2. Drag the pointer over an edge of **Trim Plane 1**. When the pointer changes to , click and drag the pointer up and down. Only the material between the trim planes moves.
3. Click .

Congratulations! You have completed this tutorial.
Return to the tutorials overview page.

Summary

- ▶ Topics Covered in this exercise:
 - Creating parallel planes.
 - Sketching on specific planes and copying from one plane to another.
 - Lofted base–base feature and connecting profiles.
 - Flex feature, moving trim planes and bending material