SolidWorks Part 3

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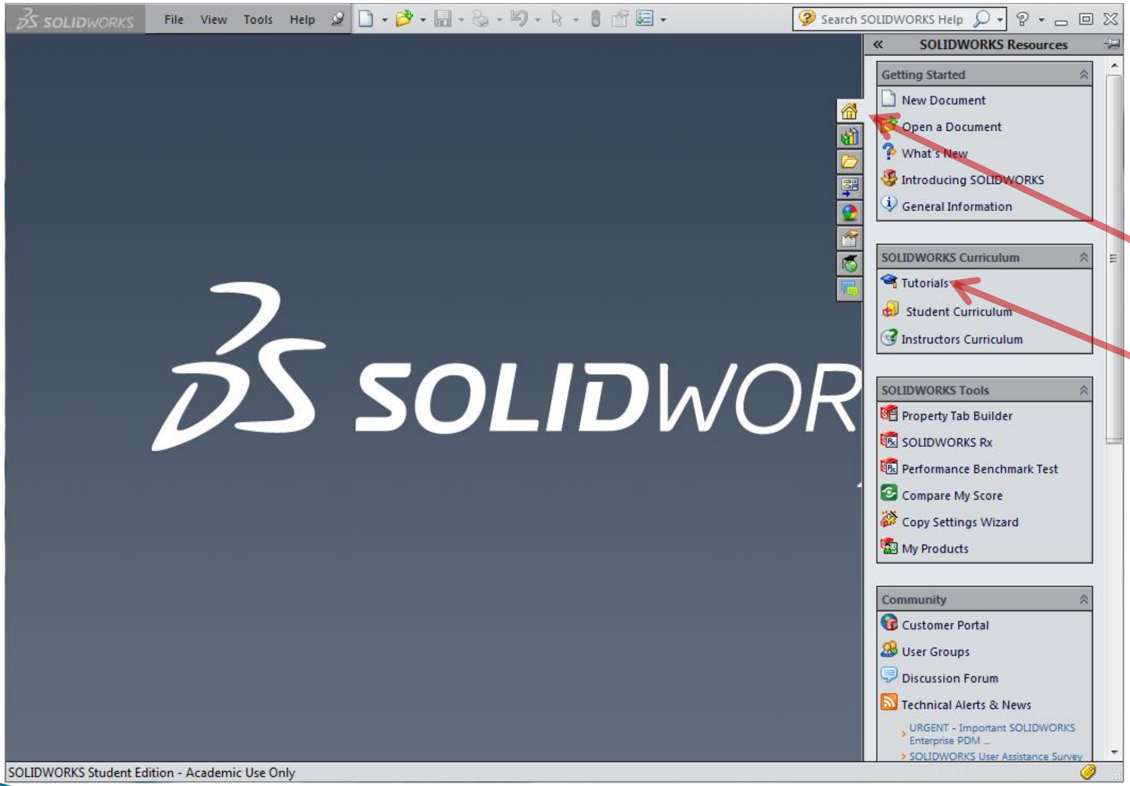
Starting The Tutorials

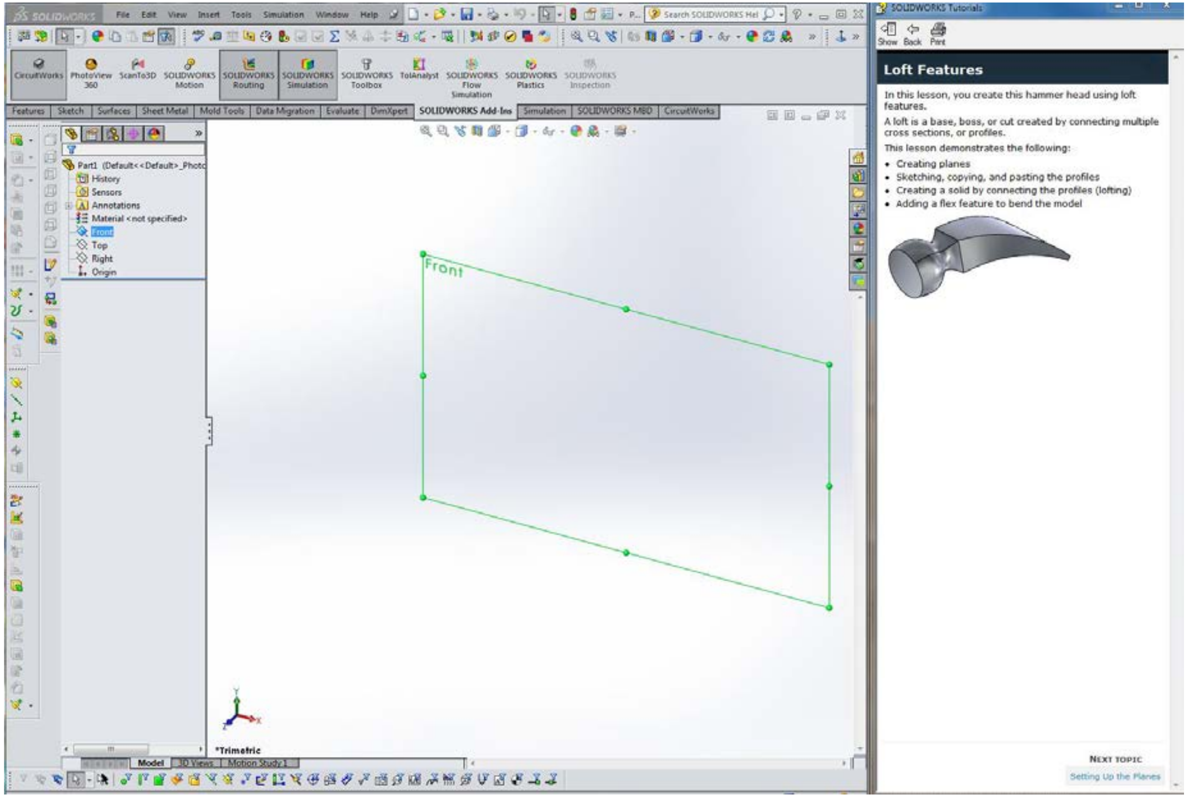


- Launch SolidWorks
- Select Resources to open the Task Pane.
- Select Tutorials

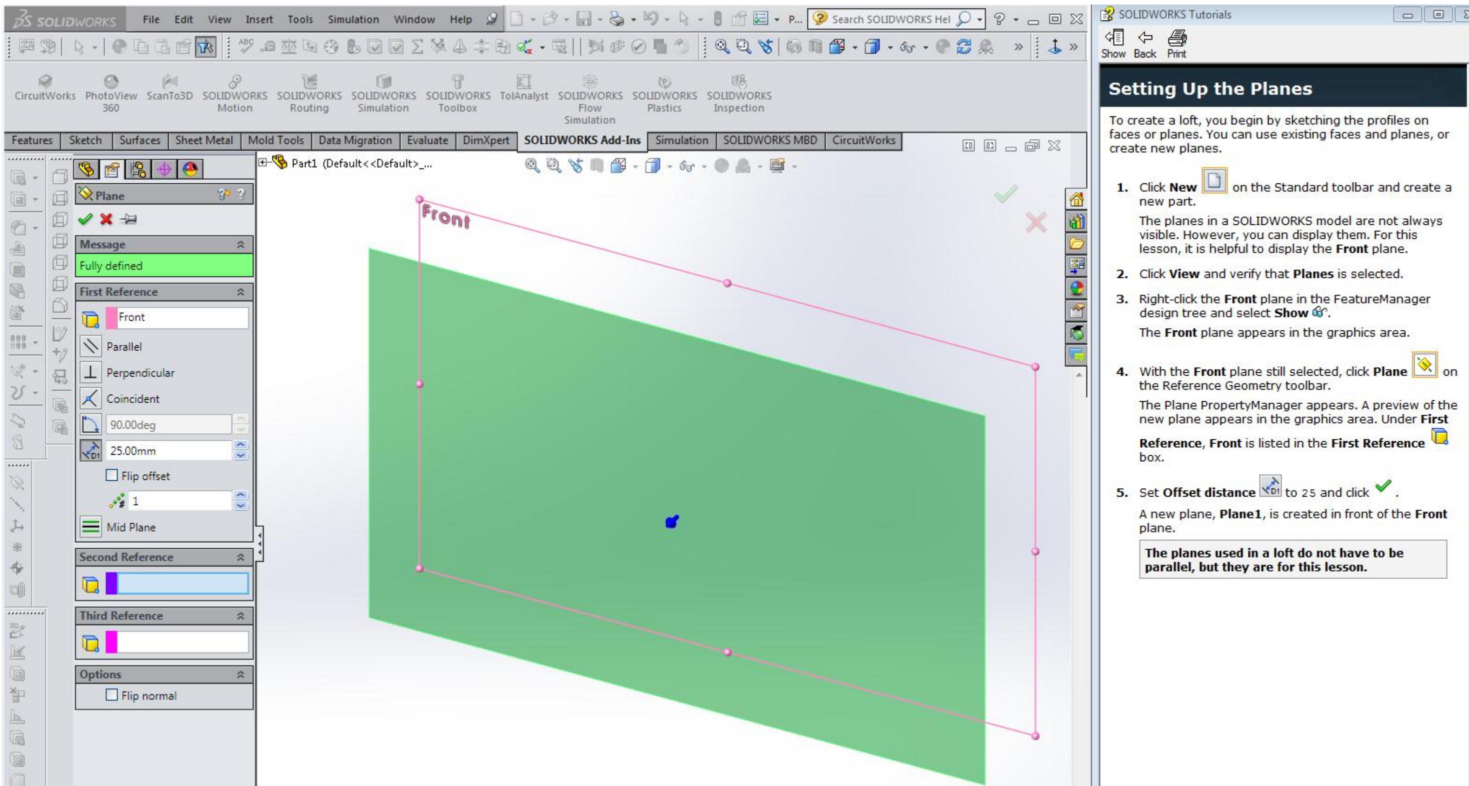
Select Basic Techniques

Open Lofts

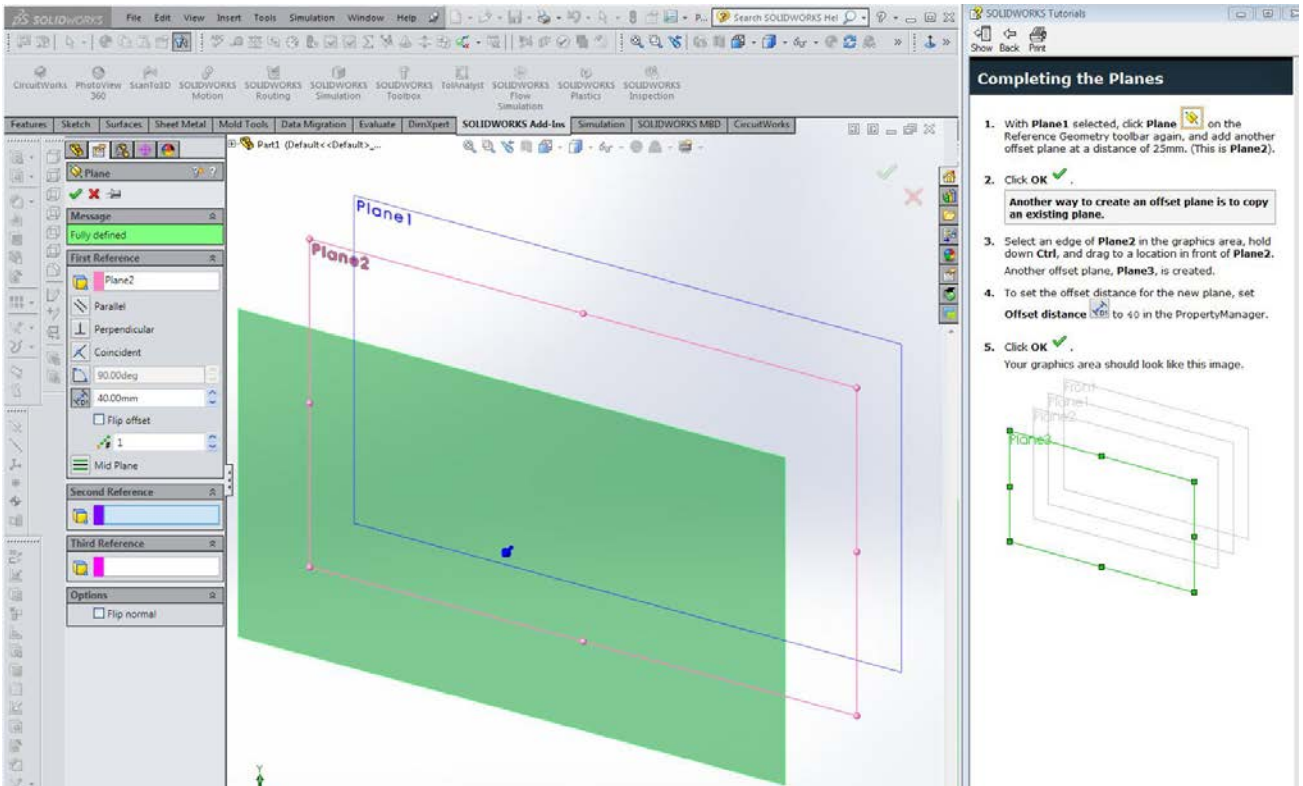
Hammer Head


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
Front
*Trimetric
Next topic
Setting Up the Planes

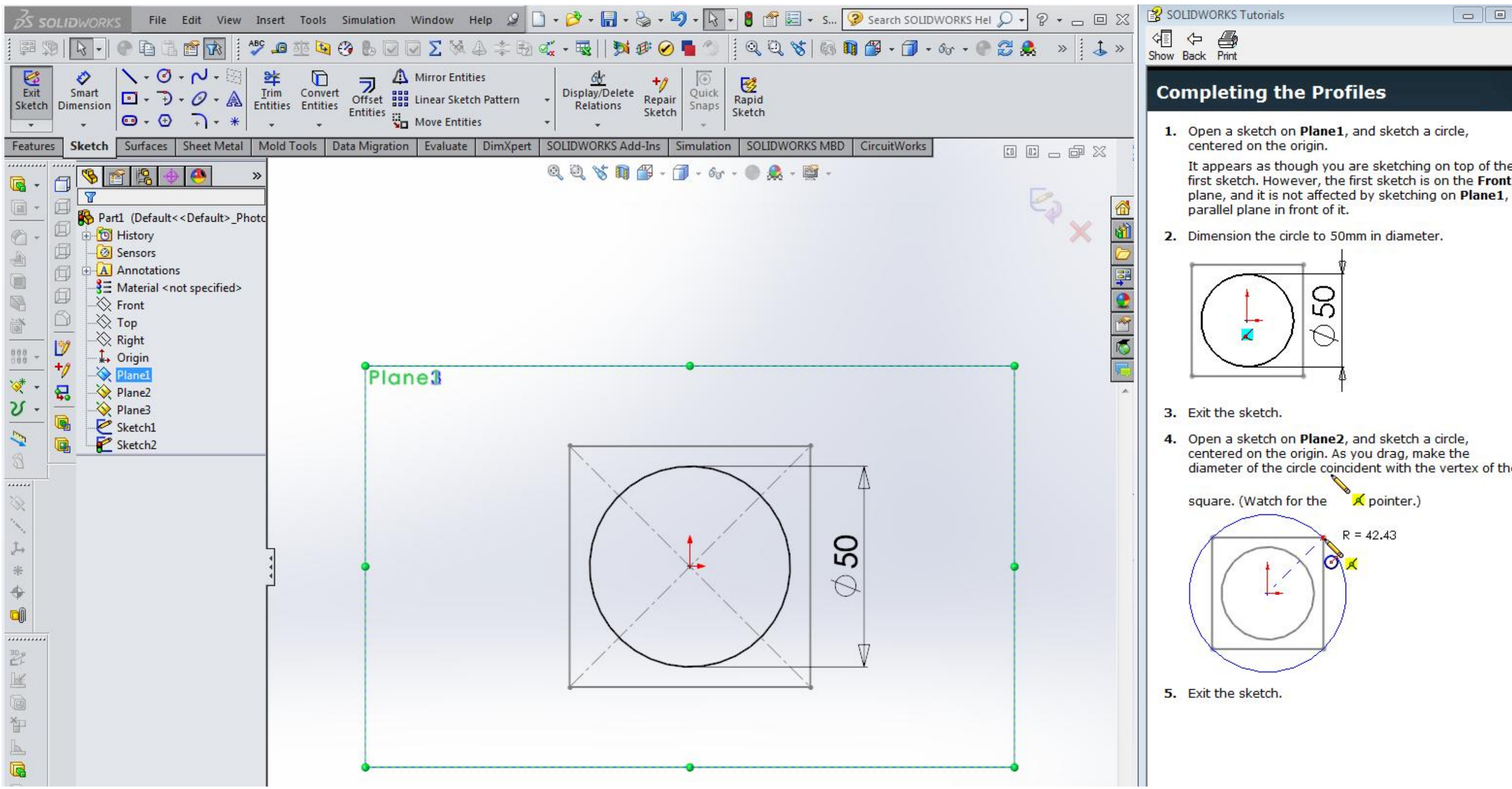
Creating a Parallel Plane



Adding Two More Planes



Sketch on Plane 2



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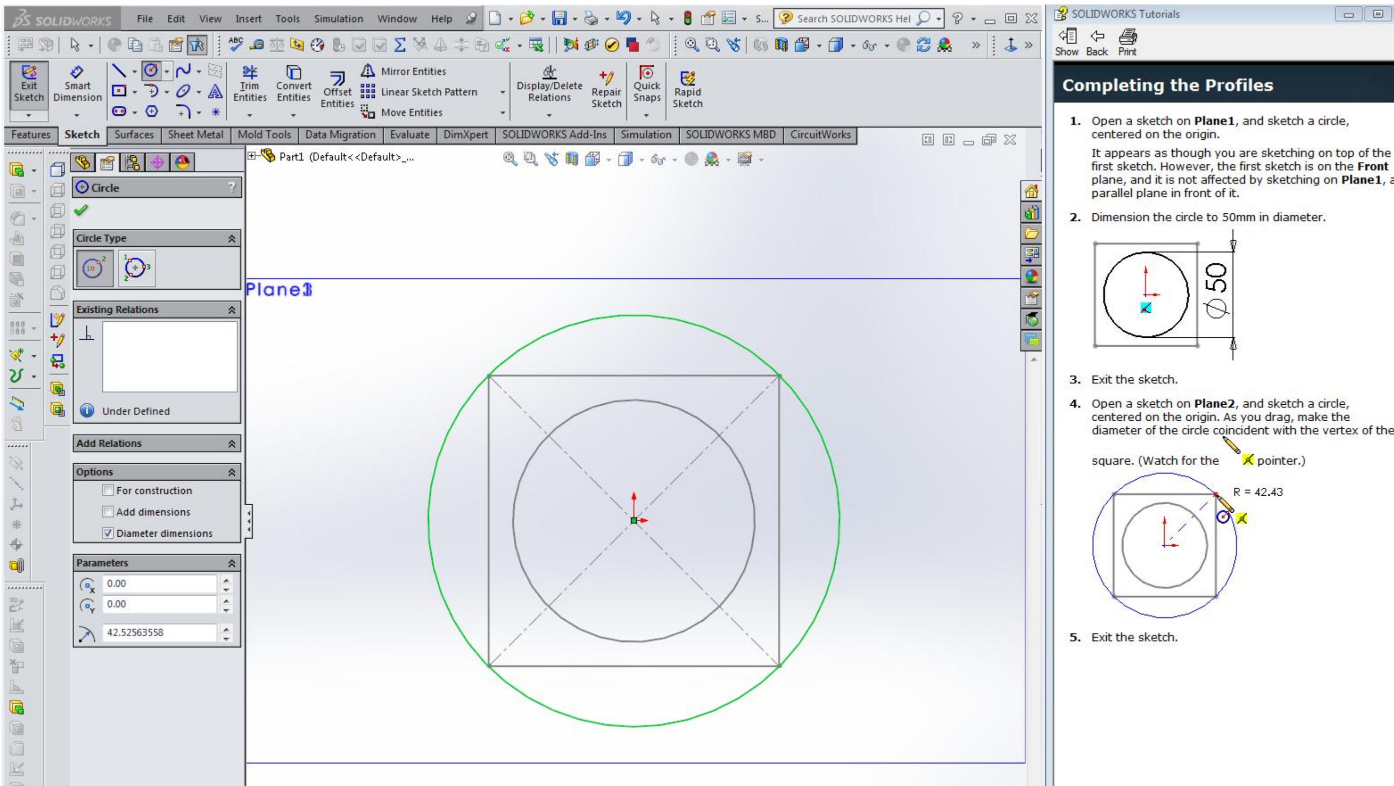
Circle Coincident with Vertices



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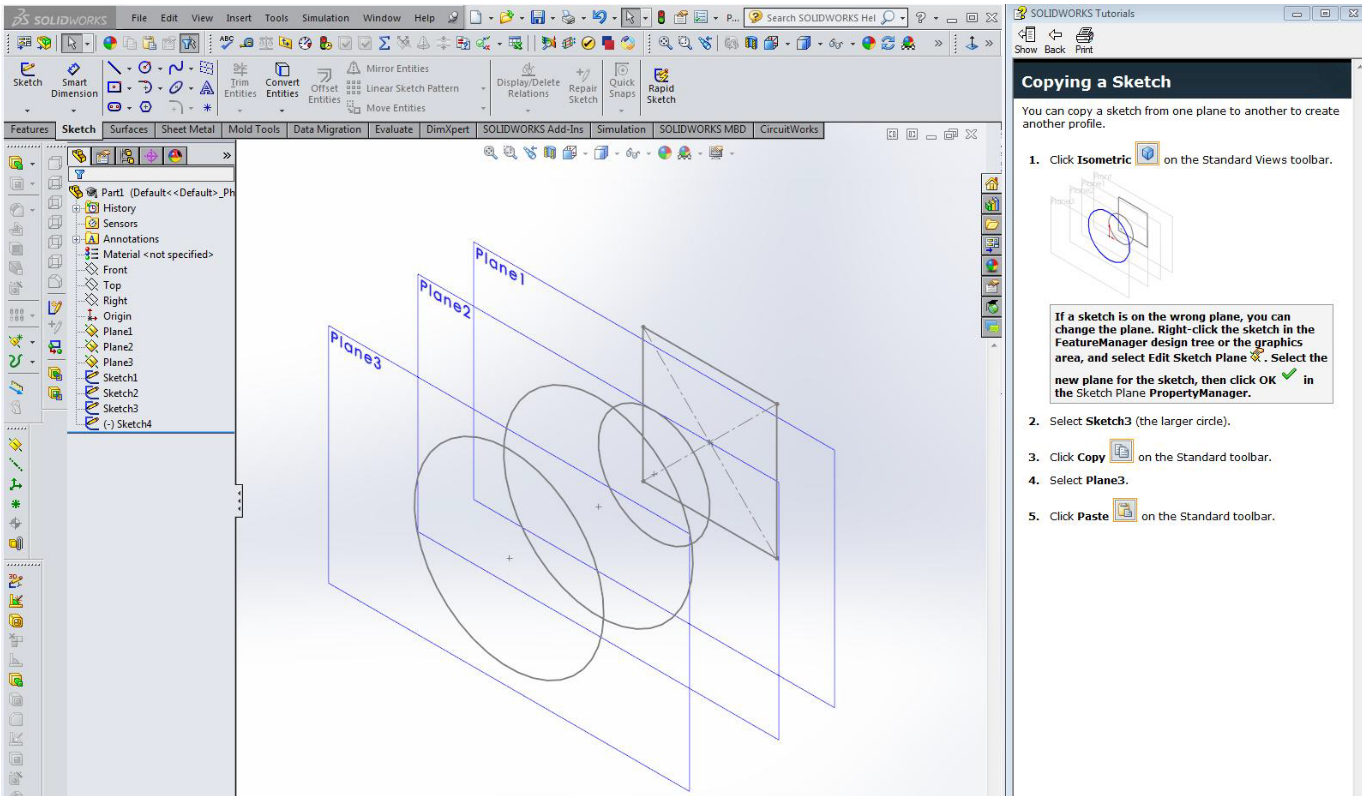
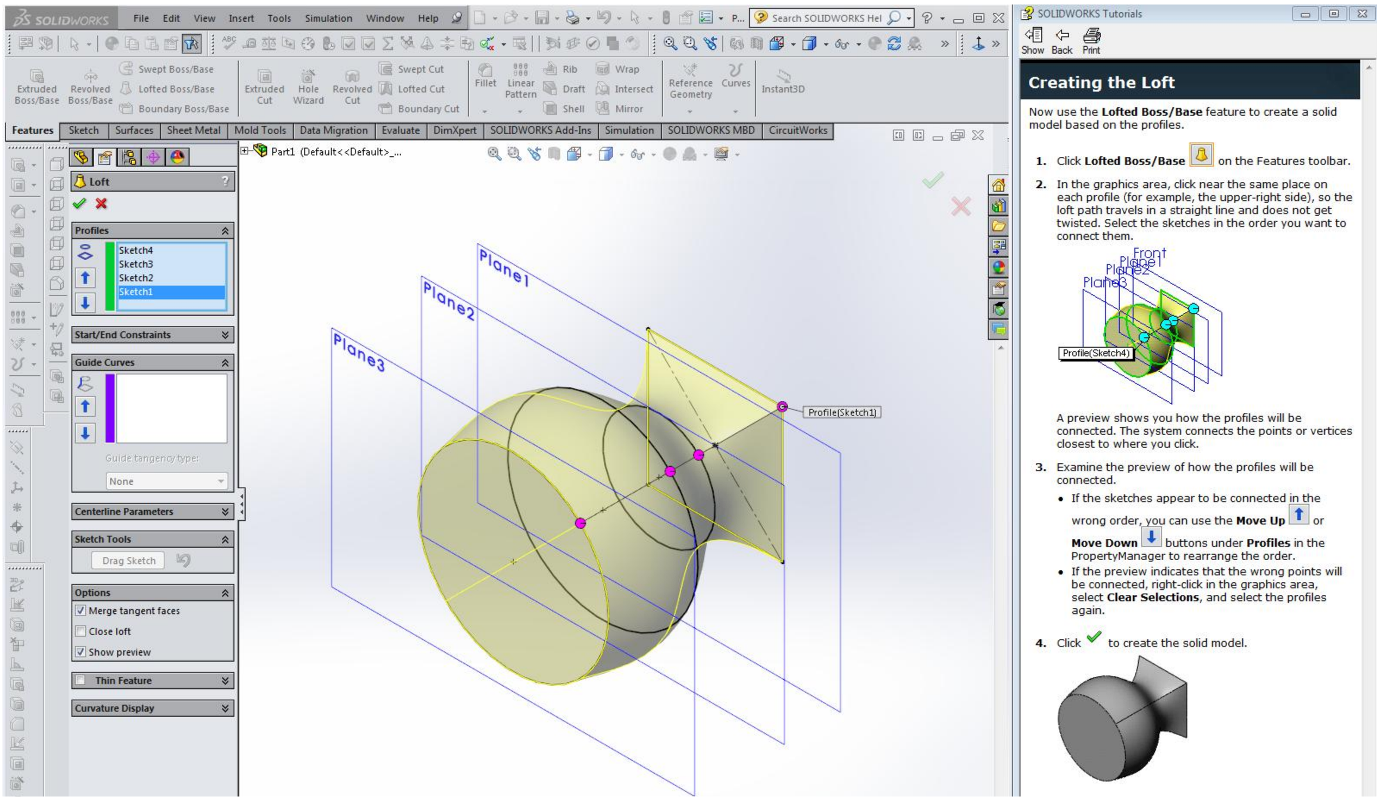
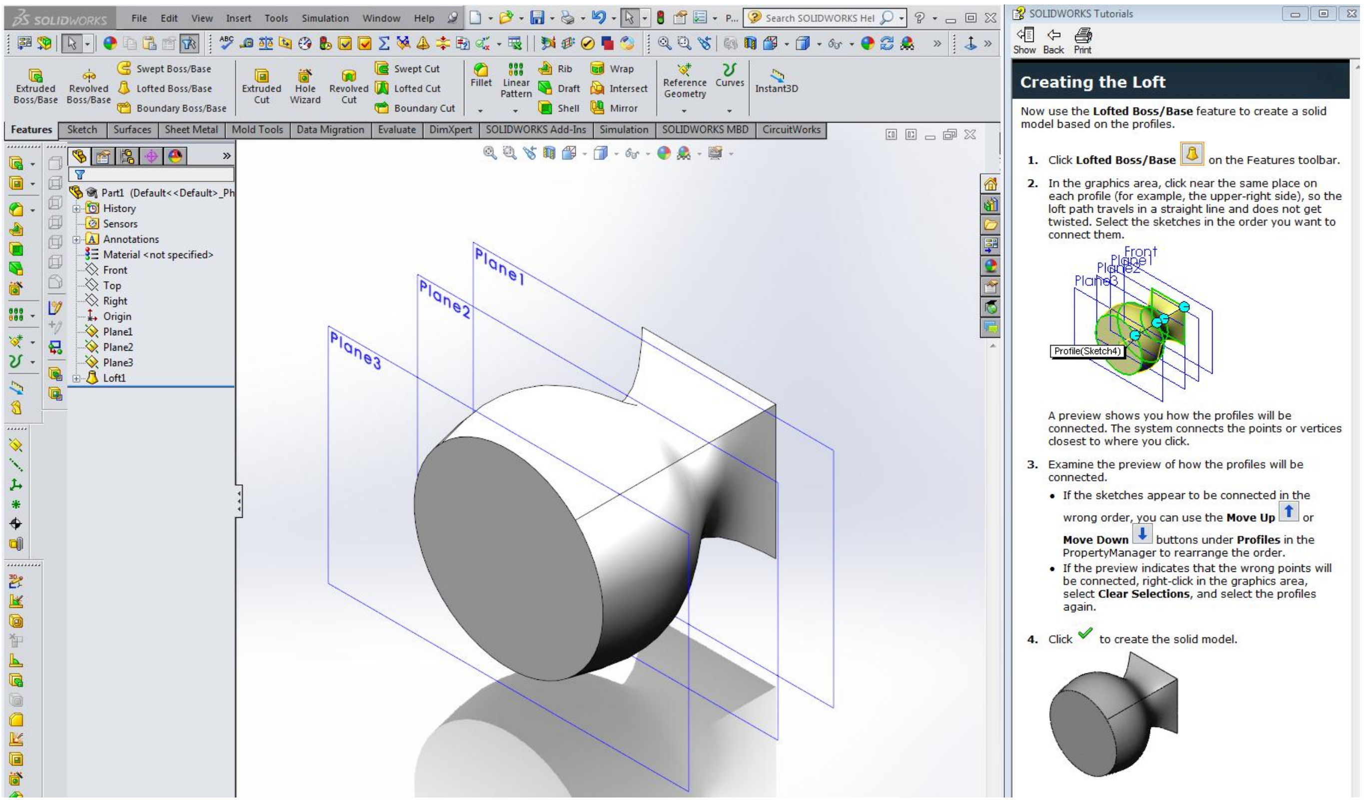
Copy-Paste Sketch from Plane 2 to 3



Lofted Bose-Base Feature

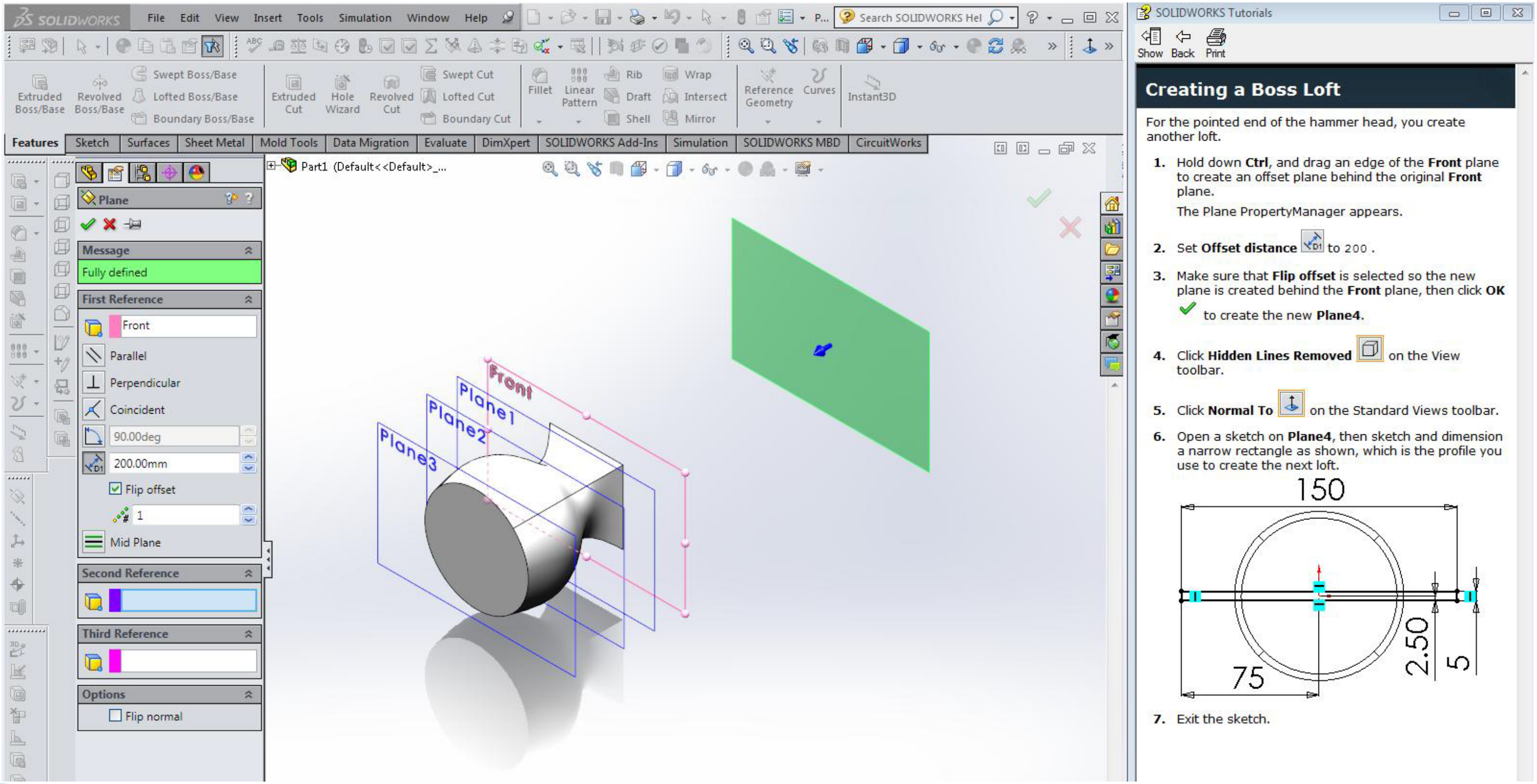


Solid Model

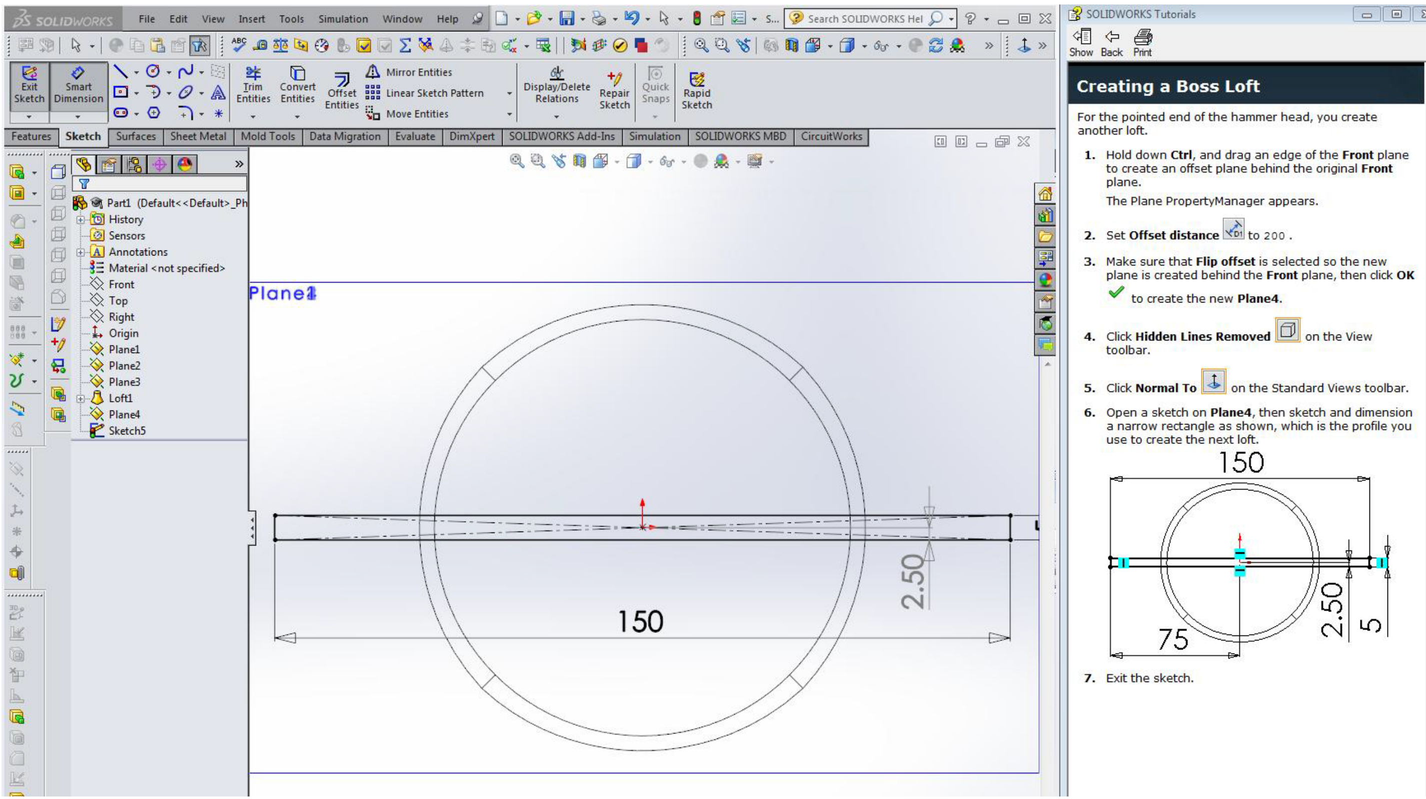


SOLIDWORKS
File Edit View Insert Tools Simulation Window Help
Extruded Boss/Base
Revolved Boss/Base
Swept Boss/Base
Lofted Boss/Base
Boundary Boss/Base
Extruded Cut
Hole Wizard
Revolved Cut
Swept Cut
Lofted Cut
Boundary Cut
Fillet
Linear Pattern
Rib
Draft
Shell
Wrap
Intersect
Mirror
Reference Geometry
Curves
Instant3D
Features Sketch Surfaces Sheet Metal Mold Tools Data Migration Evaluate DimXpert SOLIDWORKS Add-Ins Simulation SOLIDWORKS MBD CircuitWorks
Part1 (Default<<Default>_Ph
History
Sensors
Annotations
Material <not specified>
Front
Top
Right
Origin
Plane1
Plane2
Plane3
Loft1
Plane1
Plane2
Plane3
SOLIDWORKS Tutorials
Show Back Print
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.
Front
Plane1
Plane2
Plane3
Profile(Sketch4)
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.

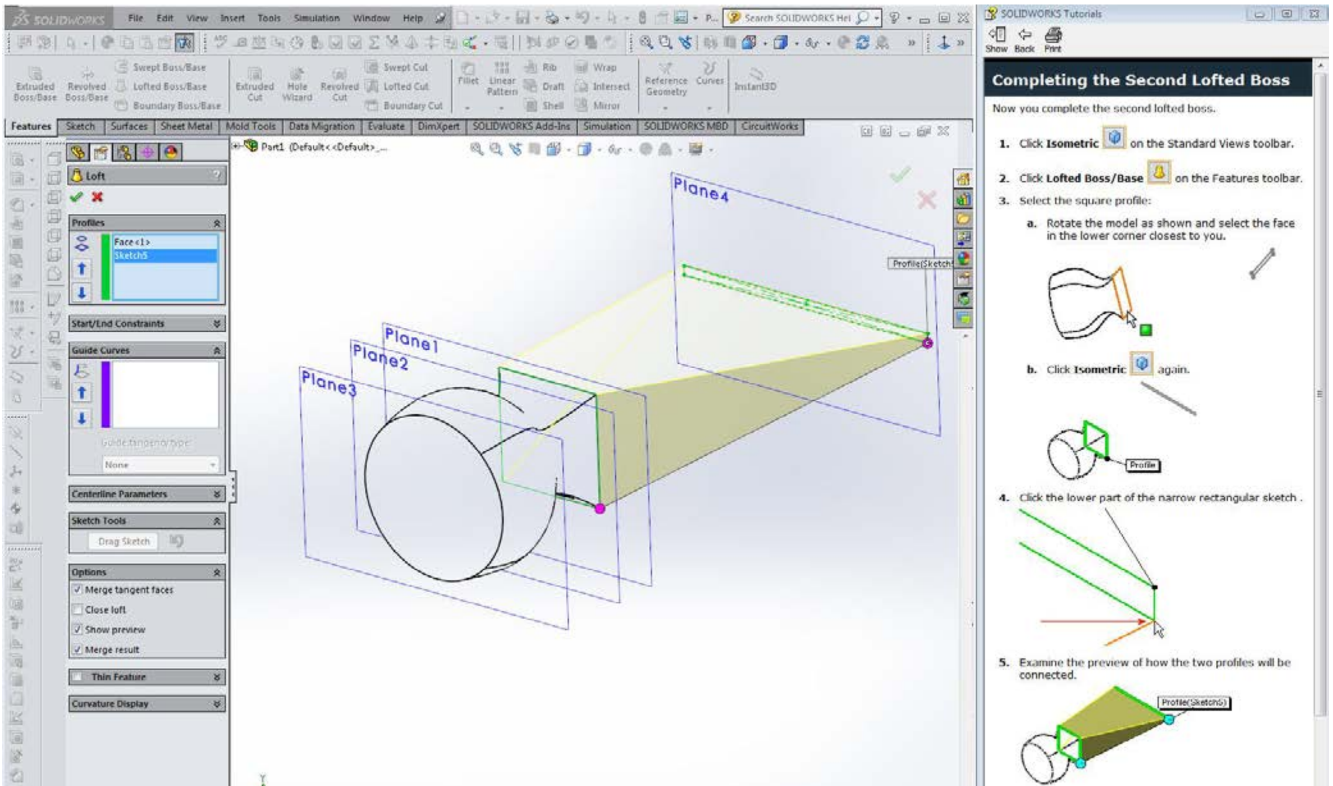
Creating Plane 4



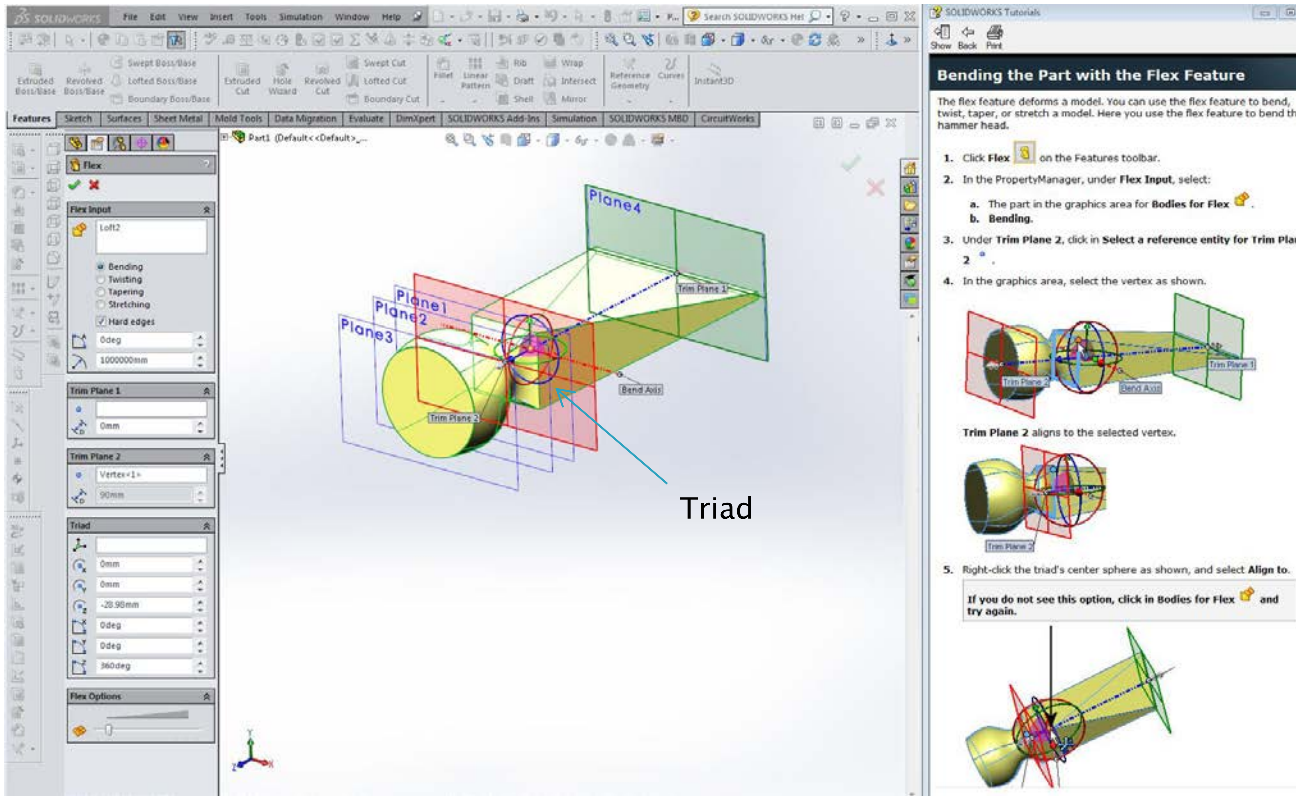
Sketch on Plane 4



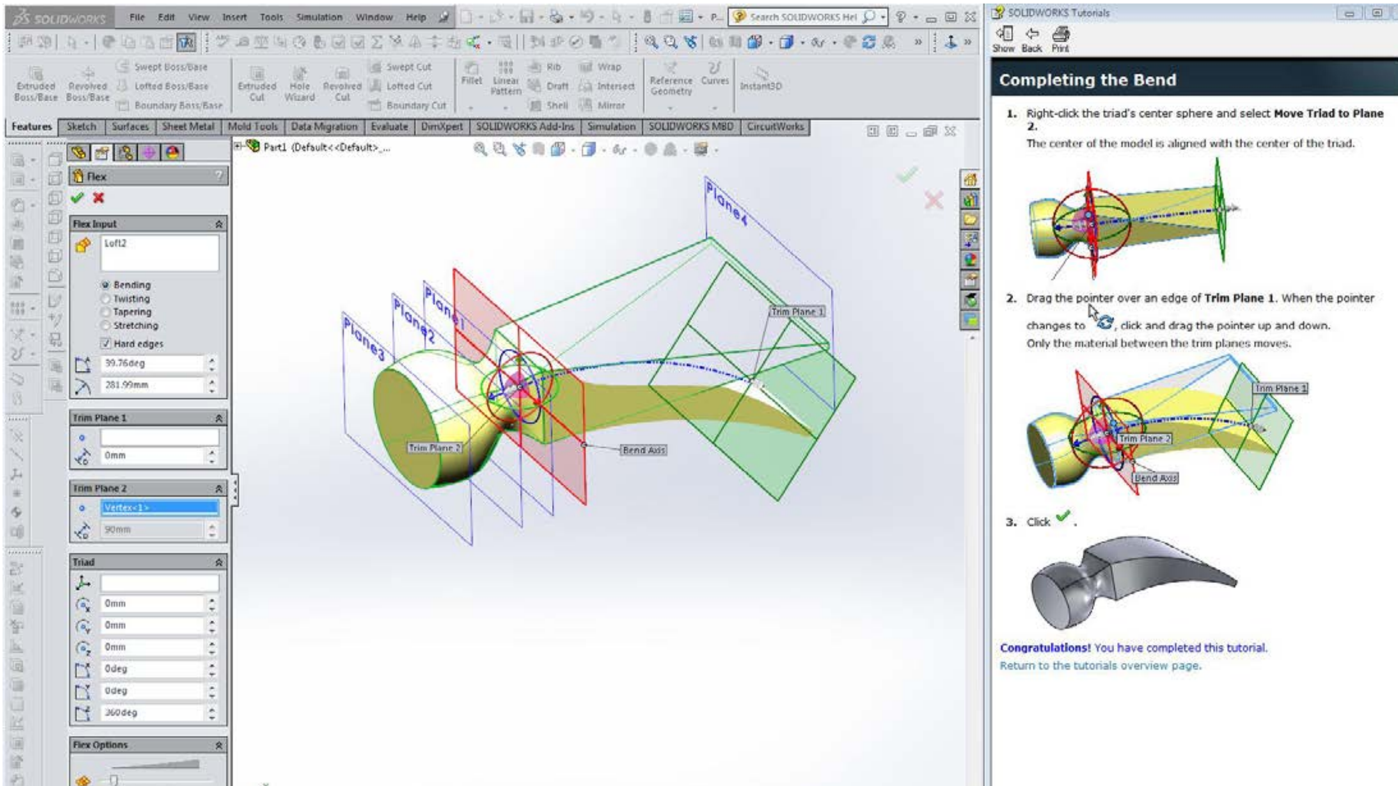
Lofted Bose-Base Connected Profiles



Flex Feature and Trim Planes



Moving Material Between Trim Planes



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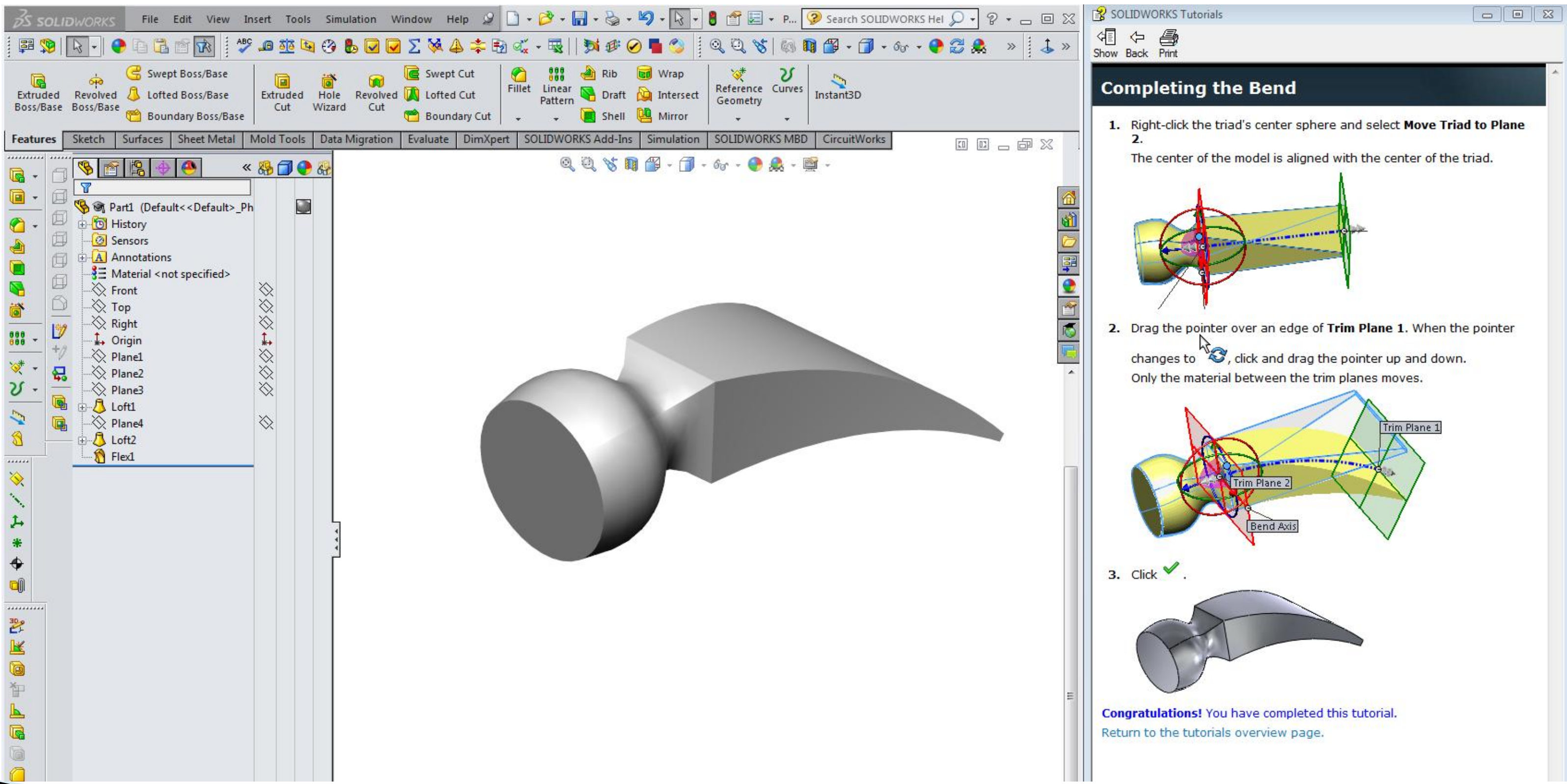
Finished Hammer Head



Summary

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