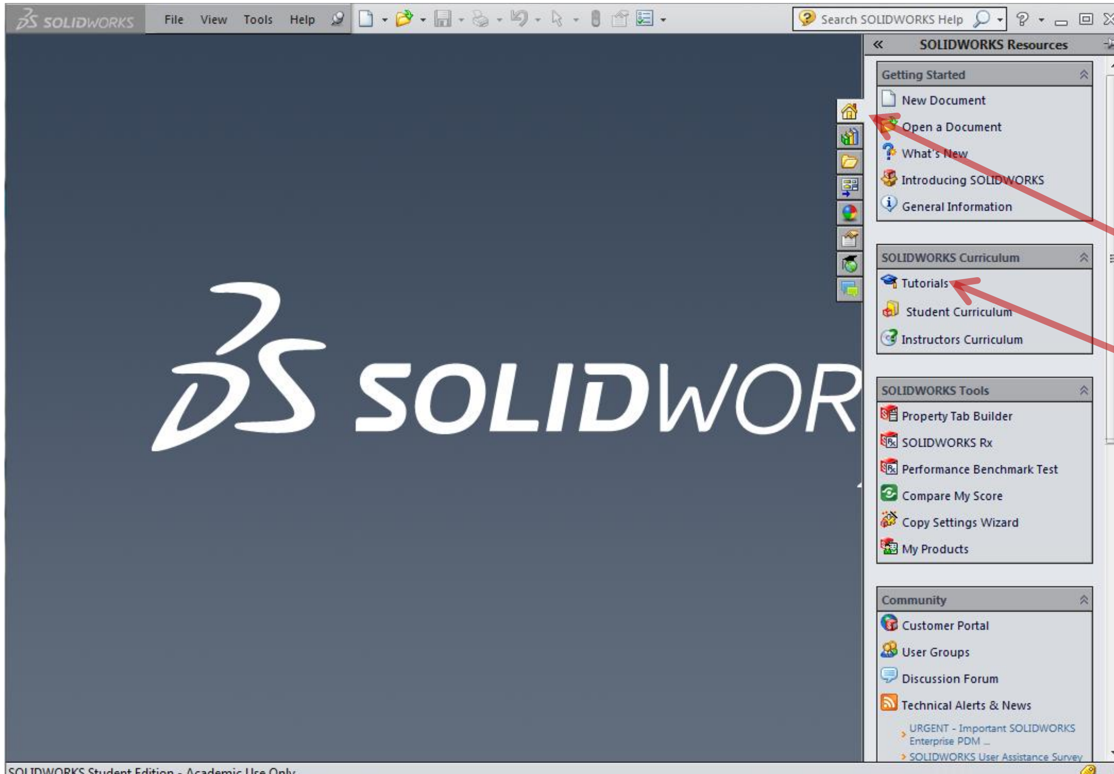
# SolidWorks Part 2

Prof. Steven S. Saliterman
Introductory Medical Device Prototyping
Department of Biomedical Engineering, University of Minnesota
http://saliterman.umn.edu/

# Starting The Tutorials

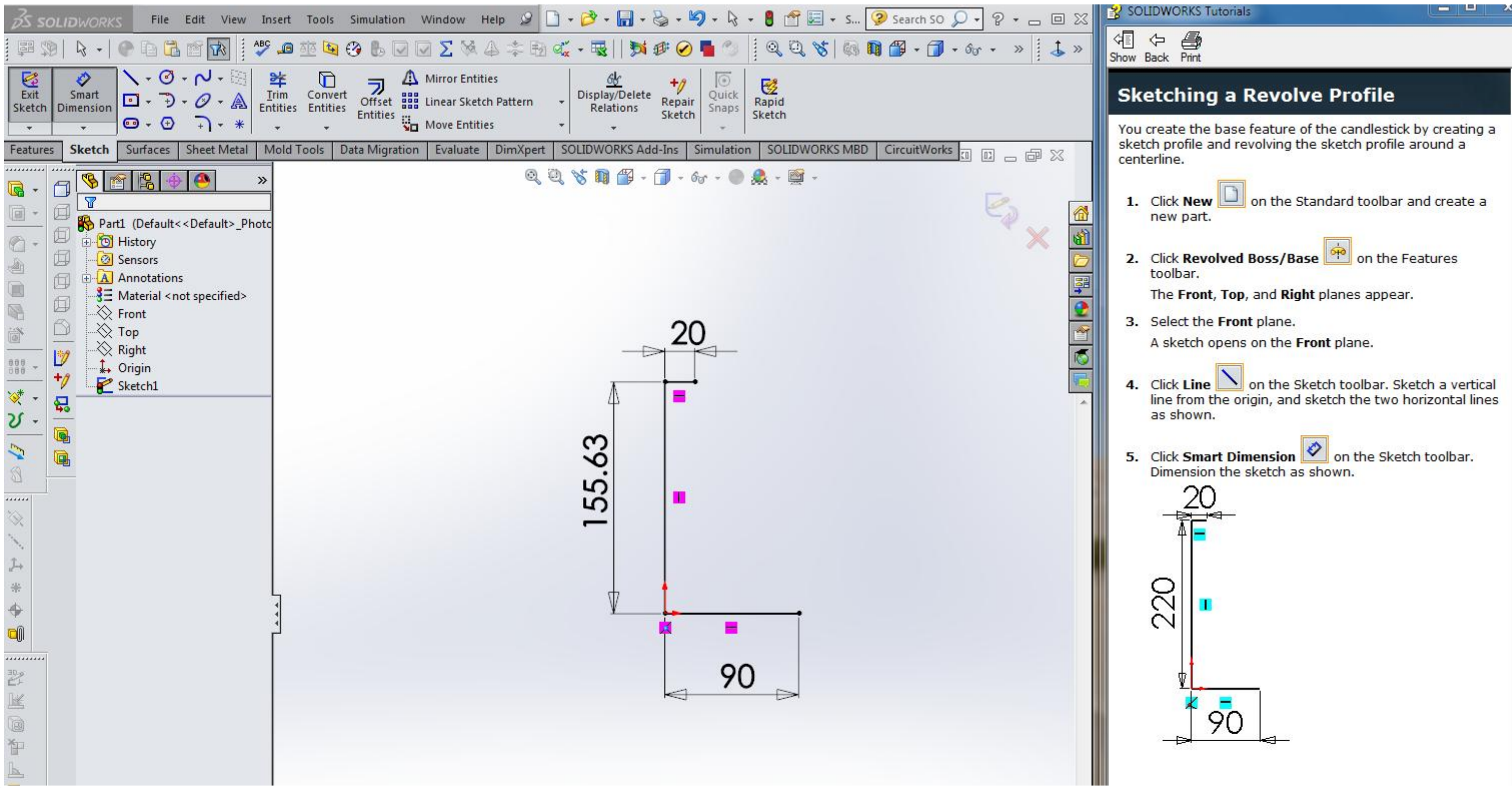



- Launch SolidWorks
- Select Resources to open the Task Pane.
- Select Tutorials

# Select Basic Techniques


Open Revolves and Sweeps

# Sketch the Revolve Profile

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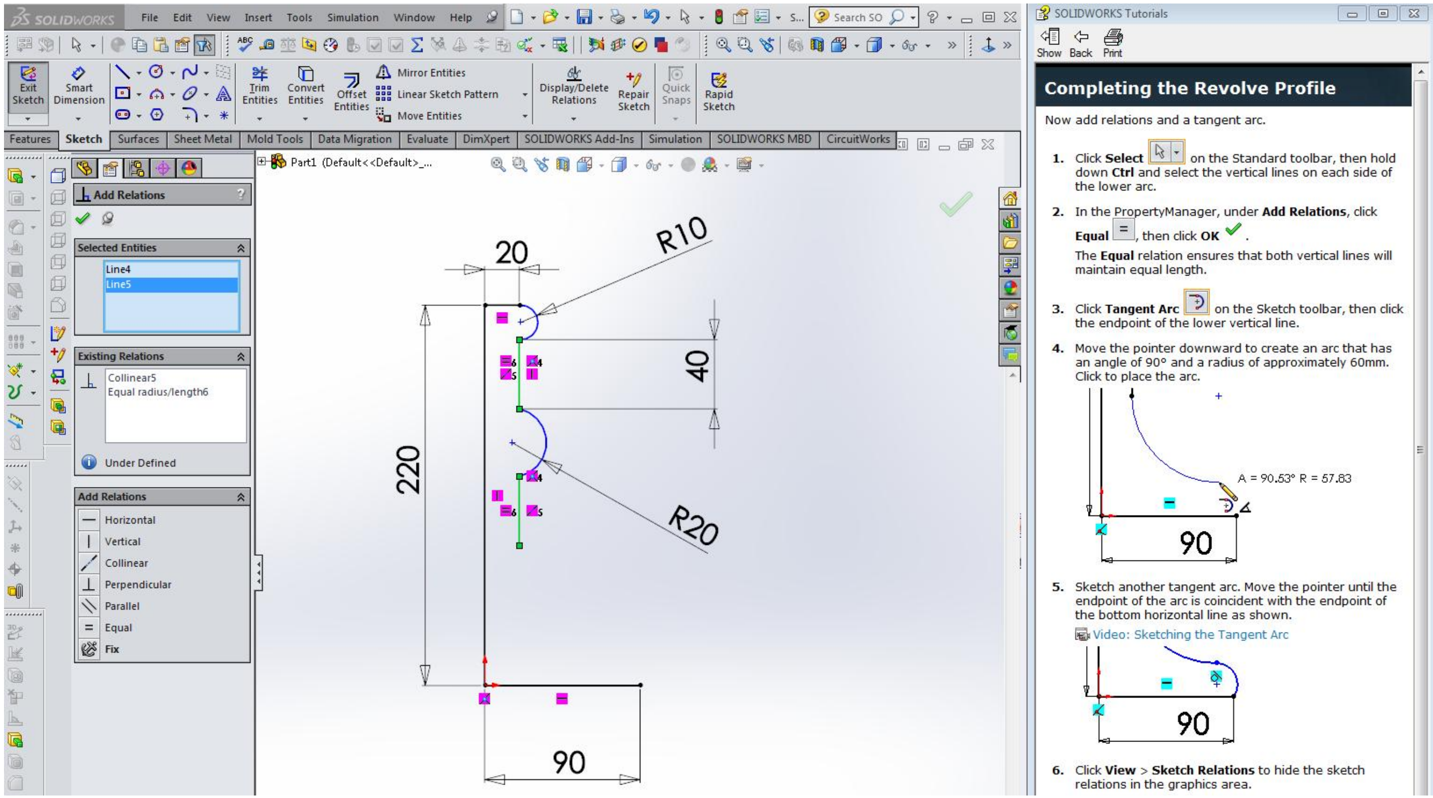
# Draw Arcs, Dimension & Trim

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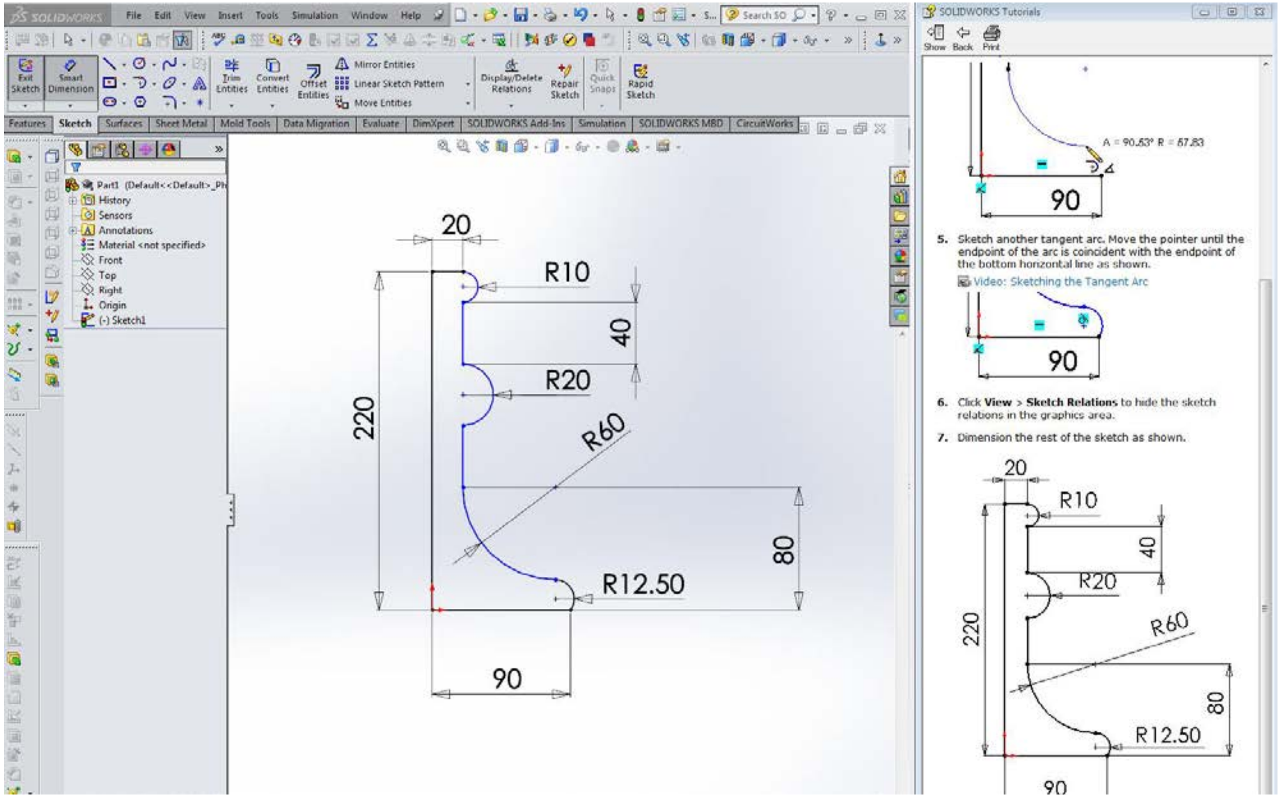
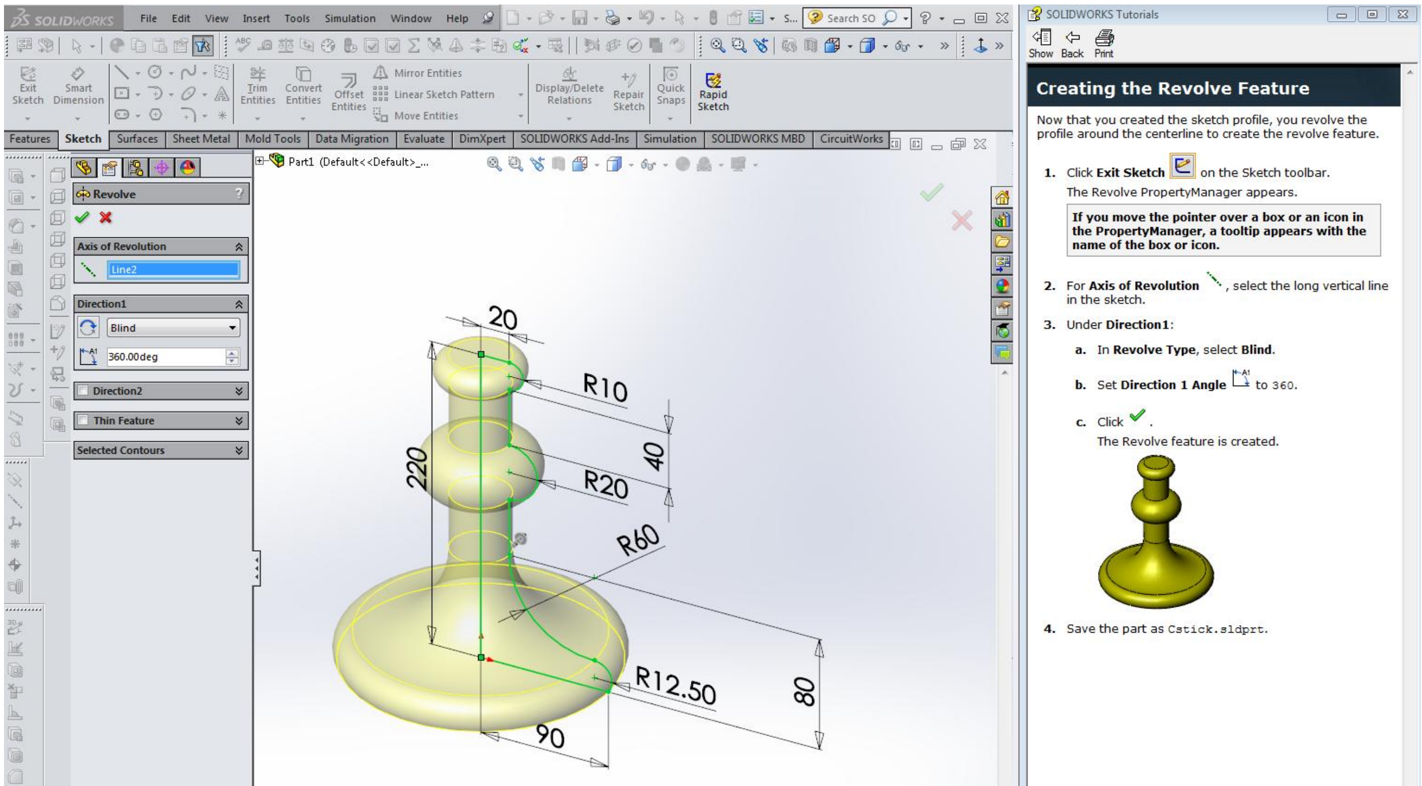
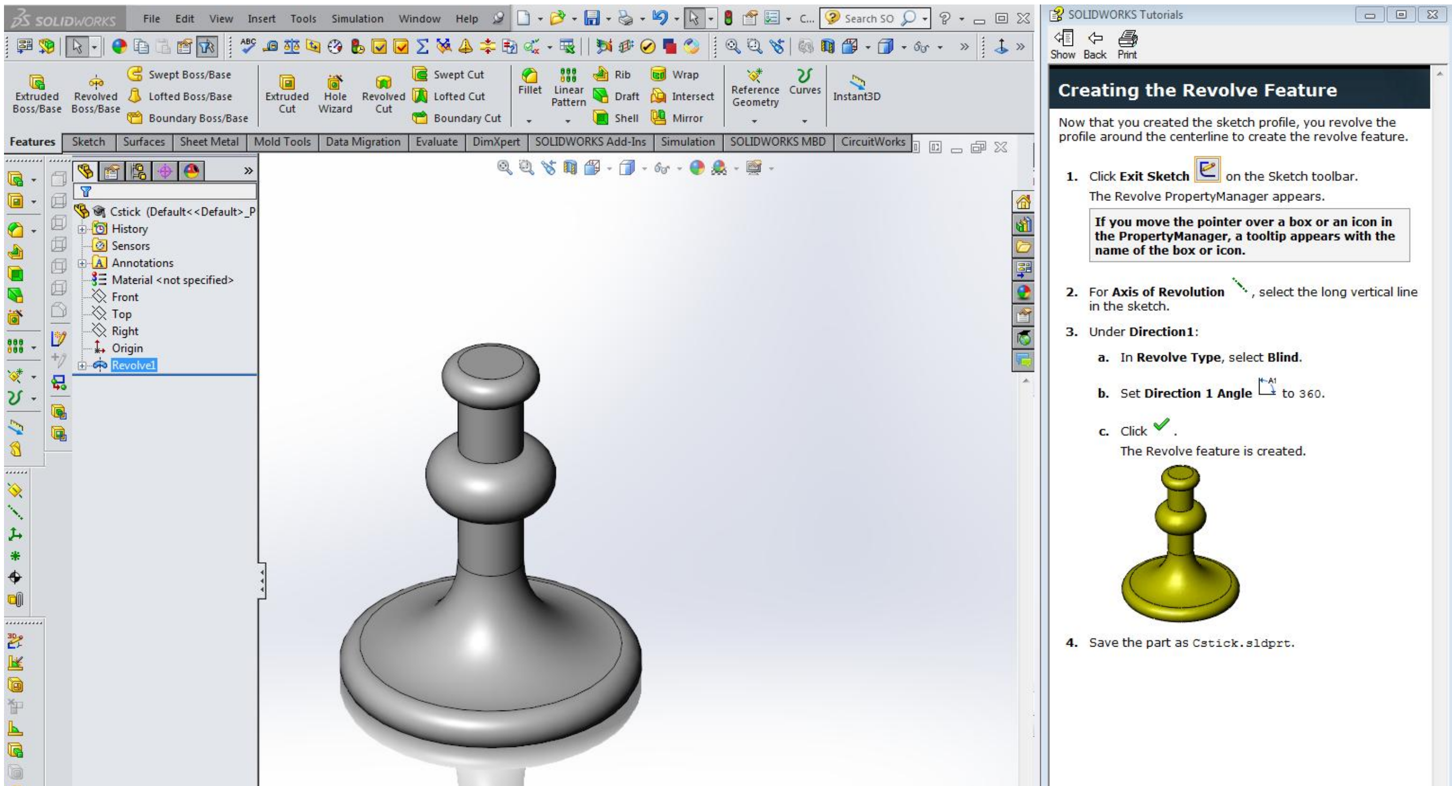
# Fully Dimensioned Profile



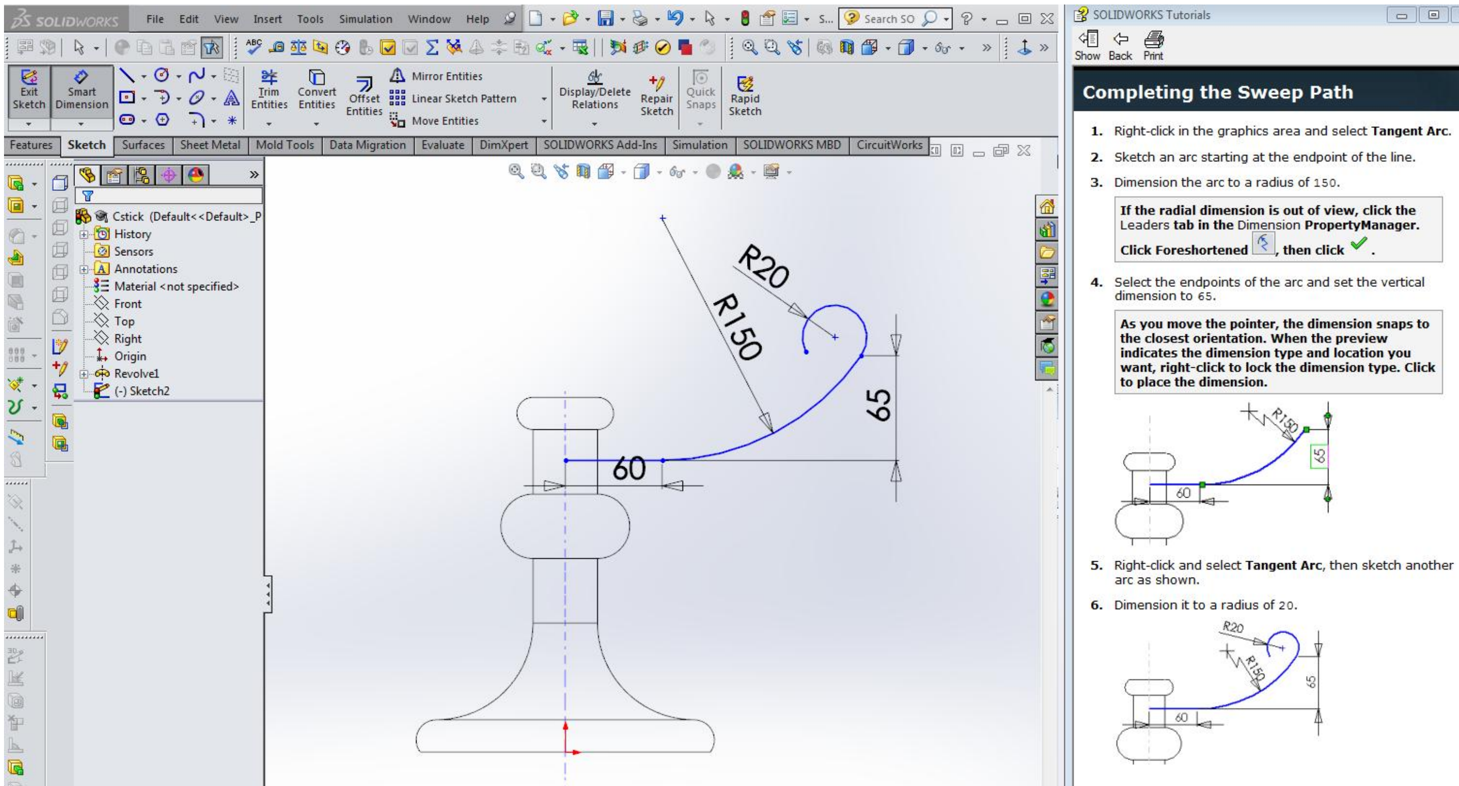
# Creating the Revolve Feature



# Revolve Appearance



# Completing the Sweep Path

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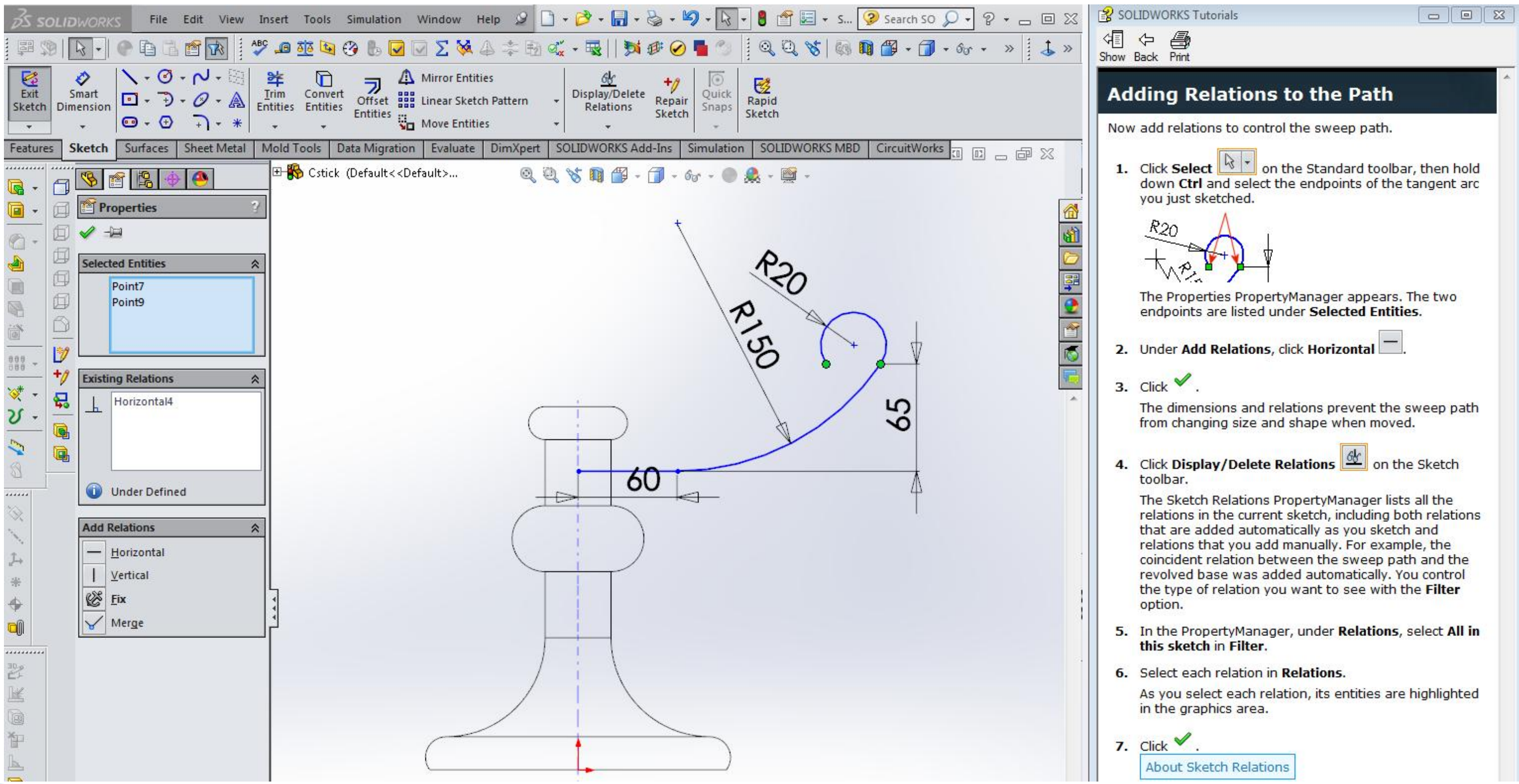
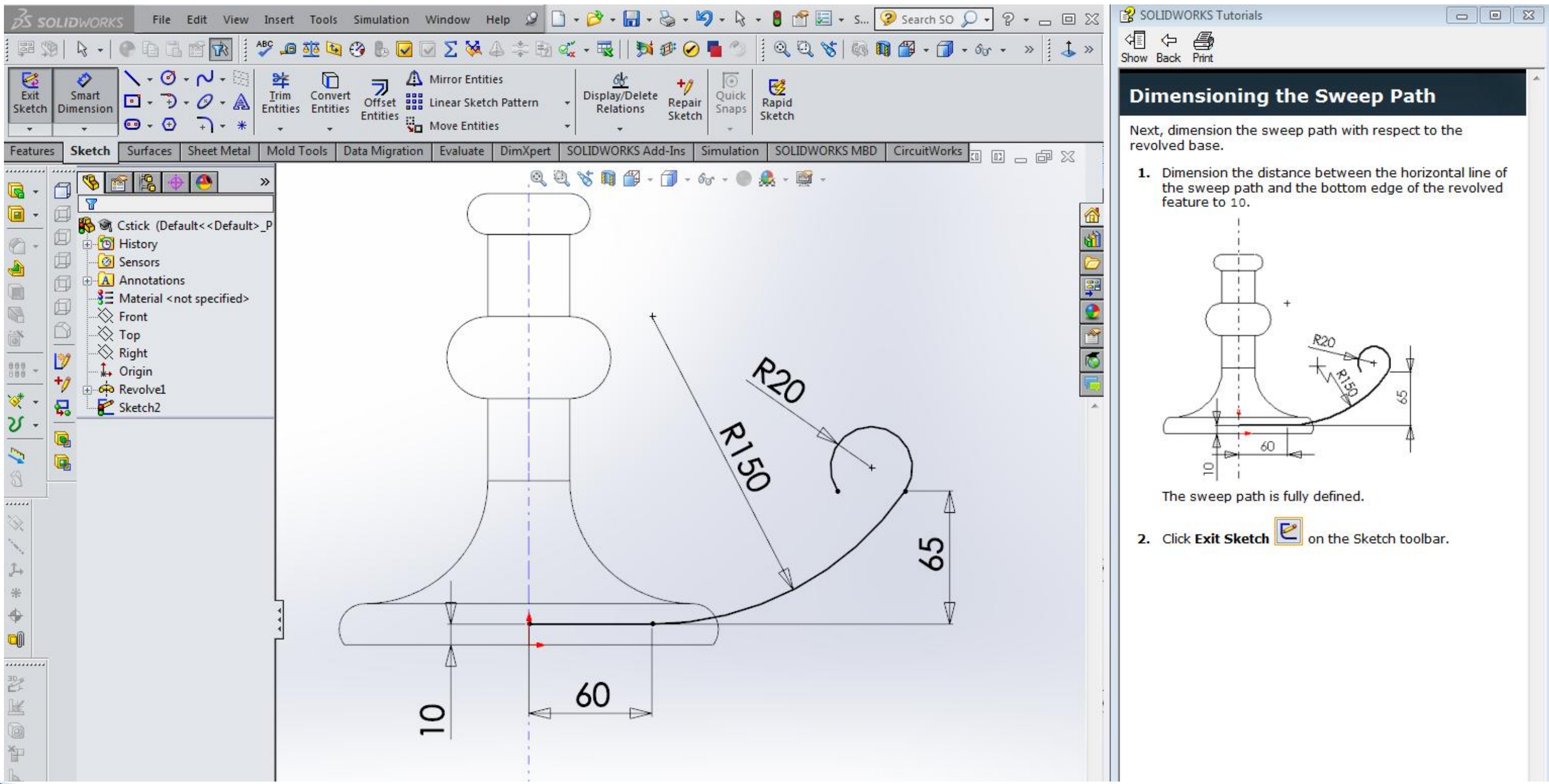
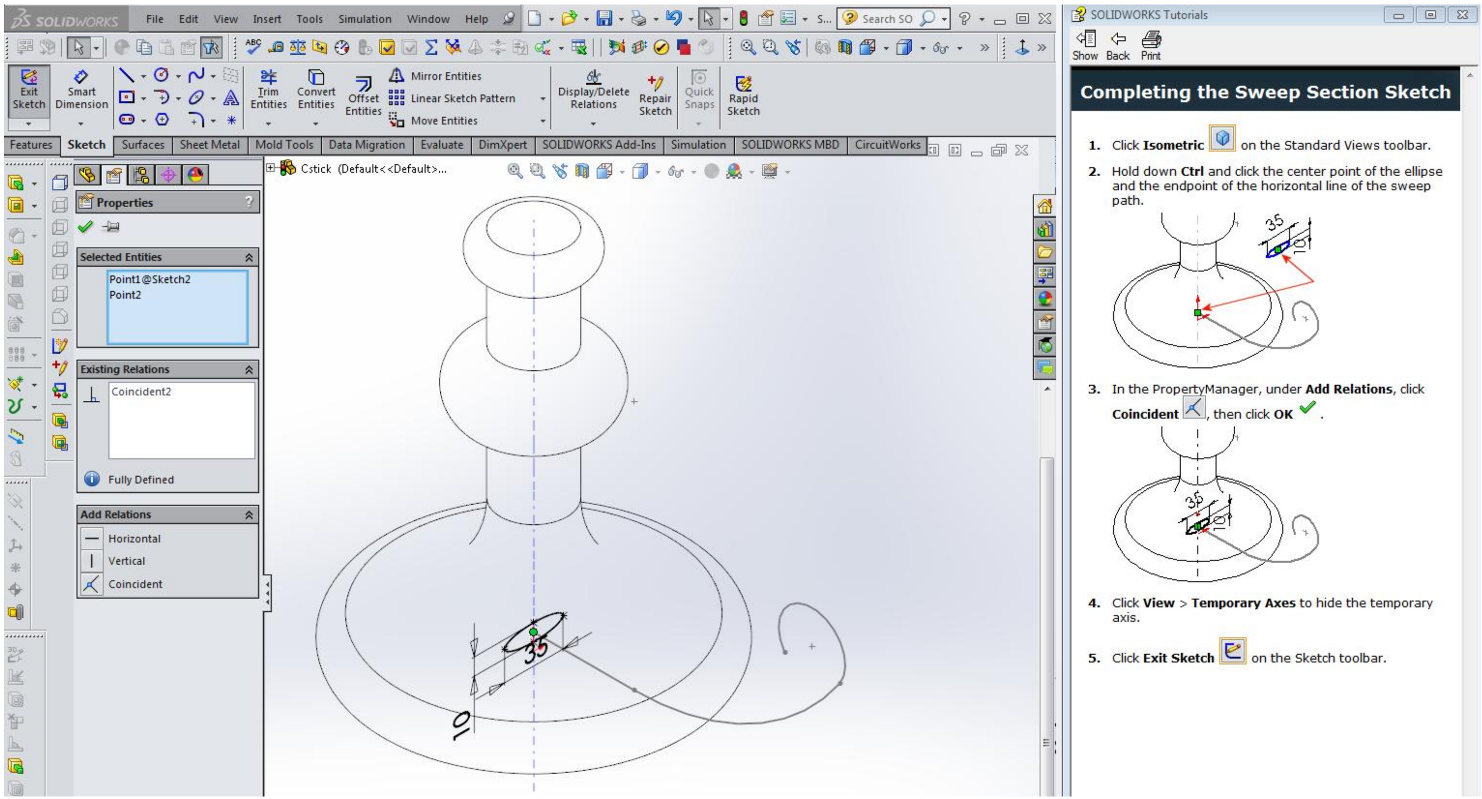
# Add Horizontal Relationship to Arc End Points



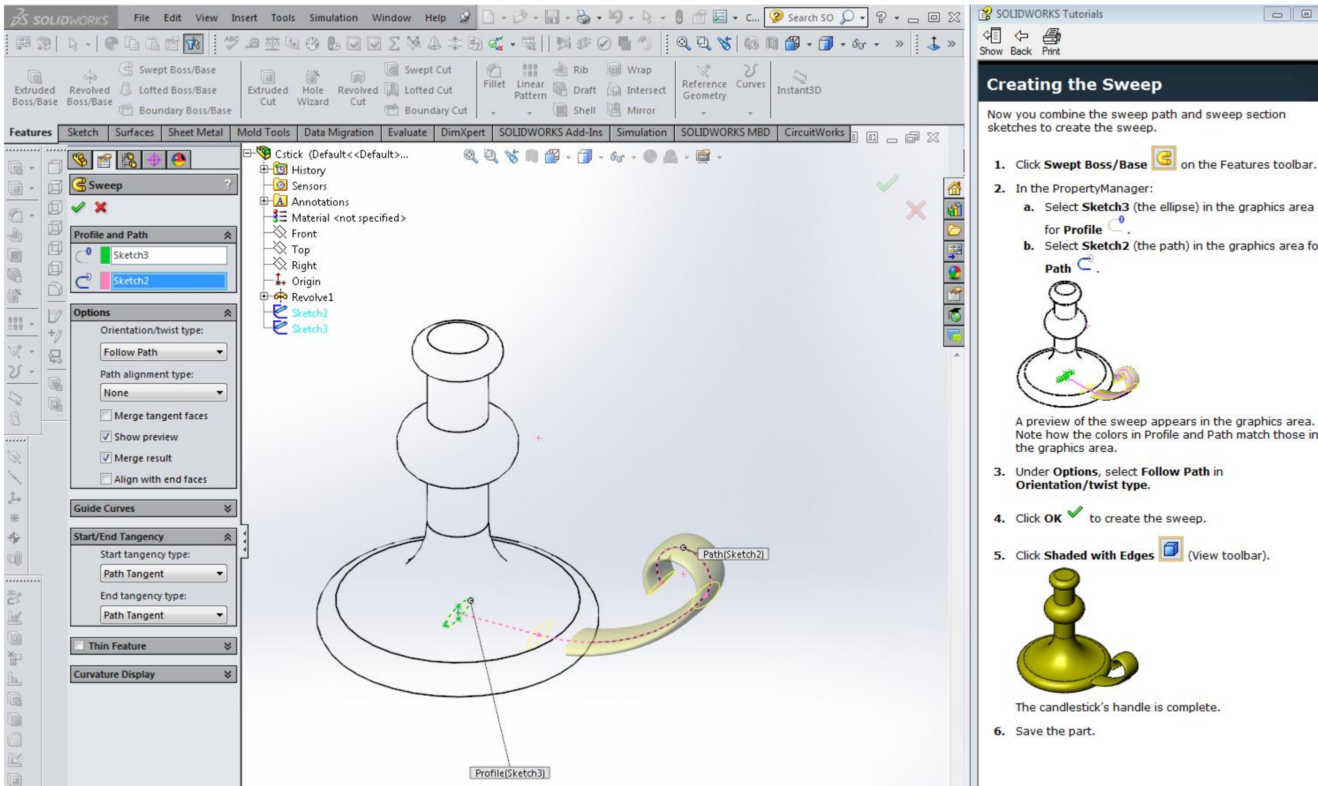
# Dimensioning Sweep Path Relative to the Base



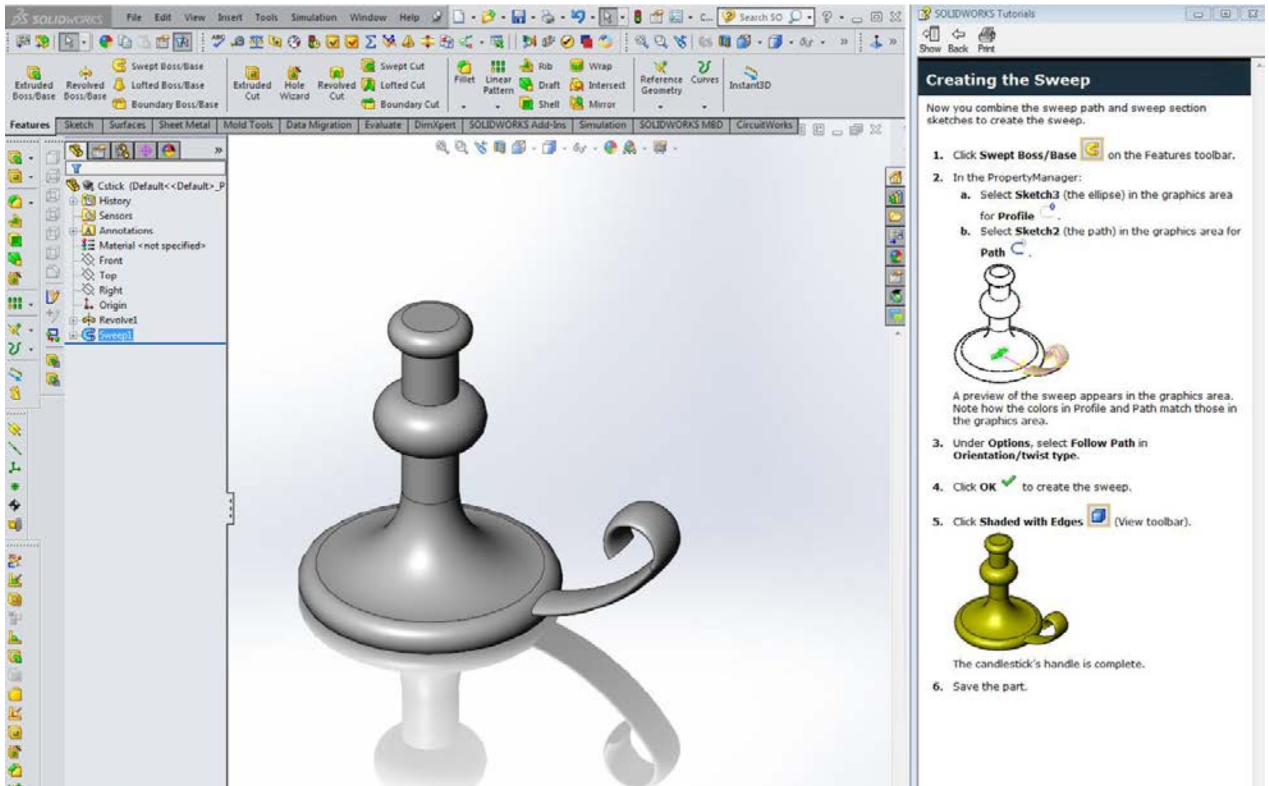
# Adding Coincident Relationship



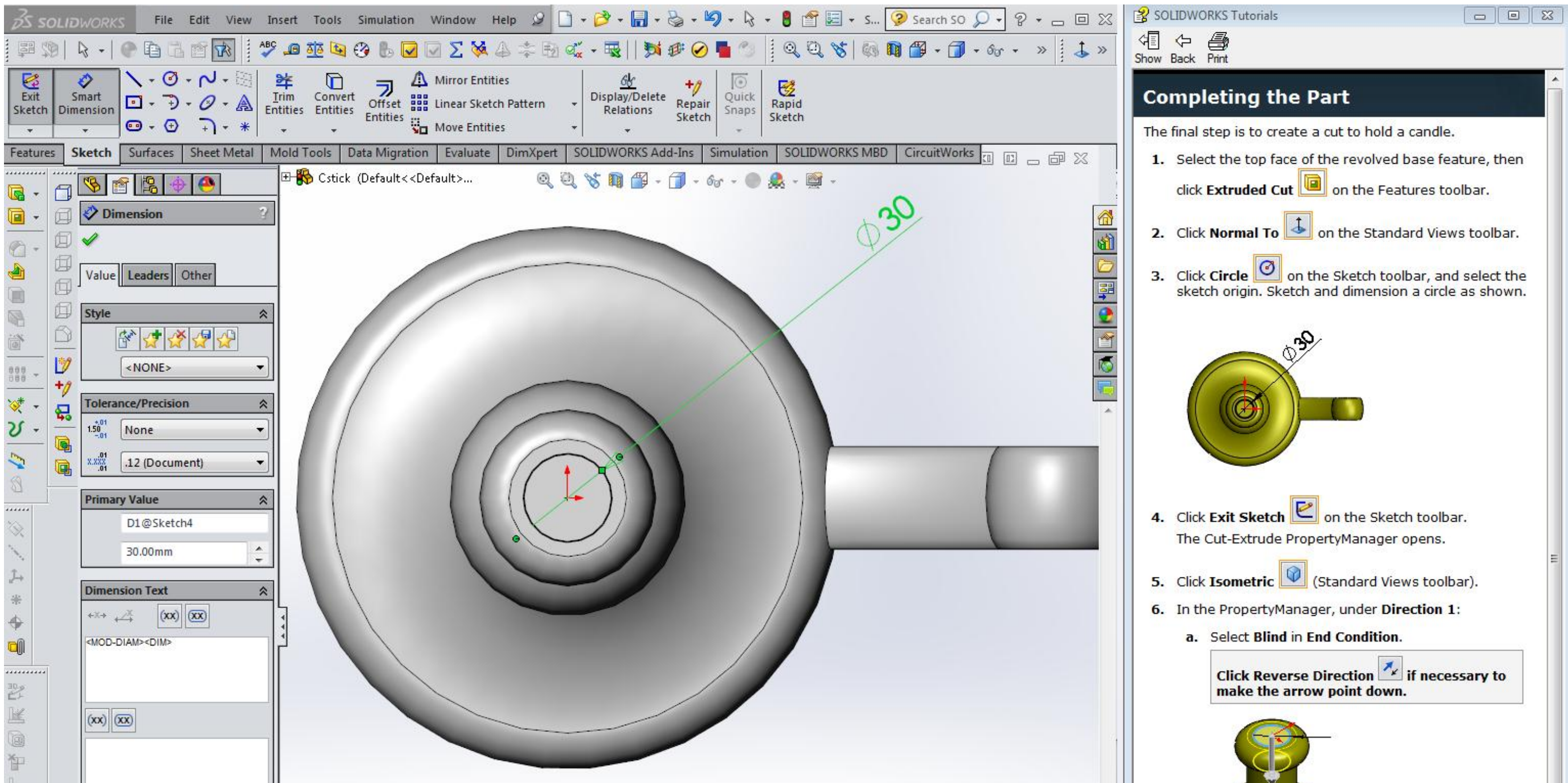
# Creating the Handle Sweep



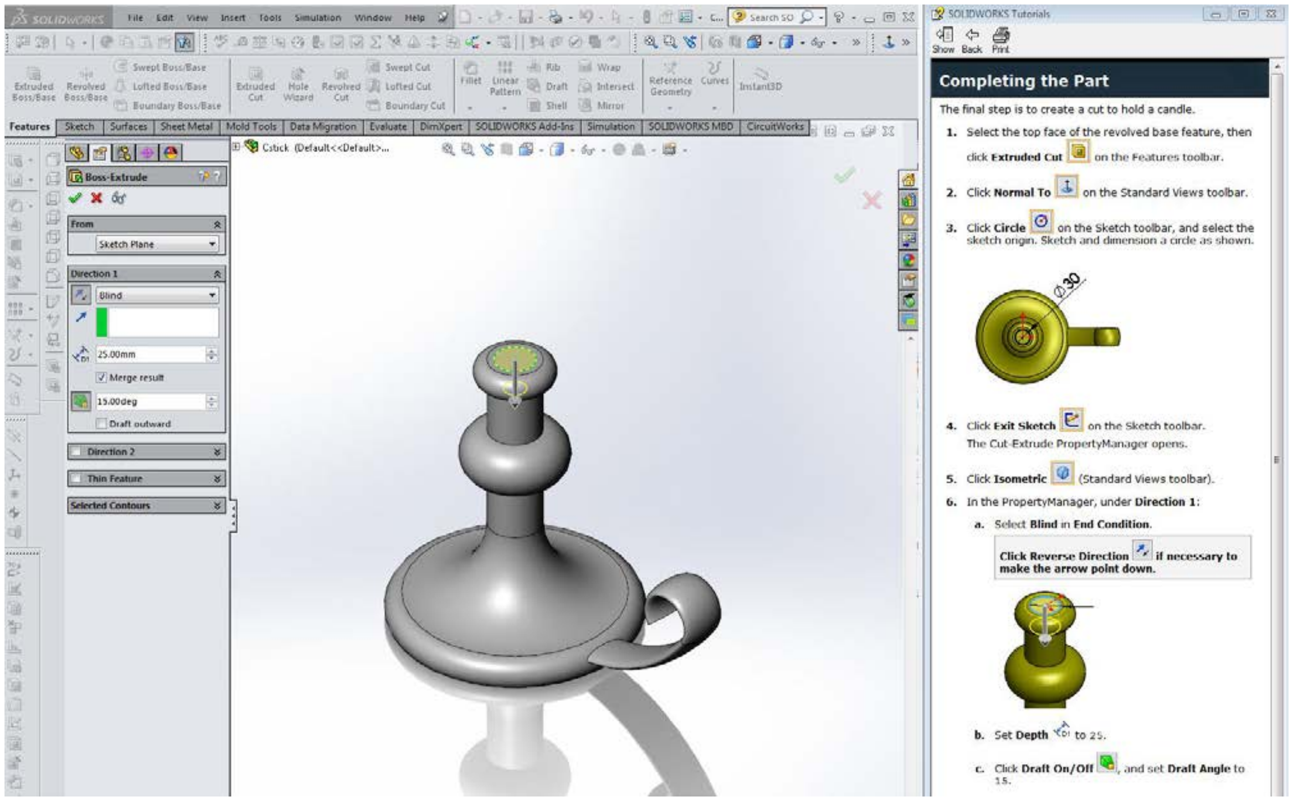
# Completion of Handle



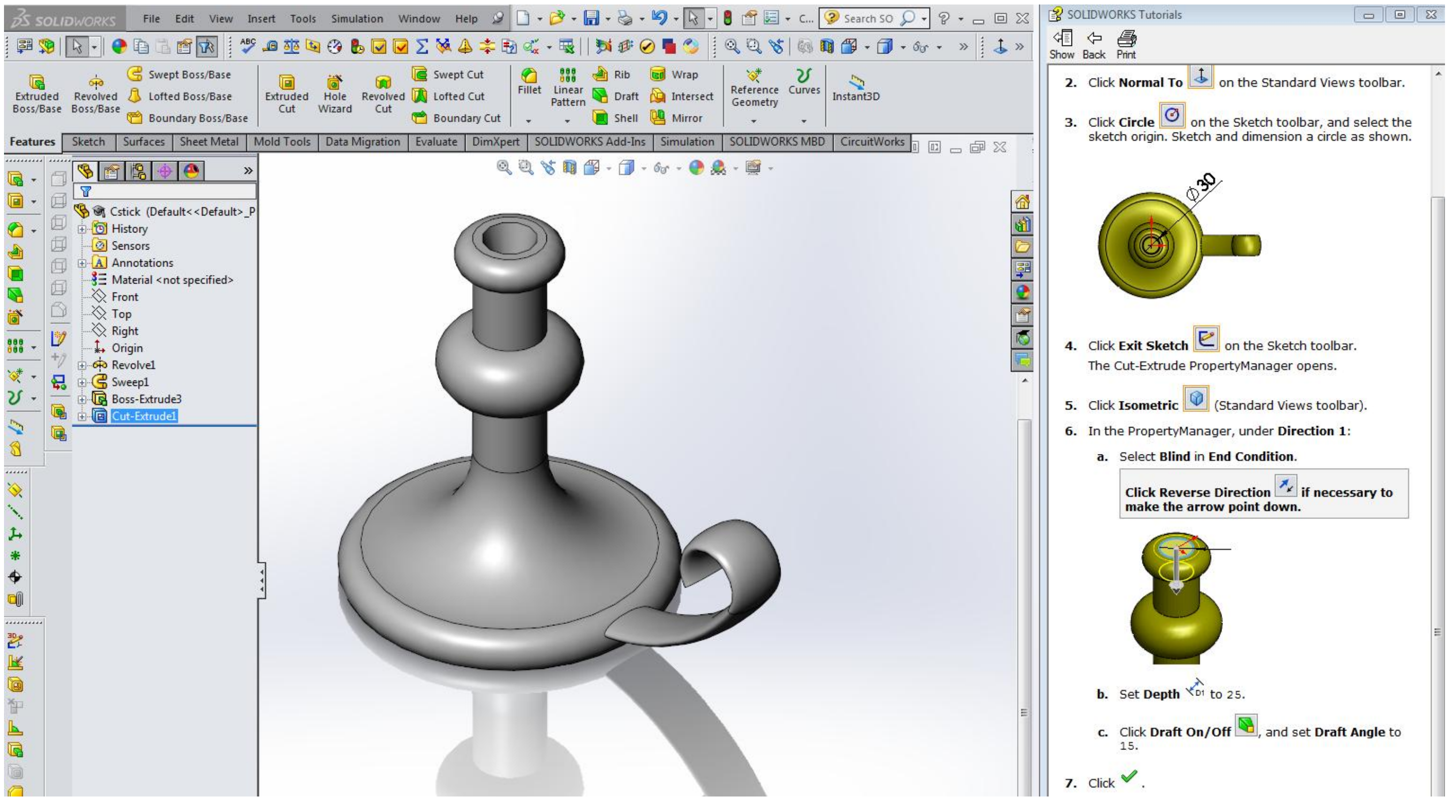
# Sketch Circle for Extruded Cut Feature



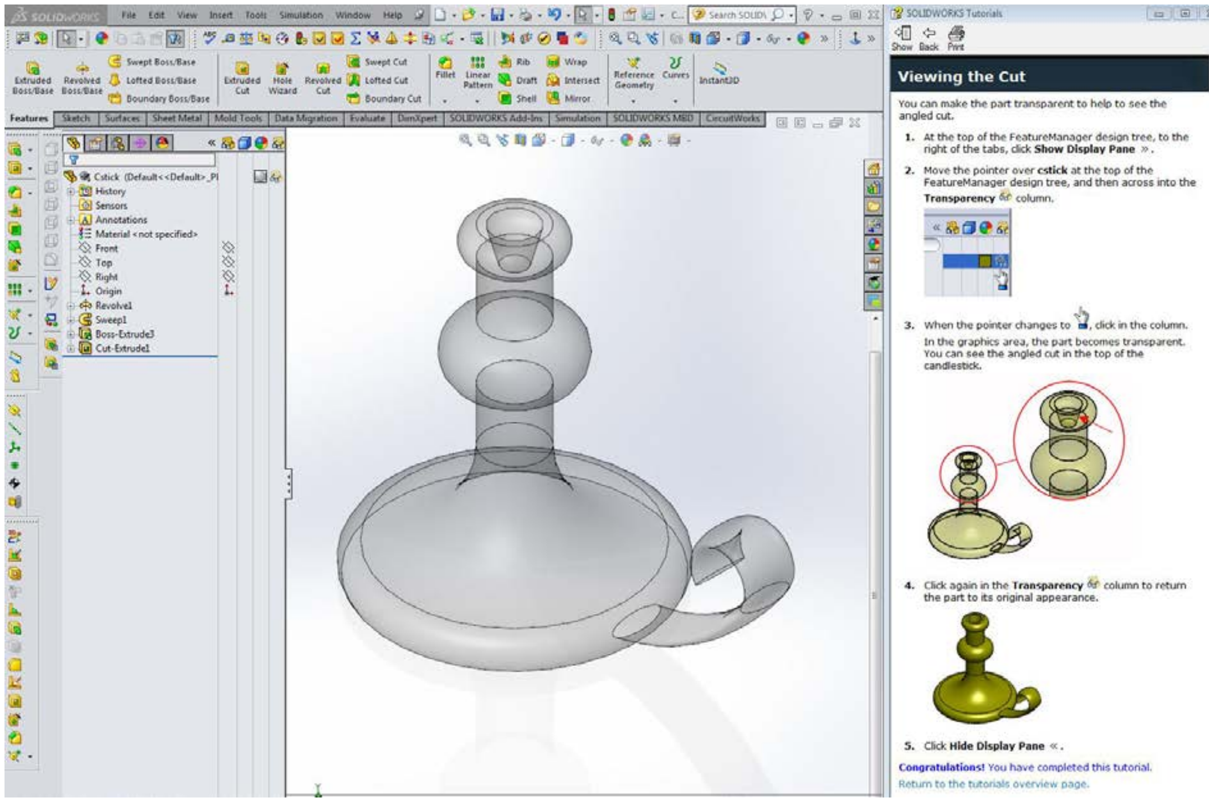
# Extruded Cut with Draft Angle



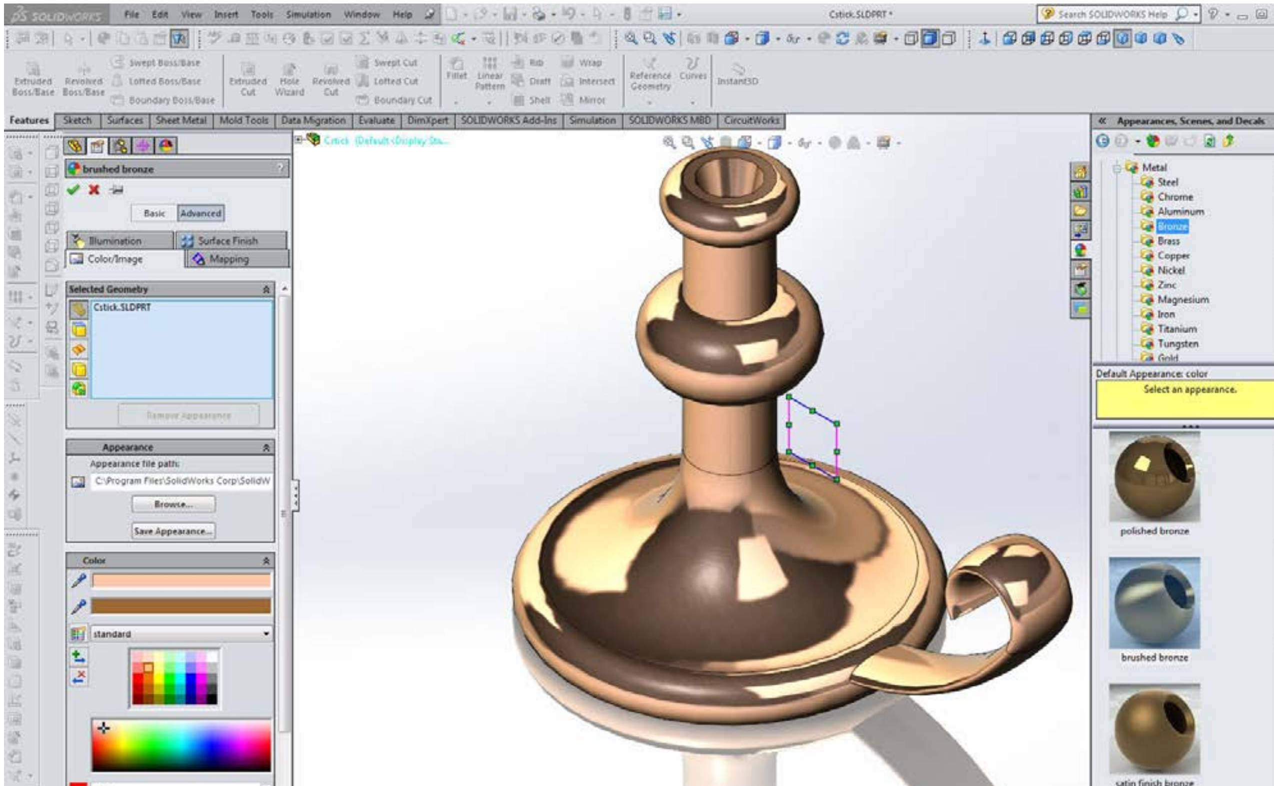
# Cut Added for Candle



# Making Cut Transparent



# Adding a Satin Bronze Finish

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# Summary

- Topics Covered in this exercise:
  - Sketching a revolve profile with lines, arcs, dimensioning, and trimming.
  - Adding dimensional relationships.
  - Creating a revolve.
  - Creating a sweep path and coincidental relationship.
  - Sweeping along the path.
  - Making an extruded-cut with draft.
  - Making a transparent view to see the cuts.
  - Making a Real View appearance.