

Module 12: Sweep Features I; Cranking Lever

The cranking lever details are shown in Figure 1.

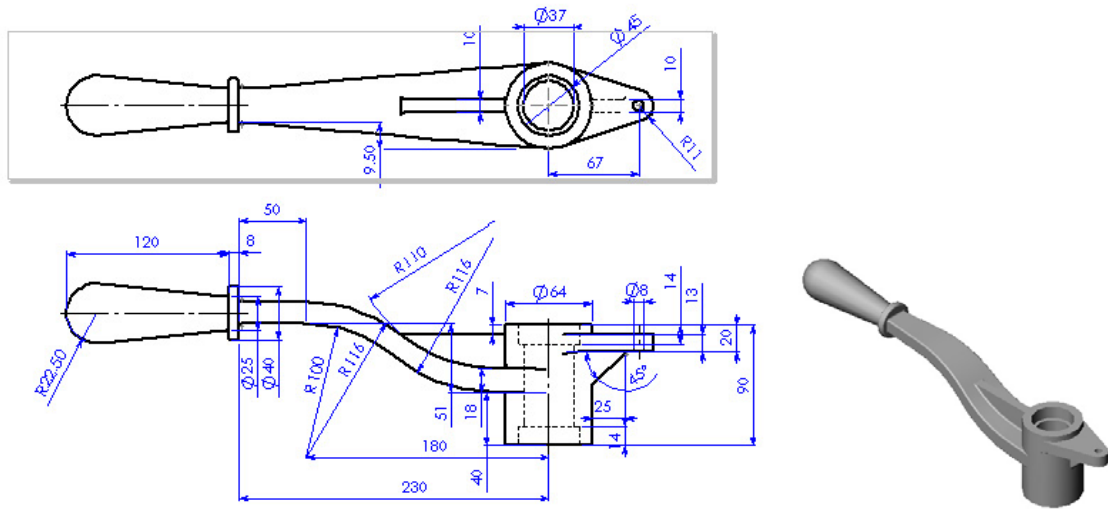


Figure 1 Cranking lever

It can be modeled by the following steps as shown in Figure 2.

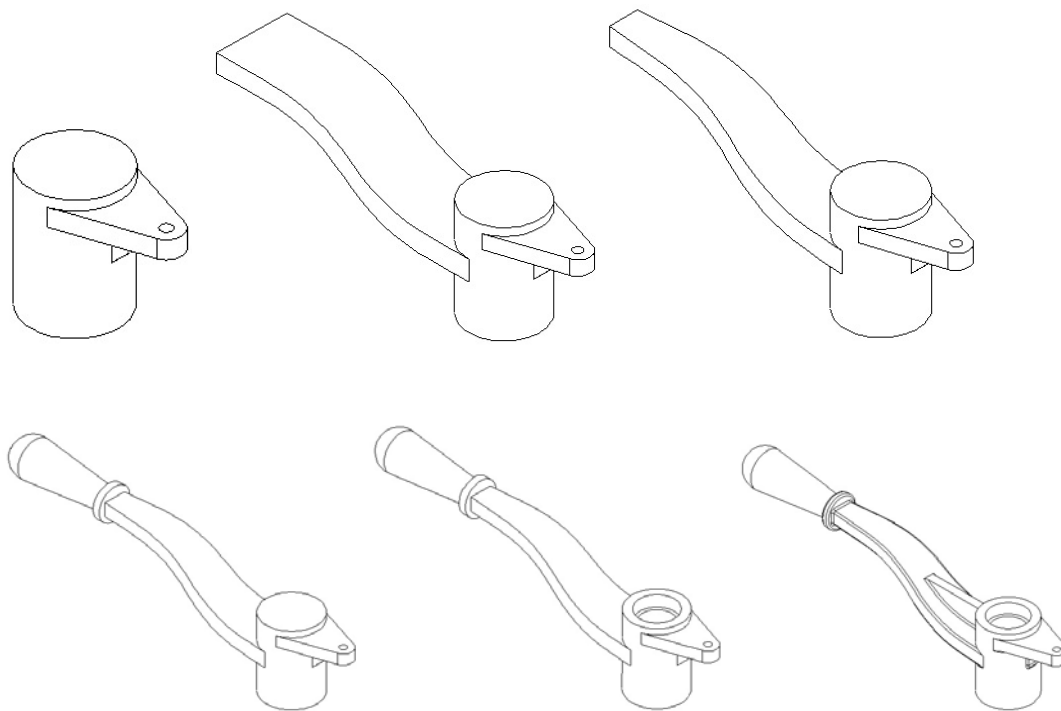







Figure 2 Cranking lever modeling process.

1. Create a cranking lever head using a revolved boss/base feature.
2. Create a crank lever head with an extruded boss/base and revolved cut
3. Add a handle to the handle body using the revolved boss/base.
4. Create two ribs fillets on the crank lever.

Create a Cranking Lever Head

1. Start SolidWorks.
2. Select **New**  on the Standard toolbar, or click **File, New** on the menu bar.
3. Select **Part**  from the **Template** tab in the dialog box, and click **OK**.
4. Click **File, Properties** on the menu bar.
5. Fill in the necessary properties in the dialog box, and click **OK** to accept the properties and close the dialog box.
6. Click **Tools, Options** on the menu bar to open the option dialog box.
7. Select the **Document Properties** tab.
8. Click **Unit** in the properties tree text box.
9. Select **Millimeter** and set the decimal places to 2.
10. Click **OK** to close the Properties dialog box.
11. Select **Sketch**  on the Sketch toolbar to open a new sketch on the **Front** Plane.
12. Select **Centerline**  on the Sketch Tools toolbar to draw a vertical centerline through the origin as the axis of the revolution.
13. Select **Line**  on Sketch Tools toolbar, and draw and dimension a profile of the cranking lever head as shown in Figure 3.

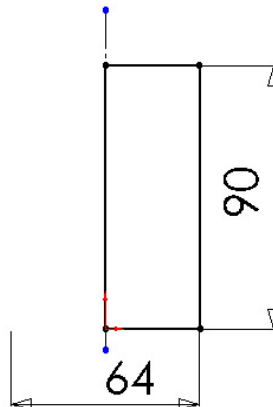






Figure 3 Cranking Lever Head Sketch

14. Select **Revolved Boss/Base**  on the Features toolbar, and click **OK** .
15. Select the **Front** plane on the FeatureManager design tree.
16. Select **Sketch**  on the Sketch toolbar to open a new sketch for the lever head rib.
17. Click **Normal To**  on the Standard Views toolbar to make the sketch plane parallel to the screen.
18. Sketch and dimension the lever head rib as shown in Figure 3.

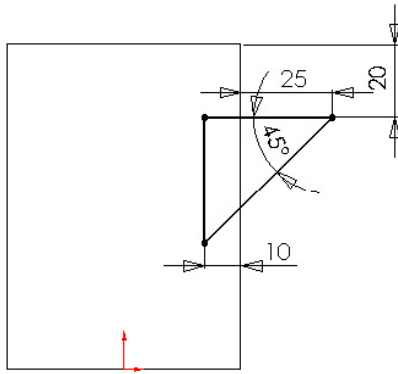




Figure 3 Lever head rib sketch.

19. Select **Extruded Boss/Base**  on the Features toolbar.
20. Select **Mid Plane** on the **End Condition** drop down list box.
21. Type **10mm** in the depth box under **Direction 1**, and click **OK** .
22. Select the top surface of the rib as shown in Figure 4, and open a sketch.

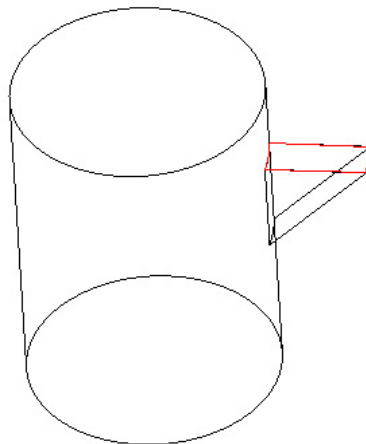


Figure 4 Sketch plane for lever head ear.

23. Sketch and dimension the lever head ear as shown in Figure 5.

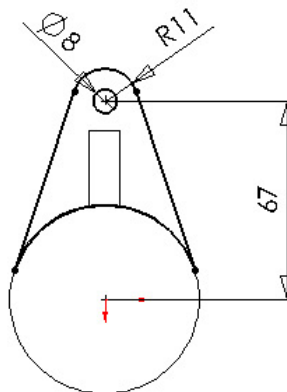




Figure 5 Lever Head Ear Profile

24. Select **Extruded Boss/Base**  on the Features toolbar.
25. Enter **13 mm** in the depth box under **Direction 1** as in Figure 6.
26. Click **OK** .

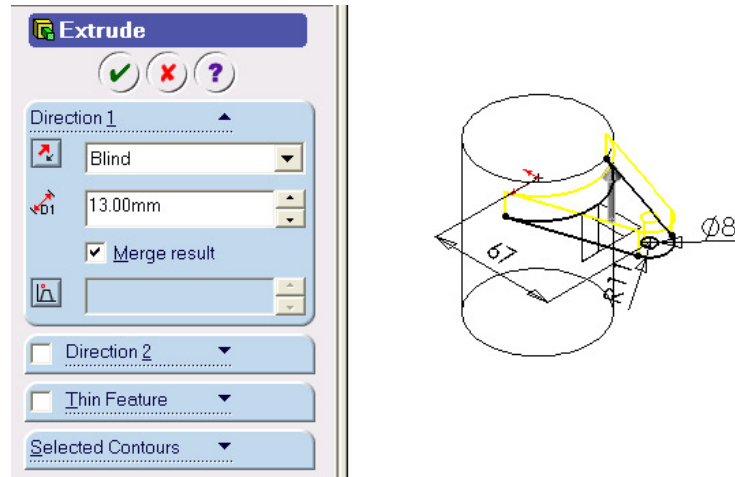



Figure 6 Lever Head Ear Extrusion

Create a Handle Body using Sweep

1. Select the **Front** on the FeatureManager design tree.
2. Select **Sketch**  on the Sketch toolbar to open a new sketch for the sweep guide curve.
3. Sketch and dimension the sweep guide curve as shown in Figure 7.

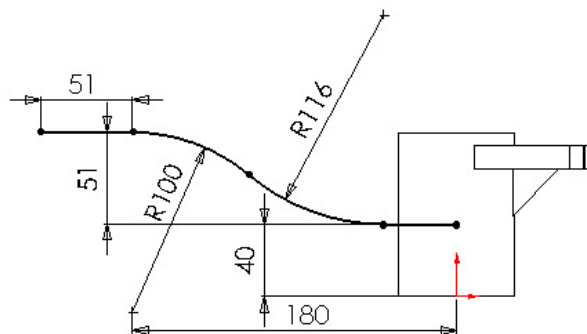





Figure 7 Sweep path

4. Click **Confirmation Corner**  or select **Rebuild**  on the Standard toolbar to exit the sketch.
5. Select **Front** on the FeatureManager design tree.
6. Select **Sketch**  on the Sketch toolbar to open a new sketch for the sweep guide curve.
7. Sketch and dimension the sweep guide curve as shown in Figure 8.

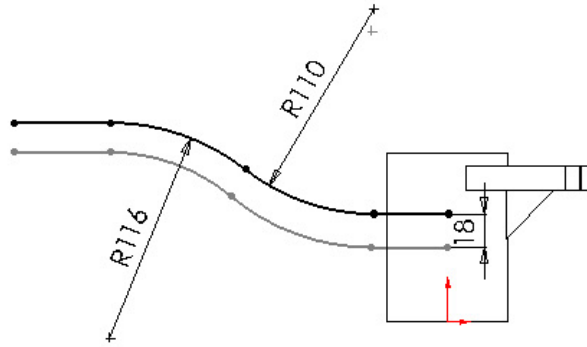







Figure 8 Sweep guide curve.

8. Click **Confirmation Corner**  or select **Rebuild**  on the Standard toolbar to exit the sketch.
9. Select **Insert, Reference Geometry, Plane** on the menu bar or Click **Plane**  on Reference Geometry Toolbar.
10. Click the **Offset Distance**  button.
11. Select the **Right** plane on the FeatureManager design tree, and enter **230 mm** in the Distance  box.
12. Check the **Reverse direction** check box as shown in Figure 9.

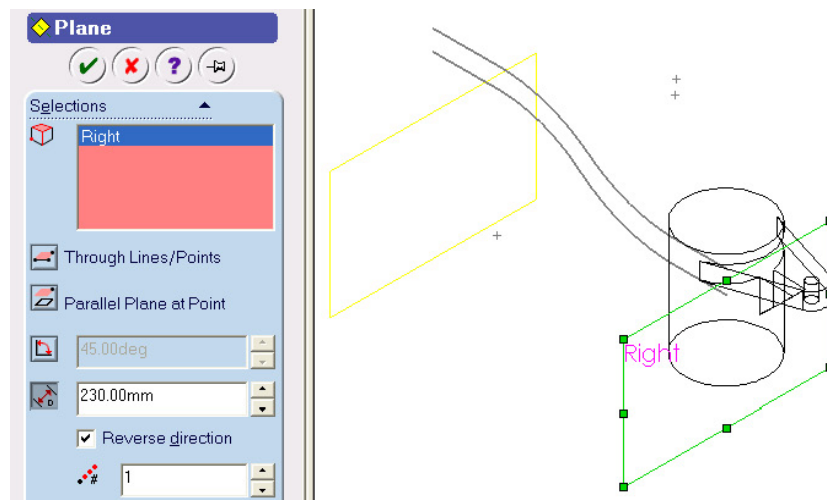







Figure 9 Reference plane for sweep profile

13. Click **OK** .
14. Select the **Plane1** on the FeatureManager design tree or in the graphic window.
15. Select **Sketch**  on the Sketch toolbar to open a new sketch for the sweep profile.
16. Sketch a rectangle. Then, sketch 2 points on the middle of the horizontal edges.
17. Click **Add Relation**  on the Sketch Relation toolbar.
18. Apply **Midpoint**  constraint to these two points.

19. Select the sweep path, and the bottom point, and apply **Pierce**  constraint as shown in Figure 10.

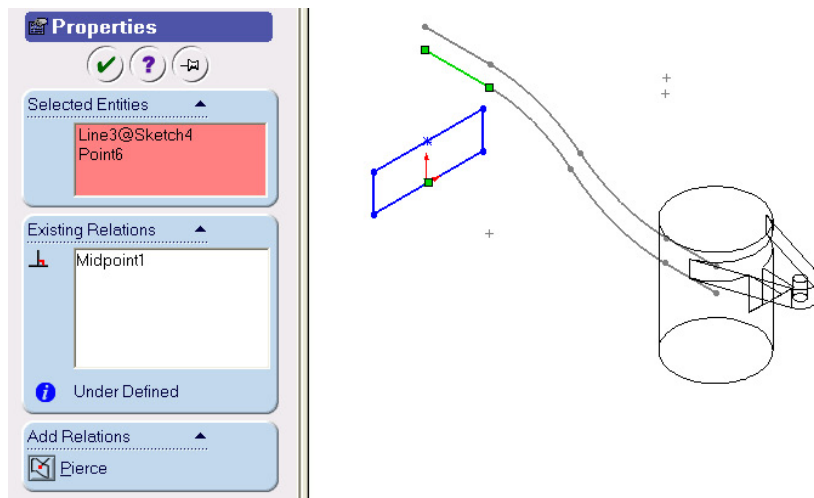



Figure 10 Pierce Constraints between Profile and the Path

20. Select the guide curve, and the top point, and apply **Pierce**  constraint as shown in Figure 11.

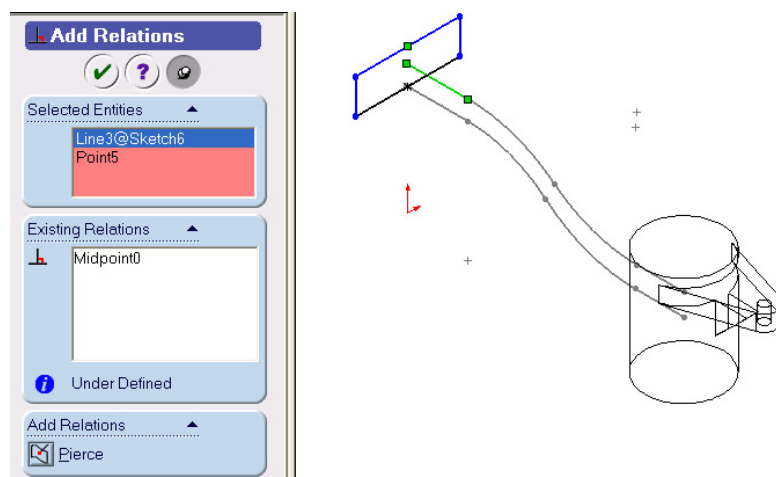


Figure 11 Pierce Constraint between Profile and Guide Curve

21. Dimension the width of the rectangle to **64 mm** as shown in Figure 12.

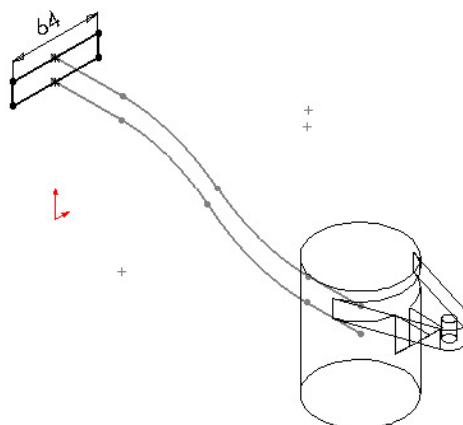





Figure 12 Profile dimension

22. Click **Confirmation Corner**  or click **Rebuild**  on the Standard toolbar to exit the sketch.
23. Select **Sweep Boss/Base**  on the Features toolbar.
24. Select the path (**Sketch 4**) in the path box under the Profile and Path frame.
25. Select the guide curve in the **Guide Curves 4** box as in Figure 13.

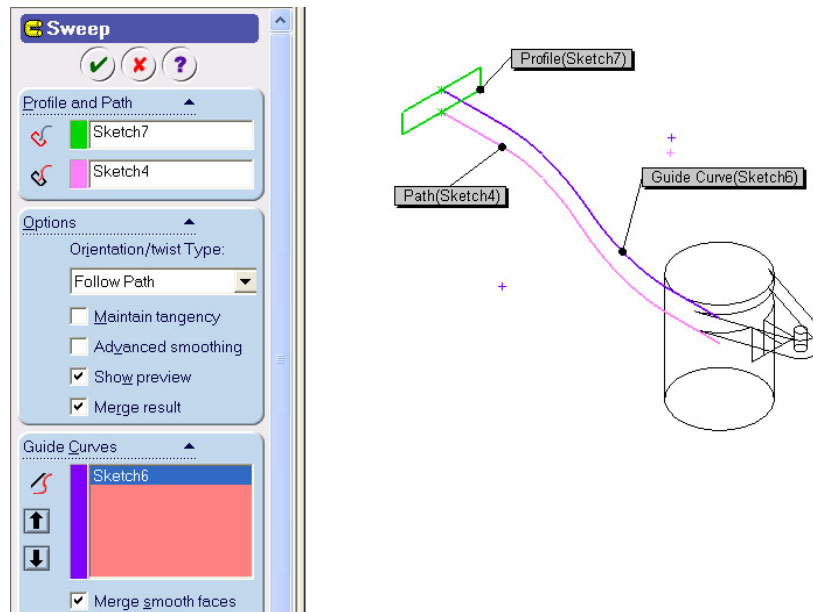



Figure 13 Sweep parameters.

26. Select **OK** .
27. Select the bottom surface of the lever head as shown in Figure 14, and open a sketch.

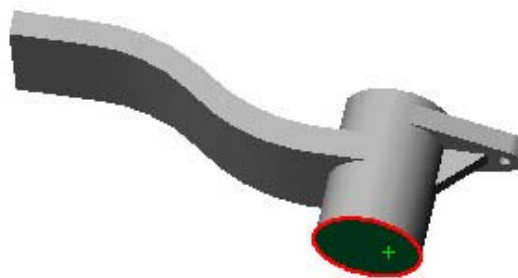




Figure 14 Sketch plane for lever handle cut.

28. Select the head circle, and click **Convert Entities**  on the Sketch Tools toolbar to create a circle on the sketch plane.
29. Sketch three lines, and **Trim**  the circle to form the cut profile as shown in Figure 15.

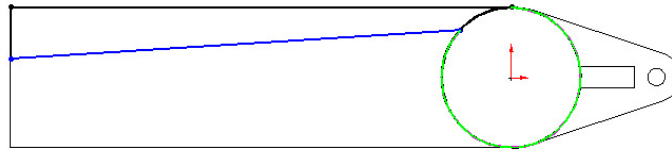




Figure 15 Lever Body Cut Sketch

30. Select **Add Relation**  on the Sketch Relations toolbar. Select the arc and the angled line, and apply a **Tangent**  constraint to them.
31. Dimension the sketch as shown in Figure 16.

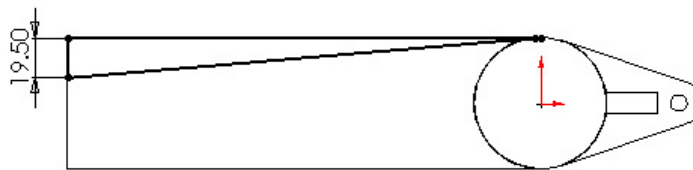


Figure 16 Lever body cut profile





32. Select **Extruded Cut**  on the Features toolbar. Select **Through All** on the end condition drop down list box, then click **OK** .
33. Select the last Feature created in FeatureManager Design Tree, i.e., **Cut-Extrude 2**, you may or may not have a different name for this Feature.
34. Select **Mirror**  on the Features toolbar. The **body cut (Cut-Extrude 2)** should be selected as the **mirror feature**. If not, select it. Then select **Front** plane on the FeatureManager design tree as the **mirror plane**.
35. Click **OK**  to mirror the cut to the other side as shown in Figure 17.



Figure 17 Lever body

Create Lever Handle and Head Hole Using Revolved Features

1. Select the **Front** plane on the FeatureManager design tree to open a sketch for the lever handle.
2. Sketch and dimension the profile as shown in Figure 18.

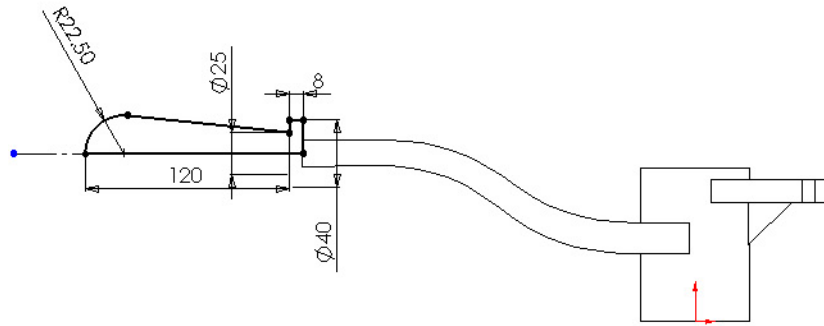




Figure 18 Lever handle profile.

3. Select **Revolved Boss/Base**  on the Features toolbar.
4. Click **OK**  to create the revolved lever handle.
5. Select the **Front** plane again in the FeatureManager design tree to open a sketch for the head hole cut.
6. Sketch the hole cut profile as shown in Figure 19.

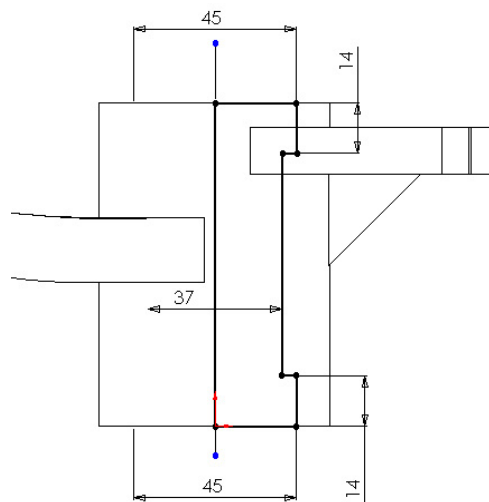



Figure 19 Head hole profile.

7. Select **Revolved Cut**  on the Features toolbar, and Click **OK**  create the head hole

Create Handle Rib and Fillets

1. Select the **Front** plane in the FeatureManager design tree to open a sketch for the rib.
2. Sketch and dimension the profile using **Line**  as shown in Figure 20.

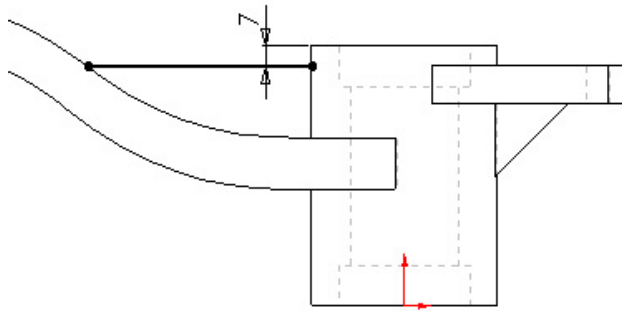






Figure 20 Handle rib profile.

3. Select **Rib**  on the Features toolbar. Select **Both Sides** , enter **10 mm** in the **Rib Thickness**  box and check **Flip material side** as shown in Figure 21.
4. Click **OK**  to create the rib.

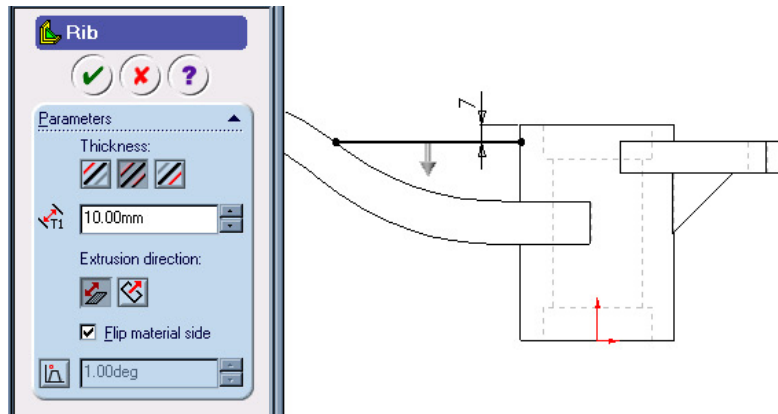



Figure 21 Handle rib parameters.

5. Select **Fillet** , and enter **2 mm** in the **Radius**  box.

Select edges around the rib edges, handle edges as in Figure 22. Then click **OK**  to create fillets. The final model for Cranking Lever is shown in Figure 23.

6. **Save the Model.**

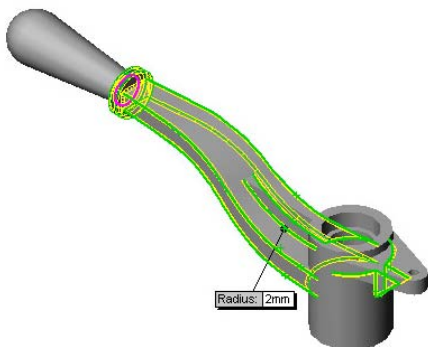


Figure 22 Cranking Lever fillets.

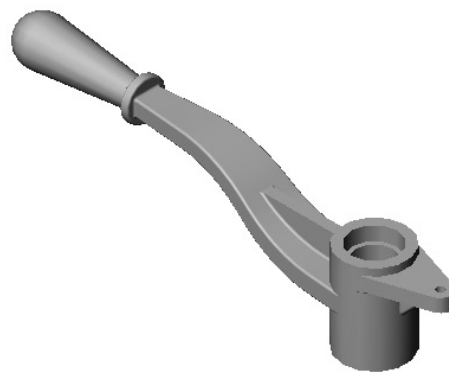


Figure 23 Cranking Lever model.