

Module 11: Reference Planes; Vertical Bearing Support

In this Tutorial, the creation and usage of reference geometries are demonstrated. The model of Vertical Bearing Support is as shown in Figure 1.

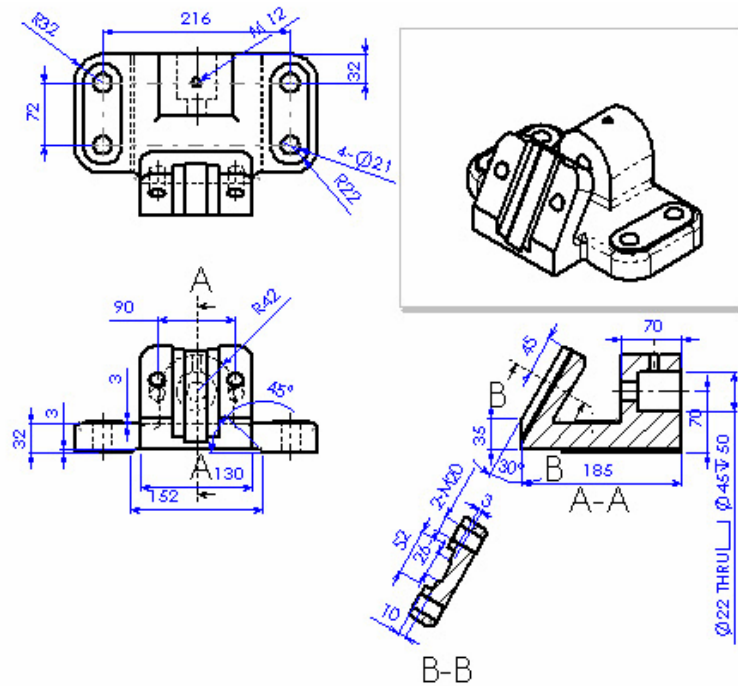


Figure 1 Vertical Bearing Support

It can be built by following the procedures as shown in Figure 2.

1. Create a base using Extruded Boss/Base.
2. Add a slot boss for creating the slot body.
3. Create a plane for slot body with **At Angle**, and then a Slot body.
4. Cut the Slots on the slot body and the base.
5. Add hole bosses for drilling holes.
6. Drill holes and add fillets.

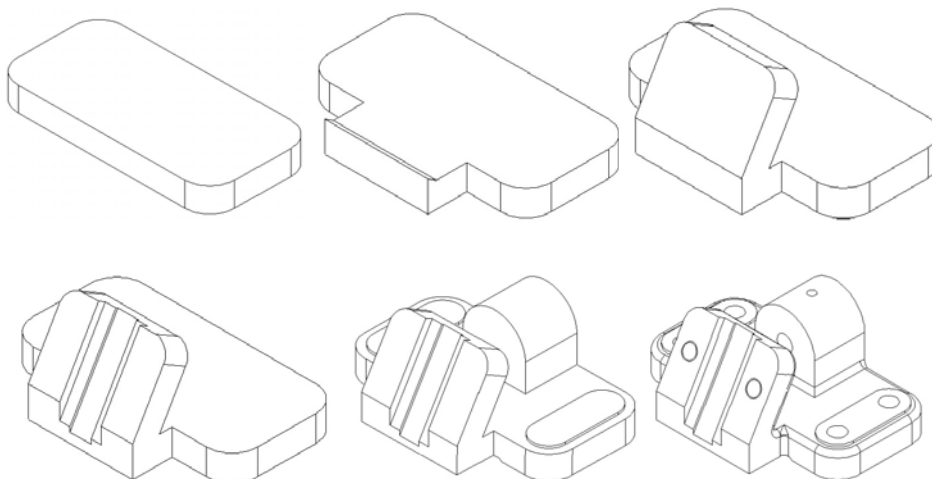










Figure 2 Modeling process of Vertical Bearing Support.

Create Support Base

1. Start Solid Works
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, and click **Unit** in the properties tree text box.
8. Select **Millimeter** and set the decimal places to **2**.
9. Click **OK** to close the Properties dialog box.
10. Select **Top** Plane in **FeatureManager** design Tree and click **Normal To**  to bring Top plane parallel to the screen.
11. Select **Sketch**  on the Sketch toolbar to open a new sketch on the **Top** Plane.
12. Select **Rectangle**  on the Sketch Tools toolbar, and draw a rectangle.
13. Select **Fillet**  on the Sketch Tools toolbar to fillet the four corners of the rectangle.
14. Select **Dimension**  on the Sketch Relation toolbar, and dimension the sketch as shown in Figure 3. To change the fillet radius to **32 mm**, double click on fillet dimension, **Modify** dimension window will open, type **32** and press Enter.
15. Click **Zoon to Fit**  to see the entire sketch.

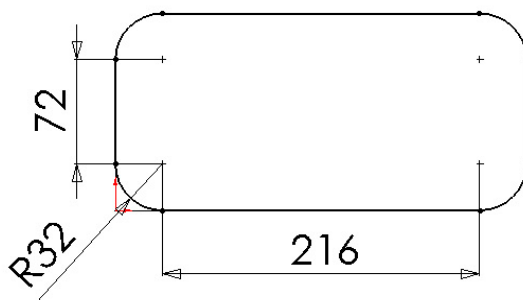




Figure 3 Base sketch and dimensions

16. Select **Extrude Boss/Base**  on the Features toolbar. The Base-Extrude PropertyManager appears.
17. Select **Blind** in the End Condition drop down list box.
18. Type **32mm** in the distance 1 box as shown in Figure 4.
19. Click **OK**  to finish the base extrusion.

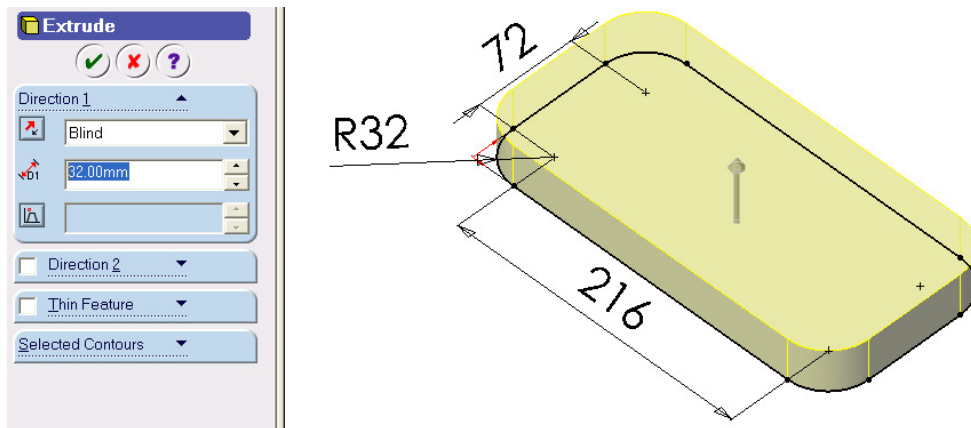





Figure 4 Base extrude

Create an Offset Plane and a Support Boss

1. Click **Plane**  on the Reference Geometry toolbar, or click **Insert, Reference Geometry, Plane** on the menu bar.
2. Select **Offset Distance**  on the PropertyManager.
3. Select the side face of the base as shown in Figure 5. Its name appears in the **Selection** text box.
4. Enter **140mm** in the Distance text box.
5. Select the **Reverse Direction** check box.
6. Click **OK** . The **Planel** is created and displayed in the FeatureManager Design Tree.

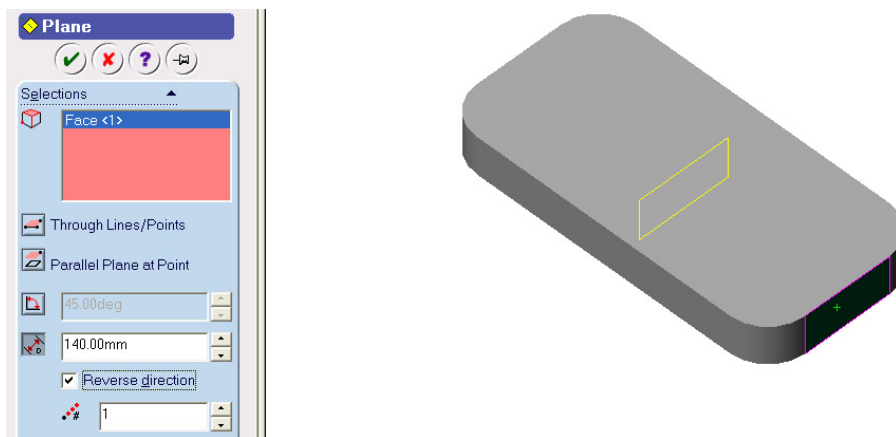






Figure 5 Offset plane

7. Select the **Planel** in the FeatureManager Design Tree or graphics window.
8. Select **Sketch**  on the Sketch toolbar to open a new sketch.
9. Click **Normal To**  on the Standard Views toolbar to make the sketch plane parallel to the screen.
10. Select **Line**  on the Sketch Tools toolbar to draw a polygon at the left edge of the part. The right vertical line of the sketch should align with the left edge of the part. Its two endpoints should be coincident with the endpoints of left edge of the part.

11. Select **Dimension**  on the Sketch Relation toolbar, and dimension the sketch as shown in Figure 6.

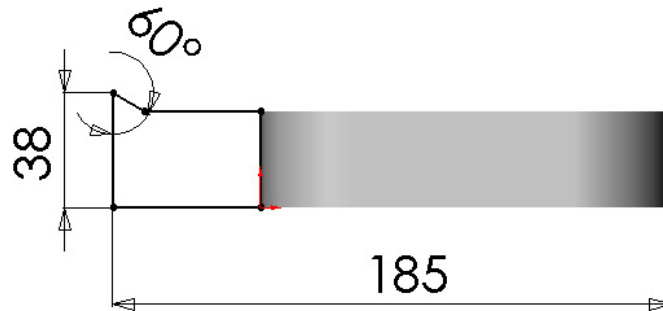





Figure 6 Boss sketch

12. Select **Extrude Boss/Base**  on the Features toolbar.
13. Click **Isometric**  to have 3-D visualization.
14. Select **Mid Plane** in the end condition drop down list box.
15. Type **130 mm** in the distance 1 text box as shown in Figure 7.
16. Click **OK**  to finish the boss extrusion.

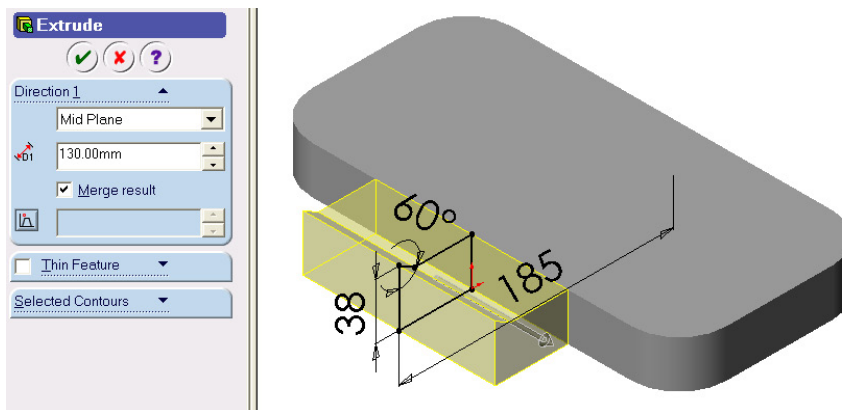





Figure 7 Boss extrusion parameters

Create an Angle Plane and a Slot Body

1. Click **Plane**  on the Reference Geometry toolbar, or click **Insert, Reference Geometry, Plane** on the menu bar.
2. Click **At Angle**  in the dialog box, and then select the front face of the boss in the graphics window. Its name appears in the Selections text box.
3. Select the edge of the boss tip as shown in Figure 8.
4. Type **30** in the Angle text box.
5. Click **OK** . The **Plane2** is created and displayed in the FeatureManager Design Tree.

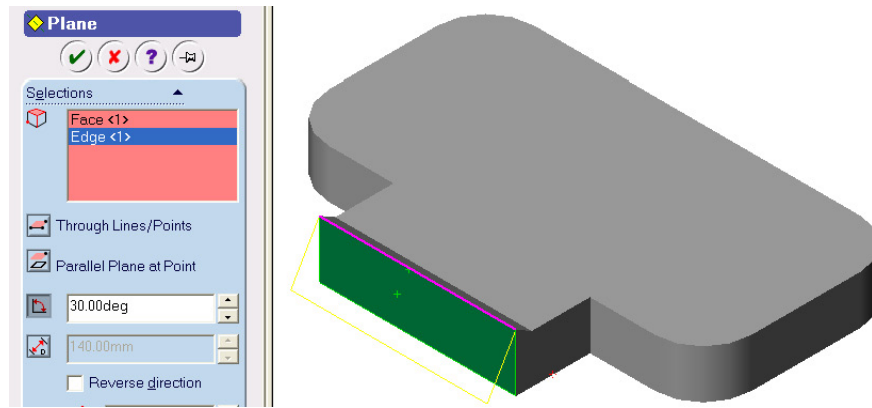






Figure 8 Plane At Angle for slot body

6. Select the **Plane2** in the FeatureManager Design Tree or in the graphics window.
7. Select **Sketch**  on the Sketch toolbar to open a new sketch.
8. Click **Normal To**  on the Standard Views toolbar to make the sketch plane parallel to the screen.
9. Select **Rectangle**  on the Sketch Tools toolbar and draw a rectangle whose top edge aligns with the top edge of the part.
10. Select **Dimension**  on the Sketch Relation toolbar. Dimension the rectangle as shown in Figure 9. If the sketch is not fully constrained, apply collinear constraint to the vertical lines and edges.

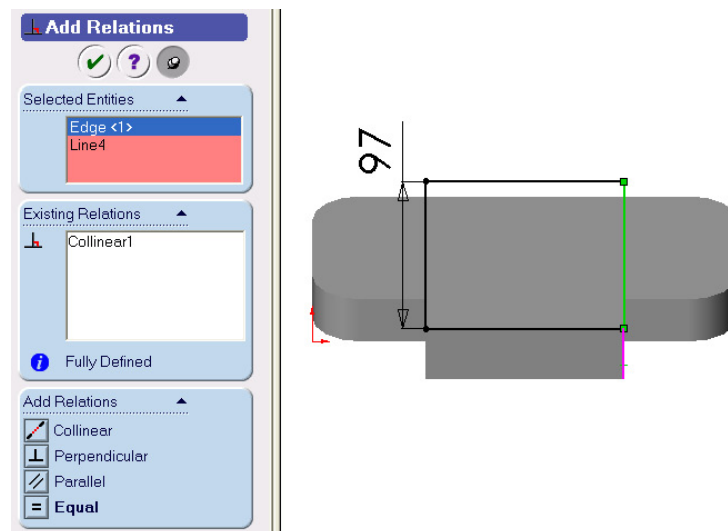




Figure 9 Slot body sketch

11. Select **Extrude Boss/Base**  on the Features toolbar.
12. Select **Blind** in the end condition drop down list box.
13. Type **32 mm** in the distance 1 text box as shown in Figure 10.
14. Click **Reverse Direction** to flip the extrusion direction
15. Click **OK**  to finish the slot body extrusion.

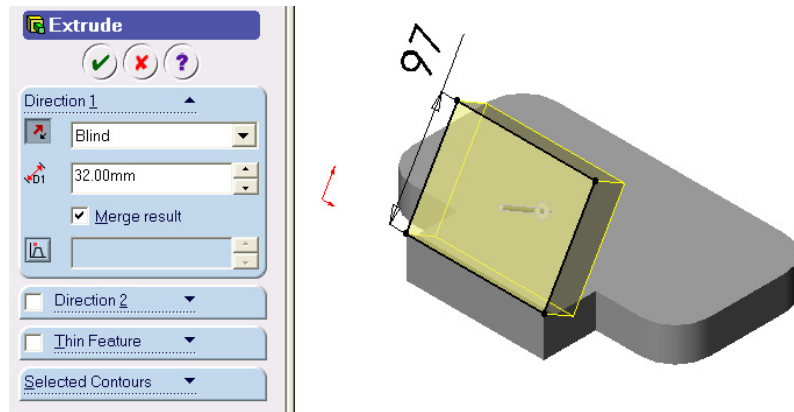




Figure 10 Slot Body extrusion parameters

16. Select **Fillet**  on the Features toolbar.
17. Select two top side edges of the slot body as shown in Figure 11.
18. Type **20 mm** in the Radius text box.
19. Click **OK**  to finish the fillet.

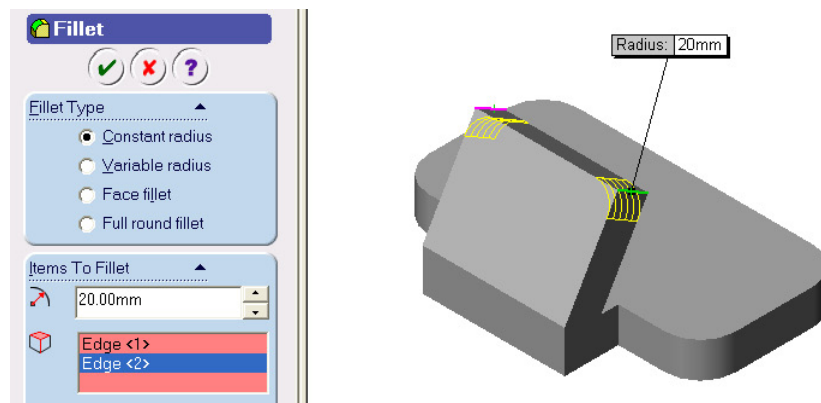


Figure 11 Slot body fillets

Cut Slots

1. Select the top face of the slot body as shown in Figure 12.

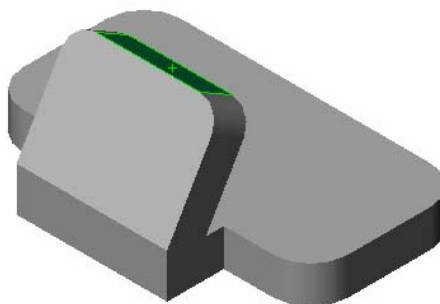








Figure 12 Sketch surface for slot cut

2. Select **Sketch**  on the Sketch toolbar to open a new sketch.
3. Click **Normal To**  on the Standard Views toolbar to make the sketch plane parallel to the screen.

4. Select **Centerline**  on the Sketch Tools toolbar to draw a horizontal centerline as the symmetric axis of the sketch.
5. Select **Line**  on the Sketch Tools toolbar to draw the sketch of the slot that is symmetric to the sketched centerline.
6. Select **Add Relation**  on the Sketch Relation toolbar to add **symmetric** constraints to the sketch. (**Note:** Apply the symmetric constraint first to the centerline with the part edges to make the centerline symmetric to the part).
7. Select **Dimension**  on the Sketch Relation toolbar, and dimension the sketch as shown in Figure 13. The sketch will be fully constrained. If not, apply more constraints.

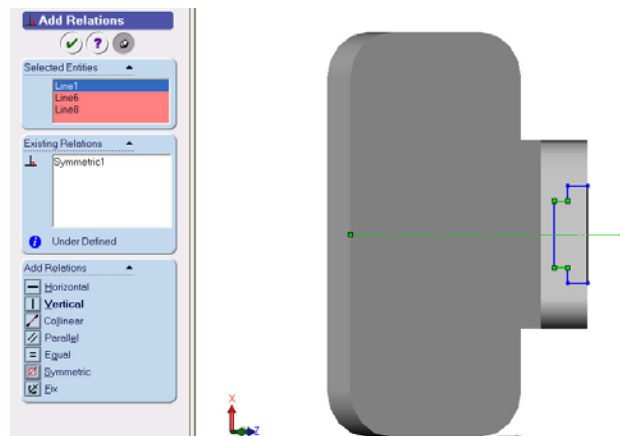




Figure 13 Slot sketch

8. Select **Extruded Cut**  on the Features toolbar.
9. Select **Through All** in the end condition drop down list box as shown in the Figure 14.
10. Click **OK**  to finish the slot cut.

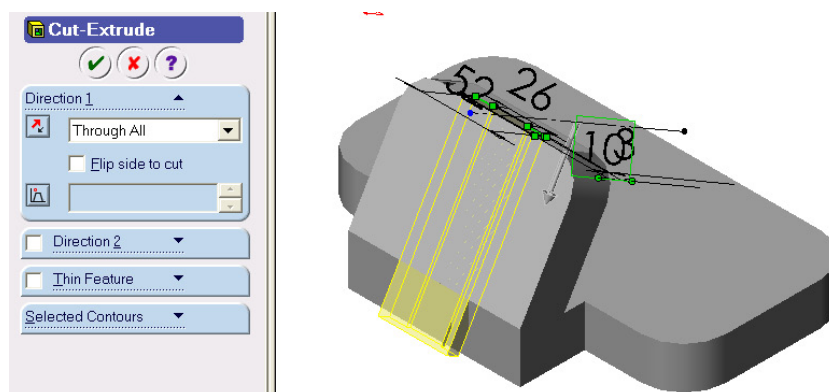



Figure 14 Slot cut parameters

11. Select back surface of the base feature as the sketch plane of the bottom as shown in Figure 15. You may rotate the part using **Rotate View**  to make back surface directly visible.

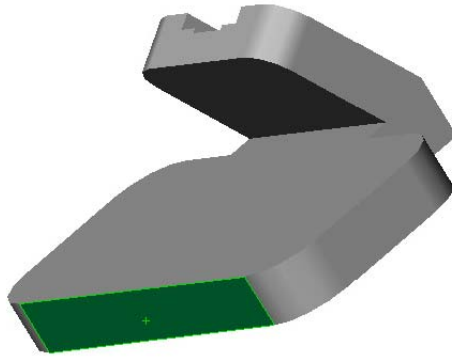








Figure 15 Sketch plane for bottom slot

12. Select **Sketch**  on the Sketch toolbar to open a new sketch.
13. Click **Normal To**  on the Standard Views toolbar to make the sketch plane parallel to the screen.
14. Select **Centerline**  on the Sketch Tools toolbar to draw a horizontal centerline as the symmetric axis of the sketch.
15. Select **Line**  on the Sketch Tools toolbar to draw a sketch of the slot that is symmetric to the sketched centerline.
16. Select **Add Relation**  on the Sketch Relation toolbar to add symmetric constraints to the sketch. (**Note:** Apply the symmetric constraint first to the centerline of the part edge to make the centerline symmetric to the part).
17. Select **Dimension**  on the Sketch Relation toolbar, and dimension the sketch as shown in Figure 16. The sketch will be fully constrained. If not, apply more constraints.

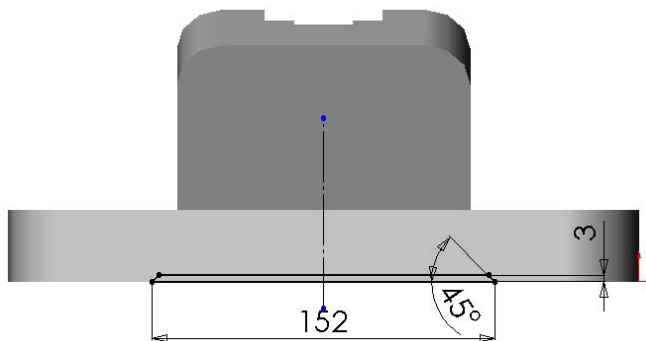




Figure 16 Bottom slot sketch

18. Select **Extruded Cut**  on the Features toolbar.
19. Select **Through All** in the end condition drop down list box as shown in the Figure 17.
20. Click **OK**  to finish the bottom slot cut.

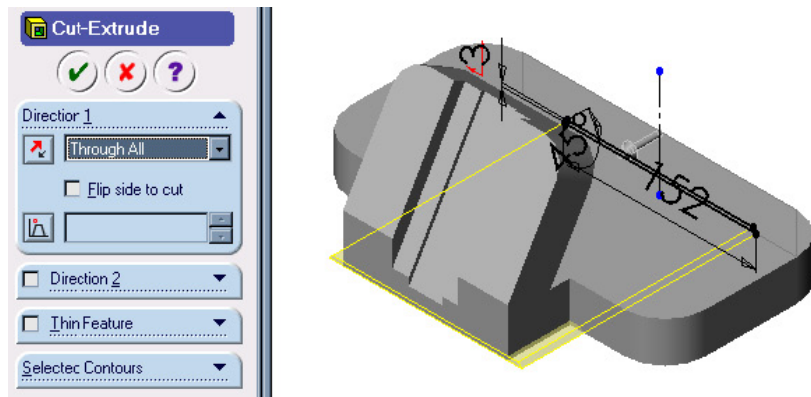


Figure 17 Bottom slot cut parameters

Create Bosses on the Base

1. Select the top face of the base as the sketch plane of the hole bosses as shown in Figure 18.

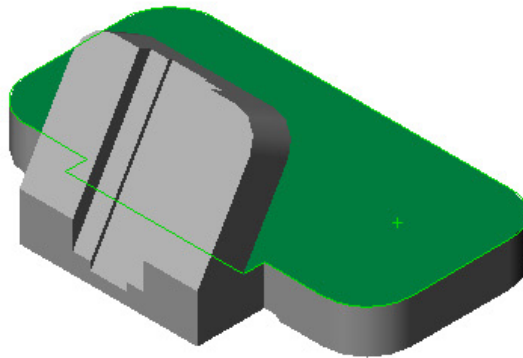















Figure 18 Sketch plane for hole boss

2. Select **Sketch**  on the Sketch toolbar to open a new sketch.
3. Click **Normal To**  on the Standard Views toolbar to make the sketch plane parallel to the screen.
4. Select **Centerline**  on the Sketch Tools toolbar to draw a horizontal centerline as the symmetric axis of the sketch.
5. Select **Centerpoint Arc**  on the Sketch Tools toolbar, and draw two arcs (two half-circles) to form the top and bottom edges of the sketch.
6. Select **Add Relation**  on the Sketch Relation toolbar.
7. Click **Keep Visible**  on the PropertyManager to activate the command.
8. Select two arcs, and apply **equal**  constraint.
9. Select the top arc and the top edge arc of the part, and apply **Concentric**  constraint.

10. Select the bottom arc and the bottom edge arc of the part, and apply **Concentric**  constraint.
11. Select the centerline and the two outside edges of the part, and apply **symmetric**  constraint.
12. Select **Line**  on the Sketch Tools toolbar, and draw two lines to close the sketch.
13. Select the sketch and the centerline by drawing a window around them (you may add entities to selection by holding the 'Control' key and clicking on the required entities one by one), and click **Mirror**  on the Sketch Tools toolbar to mirror the sketch to the other side.
14. Select **Dimension**  on the Sketch Relation toolbar. Dimension the sketch as shown in Figure 19.

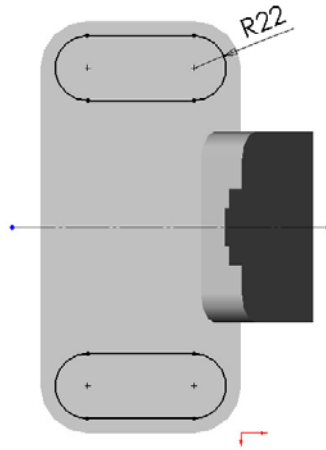





Figure 19 Hole boss sketch

15. Select **Extruded Boss/Base**  on the Features toolbar.
16. Click **Isometric**  to have 3-D visualization.
17. Select **Blind** in the end condition drop down list box.
18. Type **3 mm** in the **distance 1** box as shown in Figure 20.
19. Click **OK** .

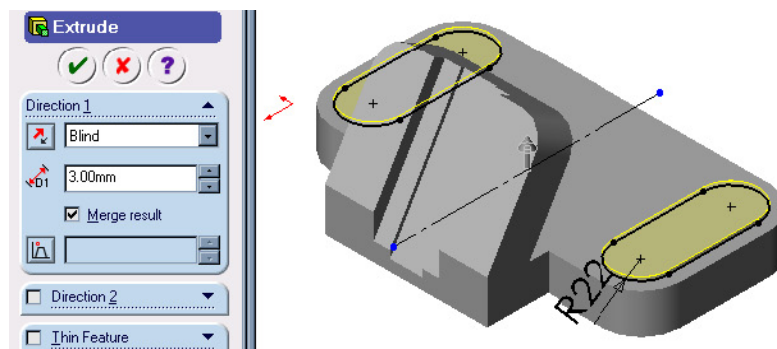


Figure 20 Hole Boss Extrusion Parameters

20. Select the back surface of the base as shown in Figure 21.

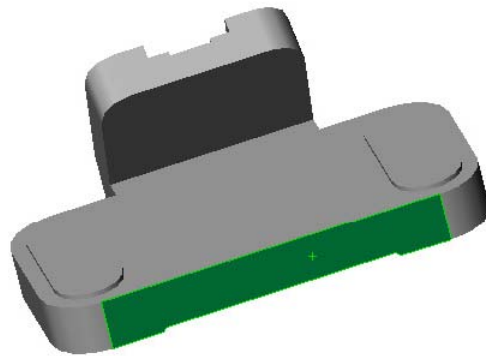










Figure 21 Sketch plane for the counterbored hole boss

21. Select **Sketch**  on the Sketch toolbar to open a new sketch.
22. Click **Normal To**  on the Standard Views toolbar to make the sketch plane parallel to the screen.
23. Select **Centerline**  on the Sketch Tools toolbar to draw a horizontal centerline as the symmetric axis of the sketch.
24. Select **Add Relation**  on the Sketch Relation toolbar. Select the centerline and the two side edges of the part, and apply **Symmetric**  constraint.
25. Select **Centerpoint Arc**  on the Sketch Tools toolbar, and draw a half circle whose center is coincident with the centerline.
26. Select **Line**  on the Sketch Tools toolbar to draw three lines to close the sketch.
27. Select **Dimension**  on the Sketch Relation toolbar, and dimension the sketch as shown in Figure 22.

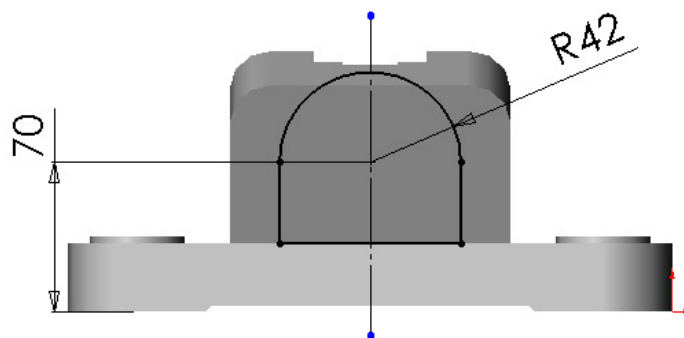






Figure 22 Sketch for counterbored hole boss

28. Select **Extruded Boss/Base**  on the Features toolbar.
29. Click **Isometric**  to have 3-D visualization.
30. Select **Blind** in the end condition drop down list box.
31. Type **70mm** in the **direction 1** box.

32. Click **Reverse Direction**  to flip the extrusion direction as shown in Figure 23.
33. Click **OK** .

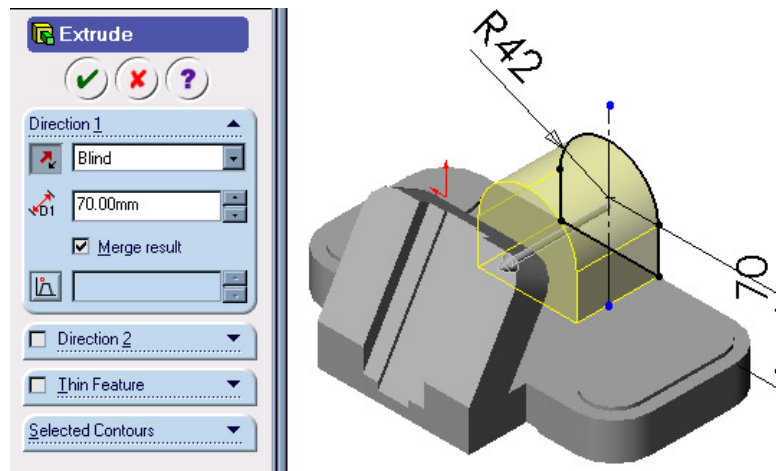



Figure 23 Counterbored hole boss

Create Holes and Fillets

1. Select the end surface of the counterbored hole boss.
2. Select the **Hole Wizard**  on the Features toolbar, and select the **Counterbore** tab.
Note: Various types of holes can be created using corresponding tab.
3. Select **Up To Next** in the End Condition & Depth cell.
4. Type **22mm** in the Hole Fit & Diameter cell. **45 mm** and **50 mm** in the Counterbore Diameter & Depth cells as shown in Figure 24.
5. Click Next.

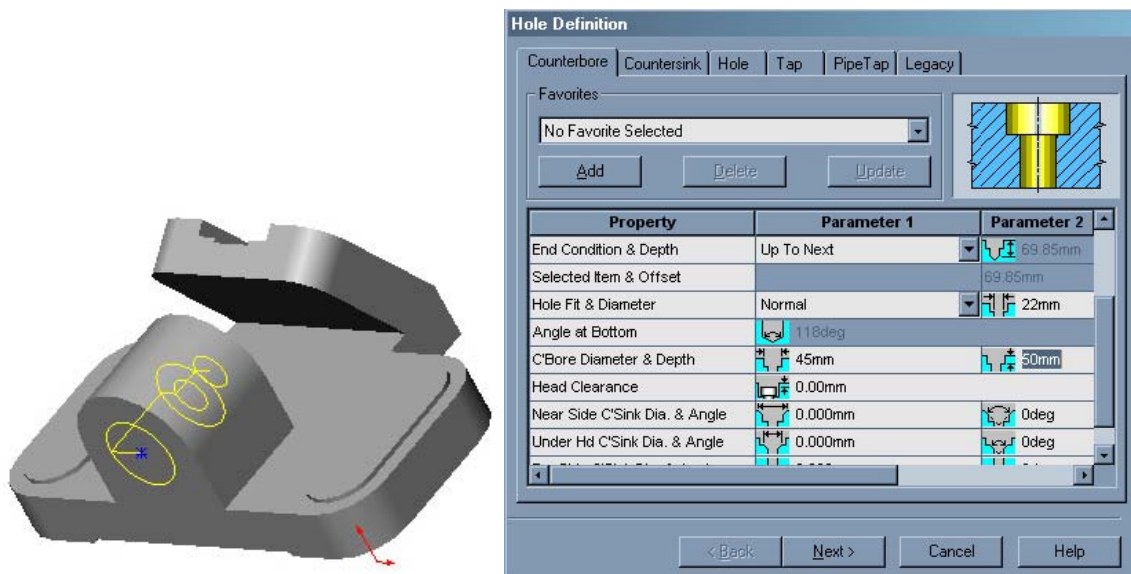




Figure 24 Counterbored Hole Parameters

6. Select **Add Relation**  on the Sketch Relation toolbar.
7. Select the hole center point and the arc edge of the boss, and click **Concentric** .
8. Click **Finish** to create the counterbored hole.
9. Select the hole boss surface as shown in Figure 25.

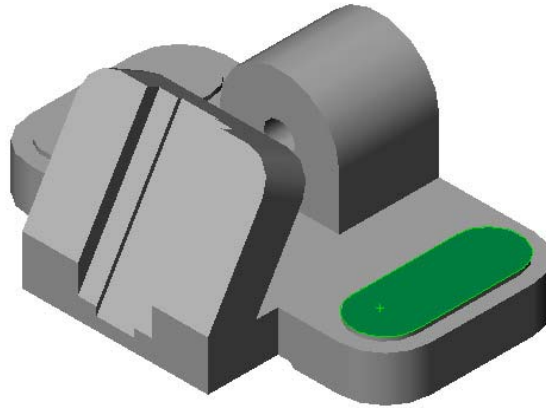








Figure 25 Sketch Plane for Base Holes

10. Select **Sketch**  on the Sketch toolbar to open a new sketch.
11. Click **Normal To**  on the Standard Views toolbar to make the sketch plane parallel to the screen.
12. Select **Circle**  on the Sketch Tools toolbar, and draw **one** circle for the hole.
13. Select **Add Relation**  on the Sketch Relation toolbar, and add concentric constraint to the circle with the arc edge of the base.

Remark: You may similarly draw **three** other circles, Add **Equal**  constraint to the circles, but we will use another useful command to produce these circles.)

14. Select **Dimension**  on the Sketch Relation toolbar. Dimension the circle diameter to **21mm** as shown in Figure 26.

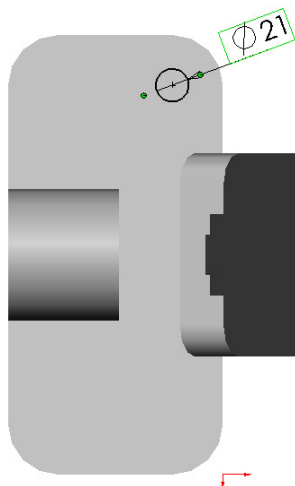



Figure 26 Base Hole Sketch

15. Select **Extruded Cut**  on Features toolbar. Select **Through All** in the end condition drop down list box as shown in Figure 27.

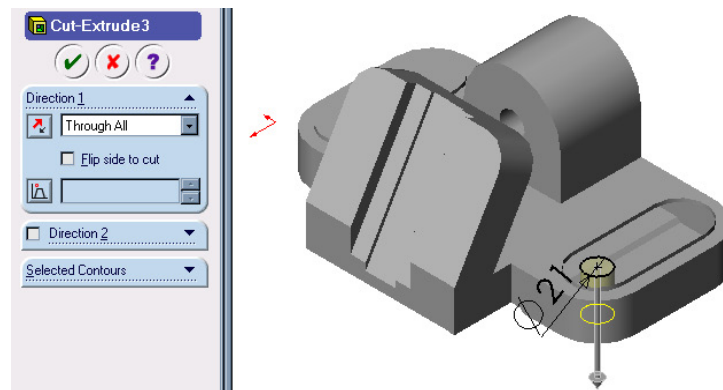


Figure 27 Base holes

16. Click **OK** .

Copying Holes using Linear Pattern

17. Make sure that Hole you have just created is selected in FeatureManager Design Tree. If not, select it by Clicking on it either in Graphic area or from FeatureManager Design Tree.

18. Click **Linear Pattern**  on Feature Toolbar, Linear Pattern Property Manager opens as shown in Figure 28.

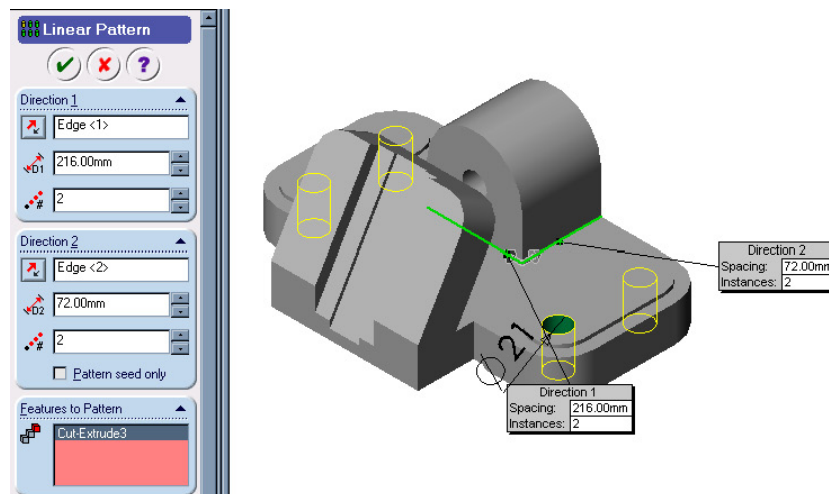









Figure 28 Copying Features using Linear Pattern.

19. Click on any edge parallel to Direction 1 as shown in Figure 28.
20. Type **216 mm** for **Spacing** .
21. Type **2** for **Number of Instances** . A preview of the copied hole will appear as shown in Figure 28. Make sure that it is being copied in correct direction. If not, click on **Reverse Direction** .
22. Click on any edge parallel to Direction 2 as shown in Figure 28.

23. Type **72 mm** for **Spacing** .
24. Type **2** for **Number of Instances** . A preview of the copied hole will appear as shown in Figure 28. Make sure that it is being copied in correct direction. If not, click on **Reverse Direction** .
25. Click **OK** .
26. Select the end face of the counterbored hole boss as shown in Figure 29. You may need to rotate the part.

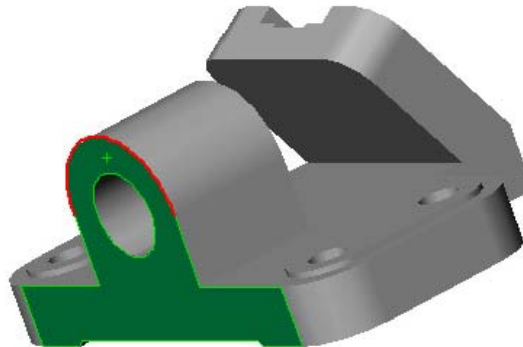










Figure 29 Sketch Plane for Reference Point

27. Select **Sketch** on the Sketch toolbar to open a new sketch.
28. Click **Normal To**  on the Standard Views toolbar to make the sketch plane parallel to the screen.
29. Select **Point**  on the Sketch Tools toolbar, and draw a point on the top edge of the counterbored hole boss.
30. Select **View, Temporary Axes** on the menu bar to show the temporary axis for adding relation.
31. Select **Add Relation**  on the Sketch Relation toolbar.
32. Select the point and the edge. Click **Coincident**  to make the point coincident with the arc edge.
33. Select the hole axis and the point as shown in Figure 30. Select **Vertical** .
34. Click **Rebuild**  on the Standard toolbar to exit sketch.
35. Click **Plane**  on the Reference Geometry toolbar, or click **Insert, Reference Geometry, Plane** on the menu bar.
36. Select **On Surface** .

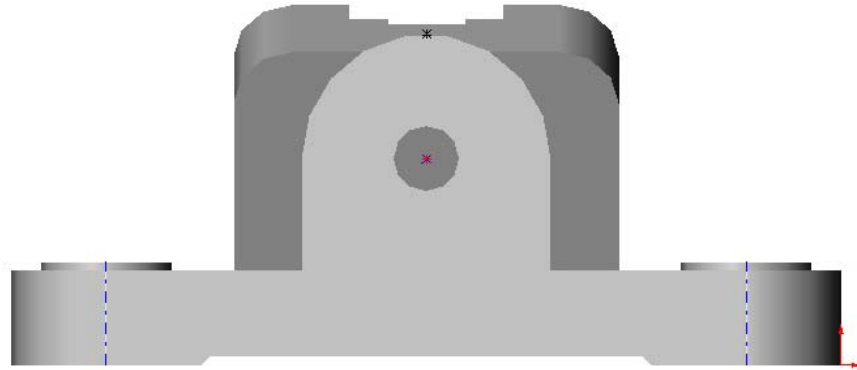


Figure 30 Sketch Point for Creating On Surface Plane

37. Select the cylindrical face of the boss and the sketch point as shown in Figure 31. Their names appear in the Selections text box.

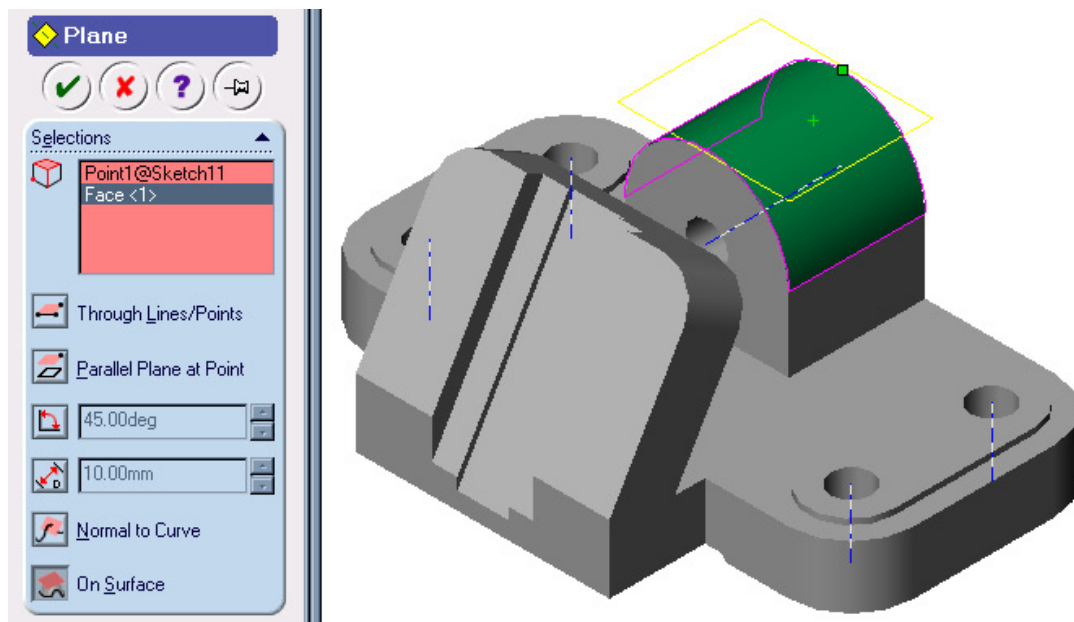




Figure 31 Top hole placement plane

38. Click **OK** . The **Plane3** is created and displayed in the FeatureManager Design Tree.
39. Select **Plane 3** in the Feature Manager Design Tree or in the graphics windows.
40. Select **Hole Wizard**  on the Features toolbar.
41. Select **Tap** tab in the Hole Definition dialog box.
42. Select the parameters as shown in Figure 32. Click **Next**.

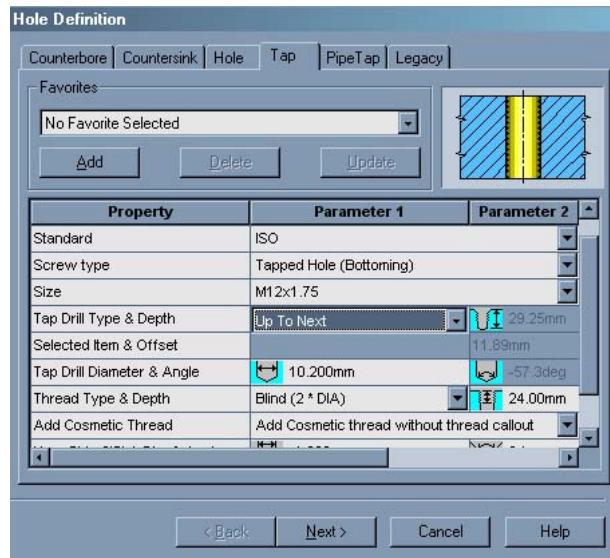



Figure 32 Top hole definition.

43. Click **Normal To**  on the Standard Views toolbar to make the sketch plane parallel to the screen.
44. Add constraints and dimension the hole center as shown in Figure 33. Then click **Finish** in the Hole Placement dialog box. (**Note:** There will be an error message in the dialog box. Click **OK** to ignore it.)

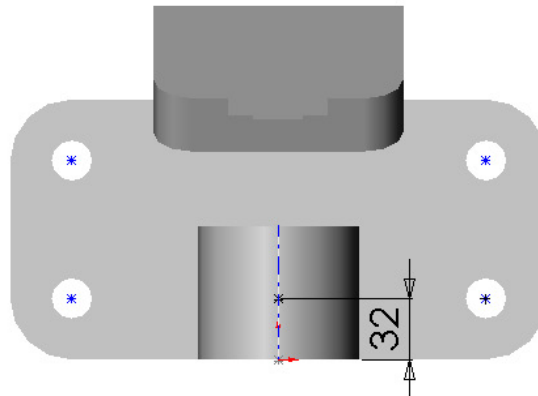


Figure 33 Top hole placement.

45. Select the front face of the slot body as shown in Figure 34.

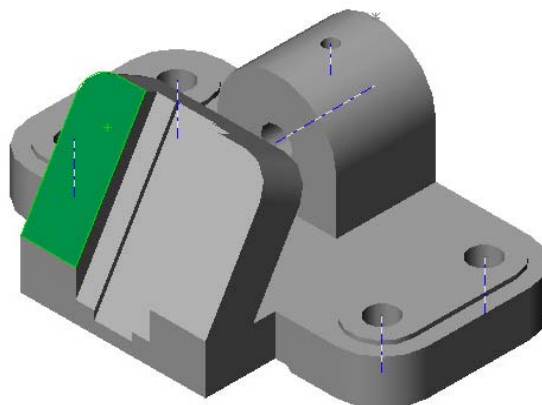



Figure 34 Slot hole sketch plane

46. Select **Hole Wizard**  on the Features toolbar.
47. Set up the Hole Definition Dialog box as shown in Figure 35. Then click **Next**.

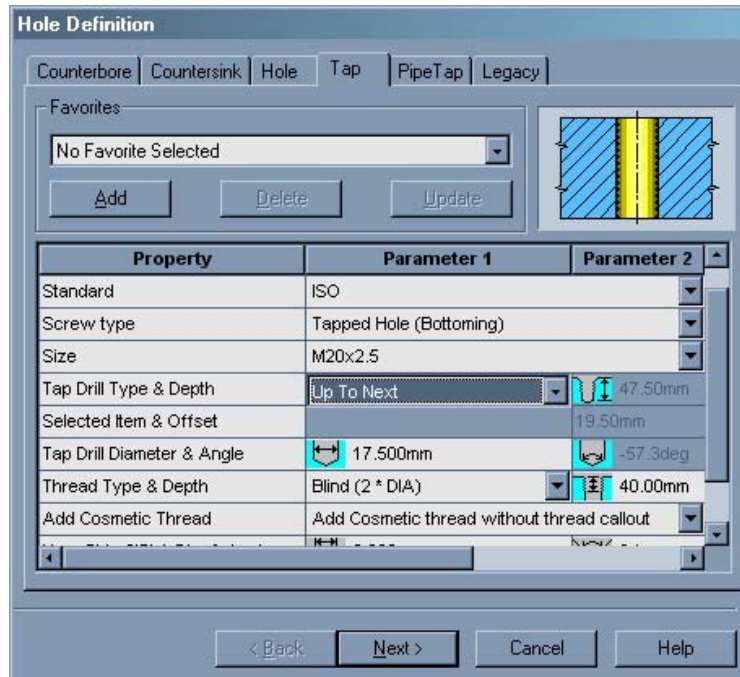



Figure 35 Slot hole definition

48. Sketch a centerline and a point for the other hole. Add Symmetric  relation between two points and centerline. Dimension the hole centers as shown in figure 36.

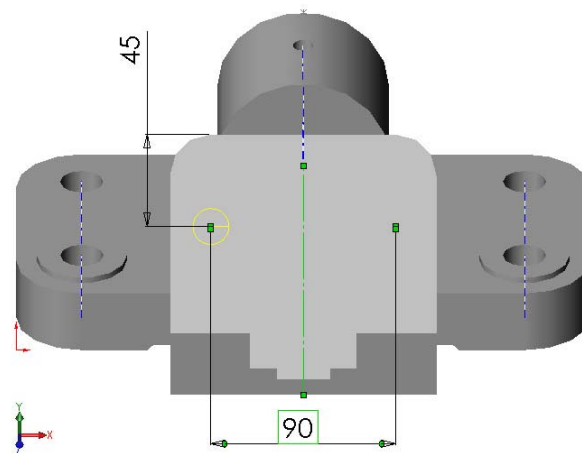




Figure 36 Slot hole placement.

49. Click Finish in the Hole Placement dialog box.
50. Add **fillets** as shown in Figure 37 and Figure 38 by using **Fillet**  on Feature toolbar. Use fillet radius as **6 mm**. Final model for Vertical Bearing Support is shown in Figure 39.
51. **Save**  the file.

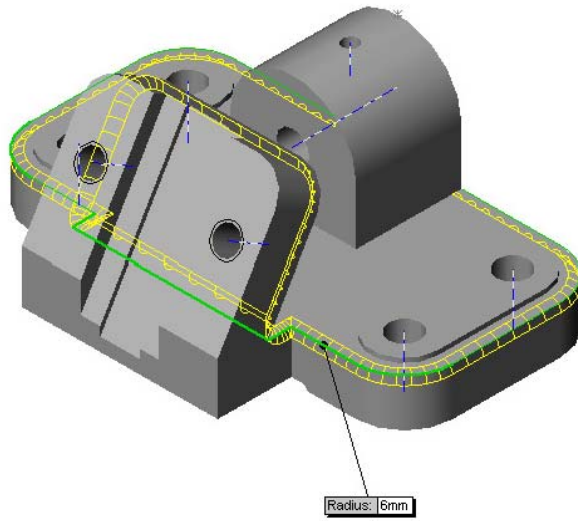


Figure 37 Base edge fillet.

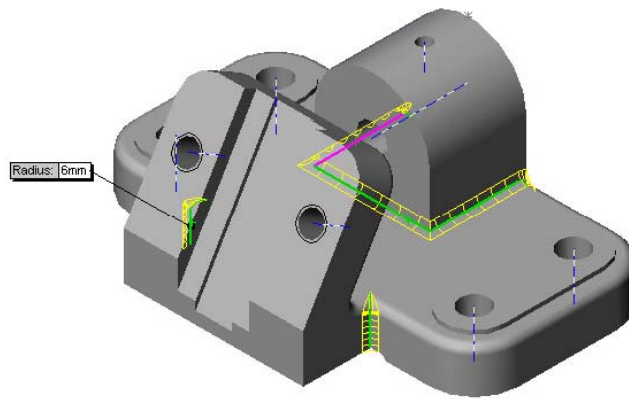


Figure 38 Slot body fillet.

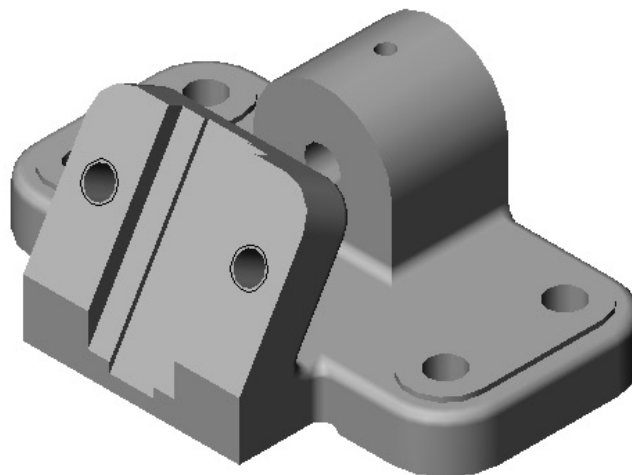


Figure 39 Vertical bearing support model