

Test-Drive
nanoCAD 3D Solid Modeling Module



General information

This tutorial contains illustrative examples of the 3D module's capabilities in the nanoCAD Platform.

This document will help end users to master their basic skills of 3D modeling in a short time. For the demo of 3D construction, we will use the 2D drawing presented at the end of this guide.

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1. System Settings for 3D Modeling

1.1. Open the **3D History** function panel: in the **Manage** tab activate the **3D History** by clicking the icon shown in Fig. 1.

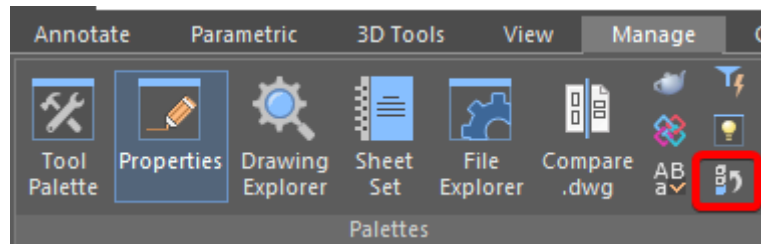


Fig. 1. 3D History function panel

1.2. Place the **Properties** and the **3D History** panels on opposite sides of the screen. (Fig. 2).

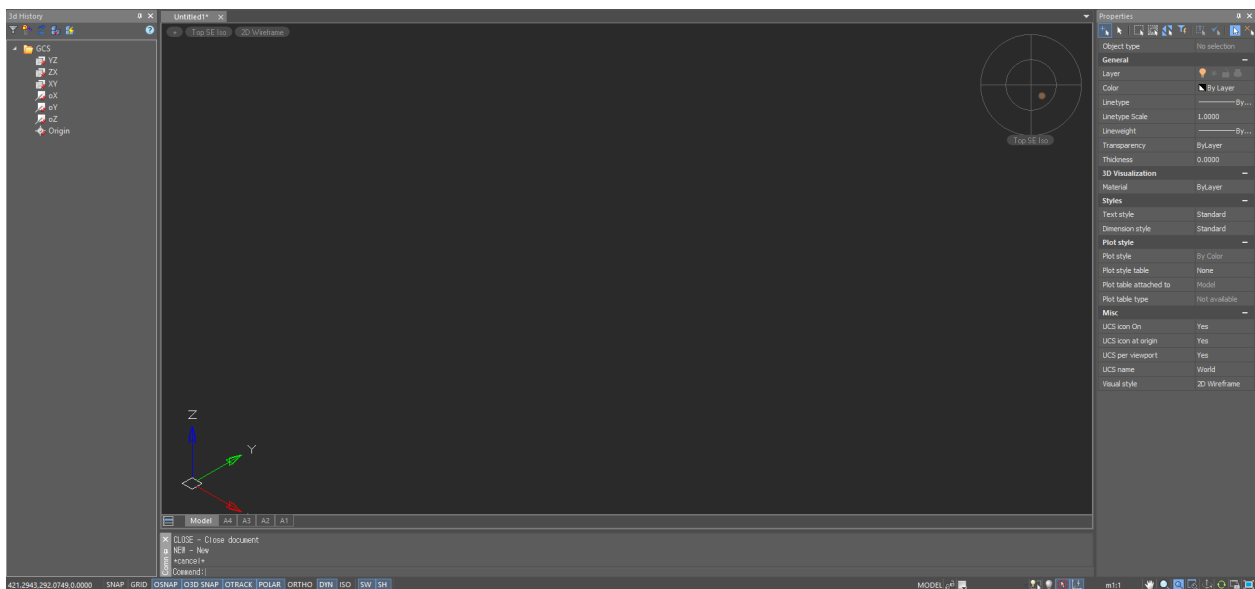


Fig. 2. Function panels arrangement on the screen

1.3. Save the file.

2. 2D Sketch Mode

First, select a plane in which you want to build a model.

2.1. On the **3D History** function panel, expand the **GCS**. Call the context menu of the **ZX** plane with the right-click (RMB) and select **Create 2d-sketch** (Fig. 3).

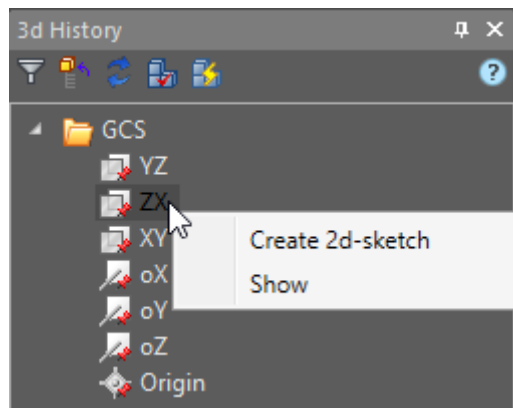


Fig. 3. Creating a 2D sketch

2.2. On the **Draw** tab choose the **Circle** command (Fig. 4) to draw a circle of any diameter with a center in the origin of the coordinates.

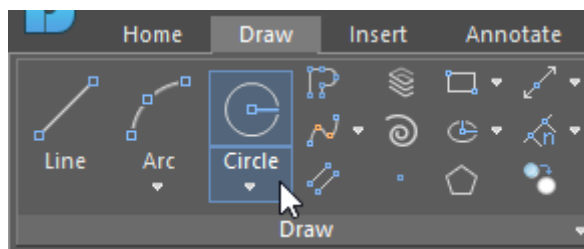


Fig. 4. The **Circle** command

2.3. Press **Shift + RMB** to access the local context menu, and select **Node** to specify the origin.

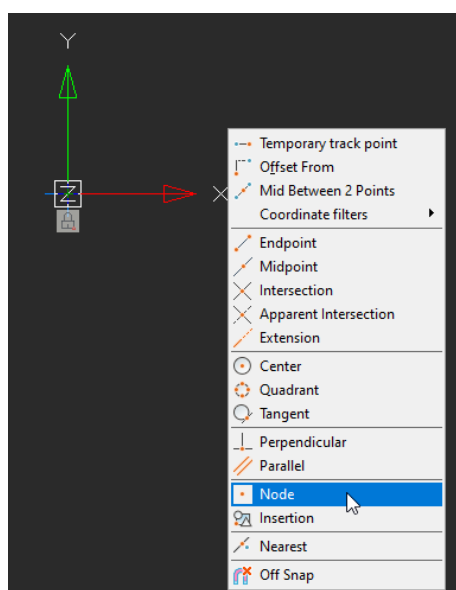


Fig. 5. The Context menu

2.4. On the **Parametric** tab, select **Diameter** and specify the constructed circle. In the window that has appeared, enter the value for the diameter: $\varnothing 140$. Click **OK** to confirm (Fig.6).

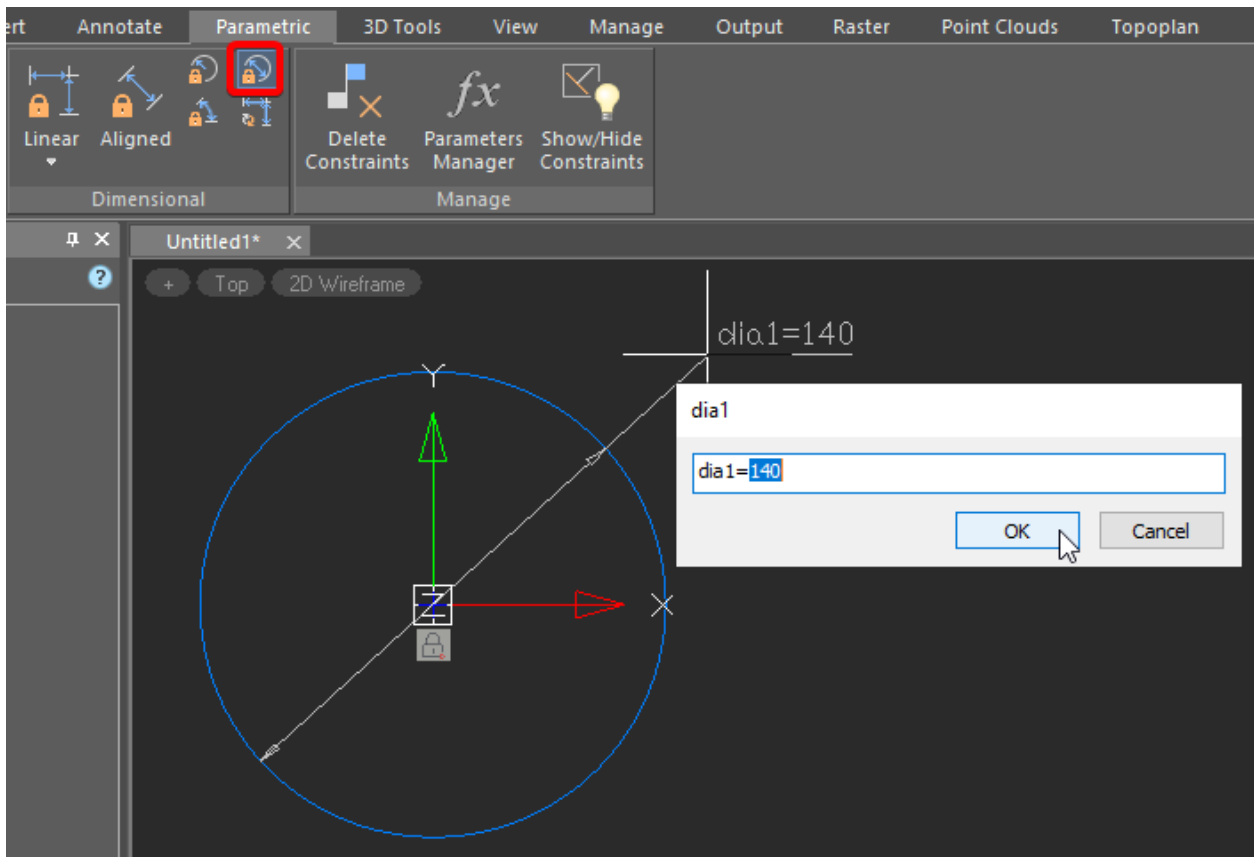


Fig. 6. Specifying the circle size.

2.5. Now, the drawing is regenerated per the entered diameter value.

3. 3D Extrude Operation

3.1. Click the **3D Extrude** command on the **3D Tools** tab (Fig. 7). Move the cursor to the circle area and confirm the sketch selection with the left-click (LMB).

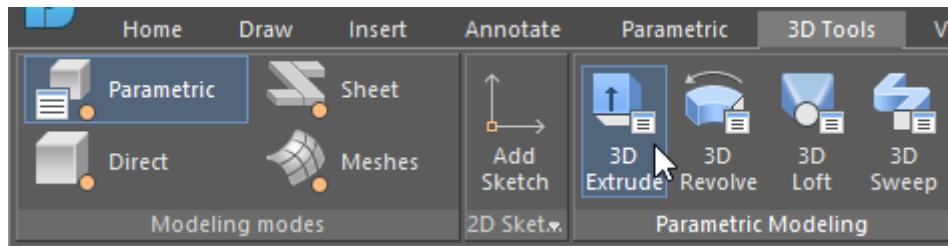


Fig. 7. The 3D Extrude operation

3.2. In the dialog box, enter the distance value 255 and click the **Negative Direction** button. Confirm with **OK** (Fig. 8).

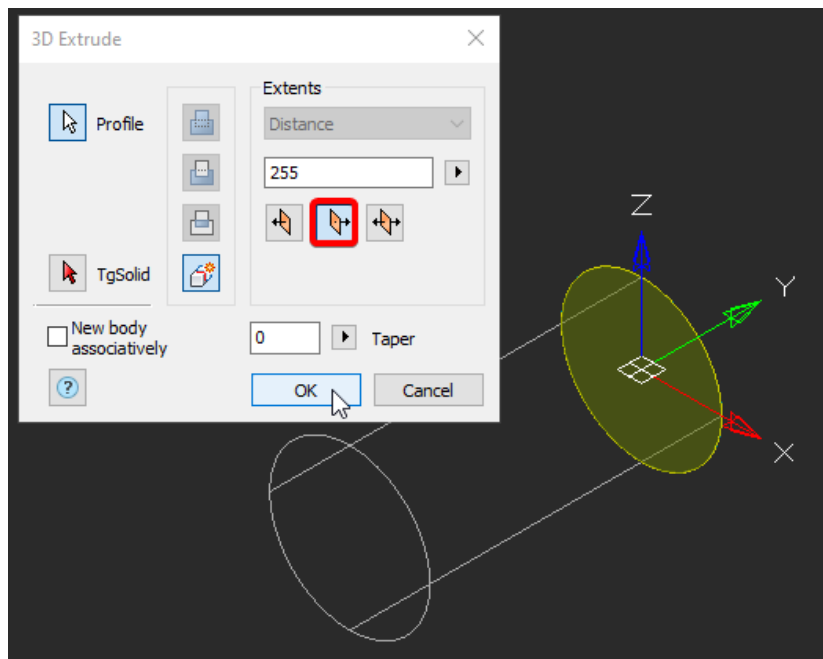


Fig. 8. The dialog box of the 3D Extrude operation

3.3. Next, we will use the cut option in the 3D Extrude operation.

Switch the view orientation to **Top NW Isometric** using the **Locator** for your convenience. Then, switch the visual style to **Flat shaded with edges** (Fig. 9).

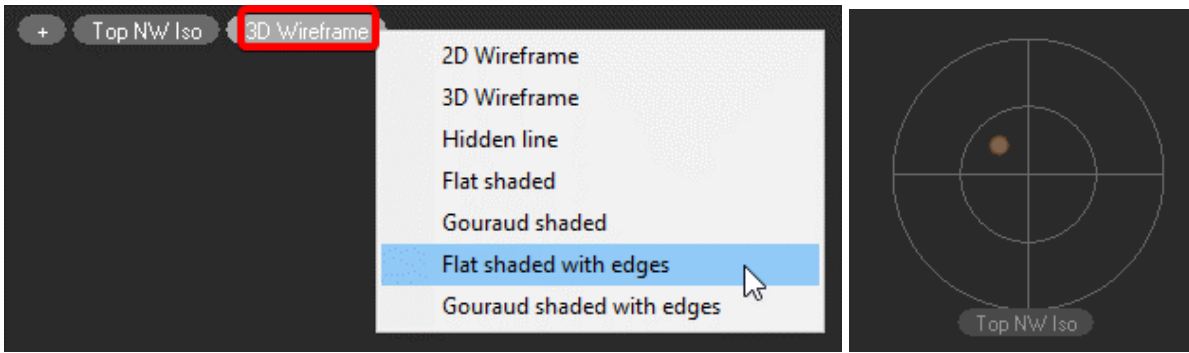


Fig. 9. Switching views and visual styles

3.4. Let's design a sketch on the front end of the object. On the **3D Tools** tab, click **Add Sketch** command (Fig. 10) and press LMB to select the front end of the 3D model. The system has switched to sketch mode. Draw a circle with a $\varnothing 95$ diameter.

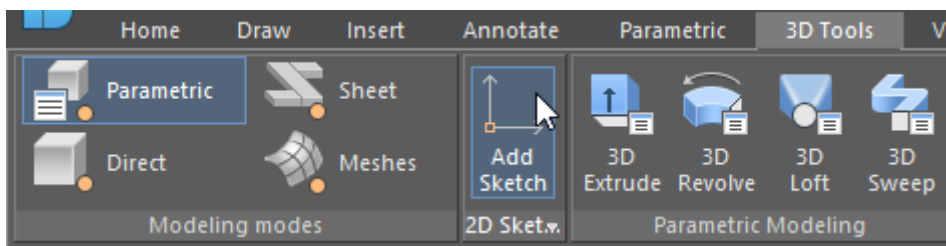


Fig. 10. Add Sketch command on the 3D Tools tab

Once again, activate the **3D Extrude** operation on the **3D Tools** tab. Choose the sketch area between the big and small circles with a click. In the window that has appeared, activate the **Cut** option (Fig. 11). In the **Distance** field enter 255-20 and choose the **Negative control direction**. Click **OK** to confirm the operation.

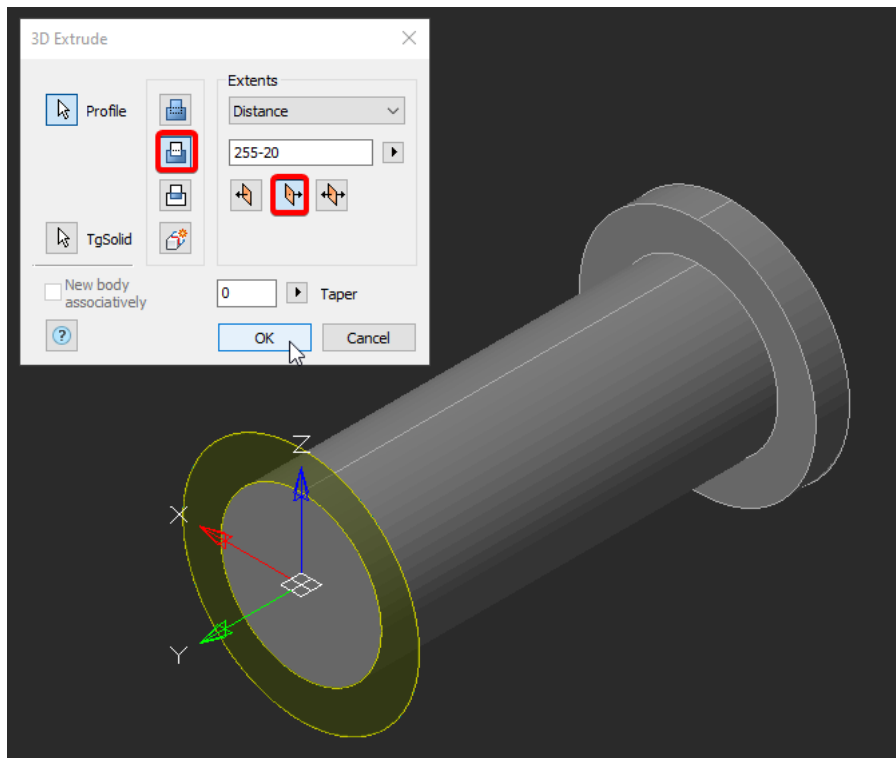


Fig. 11. Option cut of the 3D extrude operation

3.5. Similar to the previous step, construct a surface with $\varnothing 75$ diameter and 165 long.

3.6. Construct a hole with $\varnothing 36$ diameter. Select the **Through** option in the pop-up menu in the parameters of the **3D Extrude** operation.

3.7. Draw a hole with $\varnothing 45$ diameter for the M48x2 thread. Type 45-18.5 in the **Distance** field.

The result is shown in Fig. 12.

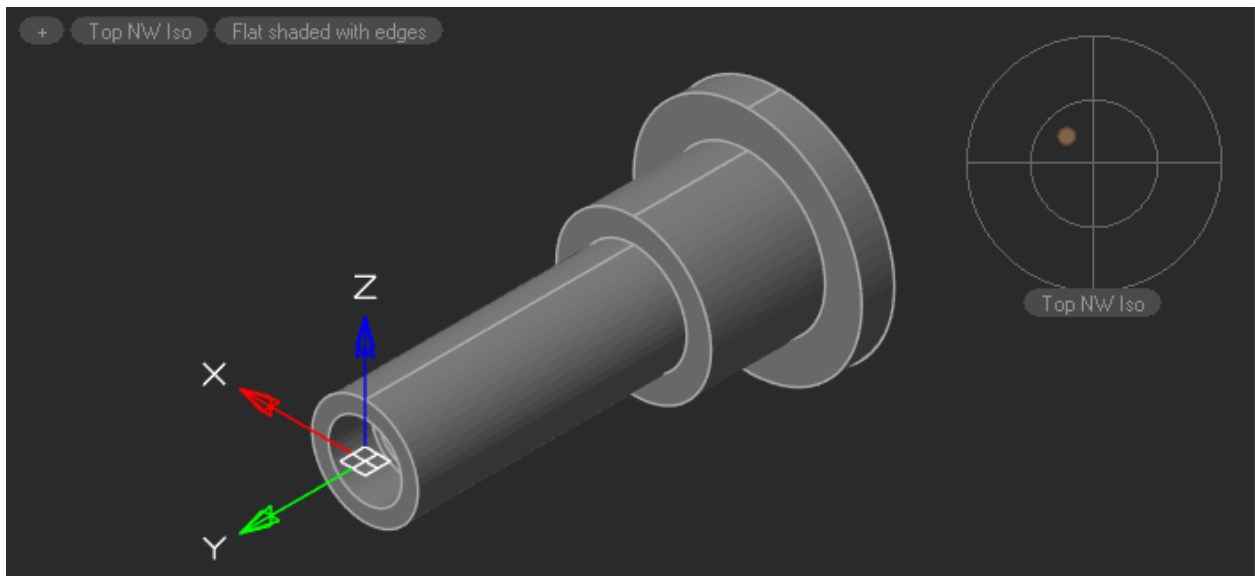


Fig. 12. The result of construction using the 3D Extrude operation

The next element will be designed with the **3D Revolve** operation. But first, let's generate a section.

4. 3D Section

4.1. Click **Section** on the **3D Tools** tab to create a 3D section (Fig. 13). Then go to the **3D History** and open **GCS** to specify the **YZ** plane.

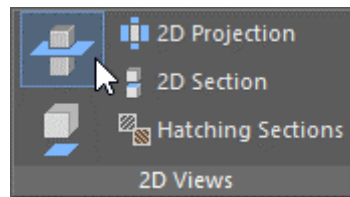


Fig. 13. The Section command

4.2. Press **Esc**

4.3. In **3D History**, or in the model space, select the **Section** and choose **YES** in the **Is Live** drop menu (Fig. 14).

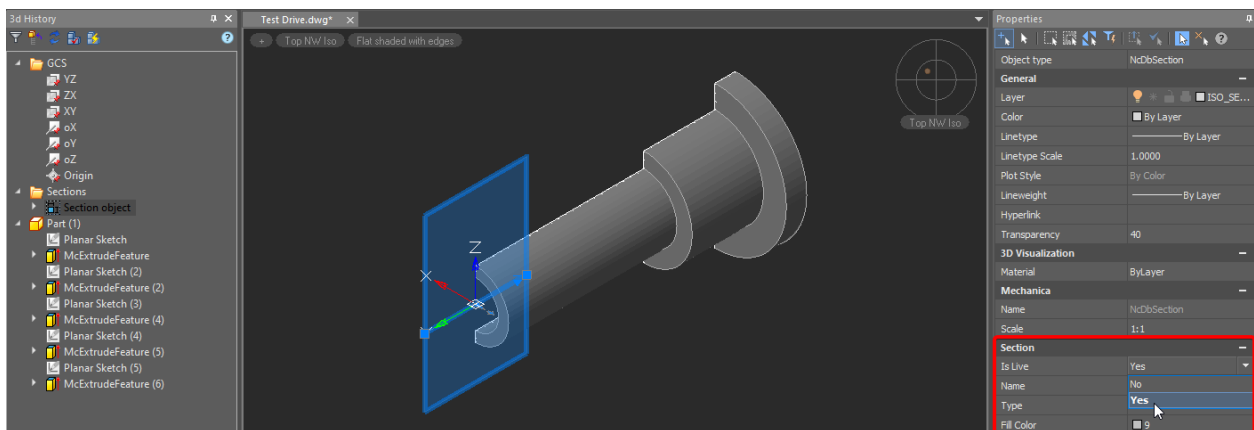


Fig. 14. Enabling the Is Live function

4.4. Set the top SE isometry by using the Locator.

5. The 3D Revolve Operation

5.1. Create a sketch on the **YZ** plane

5.2. Zoom in the previous construction and project the inner end plane of the hole. Use the **Add Projection** command in the **2D Sketch** group. If needed, you can switch to an isometric view with **Shift** + mouse wheel (Fig. 15).

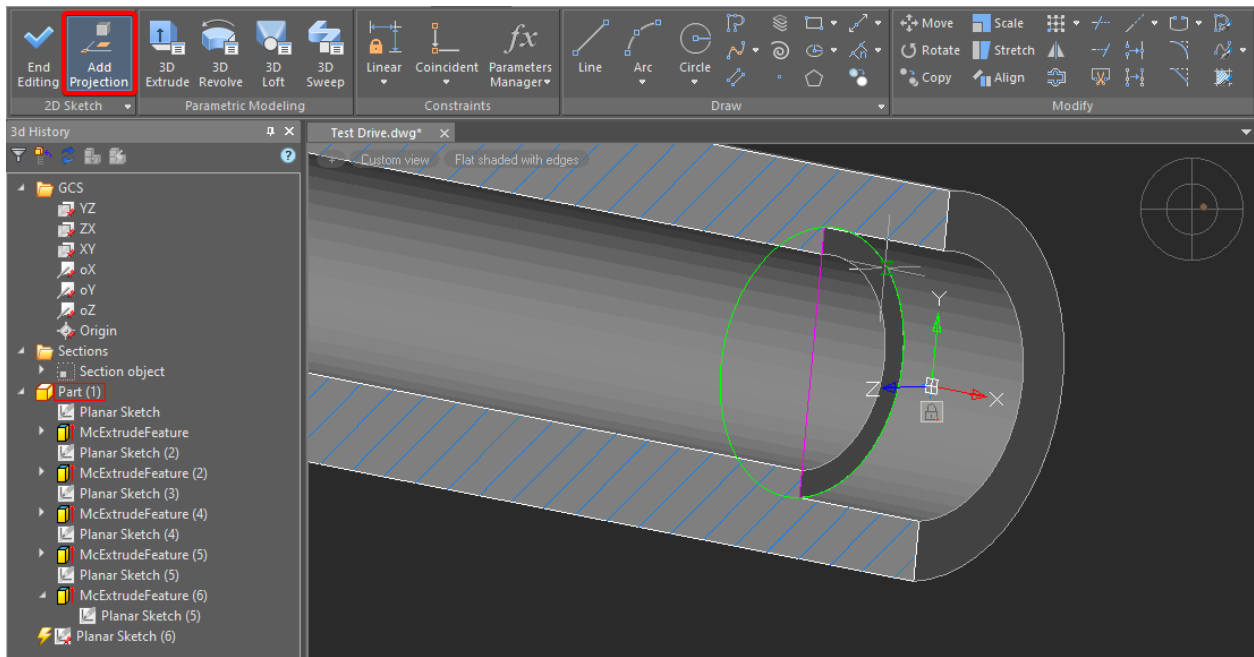


Fig. 15 Projecting an element onto the 2D sketch

The projected face (edge) will appear as a line. We will use this line for future 3D model constructions.

5.3. Design the contour of the future hole with a snap without specifying the segment dimensions. The construction should start from the projected line. The end point should be defined with the Midpoint snap. Press **Shift** for the orthogonal drawing.

5.4. When done, apply the Auto Constraint to ease line selection. You can hide the whole part: just right-click the part's name in the 3D history and choose Hide. If you want to make it visible again, right-click the part's name and choose the Show option. After the Constraint is done, use the Linear command in the Dimensional group (Fig. 16).

The fillets are also shown in the drawing. We can create them either in the sketch with the **Fillet** command on the **Draw** tab, or with the **3D Fillet** command by clicking the **Fillet Edge** button (see «3D Fillets and Chamfers»).

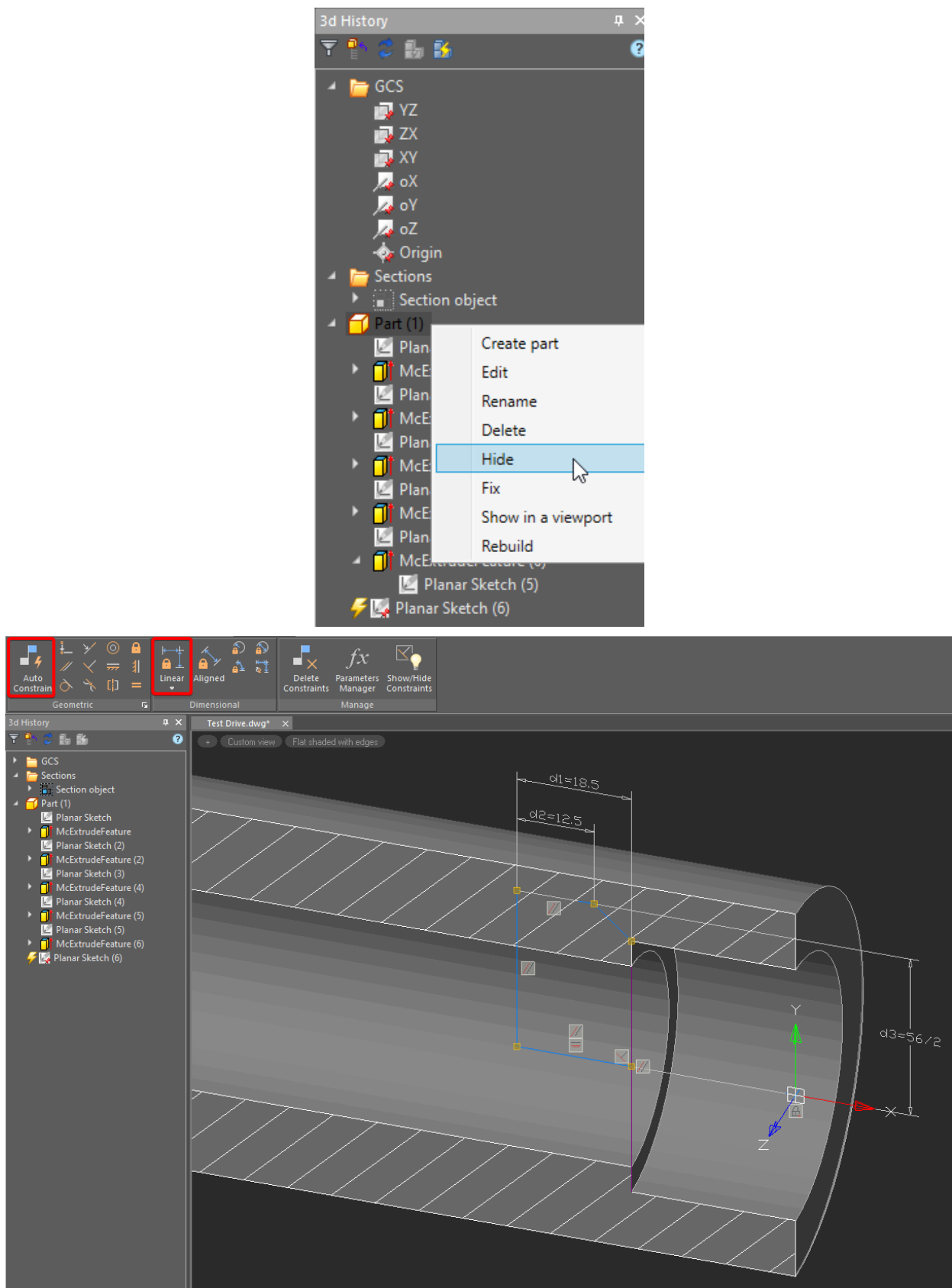


Fig. 16. The sketch with the parametric constraints

5.5. Switch to the **3D Tools** tab and click **Revolve**. Select the sketch area. In the dialog box click **TgSolid** and choose **Part (1)** in **3D History**. In the **3D Revolve** dialog box choose the **Cut** option and select **Axis**. Specify the rotation axis and click **OK** (Fig. 17).

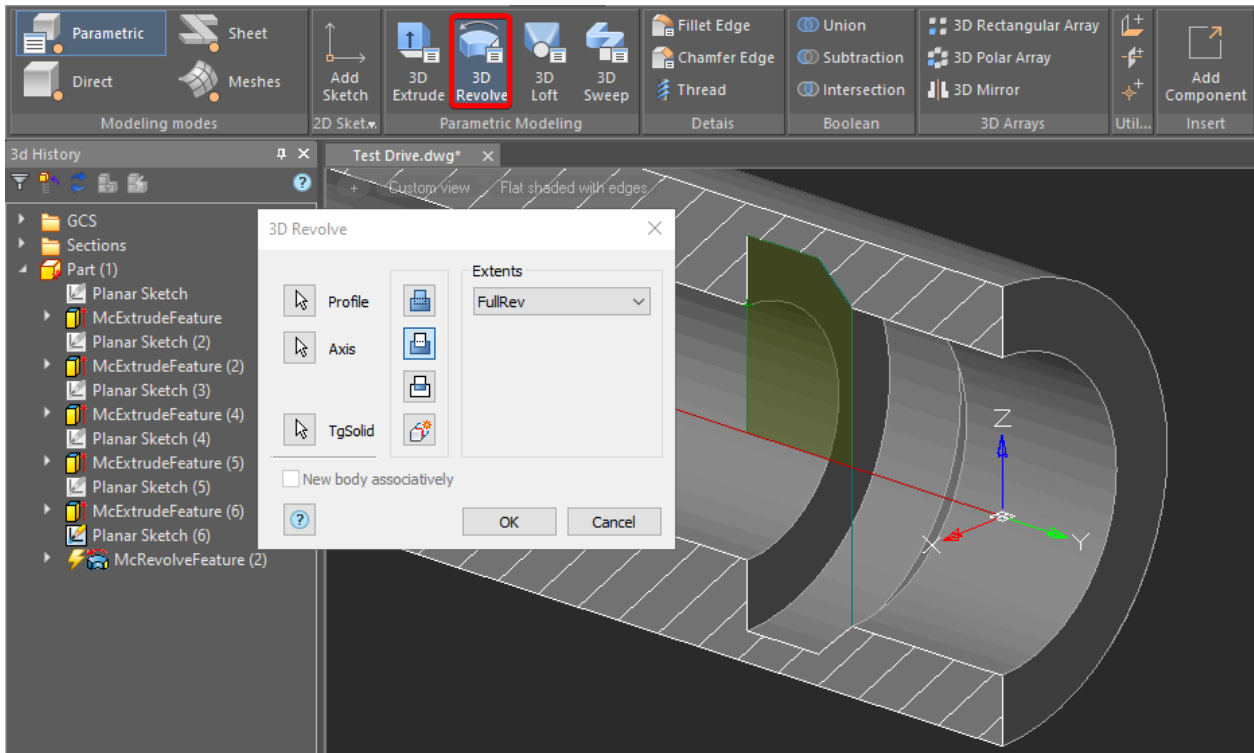


Fig. 17. The 3D Revolve operation

5.6. Construct the $\varnothing 61$, $\varnothing 47$, $\varnothing 56$ diameters by using the reference drawing. You can implement any of the methods described above. The final sketch created with the **3D Revolve** operation is shown in Fig. 18.

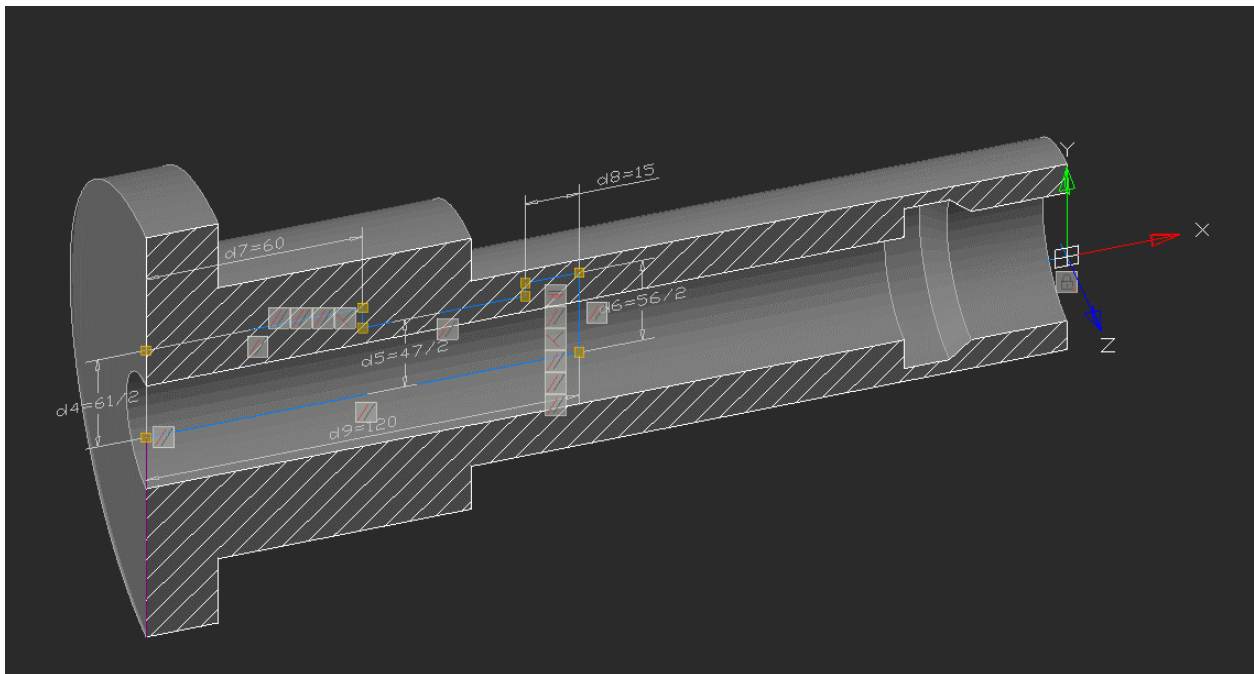


Fig. 18. The sketch with the 3D Revolve operation

6. The 3D Fillet and 3D Chamfer

Let's create fillets.

6.1. Activate the **3D Fillet** command (the **Fillet Edge** button) located on the **3D Tools** tab. Enter the radius value 2.5 in the dialog box. Then select the edges of the areas where the fillets will be drawn. Confirm the elements created with **OK** (Fig. 19).

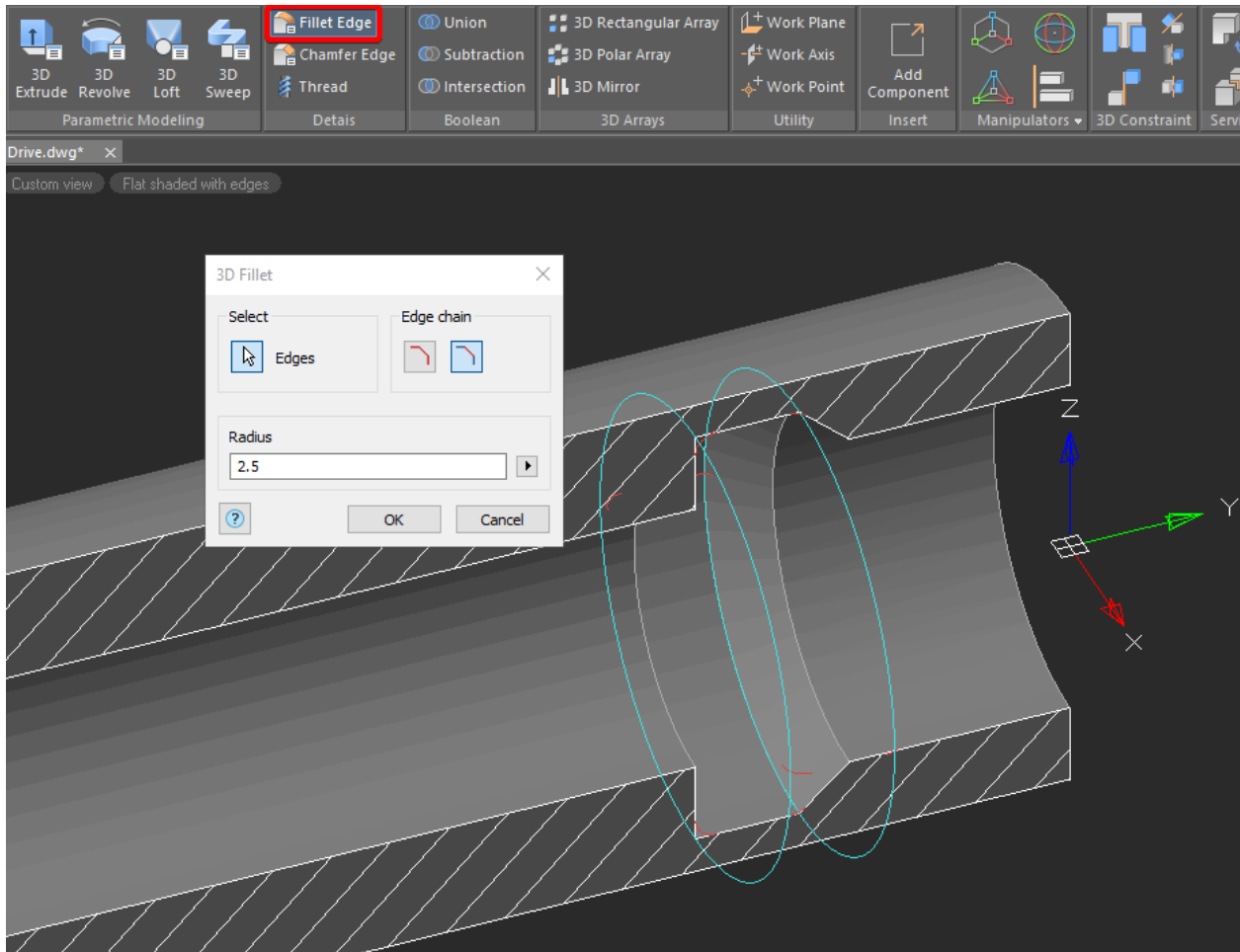


Fig. 19. Creating the Fillets

6.2. Create chamfers on the end surfaces by using the dimensions shown in the drawing.

6.3. Activate the **3D Chamfer** command and select **Distance** and **Angle** from the specified face option. Enter a value, pick a face, and then an edge you want to chamfer (Fig. 20). Confirm the action.

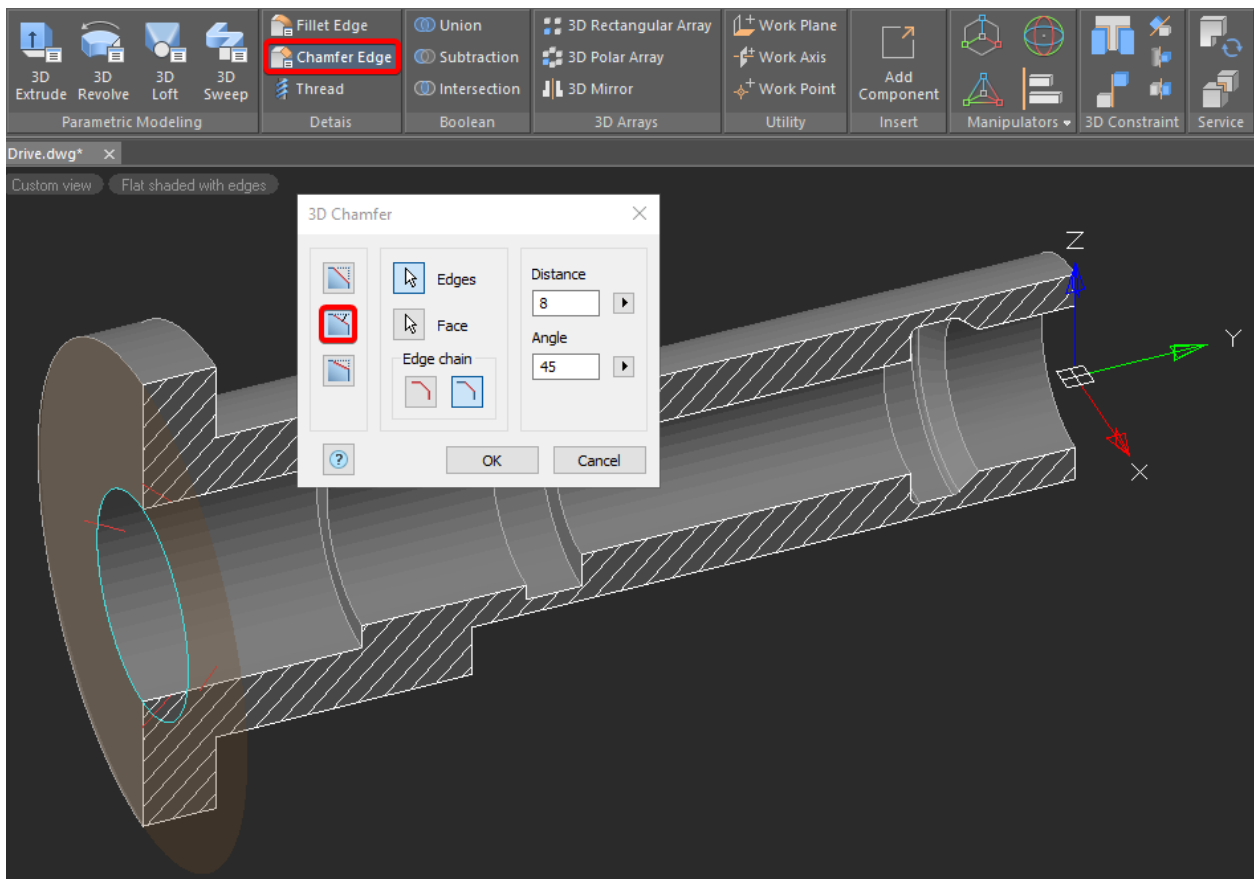


Fig. 20. Creating the chamfer

6.4. Create another chamfer on the opposite end. Follow the same instructions.

7. Creating Flats and Holes

7.1. To create a flat on the $\varnothing 75$ diameter step of the Part, use the **YZ** plane as the sketch reference. Draw a rectangle and apply **Auto Constraint** (Fig. 21). Project the end rib before you set dimensional constraints.

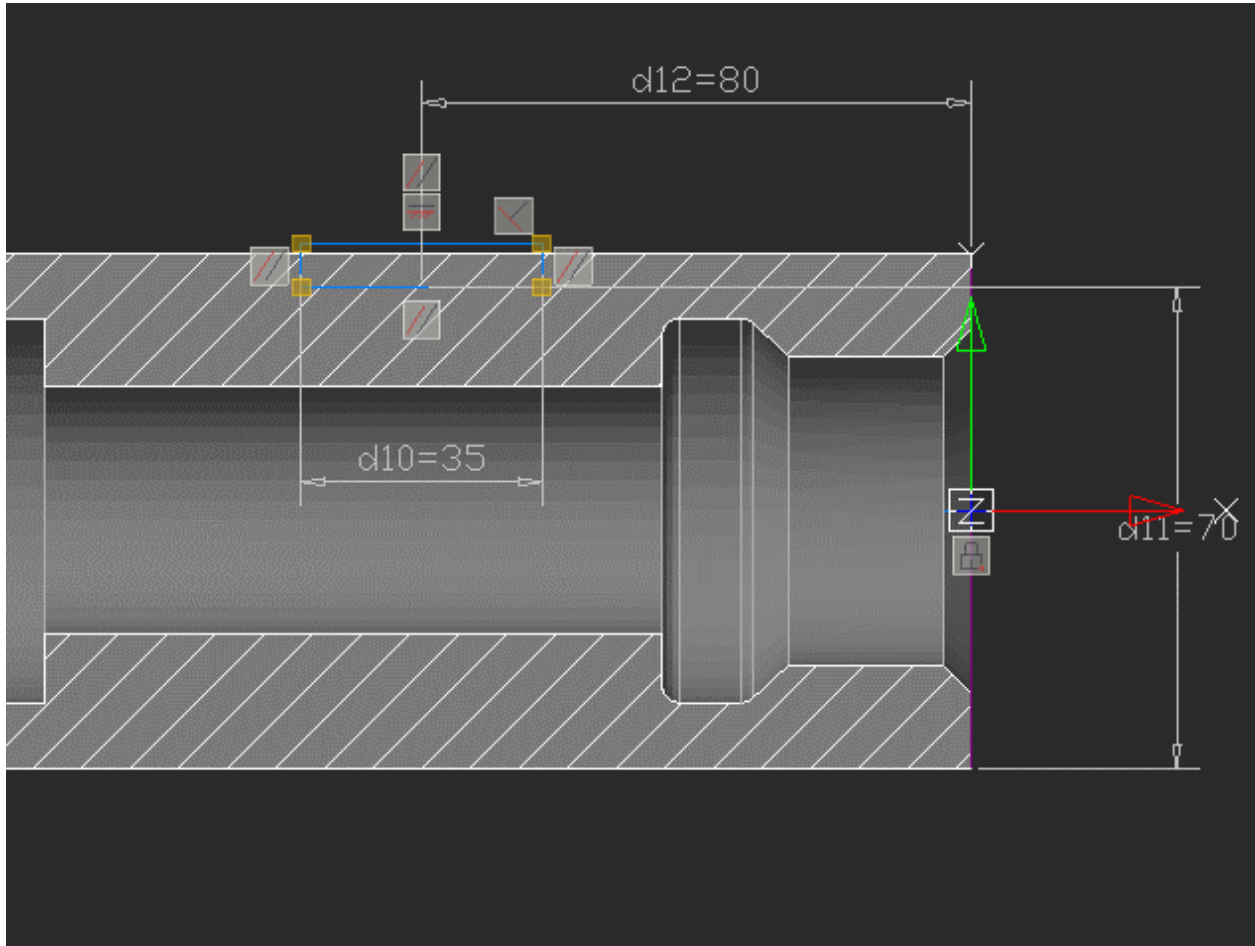


Fig. 21. The Sketch of the flat

7.2. After complete sketch construction, run the **3D Extrude** command with the following options: Cut, Through, Both Directions.

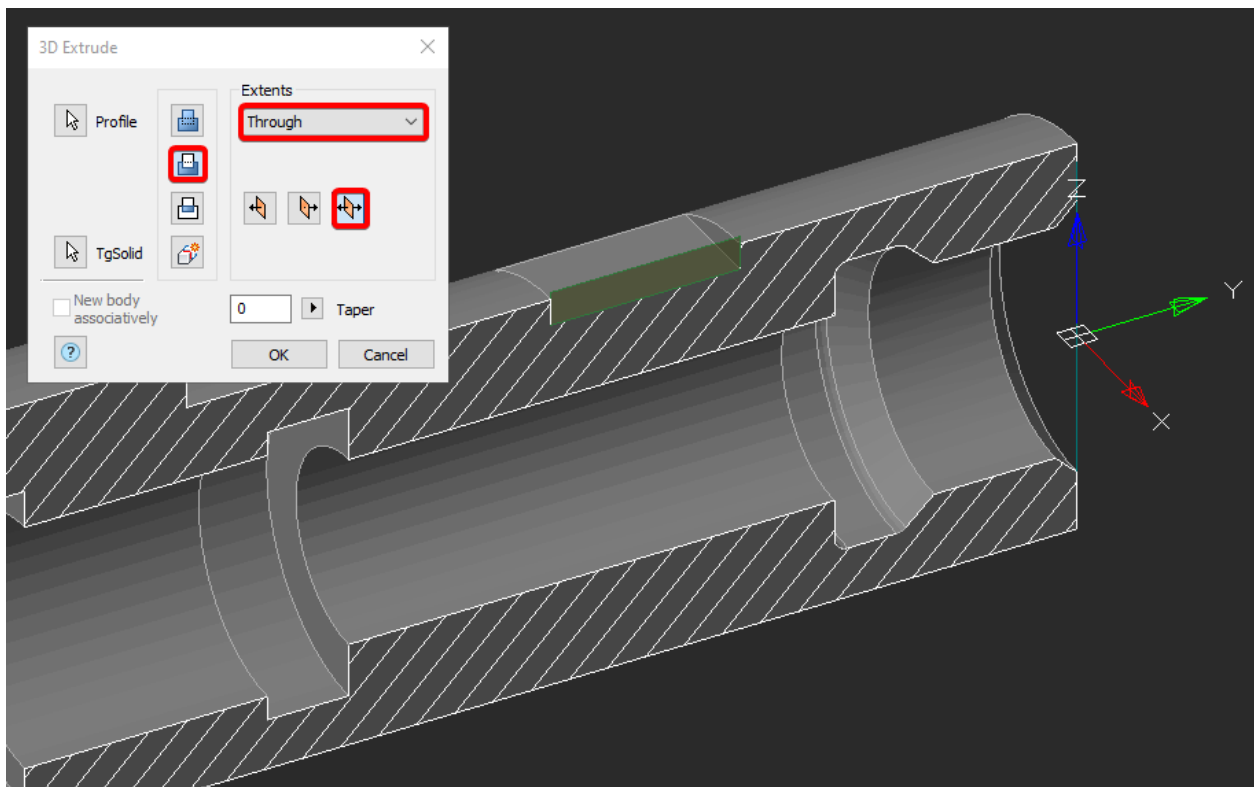


Fig. 22. The parameters of the 3D Extrude operation for the first flat

7.3. Draw a second flat on the opposite side. Use the end face of the $\varnothing 95$ diameter step as a reference plane for the sketch. The length of the rectangle for the sketch must be at least $2/3$ of the step diameter (Fig. 23).

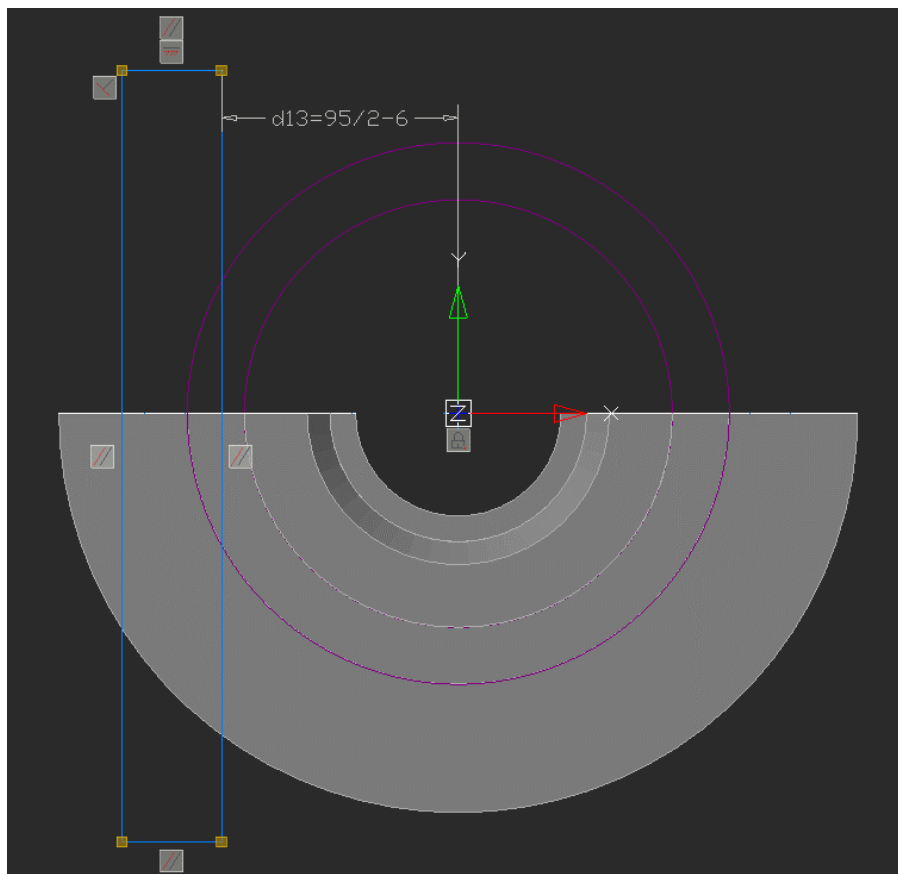


Fig. 23. The sketch of the second flat

Perform the **3D Extrude** operation at the distance of 30 mm in the negative direction with the **Cut** option enabled (Fig. 24).

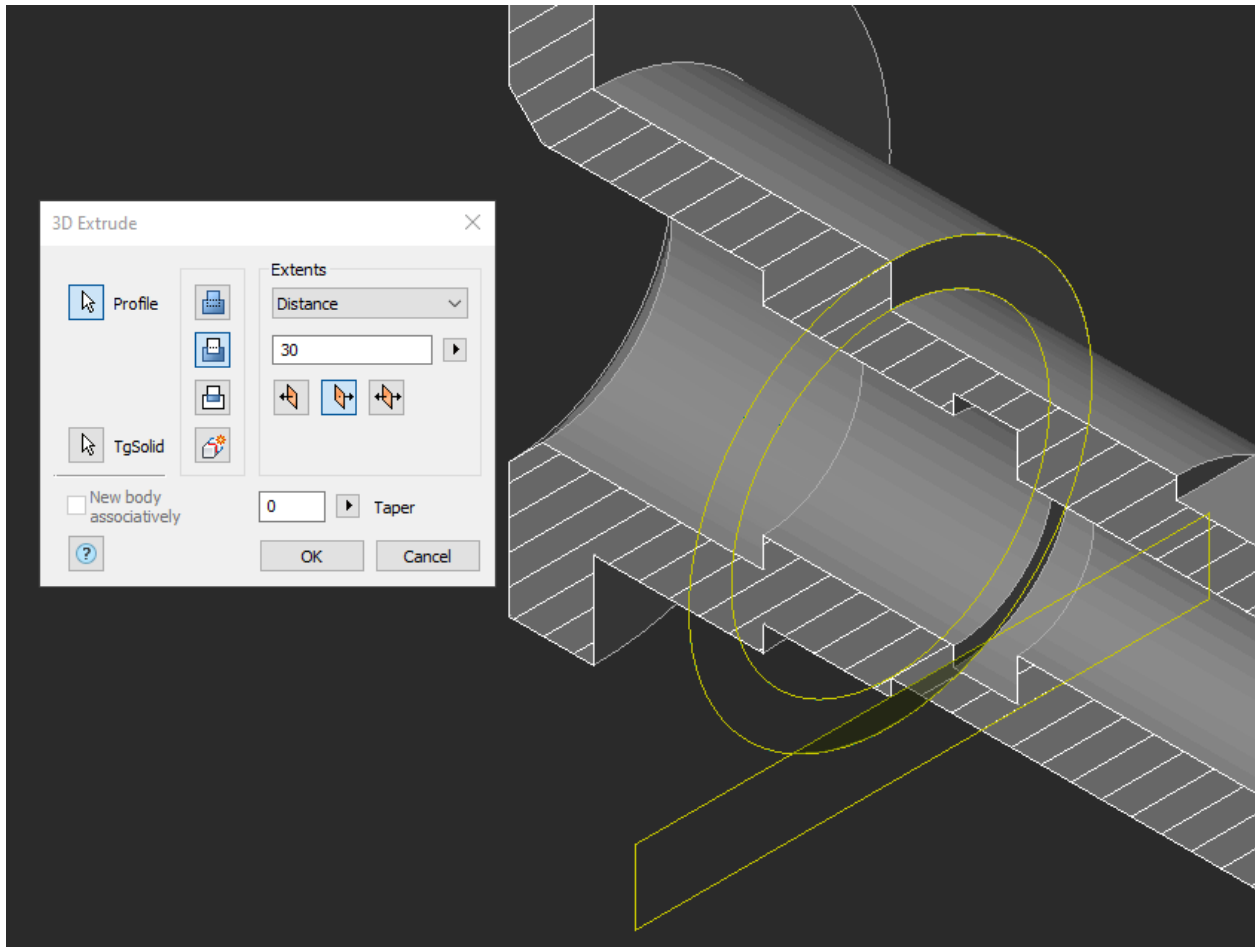


Fig. 24. Second flat draft

7.4. Create the thread holes on the constructed flats.

In the sketch mode, select the first flat and draw the hole $\varnothing 14$ in the center by using snaps (Fig. 25).

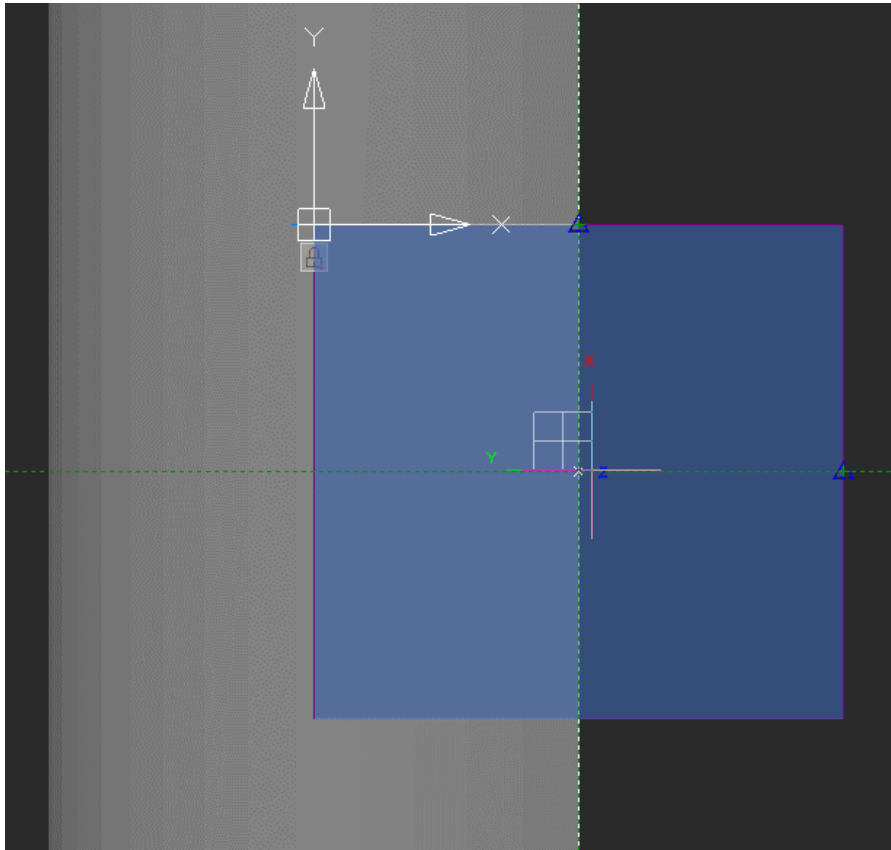


Fig. 25. Creating the hole sketch. The Midpoint snap.

Apply the **3D Extrude** operation with the **Cut** option at the distance of 30 mm in the negative direction. Create the chamfer 1,6 mm x 45° according to the dimensions shown in the drawing.

7.5. Create the hole $\varnothing 17.4$ mm in the second flat the same way.

7.6. Create the sketch for the $\varnothing 12$ hole (Fig. 26) on the **YZ** plane. Project $\varnothing 140$ diameter edge to link the dimension of the hole by X axis. Set the **3D Extrude** options to **Through**, **Negative direction**.

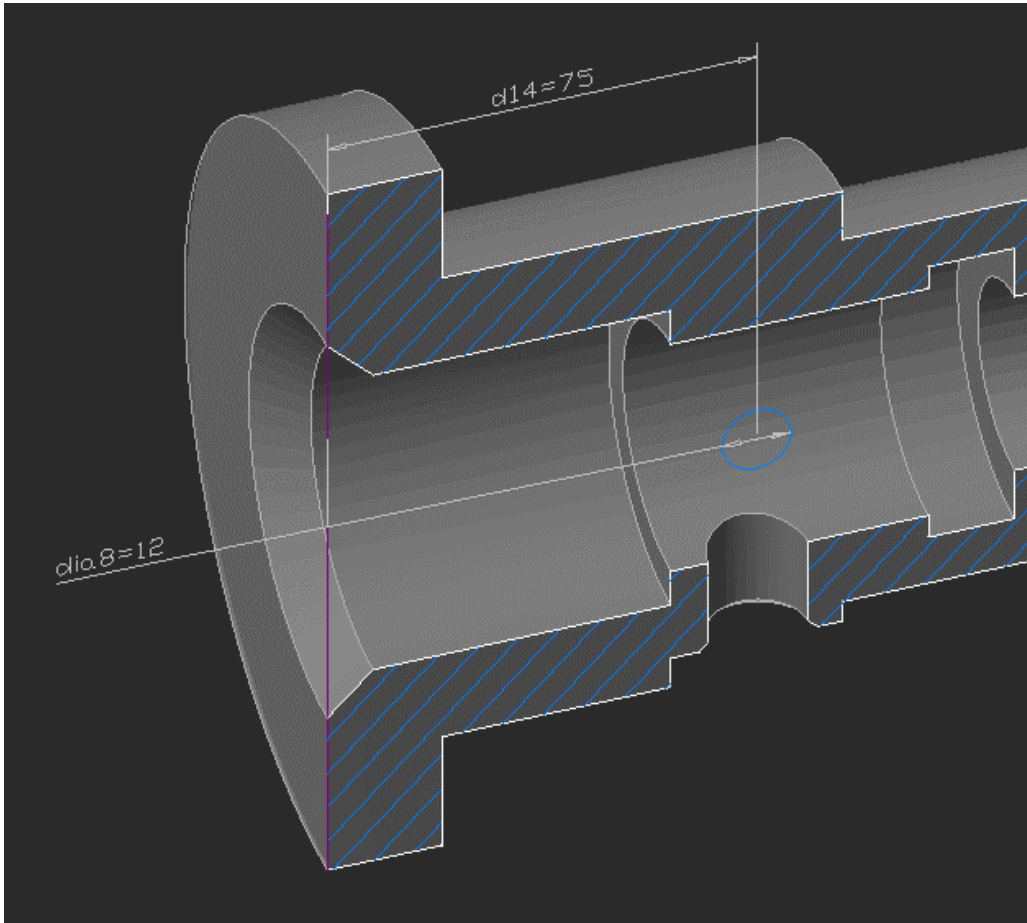


Fig. 26. The sketch for the $\varnothing 12$ hole

8. The 3D Array

Create $\varnothing 15$ holes with the 3D Polar Array operation.

8.1. Sketch a hole at the end of the largest step. Use snaps to position the hole accurately in relation to the center (Fig. 27).

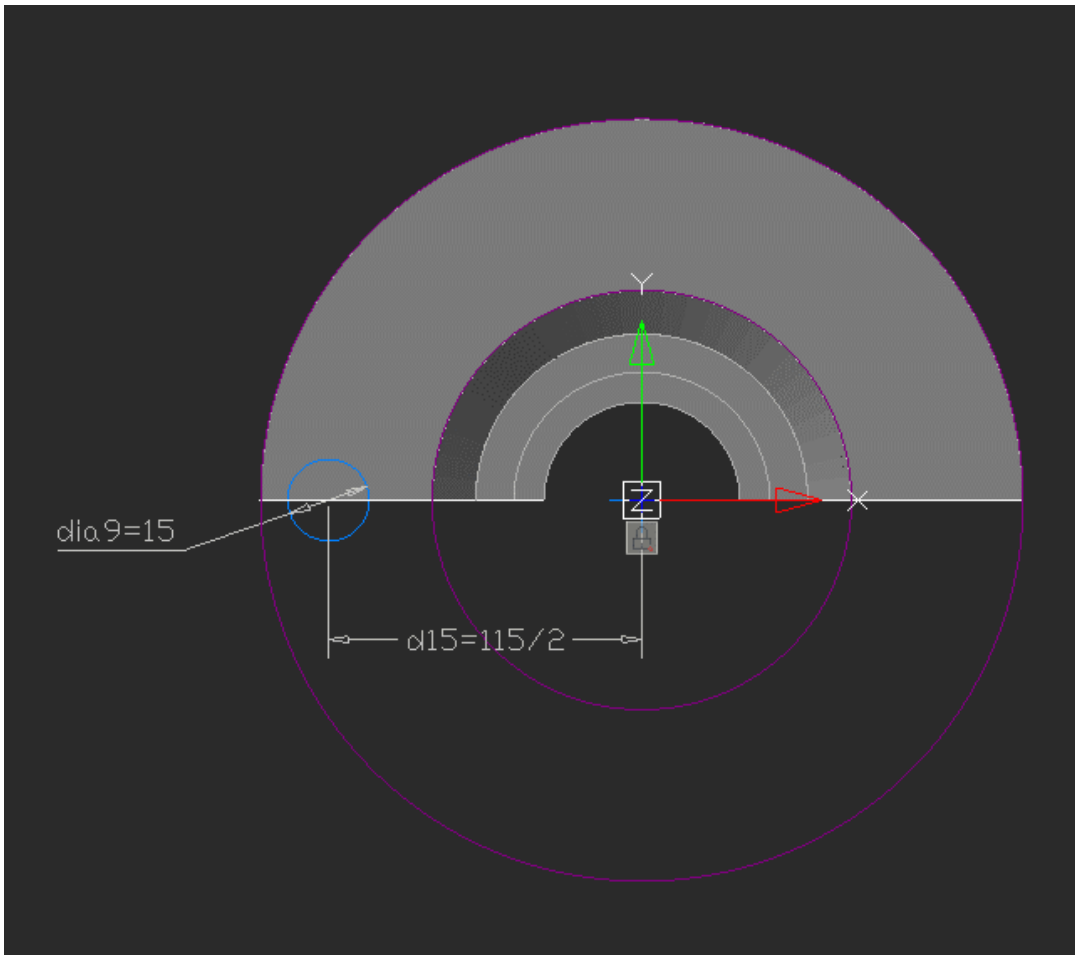


Fig. 27. The sketch of the $\varnothing 15$ mm holes

8.2. Apply the **3D Extrude** operation with the **Cut** and **Through** options enabled.

8.3. Activate the command **3D Polar Array** (Fig. 28).

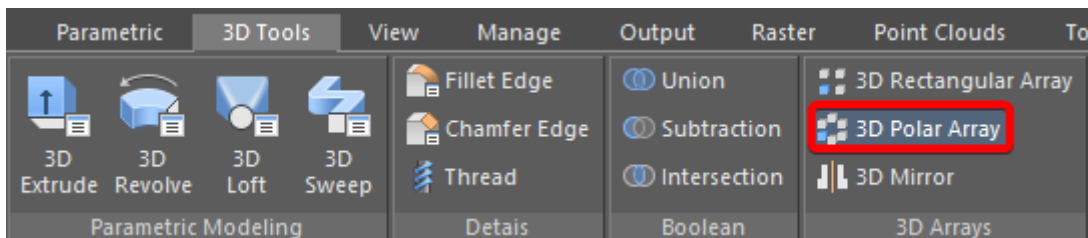


Fig. 28. 3D Polar Array command

8.3.1. Click on the hole and select the rotation axis with a click on the 3D model space.

8.3.2. Add the following parameters in the dialog box: the number of items – 6, the angle 360° (Fig. 29).

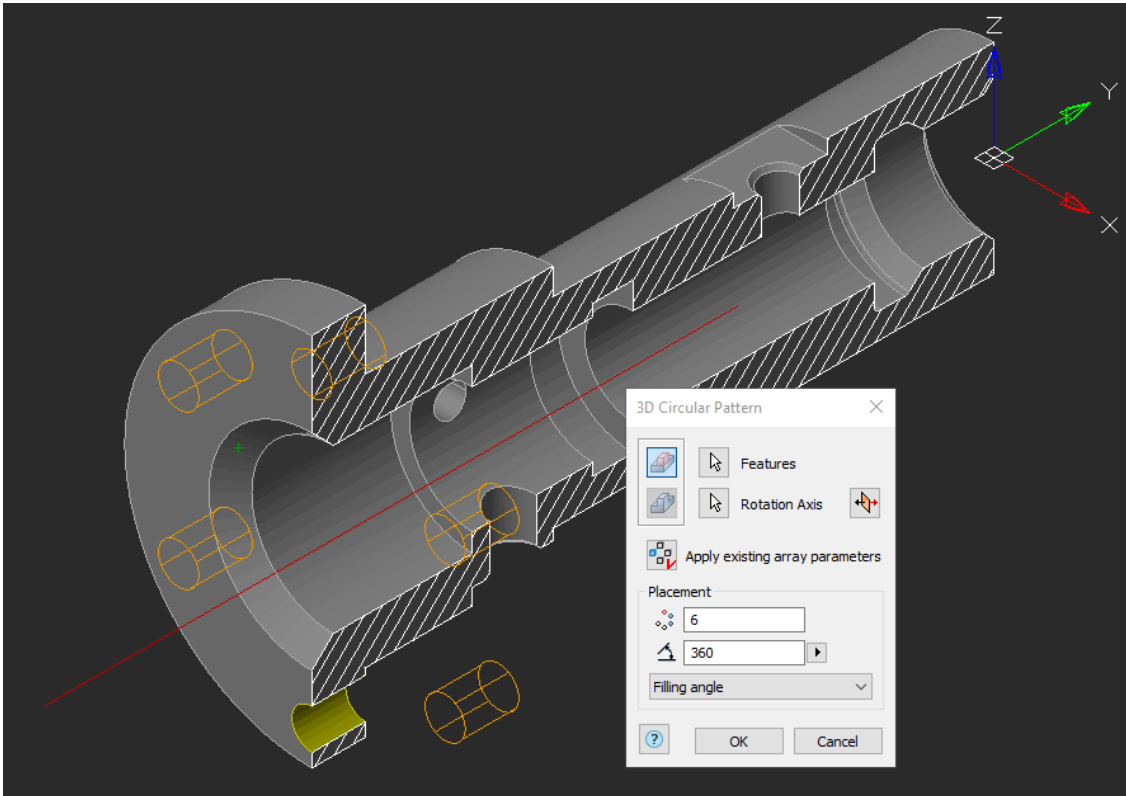


Fig. 29. The 3D Polar Array — Parameters

8.4. For a more realistic view of the model, enable the **Perspective view** in the **View** tab (Fig. 30).

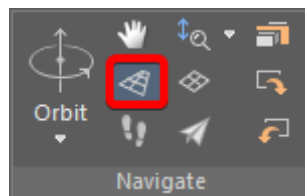


Fig. 30. Perspective view

8.5. You can change the color of the model. Select the Part in the **3D History** and choose the desired color in the **Properties**.

8.6. You can turn off the **UCS** sign in the **Properties** if needed.

8.7. Save the changes. The final view of the model is shown in Fig. 31.

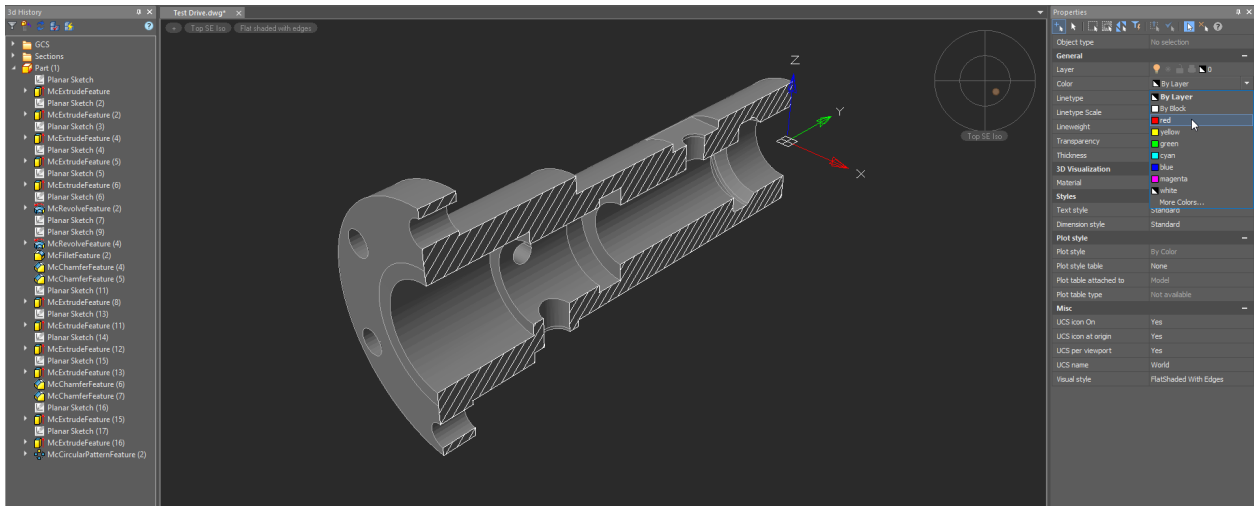


Fig. 31. The final view of the model