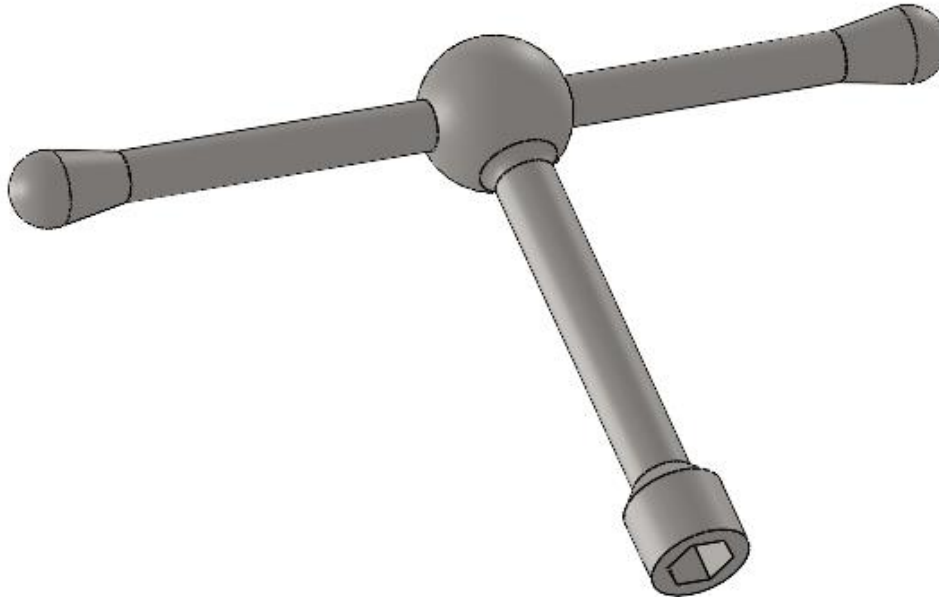
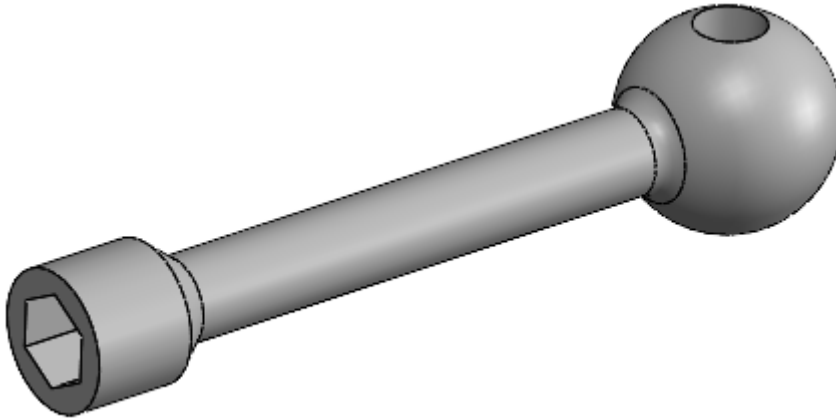




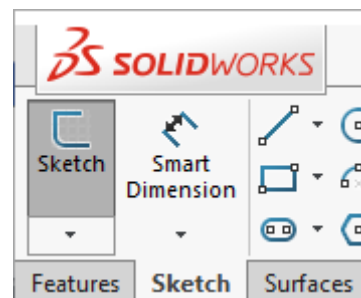
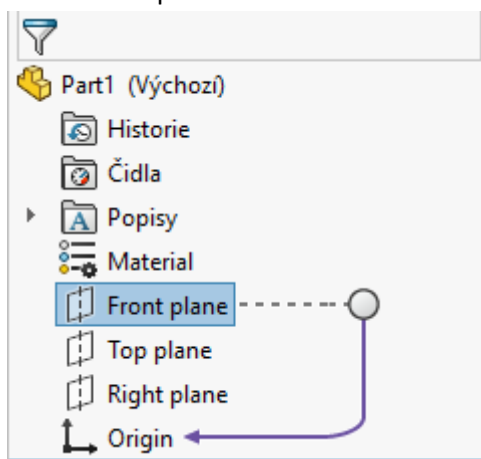
Wheel screw wrench



1. Body



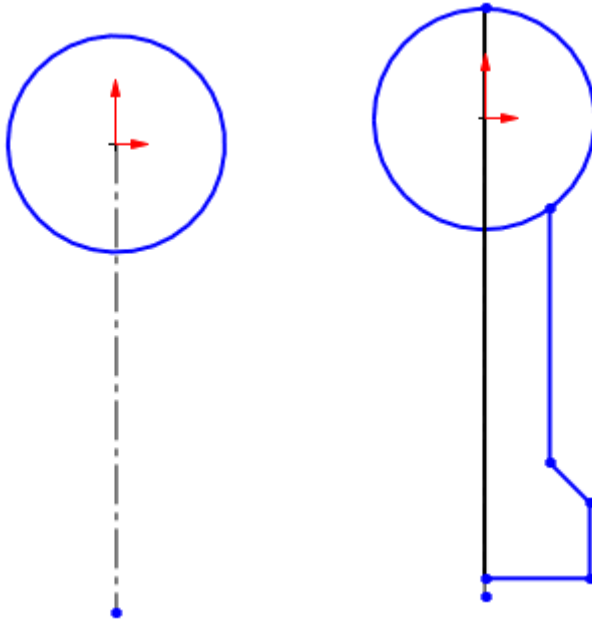
1. Open New File - Part
2. Select Front plane and Sketch



3. Draw the axis from the Origin point



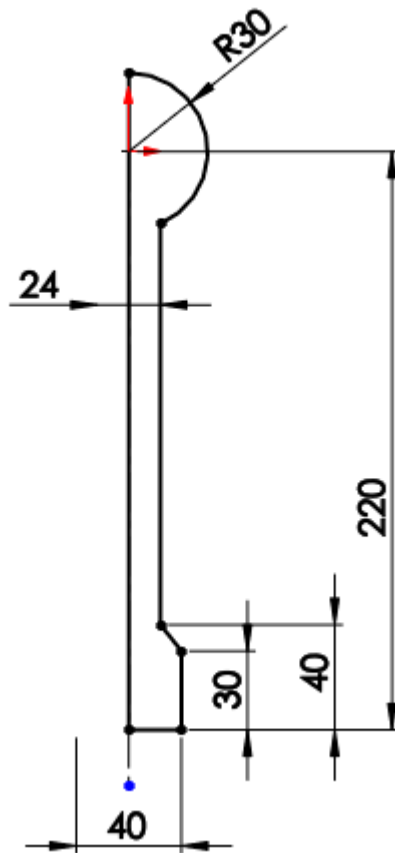
4. Draw a circle and contour edges



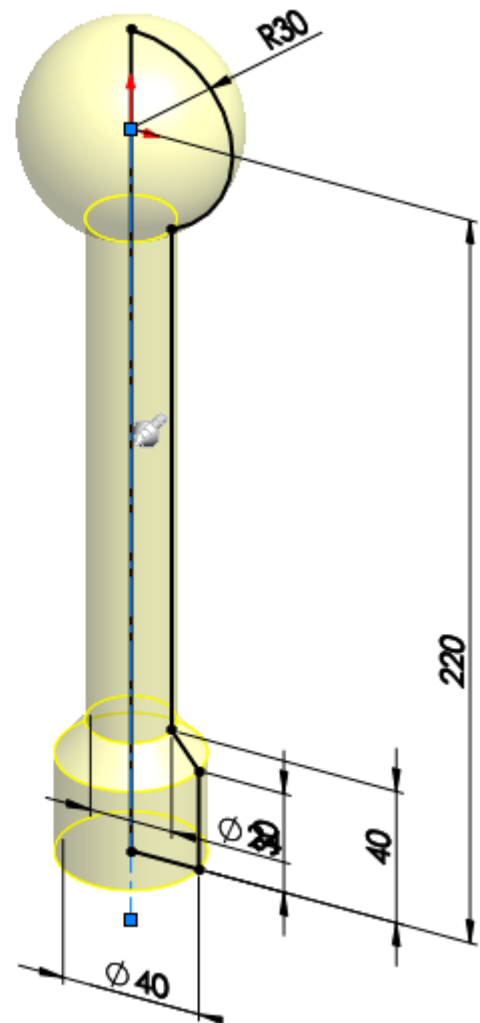
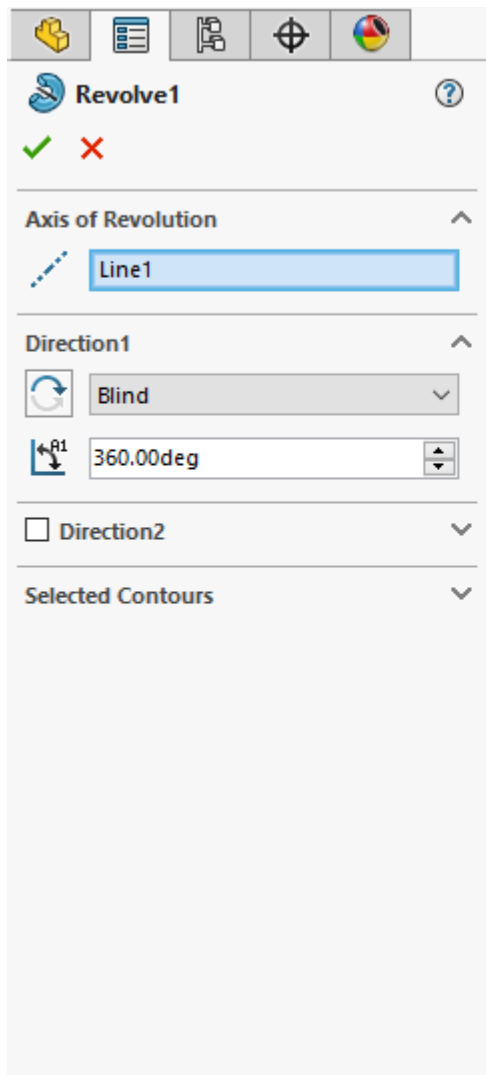
5. Remove excess elements with Trim Entities



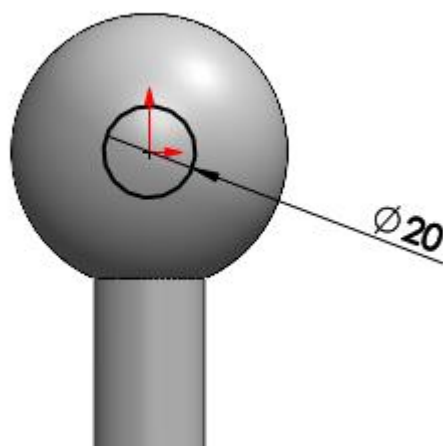
6. Create dimensions of the elements



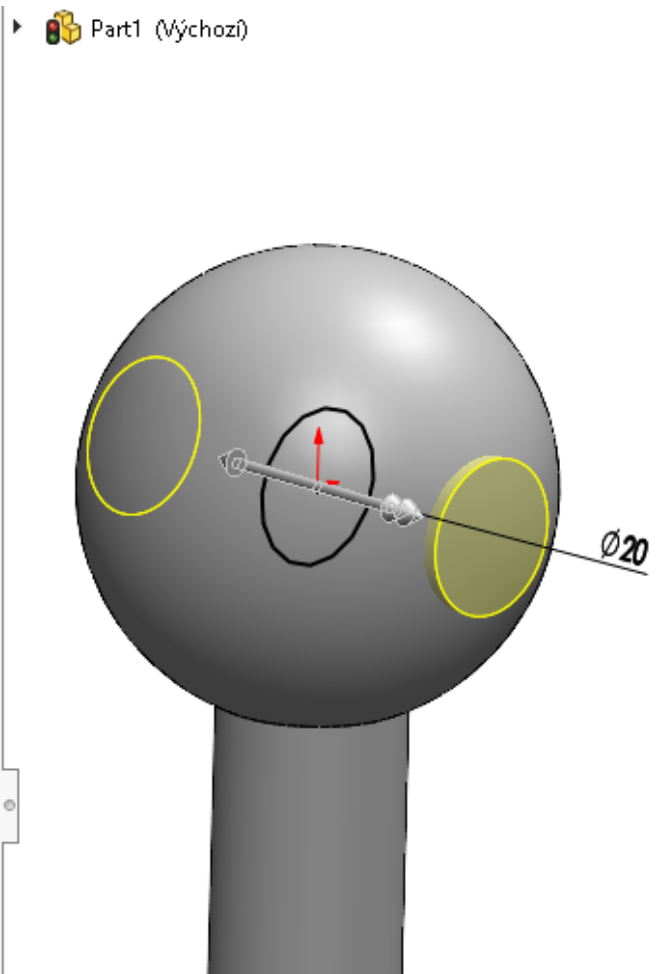
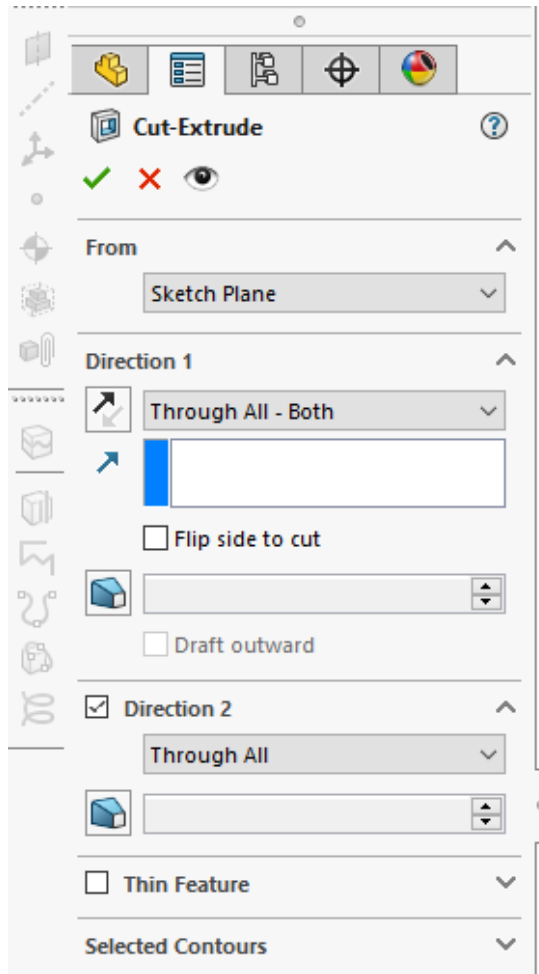
- Exit the sketch and make a Revolve along the axis by 360°



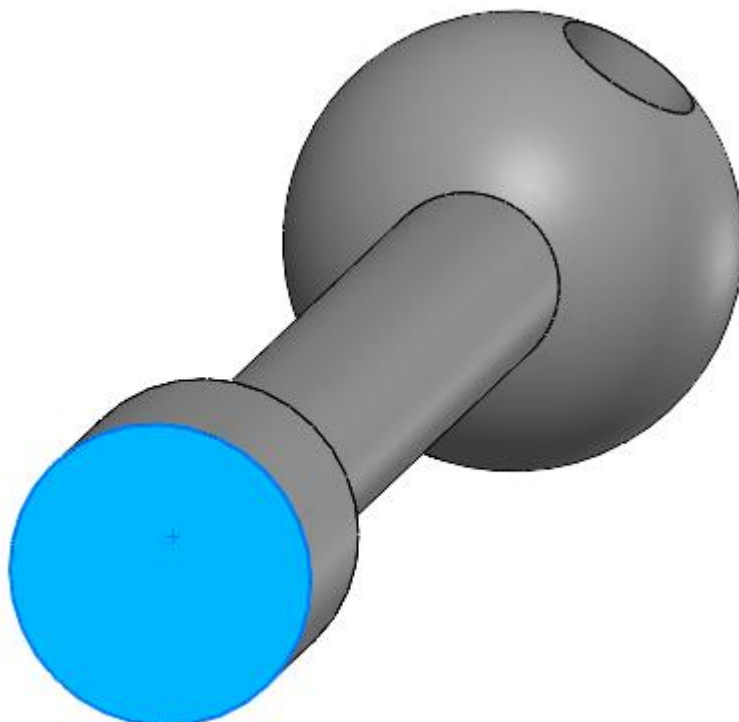
- Select Front Plane and Sketch, draw a circle to the origin point and create dimensions



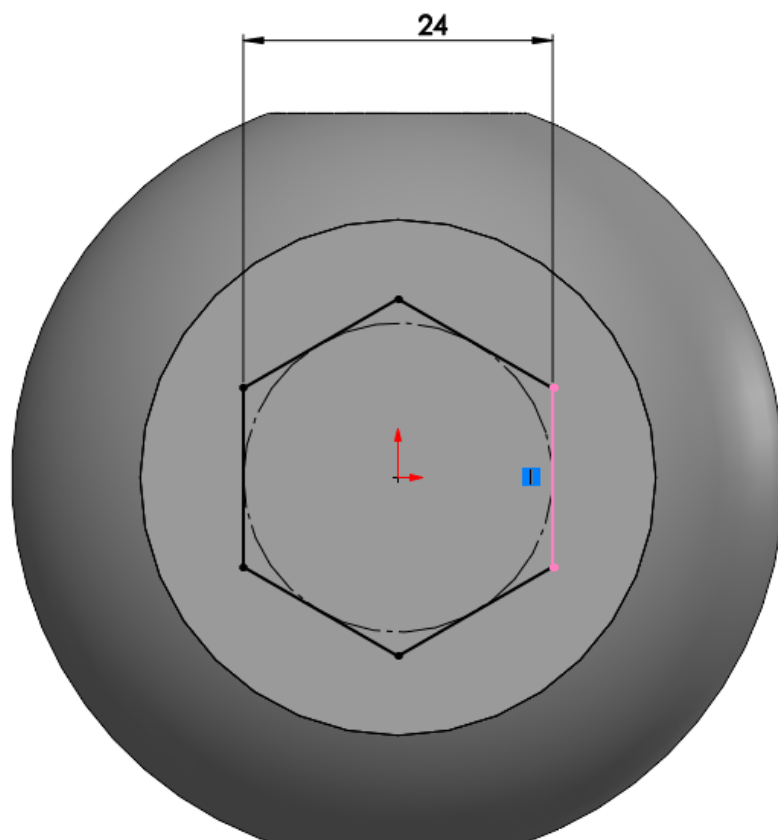
9. Make an Extrude Cut, set Through All - Both



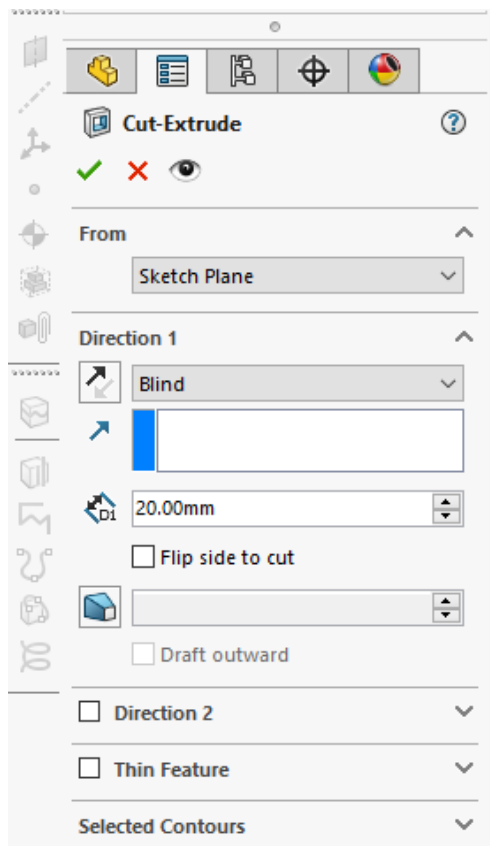
10. Select Bottom surface and Sketch



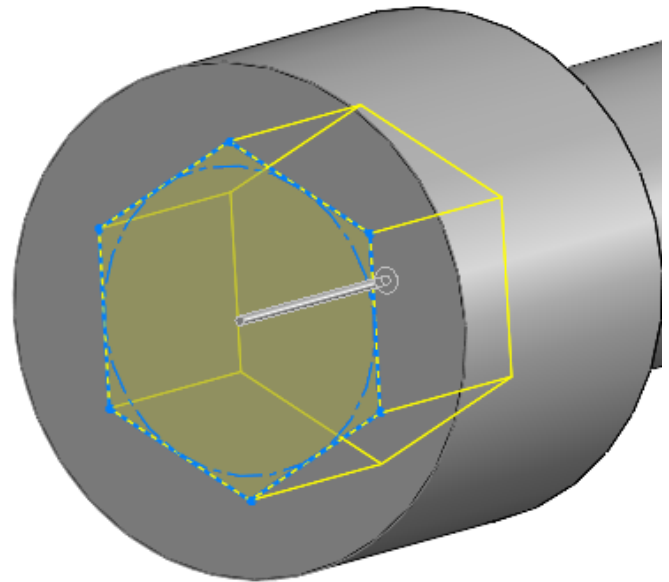
11. Draw a hexagon, assign a Vertical Snap to the edge, and create dimensions



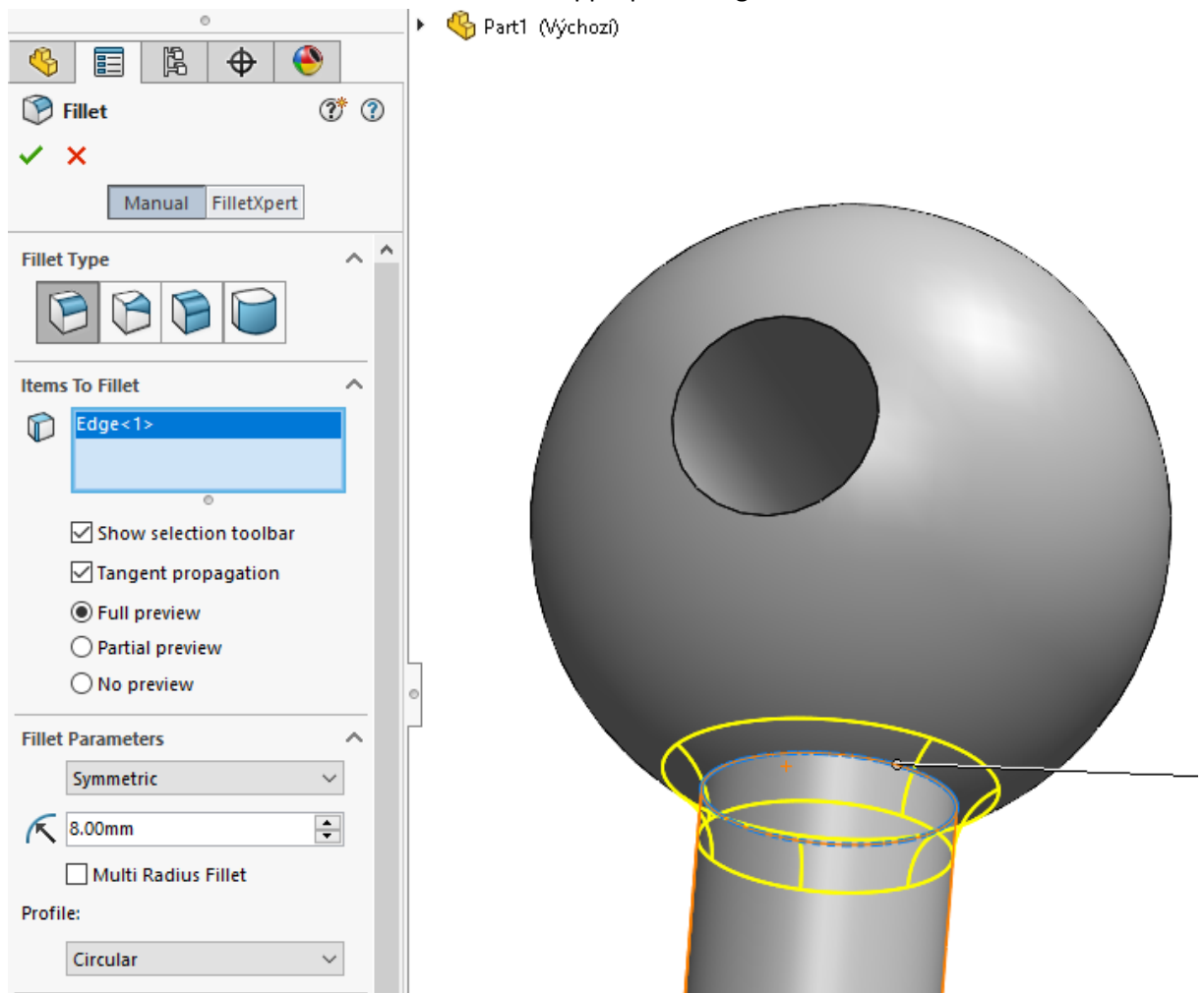
12. Exit the sketch and make Extrude Cut



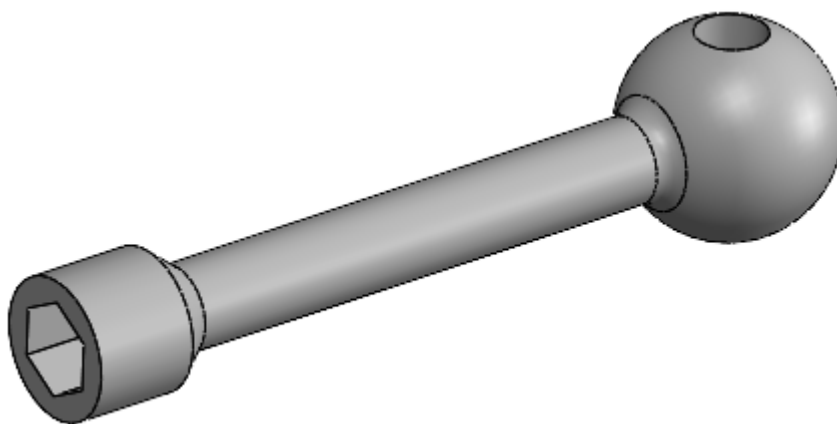
Part1 (Výchozí)



13. Create a radius with the Fillet command to the appropriate edge

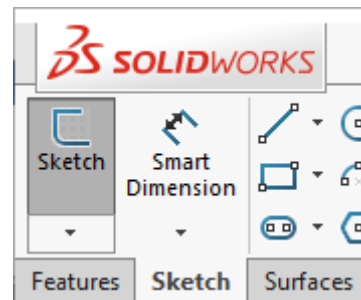
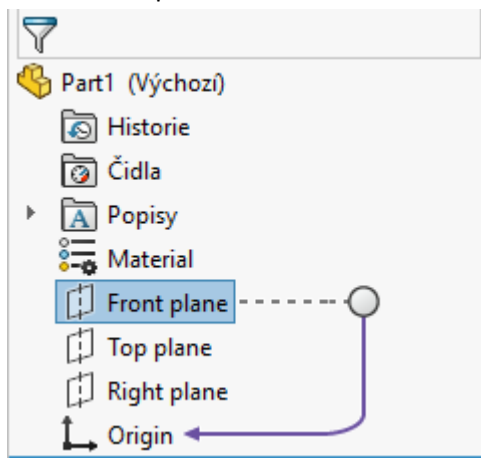


14. Save - The part is done



2. Handle

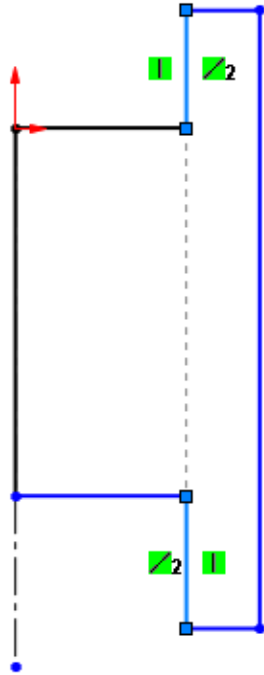
1. Open New File - Part
2. Select Front plane and Sketch



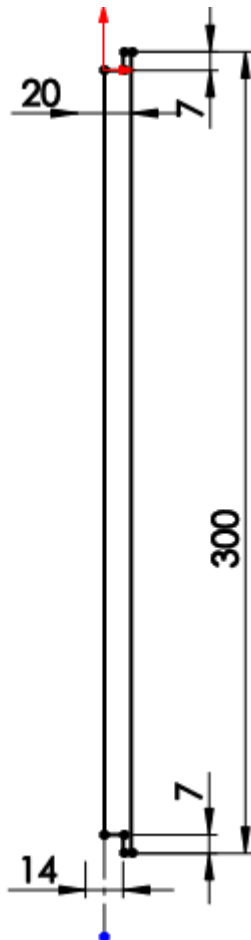
3. Draw the axis from the Origin point



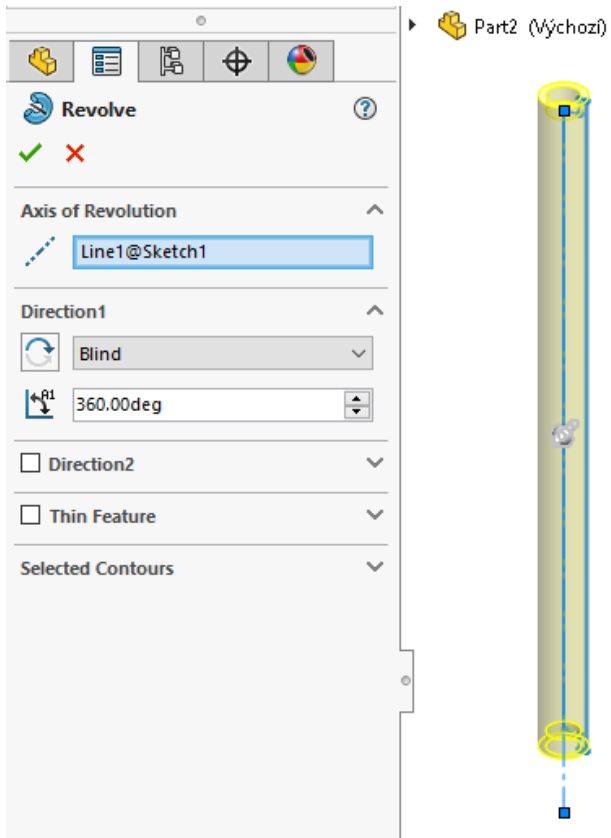
4. Draw the contour edges and assign Collinear Snap to the appropriate edges



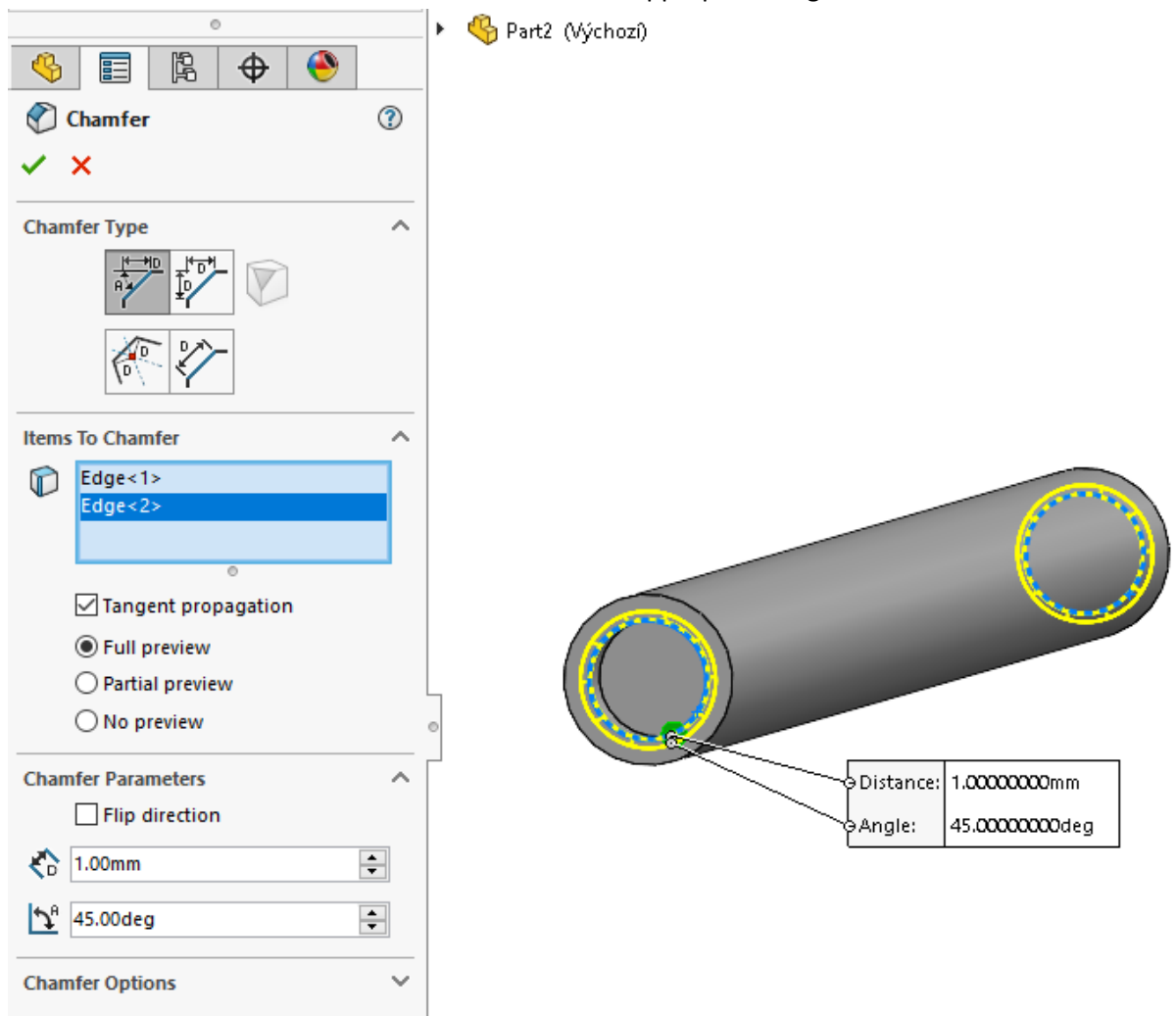
5. Create dimensions of the elements



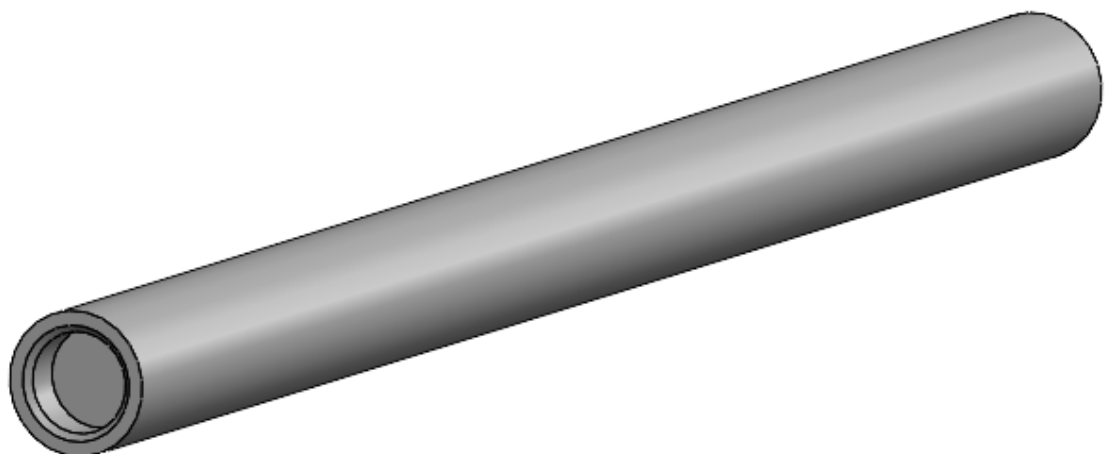
6. Exit the sketch and make a Revolve along the axis by 360°



7. Create a chamfer with the Chamfer command on the appropriate edges

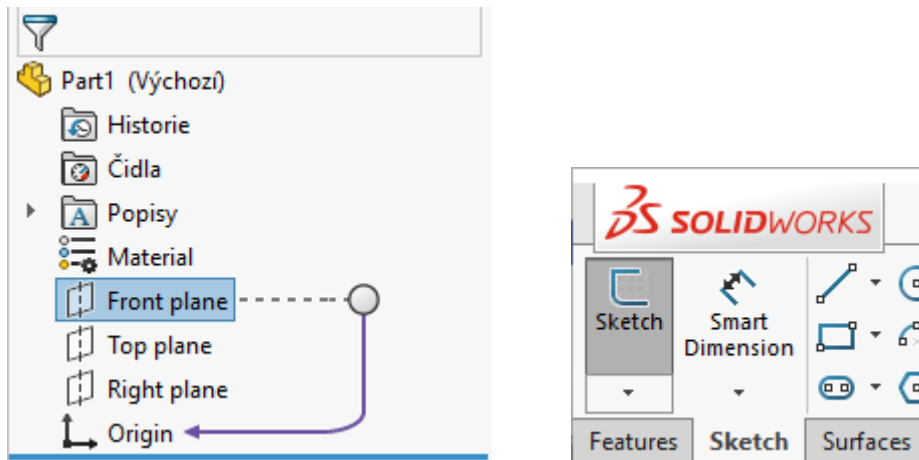


8. Save - The part is done



3. Handle

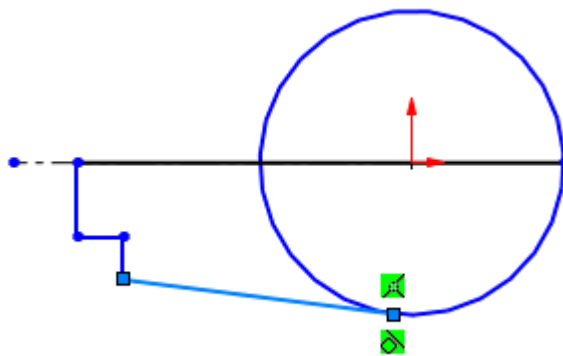
1. Open New File - Part
2. Select Front plane and Sketch



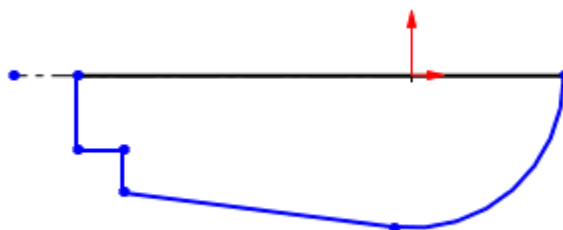
3. Draw the axis from the Origin point



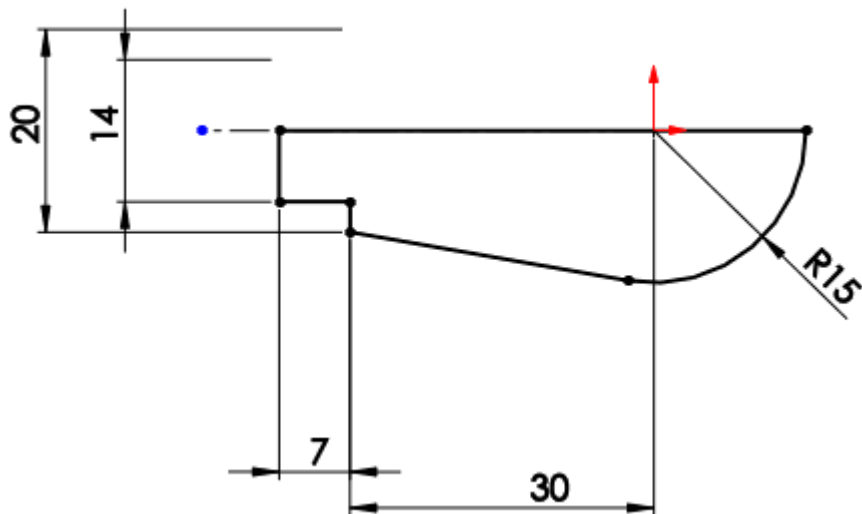
4. Draw a circle and contour edges, assign a Tangent Snap between the edge and the circle



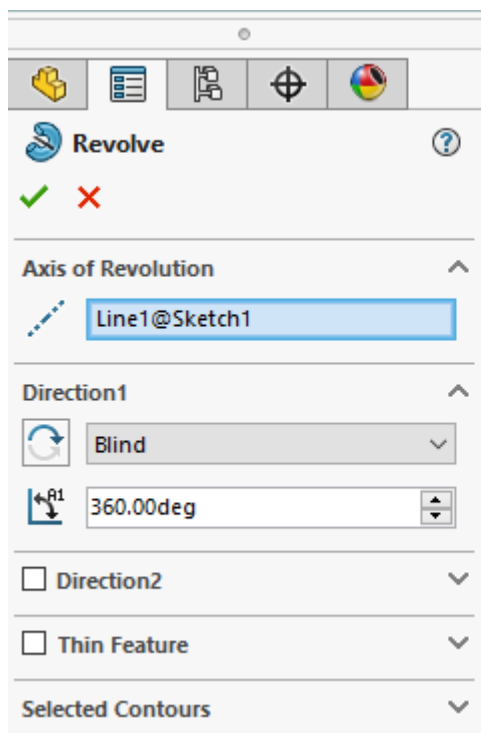
5. Remove excess edges with Trim Entities



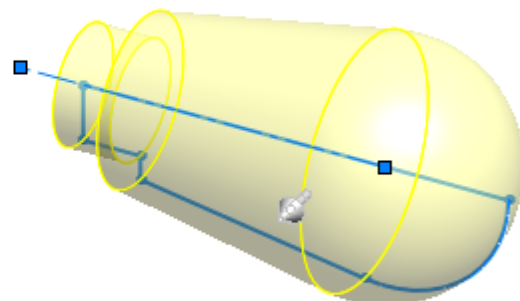
6. Create dimensions of the elements



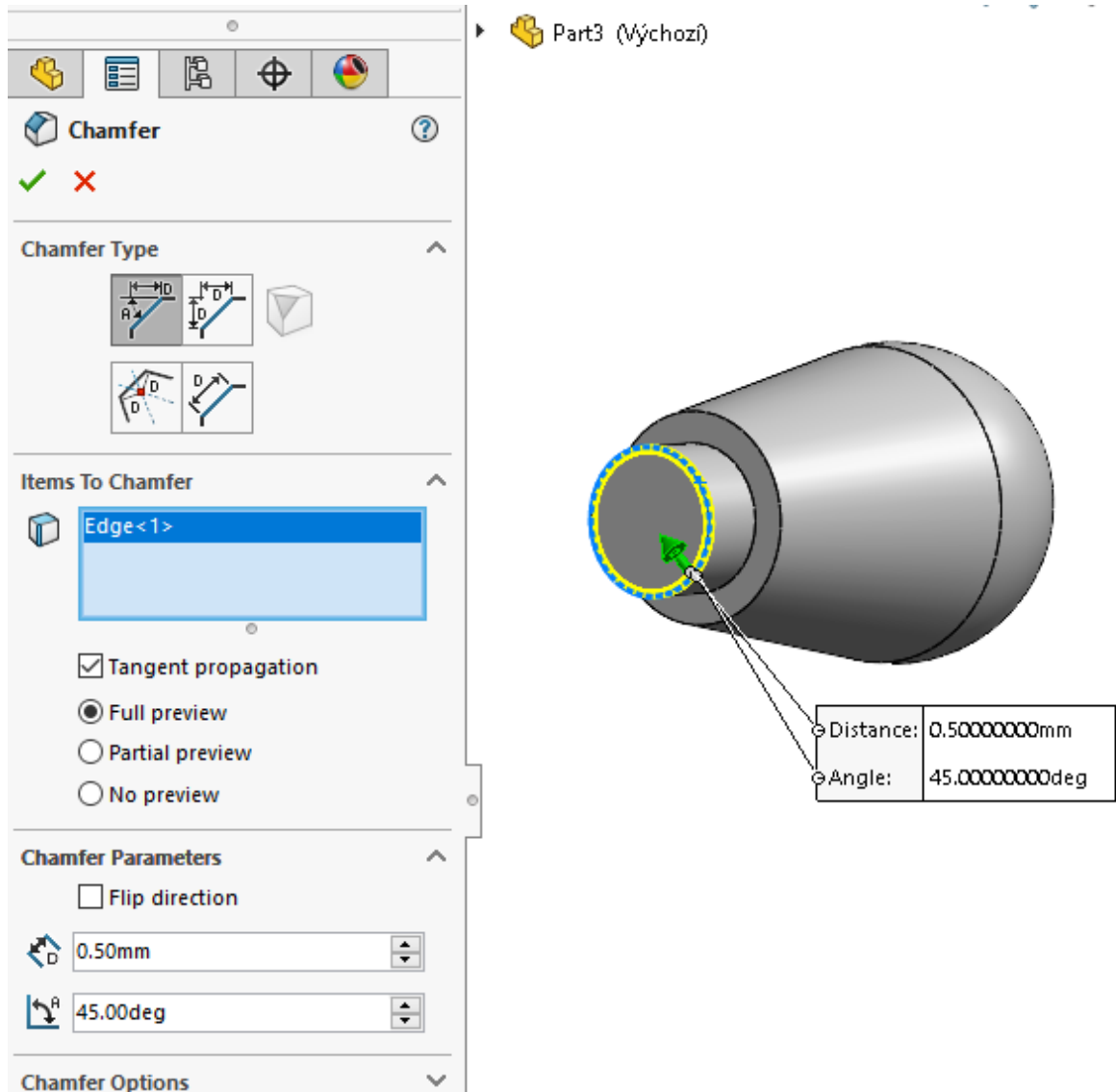
7. Exit the sketch and make a Revolve along the axis by 360°



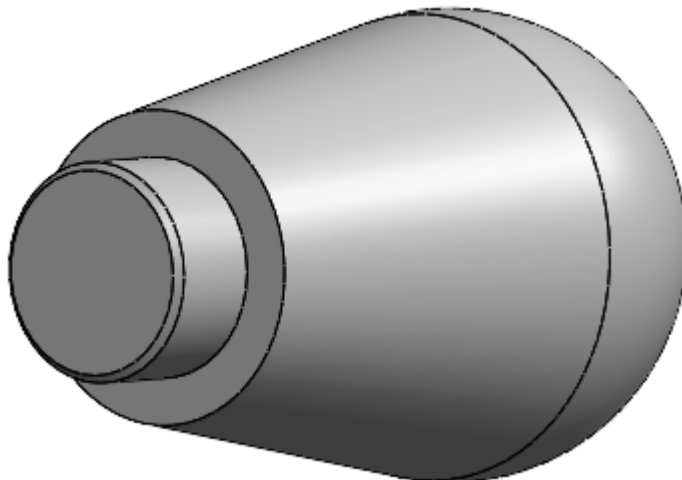
Part3 (Výchozí)



8. Create a chamfer with the Chamfer command at the appropriate edge

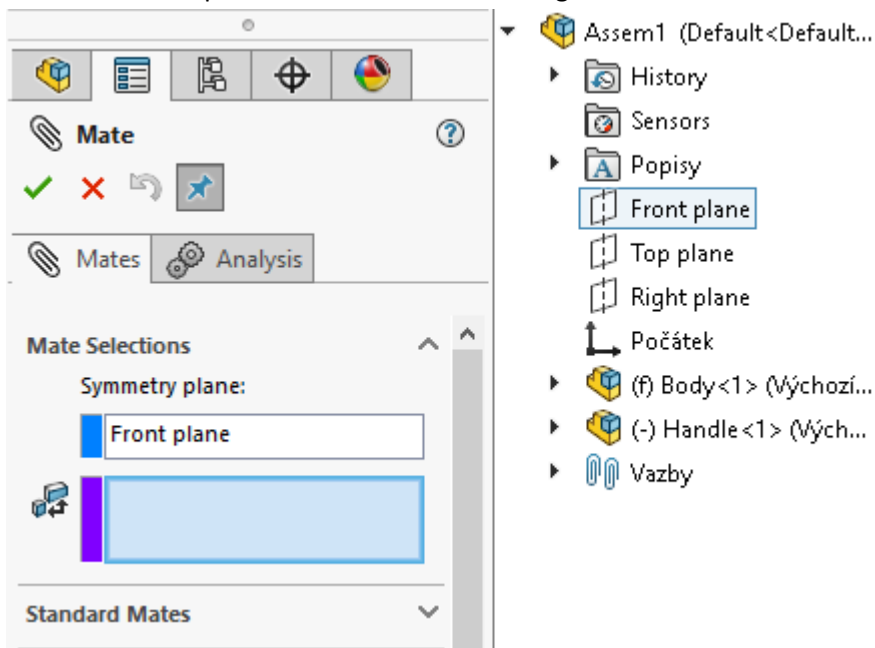


9. Save - The part is done

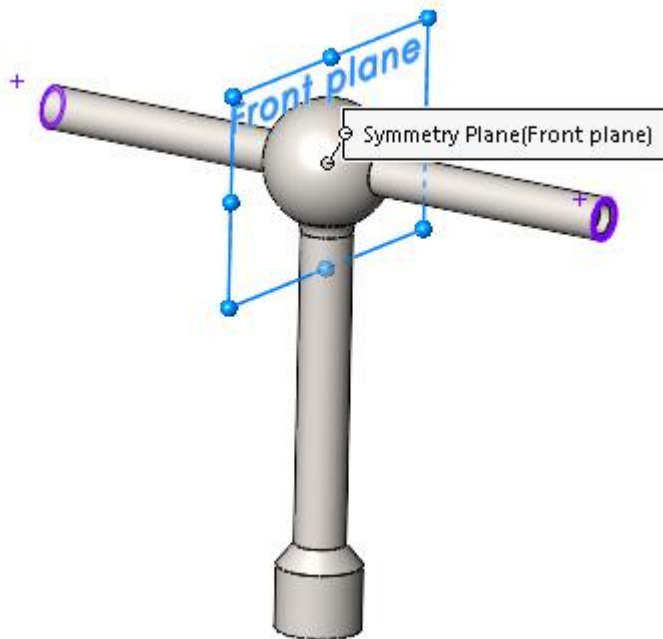


4. Assembly

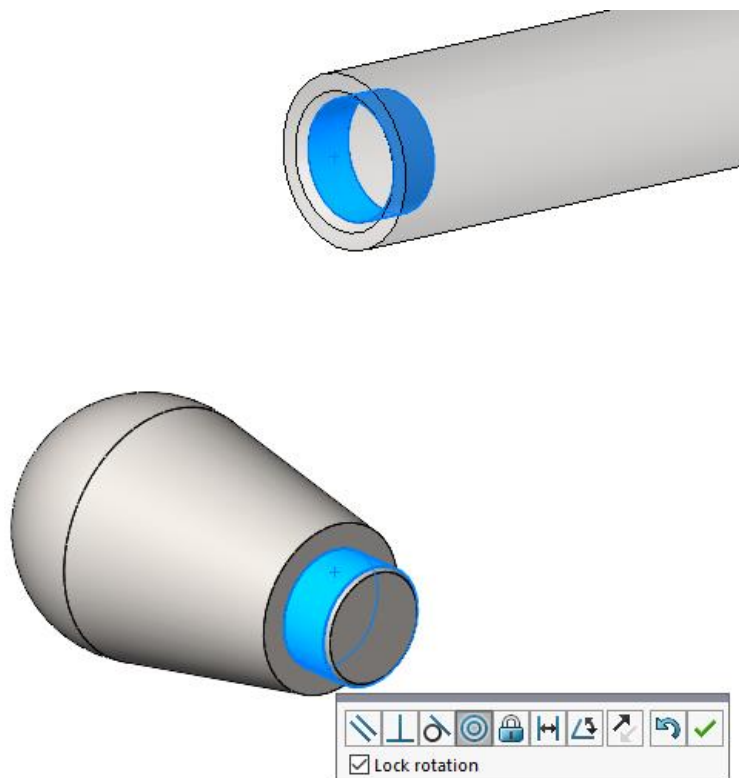
1. Open New File - Assembly
2. Find the part Body and confirm the insertion
3. Use Insert Components to find the Handle Part and insert it into the workspace
4. Run the Mate command
5. Select the appropriate surfaces on both parts. Select Standard Mates / Concentric and close the rotation option. Confirm link creation
6. Switch to Advanced Mates / Symmetric
7. Select the Front plane from the Feature Manager



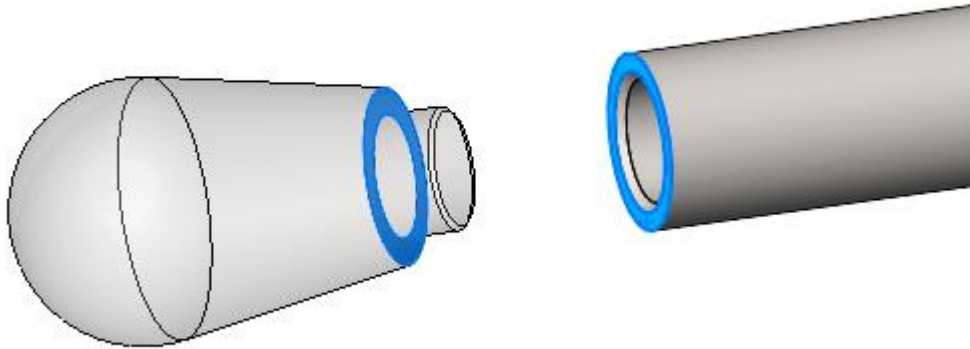
8. Select both ends surfaces at the Handle part. Confirm link creation



9. Use Insert Components to find the Head Part and insert it into the workspace
10. Run the Mate command
11. Select the appropriate surfaces on the parts. Select Standard Mates / Concentric and close the rotation option. Toggle Mate Alignment to rotate the part position. Confirm link creation



12. Select the appropriate surfaces on the parts. Select Standard Mates / Coincident. Confirm link creation



13. Repeat steps 9 - 12 for the placement of the second Head part
14. Assembly is complete

