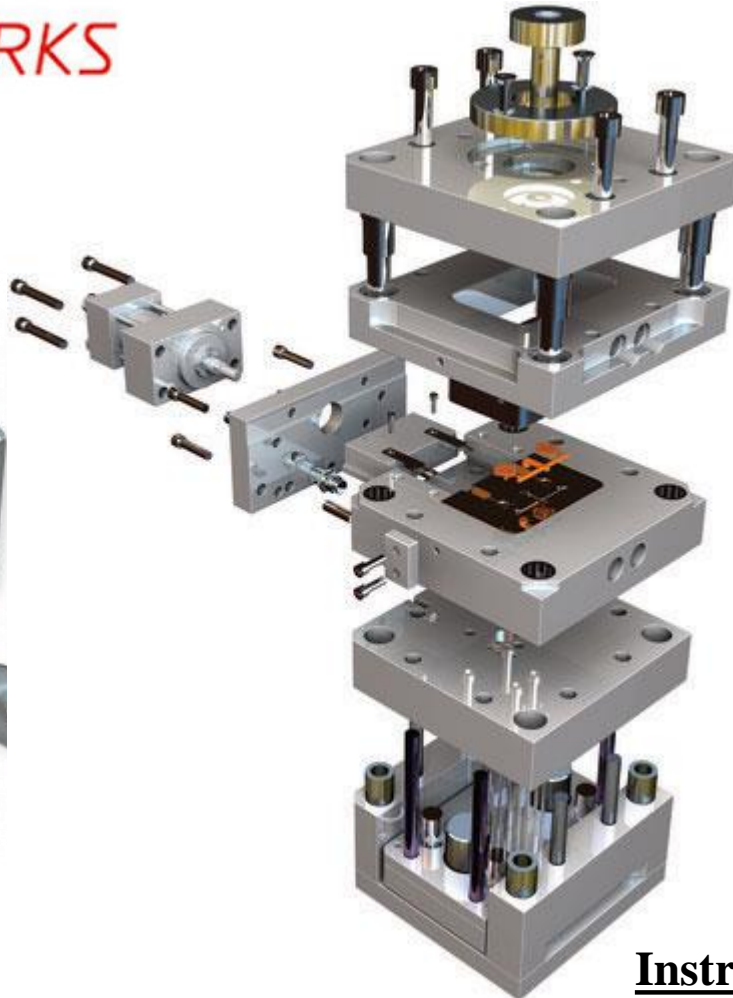


CHƯƠNG 2:

MỘT SỐ LỆNH TẠO KHỐI TRONG FEATURE

 **SOLIDWORKS**



Instructor SolidWork 2013:

Nguyễn Tấn Ý

Kiến thức cần đạt được:

■ Choose the best profile for sketching.

- Chọn mặt tốt nhất để vẽ phác

■ Choose the proper sketch plane.

- Chọn mặt phẳng phác thảo thích hợp

■ Extrude Cut Feature.

- Tính năng Extrude Cut

■ Create Hole Wizard holes.

- Tạo lỗ bằng lệnh Hole Wizard

■ Insert fillets on a solid.

- Bo cung trên khối rắn

Tham khảo: SolidWork Essentials 2011

Lesson 3: Basic Part Modeling (Trang 61)



Extrude



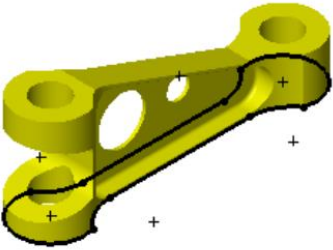
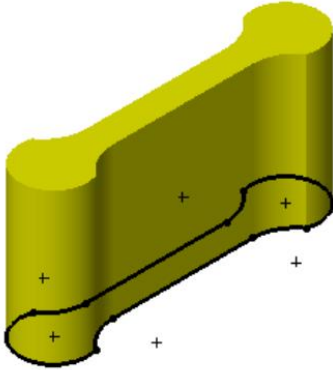

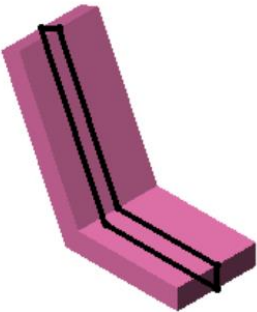
Revolve

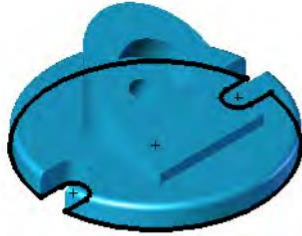
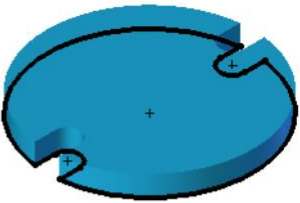
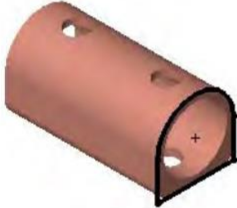
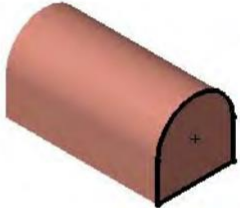
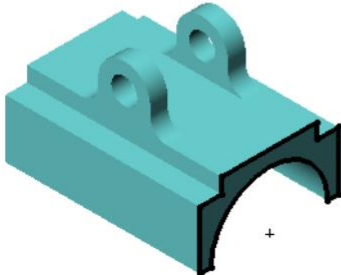
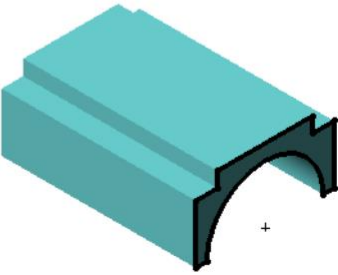


Sweep

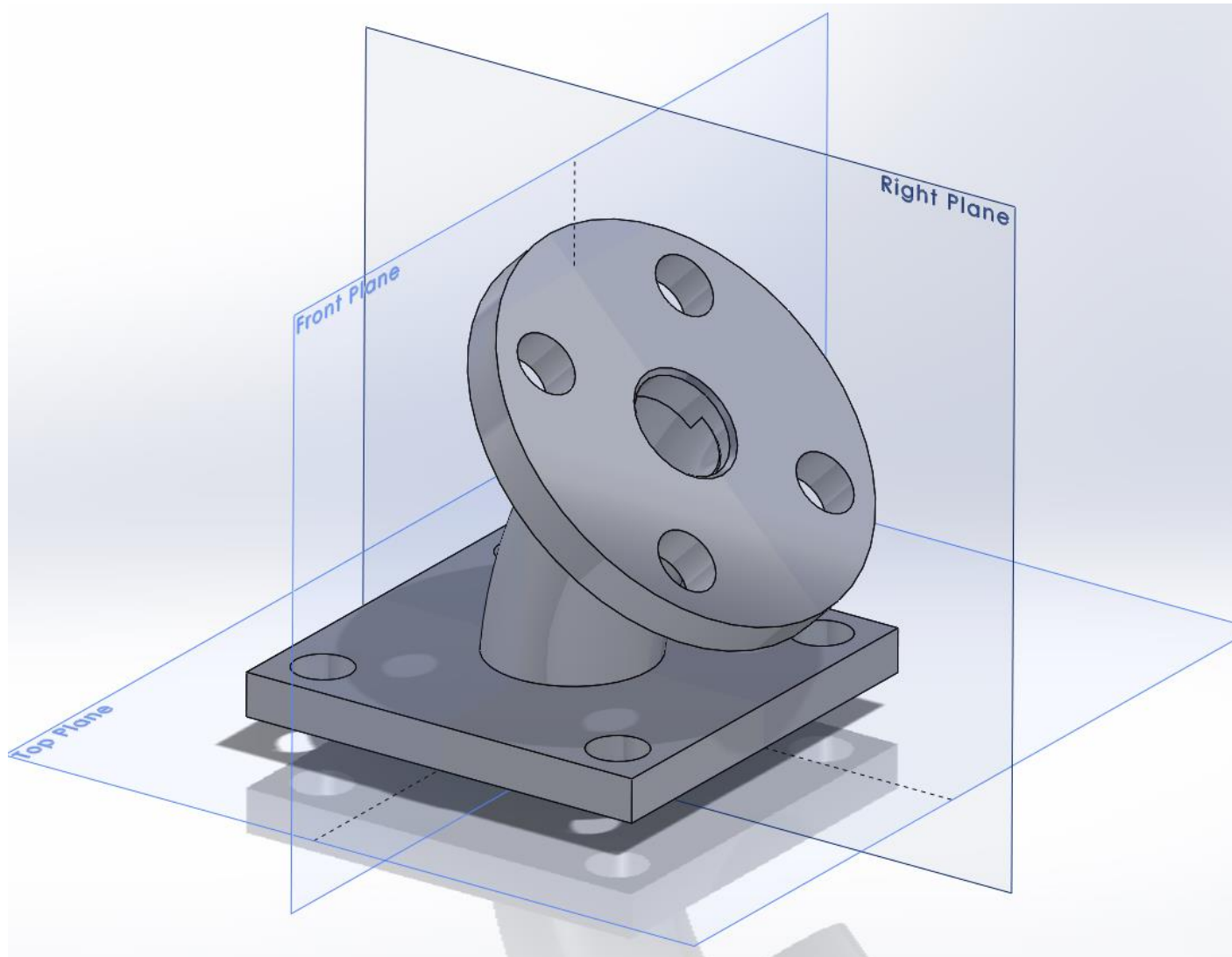


Loft

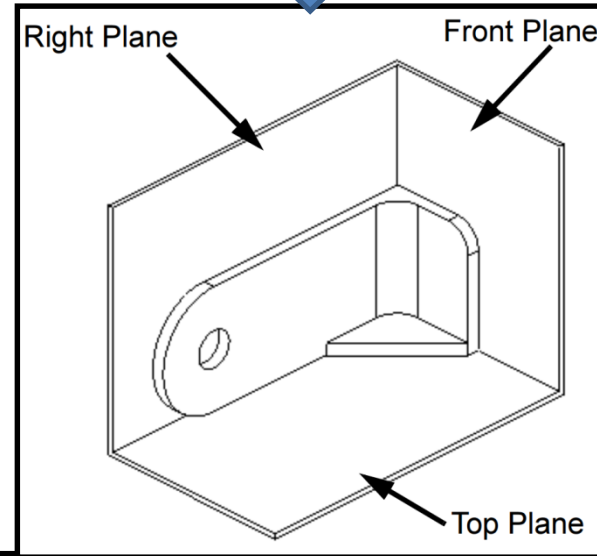
Part	Best Profile Extruded
	
	

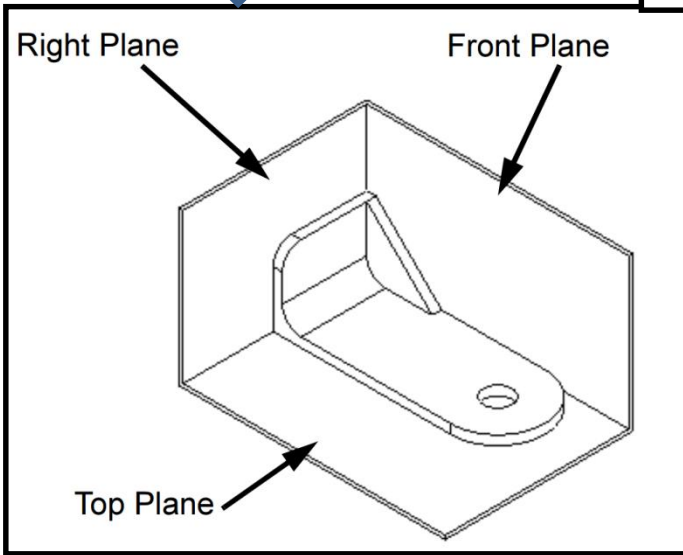
Front Plane-Top Plane -Right Plane



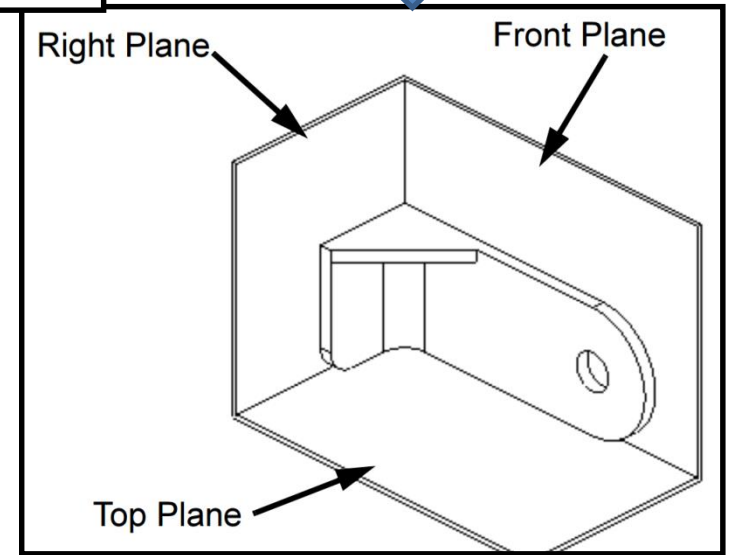
Best profile is in contact with the Right plane.



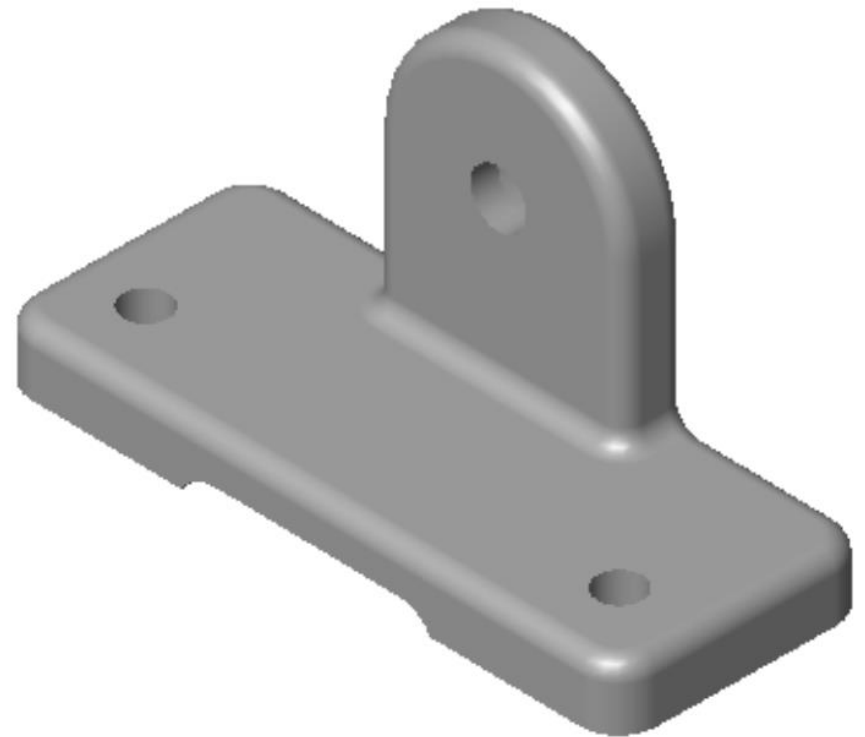
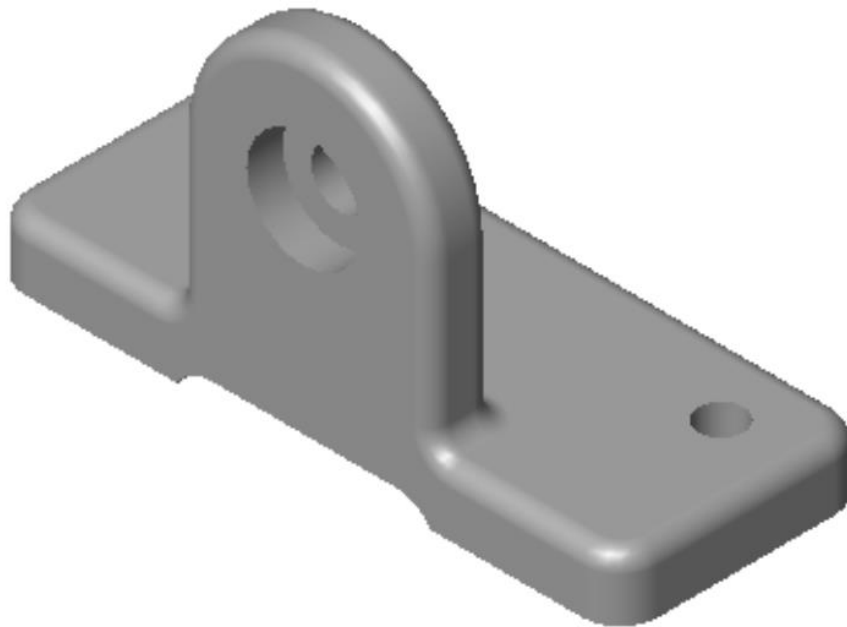
Best profile is in contact with the Top plane.



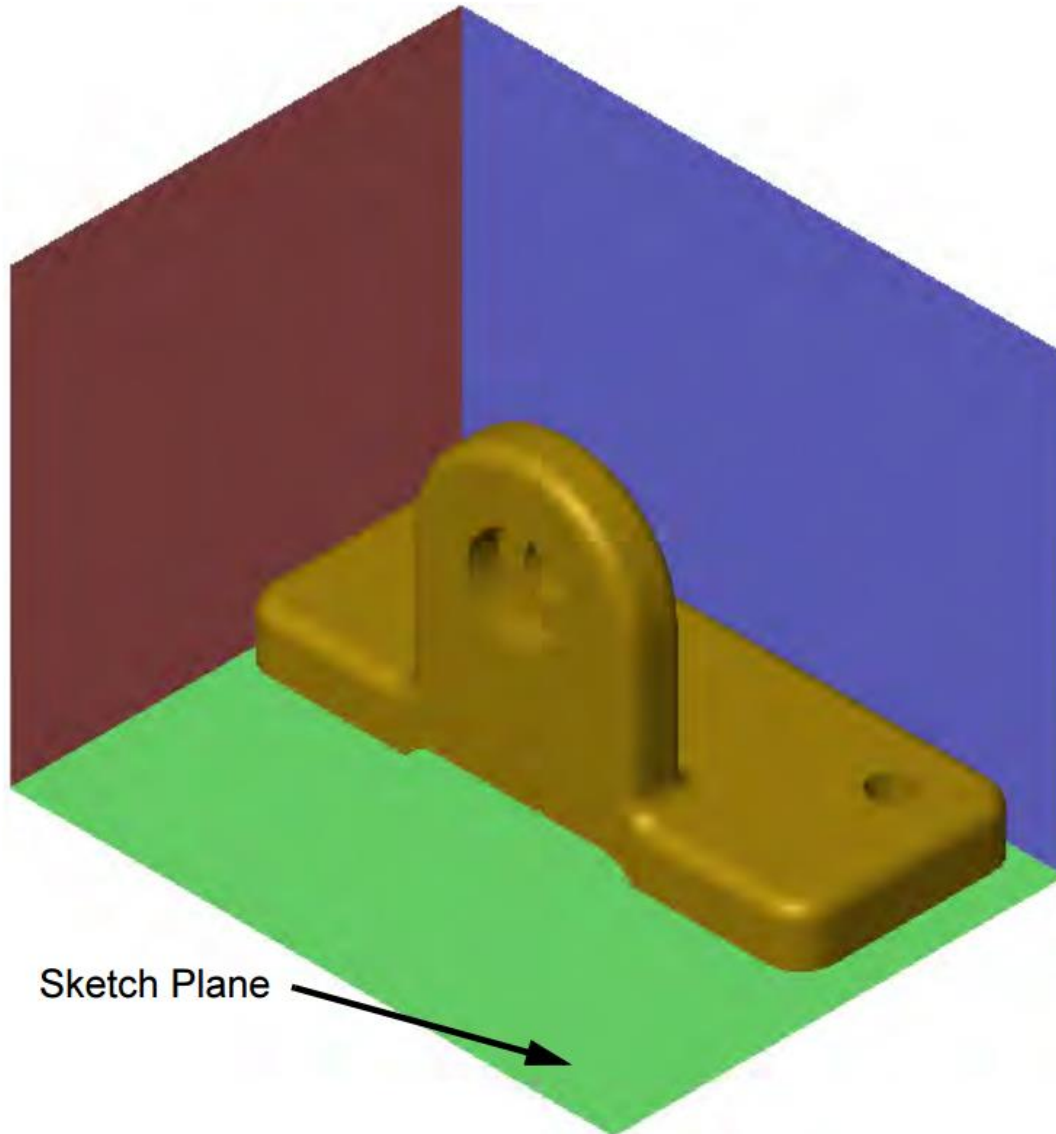
Best profile is in contact with the Front plane.

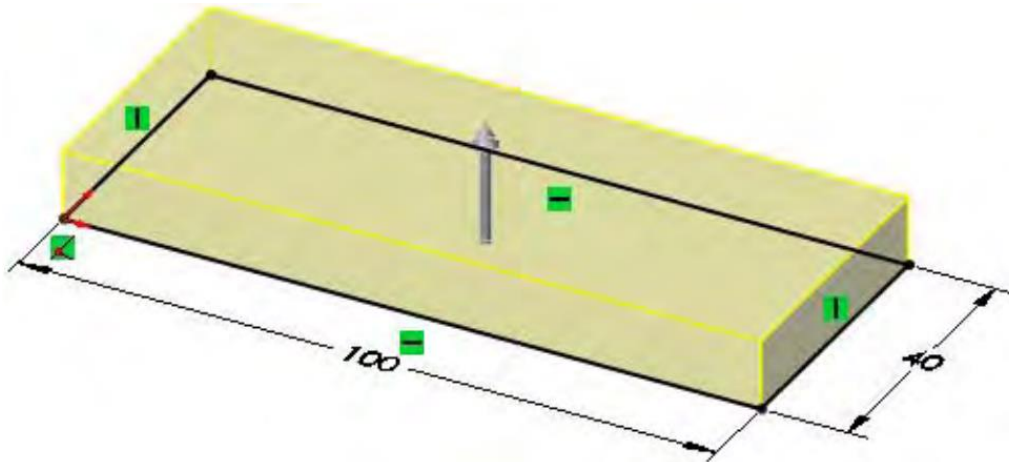
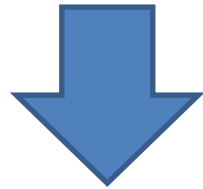
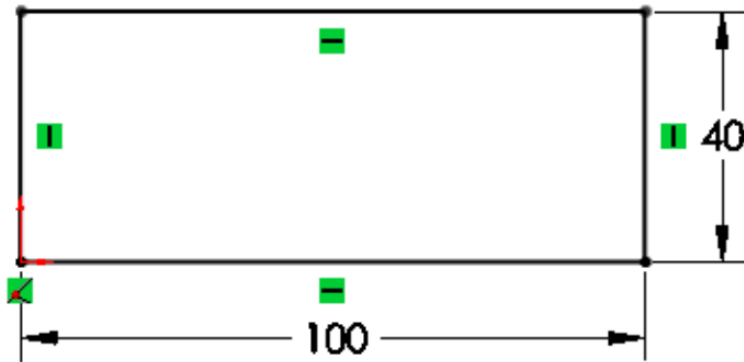


Bài tập tạo khối cơ bản: Trong bài tập này chúng ta sẽ thảo luận từng bước để tạo ra 1 chi tiết cơ bản như hình sau đây

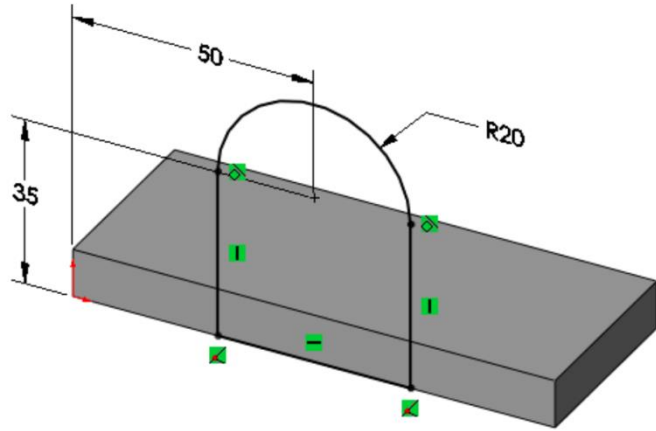


*Tham khảo: SolidWork Essentials 2011
Lesson 3: Basic Part Modeling (Trang 61)*

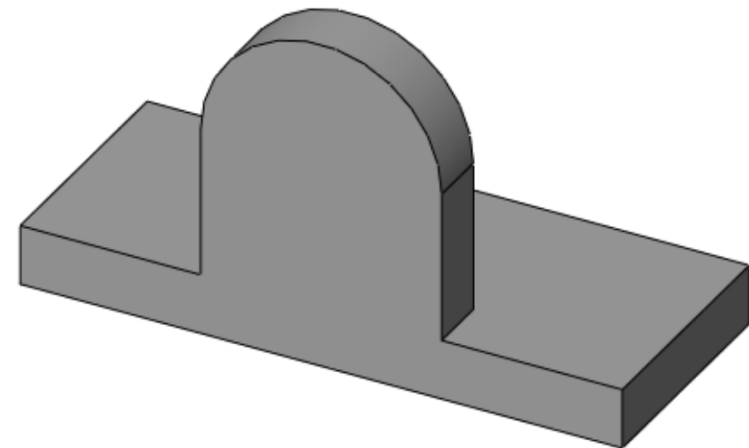
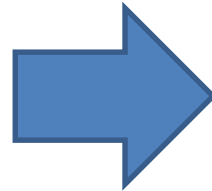
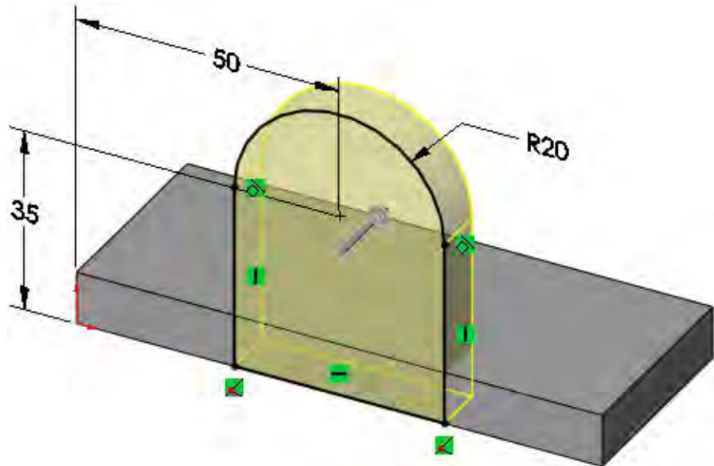


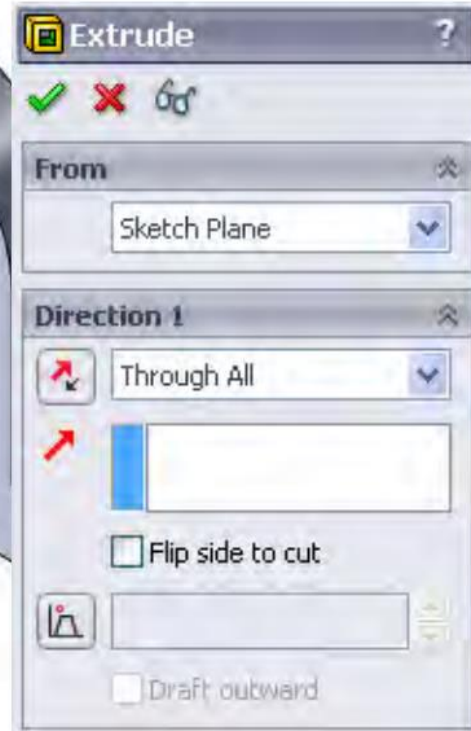
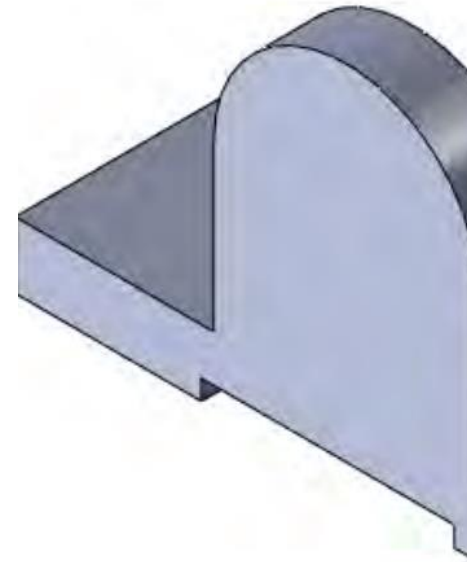
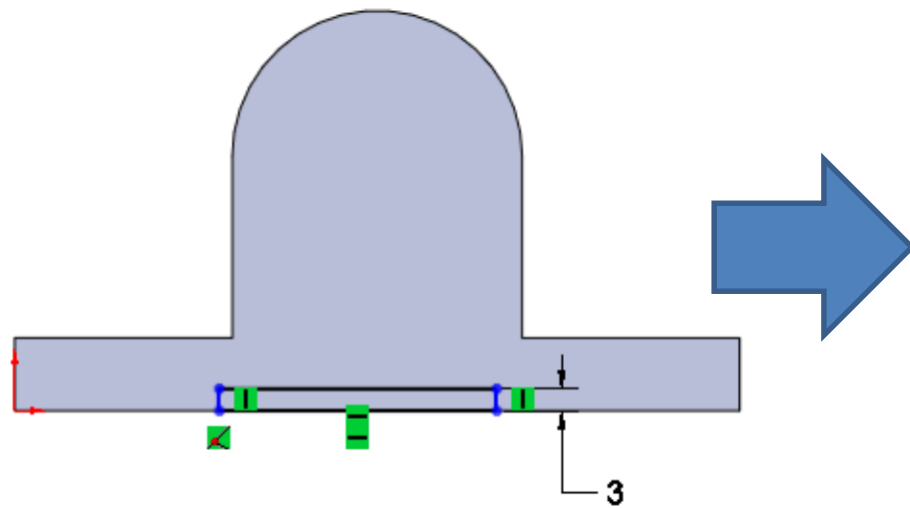


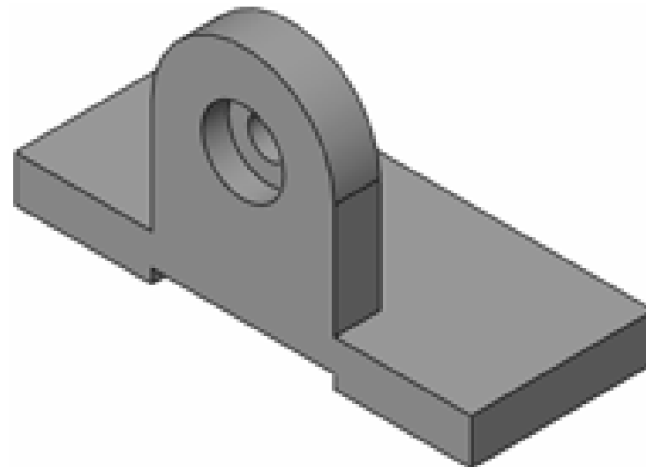
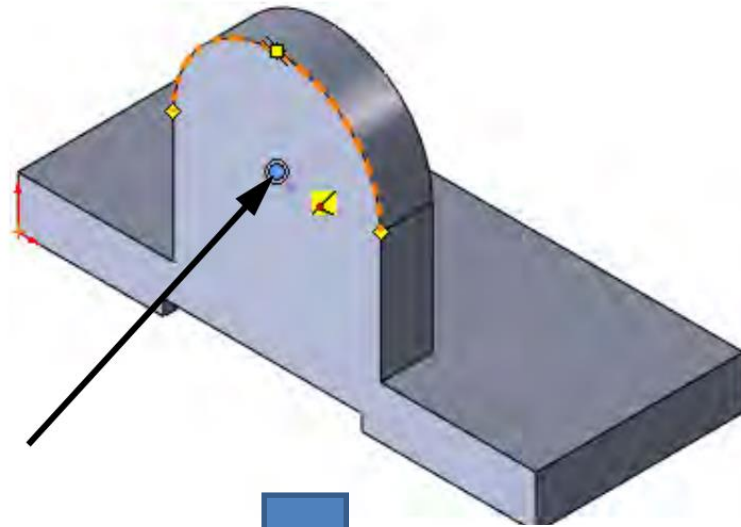
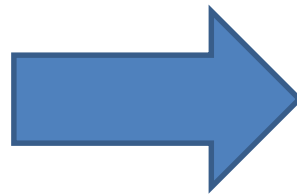
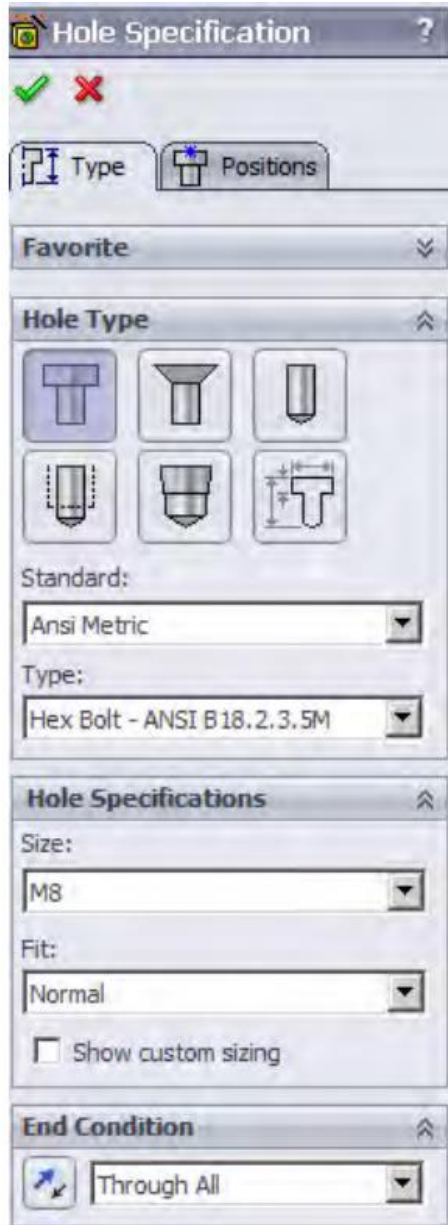
**Click Extrude
and extrude the
rectangle 10mm**

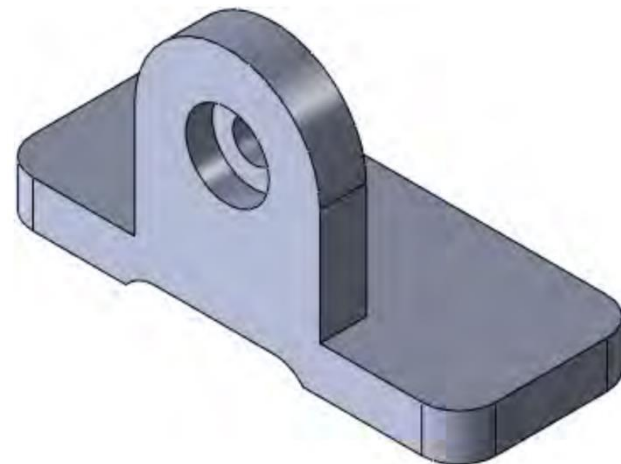
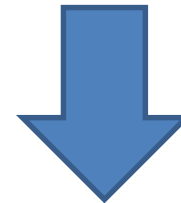
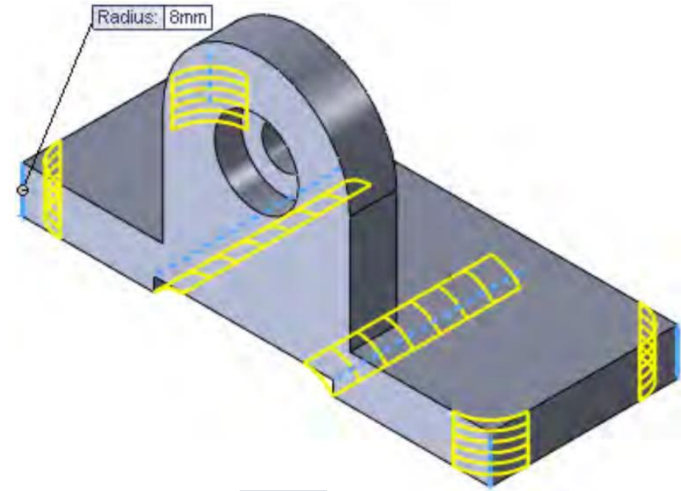
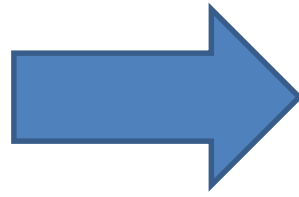
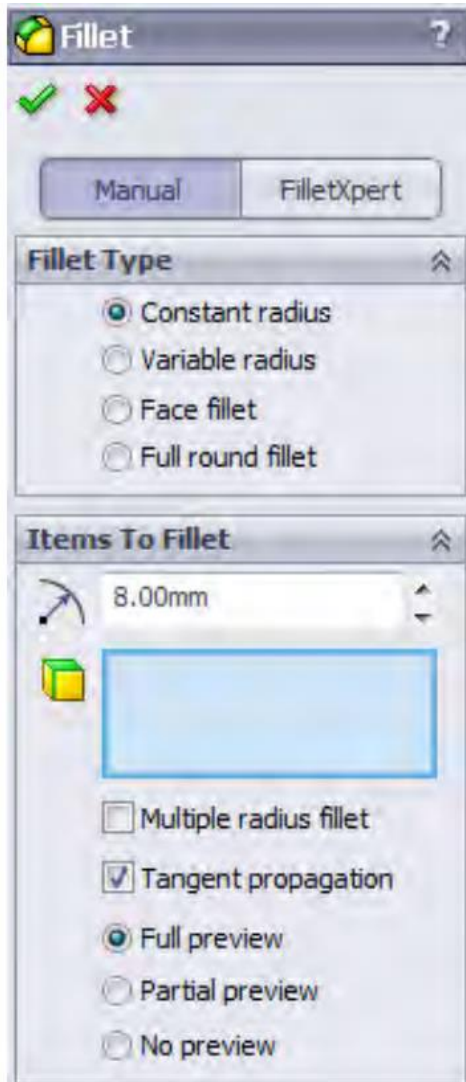


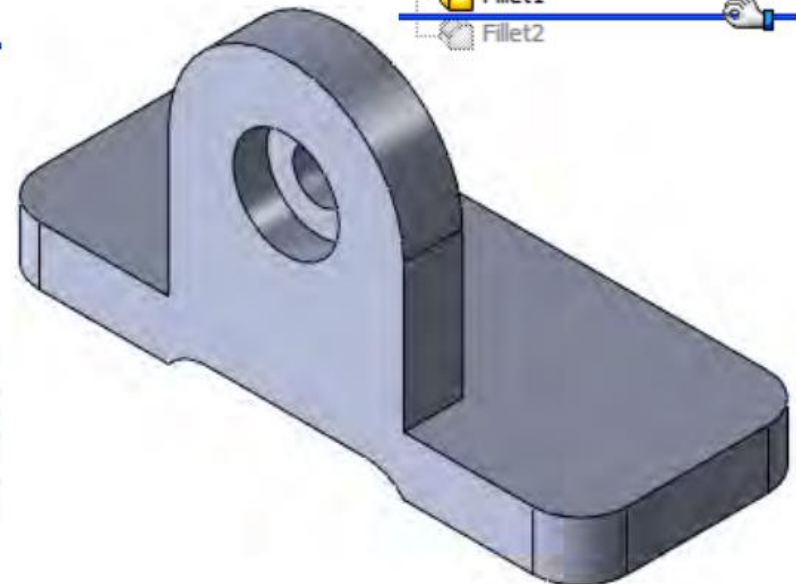
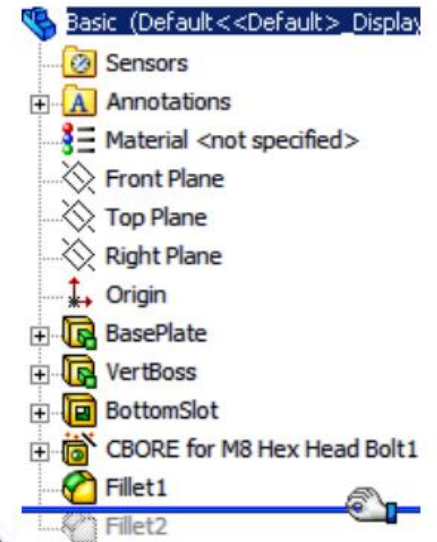
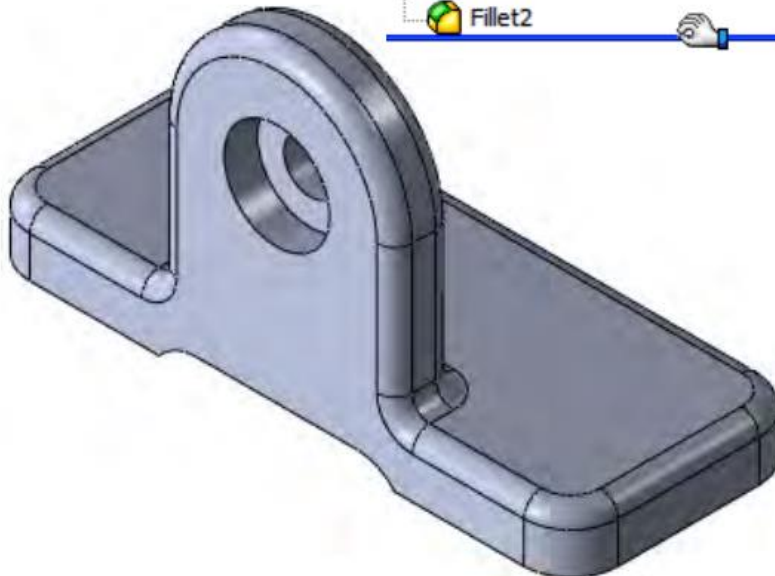
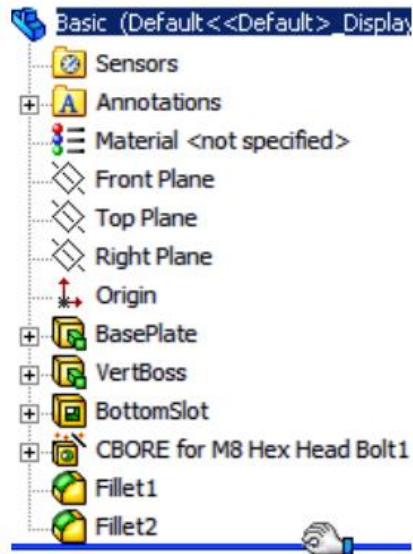
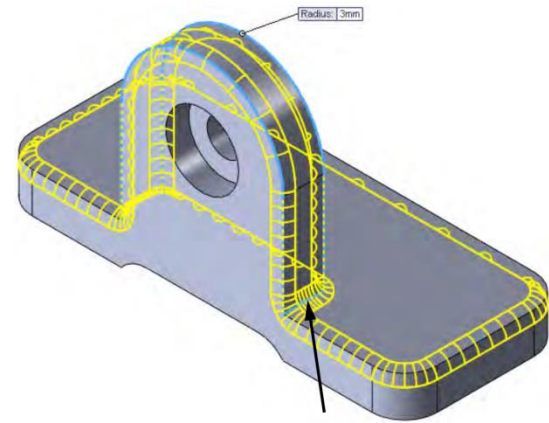
**Extrude and set the
Depth to 10mm**

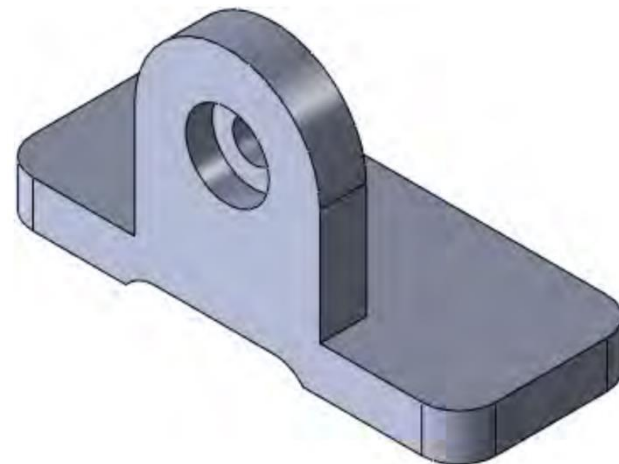
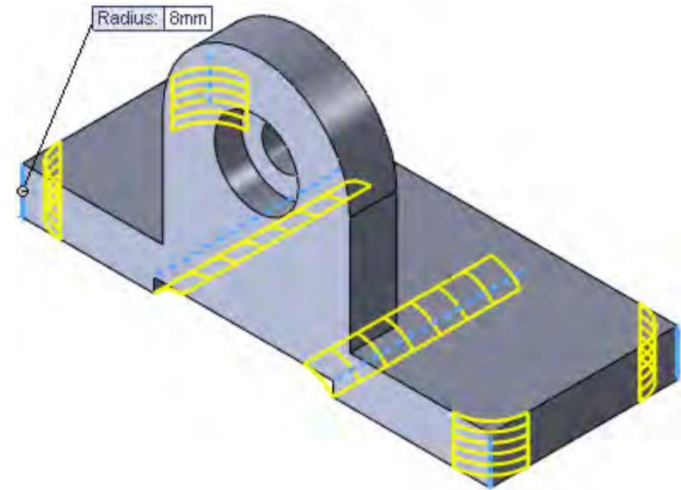
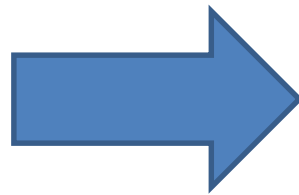
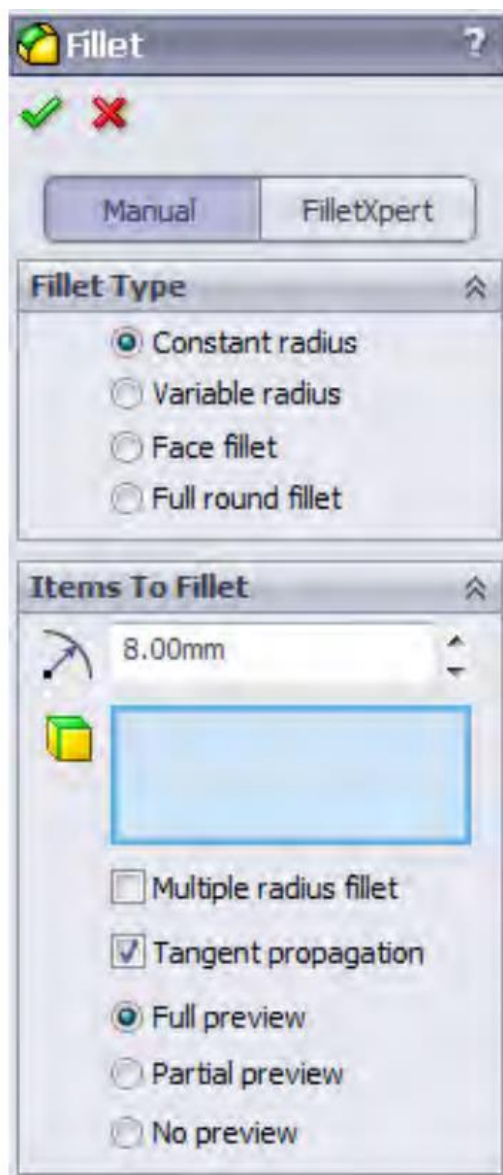


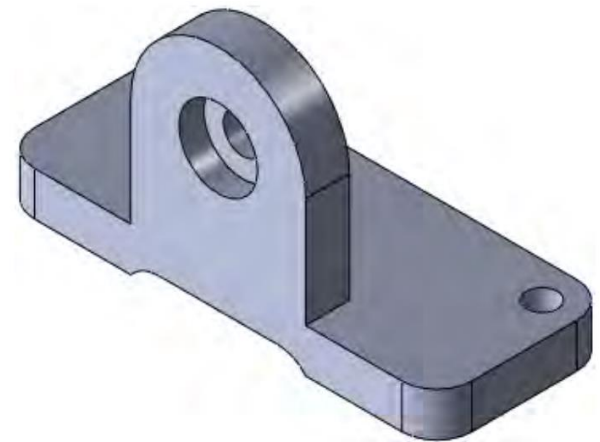
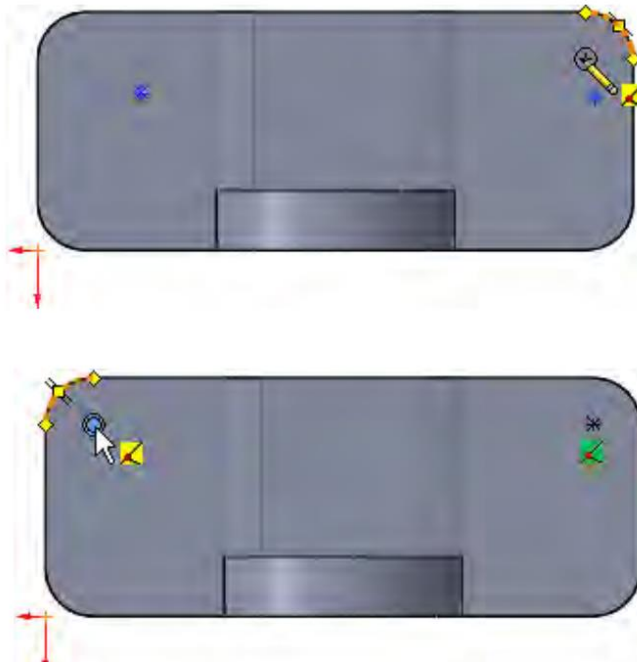
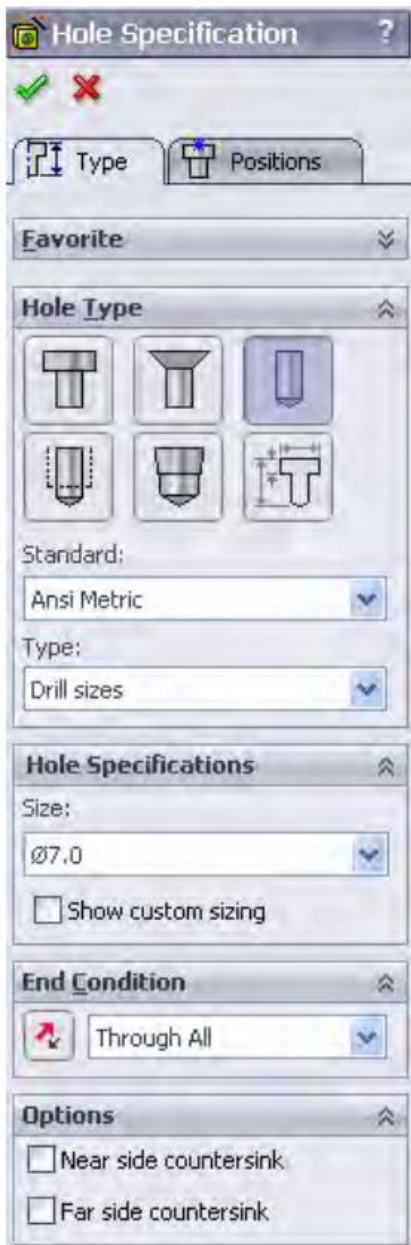


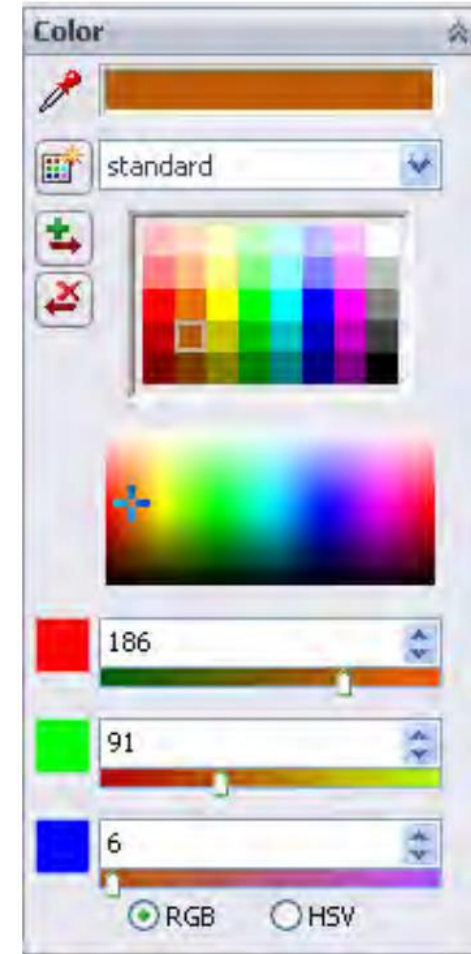
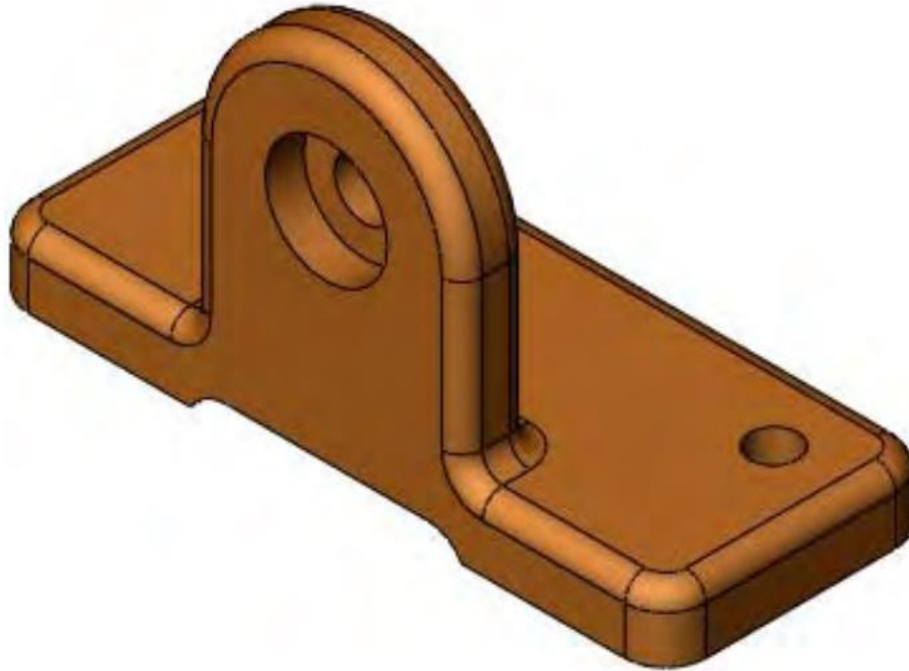


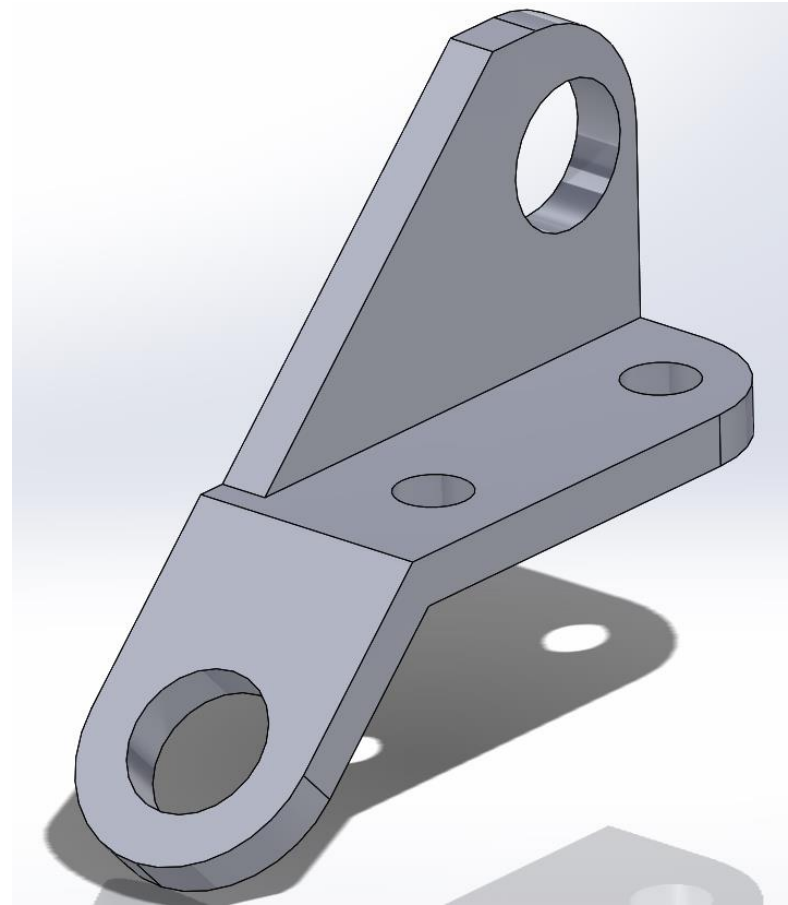
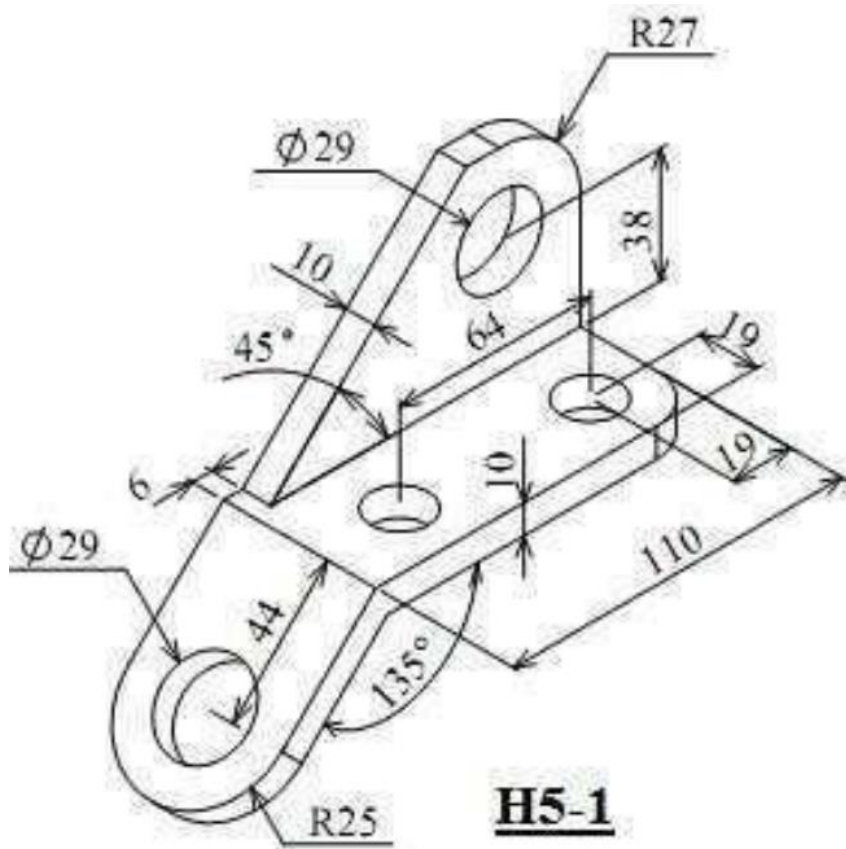


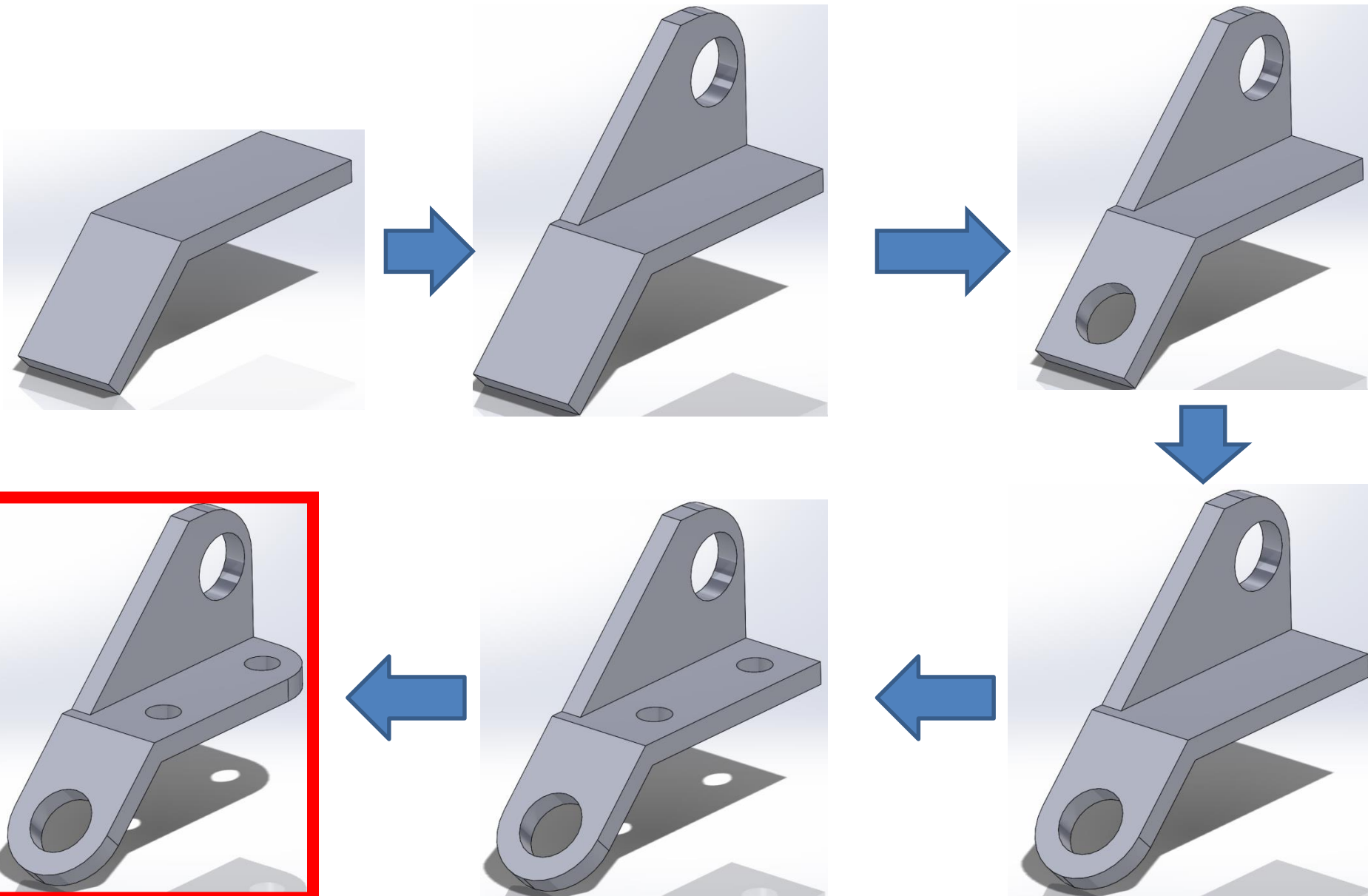


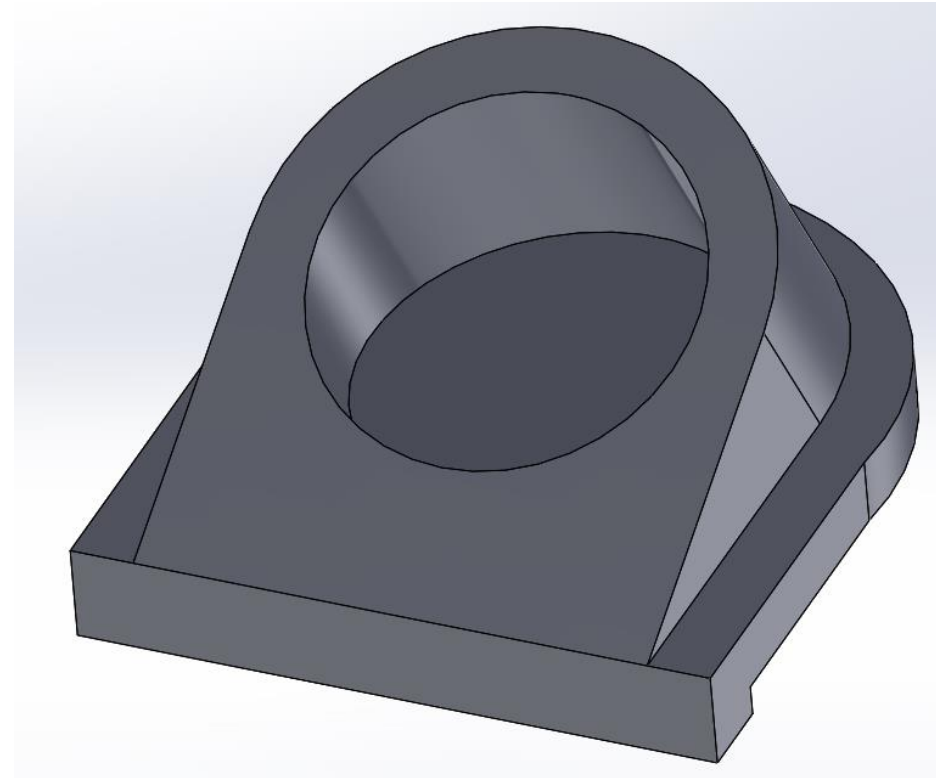
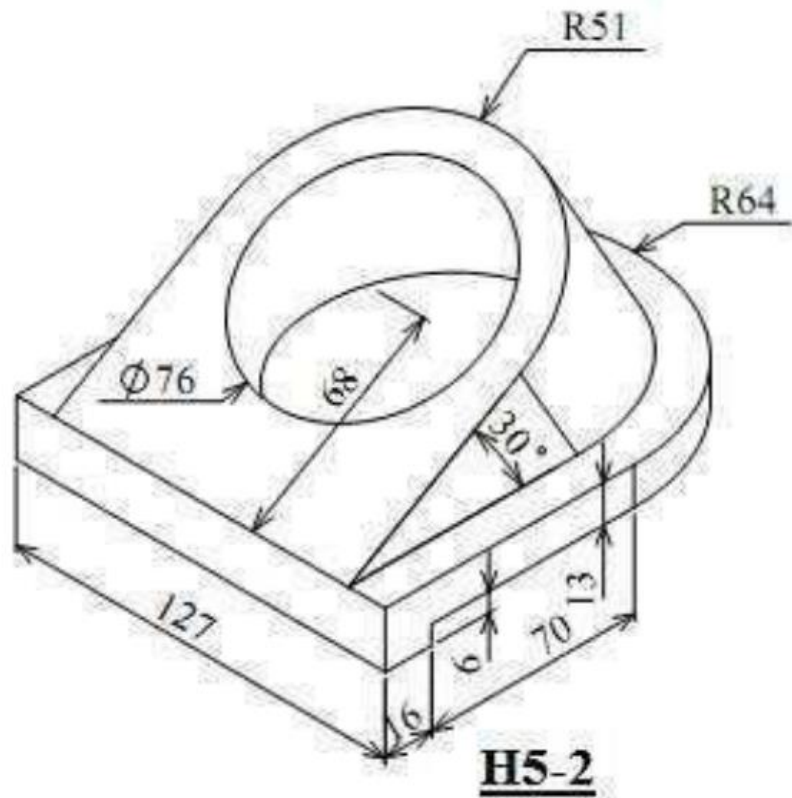


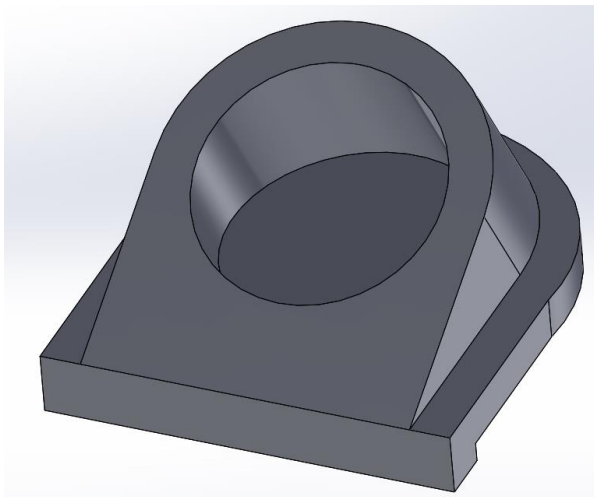
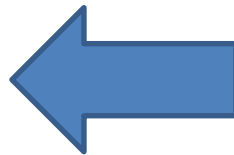
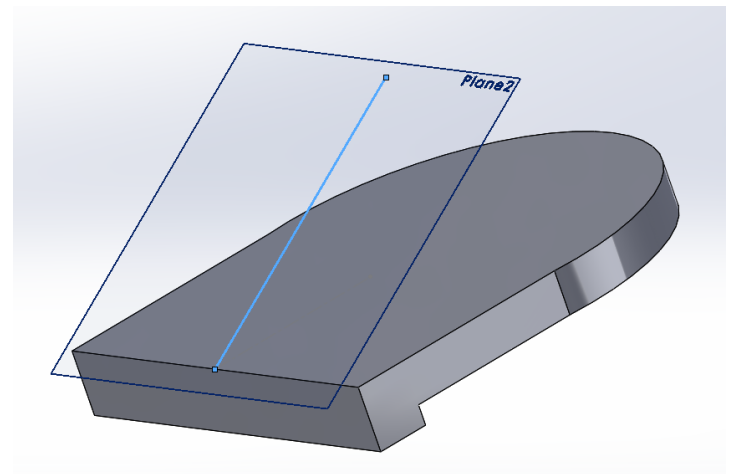
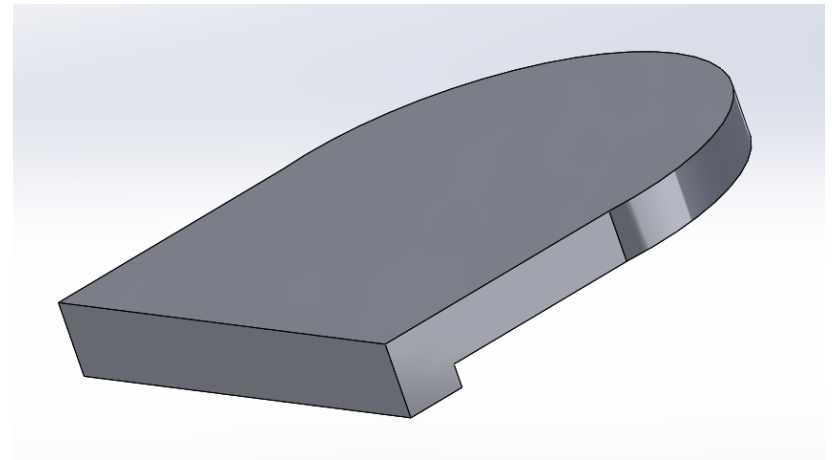
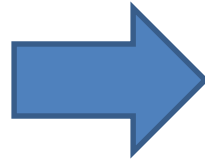
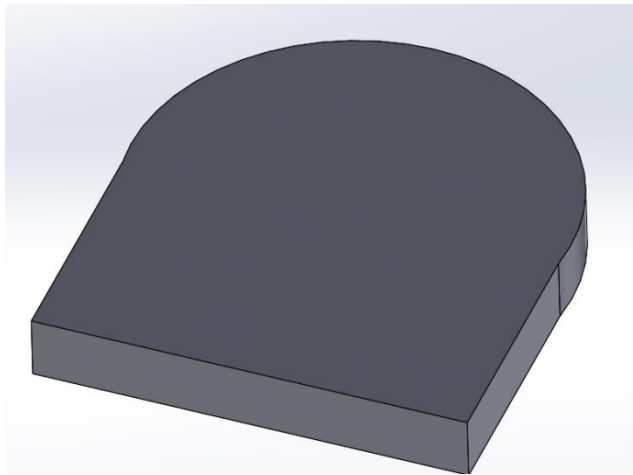


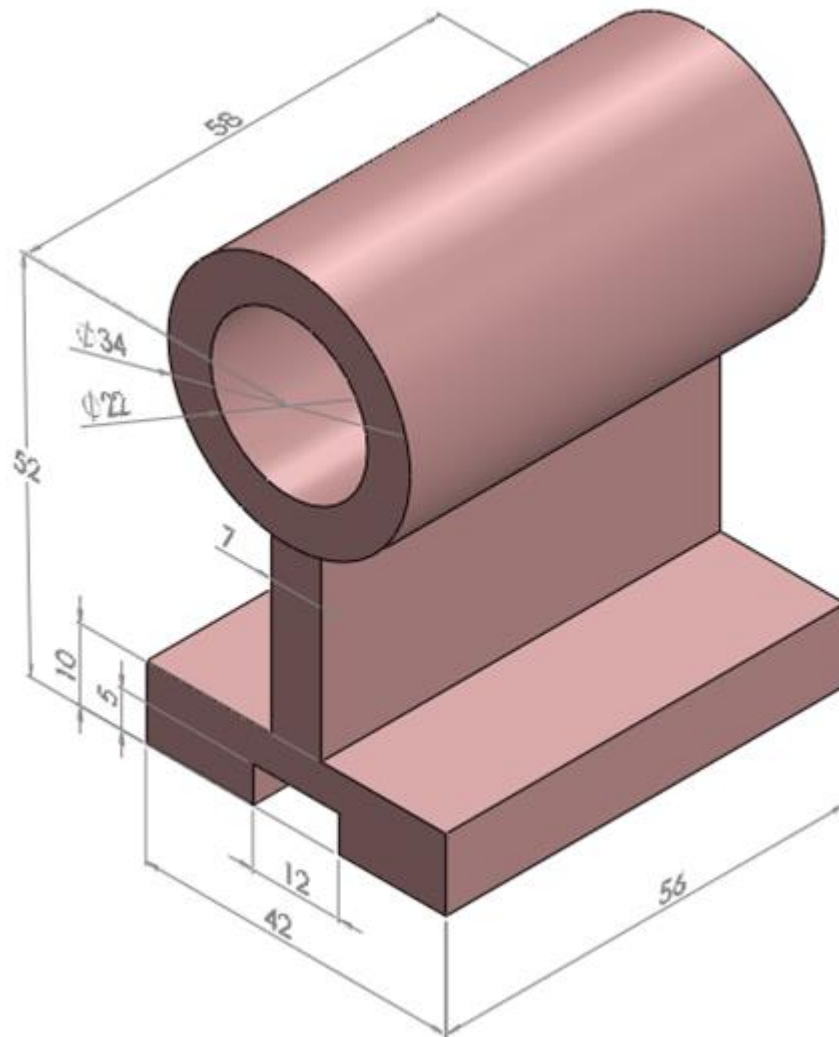


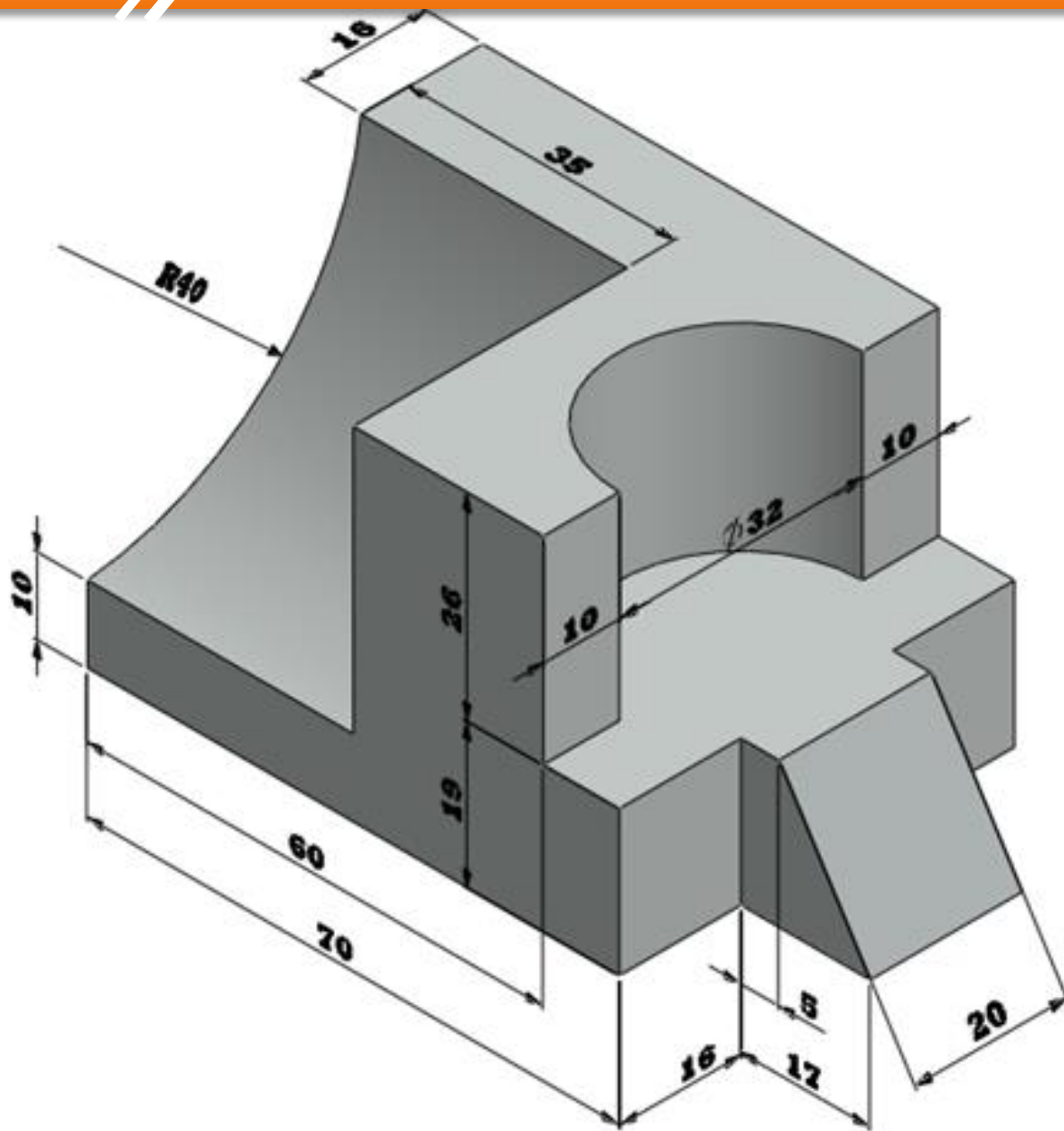


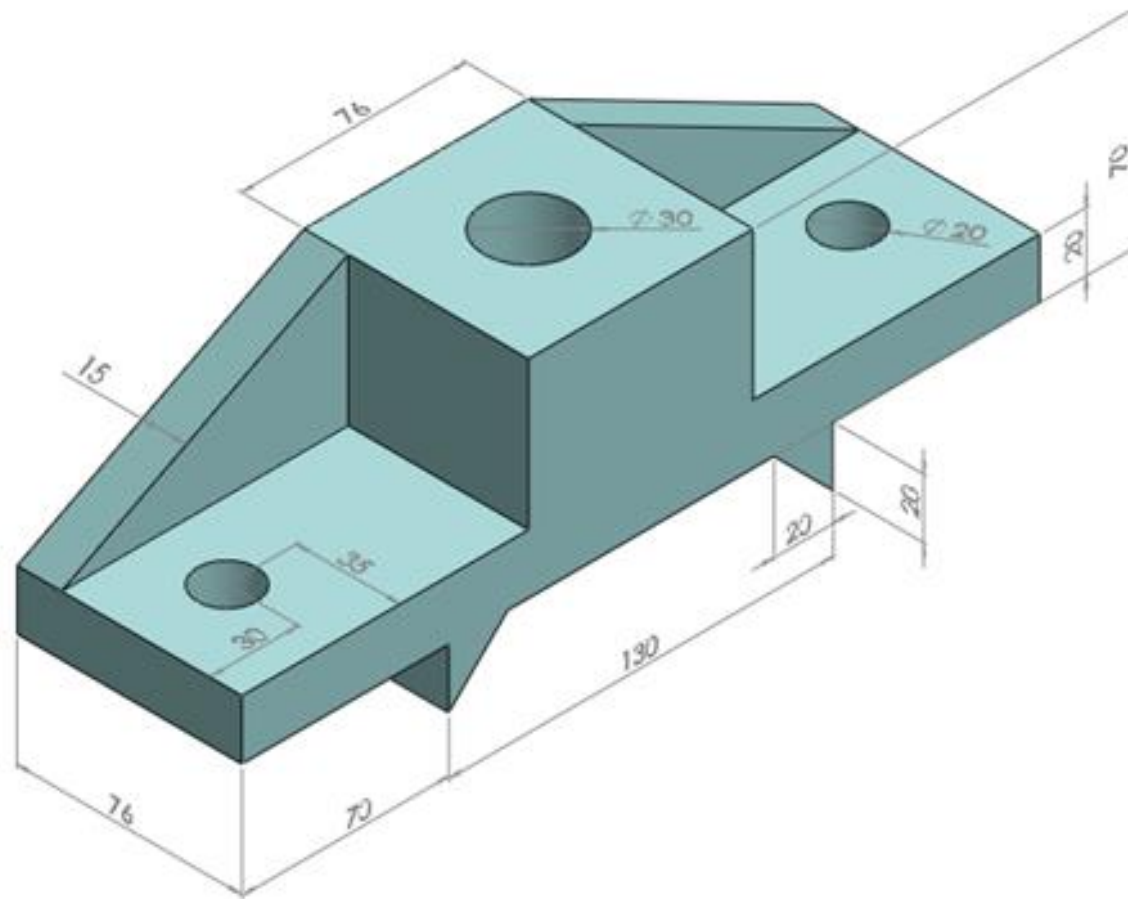


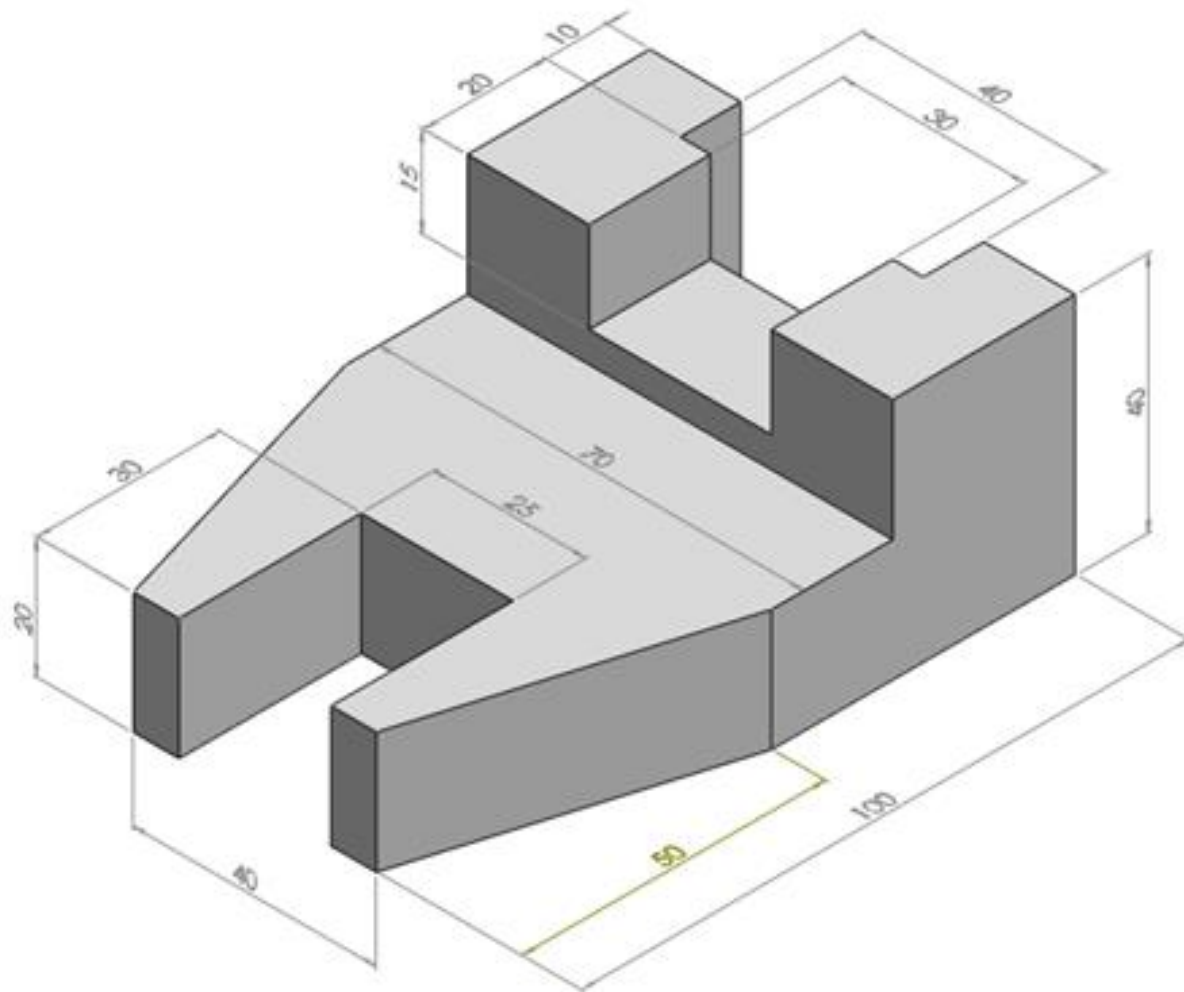


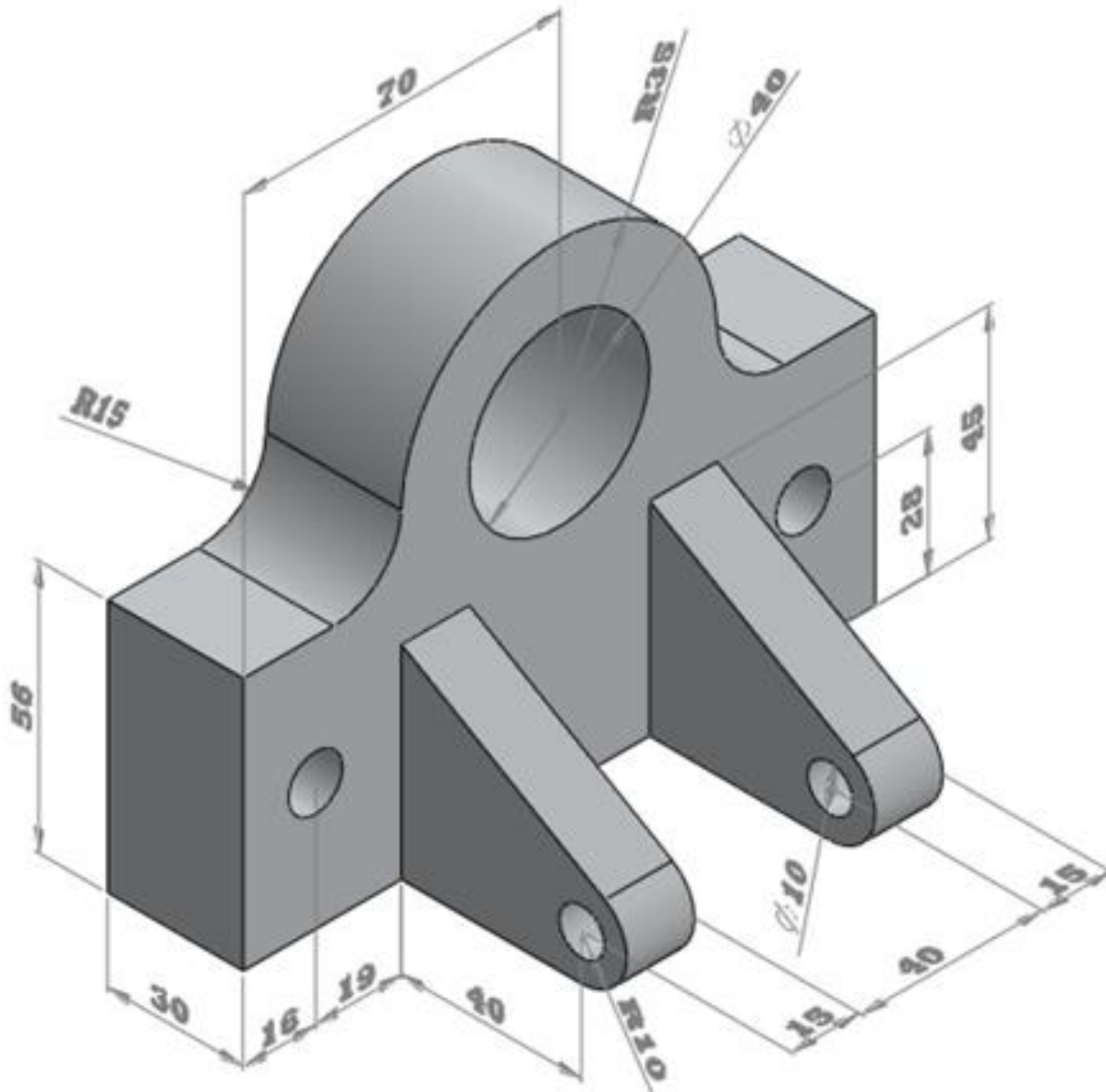


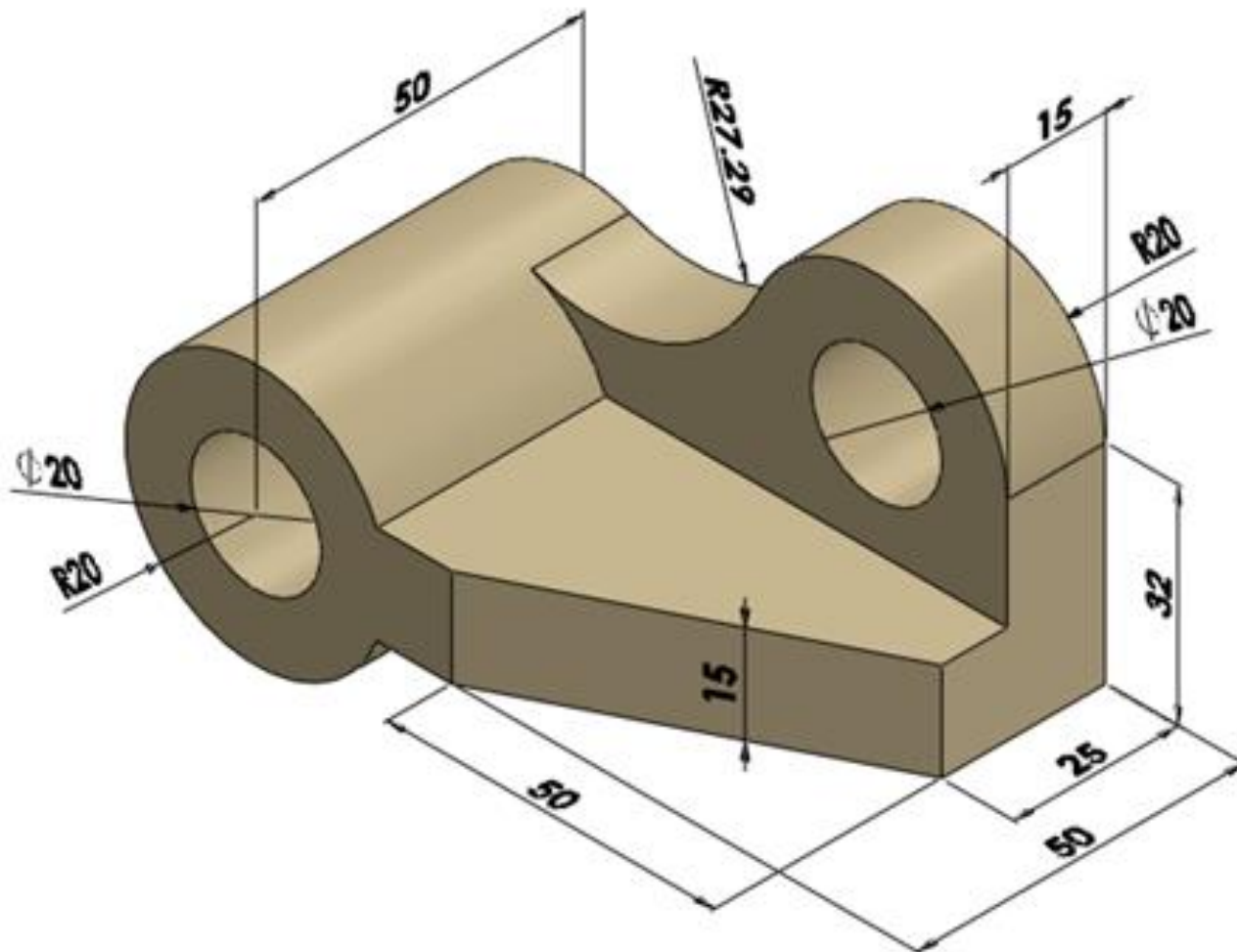


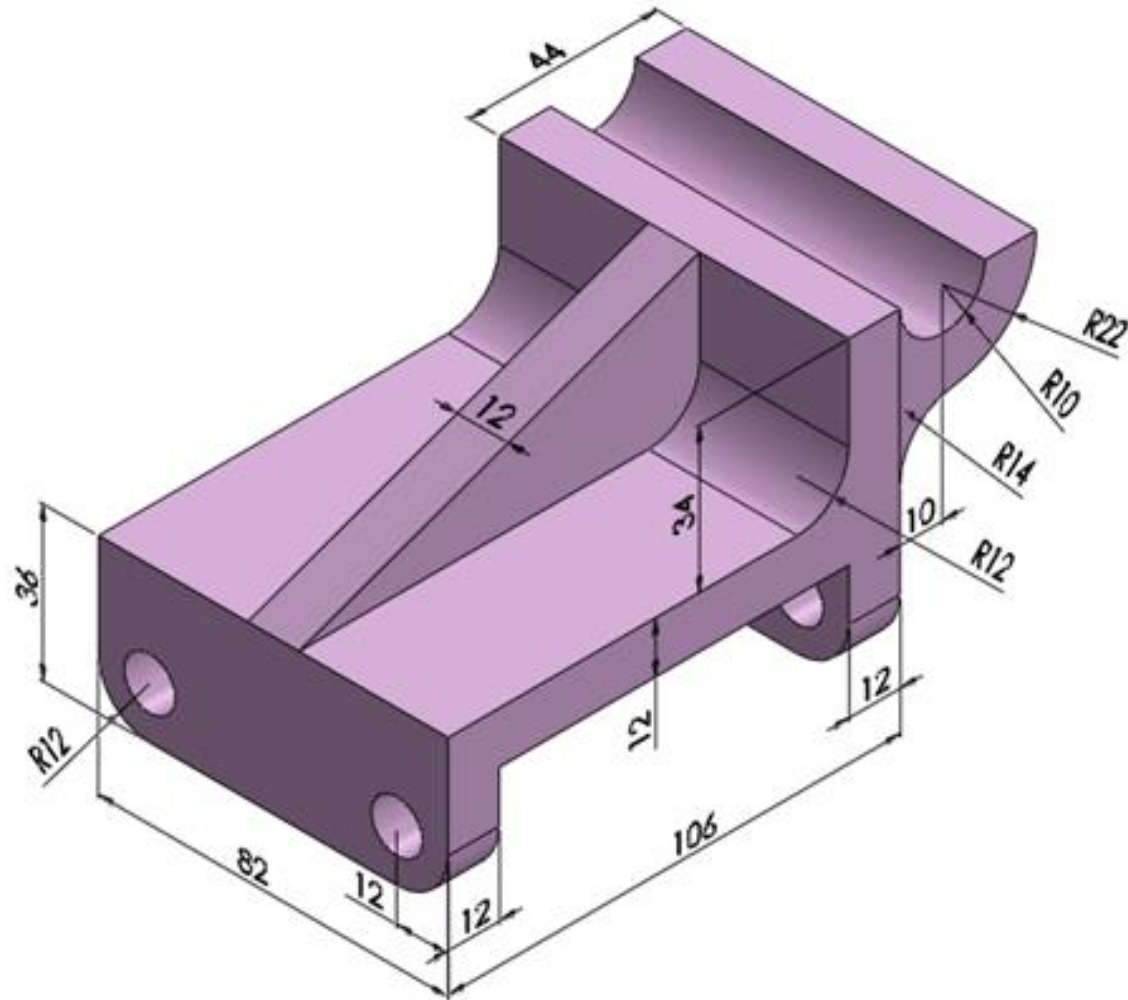


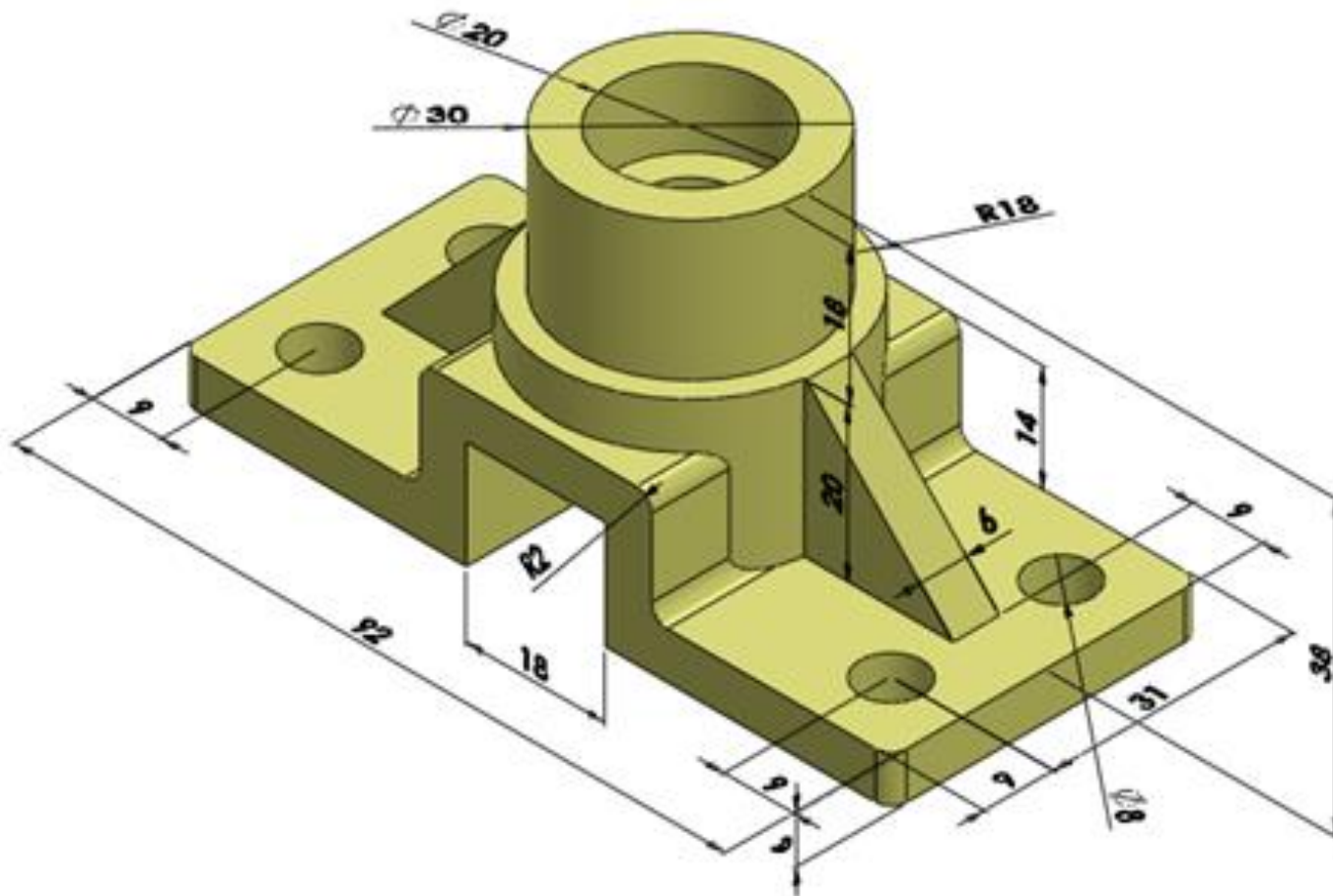


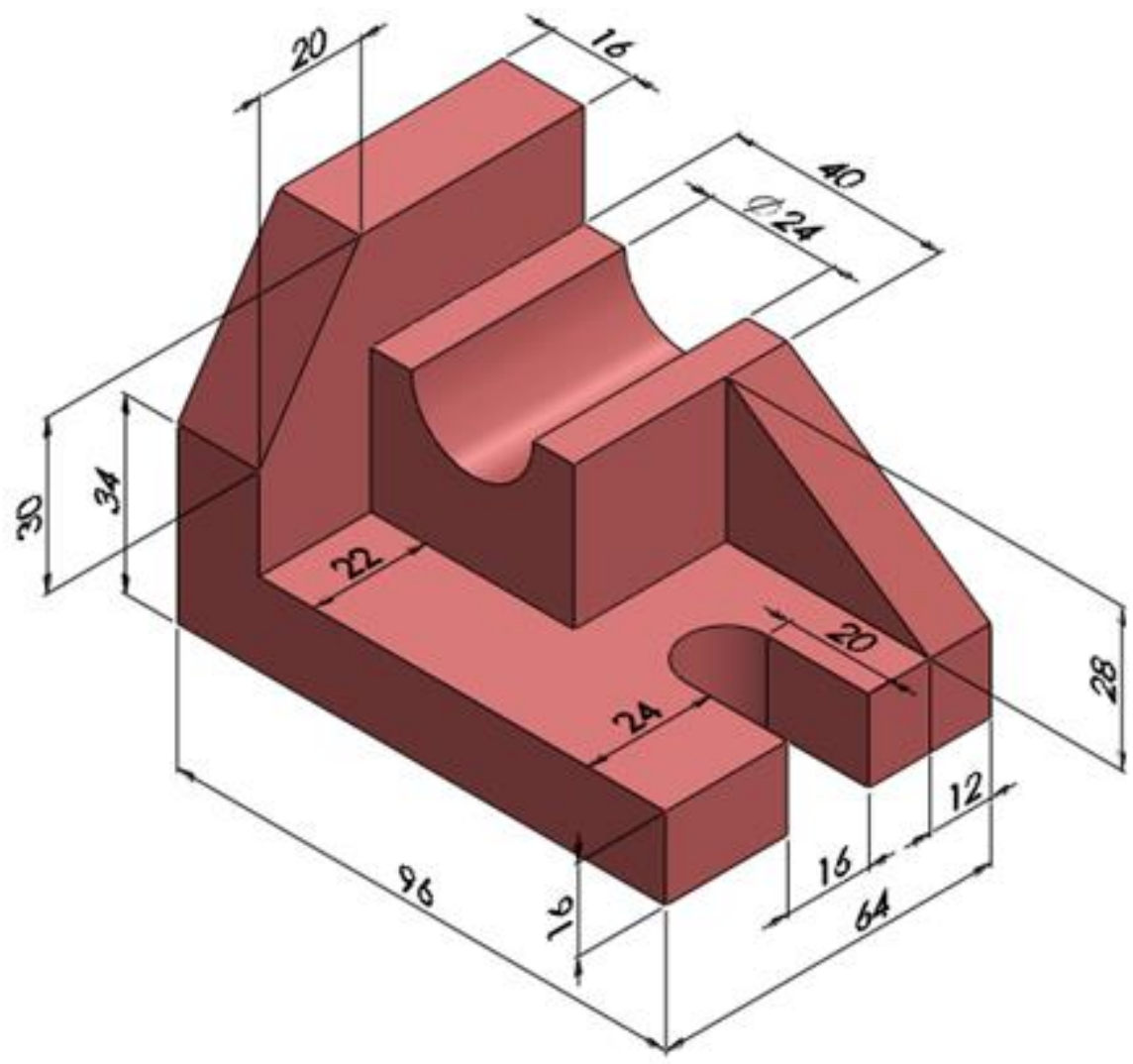


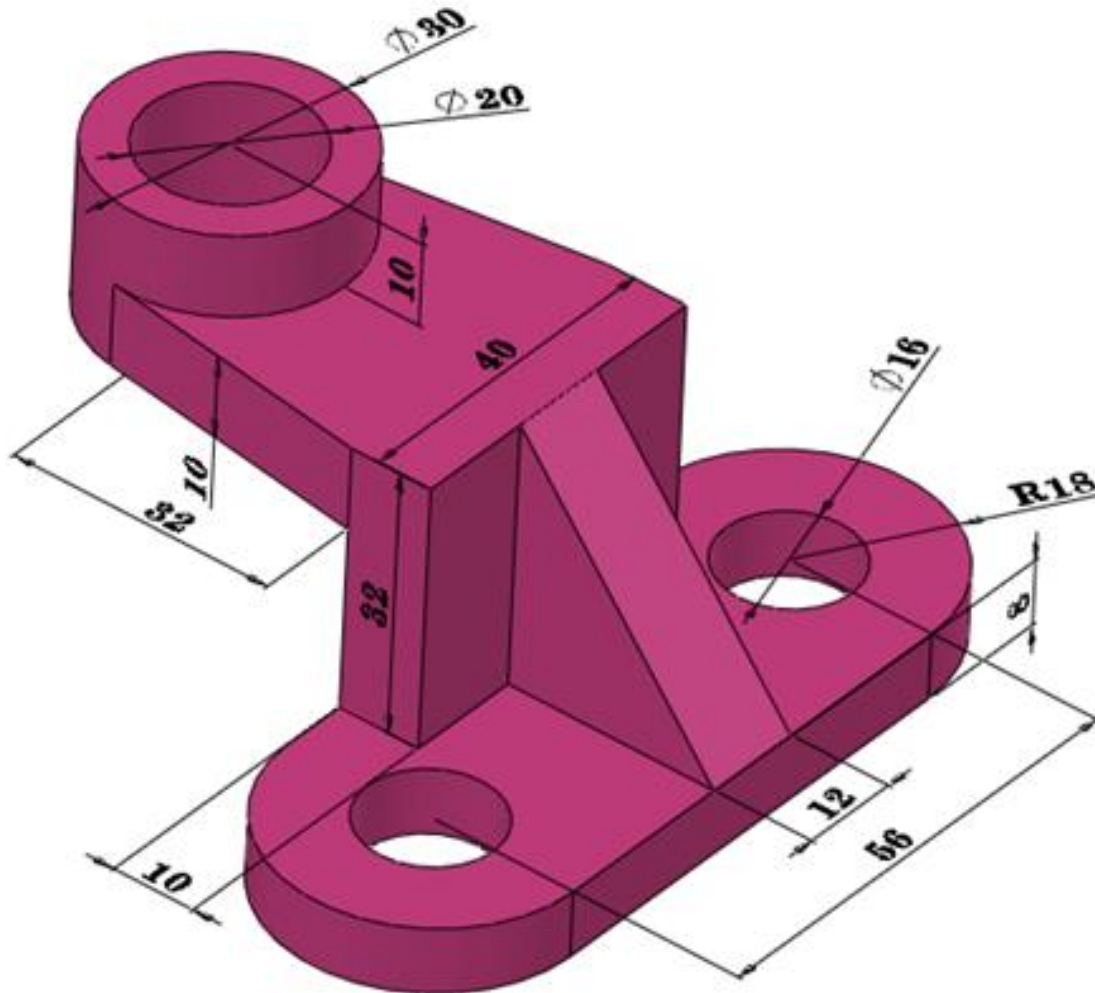


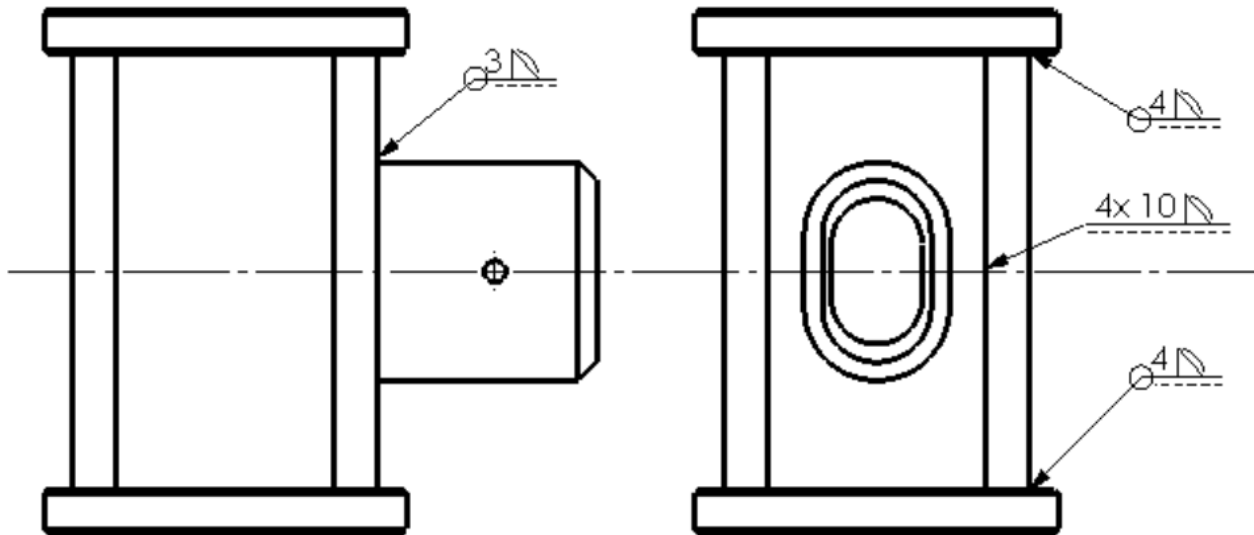
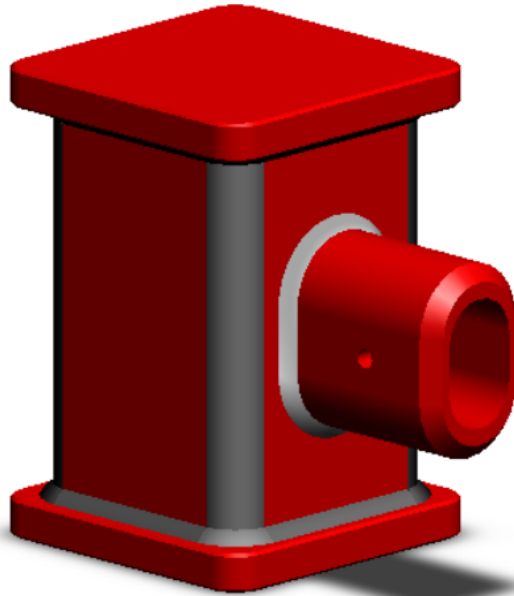


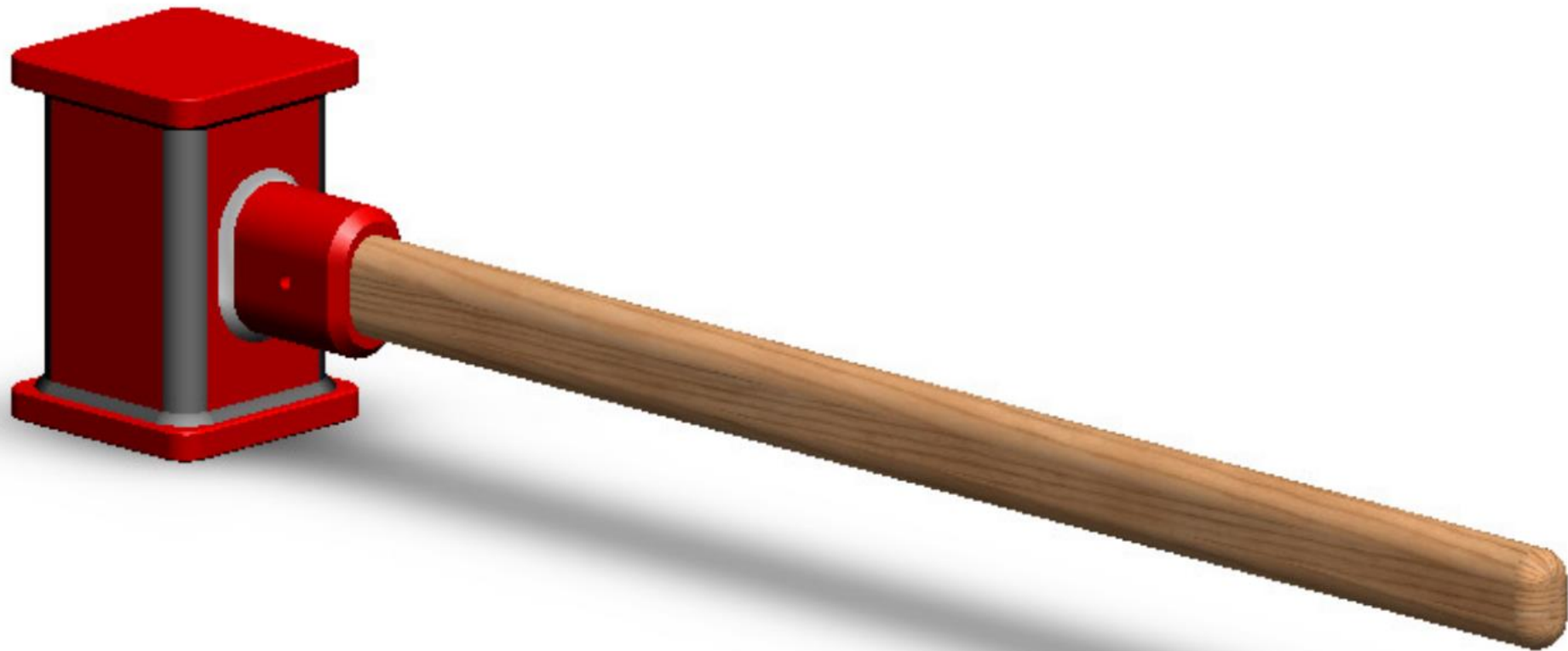


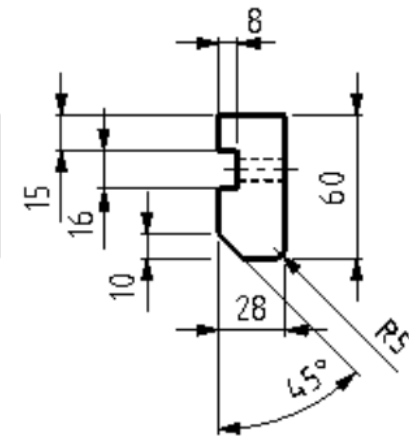
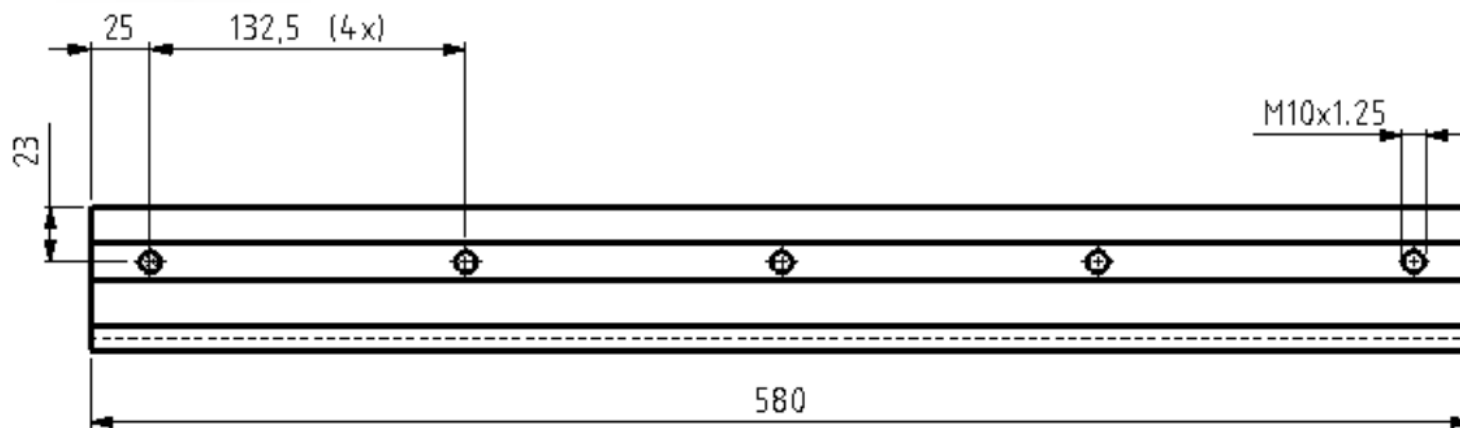
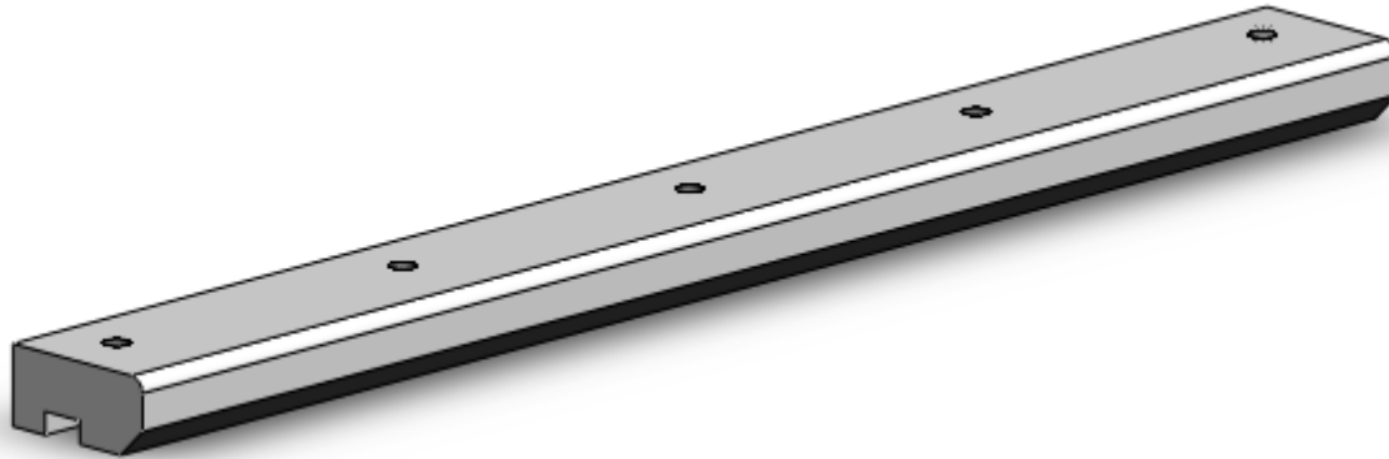


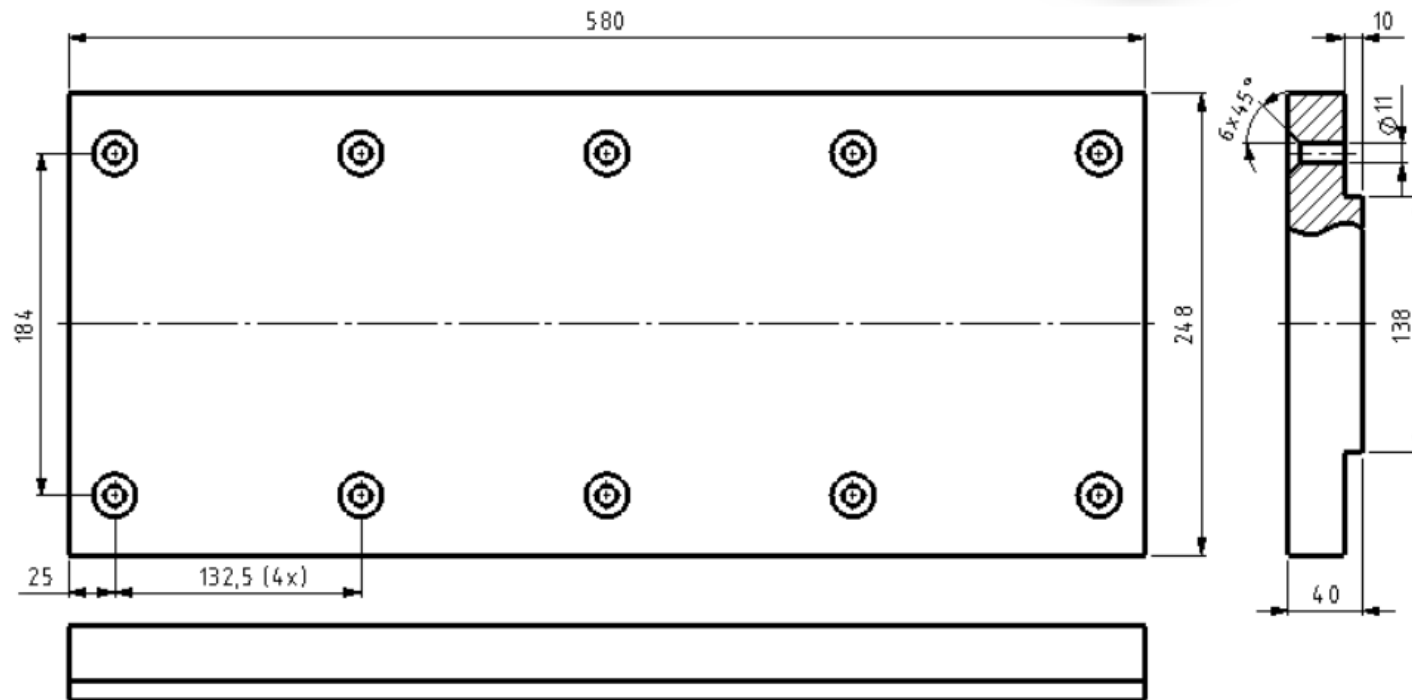
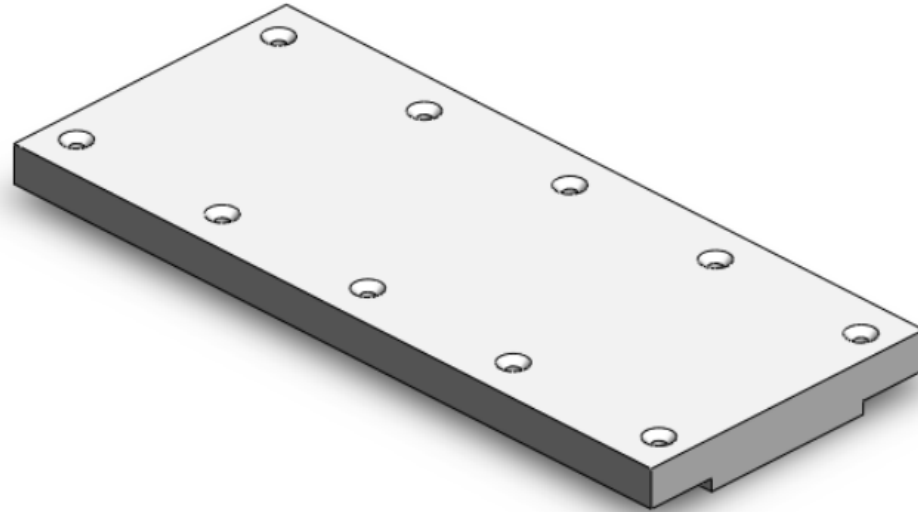




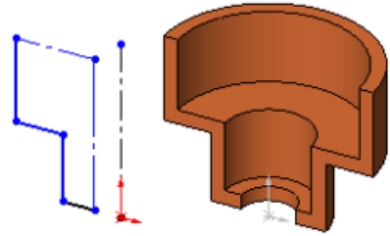
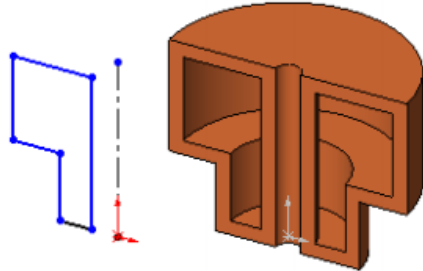
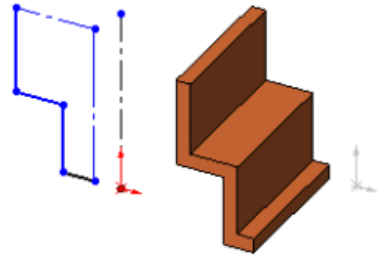
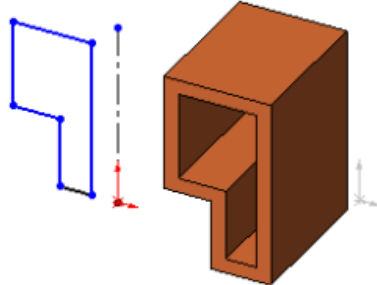


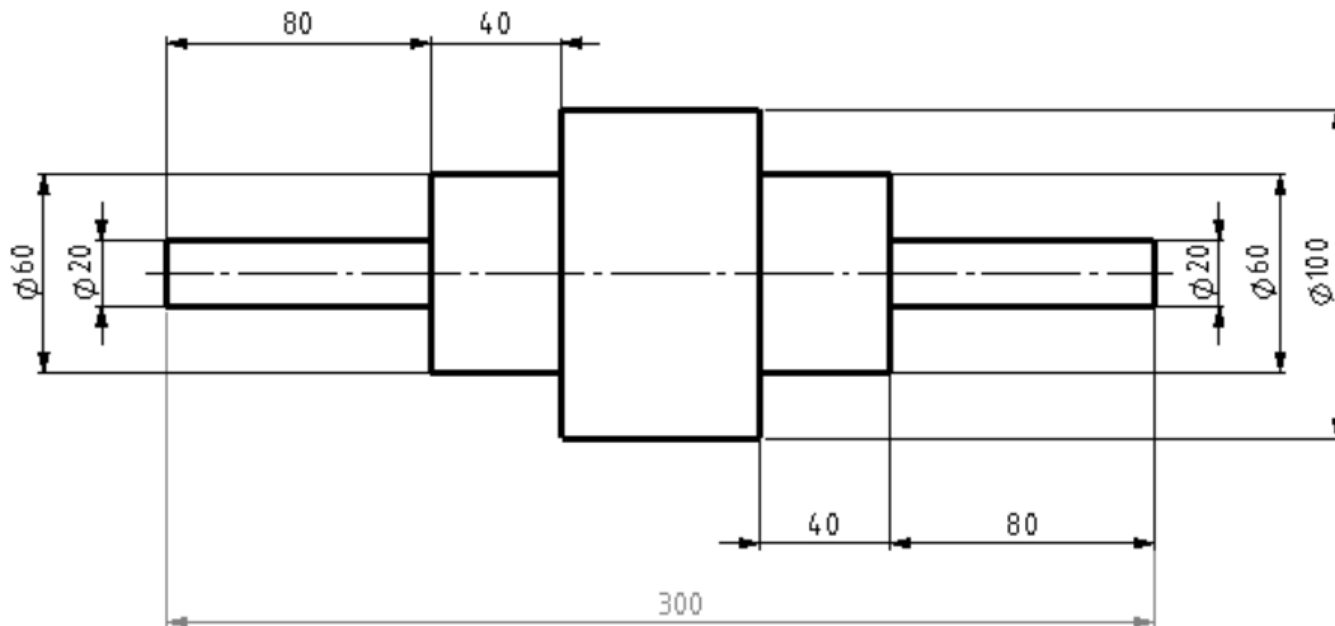
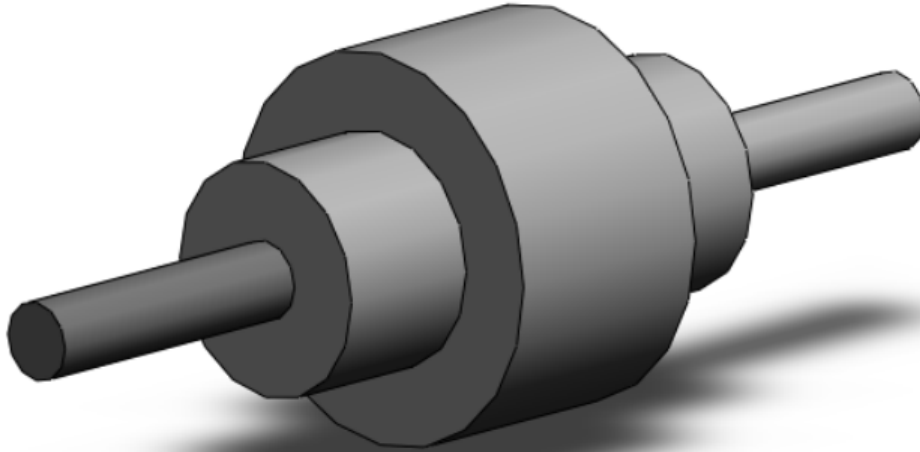


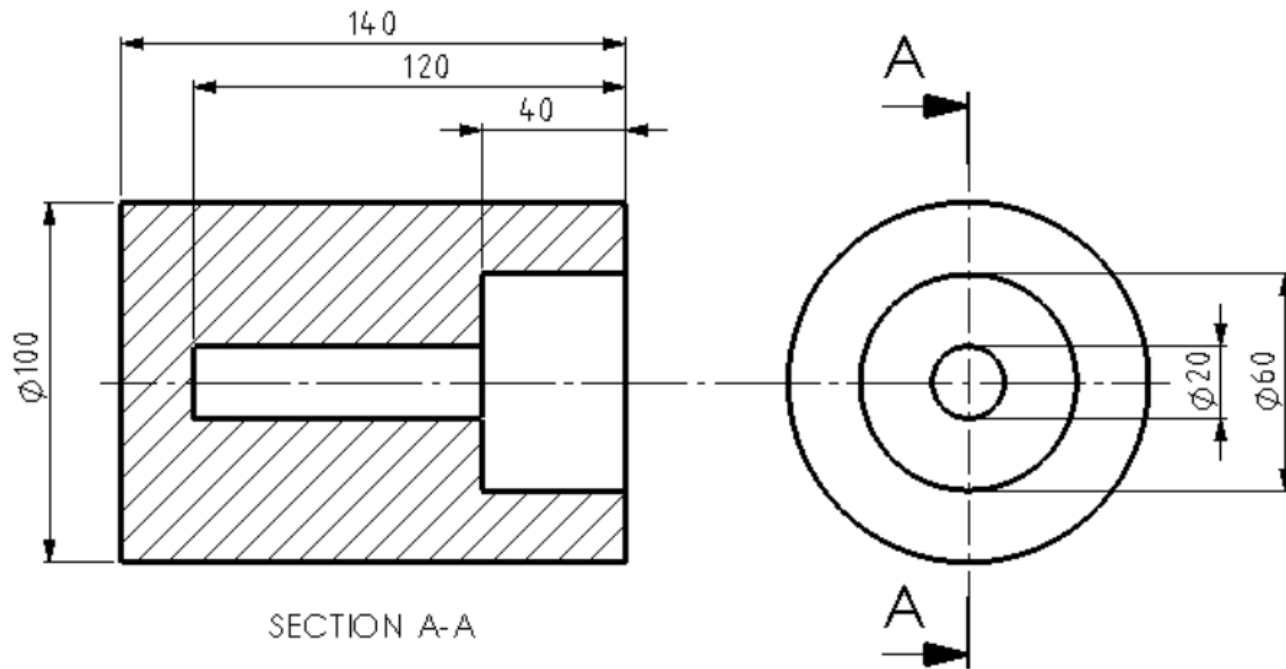
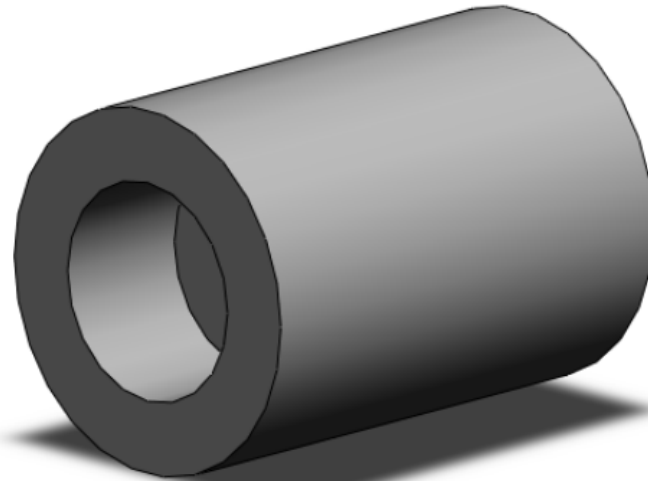


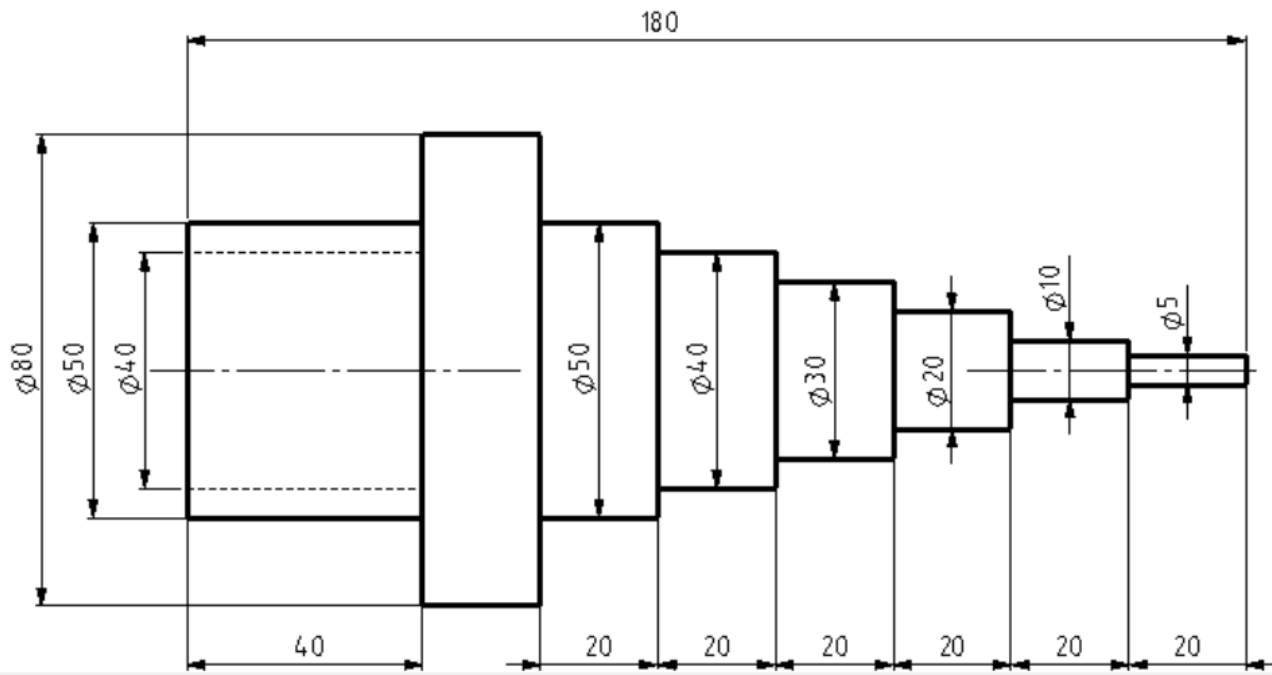
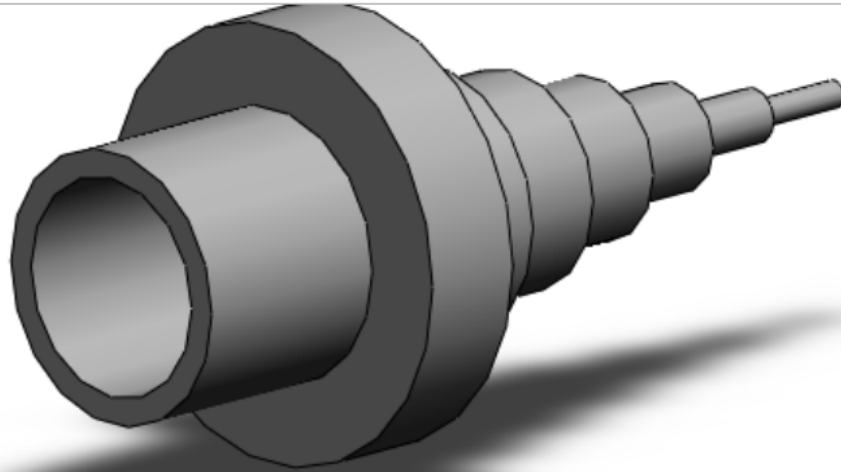


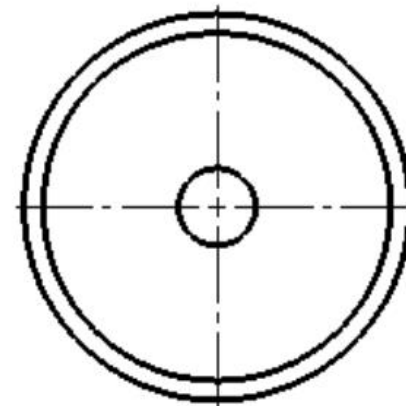
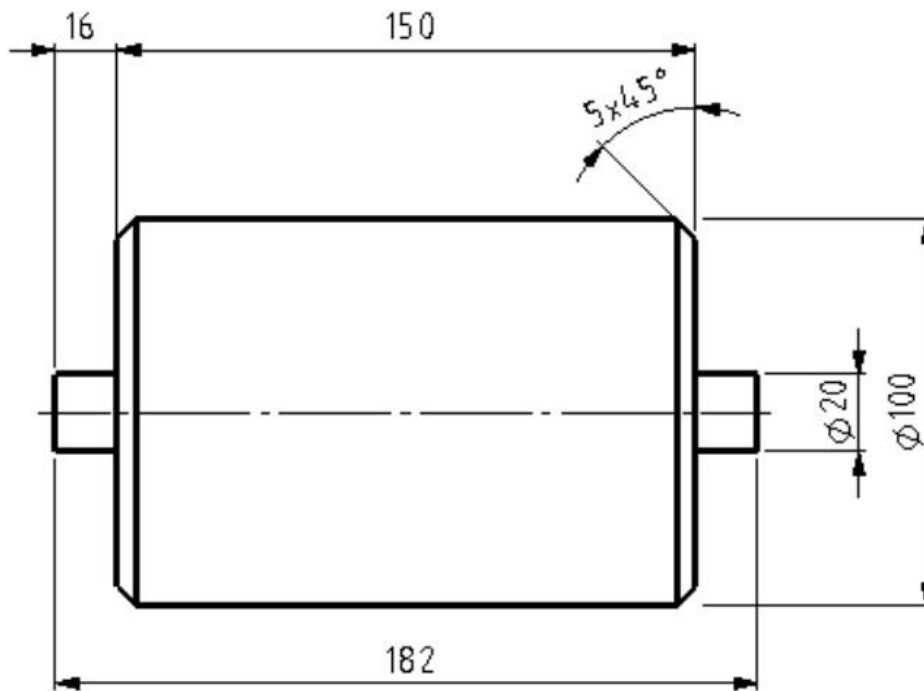
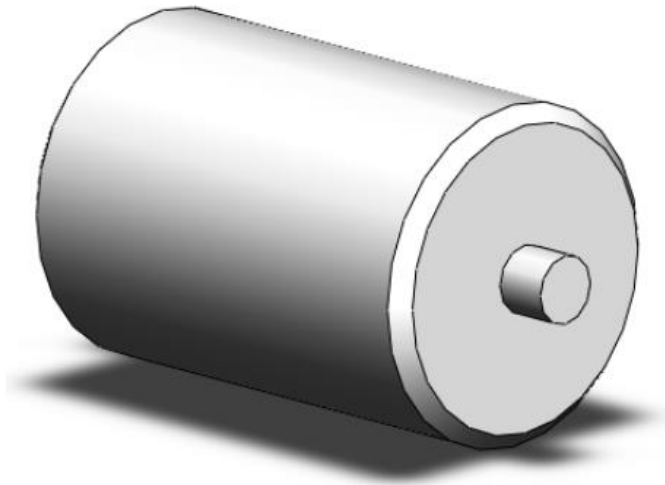
Thin Features

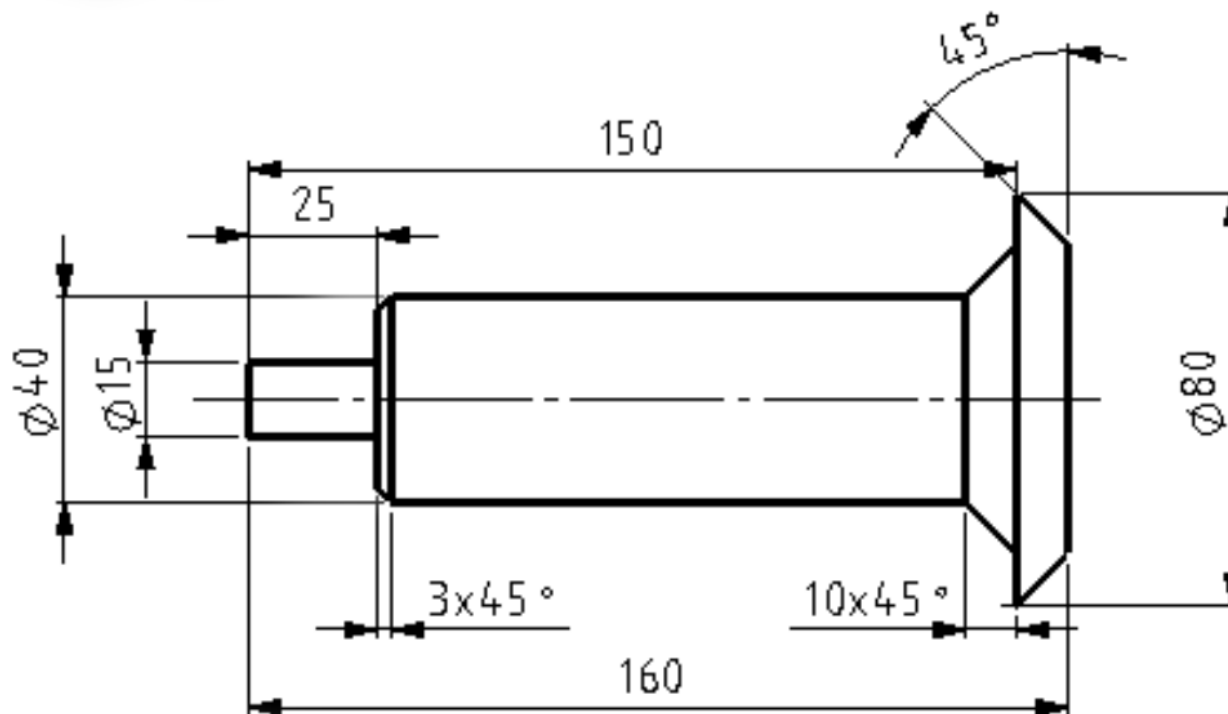
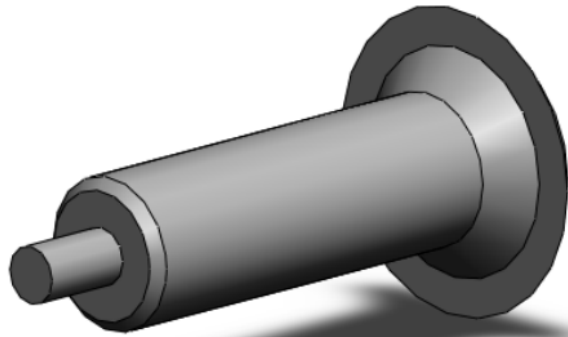
<p>Revolve, open</p>	 The diagram shows a blue 2D profile of a thin feature on the left. A vertical dashed line with a red arrow at the bottom indicates the axis of revolution. To the right, a 3D perspective view shows the resulting solid, which is a half of a bowl-like shape with a central hole, representing an open revolve.
<p>Revolve, closed</p>	 The diagram shows the same blue 2D profile and axis of revolution. The resulting 3D solid is a full, closed bowl-like shape with a central hole, representing a closed revolve.
<p>Extrude, open</p>	 The diagram shows the blue 2D profile and axis. The resulting 3D solid is a flat, open U-shaped extrusion, representing an open extrude.
<p>Extrude, closed</p>	 The diagram shows the blue 2D profile and axis. The resulting 3D solid is a closed, rectangular extrusion with a U-shaped cutout, representing a closed extrude.

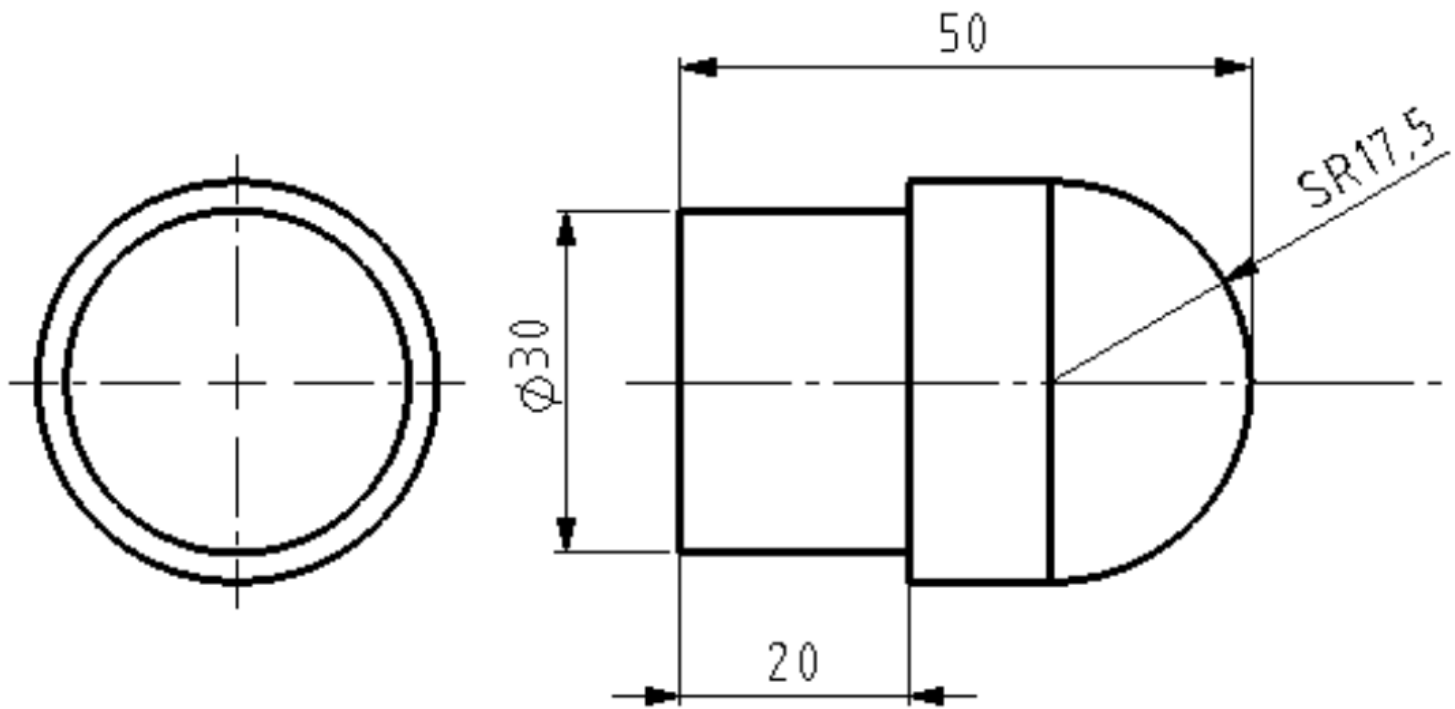
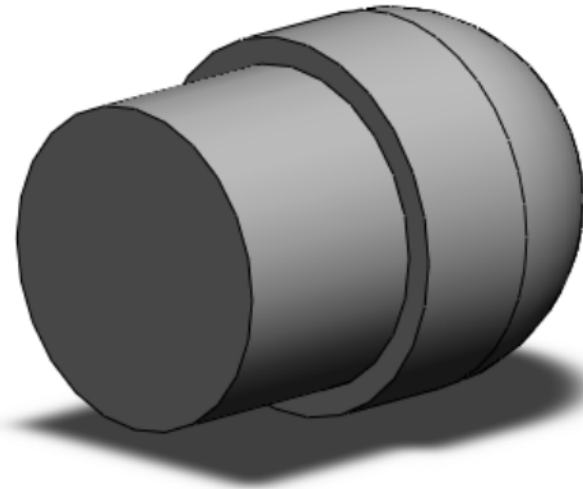


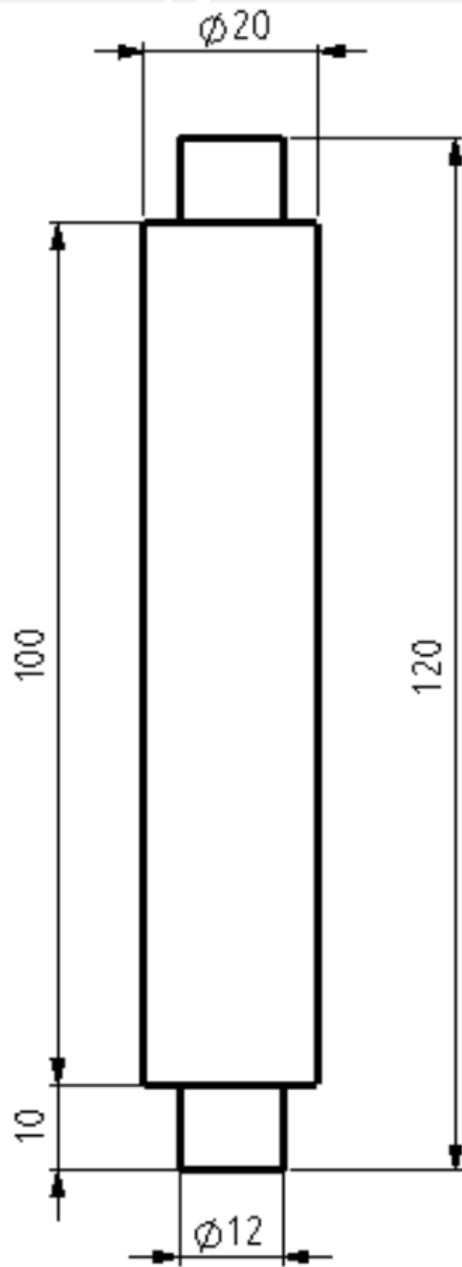


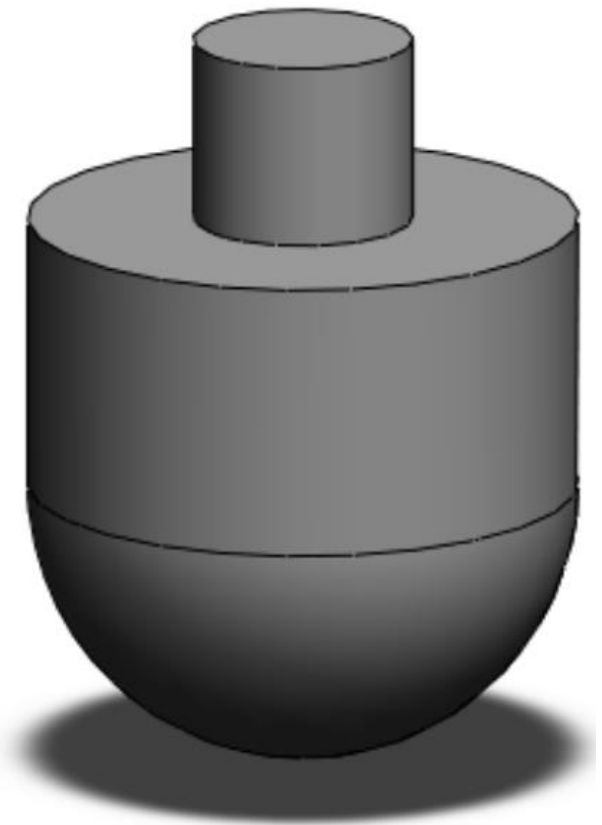
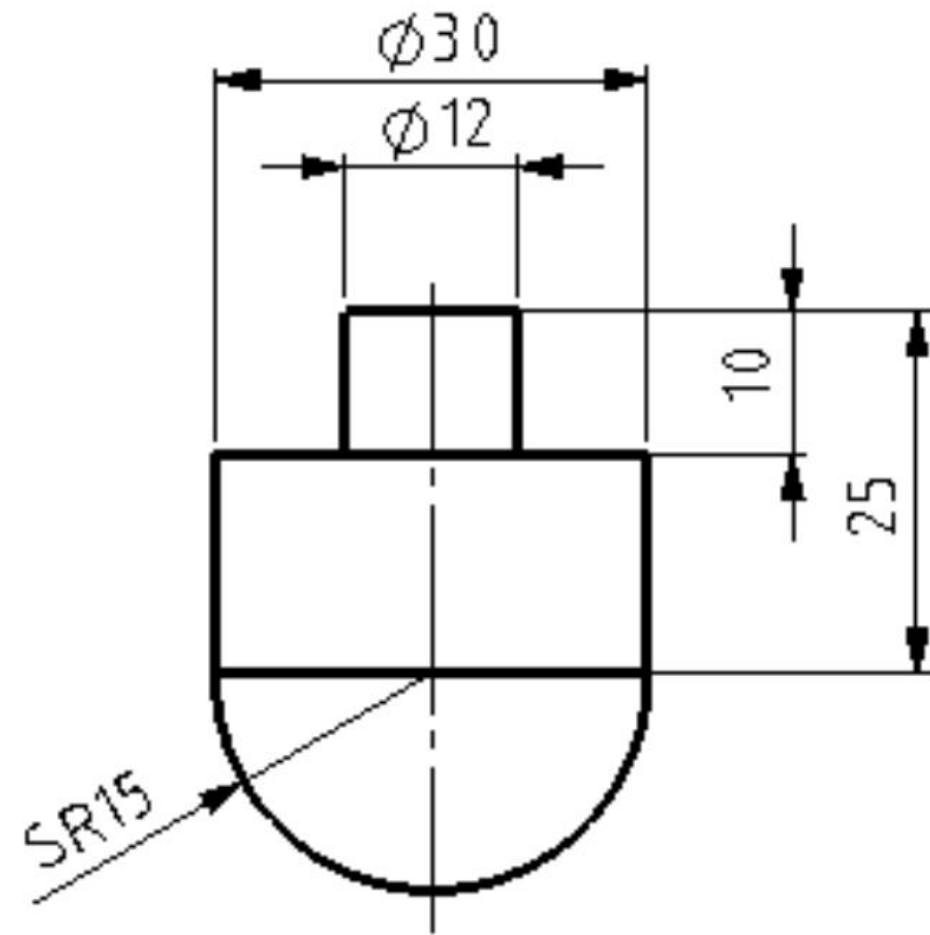




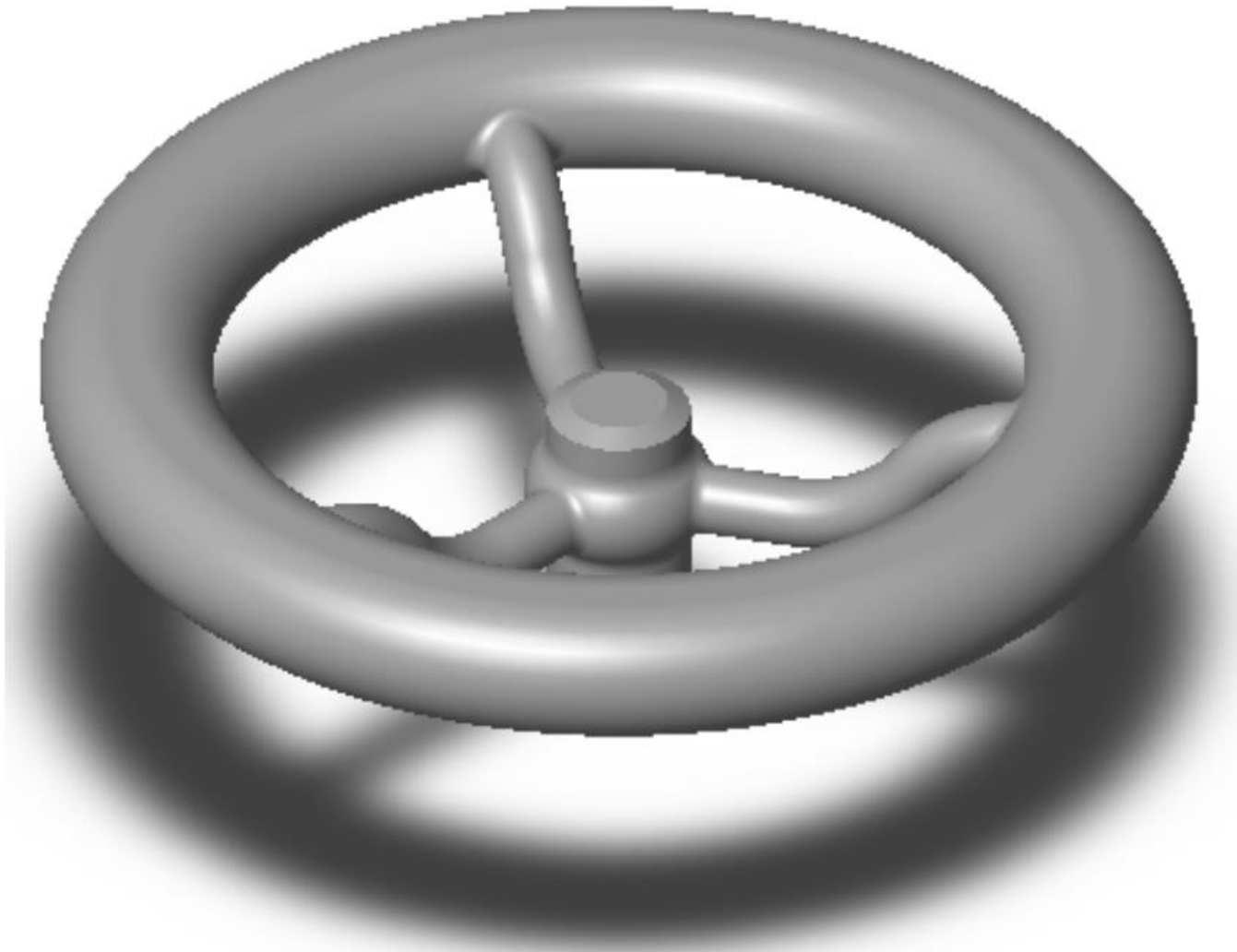


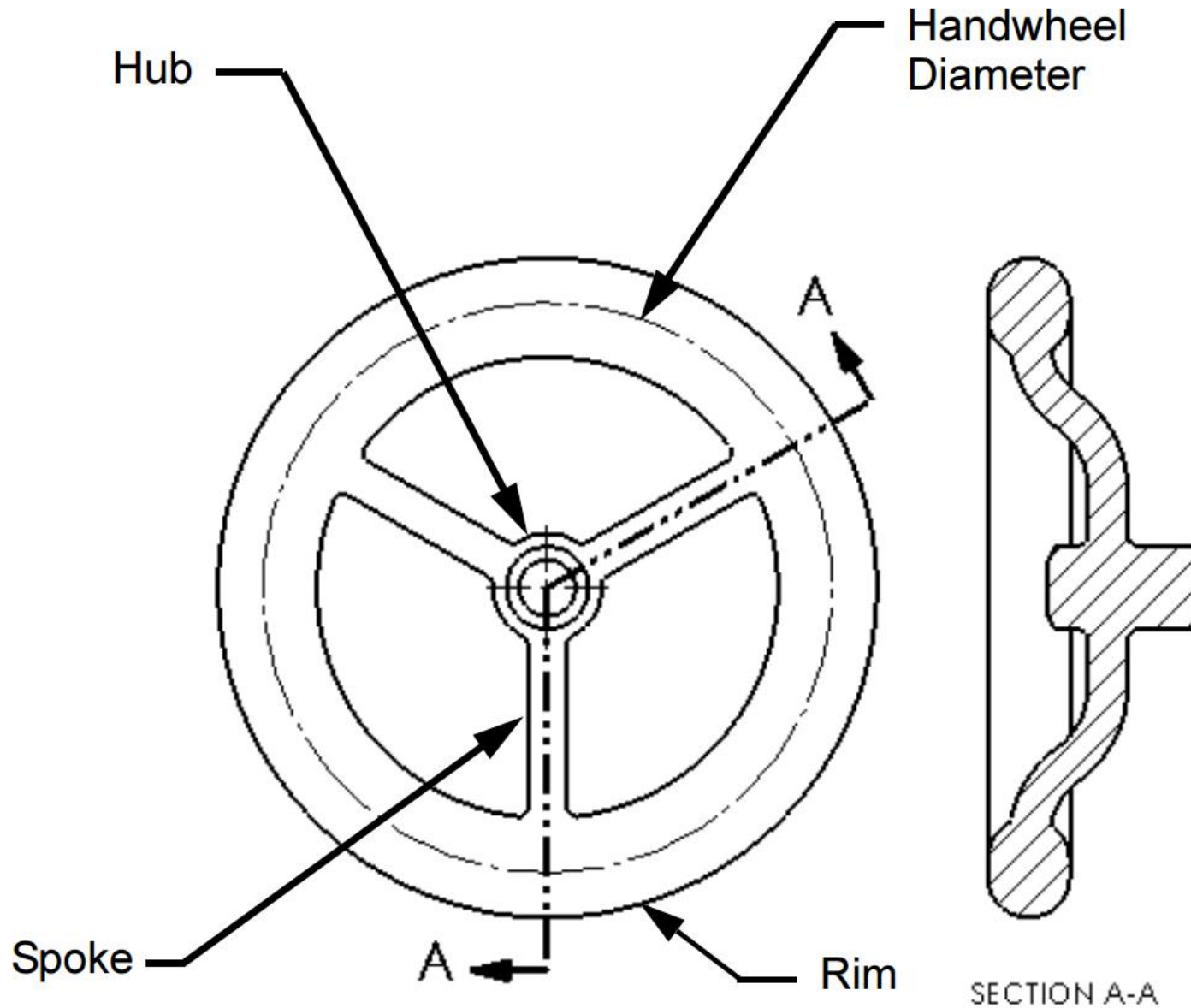




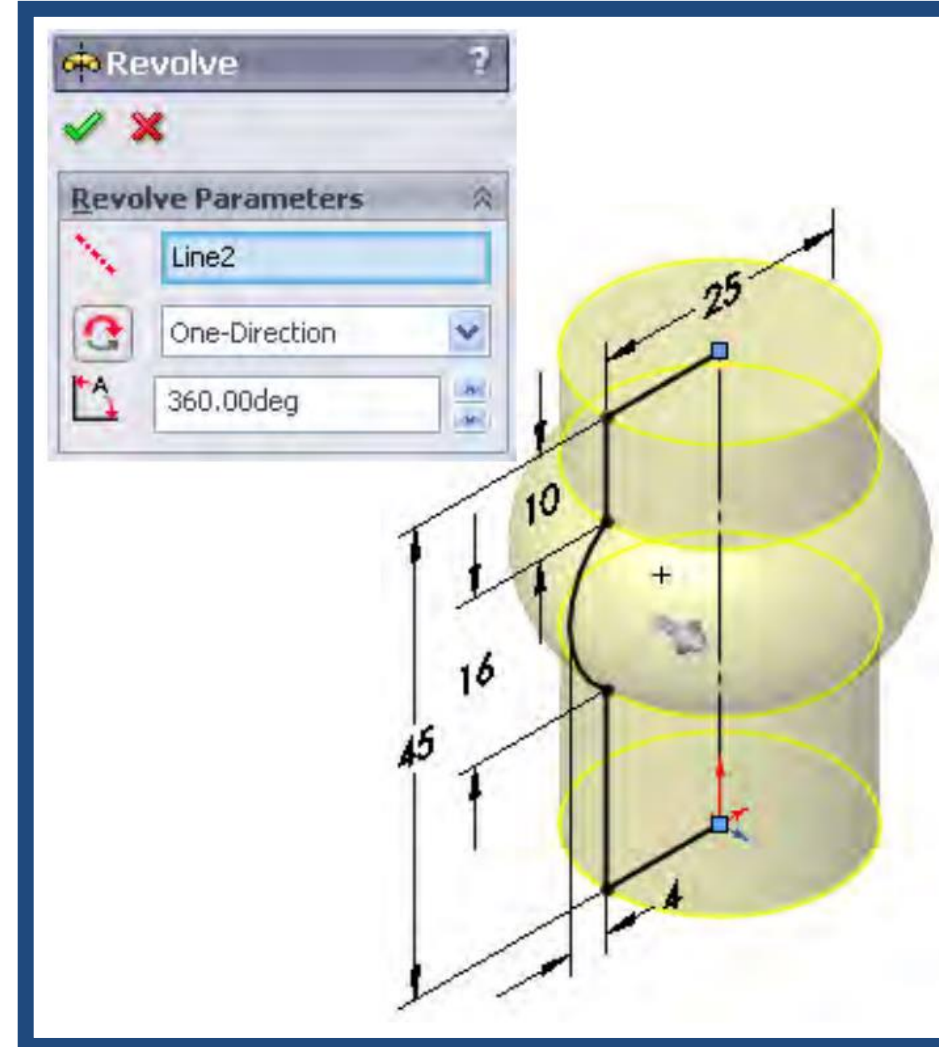
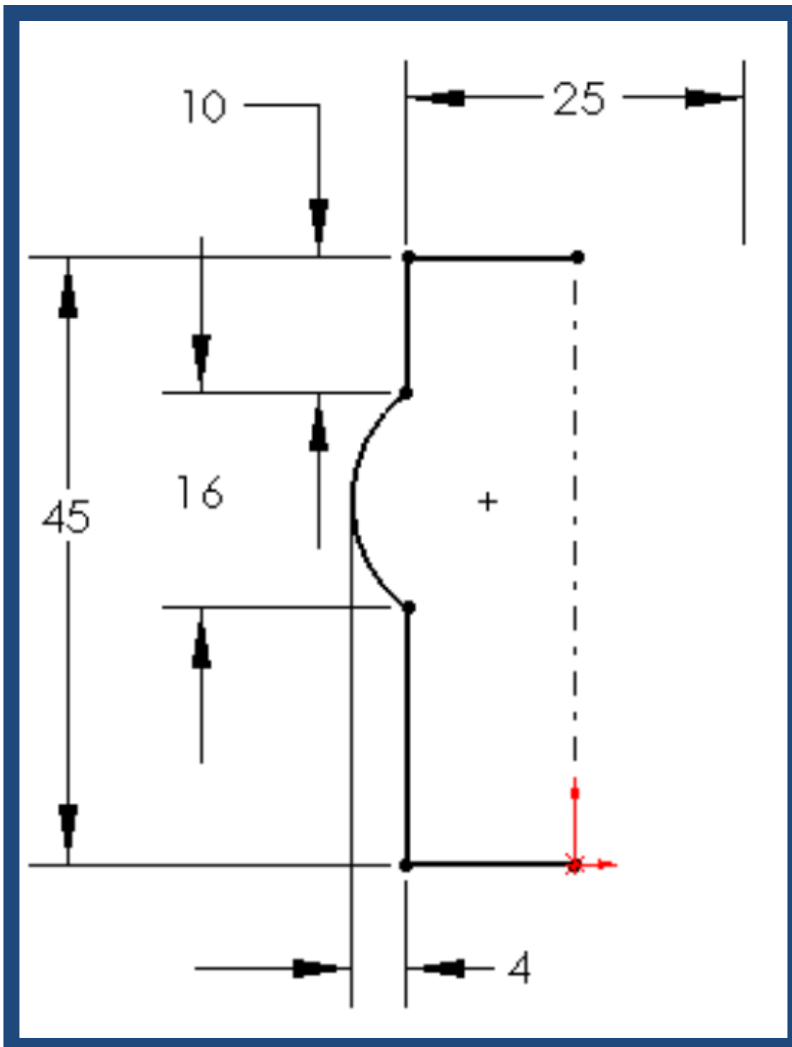


Design Handwheel

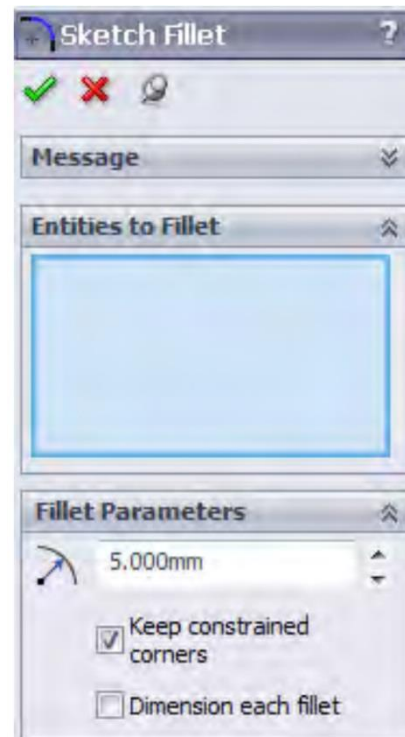
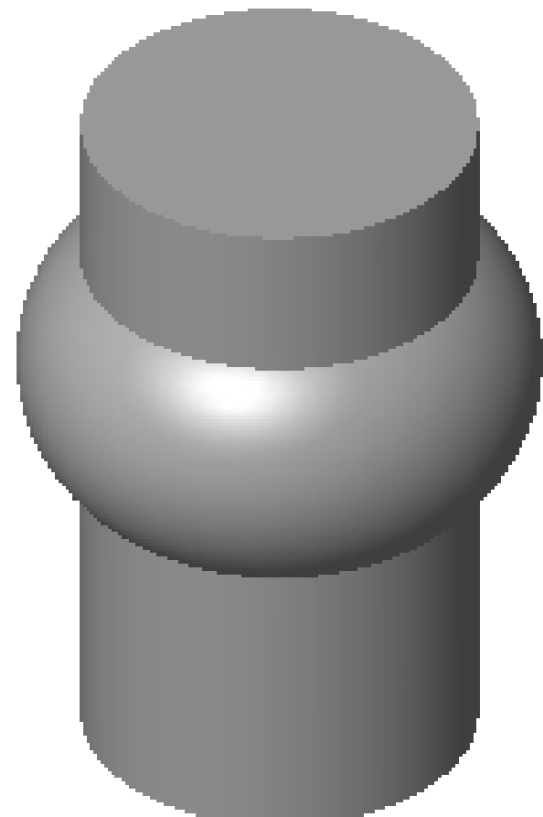




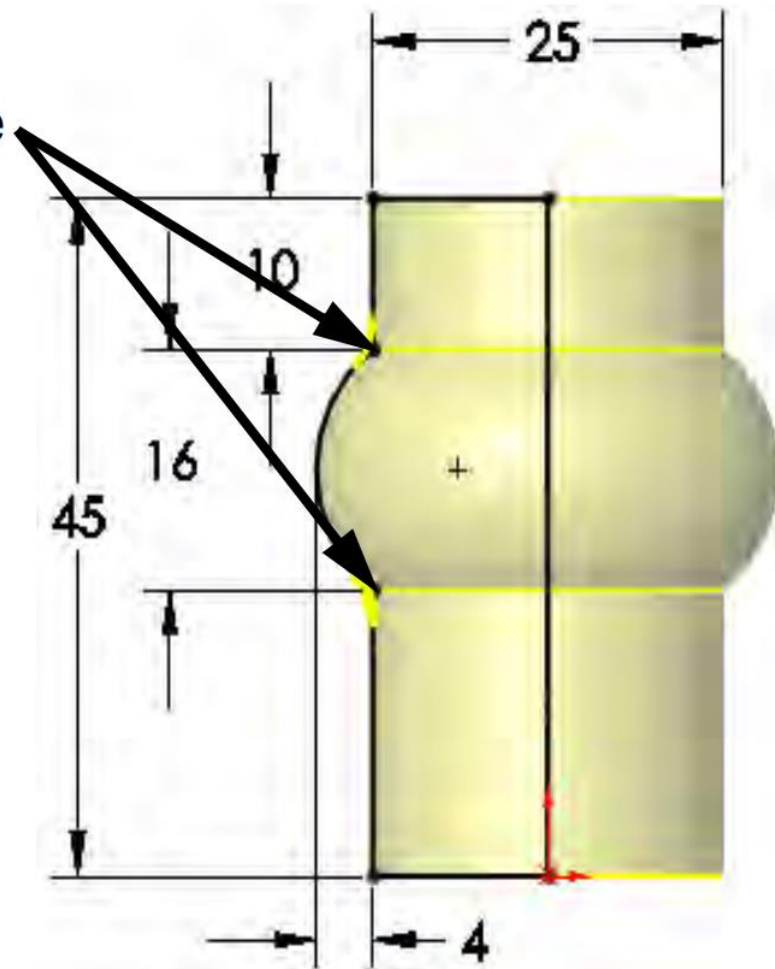
Sketches of Revolved Features



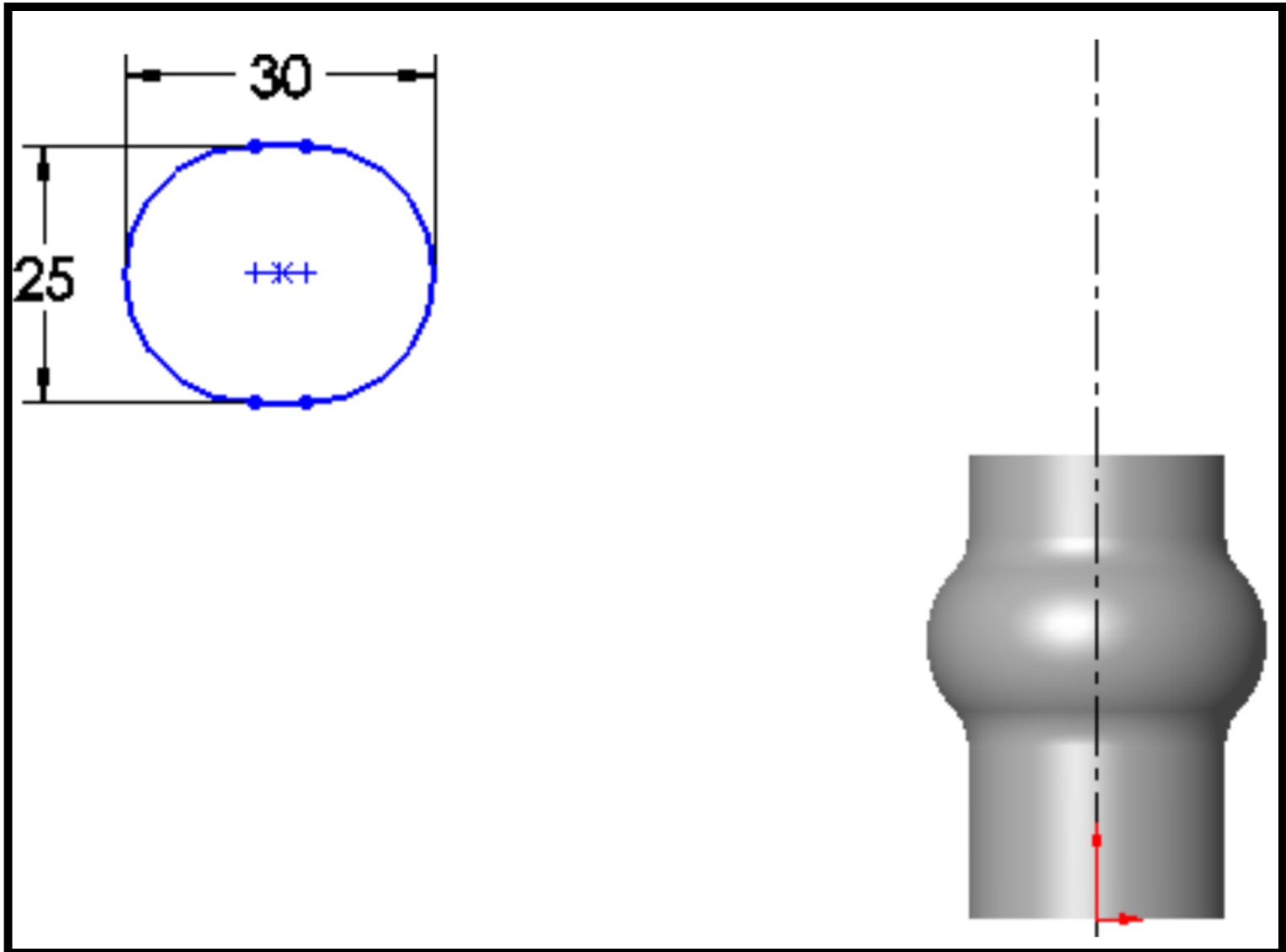
Fillet settings



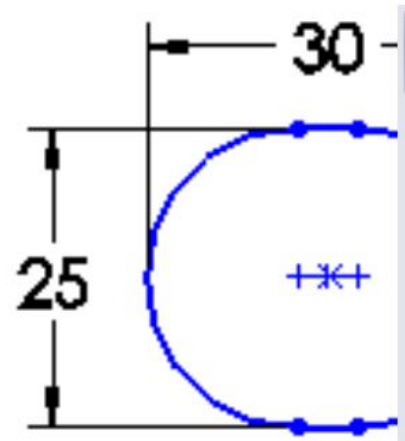
Pick here



Building the Rim



Rotation axis



Insert Line ?

✓ ✗

Message ^

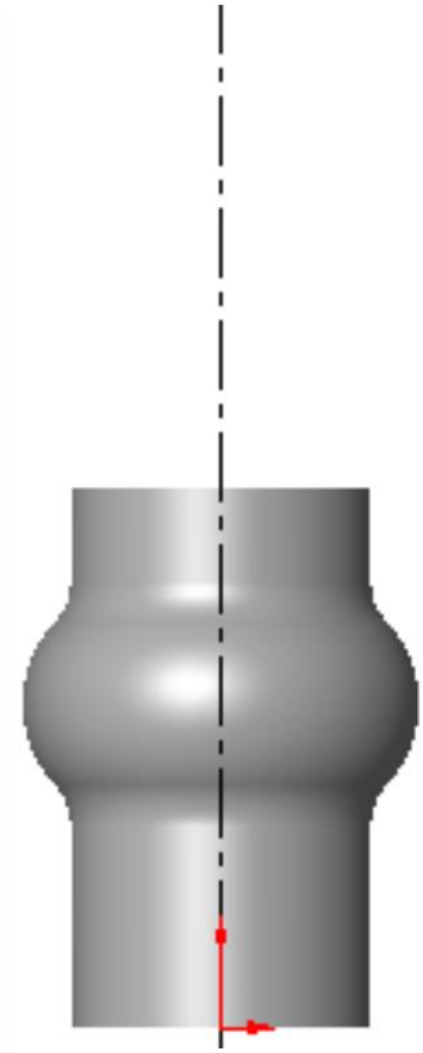
Edit the settings of the next new vertical line, or sketch a new vertical line.

Orientation ^

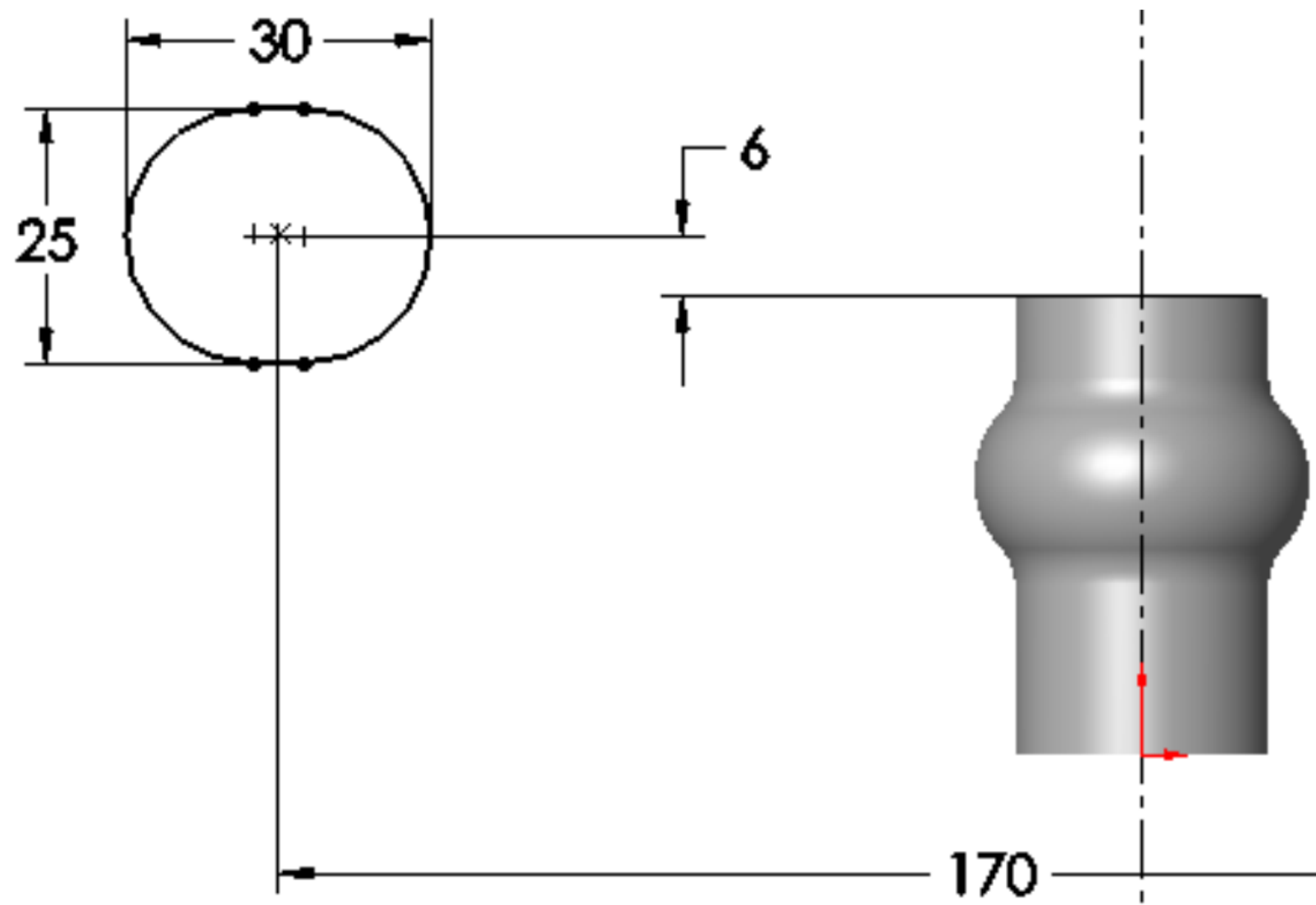
- As sketched
- Horizontal
- Vertical
- Angle

Options ^

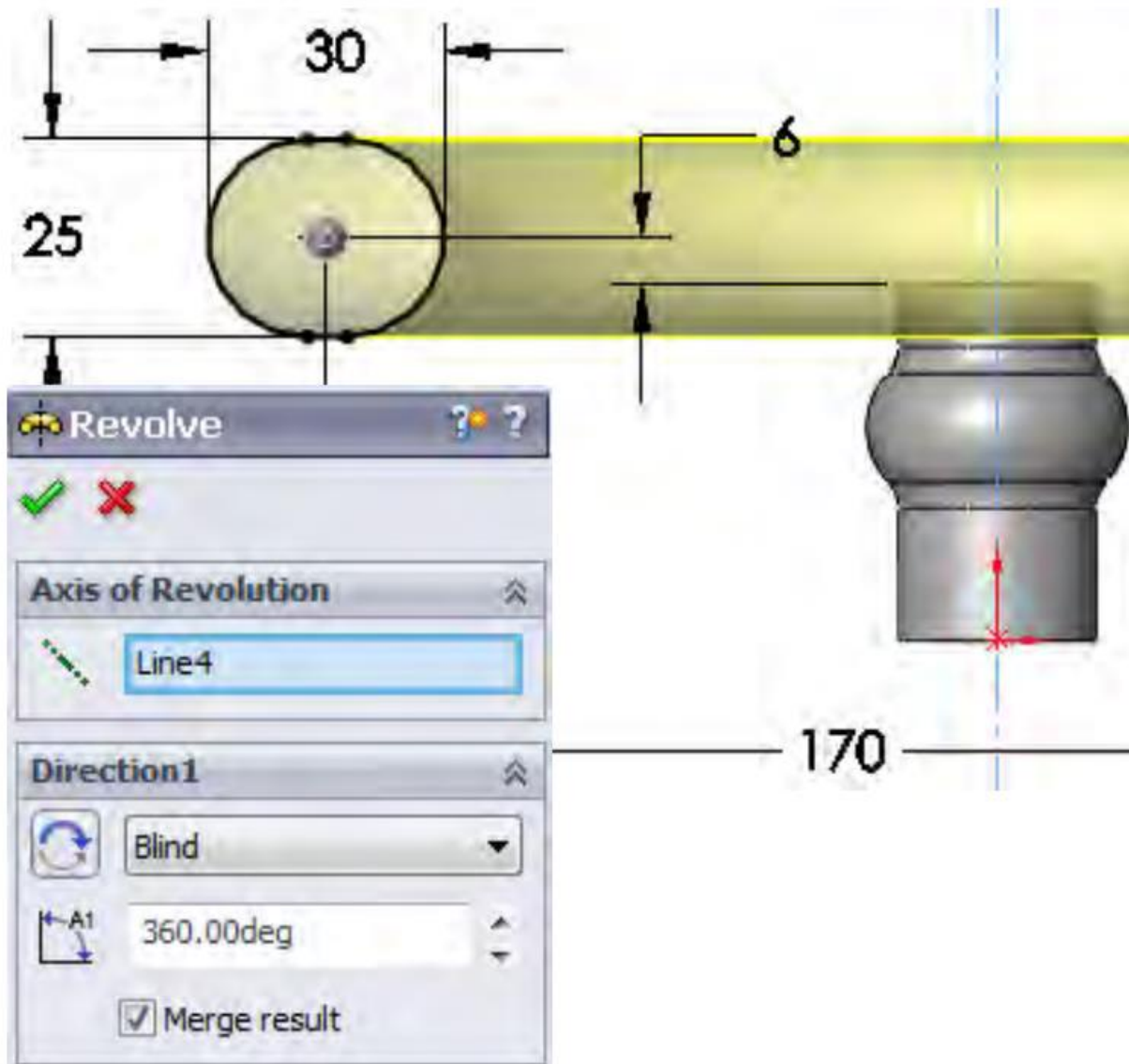
- For construction
- Infinite length



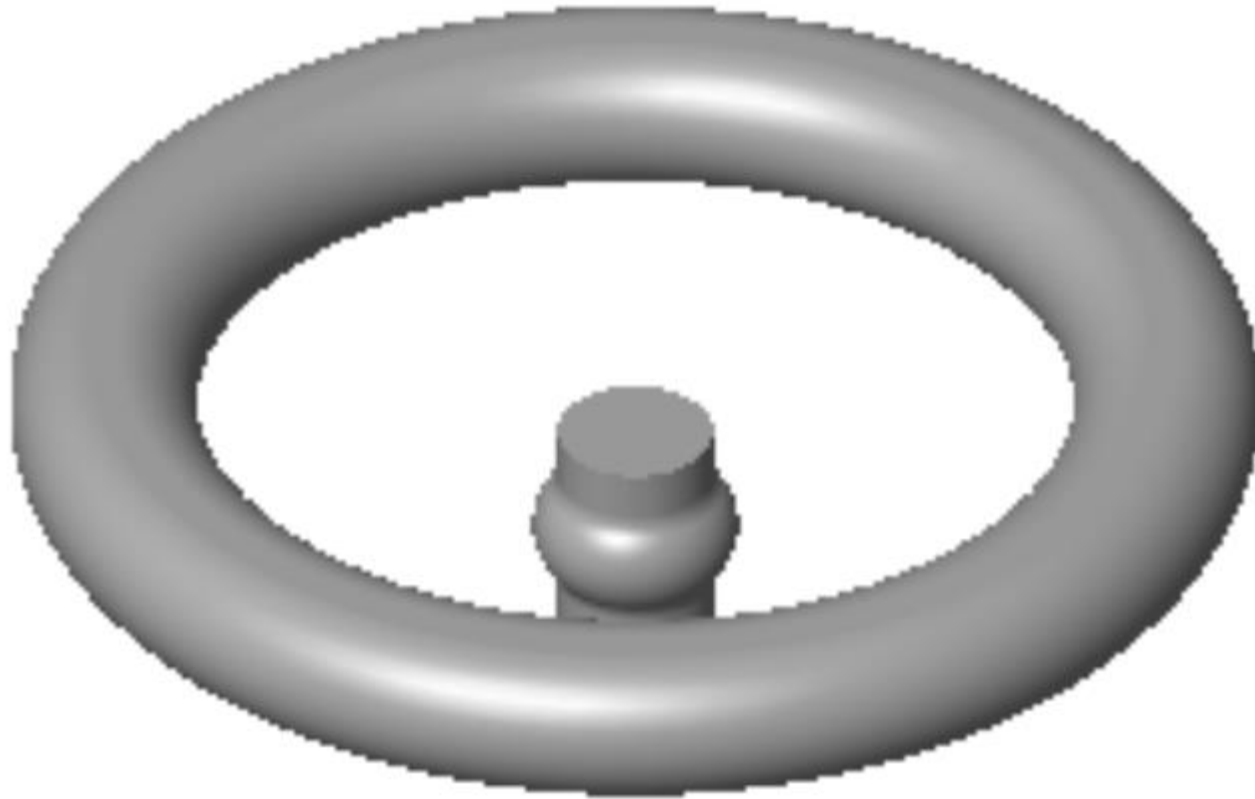
Add dimensions



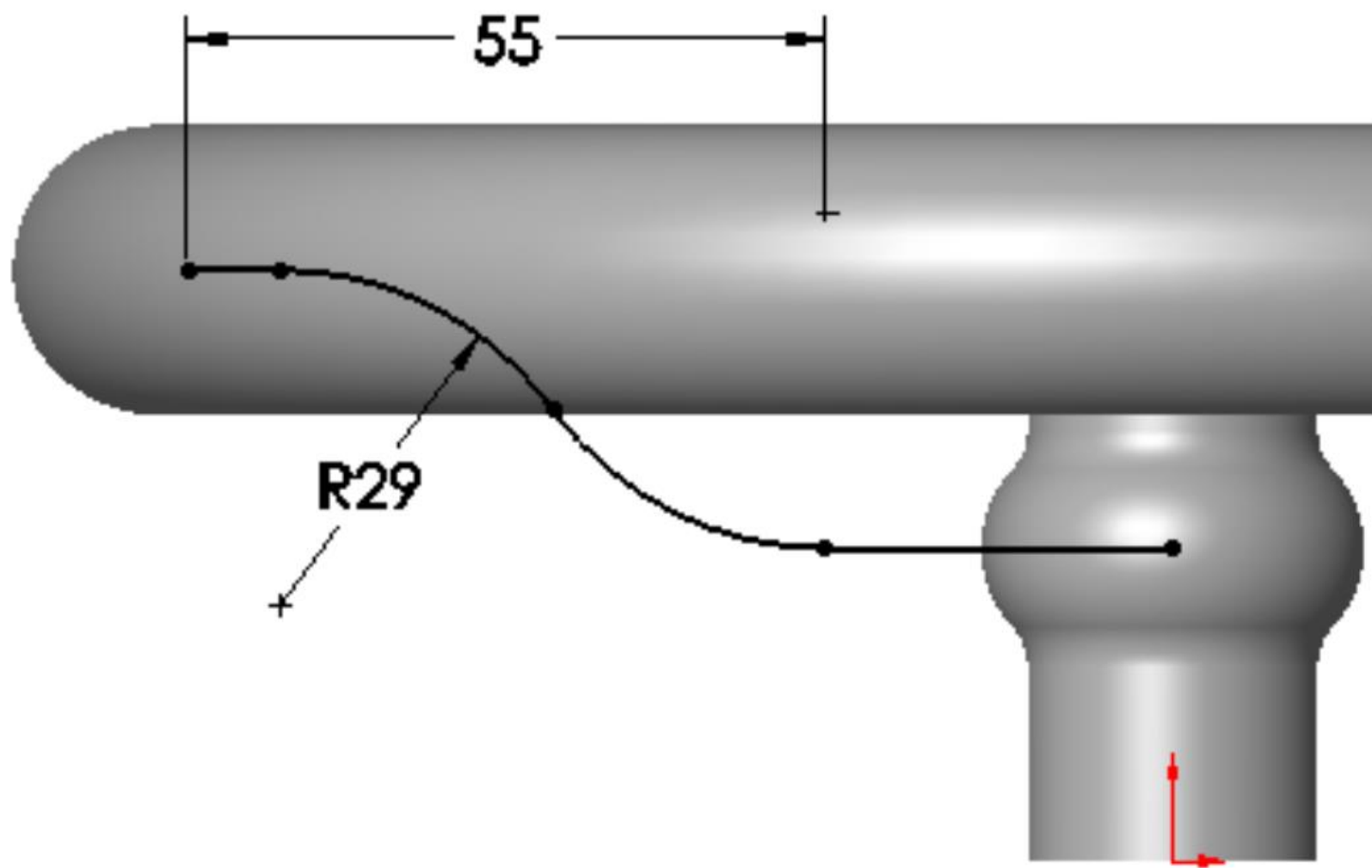
Completed feature



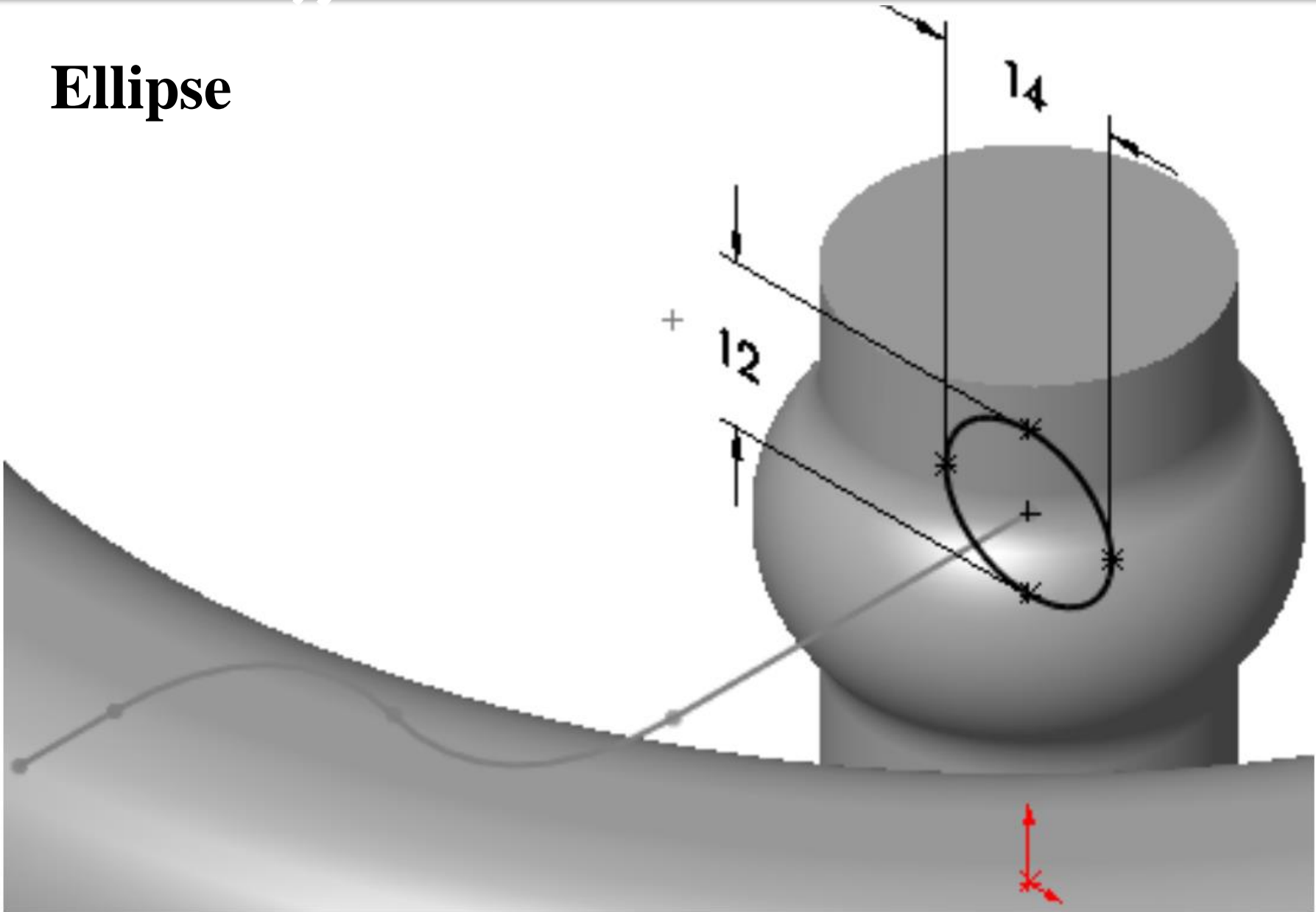
Multibody Solids



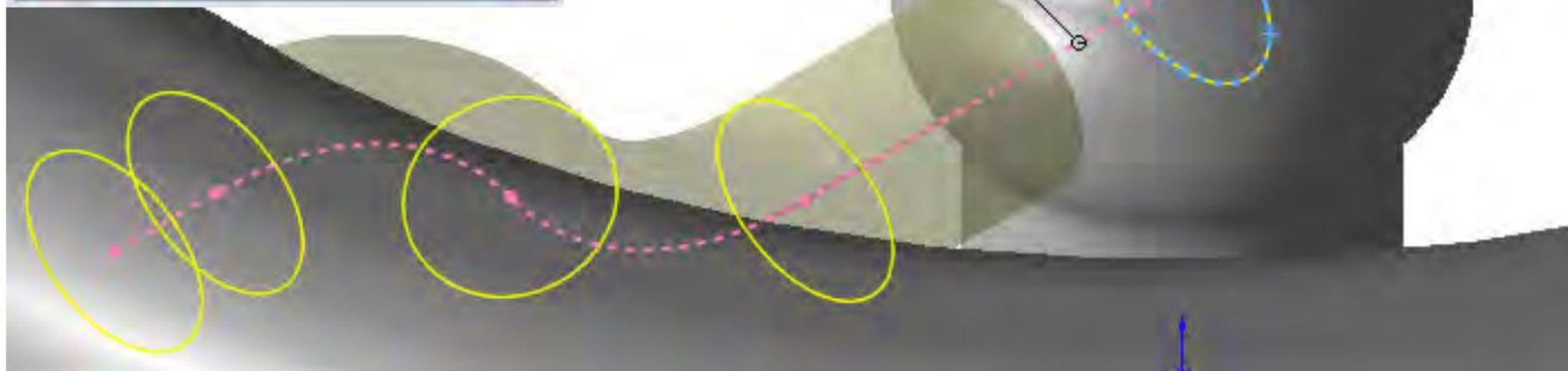
Building the Spoke



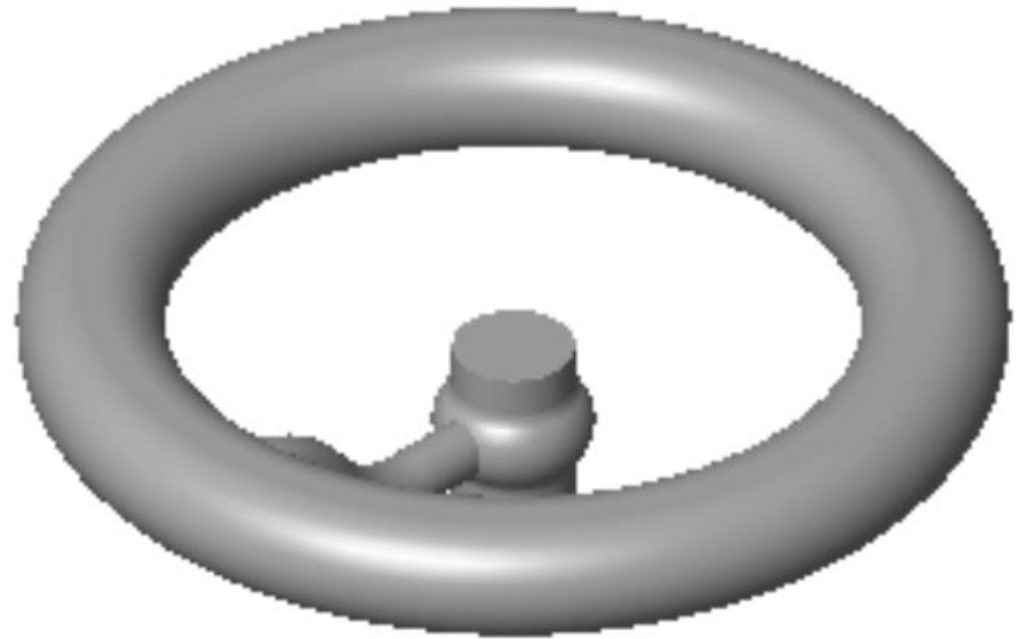
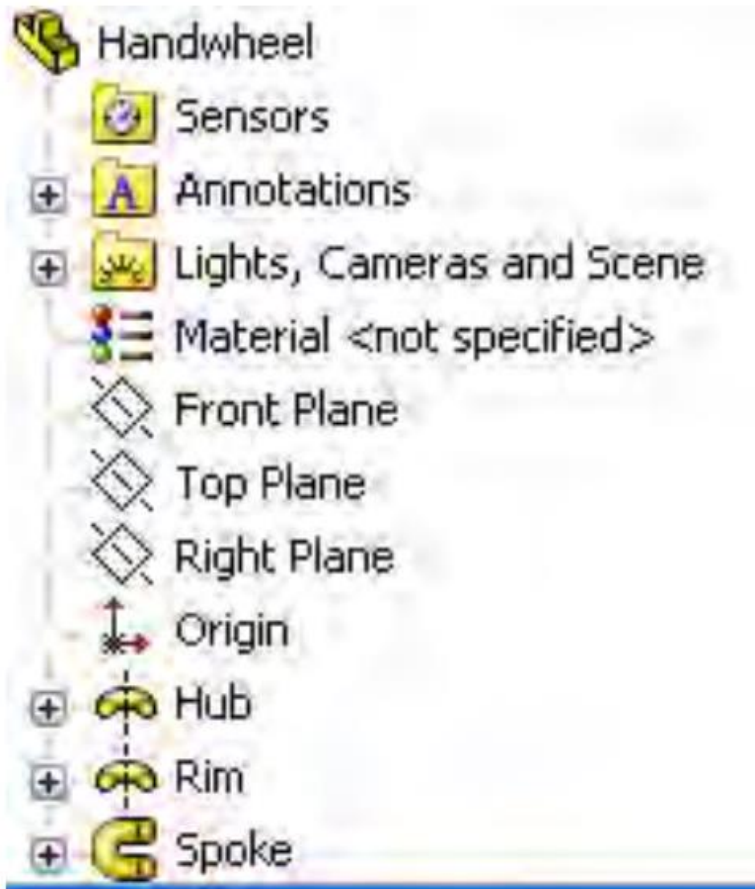
Ellipse



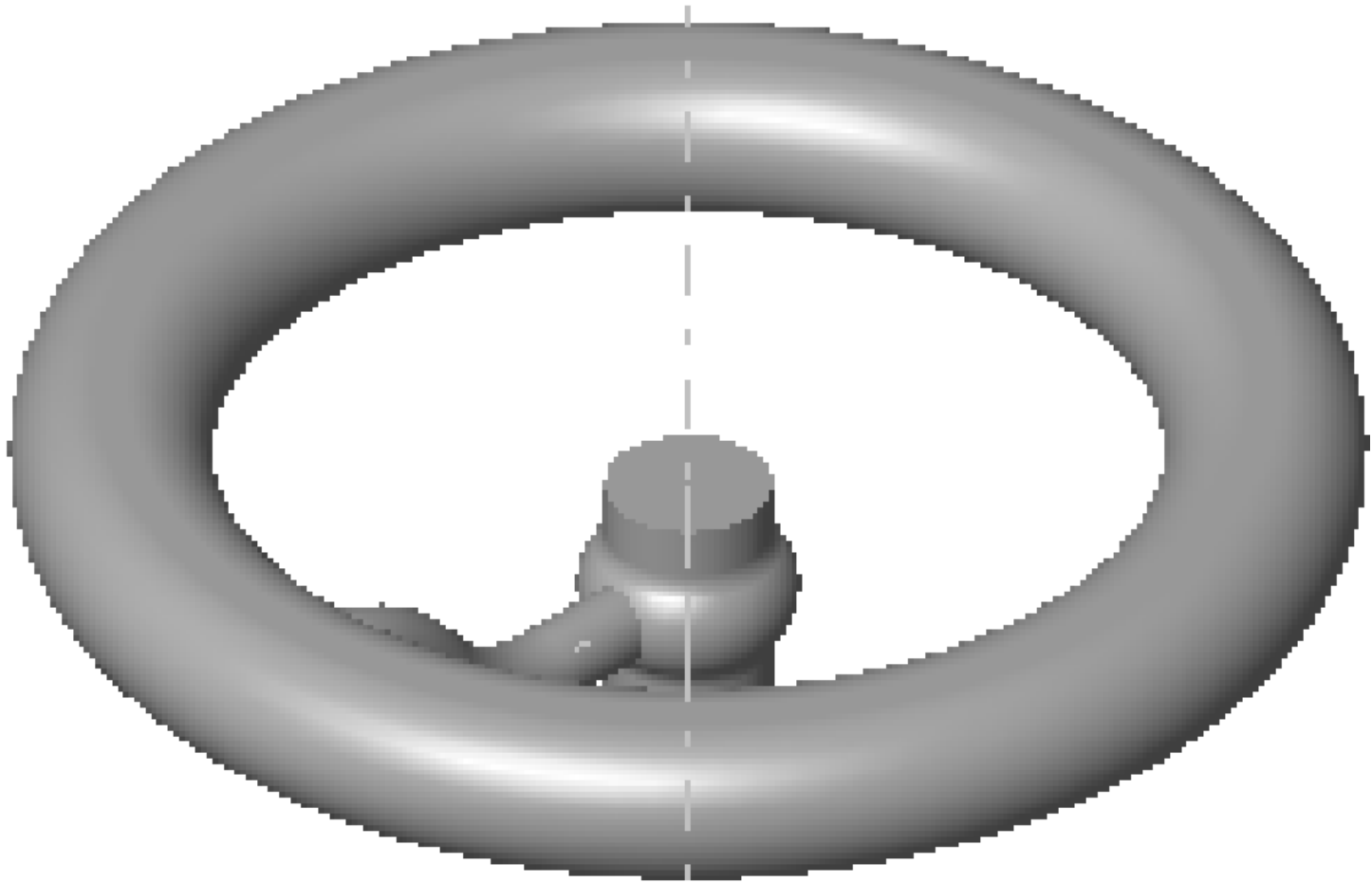
Sweep



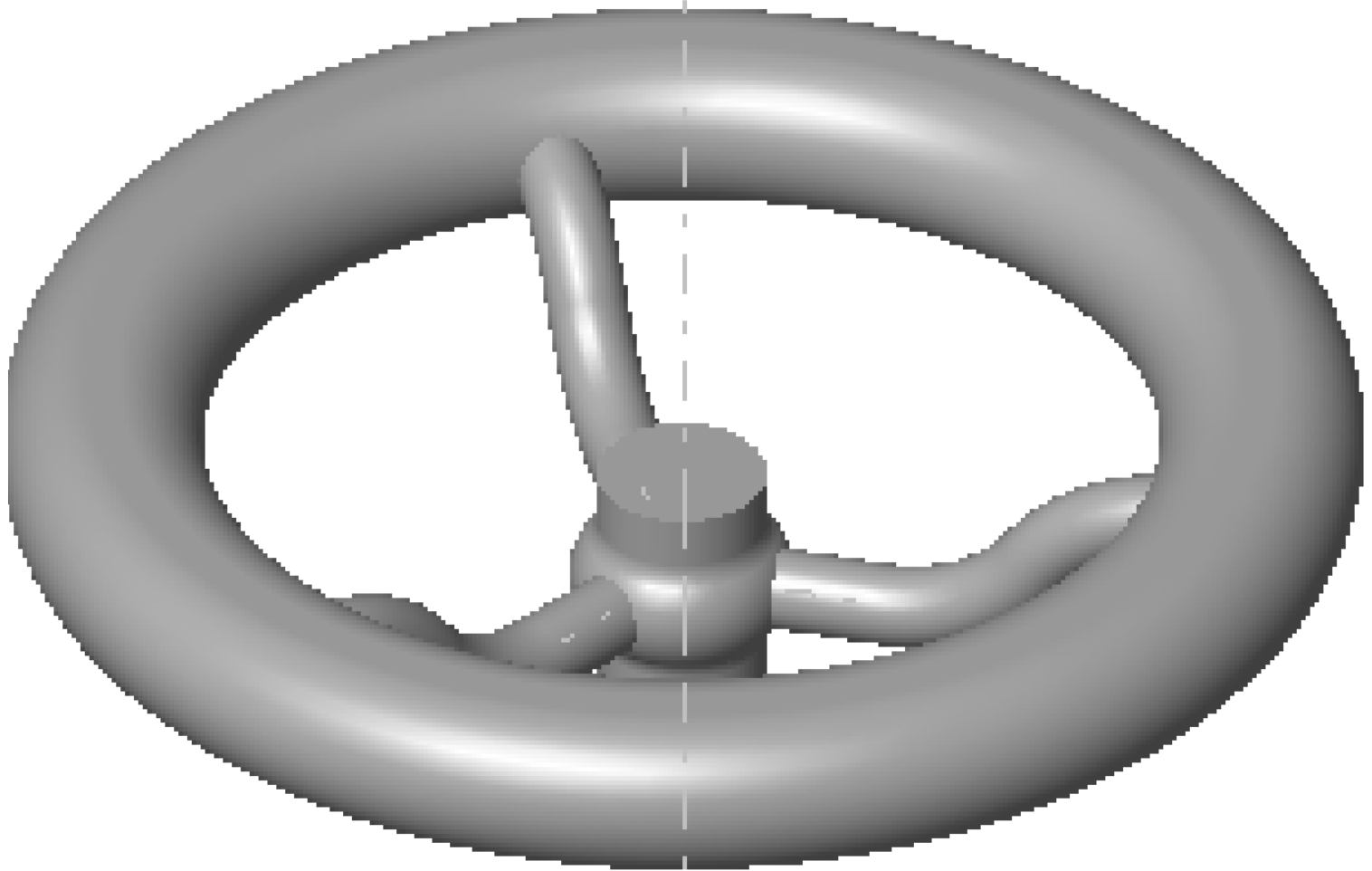
Results.



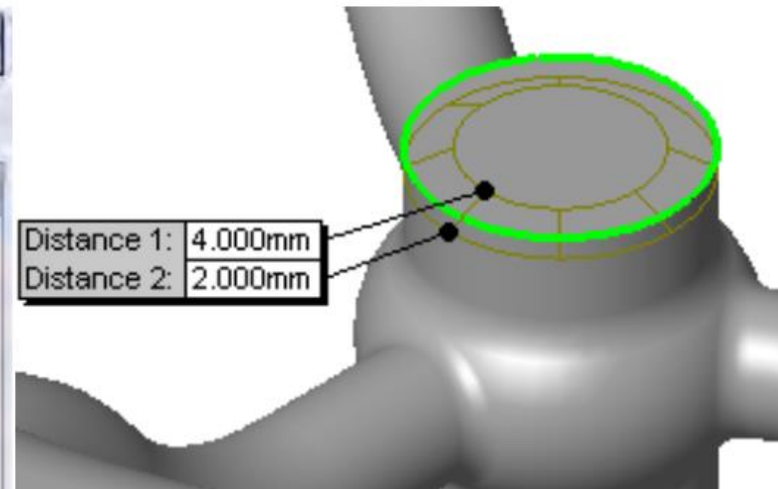
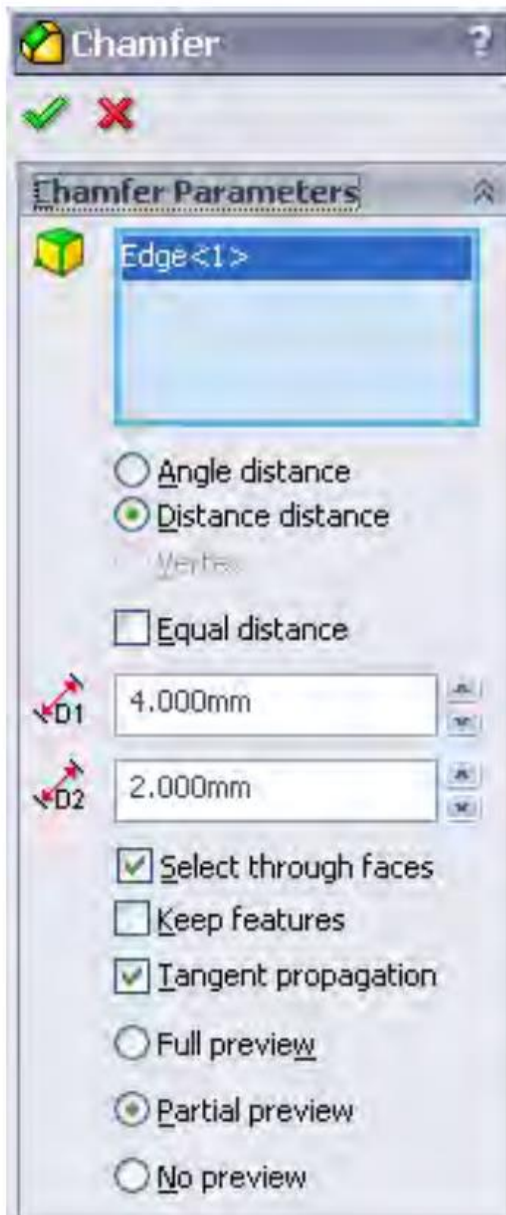
Temporary axes.



Pattern the Spoke



Chamfer



RealView Graphics

