

# Sketch Centerline Training

Sketch Centerline\_Training.zip



## Sketch Centerline - Overview



### Main Functions

- Controls Show/Hide options for centerlines using layers
- Easily change a diameter annotation of a symmetry model appeared to be half

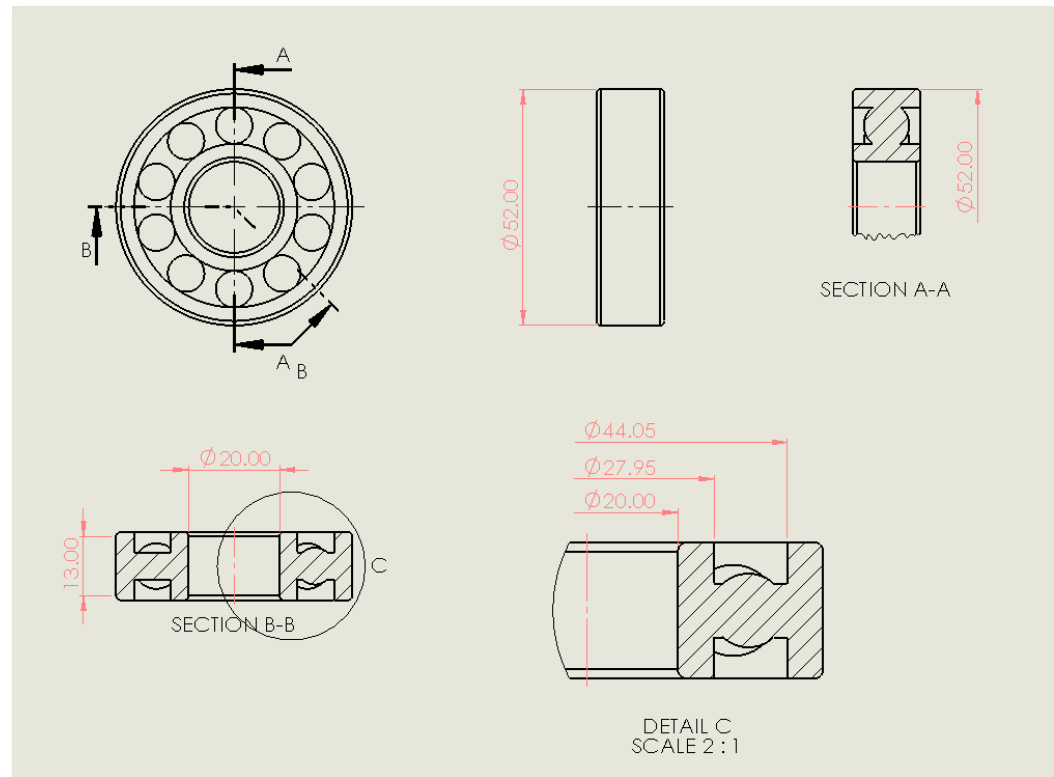
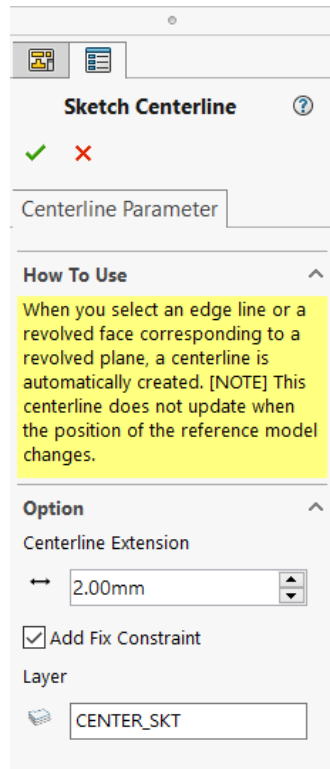


### Benefits

- Utilizes Section View of drawings
- Diverse editing options as using sketch centerlines



**Course Objective:** Follow instructions through the course and learn how to use Sketch Centerline.

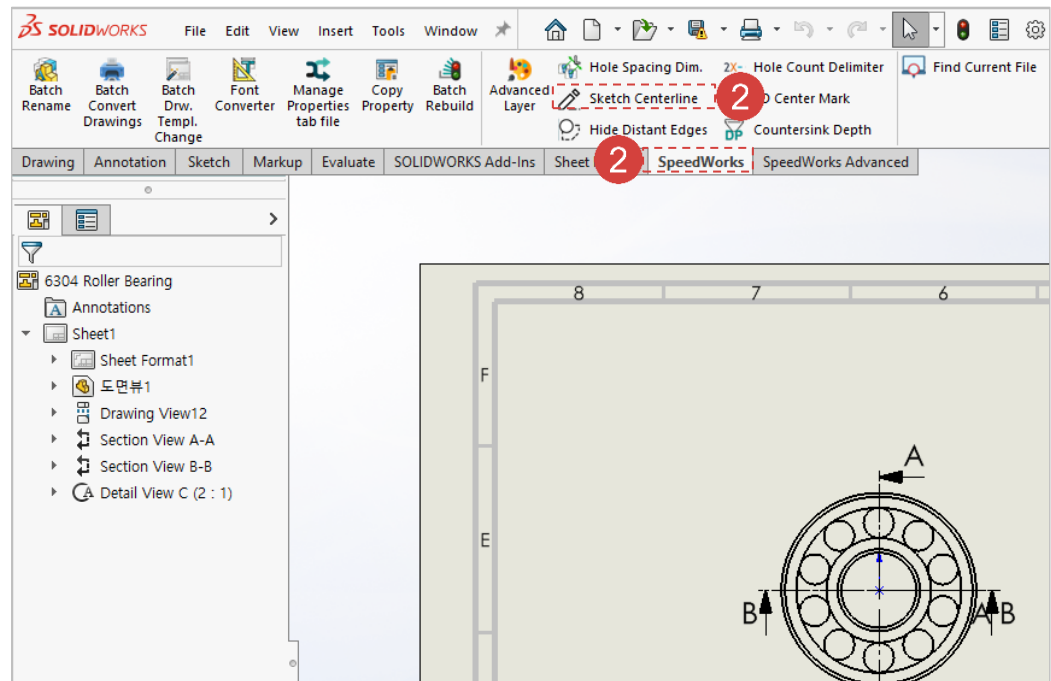
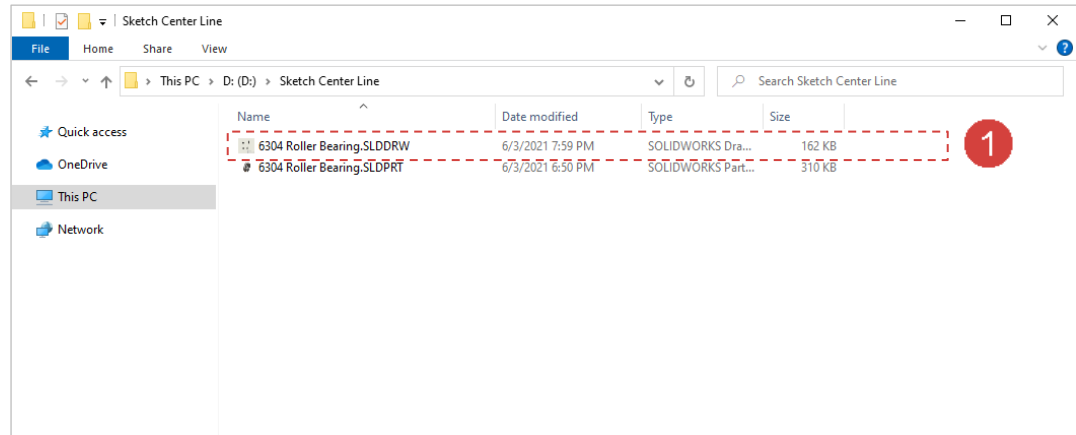


## Sketch Centerline - Execute

Work on the part with the red circle (1) following instructions by yourself.  
The part with the black square (1) is an additional explanation or a reference.

1 Execute **6304 Roller Bearing.SLDDRW** file among example files.

2 Click **[SpeedWorks]tab - [Sketch Centerline]** button to run the tool.



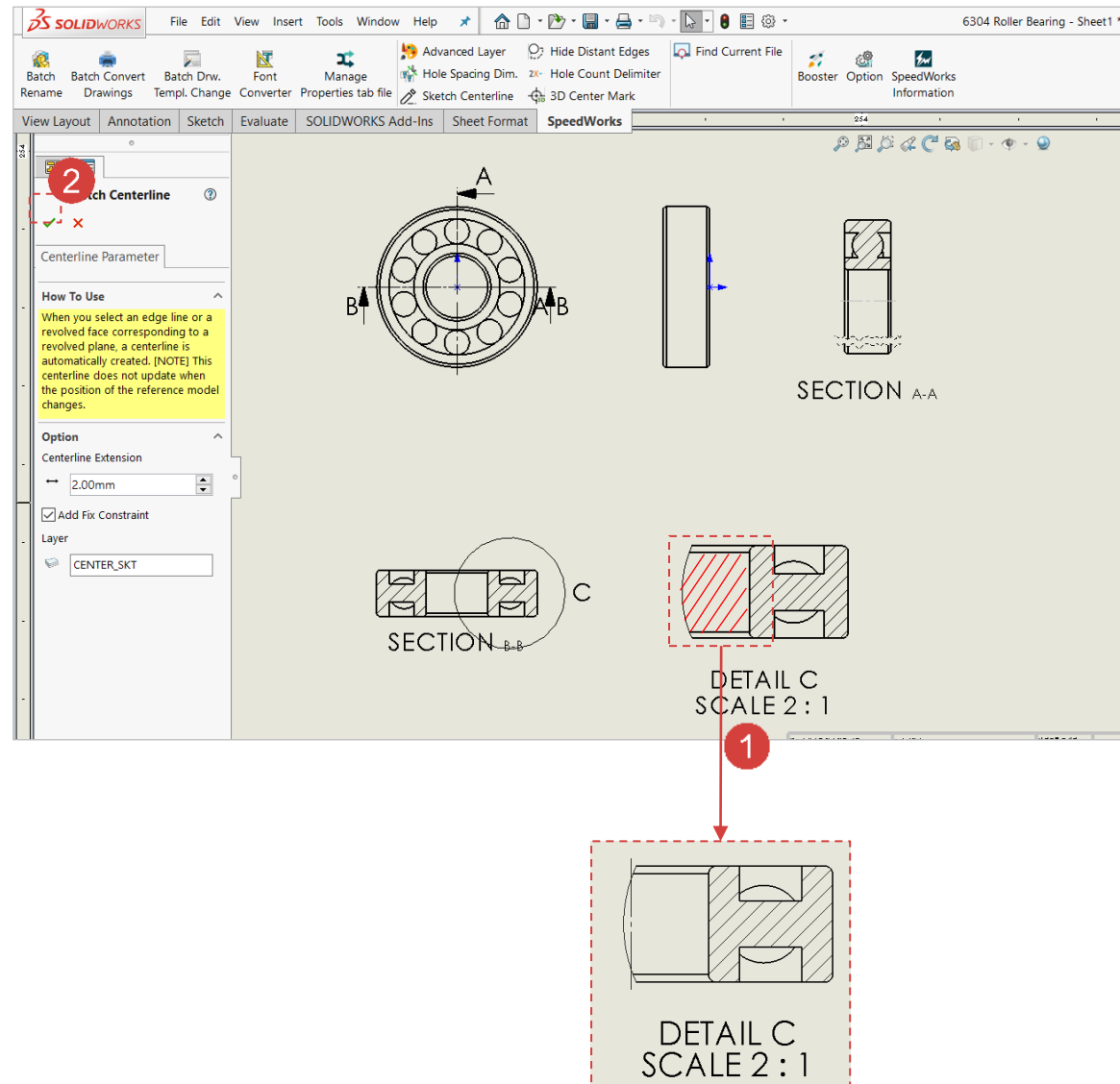
※ You can download example files at Support page in SpeedWorks Homepage(<https://speedworks.info>).

## Sketch Centerline - Add Sketch Centerline

Work on the part with the red circle (1) following instructions by yourself.  
The part with the black square (1) is an additional explanation or a reference.

1 Click inner surfaces of bearing (Area slashed in red) on [Detail C] while Sketch Centerline is running. As soon as clicking it, a sketch centerline being sketch type is created as follows.

2 Then, click [OK] button to end [Sketch Centerline].



## Sketch Centerline - Add Annotation Centerline

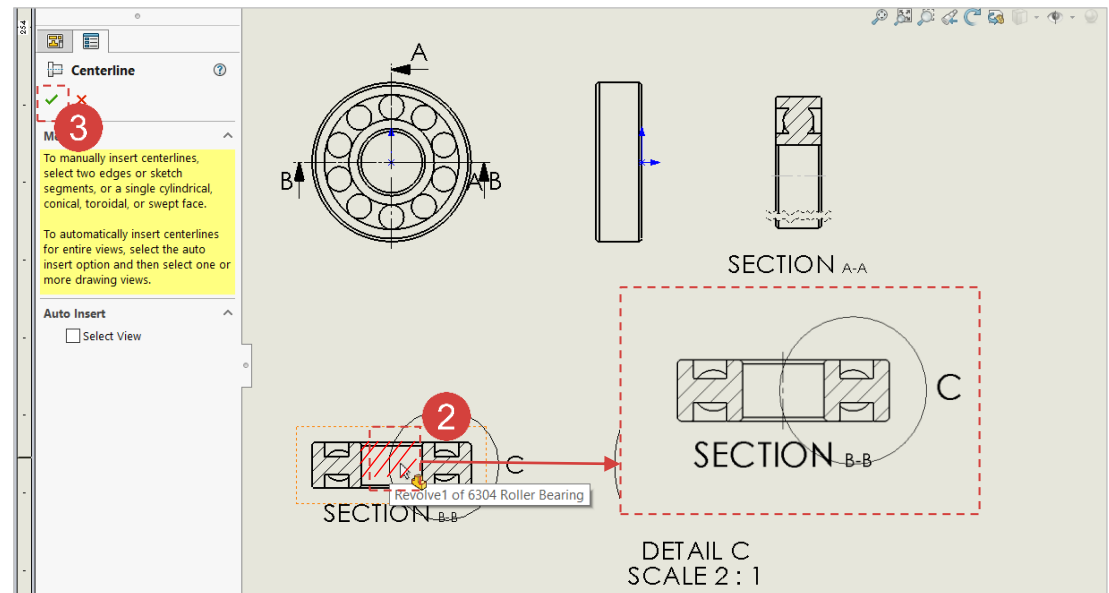
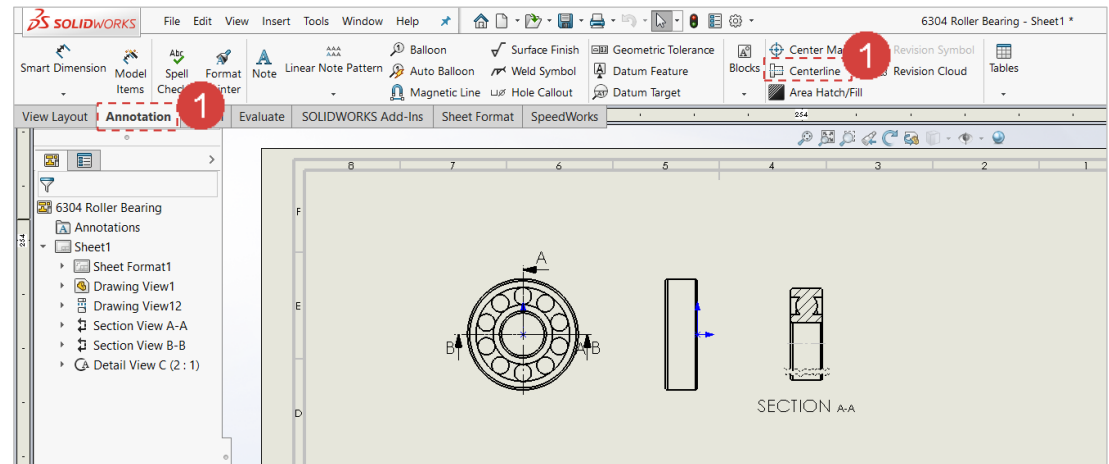
Work on the part with the red circle (1) following instructions by yourself.  
The part with the black square (1) is an additional explanation or a reference.

1 Click [Annotations] tab - [Centerline] to run the tool.

Click inner surfaces of bearing (Area slashed in red) on  
2 detail B while [Centerline] is activated. As soon as  
clicking it, a centerline being annotation type is created  
as follows.

Click [OK] button to end adding annotation centerlines.

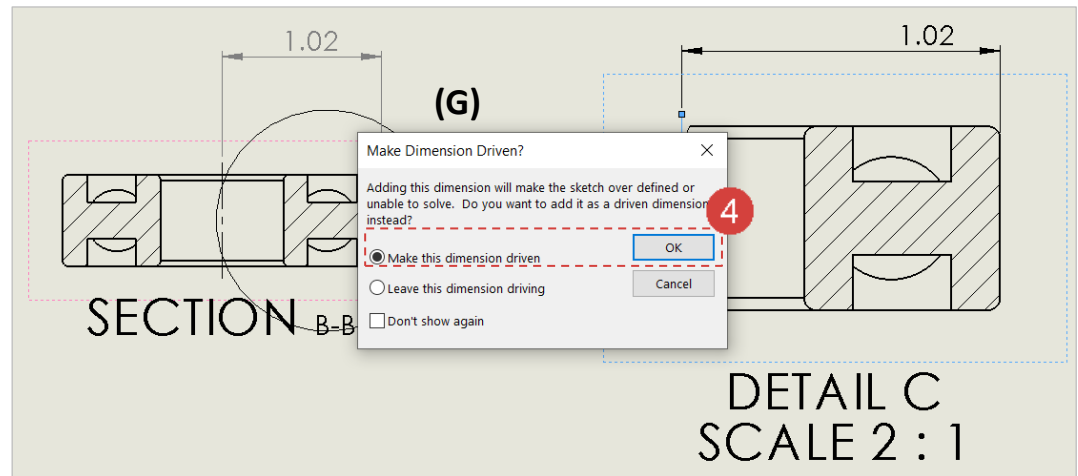
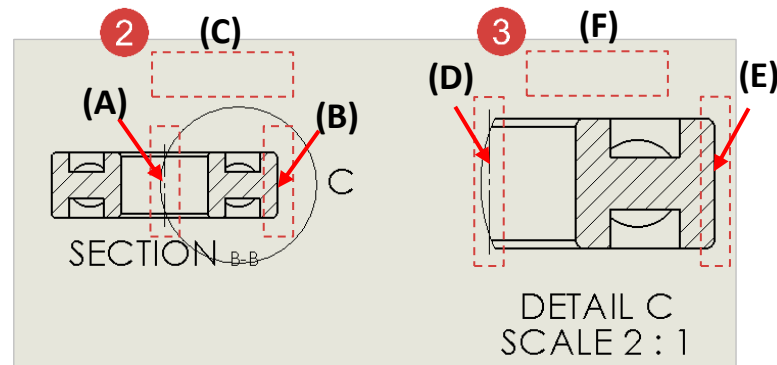
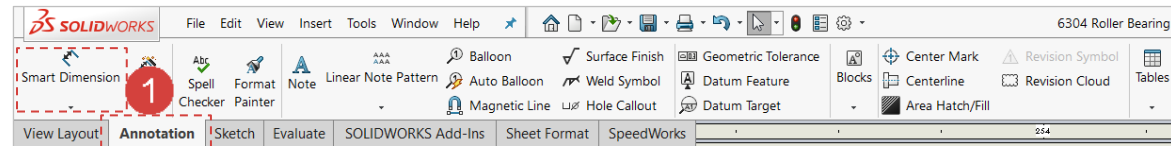
3



## Sketch Centerline - Insert Dimension



Work on the part with the red circle (1) following instructions by yourself.  
The part with the black square (1) is an additional explanation or a reference.

- 1 Click **[Annotations]tab - [Smart Dimension]** to start inserting smart dimensions.
- 2 As users click the centerline (A) and the right edge (B) on **SECTION B-B**, the distance between 2 lines is displayed. Relocate the dimension to make it fit into the box (C).
- 3 Next, as users click the centerline (D) and the right edge (E) on **DETAIL C**, the distance between 2 lines is displayed also. Relocate the dimension to make it fit into the box (F).
- 4 If the window having the title **[Make Dimension Driven?]** like (G) appears when users try inserting 2<sup>nd</sup> centerline, check **[Make this dimension driven]** and click **[OK]** button.



## Sketch Centerline - Check Difference

Work on the part with the red circle (1) following instructions by yourself.  
The part with the black square (1) is an additional explanation or a reference.

- 1 To identify the difference between **the centerline being sketch type** and **the centerline being annotation type**, click dimensions and check Feature Manager window.
- 2 Click **[Leaders]** tab at Feature Manager window.
- 3 If users compare **[Witness/Leader Display]** field, users could verify the presence of the following marks, **[Radius]** ( - Currently activated with marking dark gray),  **[Diameter]** ( - Currently deactivated with marking bright gray) .

Users would also see different options for each

- 4 notation created in different ways when users right-click on annotations.

