How to rotate part in CATIA with the compass and a defined axis

This guide shows you how to rotate a part in CATIA V5 with the compass. You first create an axis in Generative Shape Design, snap the compass onto it, and then rotate the part by an exact increment. Using a defined axis keeps the rotation accurate, which matters most in large assemblies or when you work to millimeter precision.

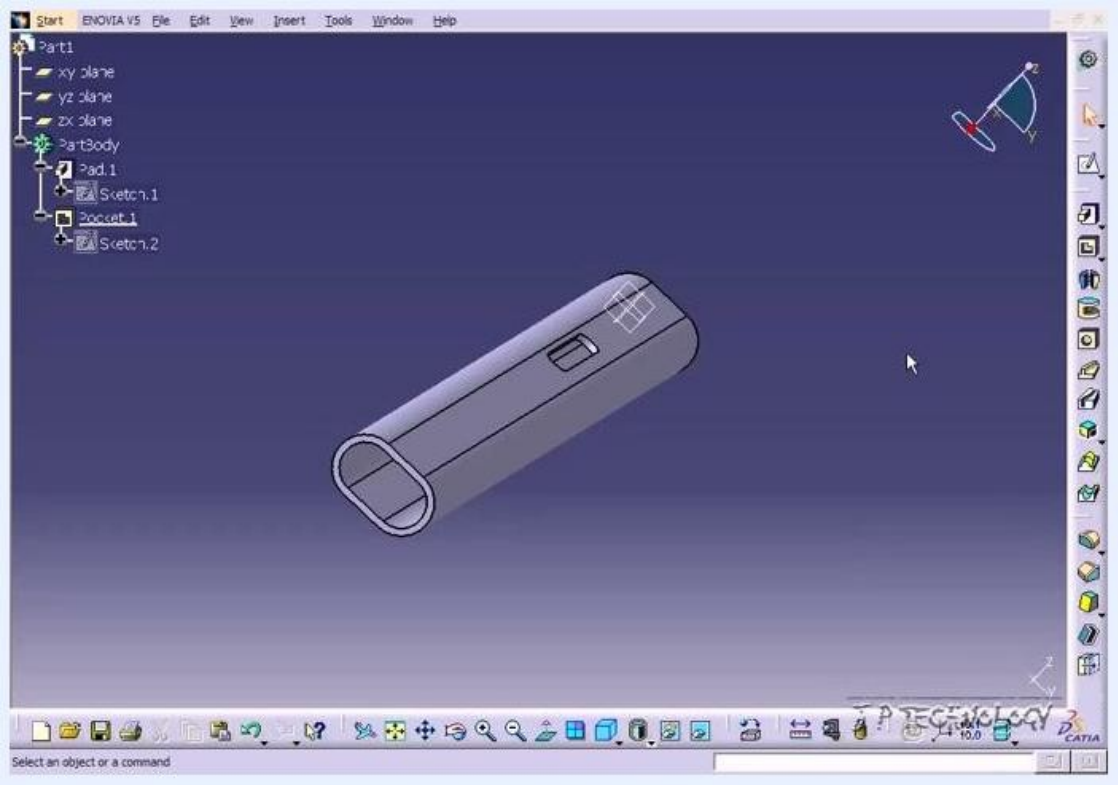
Prerequisites

- CATIA V5 installed and running.
- Tutorial part number 87 from the DVD.
- Access to the **Generative Shape Design** workbench.

Steps

1. Open the tutorial part

Open CATIA V5 and load tutorial part number 87 from the DVD. The model tree appears on the left and the 3D part appears in the center of the workspace. The compass is in the upper right corner of the workspace.



CATIA V5 interface with tutorial part 87 loaded, showing the model tree, 3D part, and compass

2. Decide where the rotation axis should be

Before you rotate the part in CATIA, decide where you want to rotate it, for example along a specific line or axis. Then plan to create an axis at that location. This way you are 100% certain about the rotation axis.

Placing the compass on different faces or locations gives different rotation results. This is true for the W axis or for a spot near the center of a circle. Even if you think you are at the center, you may not be exactly there, because the compass can still move slightly.

To get a precise result, create geometry such as a line or axis within your circles or curves. Always use a straight line or axis for compass rotation.

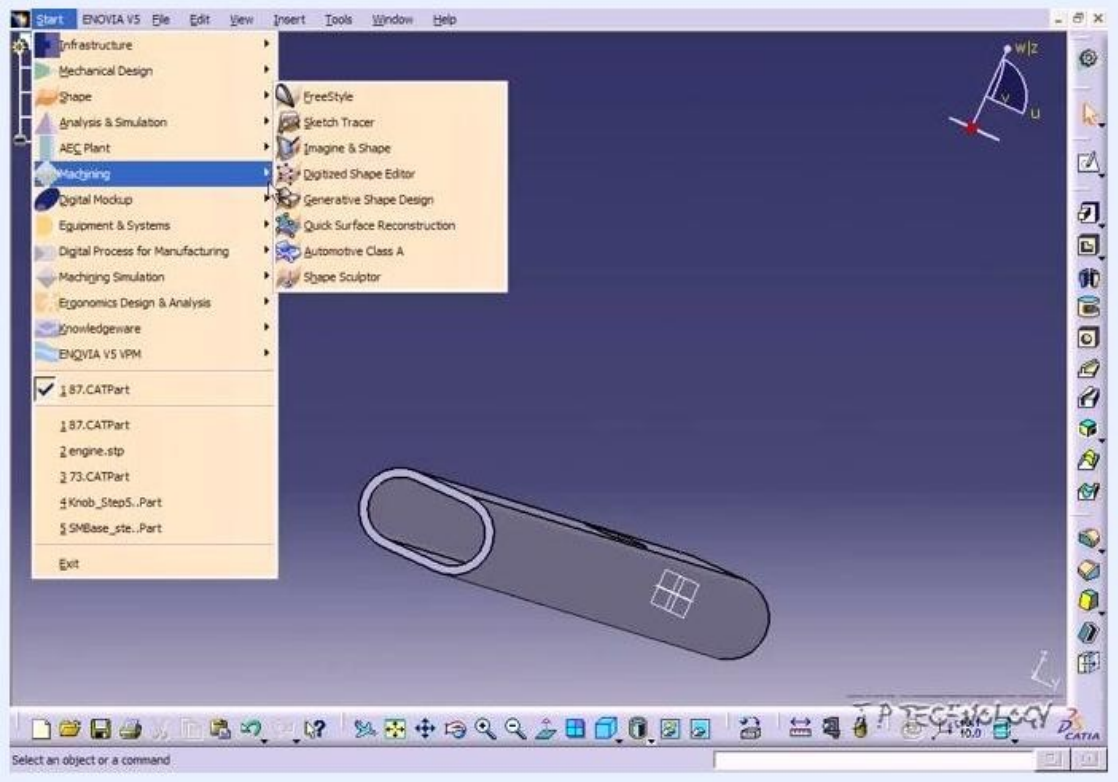


Close-up of the part with the compass being positioned and axis creation being considered

3. Switch to Generative Shape Design

1. In the top menu, click **Start**.
2. Go to **Shape**.
3. Select **Generative Shape Design** from the dropdown menu.

This module lets you create reference geometry, such as lines and axes, for precise operations.



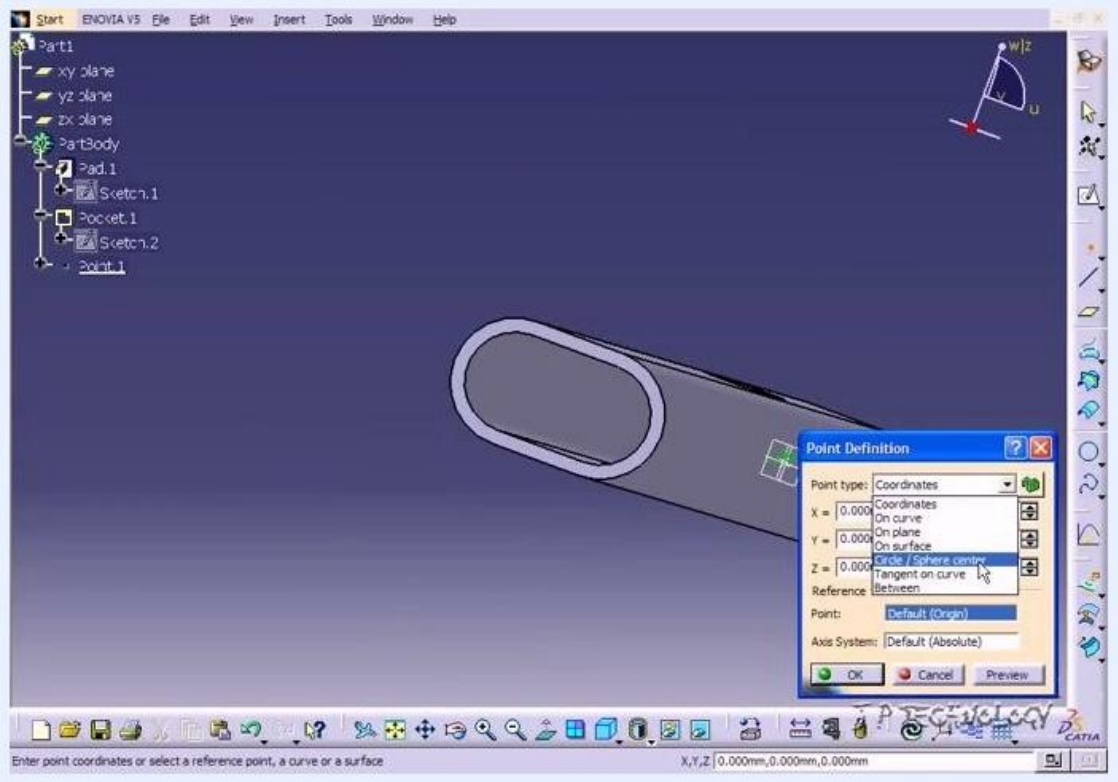
CATIA V5 menu open with Start > Shape > Generative Shape Design highlighted

4. Create a point at the center of each end curve

You create the axis so that it runs through the center of the part. Start with a point at the center of each end curve.

1. Click the **Point** tool in the toolbar.
2. In the **Point Definition** dialog, change **Point type** to **Circle/Sphere center**.
3. Click the first end curve of the part. CATIA automatically places a point at the center of the selected curve.
4. Click **OK** to create the first point.

Repeat the process for the opposite end. Click the **Point** tool again, select the other end curve, make sure **Circle/Sphere center** is selected, and click **OK** to create the second point.

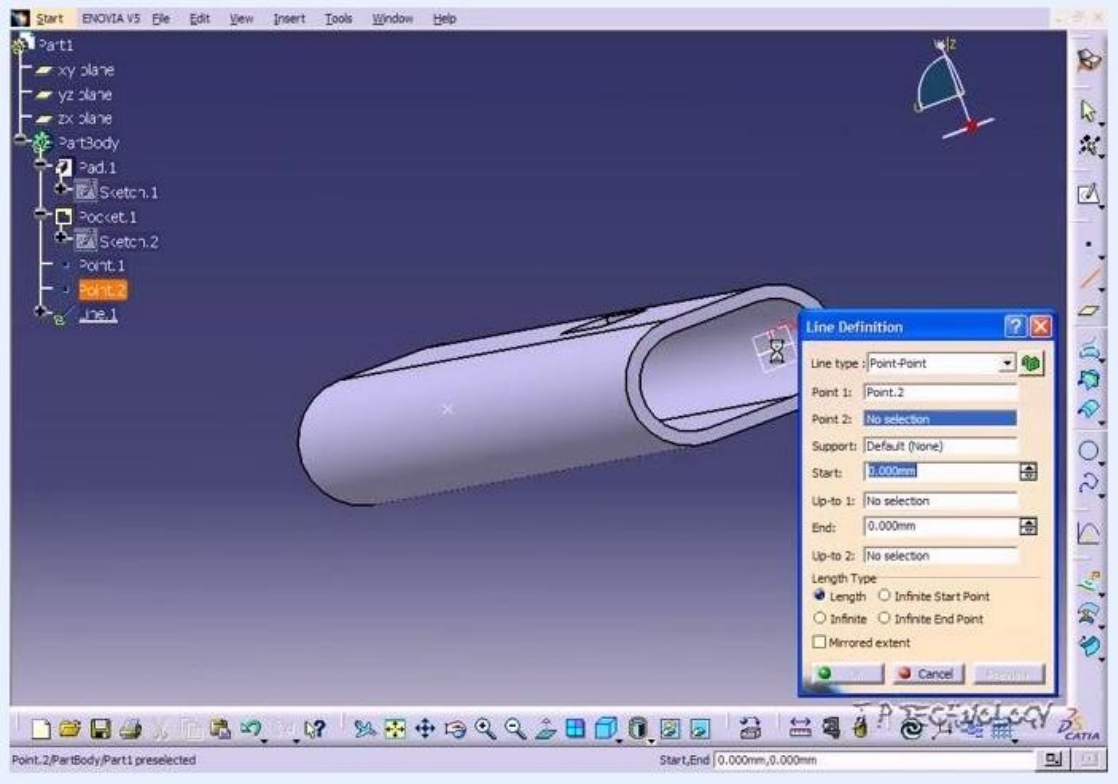


Point Definition dialog open, Circle/Sphere center selected, and first end curve selected for point placement

5. Create the axis line between the two points

1. Click the **Line** tool in the toolbar.
2. In the **Line Definition** dialog, set **Line type** to **Point-Point**.
3. For **Point 1**, select the first point you created.
4. For **Point 2**, select the second point.

5. Click **OK** to create the line (axis) between the two points.

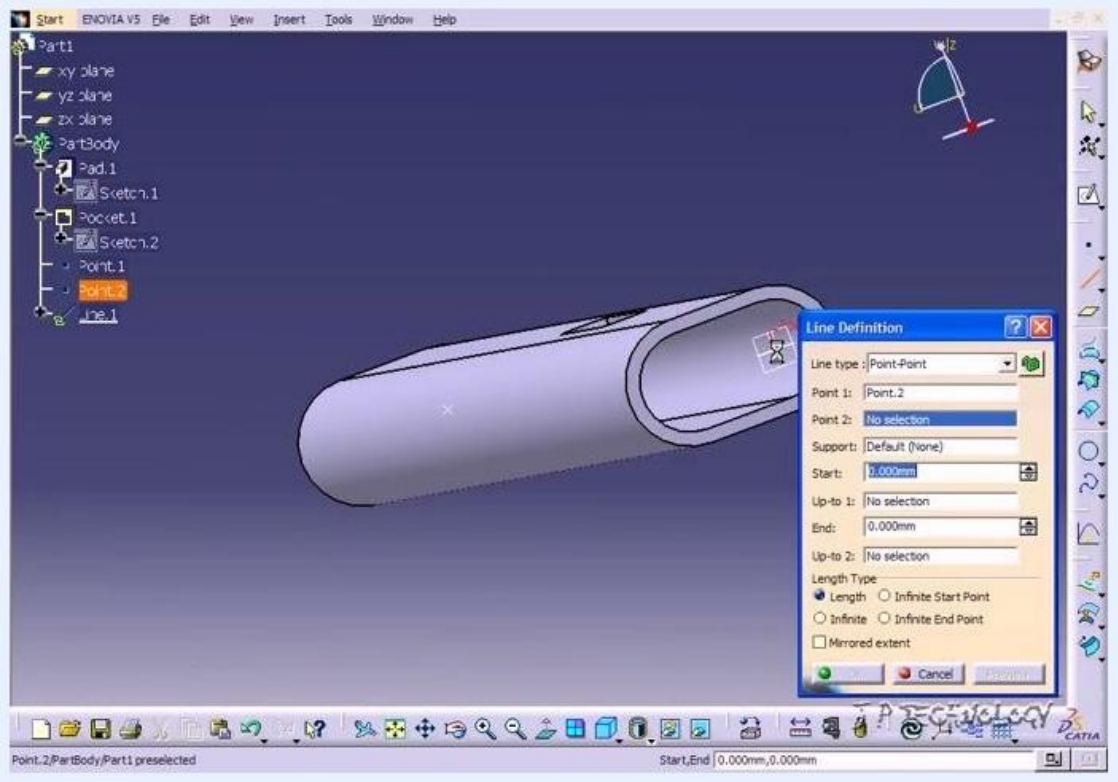


Line Definition dialog open, with Point 1 and Point 2 selected, creating the axis line

6. Verify the axis

The new line runs through the part from end to end. Switch to a front view to check the axis alignment more easily.

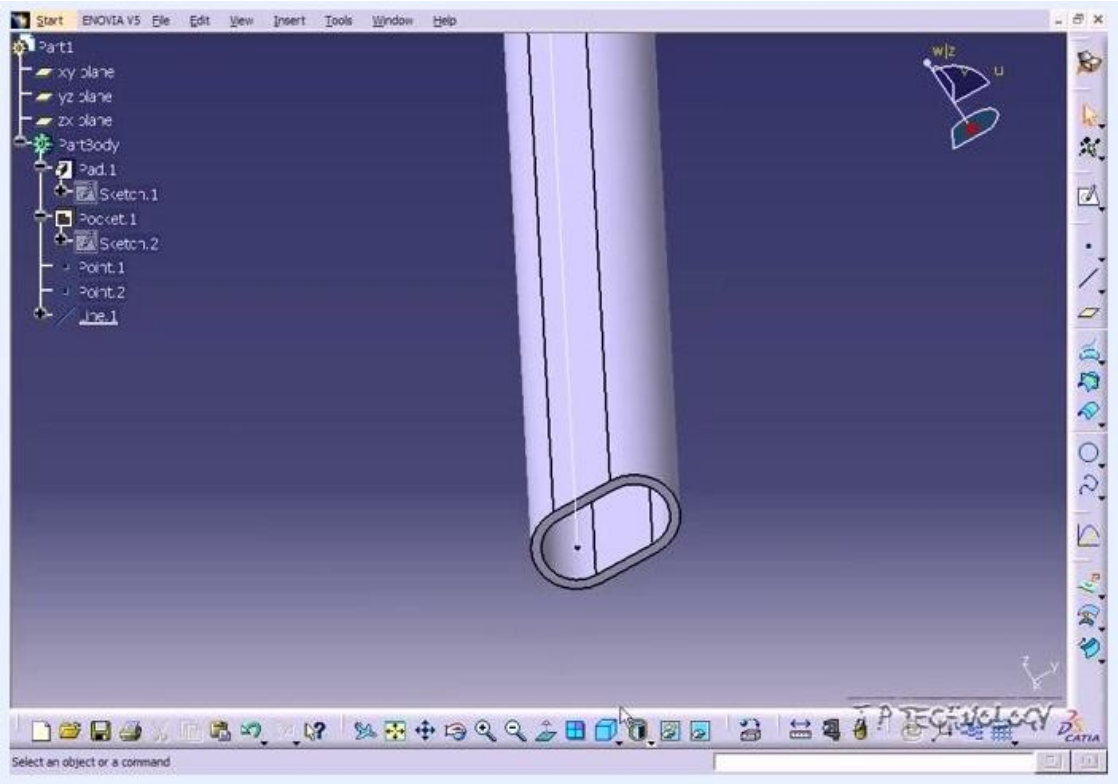
If you place the compass at various locations without using the axis line, you may get different rotation results.



Front view of the part showing the axis line running through the center

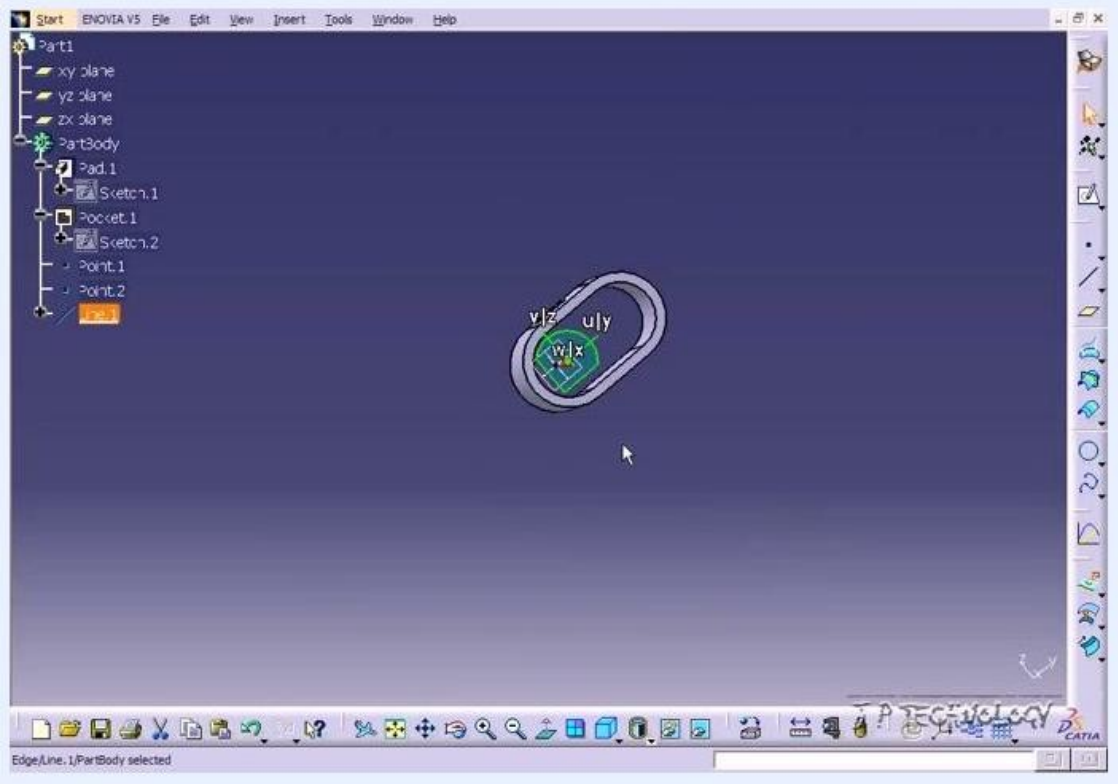
7. Snap the compass onto the axis

Drag the compass and snap it directly onto the axis line you created. The compass is now constrained to move only along the axis line. Because it cannot move off the line, the rotation about your chosen axis is 100% accurate.



Compass snapped to the axis line at the end of the part, ready for precise rotation

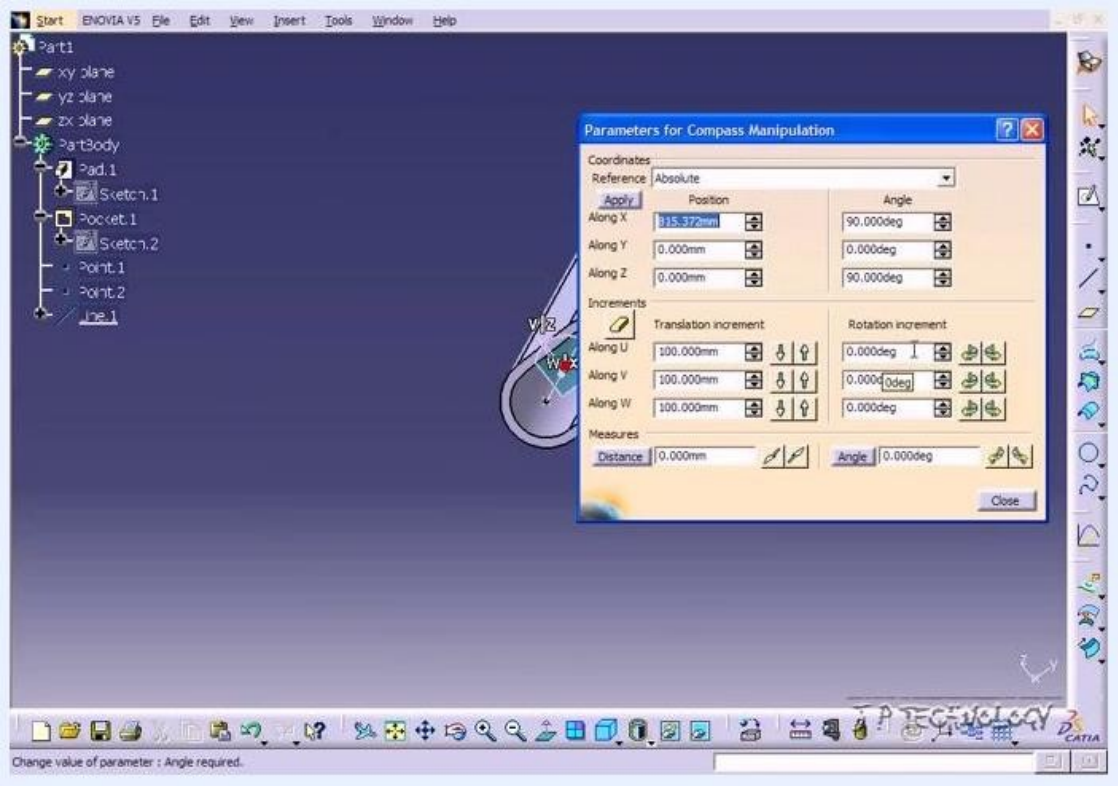
For small parts, minor inaccuracies may not be noticeable. For large assemblies or millimeter-level precision, defining the axis is critical. Always create and use an explicit axis for compass rotation to avoid cumulative errors in complex designs.



Compass and axis line shown on the part, emphasizing the importance of precise axis placement

8. Highlight all features you want to rotate

Select the entire part and all its features, either in the specification tree or directly in the 3D view. Make sure all relevant elements are highlighted before you continue. The compass should still be snapped to the axis line.



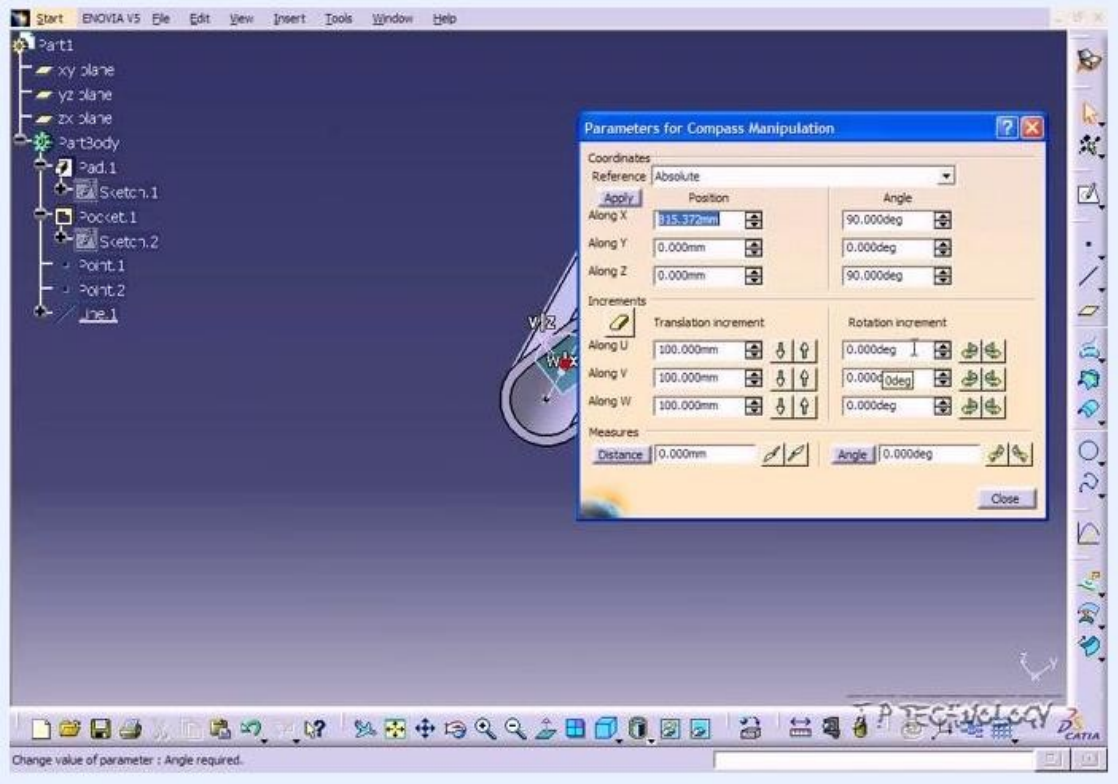
Part and features highlighted in the CATIA V5 workspace, ready for rotation, with the compass snapped to the axis line

9. Open the compass rotation parameters

To start rotating the part in CATIA, use either of these options:

- Click the plane you want to rotate around.

- Double-click the compass to open the rotation parameters dialog.

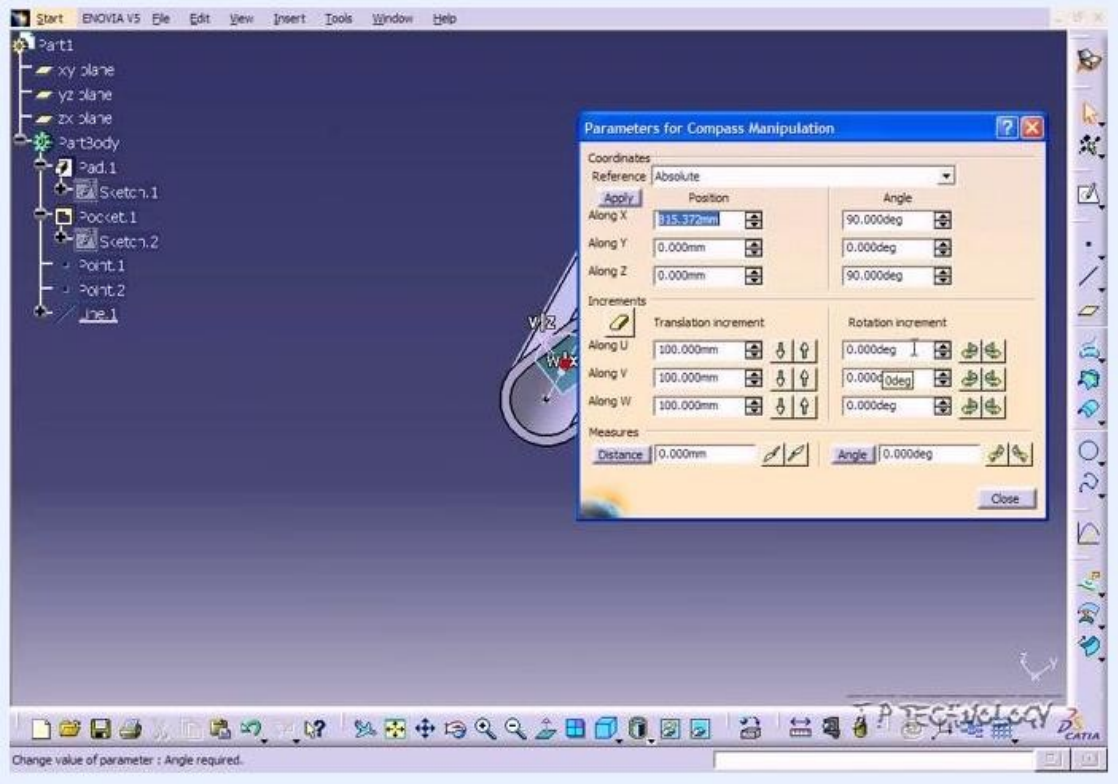


Part with compass attached and ready for rotation, with the part and axis line visible in the workspace

10. Set the rotation increment

1. In the **Parameters for Compass Manipulation** dialog, find the **Rotation increment** section.
2. In the field next to the W axis, enter 15.000deg.

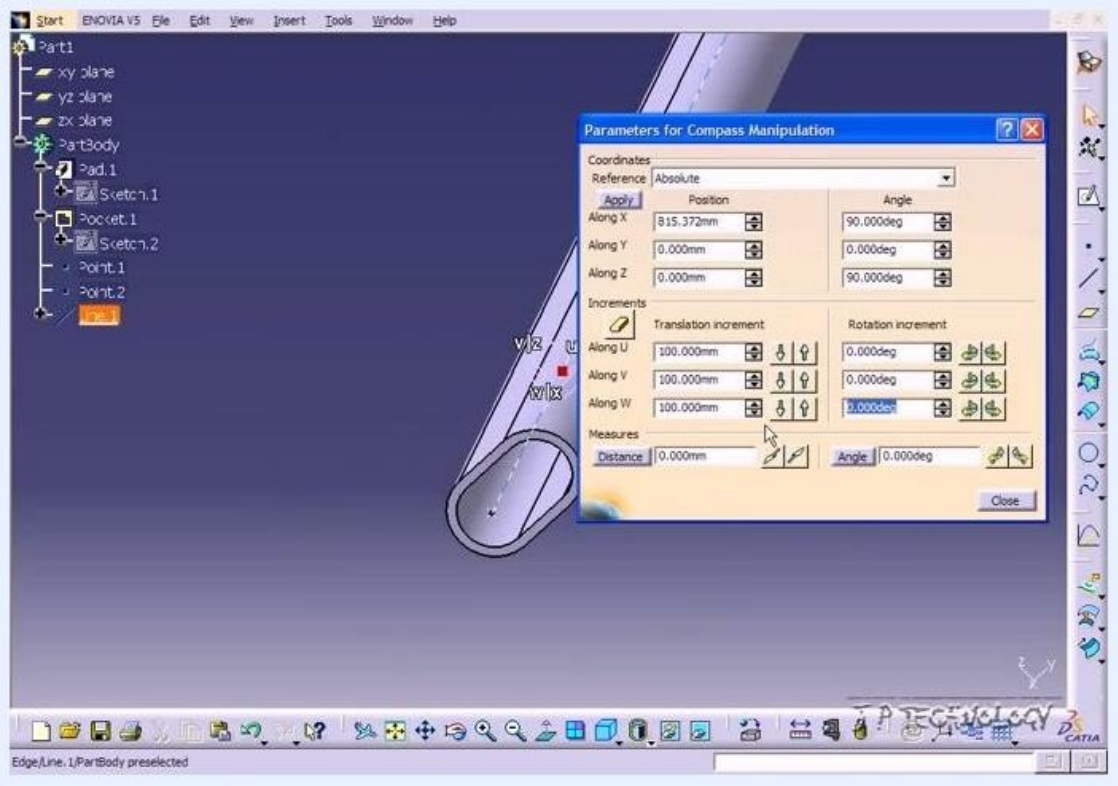
The part will now rotate exactly 15 degrees around the defined axis.



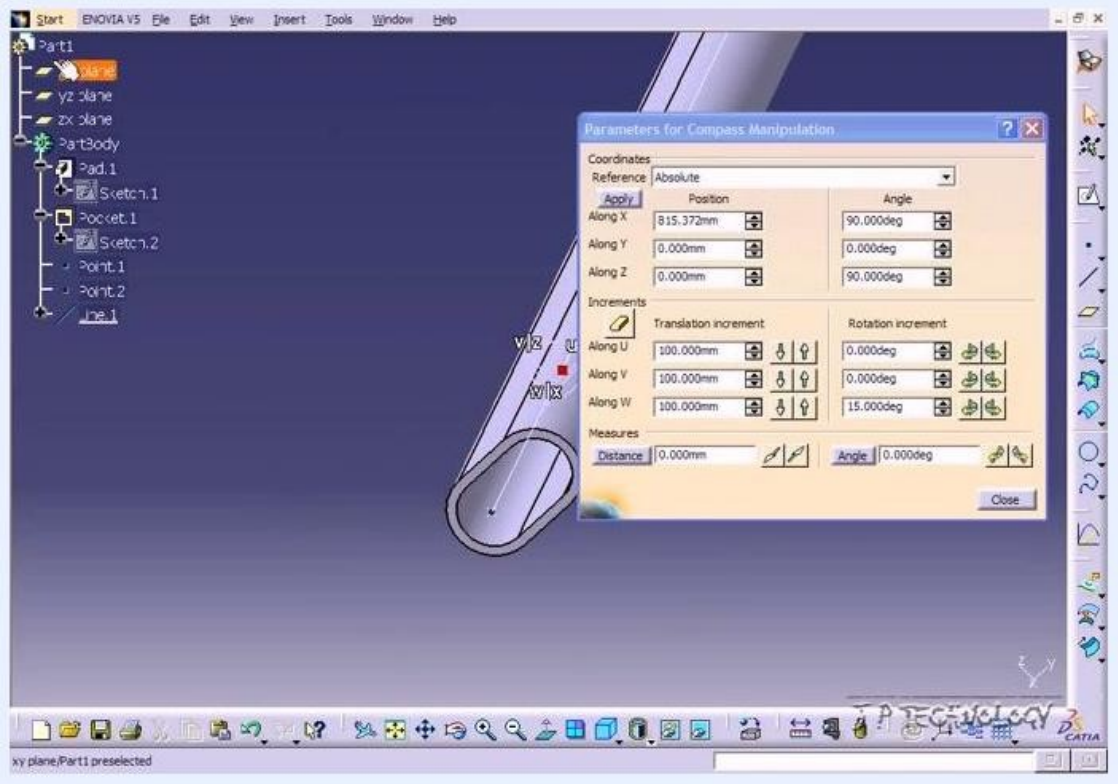
Parameters for Compass Manipulation dialog open, with 15.000deg entered for the W axis rotation increment

11. Select the part body and all features

Click **PartBody** in the specification tree to select the main part. Make sure all features, such as pads, sketches, and points, are also selected so the whole part rotates together.



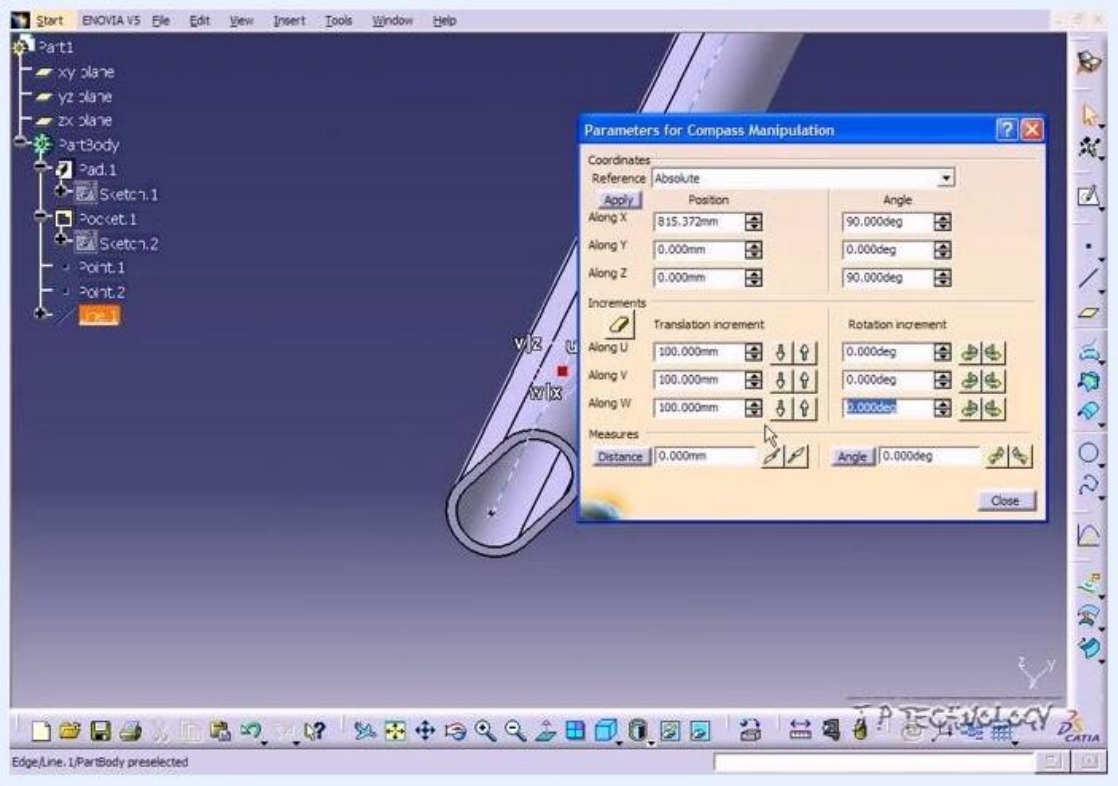
PartBody and features selected in the tree, with the Parameters for Compass Manipulation dialog still open



The xy plane selected in the tree, with the Parameters for Compass Manipulation dialog still open

12. Rotate the part

Click the rotation button (circular arrow) in the **Parameters for Compass Manipulation** dialog to apply the rotation. The part rotates exactly 15 degrees around the axis you defined.



Part rotated 15 degrees around the axis, with all features highlighted and the Parameters for Compass Manipulation dialog visible

13. Verify the rotation

Confirm that the part rotated as intended and that all features keep their correct positions relative to the axis. The axis ensures 100% accuracy and prevents misalignment or errors.



Part and features rotated, with the axis and compass still visible and the Parameters for Compass Manipulation dialog open

Creating the axis is what makes the rotation accurate.



Emphasis on the importance of axis creation for accurate rotation, with the Parameters for Compass Manipulation dialog open

If you don't use an axis, you may get mixed results.



Warning about mixed results if an axis is not used, with the Parameters for Compass Manipulation dialog open

Summary

- Always create and use a defined axis for any rotation operation in CATIA V5.
- Use the **Point** tool with **Circle/Sphere center** to place points at the centers of the end curves. Then use the **Line** tool to connect them.
- Snap the compass to the axis so it cannot move off the line.
- Use the **Parameters for Compass Manipulation** dialog to control rotation increments precisely.
- Select all relevant features so the entire part rotates as a unit.
- Double-check the result, especially in complex assemblies.

What's next

Keep using the compass along the defined axis for every rotation operation. This maintains accuracy throughout your design process and keeps your parts positioned exactly as required.