How to find the center of gravity in CATIA V5

You can find the center of gravity and mass of any part, sub-assembly, or full assembly in CATIA V5 with the **Measure Inertia** tool. This guide shows how to find the center of gravity in CATIA by building a simple assembly of two cylindrical parts and measuring it. The same technique applies to larger products, such as an automotive body.

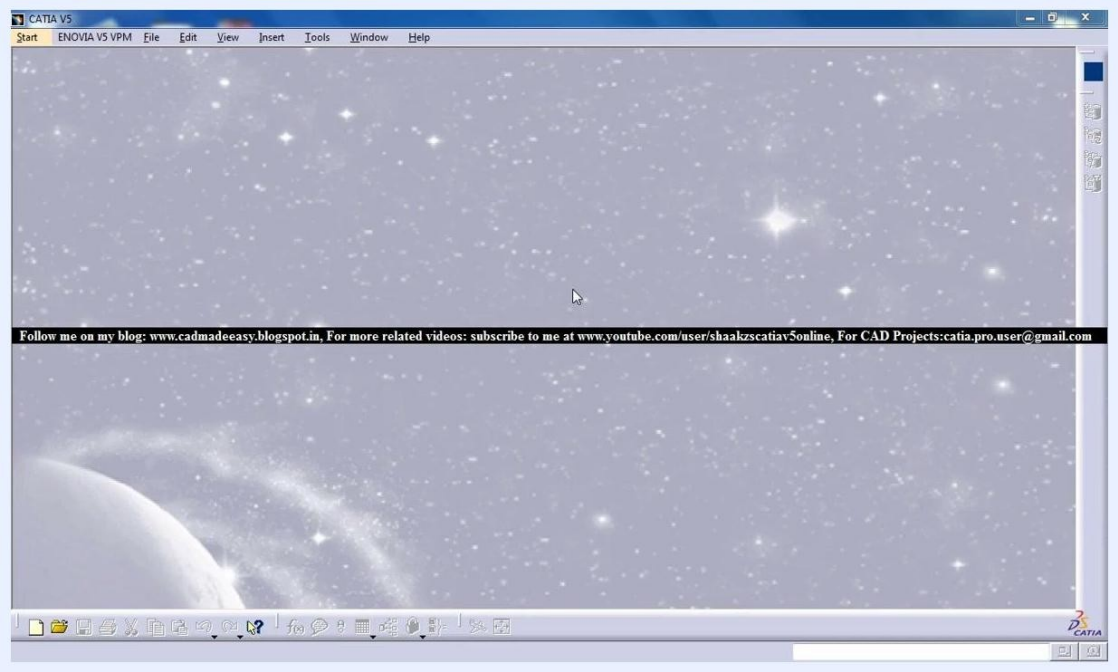
Prerequisites

Before you begin, make sure you have:

- CATIA V5 installed on your computer.
- A decision on whether to work with a single part or an assembly. This guide uses the assembly (top-down) approach.

Step 1: Launch CATIA V5

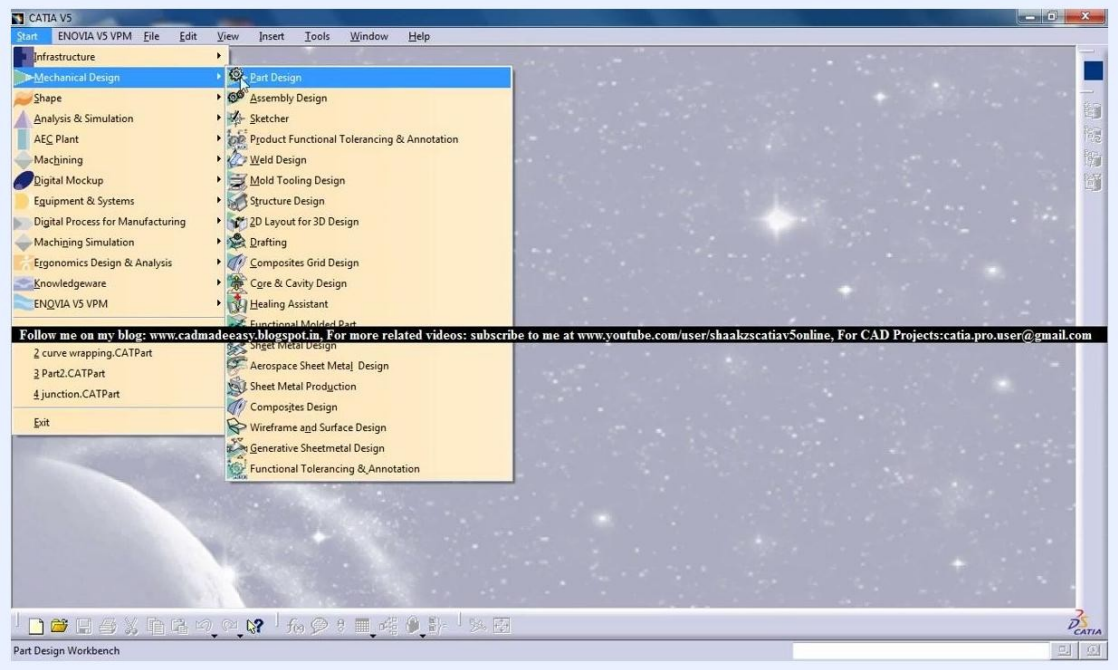
Open CATIA V5 and wait for the main interface to load. The background shows a starry sky, and a banner at the bottom links to additional resources.



CATIA V5 main interface with a starry background and a resource banner

Step 2: Start a new assembly design

1. Move your cursor to the top menu and click **Start**.
2. Hover over **Mechanical Design** in the dropdown menu.
3. Select **Part Design** to create a single part. To follow the top-down approach in this guide, select **Assembly Design** instead.

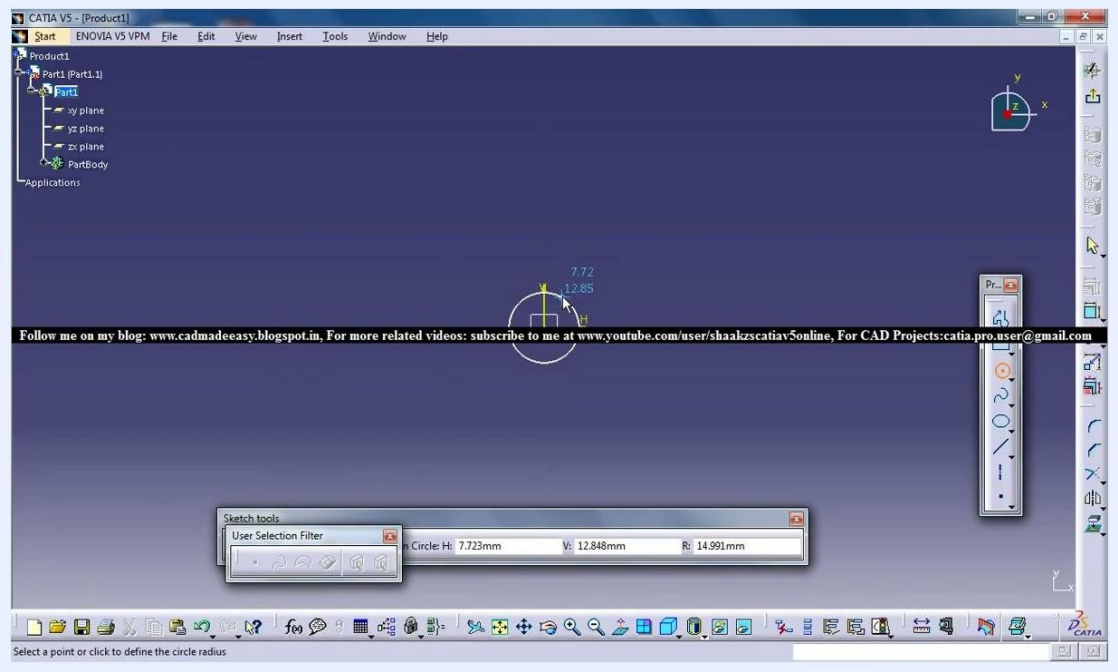


The Start > Mechanical Design dropdown menu showing Part Design and other options

Step 3: Insert a part and sketch the first cylinder

1. In the assembly workbench, go to the **Insert** menu, choose **New Part**, and insert it under the current product.
2. Confirm that the new part appears in the product tree on the left side of the screen.
3. Select the new part (for example, Part1) in the product tree.
4. Enter the sketcher workbench.
5. Draw a circle to define the base of the cylinder. Use the sketch tools at the bottom of the screen.
6. Specify the circle's dimensions. In this example, H is 7.723 mm and R is 14.991 mm.

You give the cylinder its height in a later step.

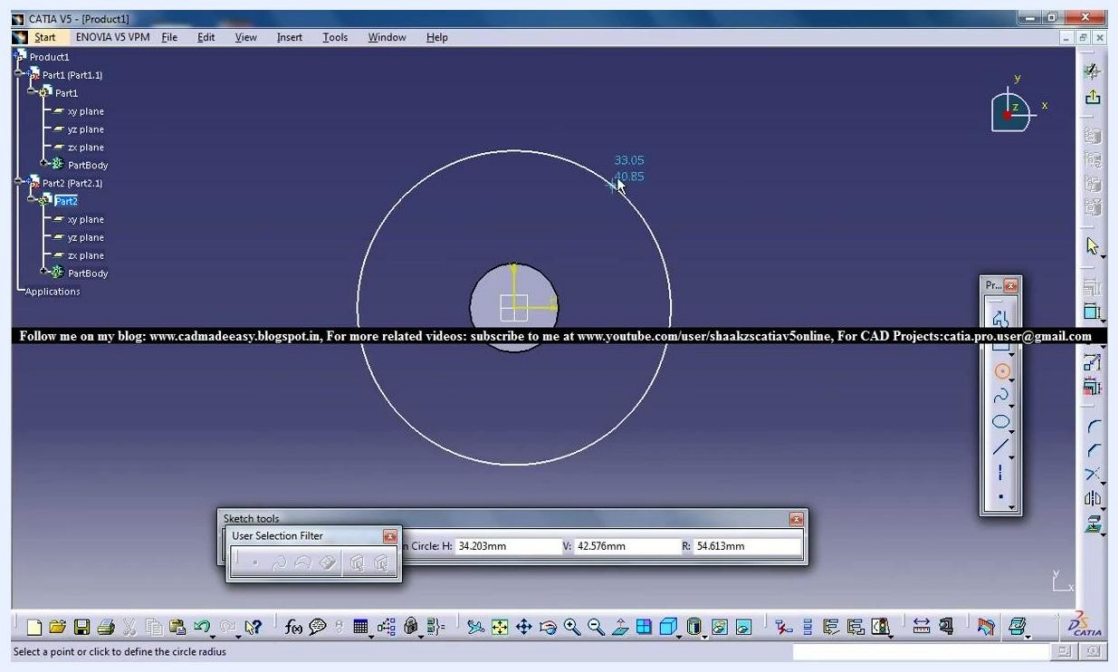


The sketcher workbench with a circle being drawn, showing dimensions and sketch tools

Step 4: Add a second part

1. Exit the current part workbench to return to the assembly workbench. The toolbars and available tools change when you switch workbenches.
2. Select **Insert** > **New Part**.
3. Confirm that the new part (for example, Part2) appears in the product tree under Product1.

4. When the "New Part: Origin Point" dialog appears, click **Yes** to define a new origin point for the new part.



The product tree showing Part1 and the new Part2, with the dialog for defining the origin point

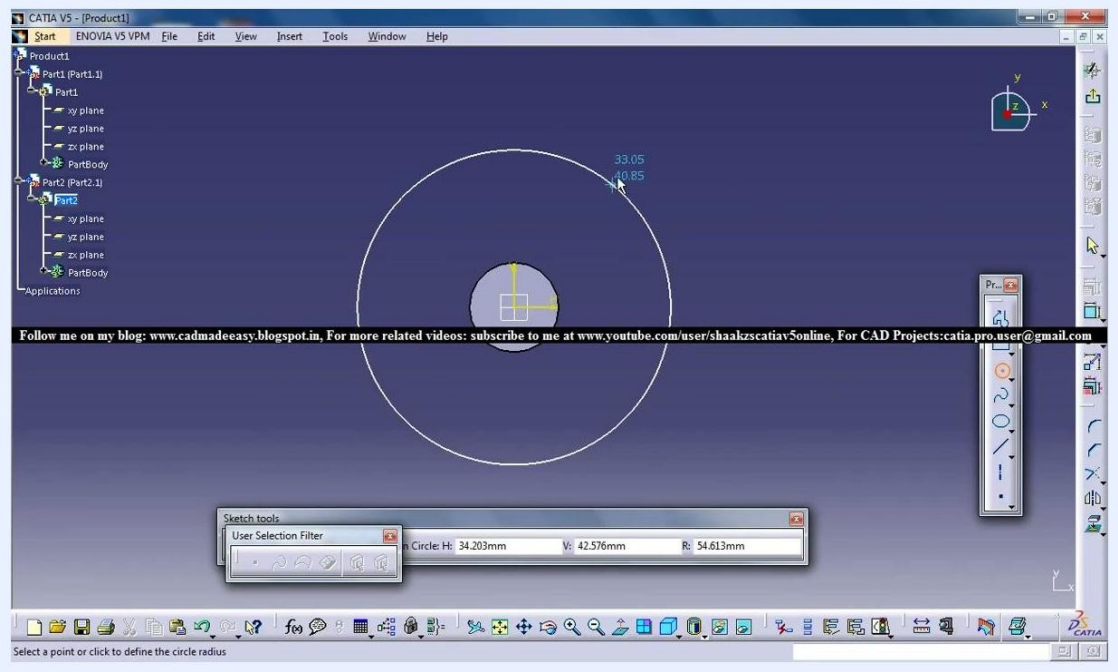
Step 5: Sketch the second cylinder

1. Enter the part workbench for Part2.
2. Begin a new sketch on the plane you want, for example the xy plane.
3. Draw a circle to define the base of the second cylinder.
4. Specify the circle's dimensions, then complete the sketch and exit the sketcher.

The source gives two example sets of values:

- Center at H: 1.419 mm, V: 0 mm, Radius: 0 mm initially, then adjusted as needed.

- Circle at H: 34.203 mm, V: 42.576 mm, R: 54.613 mm.

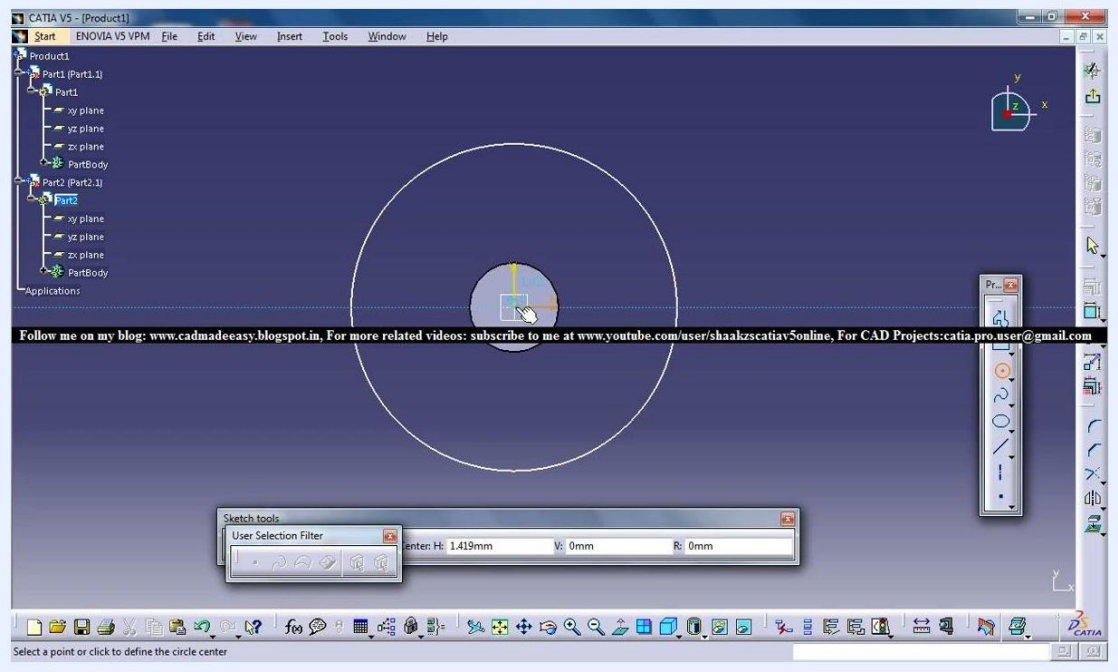


The sketcher workbench for Part2 showing a large circle with its dimensions and the sketch tools

Step 6: Pad the second cylinder

1. Use the **Pad** tool to extrude the circle into a cylinder.
2. In the **Pad Definition** dialog, set the length (height) of the cylinder. In this example, Length is 20 mm.
3. Make sure the correct profile is selected (for example, Sketch.1).

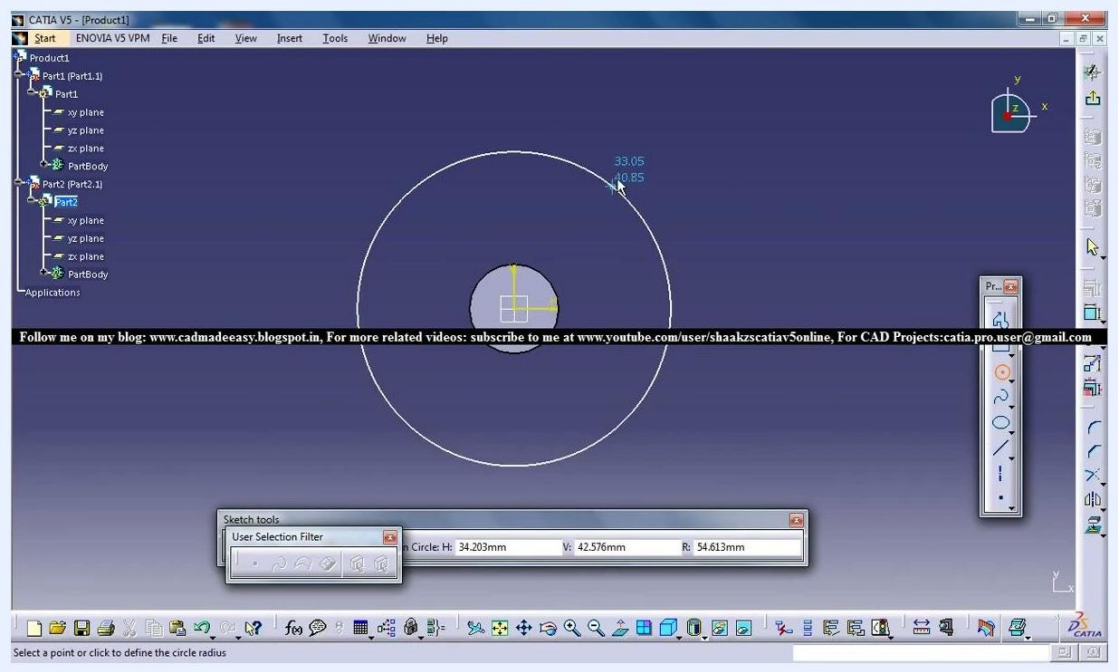
4. Click **OK** to create the pad.



The Pad Definition dialog for Part2 with Length set to 20 mm and the new cylinder being created

Step 7: Review the assembly

Rotate and inspect the assembly to confirm that both cylinders (Part1 and Part2) are present and correctly positioned. The product tree now shows both parts, each with its own body and planes.

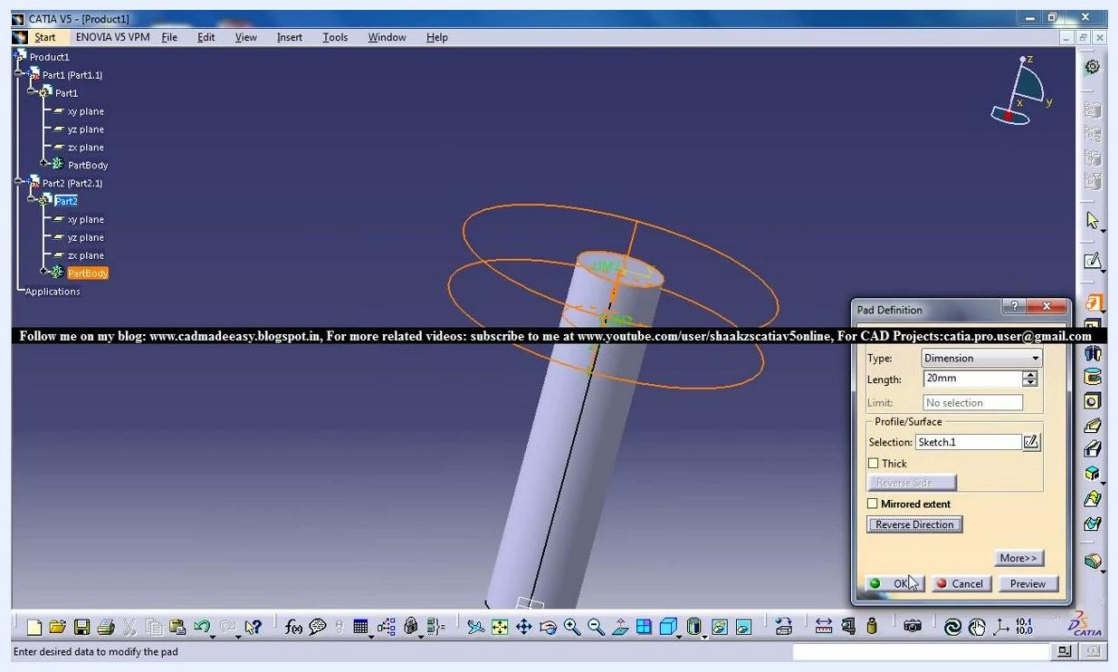


The assembly view showing both cylinders, Part1 and Part2, in the product tree and in 3D space

Step 8: Open the Measure Inertia tool

1. Locate the **Measure Inertia** tool in the **Measure** toolbar.
2. Click the **Measure Inertia** icon, a blue cube with a yellow arrow.

3. Get ready to select the part or assembly whose center of gravity and mass you want to find.



The Measure Inertia tool highlighted in the toolbar with the completed assembly visible

Step 9: Find the center of gravity and mass of the assembly

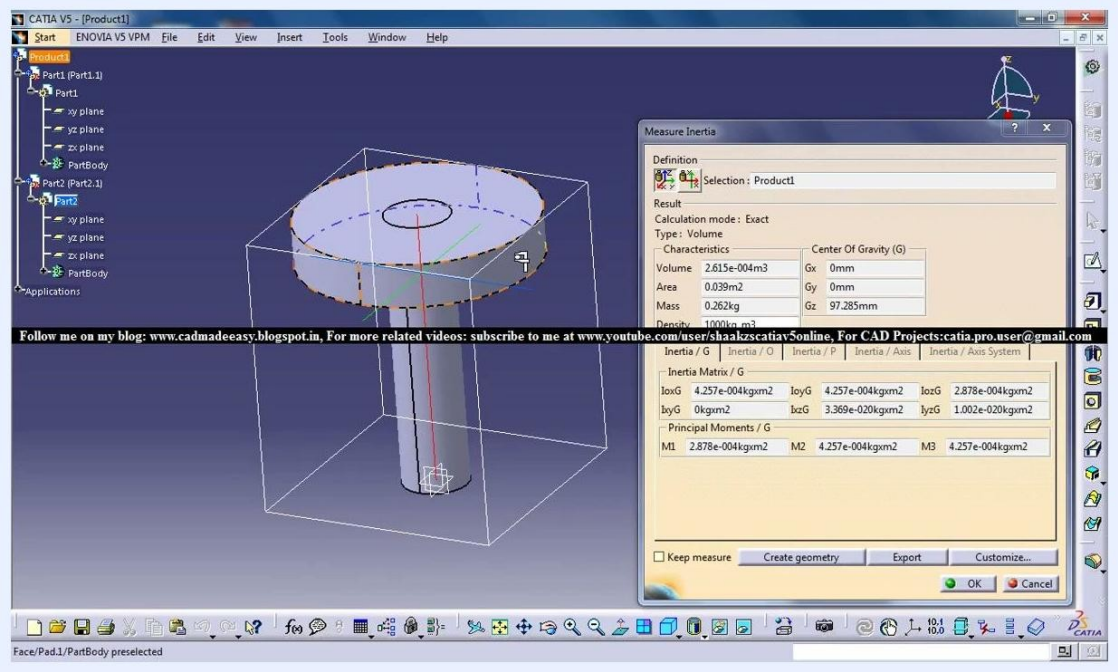
You can use this technique on any part, sub-assembly, or entire assembly. The **Measure Inertia** tool offers both 2D and 3D inertia calculations. This guide uses the 3D option.

- To measure a single part, click that part (for example, Part1 or Part2) in the product tree.
- To measure the entire assembly, click the top-level product node (for example, Product1). This analyzes the combined properties of all included parts.

To measure the assembly:

1. With the **Measure Inertia** dialog open, make sure **Measure Inertia 3D** is selected.
2. Click Product1 in the product tree.

3. Look at the 3D view. A bounding box appears around the assembly, and a cross marks the calculated center of gravity.



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The Measure Inertia dialog with Product1 selected, showing the bounding box, the cross at the center of gravity, and detailed results

Step 10: Read the results

The **Measure Inertia** dialog displays these results for the assembly:

- **Center of gravity (G):** Gx = 0 mm, Gy = 0 mm, Gz = 97.285 mm
- **Mass:** 0.262 kg
- **Density:** 1000 kg/m³ (the default value used for the calculation)
- **Volume:** 2.615e-004 m³
- **Area:** 0.039 m²

The dialog also shows the inertia matrix and principal moments for advanced analysis.

The cross in the 3D view marks the exact location of the center of gravity. The coordinates (0, 0, 97.285) give that position in the assembly's coordinate system. The mass is calculated from the default density and the combined volume of the parts. You can use these results for further engineering analysis, design validation, or documentation.

Summary

You built an assembly of two cylindrical parts and used the **Measure Inertia** tool to find its center of gravity and mass in CATIA V5. The technique is quick and works on any part or assembly where you need mass properties, including automotive bodies and other mechanical assemblies.

What's next

- Adjust the density value in the part properties if you need results for materials other than the default.
- Select individual parts, sub-assemblies, or the top-level product to compare their centers of gravity and mass.