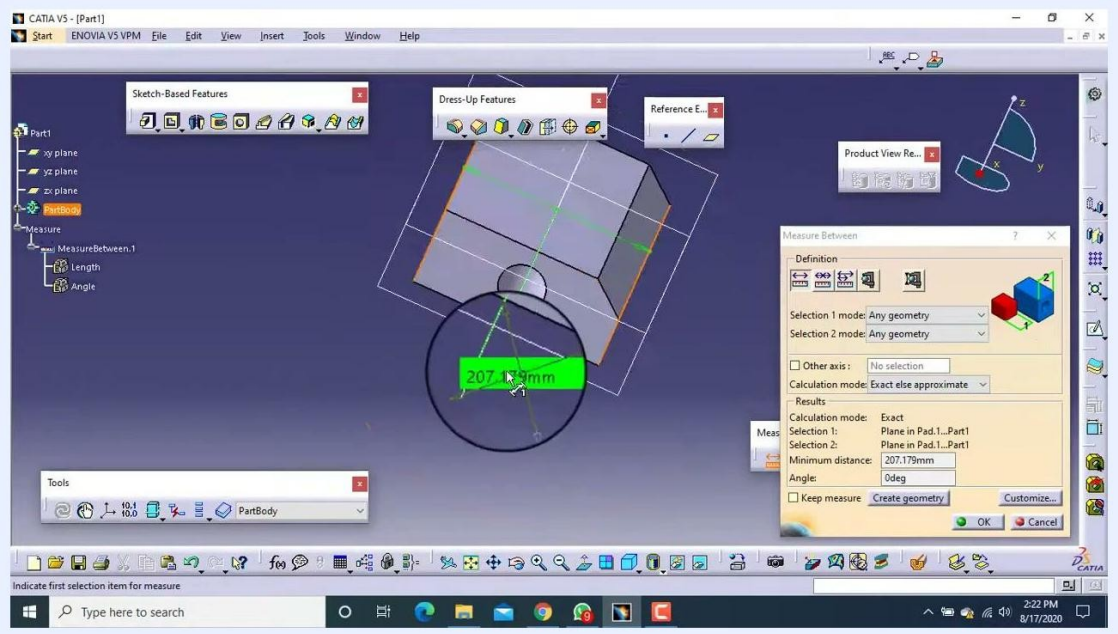
How to measure dimensions in CATIA V5: distances, angles, radii, and inertia

This guide shows you how to measure dimensions in CATIA V5 on a 3D part. You use the **Measure Between** tool for distances and angles, **Measure Item** for the radius and center of a circular edge, and **Measure Inertia** for the part's physical properties. Each dialog shows its results and highlights the measurement on the model in green.

Prerequisites

Before you start, make sure the following are in place:

- The part file is loaded in CATIA V5.
- The workspace shows the part with its planes (xy plane, yz plane, zx plane) and PartBody in the tree view.
- The toolbars **Sketch-Based Features**, **Dress-Up Features**, **Reference Elements**, and **Product View** are accessible.
- The **Tools** palette is visible at the bottom of the window.



CATIA V5 workspace with the part loaded and the toolbars and planes visible

No measurements have been performed yet at this point.

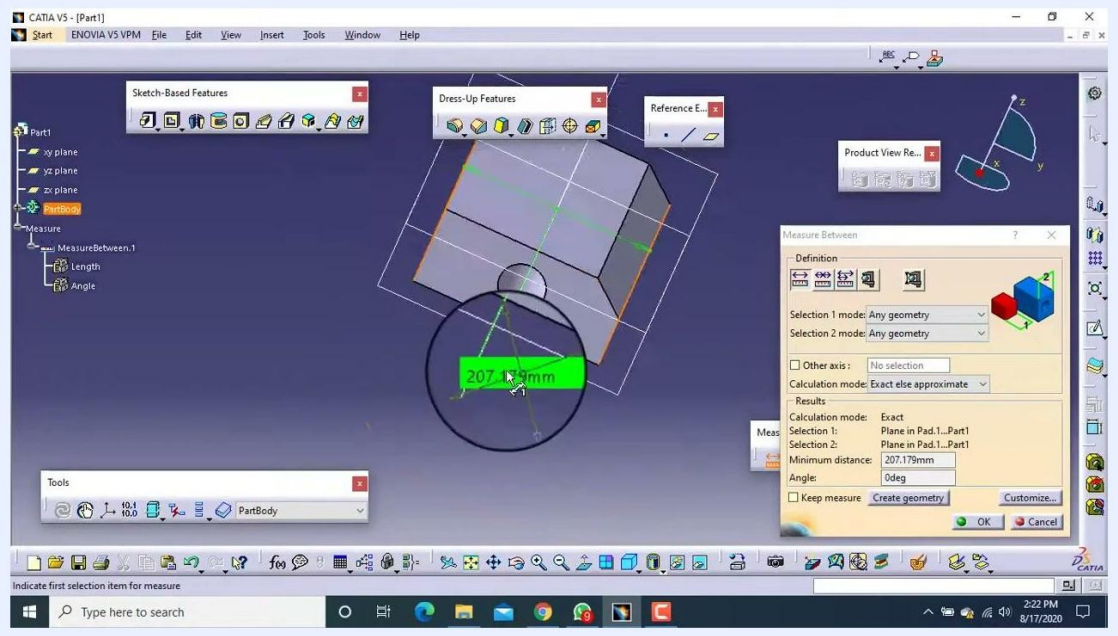
Step 1: Start the Measure Between tool

In the **Measure** toolbar, click **Measure Between**. Its icon shows two arrows pointing left and right. The cursor highlights the tool as you select it.

Step 2: Measure the distance between two planes

Select two planes on the part to measure the minimum distance between them. The **Measure Between** dialog appears with these values:

- **Selection 1 mode:** Any geometry
- **Selection 2 mode:** Any geometry
- **Calculation mode:** Exact
- **Selection 1:** Plane in Pad.1...Part1
- **Selection 2:** Plane in Pad.1...Part1
- **Minimum distance:** 207.179 mm
- **Angle:** 0 deg

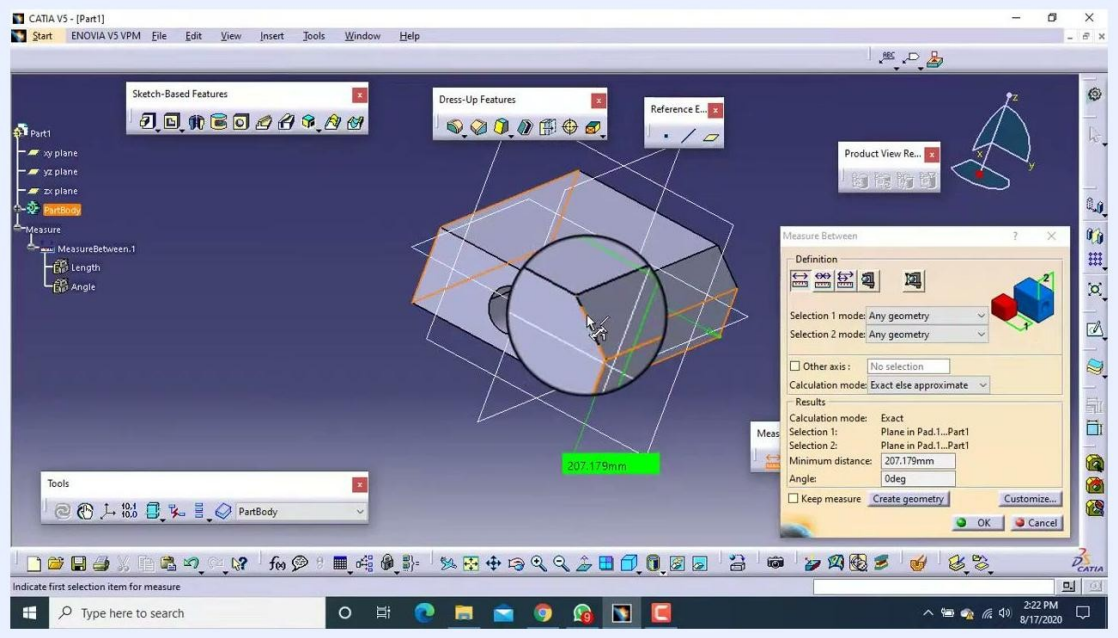


Measure Between dialog open with the 207.179 mm distance shown in green and two planes selected

The measured distance appears in green on the model.

Step 3: Rotate the model to inspect the measurement

Rotate the 3D model to see the measured distance from another angle. The selected planes and the 207.179 mm measurement stay highlighted. The **Measure Between** dialog stays open, so you can adjust or confirm the measurement.



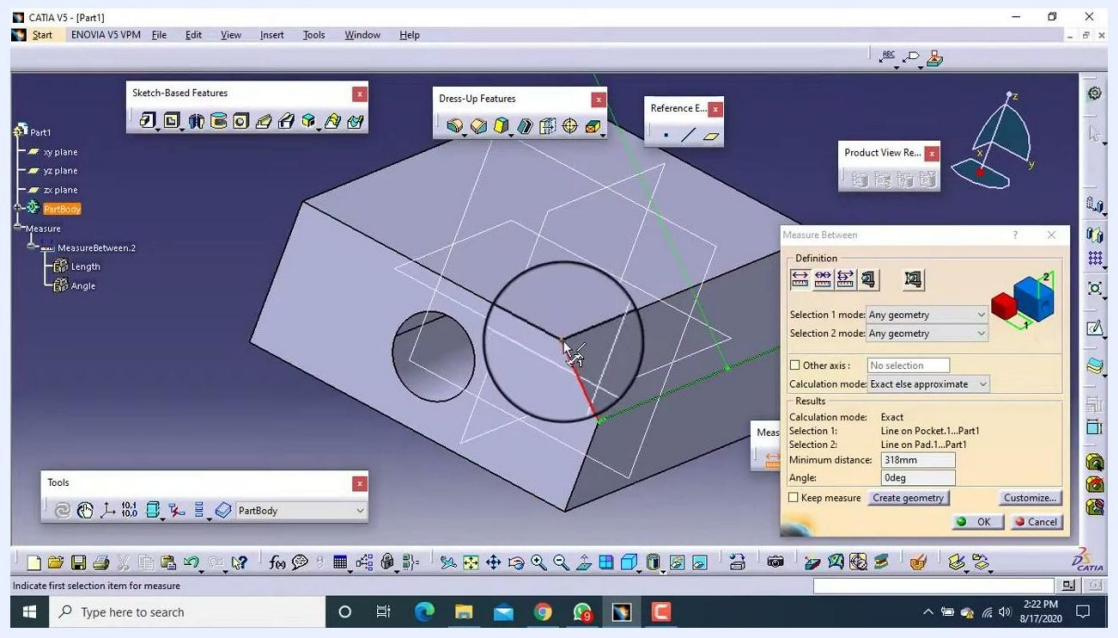
Rotated view of the part with the 207.179 mm measurement highlighted

Step 4: Measure the distance between two lines

Select a line on the pocket feature and a line on the pad feature. The dialog updates with these values:

- **Selection 1:** Line on Pocket.1...Part1
- **Selection 2:** Line on Pad.1...Part1
- **Minimum distance:** 318 mm

- **Angle:** 0 deg



Line-to-line measurement with the 318 mm distance shown in green

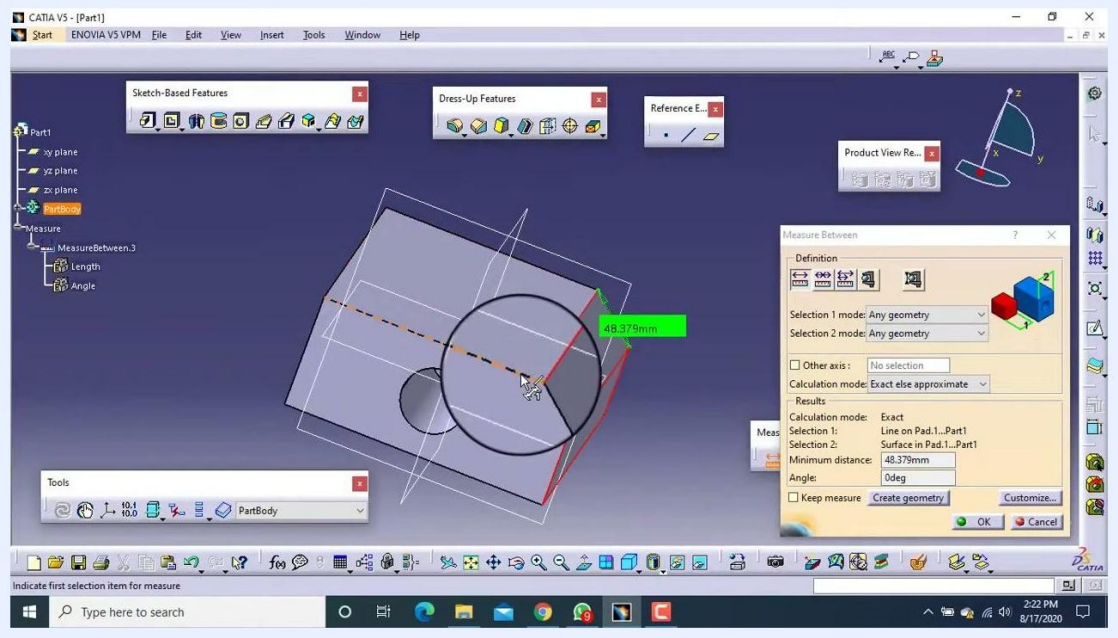
The distance appears in green on the model.

Step 5: Measure the distance between a line and a surface

Select a line on the pad and a surface on the pad. The dialog updates with these values:

- **Selection 1:** Line on Pad.1...Part1
- **Selection 2:** Surface in Pad.1...Part1
- **Minimum distance:** 48.379 mm

- **Angle:** 0 deg



Line-to-surface measurement with the 48.379 mm distance shown in green

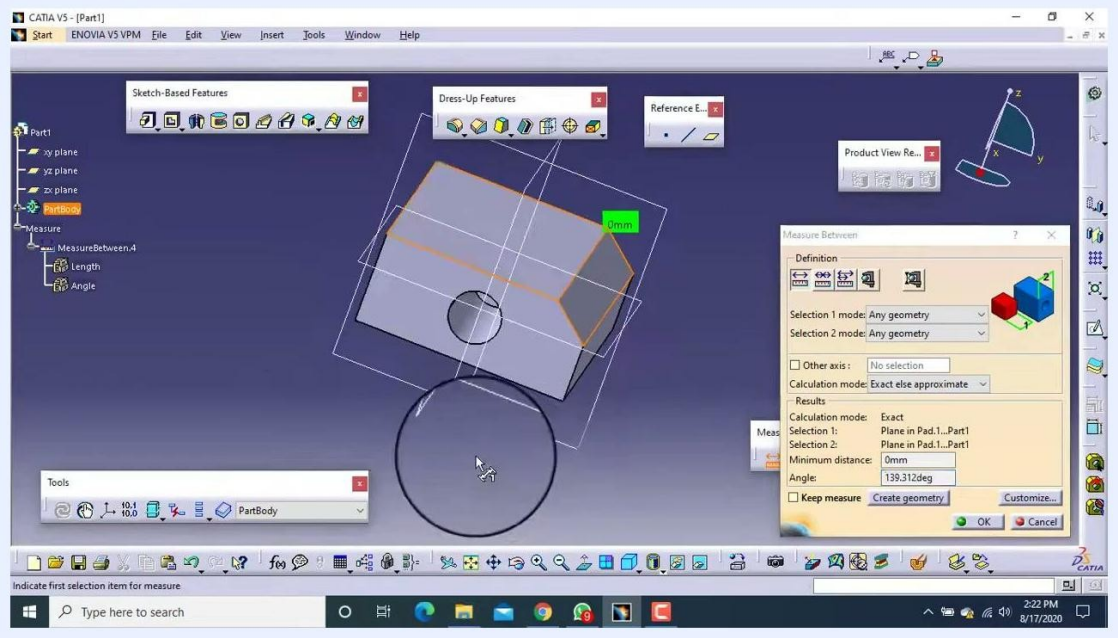
The distance appears in green on the model.

Step 6: Measure the angle between two planes

Select two planes on the part. The dialog updates with these values:

- **Selection 1:** Plane in Pad.1...Part1
- **Selection 2:** Plane in Pad.1...Part1
- **Minimum distance:** 0 mm

- **Angle:** 139.312 deg



Angle measurement between two planes with 139.312 deg shown in the dialog

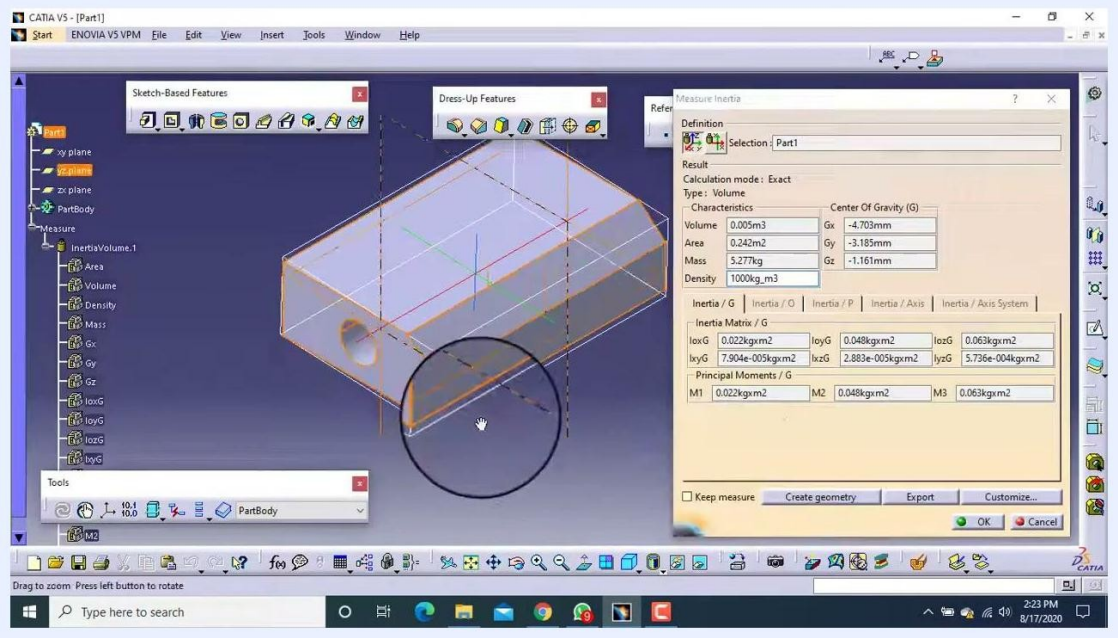
The angle appears in green on the model.

Step 7: Measure the distance between two planes in a pocket

Select two planes inside the pocket feature. The dialog updates with these values:

- **Selection 1:** Plane in Pocket.1...Part1
- **Selection 2:** Plane in Pocket.1...Part1
- **Minimum distance:** 160 mm

- **Angle:** 0 deg

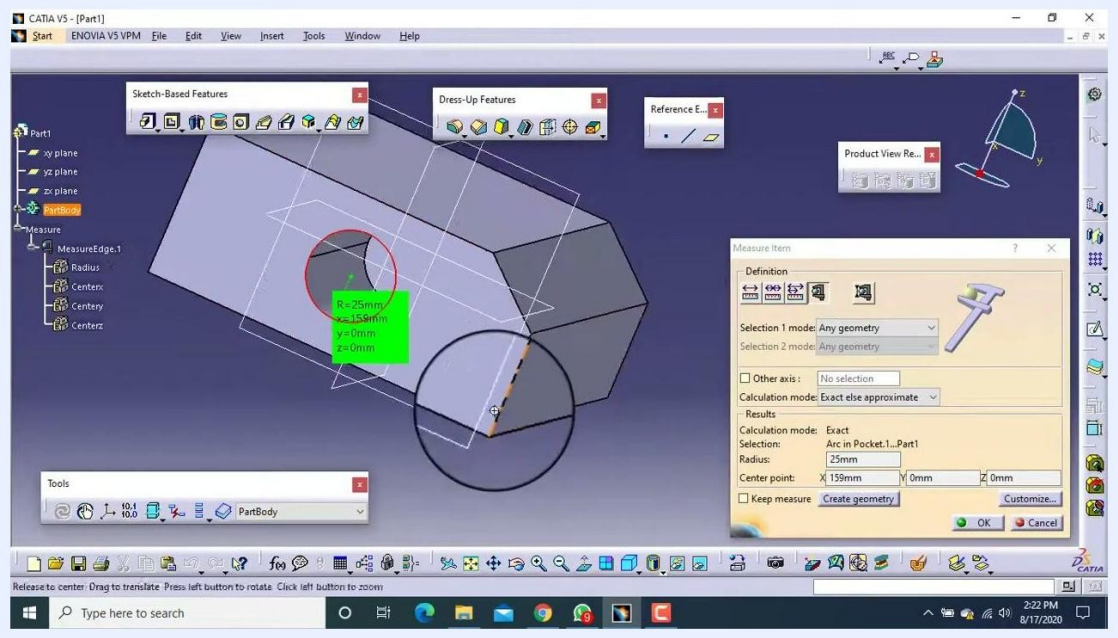


Plane-to-plane measurement in the pocket with 160 mm shown in green

The distance appears in green on the model.

Step 8: Measure the radius and center of a circular edge

Select the arc or circular edge in the pocket feature. The **Measure Item** dialog appears and shows:



Radius and center measurement showing R = 25 mm and a center at 159, 0, 0

- **Radius:** 25 mm
- **Center point:** x = 159 mm, y = 0 mm, z = 0 mm

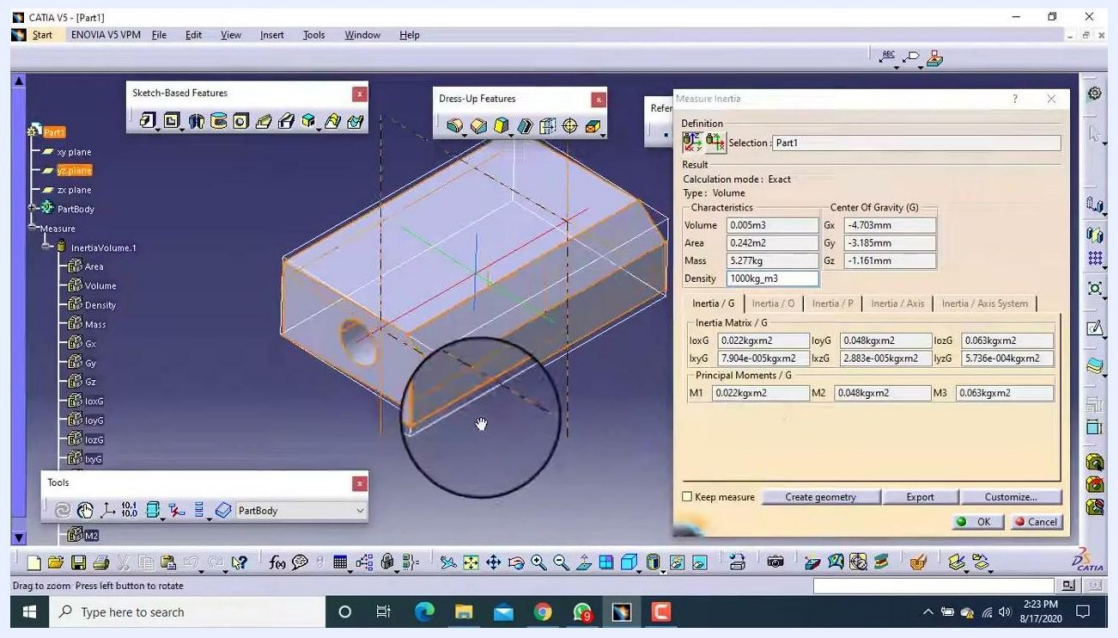
The radius and center coordinates appear in green on the model.

Step 9: Analyze the inertia and physical properties of the part

Open the **Measure Inertia** dialog for the part. The dialog shows these values:

- **Volume:** 0.005 m^3
- **Area:** 0.242 m^2
- **Mass:** 5.277 kg
- **Density:** 1000 kg/m^3
- **Center of gravity:** Gx = -4.703 mm, Gy = -3.185 mm, Gz = -1.161 mm

- The inertia matrix and principal moments

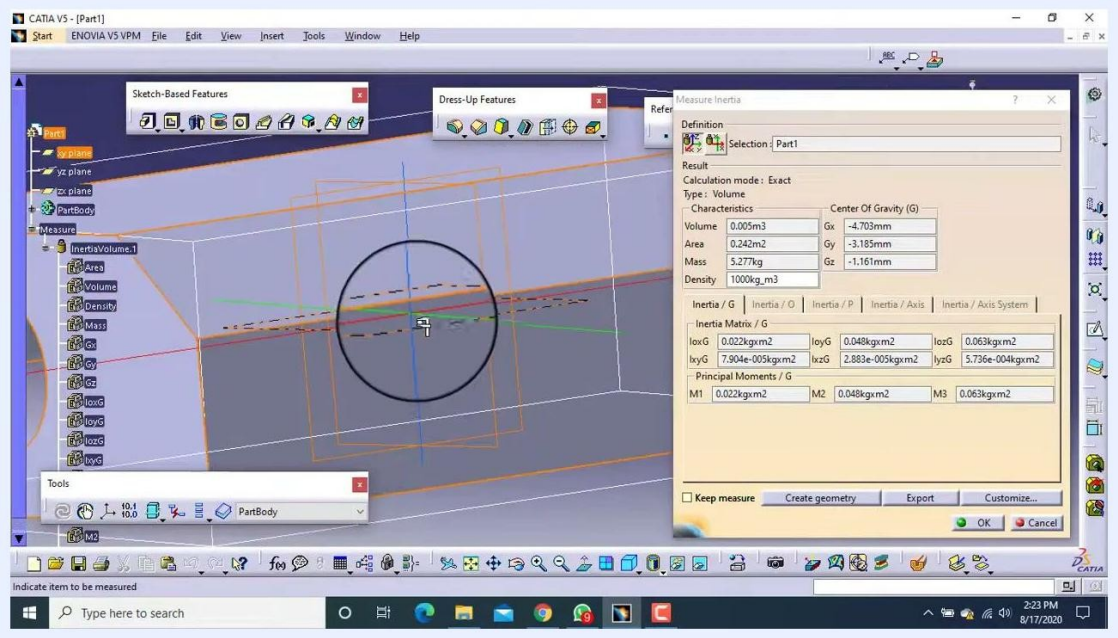


Measure Inertia dialog showing volume, area, mass, density, and inertia matrix values

The part is highlighted, and the dialog stays visible so you can review or export the results.

Step 10: Inspect the model from different angles

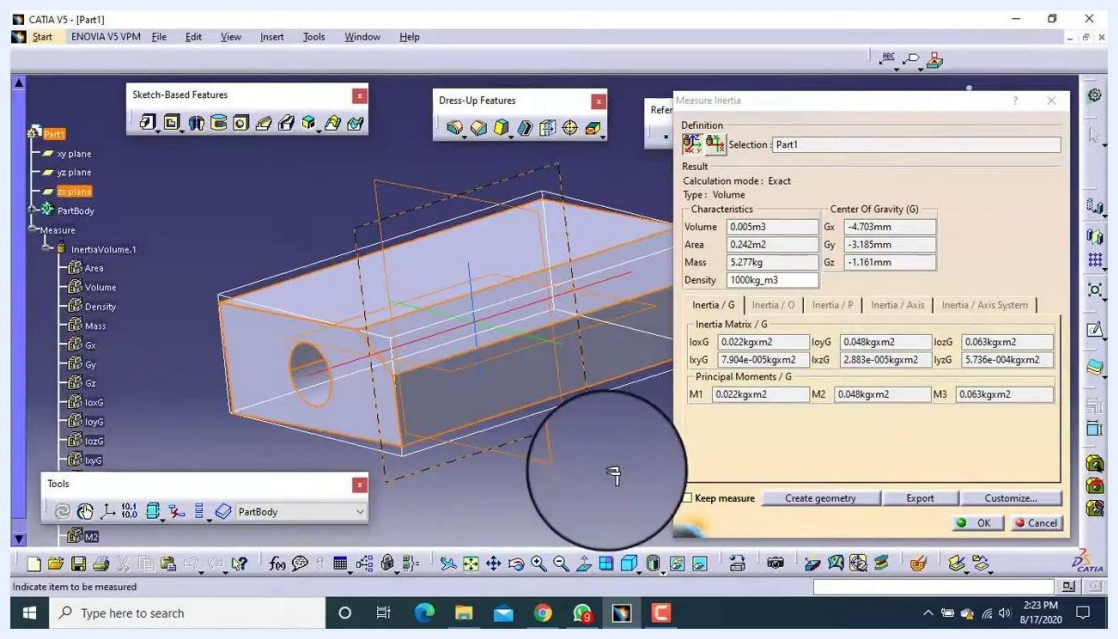
Rotate and zoom in on the model to examine the geometry and the highlighted features used in the inertia calculation. The **Measure Inertia** dialog stays open and shows the same values. The circular feature and reference planes are clearly visible, which helps you verify the geometry you selected.



Zoomed-in view of the part with reference planes and a highlighted circle while the Measure Inertia dialog is open

Step 11: Return to the default view

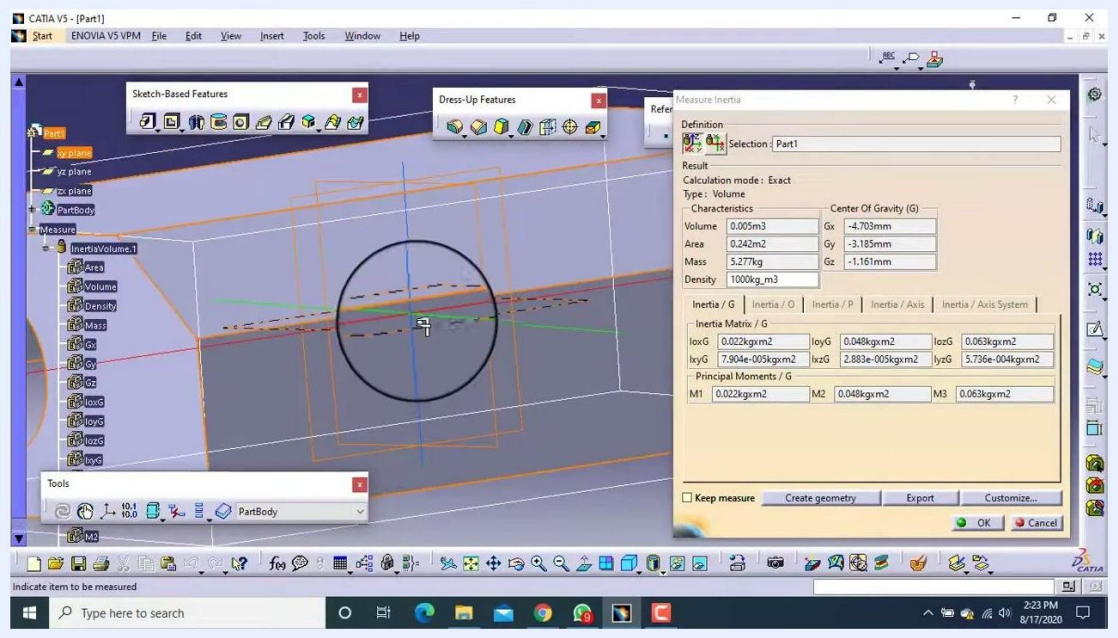
Rotate or zoom out to return to a standard isometric view of the part. The **Measure Inertia** dialog is still open. The part, reference planes, and highlighted features remain visible for a final review.



Isometric view of the part with reference planes and a highlighted circle while the Measure Inertia dialog is open

Step 12: Close the Measure Inertia dialog

Click **OK** or **Cancel**, as appropriate, to close the **Measure Inertia** dialog. The workspace returns to the standard CATIA V5 environment with the part and reference planes visible. The **Sketch-Based Features**, **Dress-Up Features**, **Reference Elements**, and **Measure** toolbars remain accessible.



CATIA V5 workspace with part and reference planes visible, cursor over screen recording icon, Measure Inertia dialog closed

Summary

You now know how to measure dimensions in CATIA V5. You measured distances between planes, lines, and surfaces, and the angle between two planes. You also measured the radius and center of a circular edge and reviewed the part's volume, area, mass, density, and center of gravity. Each measurement dialog gives detailed results and options to create geometry or export data as needed.