How to make a sphere in CATIA V5

This guide shows you how to make a sphere in CATIA V5 using the Part Design workbench. You sketch a semicircle next to an axis of revolution, revolve it 360 degrees with the Shaft tool, and then adjust the sphere's color and view.

Prerequisites

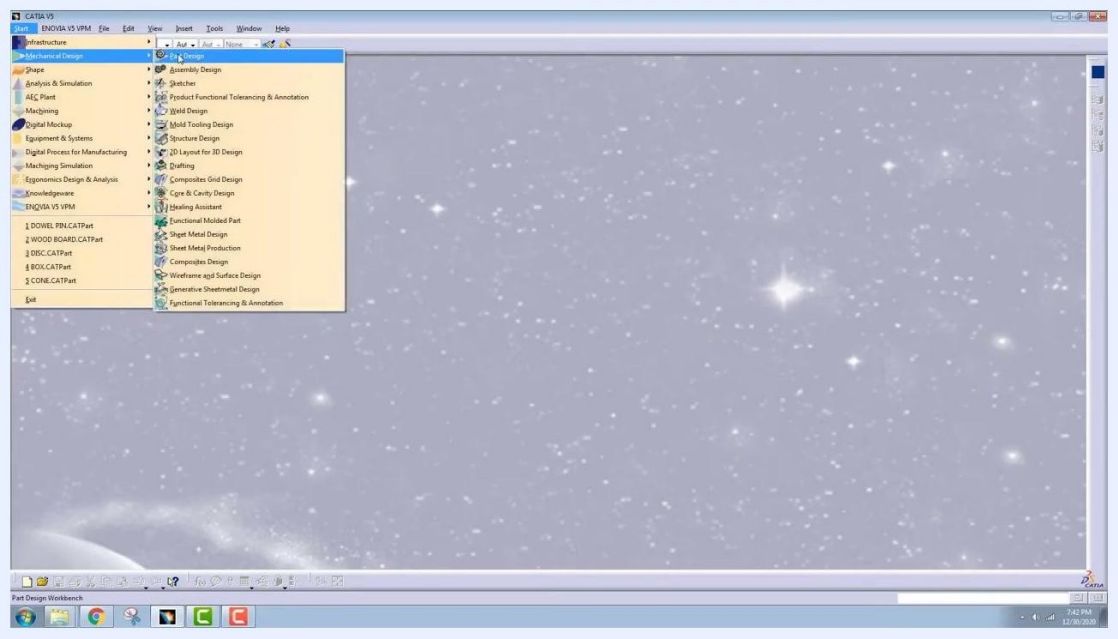
- CATIA V5 installed and ready to launch
- Access to the **Part Design** workbench under **Mechanical Design**

Steps

1. Open CATIA V5 and select Part Design

Launch CATIA V5. From the top menu, select **Start** > **Mechanical Design** > **Part Design**.

The workspace shows a starry background with a menu overlay. Several part files are listed on the left, and the **Part Design** workbench is highlighted.

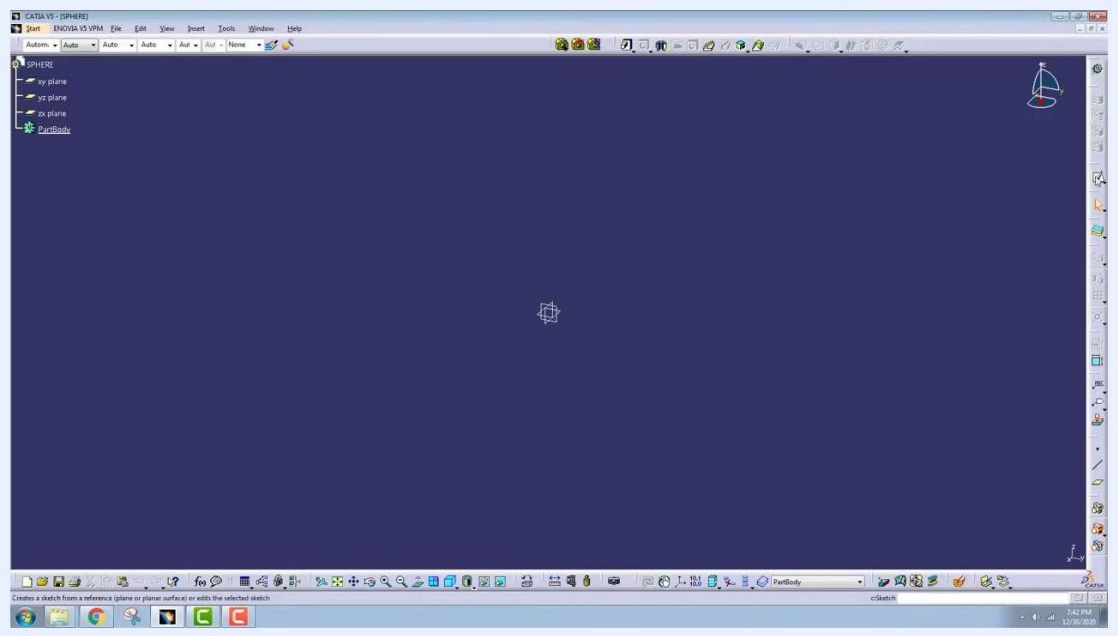


CATIA V5 main menu with Mechanical Design > Part Design selected, showing available workbenches and recent files

2. Check the Part Design workbench

After you select **Part Design**, the workspace changes to a dark blue background. The tree on the left shows the part structure: SPHERE with xy plane, yz plane, zx plane, and PartBody.

A coordinate system appears in the center of the screen, which means you can start sketching.



Part Design workbench with planes and PartBody listed, ready for sketching

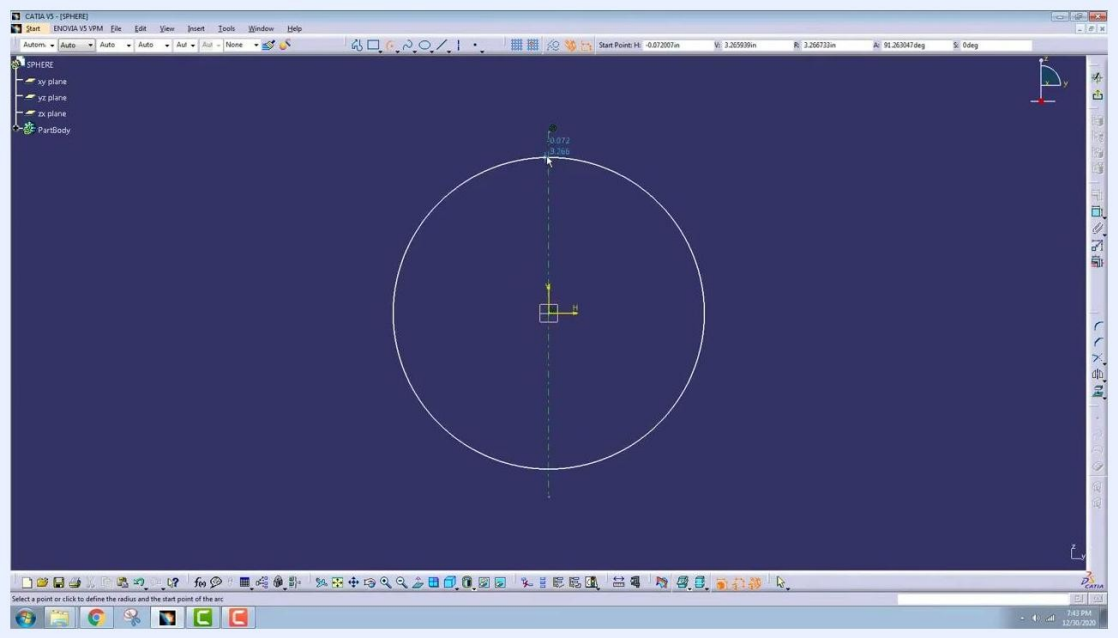
3. Start a new sketch on a plane

Select a plane to begin the sketch. The yz plane is a common choice.

The view zooms in on the selected plane, and a vertical dashed line appears. This line represents the axis of revolution. The cursor is positioned so you can begin drawing along the axis.

4. Draw the axis of revolution

Draw a vertical construction line (dashed). You will revolve the profile around this line. The line is labeled with coordinates, and you use the cursor to define its endpoints.

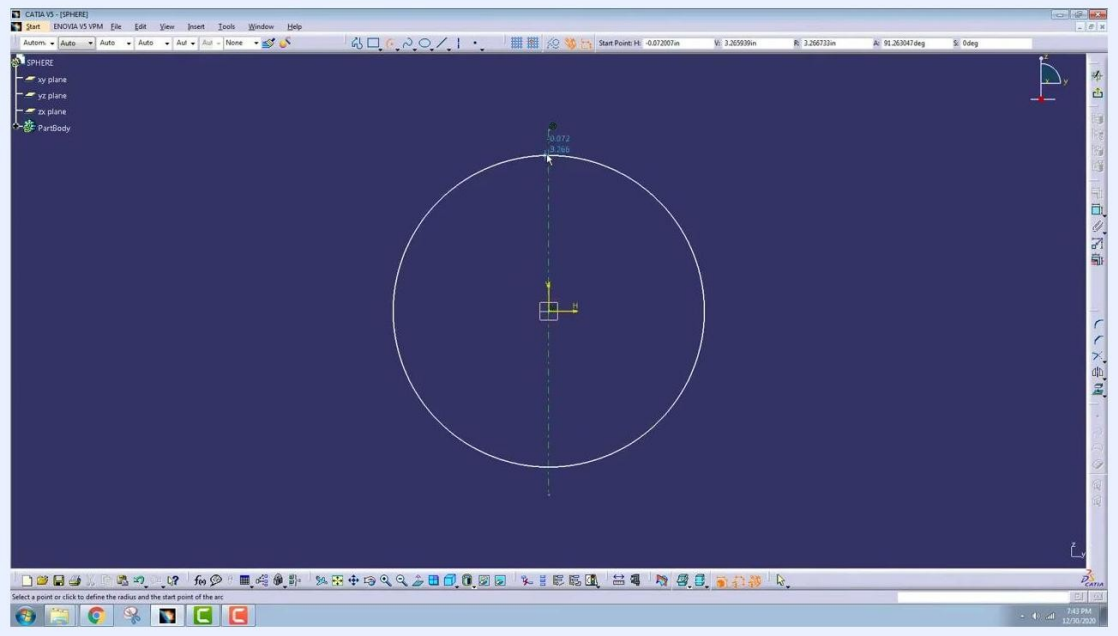


Sketching a vertical axis line on the selected plane, with coordinates visible

5. Draw a circle for the sphere profile

Select the circle tool and place the center at the intersection of the axis and the origin. Drag outward to define the radius.

The circle is centered on the axis, and its radius is displayed numerically.

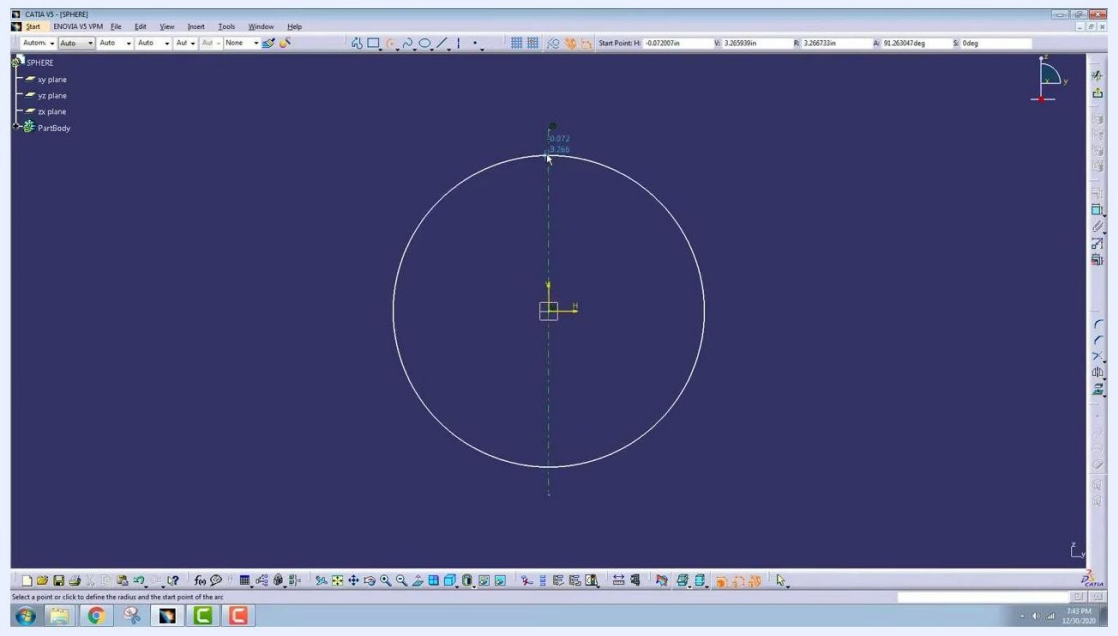


A full circle is drawn centered on the axis, with radius and coordinates shown

6. Create a semicircle profile

Use the trim or arc tool to remove half of the circle. This leaves a semicircle profile on one side of the axis, ready for revolution.

The axis line is highlighted, and the semicircle is clearly visible.

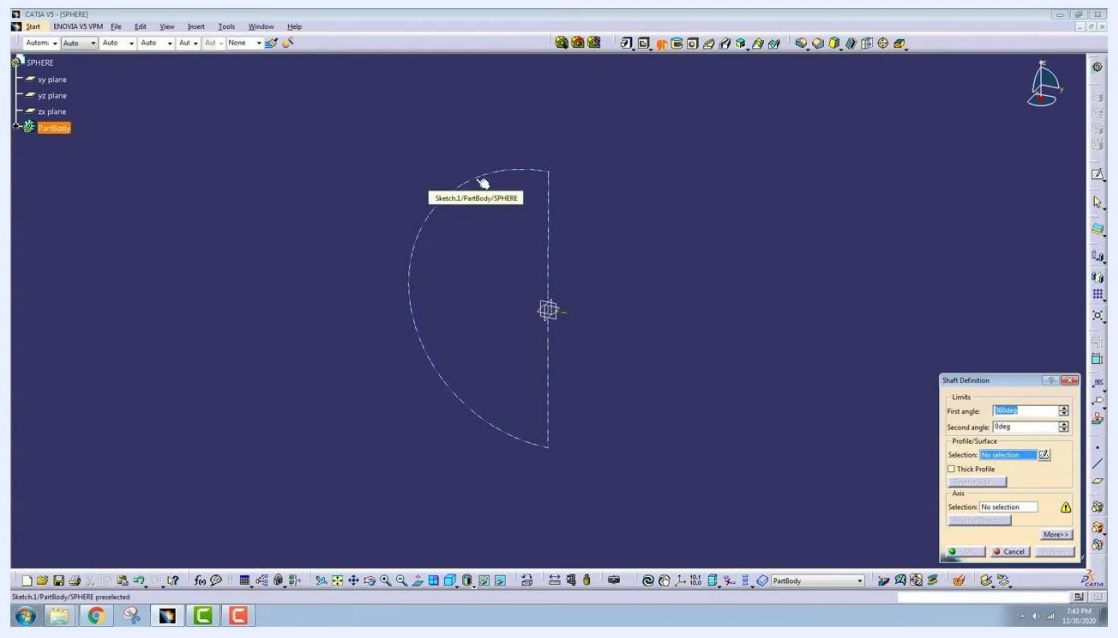


A semicircle profile is visible, with the axis line highlighted for revolution

7. Select the Shaft tool to revolve the profile

Make sure the semicircle and the axis are selected. Then click the **Shaft** (revolve) tool in the toolbar.

The semicircle and axis are now ready to be revolved into a 3D shape.

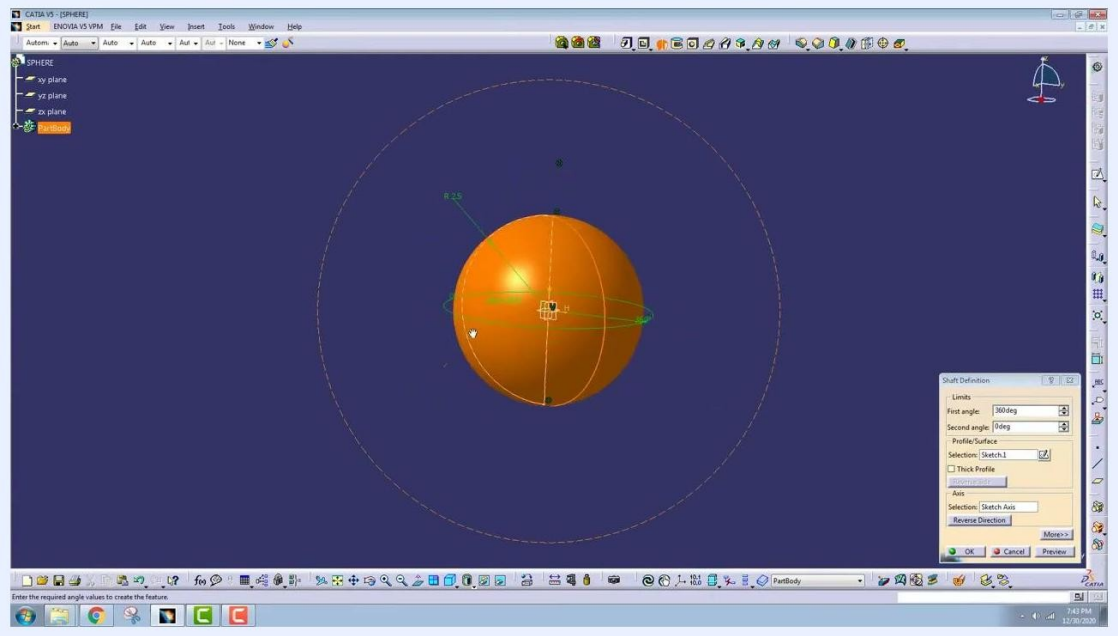


Semicircle and axis selected, Shaft tool highlighted, ready for revolution

8. Define the shaft parameters and preview the sphere

The **Shaft Definition** dialog appears. Set **First angle** to 360deg and **Second angle** to 0deg. Make sure the profile is set to Sketch.1 and the axis to Sketch Axis.

Click **Preview** to see the resulting 3D sphere.

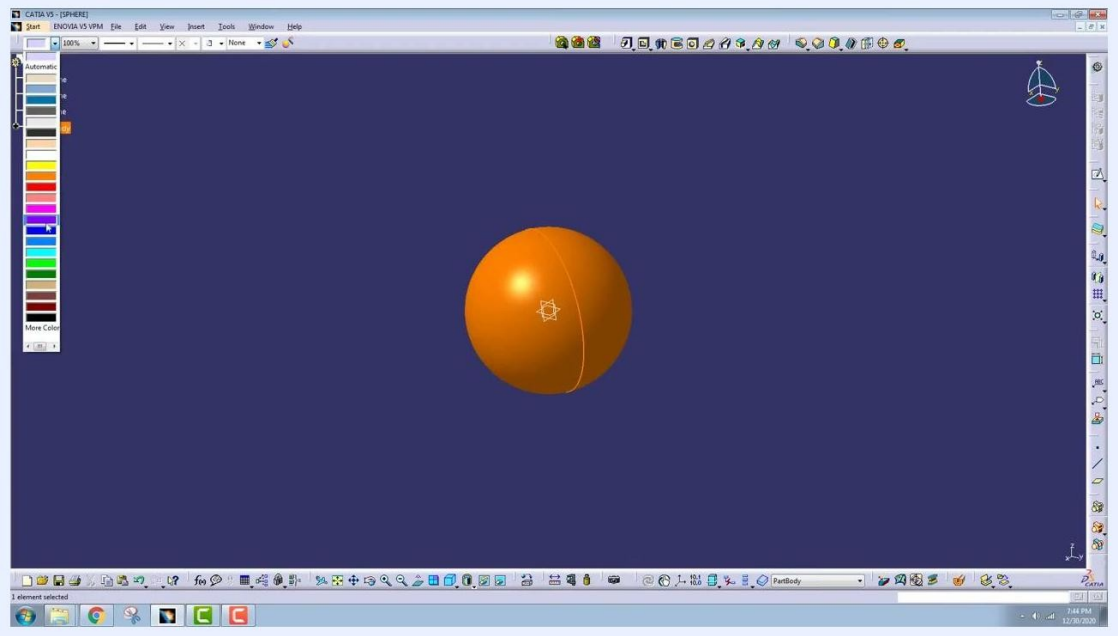


Shaft Definition dialog open, parameters set, and a preview of the orange 3D sphere visible

9. Change the sphere's color

With the sphere selected in the workspace, open the color palette on the left side of the screen. Click a color swatch, such as a shade of blue or purple.

The sphere's appearance updates immediately.

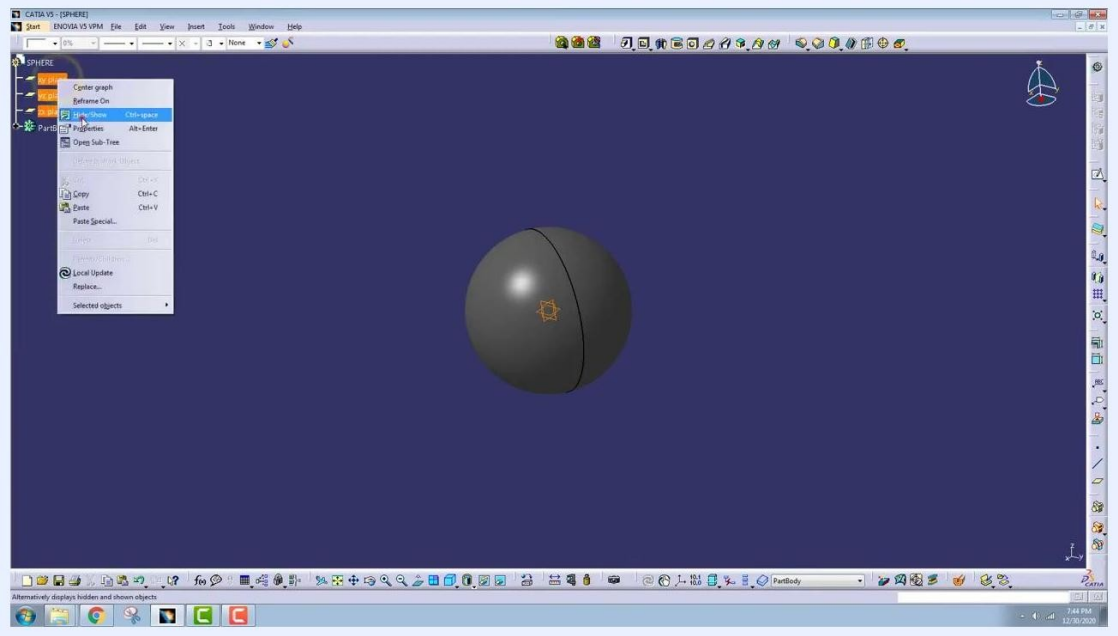


CATIA V5 workspace showing the color palette open and the sphere being recolored

10. Hide or show the reference planes

In the tree on the left, right-click a reference plane such as xy plane, yz plane, or zx plane. In the context menu, select **Hide/Show**. You can also use the shortcut Ctrl+Space.

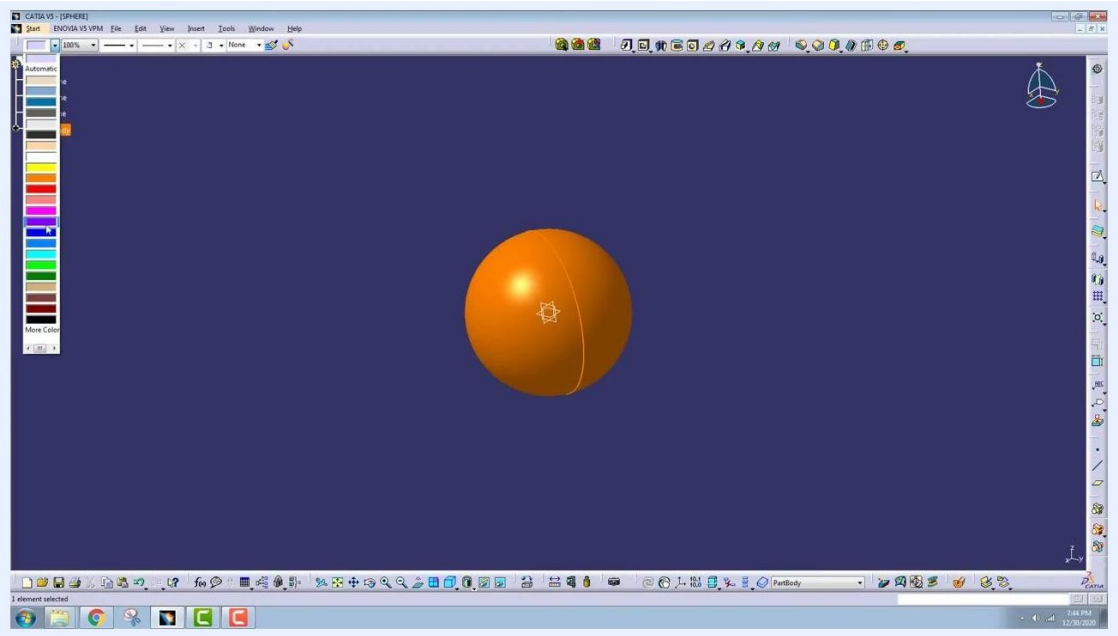
The reference planes are hidden, and only the sphere remains visible.



Context menu open on a reference plane with Hide/Show highlighted, sphere visible in workspace

11. Adjust the view and lighting

Rotate or reposition the sphere in the 3D workspace to inspect it from different angles. Lighting and shading affect the surface, and you may see shadows and highlights. These effects can give the sphere different shades and enhance its 3D appearance.



Sphere shown with advanced shading and shadow, rotated to a new angle in the workspace

Summary

You now know how to make a sphere in CATIA V5. Open the **Part Design** workbench, sketch an axis and a semicircle on a plane, and revolve the profile with the **Shaft** tool at a first angle of 360deg. You can then change the color, hide the reference planes, and rotate the sphere to check its shading.