How to make a cone in CATIA V5 using Part Design and the Shaft feature

This guide shows you how to make a cone in CATIA V5. You sketch a right triangle profile in Part Design, apply constraints, and revolve the profile 360 degrees around an axis with the Shaft tool. You then inspect the finished 3D cone.

Prerequisites

- CATIA V5 installed and ready to open.

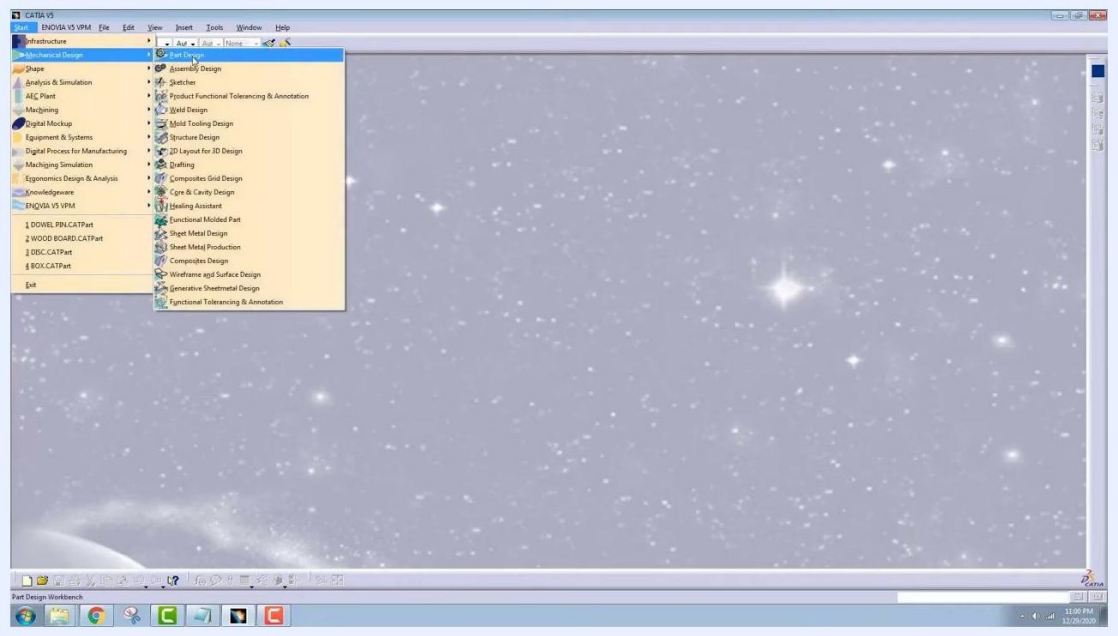
Steps

1. Start a new part in Part Design

To begin making a cone in CATIA, open the Part Design workbench:

1. Open CATIA V5.
2. In the top menu bar, select **Start > Mechanical Design > Part Design**.
3. In the dropdown menu under **Mechanical Design**, click **Part Design** to begin a new part.

Before you click, the workspace background shows a starry pattern, and the left panel lists the available files and modules.



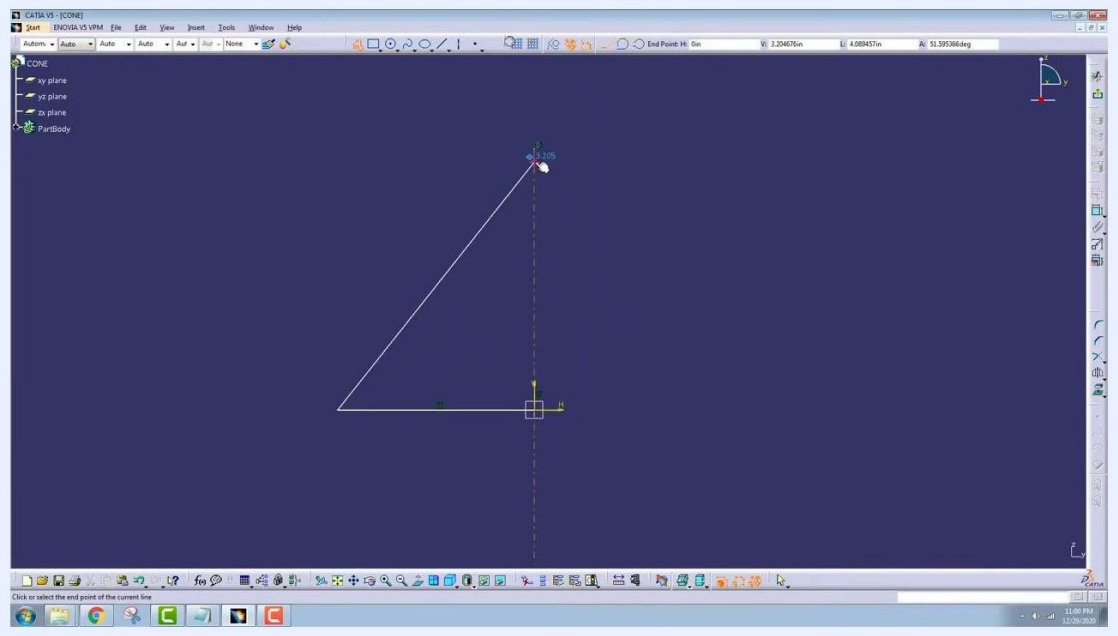
CATIA V5 main menu with Mechanical Design > Part Design selected, showing the dropdown menu and workspace background

After you click, the workspace changes to a dark blue background. In the tree on the left, under the part name (for example, "CONE"), you see three default planes (xy plane, yz plane, and zx plane) and a PartBody. A coordinate system appears in the center of the workspace. No geometry is present yet.

2. Start the sketch and draw the axis line

Select a plane to start a new sketch. The xy plane is a common choice. The workspace stays dark blue, and the sketch tools become available.

Draw a vertical construction line from the origin upwards. Use the cursor to define the endpoint. Coordinate values appear near the cursor as you draw.



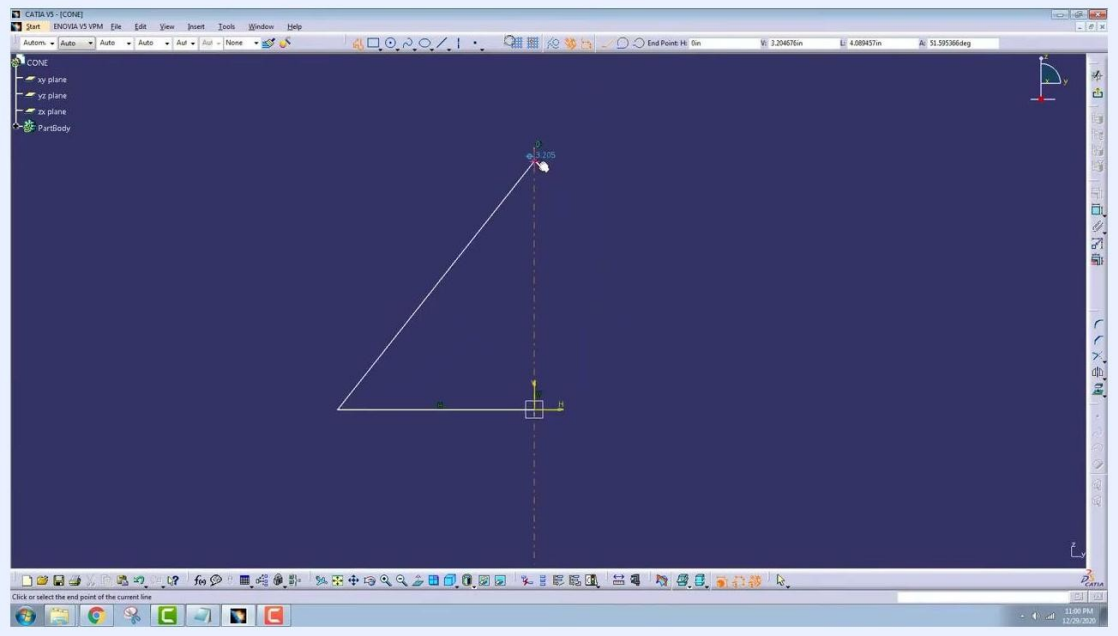
CATIA V5 sketch environment with a vertical construction line being drawn from the origin, coordinate values visible

3. Draw the cone profile

Continue sketching by drawing two lines to form a right triangle. This triangle is the cone profile that you revolve. The triangle has:

- One vertical side, which is the axis of revolution.
- One horizontal base.
- One slanted side that connects the top of the vertical line to the end of the base.

The lines connect at the origin, and the endpoints are marked.

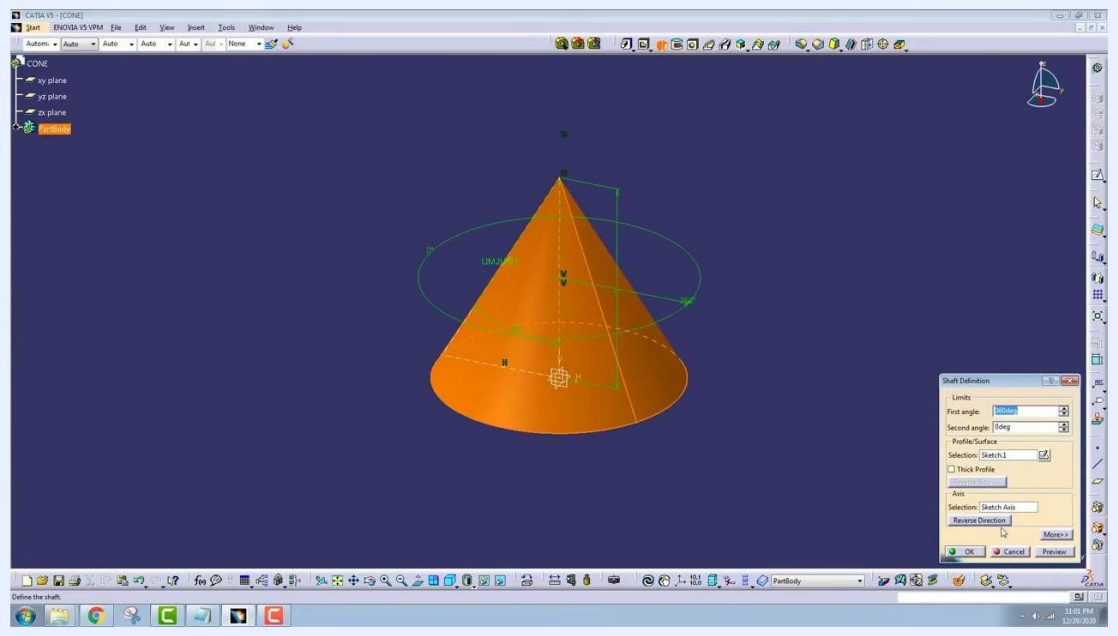


CATIA V5 sketch showing a right triangle profile for the cone, with endpoints and construction lines visible

4. Apply geometric constraints

Add constraints to the sketch to define the exact dimensions and angles. The angle between the base and the slanted side is displayed (for example, 38.404634 degrees), and the vertical and horizontal sides are dimensioned.

The **Constraint Definition** dialog box appears and lets you edit the angle value.



CATIA V5 sketch with constraints applied, angle and dimension values visible, and the Constraint Definition dialog open

5. Exit the sketch

Exit sketch mode to return to the Part Design workbench. The completed triangle profile is highlighted in orange and is ready for the next operation.

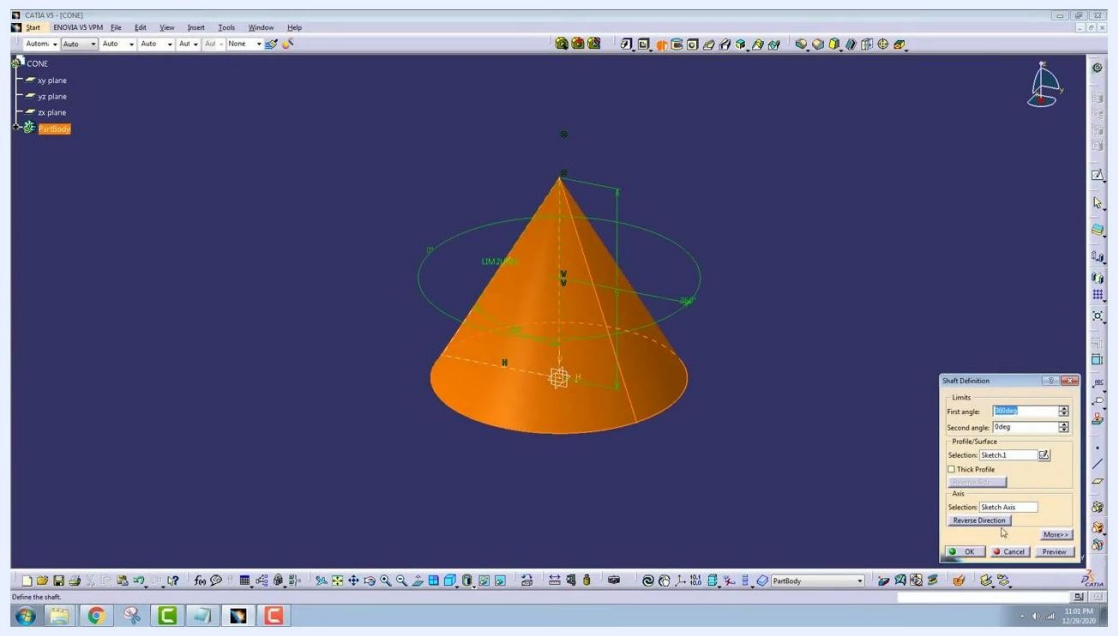
6. Make the cone with the Shaft tool

The **Shaft** tool creates a shaft by revolving a sketch around an axis. Select it, and the **Shaft Definition** dialog box appears. Set the following parameters:

- **First angle:** 360deg
- **Second angle:** 0deg
- **Profile/Surface Selection:** Sketch.1
- **Axis Selection:** Sketch Axis

If needed, select **Reverse Direction**. Click **OK** to generate the 3D cone.

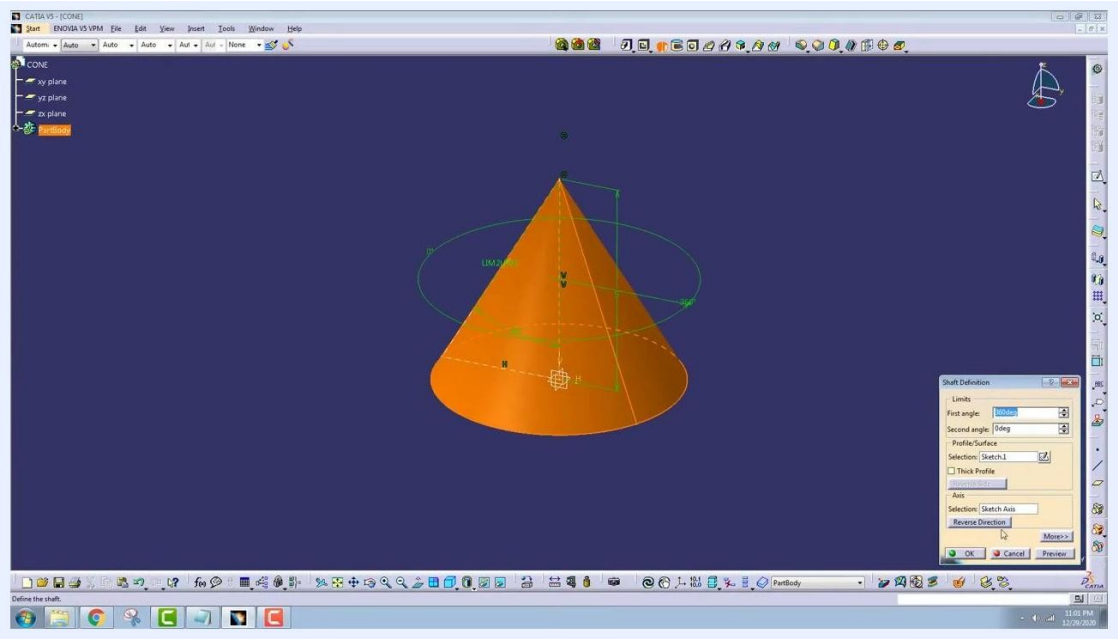
The cone appears in orange, with green construction geometry and revolution indicators.



CATIA V5 with the Shaft Definition dialog open, the cone rendered in orange, and the revolution parameters set

7. Inspect the 3D cone

Rotate and zoom the view to inspect the cone. The cone appears in a shaded view, and the revolution path is visible as a dashed circle. The part tree on the left shows the updated structure, and the workspace is clear for further operations.



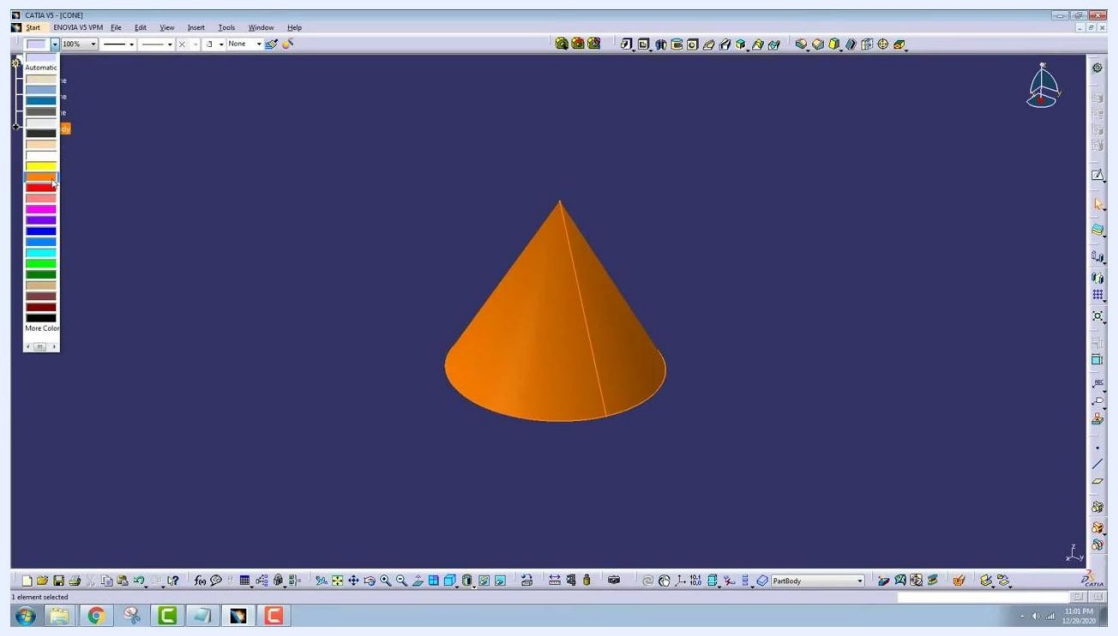
CATIA V5 showing the completed 3D cone, rotated for inspection, with the revolution path visible

8. Check shading and lighting effects

Review the cone with enhanced shading. A visible shadow appears on the workspace floor, which indicates a change in the rendering mode or lighting settings.

The left panel lists the part structure: CONE with xy plane, yz plane, zx plane, and PartBody. The cone remains selected, and the workspace background is slightly lighter, which emphasizes the shadow.

Use this view to inspect the cone's surface quality and how it interacts with light. This is useful for presentation or further design evaluation.



CATIA V5 workspace showing the cone with a realistic shadow, the part tree visible on the left, and enhanced lighting effects

Summary

You now know how to make a cone in CATIA V5. You started a new part in Part Design, sketched and constrained a right triangle profile, and exited the sketch. You then revolved the profile with the **Shaft** tool and inspected the resulting 3D cone, including its shading and shadow.