How to create a plane in CATIA V5

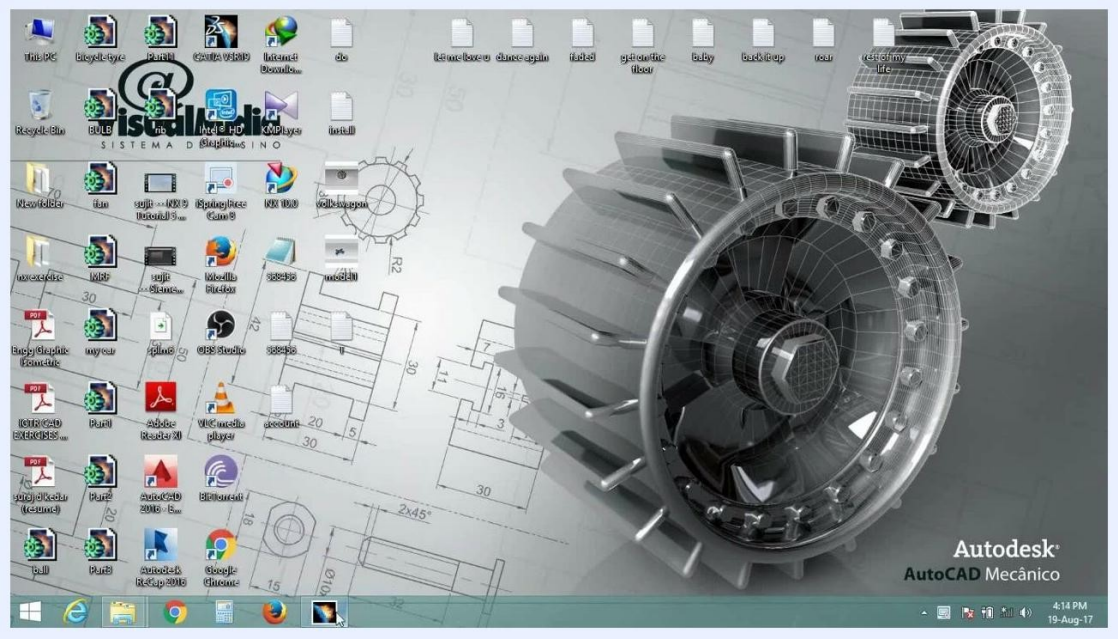
This guide shows you how to create a plane in CATIA V5 in the Part Design workspace. You start a new part, open the **Plane Definition** dialog, and choose a plane type. The types covered are offset, parallel through point, angle/normal, through three points, through two lines, through point and line, through planar curve, normal to curve, and tangent to surface. Each type uses different references, so you can position a plane precisely on your model.

Prerequisites

- CATIA V5 installed on your computer, with its icon on the desktop.
- For the point, line, curve, and surface plane types, a part with geometry to reference (for example, Pad.1 vertices and edges).

Step 1: Open CATIA V5

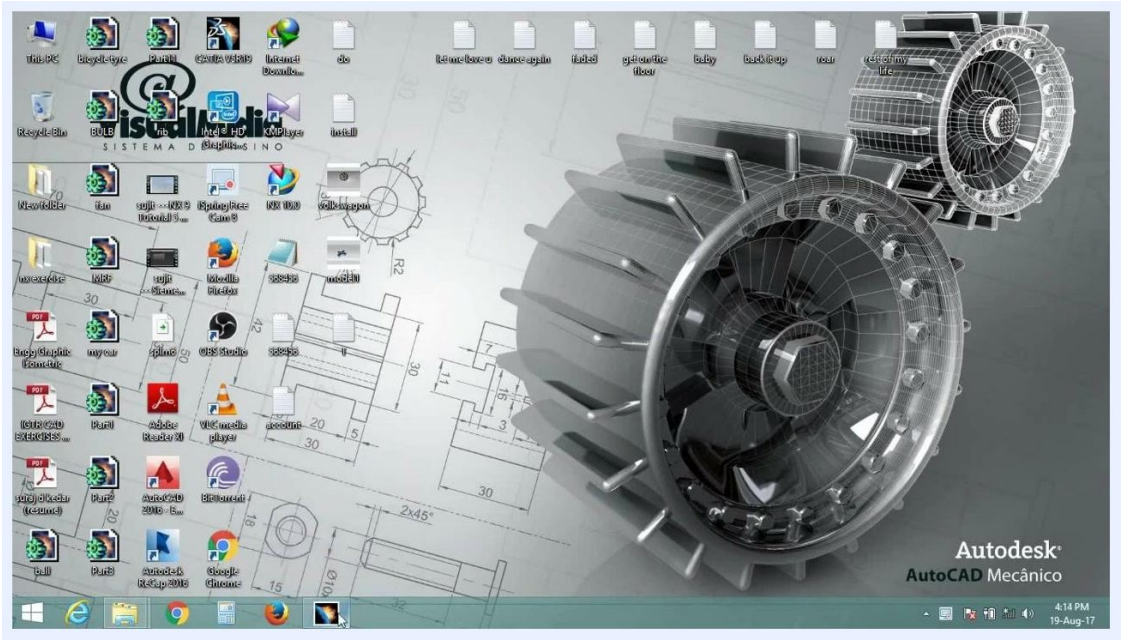
1. Make sure your computer is on the desktop.
2. Double-click the CATIA V5 icon to launch the application.



Windows desktop with an engineering-themed wallpaper, application and document icons including CATIA V5R19, and the taskbar at the bottom

Step 2: Start a new part in Part Design

1. In the top menu of CATIA V5, click **Start**.
2. Hover over **Mechanical Design**.
3. Select **Part Design** from the dropdown.



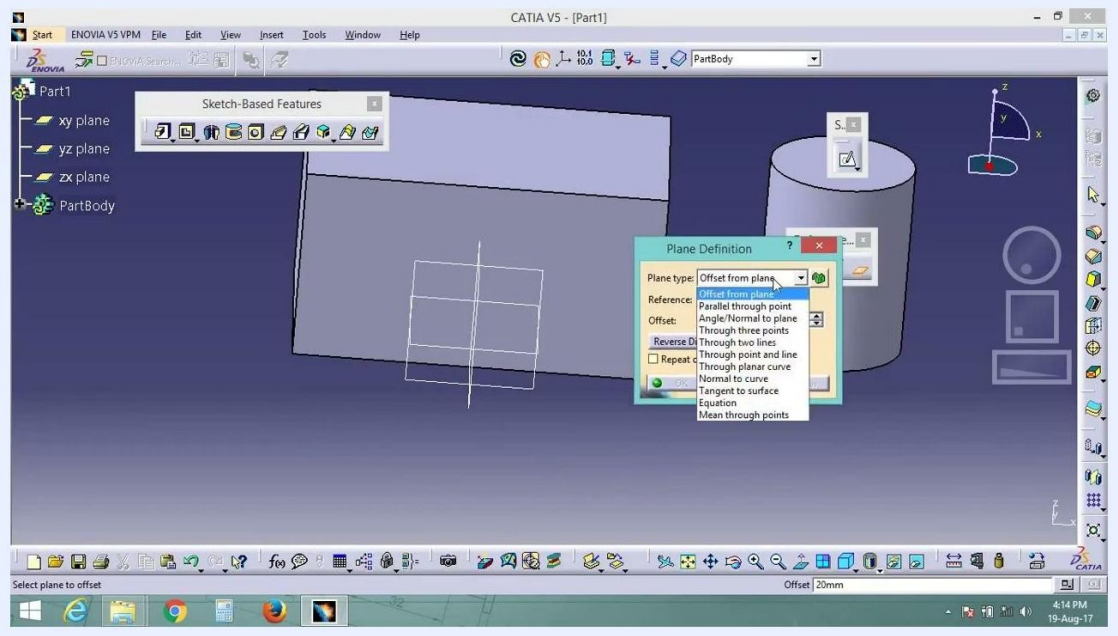
CATIA V5 with the Start menu expanded and Mechanical Design and Part Design highlighted

The Part Design workspace displays the default planes (xy plane, yz plane, and zx plane) and a PartBody. If you need basic geometry to reference, use the sketch tools to create it.

Step 3: Create a plane in CATIA with an offset from a plane

1. In the Part Design workspace, select the **Plane** tool.
2. In the **Plane Definition** dialog, set **Plane type** to **Offset from plane**.
3. For **Reference**, select the base plane you want, for example xy plane.
4. Enter the offset value in the **Offset** field, for example 20mm or 100mm.

5. Click **OK**.

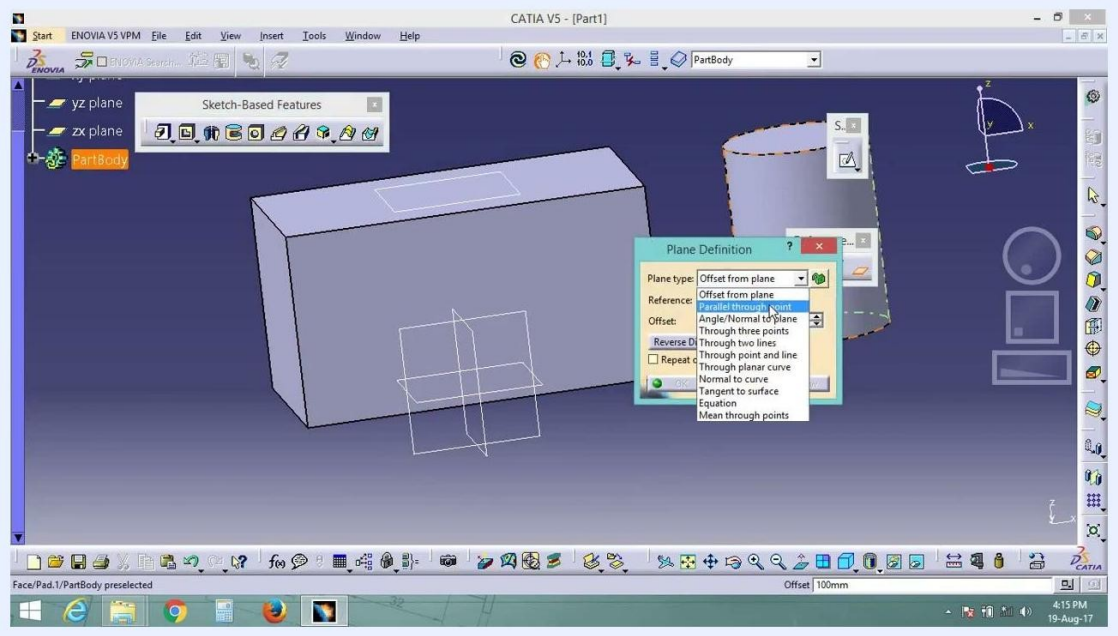


CATIA V5 workspace with a 3D part, the default planes, and the Plane Definition dialog showing Offset from plane, reference xy plane, and the offset value being edited

Step 4: Create a plane parallel through a point

1. Open the **Plane Definition** dialog again.
2. Set **Plane type** to **Parallel through point**.
3. For **Reference**, select the plane you want, for example yz plane.
4. For **Point**, select a vertex or point on your model, for example Pad.1\Vertex.
5. Click **OK**.

CATIA creates a plane parallel to the reference plane that passes through the selected point.

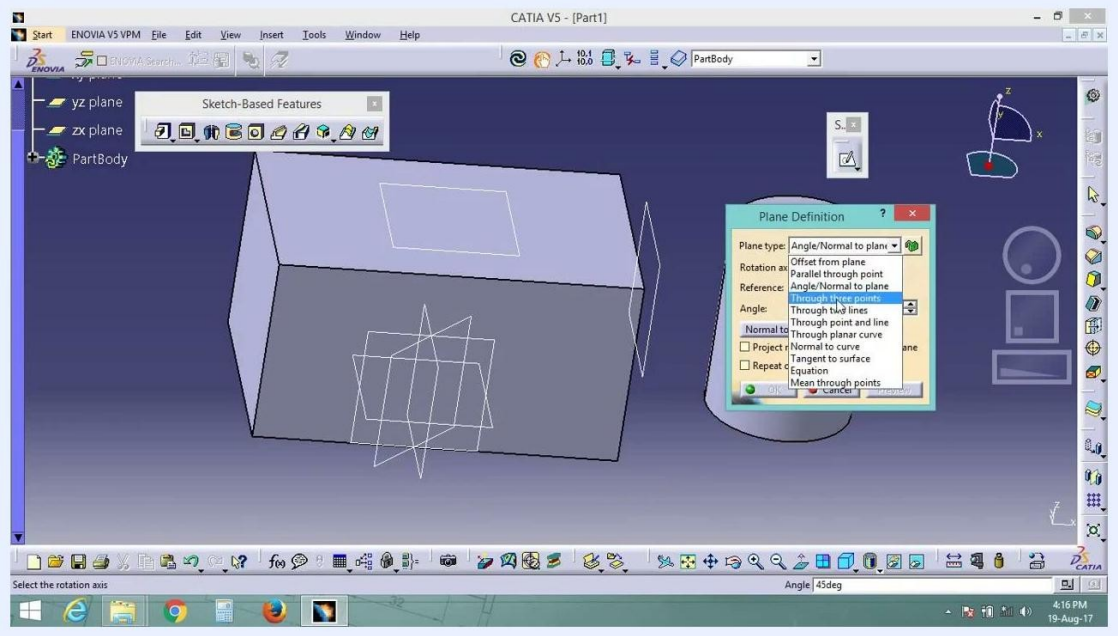


Plane Definition dialog set to Parallel through point, with reference yz plane and point Pad.1\Vertex selected

Step 5: Create a plane at an angle to a reference plane

1. In the **Plane Definition** dialog, set **Plane type** to **Angle/Normal to plane**.
2. For **Rotation axis**, select **Z Axis** from the dropdown menu.
3. For **Reference**, select zx plane.
4. In the **Angle** field, enter 45deg.

5. Click **OK**.

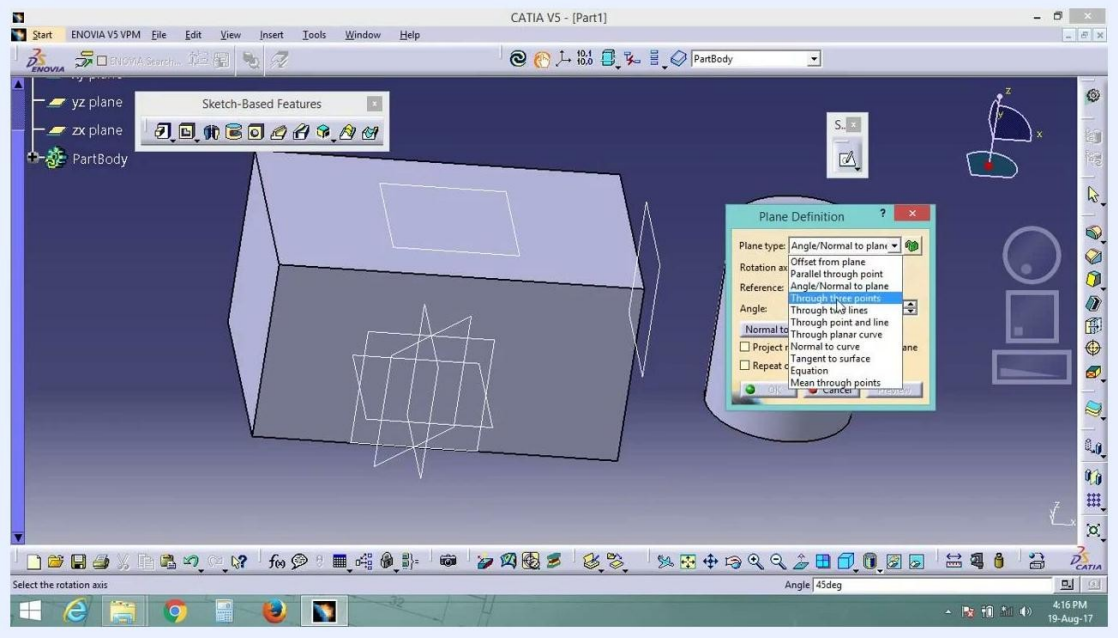


Plane Definition dialog with Angle/Normal to plane selected, Z Axis as the rotation axis, zx plane as the reference, an angle of 45deg, and the OK button highlighted

Step 6: Create a plane through three points

1. Open the **Plane Definition** dialog.
2. Set **Plane type** to **Through three points**.
3. For **Point 1**, select Pad.1\Vertex.
4. For **Point 2**, select another Pad.1\Vertex.
5. For **Point 3**, select a third Pad.1\Vertex.
6. Click **OK**.

CATIA creates a plane that passes through the three selected points.

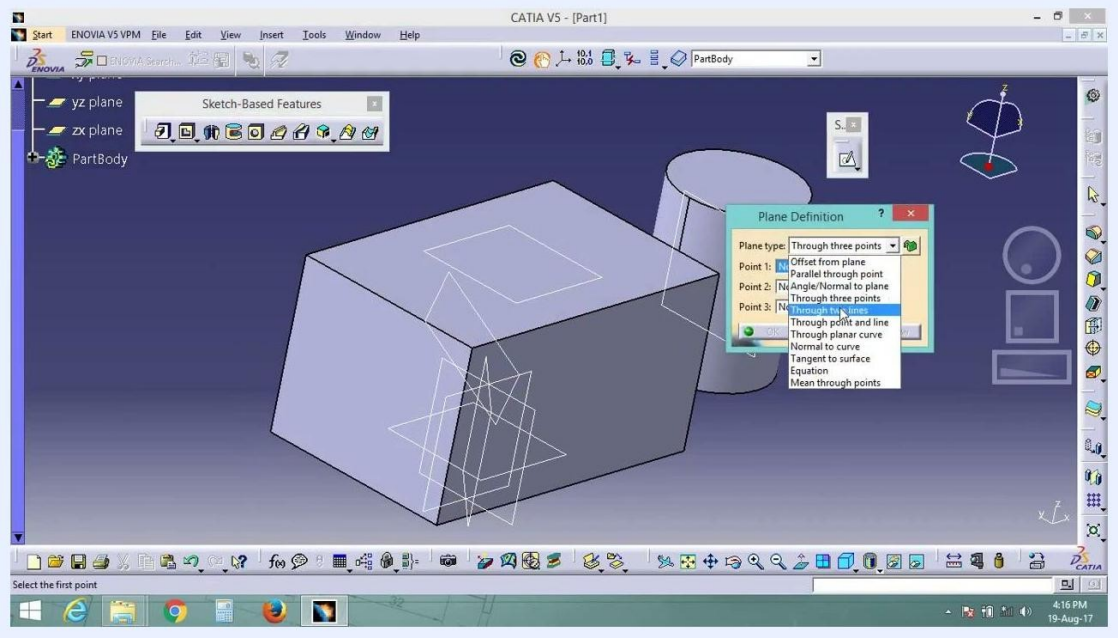


Plane Definition dialog with Through three points selected and three vertices from Pad.1 entered as points

Step 7: Create a plane through two lines

1. Open the **Plane Definition** dialog.
2. Set **Plane type** to **Through two lines**.
3. For **Line 1**, select Pad.1\Edge.
4. For **Line 2**, select another edge or line as required.
5. (Optional) Select **Forbid non coplanar lines** to restrict the selection to coplanar lines only.

6. Click **OK**.

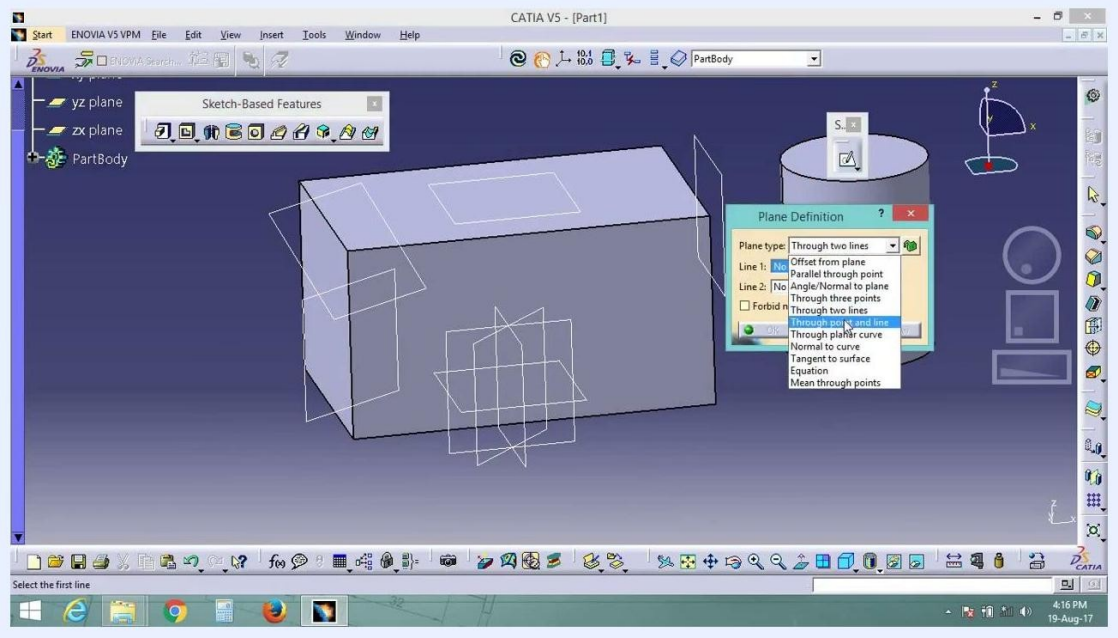


Plane Definition dialog with Through two lines selected and one edge chosen as Line 1

Step 8: Create a plane through a point and a line

1. Open the **Plane Definition** dialog.
2. Set **Plane type** to **Through point and line**.
3. For **Point**, select Pad.1\Vertex.
4. For **Line**, select Pad.1\Edge.
5. Click **OK**.

CATIA creates a plane that passes through the selected point and line.



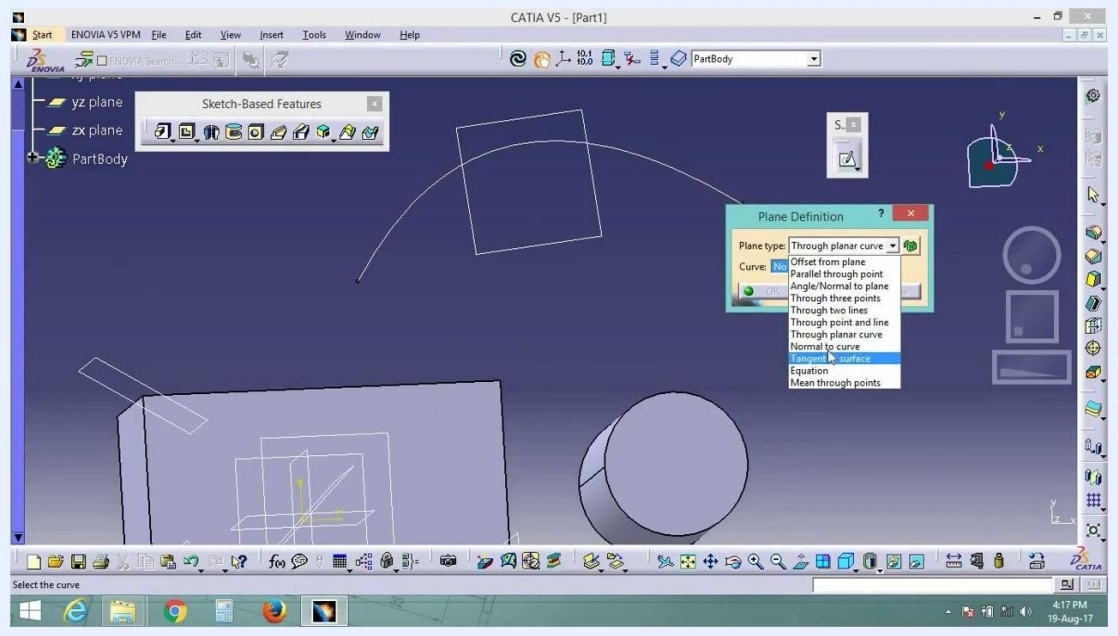
Plane Definition dialog with Through point and line selected, showing a vertex and an edge entered

Step 9: Create a plane through a planar curve

You can also create a plane in CATIA from curves and surfaces, starting with a planar curve.

1. Open the **Plane Definition** dialog.
2. Set **Plane type** to **Through planar curve**.
3. For **Curve**, select the planar curve from your model.

4. Click **OK**.

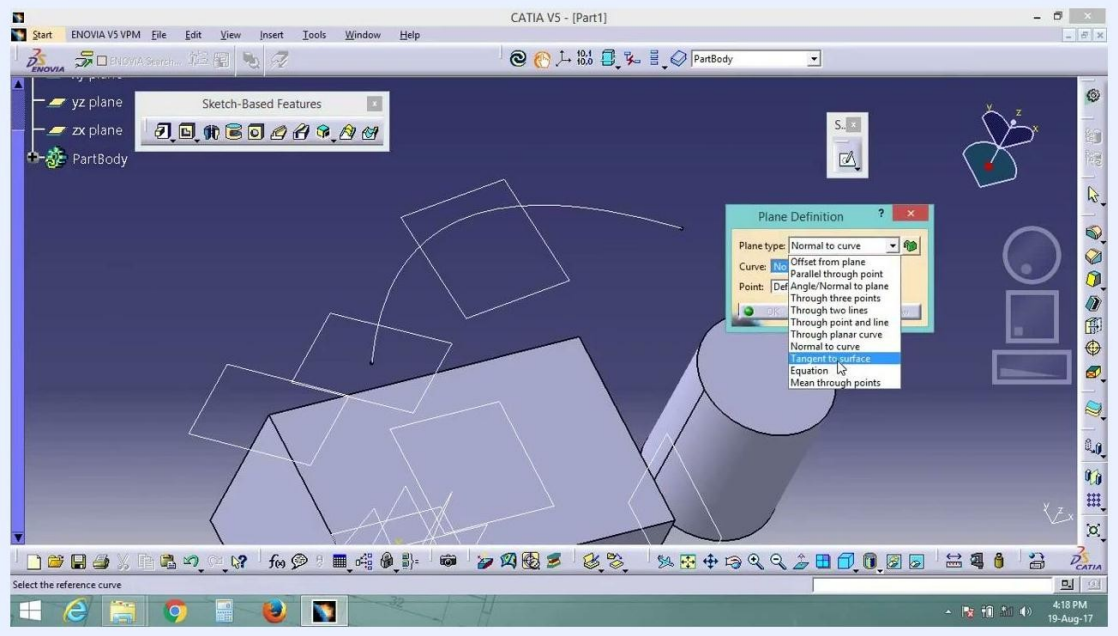


Plane Definition dialog open with Through planar curve selected and a curve being chosen as the reference

Step 10: Create a plane normal to a curve

1. In the **Plane Definition** dialog, set **Plane type** to **Normal to curve**.
2. For **Curve**, select the curve from your model.
3. For **Point**, select the point on the curve where the plane should be normal.

4. Click **OK**.



Plane Definition dialog with Normal to curve selected and the Curve and Point fields ready for input

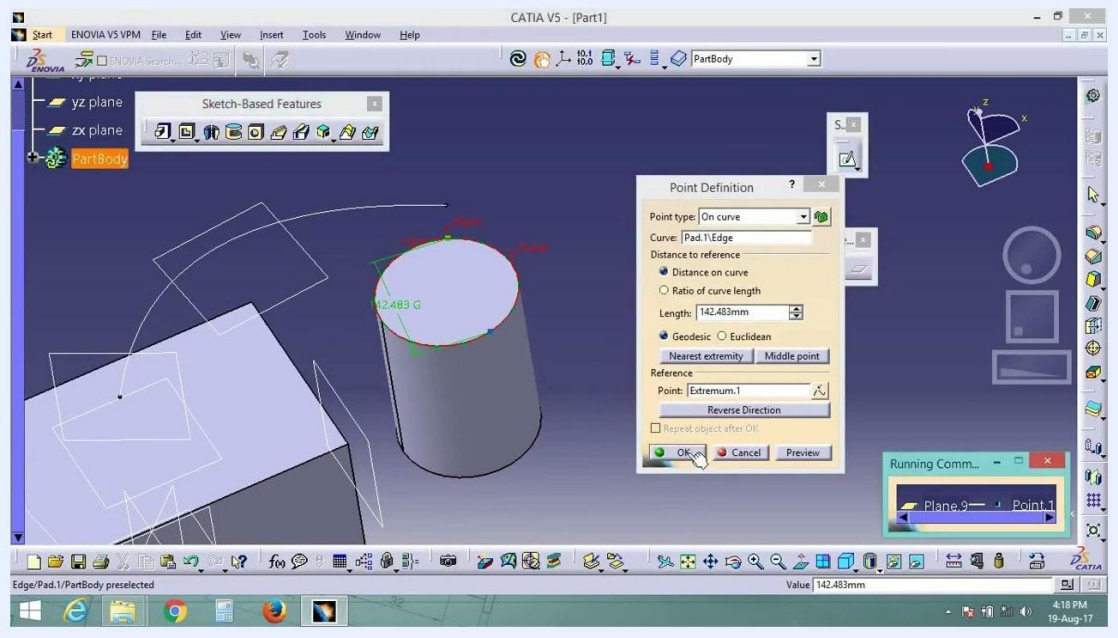
Step 11: Create a plane tangent to a surface

1. In the **Plane Definition** dialog, set **Plane type** to **Tangent to surface**.
2. For **Surface**, select the surface from your model.
3. For **Point**, select or create the point on the surface where the plane should be tangent.

If a point does not exist yet, right-click in the **Point** field and choose to create a new point. Then complete the **Point Definition** dialog:

- Set **Point type** to **On curve**.
- For **Curve**, select the edge or curve on the surface, for example Pad.1\Edge.
- Set **Distance to reference** as needed, for example Length = 142.483mm.
- Choose **Nearest extremity** or **Middle point** as the reference.
- Click **OK** to create the point.

Finally, click **OK** in the **Plane Definition** dialog to create the tangent plane.

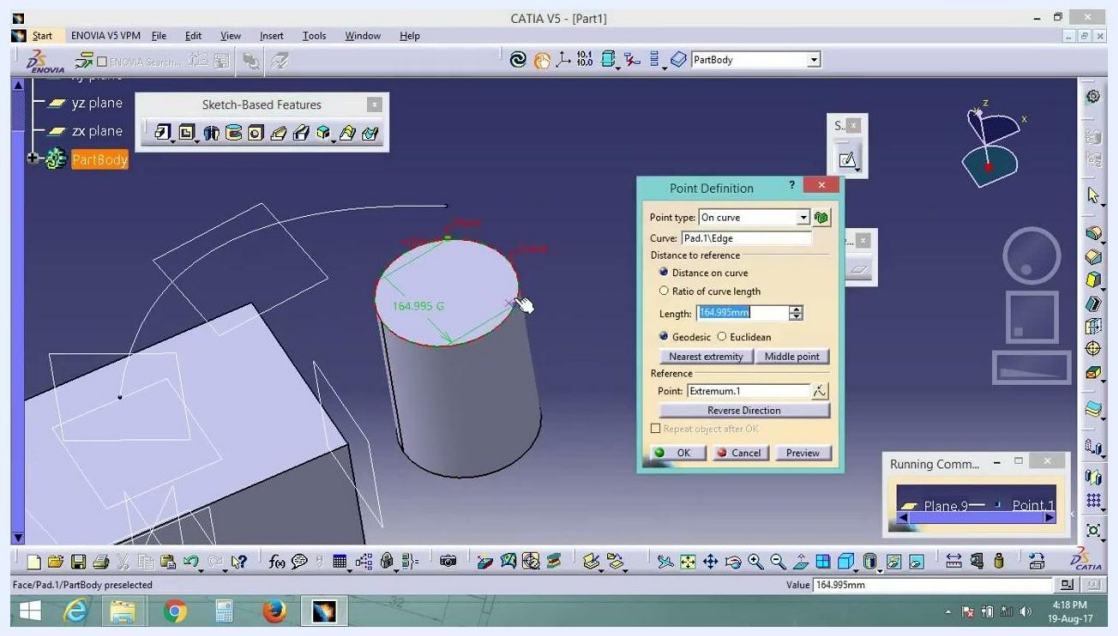


Point Definition dialog open with On curve selected, the curve set to Pad.1\Edge, the length set to 142.483mm, and the reference point set to Extremum.1

Step 12: Review the planes you created

1. Review the workspace and confirm that the new planes (through planar curve, normal to curve, and tangent to surface) are visible and correctly positioned relative to your model.

2. Close any open dialogs you no longer need, such as **Reference** or **Point Definition**.



CATIA V5 workspace showing multiple planes, a cylinder, a rectangular block, and the Reference dialog open, with all created planes visible in the 3D view

Summary

You now know how to create a plane in CATIA V5 using offsets, points, lines, angles, curves, and surfaces. These reference planes support precise modeling of complex geometries.

What's next

- Save your work.
- Review each plane for accuracy before you continue with further modeling operations.