How to apply a coincidence constraint in CATIA V5

This guide shows you how to apply a coincidence constraint in CATIA V5 to anchor a line's endpoints to vertical and horizontal reference lines. You draw a slanted line in the sketch environment, constrain each endpoint to a reference line, and test how the constraint limits movement. You then add dimensions to define the line fully.

Prerequisites

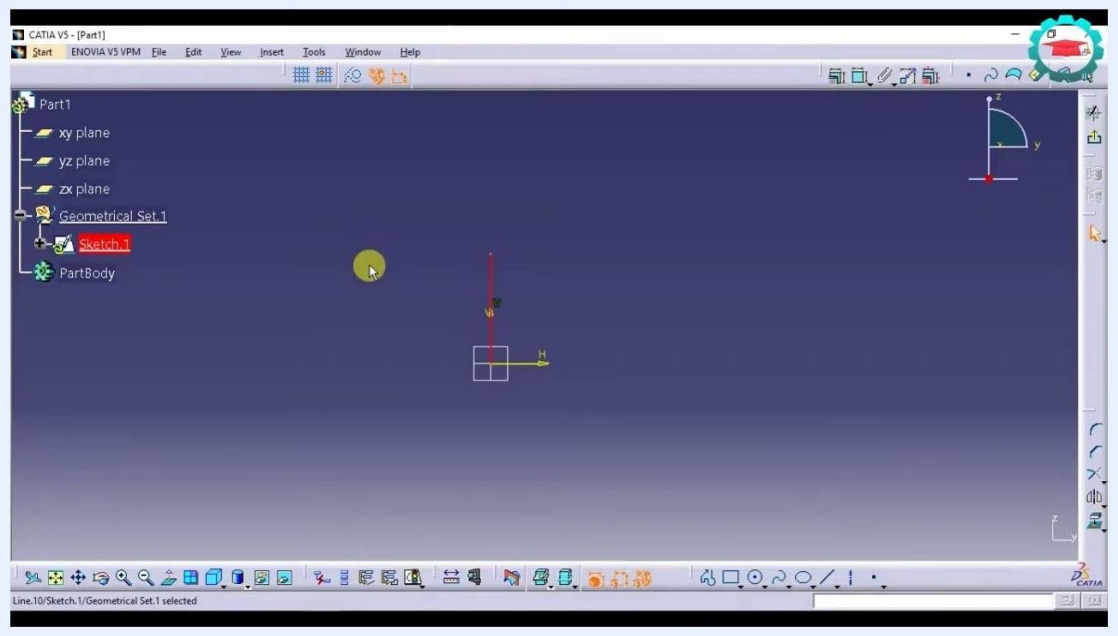
- CATIA V5 is installed and running.
- You have a part file that contains Sketch.1 under Geometrical Set.1.

Open the sketch environment

Step 1: Open the sketch

1. Launch CATIA V5 and open your part file.
2. In the tree on the left, double-click Sketch.1 under Geometrical Set.1 to enter the sketching environment.

The workspace shows the coordinate system and the reference planes (xy plane, yz plane, zx plane). A vertical reference line is visible in the sketch.



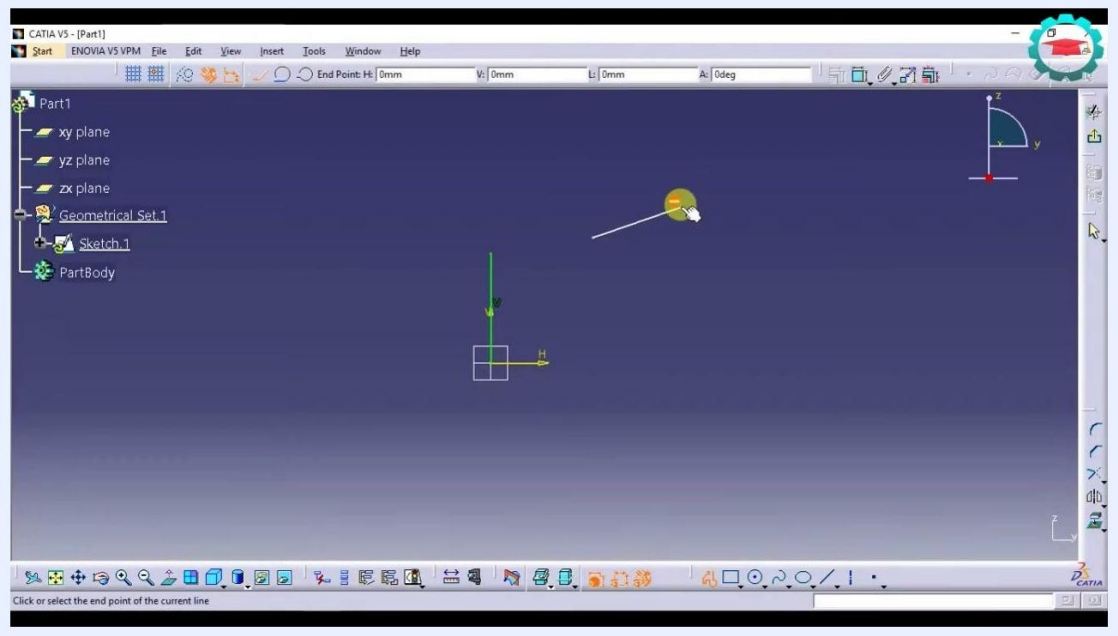
CATIA V5 sketch environment with a vertical reference line and coordinate axes. The tree on the left shows Part1, Geometrical Set.1, and Sketch.1 highlighted.

Draw the line to constrain

Step 2: Draw a new line in space

1. Select the line tool from the toolbar.
2. Click in the workspace to start a new line away from the origin and the reference lines.
3. Click again to complete the line.

You now have a slanted line in space.

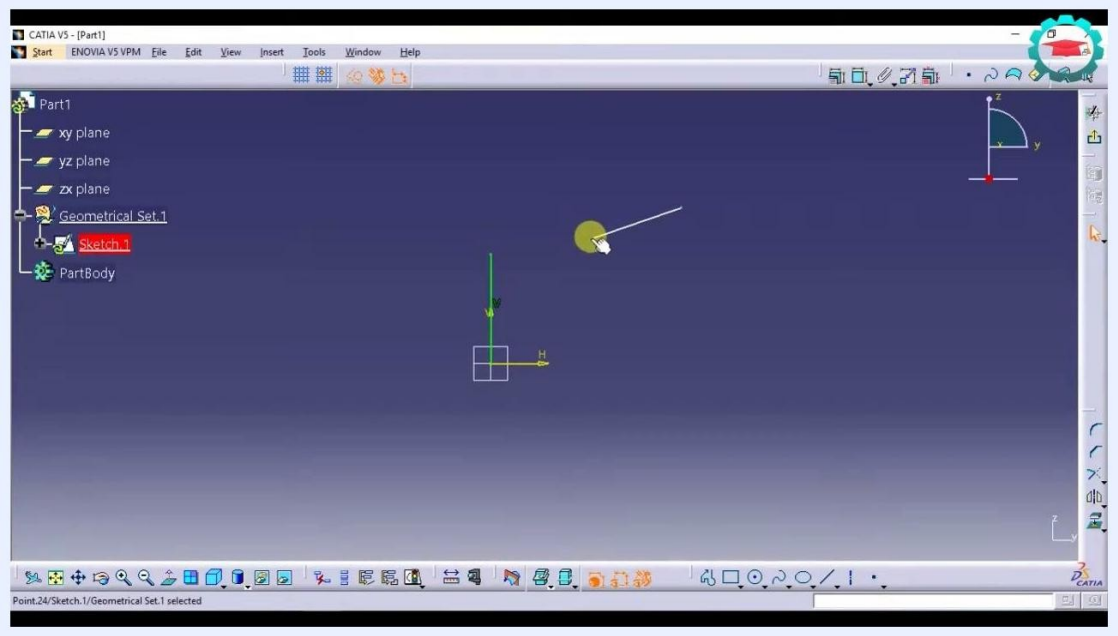


A new slanted line is drawn in the sketch, away from the vertical reference line. The cursor is at the endpoint of the new line.

Apply the first coincidence constraint

Step 3: Select the point and the reference line

1. Click the endpoint of the slanted line that you want to constrain. The selected point is highlighted.



The endpoint of the slanted line is selected and shown with a highlighted marker. The vertical reference line and the slanted line are both visible.

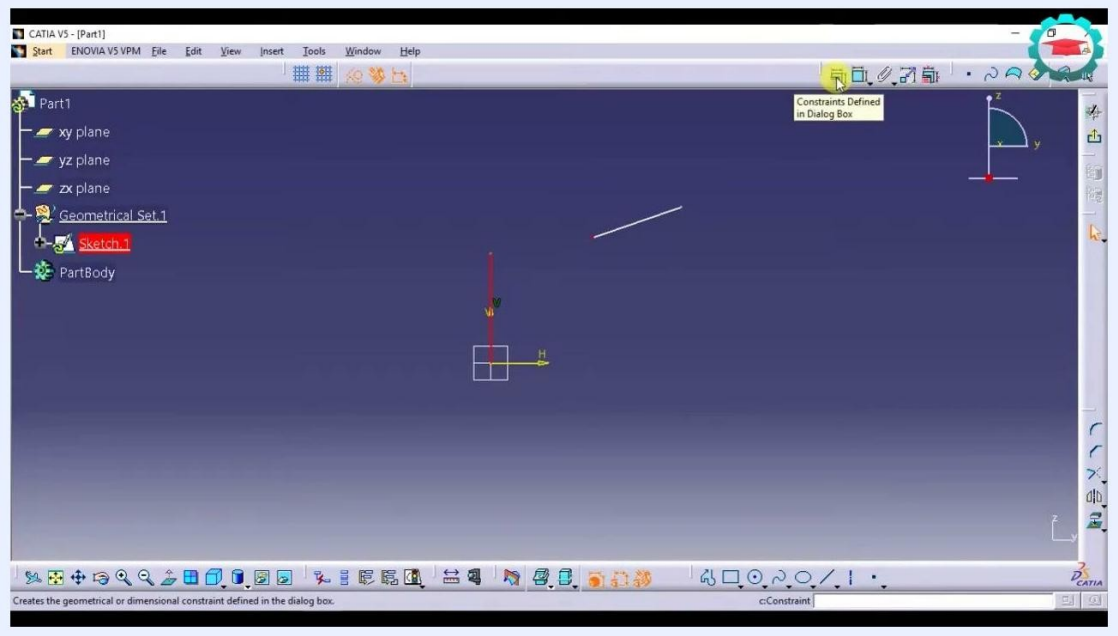
2. Hold Ctrl and select the vertical reference line in the sketch.

Both the endpoint and the vertical line are now selected.

Step 4: Apply the coincidence constraint

1. In the top toolbar, click the **Constraints Defined in Dialog Box** button (the icon with a constraint symbol).

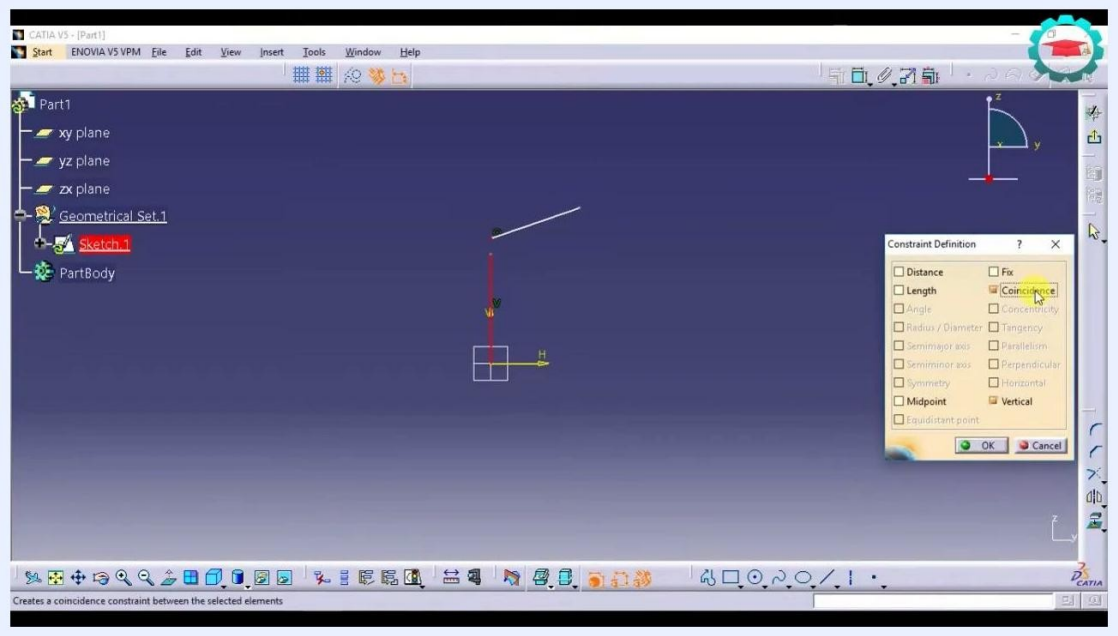
The **Constraint Definition** dialog box appears.



The Constraints Defined in Dialog Box button is highlighted in the toolbar. The workspace shows the selected point and the vertical line.

2. In the **Constraint Definition** dialog box, check **Coincidence**.
3. Click **OK**.

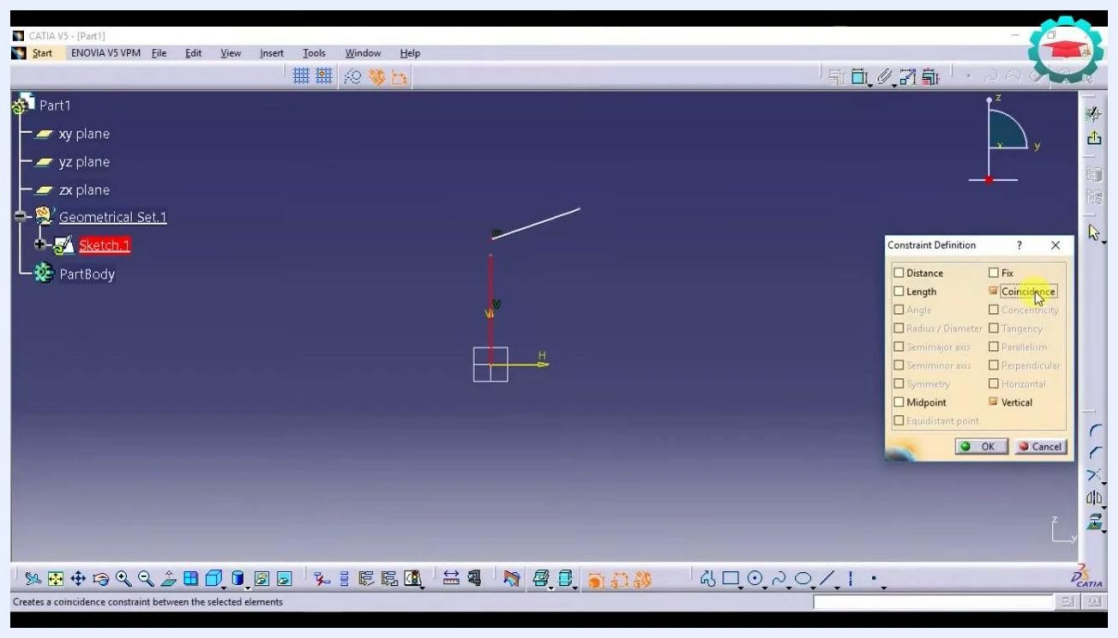
The endpoint of the slanted line is now coincident with the vertical reference line.



The Constraint Definition dialog box is open with the Coincidence option checked. The endpoint of the slanted line is coincident with the vertical reference line.

Step 5: Test the constraint

Try to move the slanted line. It moves only along the vertical reference line because its endpoint is constrained.



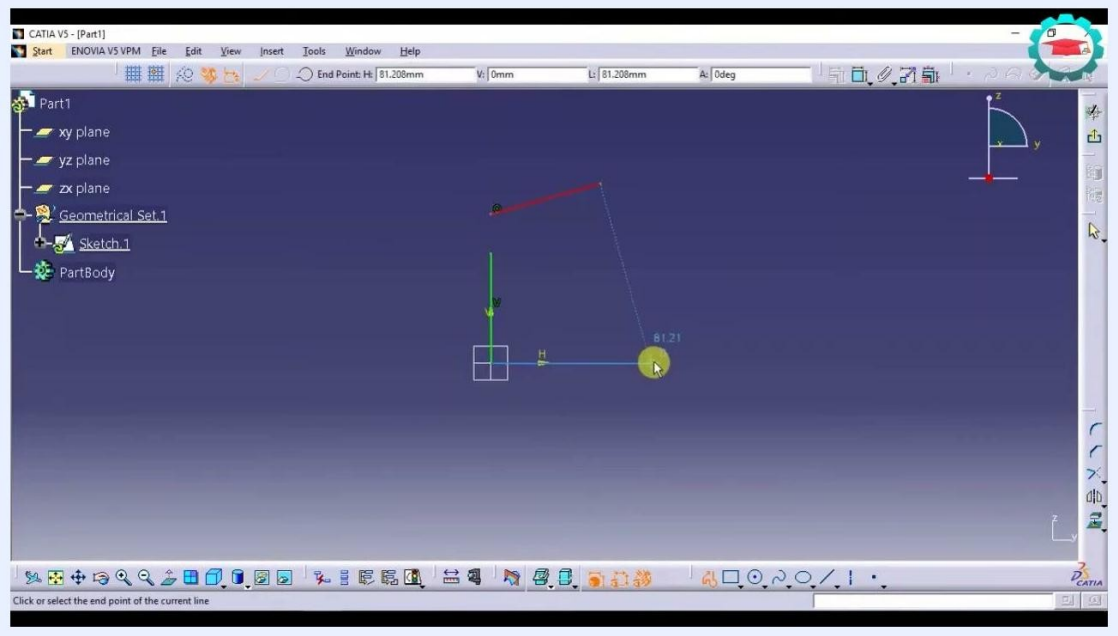
The slanted line is being moved, but its endpoint stays attached to the vertical reference line, which shows the effect of the coincidence constraint.

Apply a second coincidence constraint

Step 6: Draw a horizontal reference line

1. Select the line tool again.
2. Draw a horizontal line from the origin or another reference point.

The horizontal line appears in the sketch.



A horizontal reference line is drawn from the origin. The slanted line and the vertical reference line are visible.

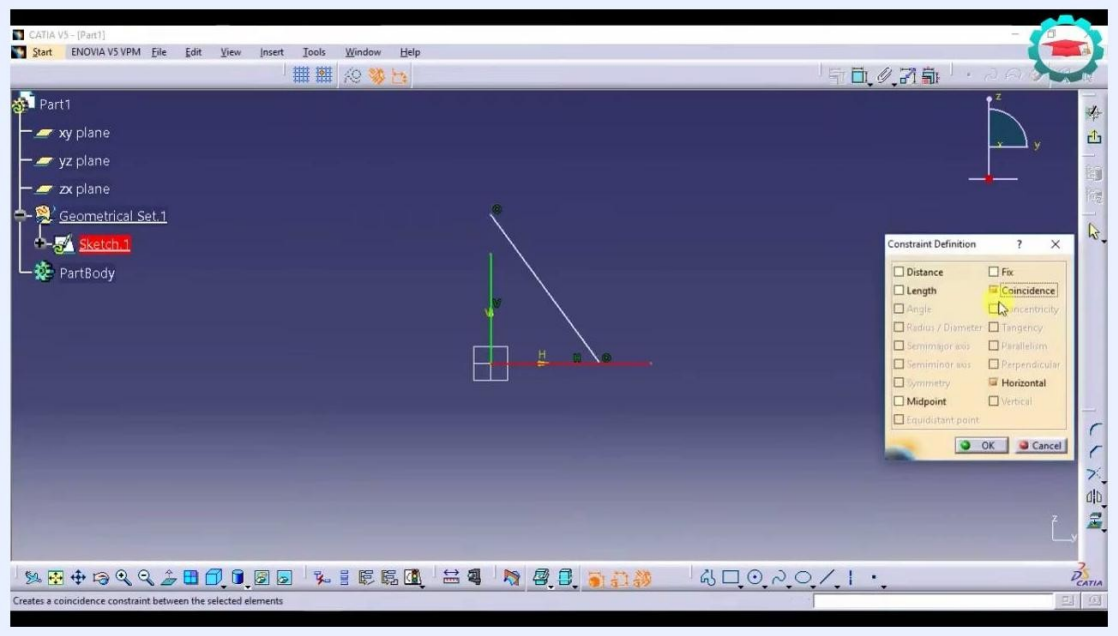
Step 7: Select the other endpoint and the horizontal line

1. Click the other endpoint of the slanted line.
2. Hold Ctrl and select the horizontal reference line.

Step 8: Apply the second coincidence constraint

1. Open the **Constraint Definition** dialog box again.
2. Check **Coincidence**.
3. Click **OK**.

Both endpoints of the slanted line are now coincident: one with the vertical line and one with the horizontal line.



The Constraint Definition dialog box is open with Coincidence selected. Both endpoints of the slanted line are coincident with the vertical and horizontal reference lines.

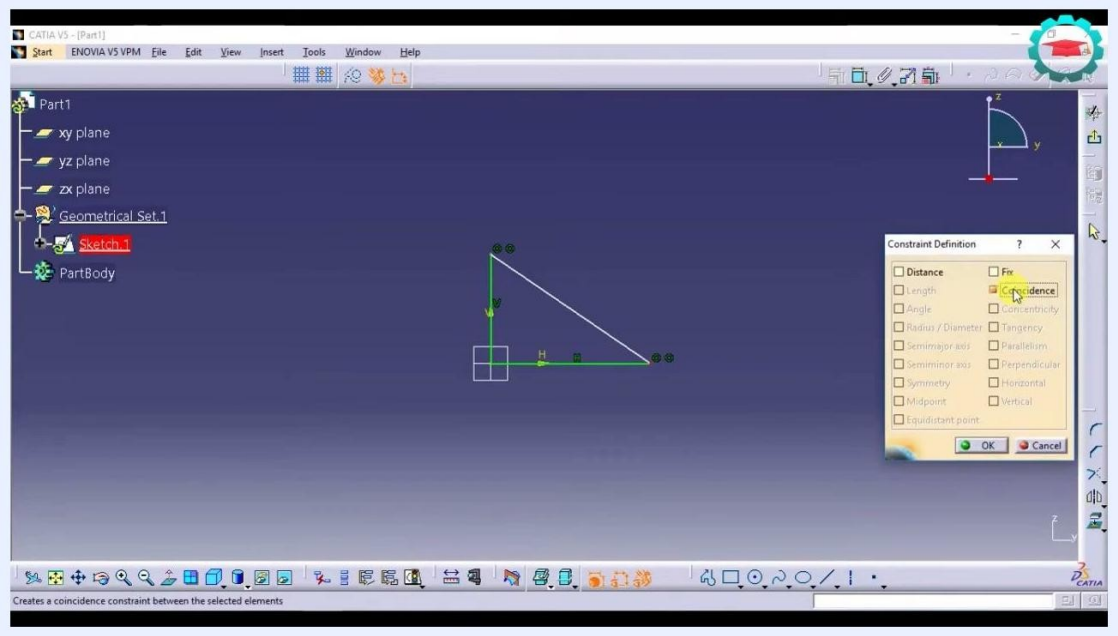
Add dimensions to complete the definition

Step 9: Dimension the constrained line

The slanted line is now fully constrained and cannot move independently. Its endpoints are locked to the intersection points of the reference lines. You can add dimensions to define it completely.

1. Select the dimension tool from the toolbar.
2. Click the vertical reference line and the coincident point to add a vertical dimension (for example, 50).
3. Click the horizontal reference line and the coincident point to add a horizontal dimension (for example, 85).

The line is now fully defined by both the coincidence constraints and the dimensions.



The sketch shows the slanted line with its endpoints coincident to the reference lines, and the dimensions 50 (vertical) and 85 (horizontal).

Summary

You applied coincidence constraints to a line in CATIA V5 and anchored its endpoints to a vertical and a horizontal reference line. These constraints restrict how the line can move. Adding dimensions then defined the line fully.

What's next

- Repeat the steps with other lines or points in your sketch.
- Use dimensions with coincidence constraints to define your sketch geometry fully.