How to use stiffener in CATIA V5

The Stiffener command adds a rib or stringer to a part or wall. It increases the part's strength without breaking the part. This guide shows you how to use stiffener in CATIA V5 in two scenarios: between two flat surfaces, and between a flat surface and a round surface.

Prerequisites

- CATIA V5 with access to the Part Design workbench
- Basic familiarity with sketching, such as drawing profiles and applying constraints

Scenarios covered

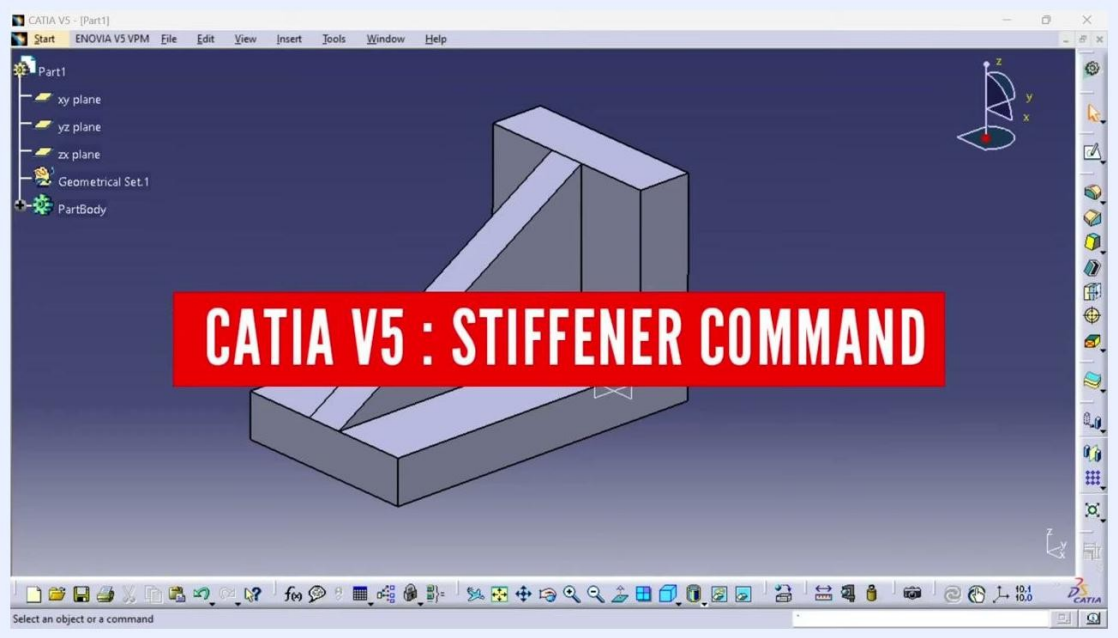
- Both surfaces are flat.
- One surface is flat and the other is round.

Create a stiffener between two flat surfaces

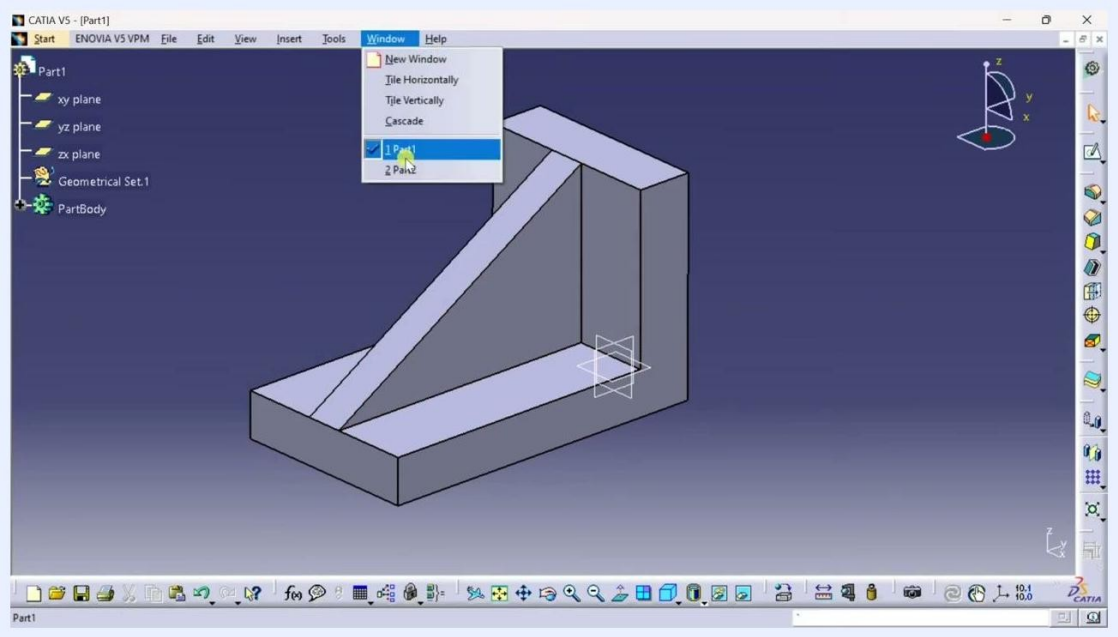
In this scenario, you create a pad, sketch a line across it, and use that line to build the stiffener.

Step 1: Open the Part Design workbench

Open the Part Design workbench in CATIA V5. You see the default interface, with the model tree on the left and the 3D workspace in the center.



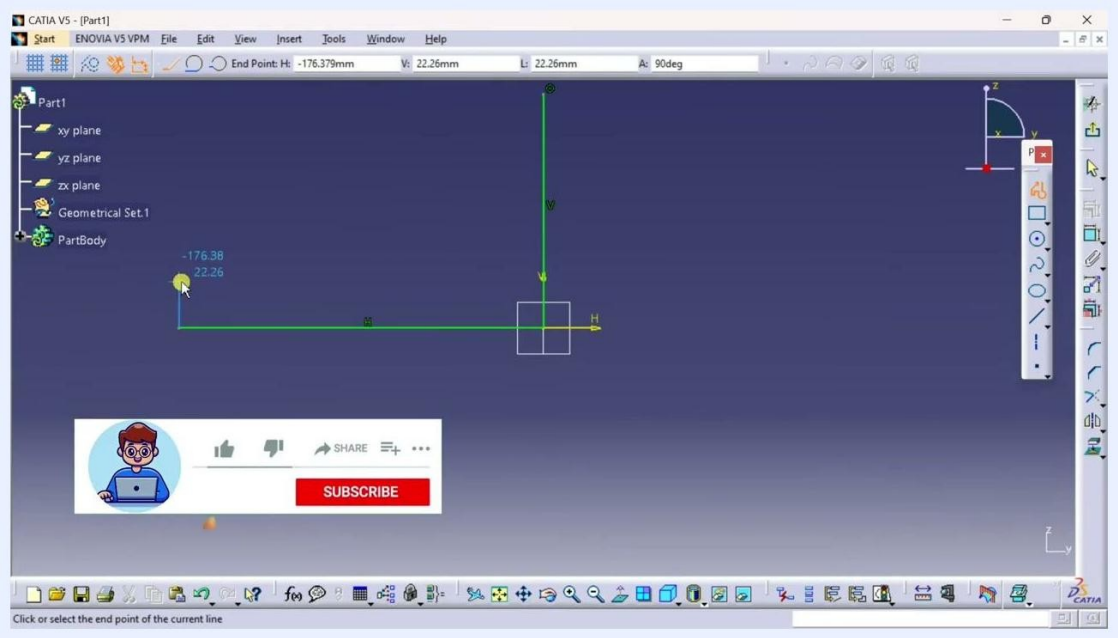
CATIA V5 interface showing a 3D model with a stiffener and the text "CATIA V5: STIFFENER COMMAND" overlaid



The CATIA V5 Part Design workbench with an empty workspace and the model tree on the left

Step 2: Make PartBody the active work object

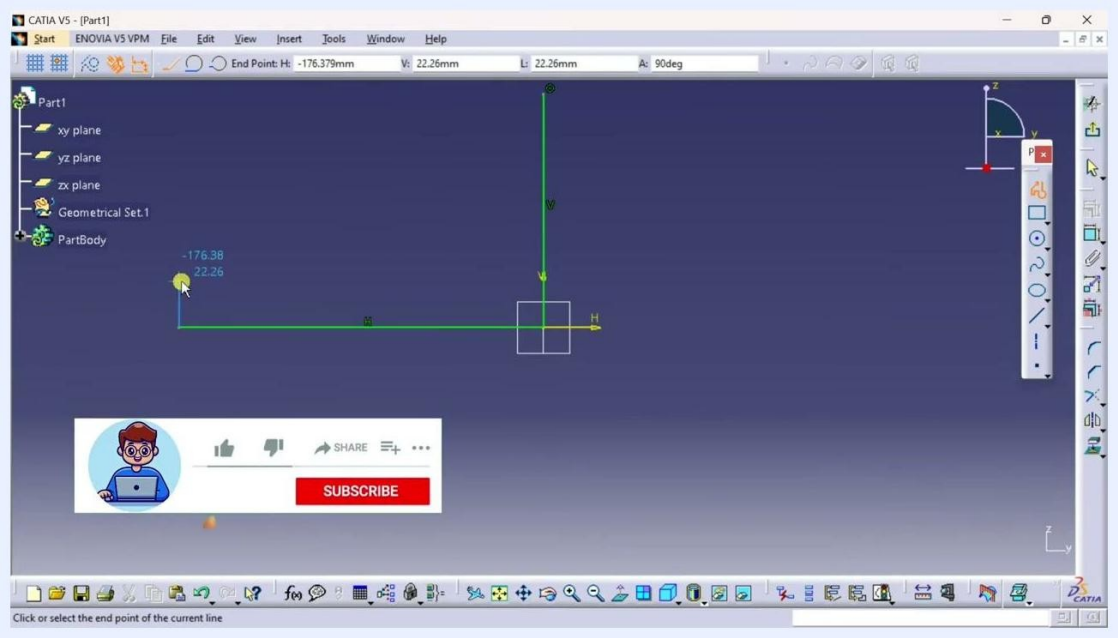
Before sketching, right-click **PartBody** in the model tree and select **Define In Work Object**. This ensures all new features are added to the correct part body.



The right-click menu on PartBody with Define In Work Object highlighted

Step 3: Start a sketch on a plane

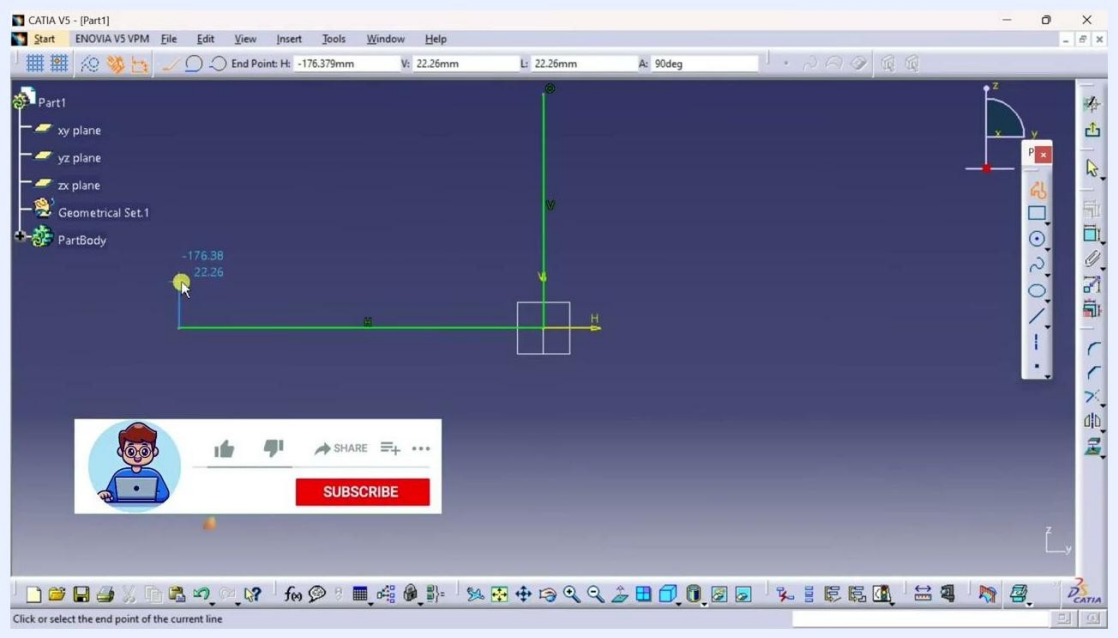
Click the **Sketch** command and select a plane (xy, yz, or zx). The selected plane is highlighted, and the sketching grid appears in the workspace.



Selecting a sketch plane in CATIA V5 with the cursor over a plane

Step 4: Draw the profile

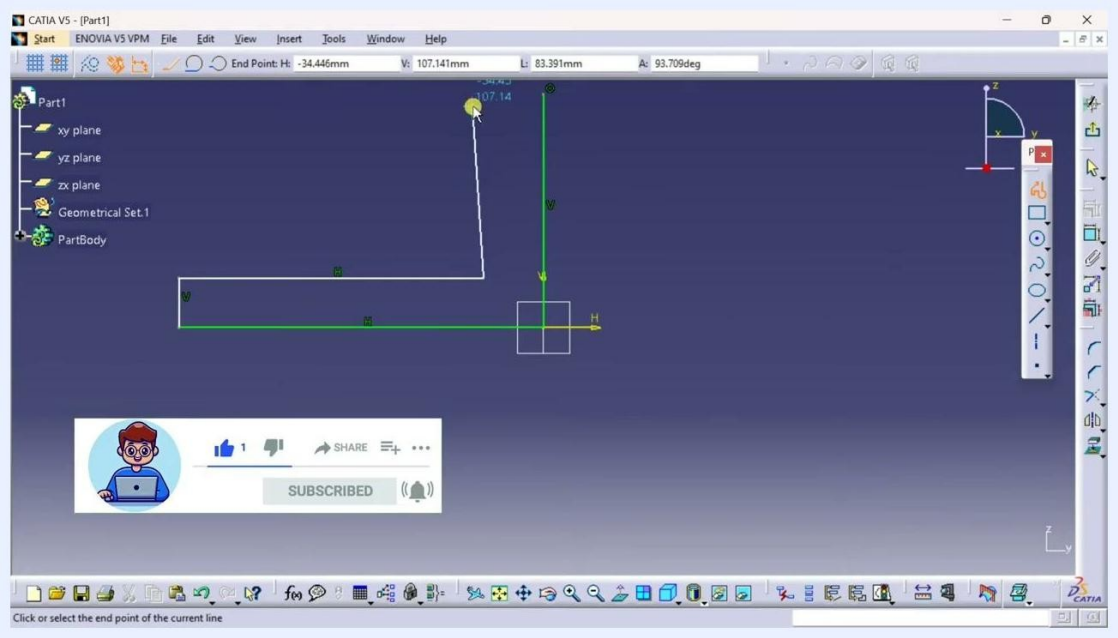
Use the **Profile** command to draw the sketch on the selected plane. Draw lines as needed to define the profile. The interface displays coordinates and dimensions as you draw.



A profile sketch in progress with green lines and dimension indicators

Step 5: Exit the sketch

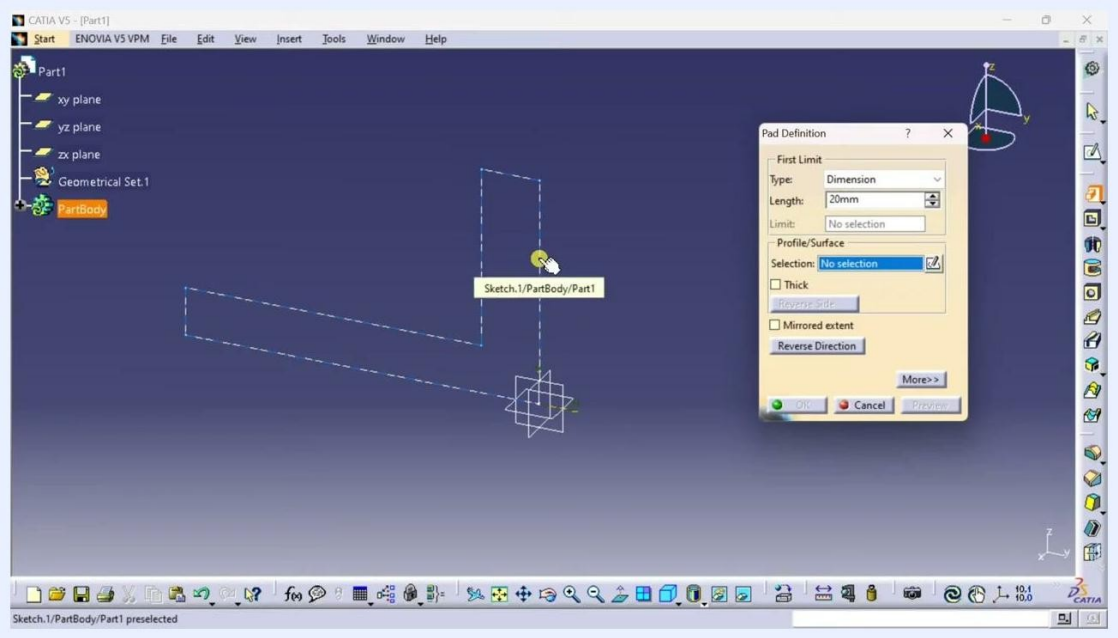
Press **Esc**, then click **Exit Workbench** to return to the main modeling environment. The sketch is saved and ready to use.



A completed profile sketch with dimensions and the Exit Workbench option visible

Step 6: Select the sketch for the pad

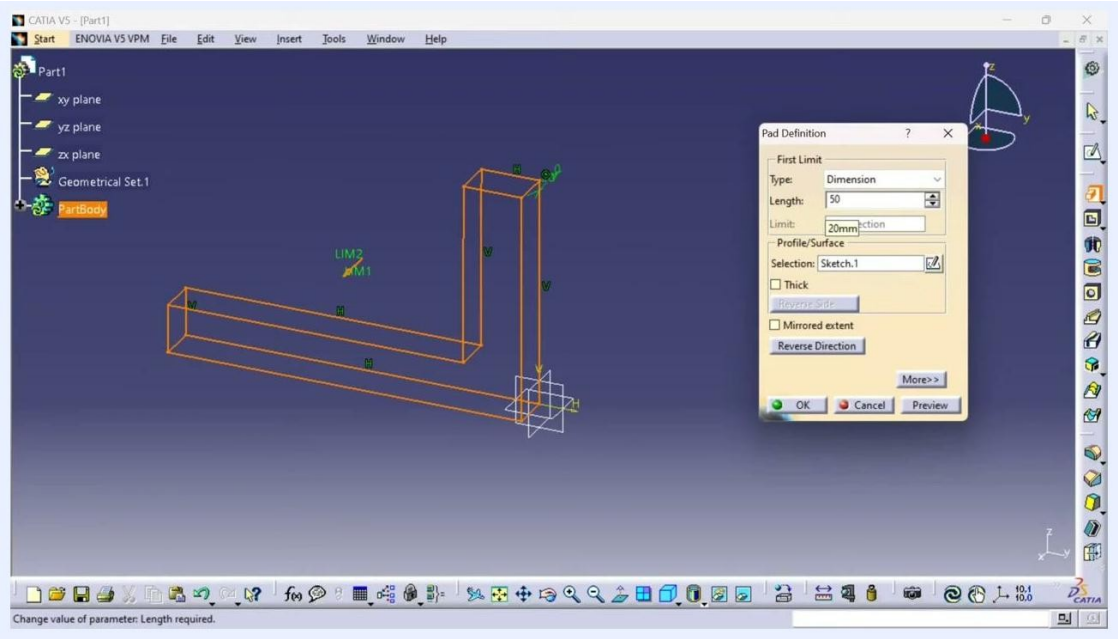
Click the **Pad** command in the Part Design workbench. In the **Pad Definition** dialog, click the **Selection** box under **Profile/Surface**. Select your sketch in the workspace or model tree. The selected sketch is highlighted.



The Pad Definition dialog open with the sketch selected and the profile highlighted

Step 7: Set the pad length and mirrored extent

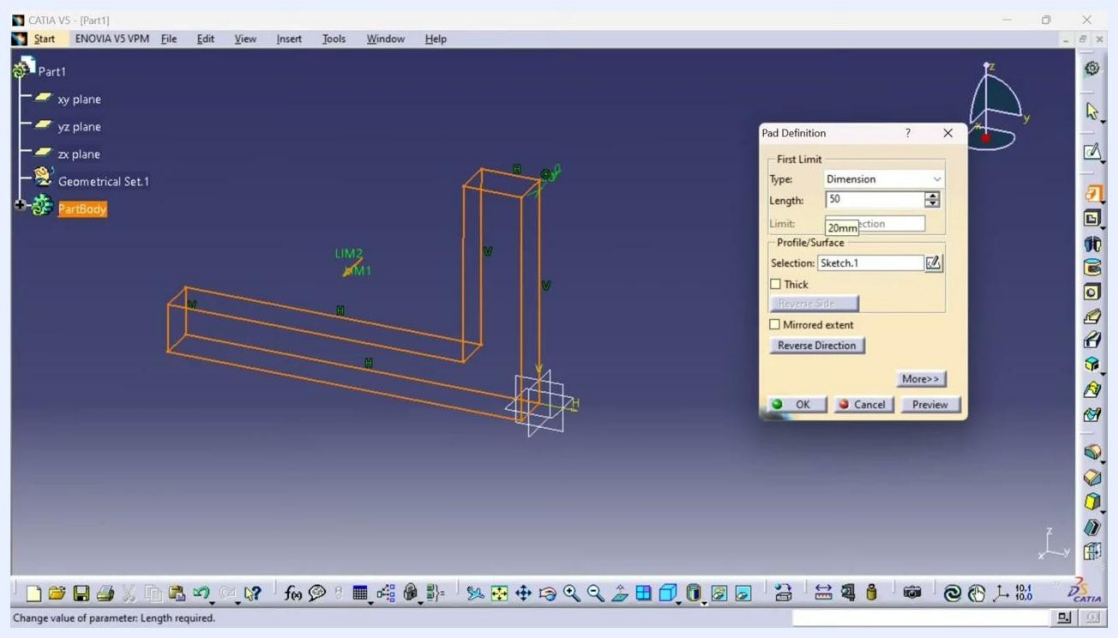
In the **Length** field, enter the pad length, for example **50 mm**. To extend the pad equally in both directions from the sketch plane, check **Mirrored extent**. The preview shows the pad extending equally on both sides of the sketch plane.



The Pad Definition dialog with Length set to 50 mm, Mirrored extent checked, and a preview of the pad

Step 8: Create the pad

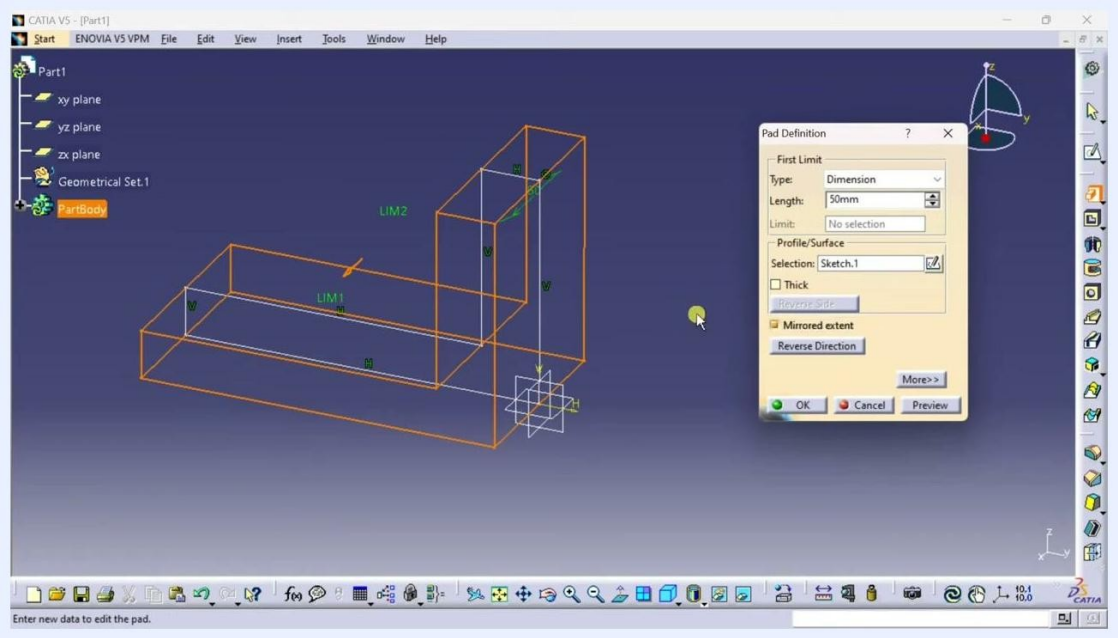
Click **Preview** to check the result, then click **OK**. The 3D model updates to show the new solid feature.



The resulting 3D pad after applying the Pad command

Step 9: Sketch the stiffener line

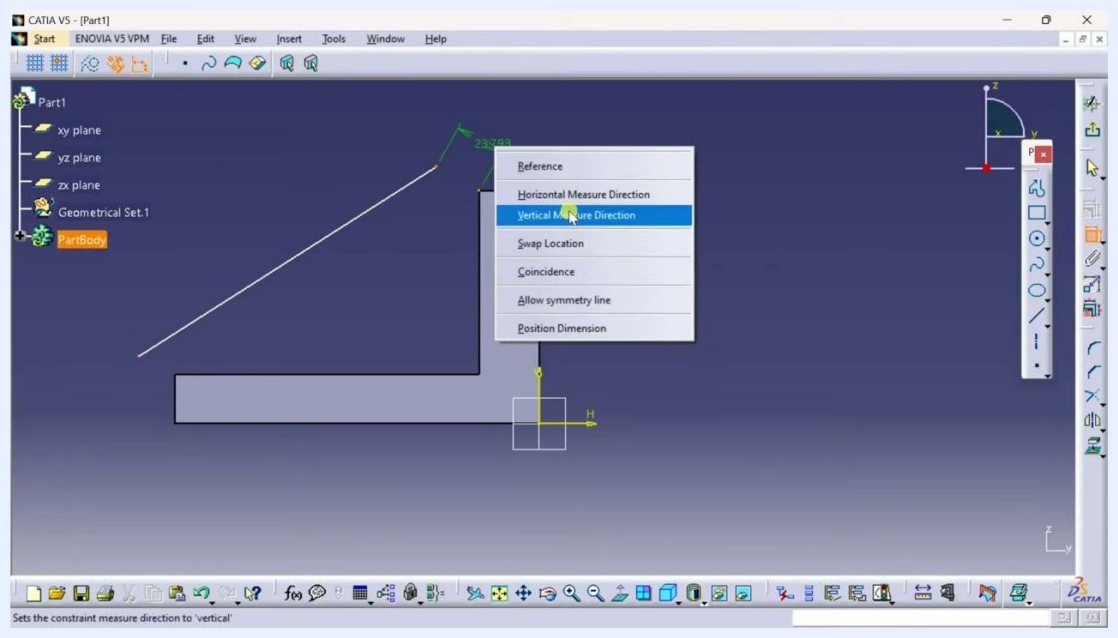
Click the **Sketch** command and select the plane or planar face you want on your model. Click the **Line** tool in the sketch toolbar, then click the start and end points of your stiffener line. Press **Esc** to stop the line command.



The sketch environment with a diagonal line drawn for the stiffener, showing coordinates and dimensions

Step 10: Open the Constraint command

Constrain the line so that it is fully defined. Click the **Constraint** command in the sketch toolbar.

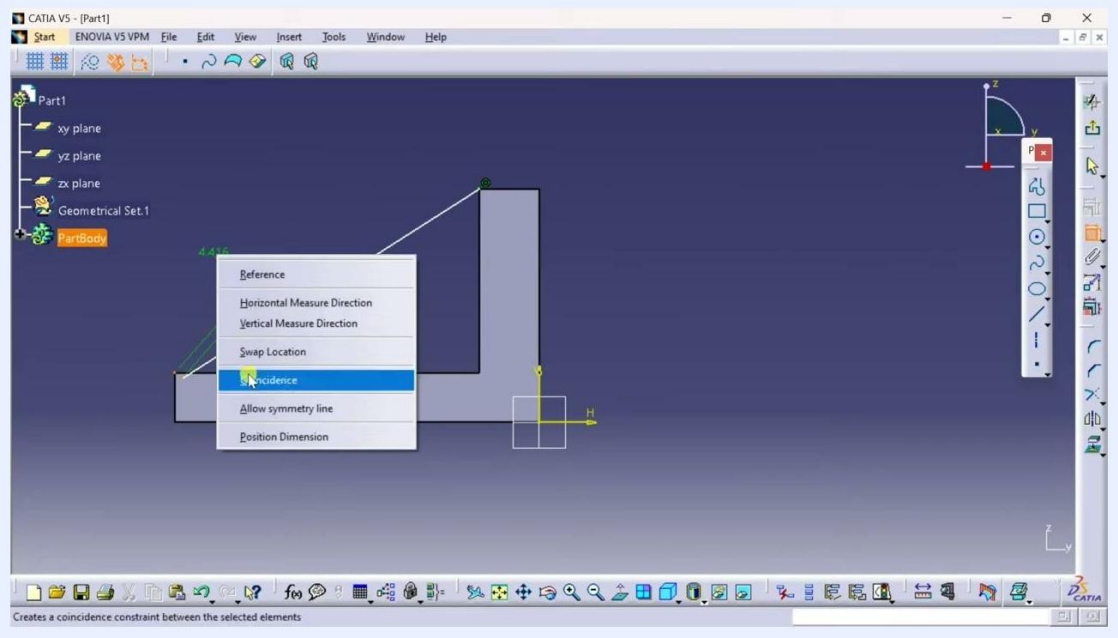


The sketch environment with the Constraint tool highlighted, ready to constrain the line

Step 11: Apply coincidence constraints

Right-click an endpoint of the line and select **Coincidence** from the context menu. Then select the edge of the sketch or model that you want to attach the line to. Repeat for the other endpoint as needed.

When the line turns green, it is fully constrained and fixed in position.

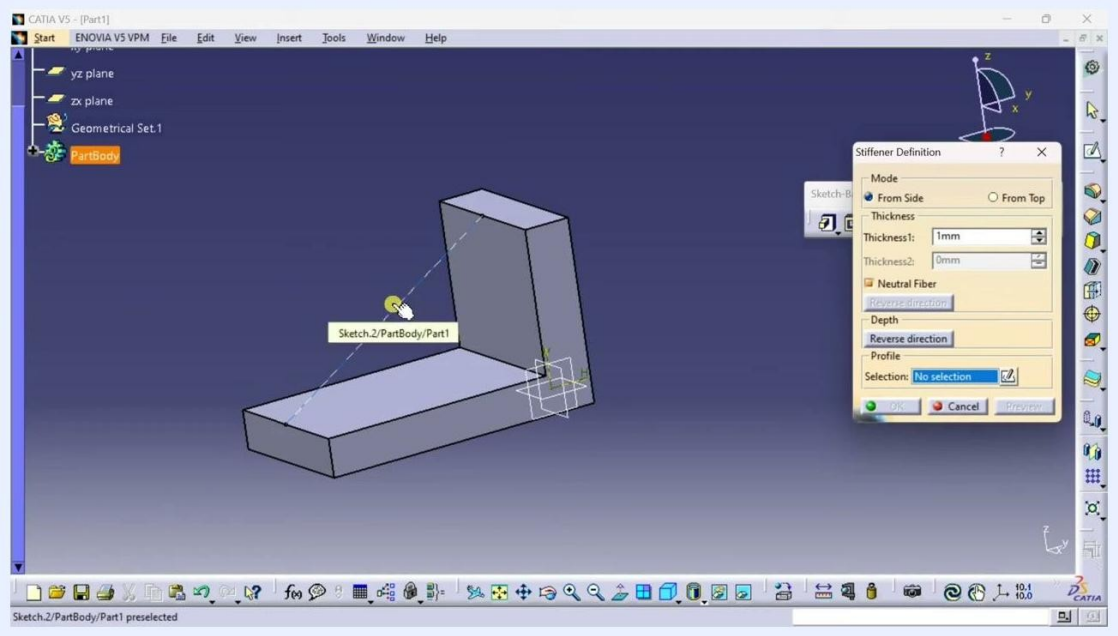


The line fully constrained and green, with the context menu open on an endpoint

Step 12: Exit the sketch and open the Sketch-Based Features toolbar

Click **Exit Workbench** to leave the sketch environment and return to the Part Design workbench. Make sure the line is ready to use as the stiffener profile.

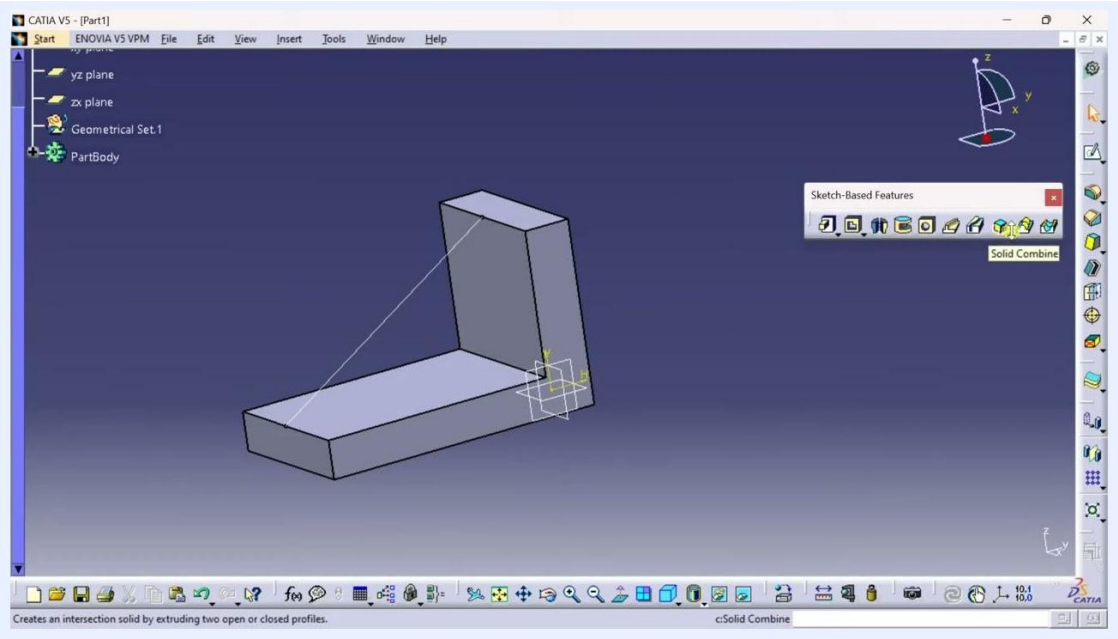
Locate the **Sketch-Based Features** toolbar. If necessary, drag it to a convenient place on your screen.



The Sketch-Based Features toolbar being accessed, with the 3D model and stiffener line visible

Step 13: Access the Stiffener command

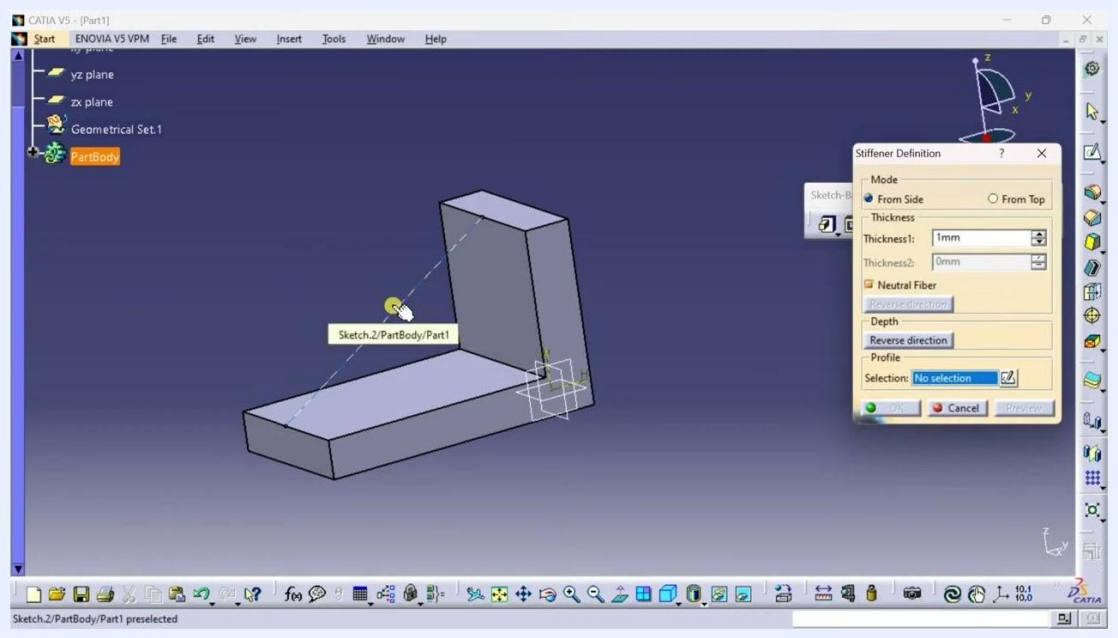
This step is the starting point for how to use stiffener in CATIA V5. In the **Sketch-Based Features** toolbar, click the **Solid Combine** icon. Click the small arrow next to it to expand more options, then select **Stiffener** from the dropdown.



The Sketch-Based Features toolbar open with the Solid Combine option highlighted

Step 14: Select the stiffener profile

In the **Stiffener Definition** dialog, go to the **Profile** section and click the **Selection** box. Click the line you drew, for example Sketch.2/PartBody/Part1. The selected line is highlighted in the 3D view.



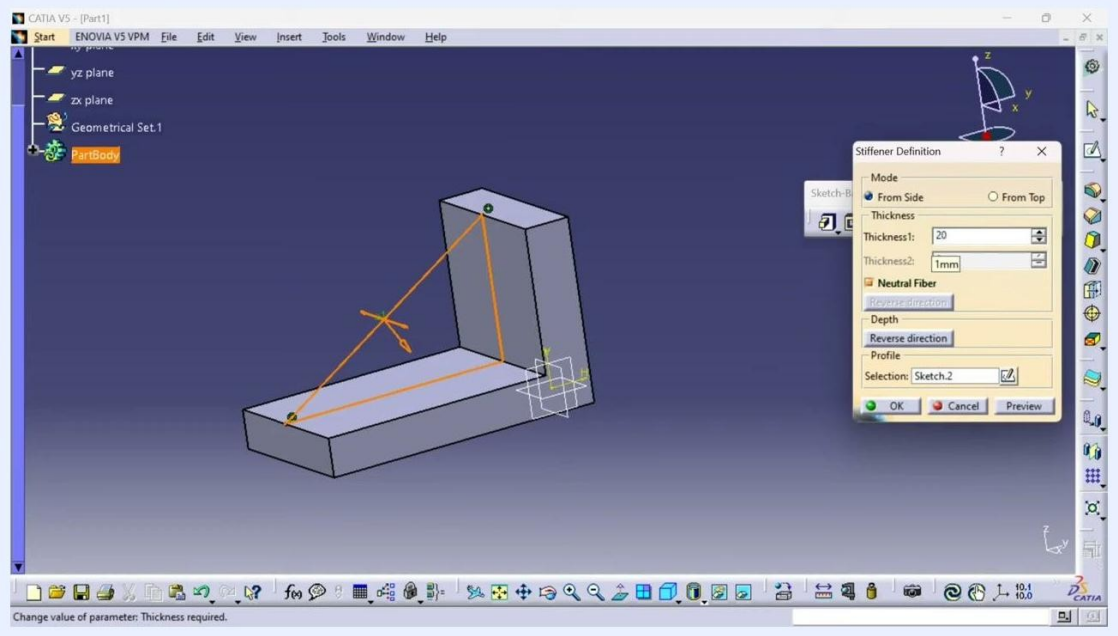
The Stiffener Definition dialog open with the stiffener line being selected as the profile

Step 15: Set the stiffener thickness

In the **Stiffener Definition** dialog, set these values:

- Set **Thickness1** to **20mm**.
- Set **Thickness2** to **0mm**, if it is not already set.
- Check **Neutral Fiber** to distribute the thickness equally on both sides of the profile, similar to the Mirrored extent option.

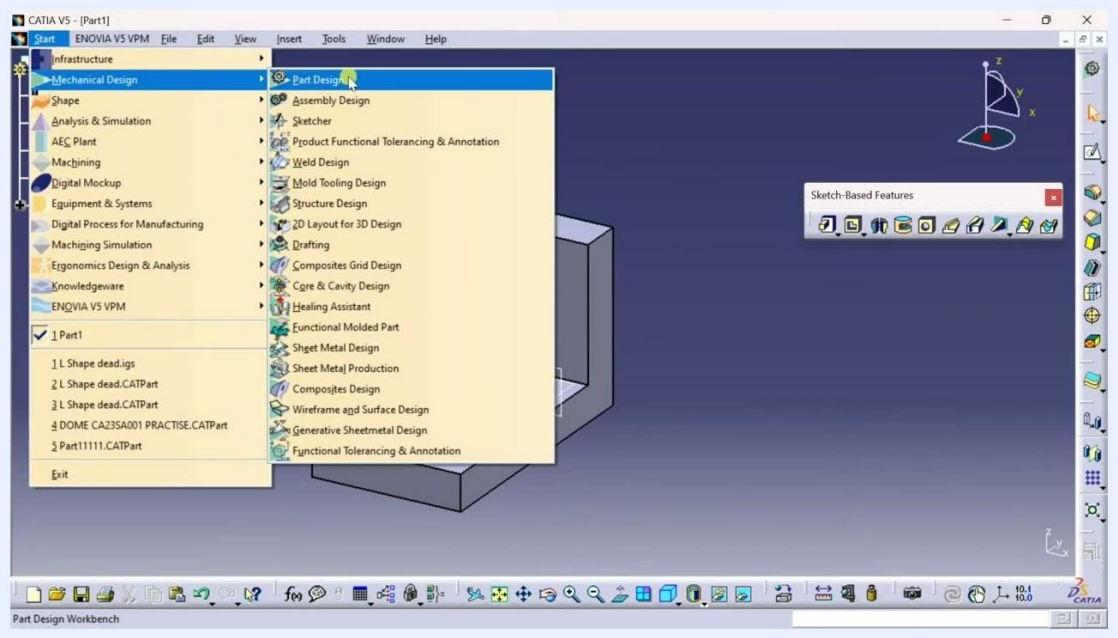
Optionally, adjust the **Mode** (From Side or From Top) and the **Depth**.



The Stiffener Definition dialog with Thickness1 set to 20mm, Neutral Fiber checked, and the profile selected

Step 16: Preview and create the stiffener

Click **Preview** in the **Stiffener Definition** dialog to see the stiffener. If you are satisfied, click **OK**. The 3D model updates to show the new stiffener connecting the two flat surfaces.



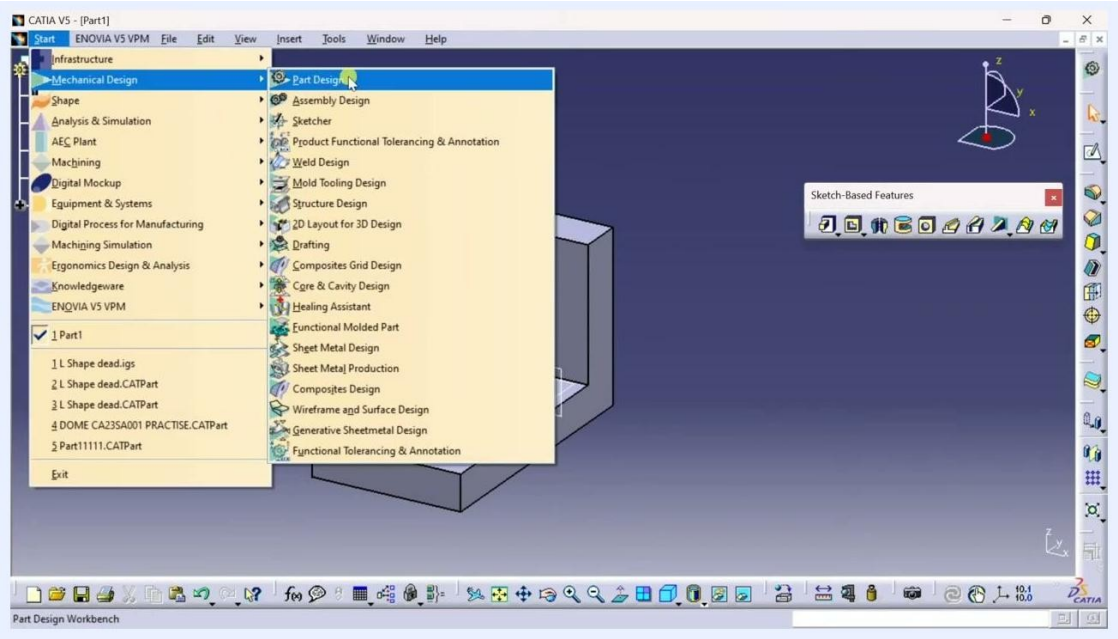
The Stiffener Definition dialog with the OK button highlighted and the stiffener preview visible in the 3D model

Create a stiffener between a flat and a round surface

In this scenario, you extrude a circle and a rectangle with Multi-Pad, then add a stiffener between the resulting cylinder and the rectangle.

Step 17: Open a new part

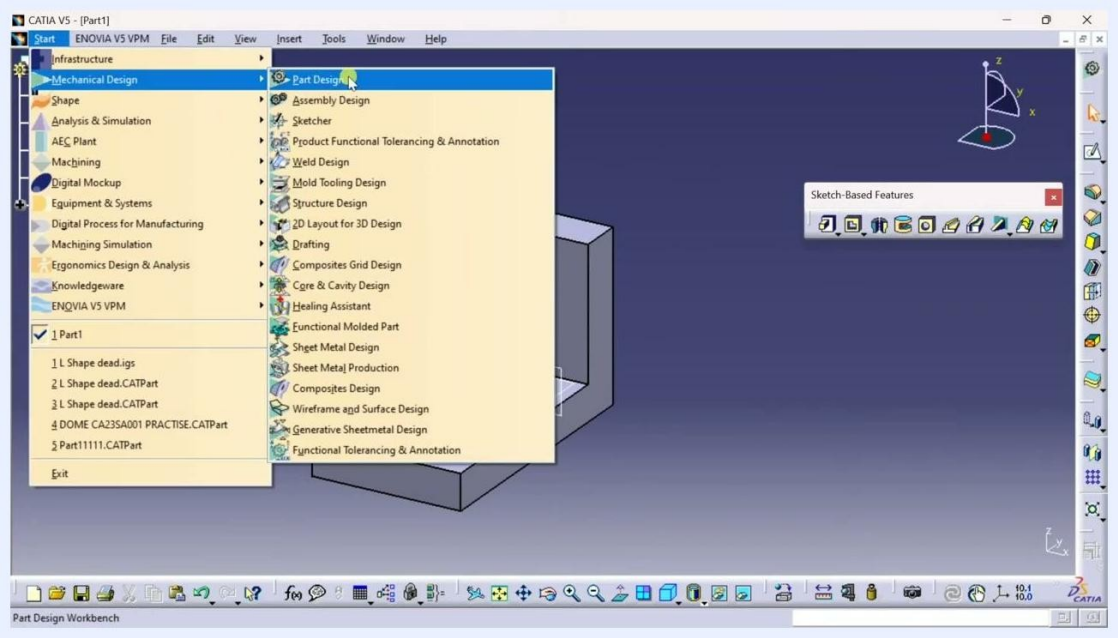
Click **Start** in the top menu and go to **Mechanical Design** > **Part Design**. In the **New Part** dialog, enter a name for your part, for example "Part2". Keep the default options (Enable hybrid design, Create a geometrical set) as needed, then click **OK**.



The Start menu open with Mechanical Design and Part Design highlighted

Step 18: Prepare the new part for sketching

In the new part, right-click **PartBody** in the tree and select **Define In Work Object**. This makes PartBody the active work object.



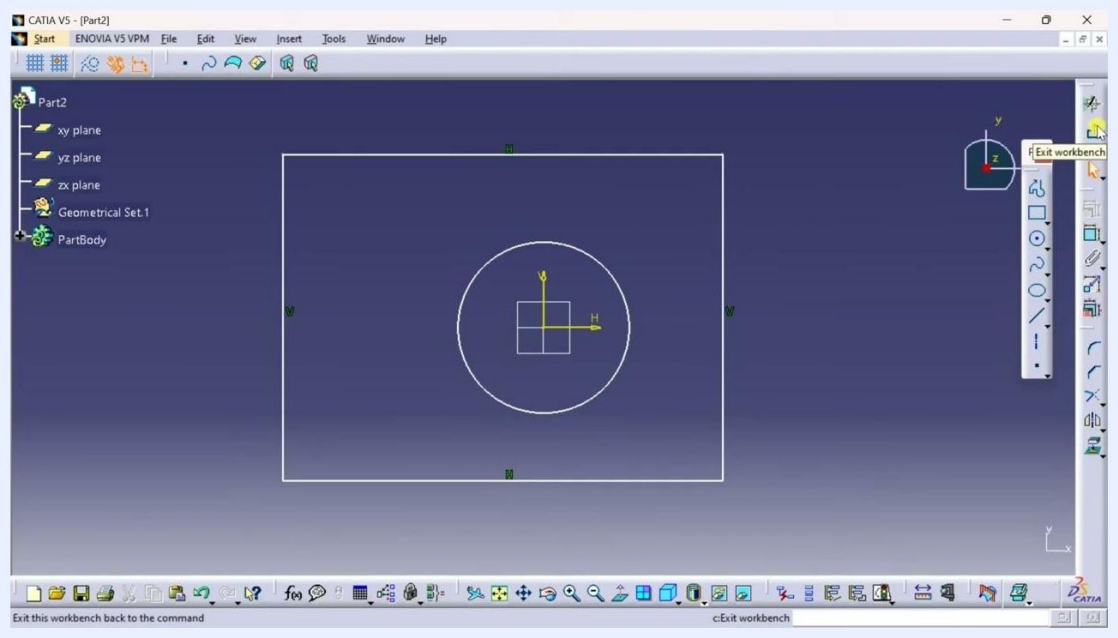
The right-click menu on PartBody with Define In Work Object highlighted

Step 19: Remove unwanted sketches

If an unwanted sketch exists on the selected plane, right-click it and delete or clear it before you continue. The next steps use a sketch that contains a circle and a rectangle.

Step 20: Exit the sketch workbench

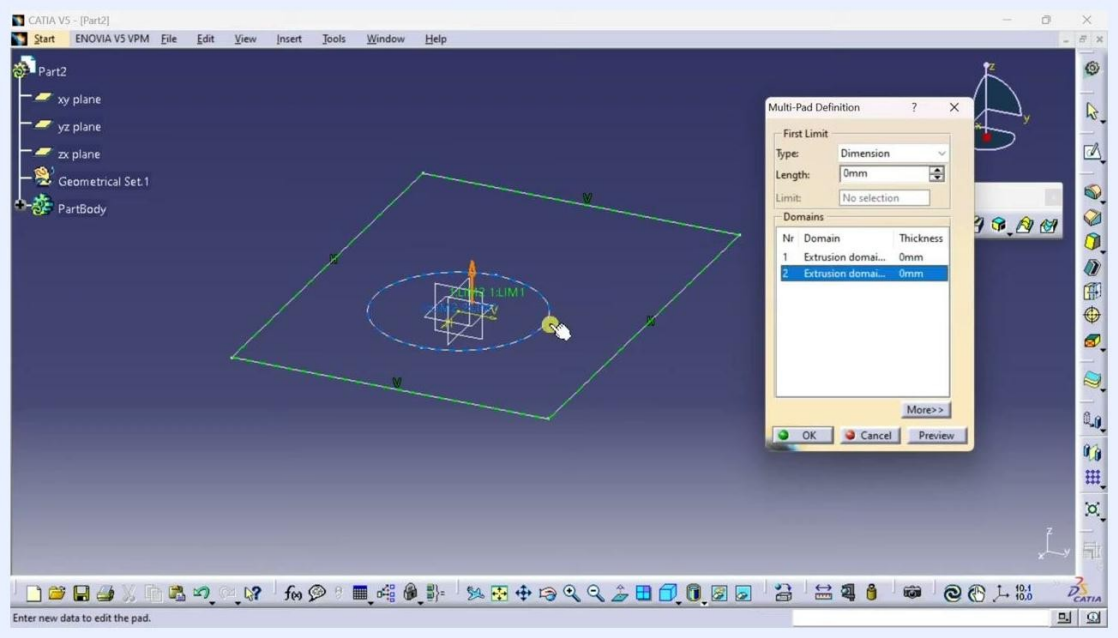
Click the **Exit workbench** icon on the right toolbar. This returns you to the main Part Design workbench. The rectangle and circle remain visible.



The Exit workbench button highlighted on the right toolbar, with the rectangle and circle still visible

Step 21: Start the Multi-Pad feature

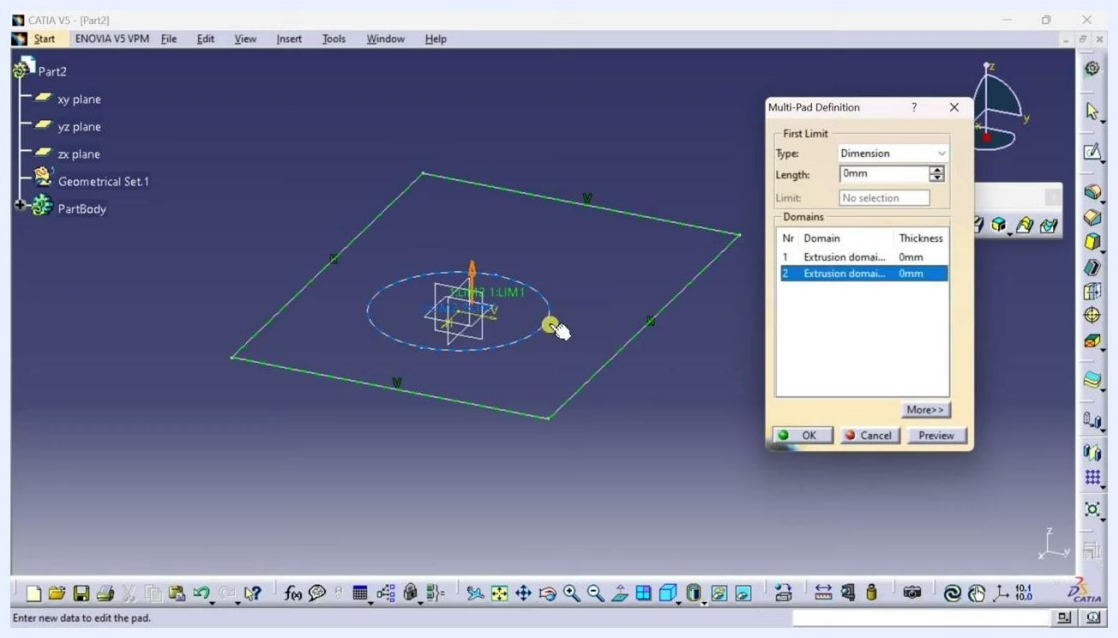
In the **Sketch-Based Features** toolbar, click the **Multi-Pad** icon. This lets you create extrusions from multiple domains in the sketch.



The Multi-Pad icon highlighted in the Sketch-Based Features toolbar, with the rectangle and circle visible in 3D

Step 22: Select the circle as a domain

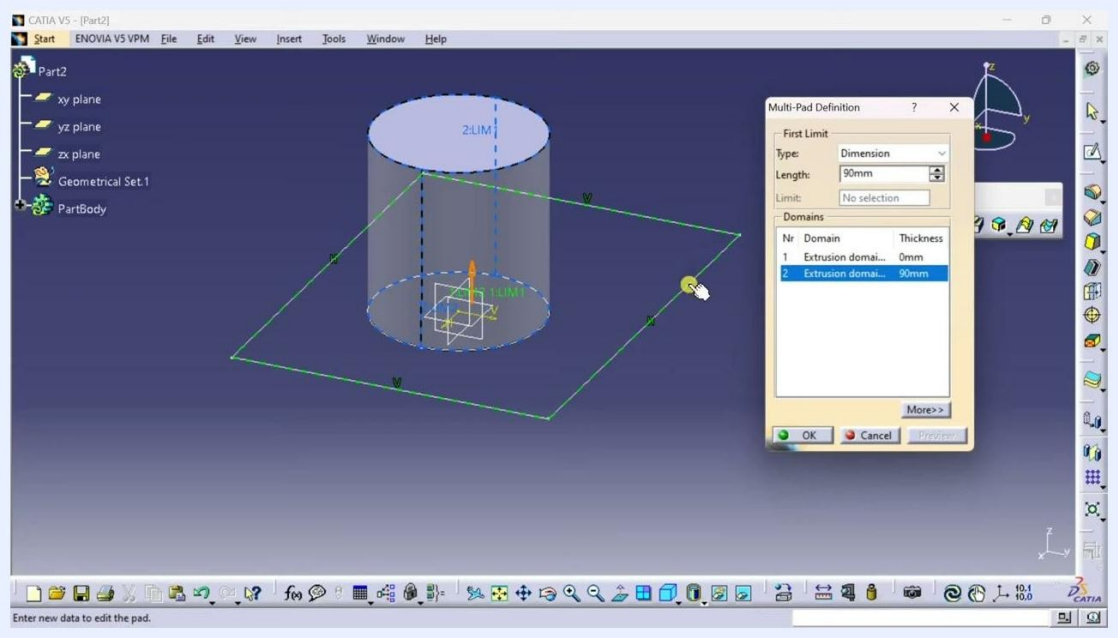
In the **Multi-Pad Definition** dialog, click the circle in the sketch. The circle turns blue, and the domain list in the dialog updates to show the selection.



The Multi-Pad Definition dialog open with the circle highlighted in blue and the domain list showing the selection

Step 23: Set the circle's extrusion length

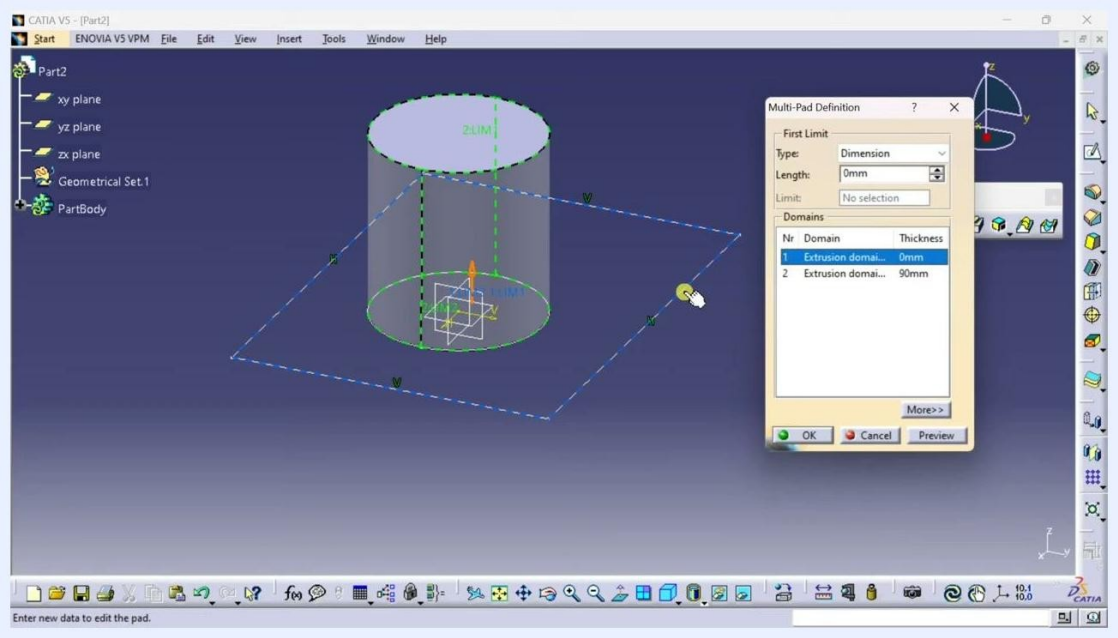
Set **Length** for the circle domain to **90mm**. The preview shows a cylinder extruded from the circle.



A cylinder extruded from the circle, with the Multi-Pad Definition dialog showing Length = 90mm

Step 24: Select the rectangle as a domain

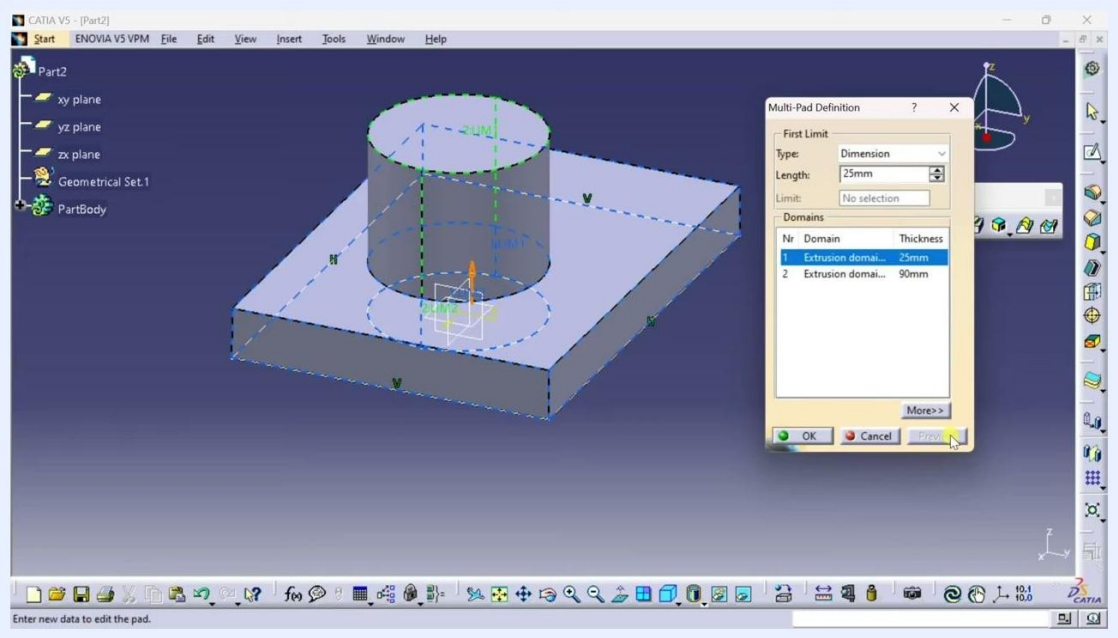
Click the rectangle in the sketch. The rectangle turns blue, and the domain list updates to show it as a domain.



The rectangle highlighted in blue, with the Multi-Pad Definition dialog showing both domains

Step 25: Set the rectangle's extrusion length

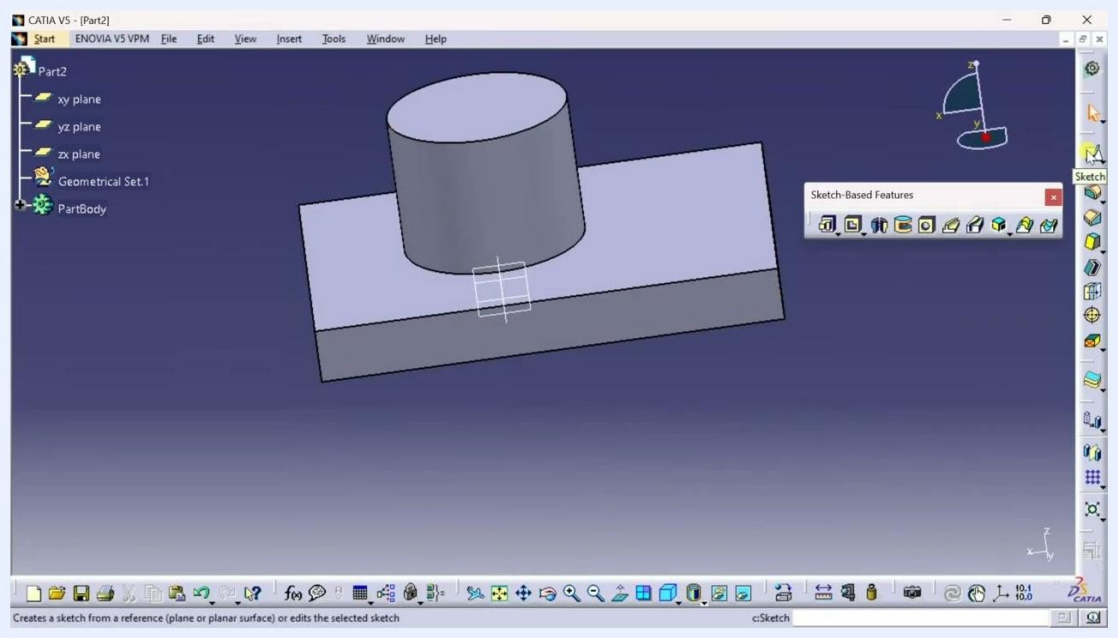
Set **Length** for the rectangle domain to **25mm**. The preview shows both the cylinder and the extruded rectangle. Click **OK** to apply the Multi-Pad operation.



Both the cylinder and the extruded rectangle, with the Multi-Pad Definition dialog showing Length = 25mm for the rectangle

Step 26: Create a new sketch

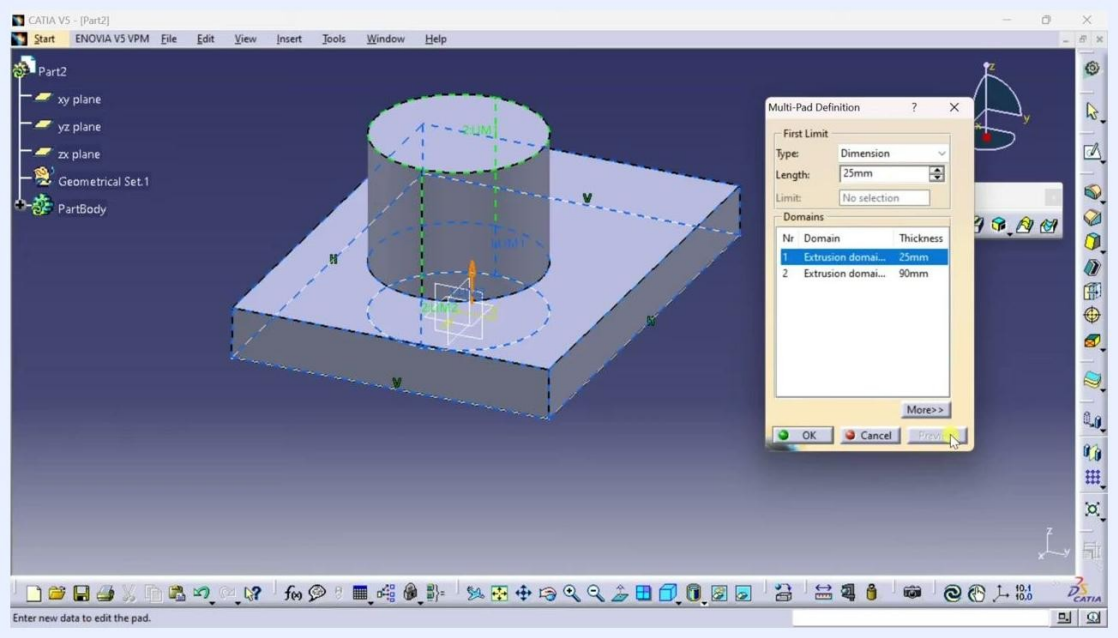
In the **Sketch-Based Features** toolbar, click the **Sketch** icon. The model is ready for a new sketch on a selected face or plane.



The Sketch-Based Features toolbar open, with the cylinder on the rectangle ready for a new sketch

Step 27: Select the rectangle's face

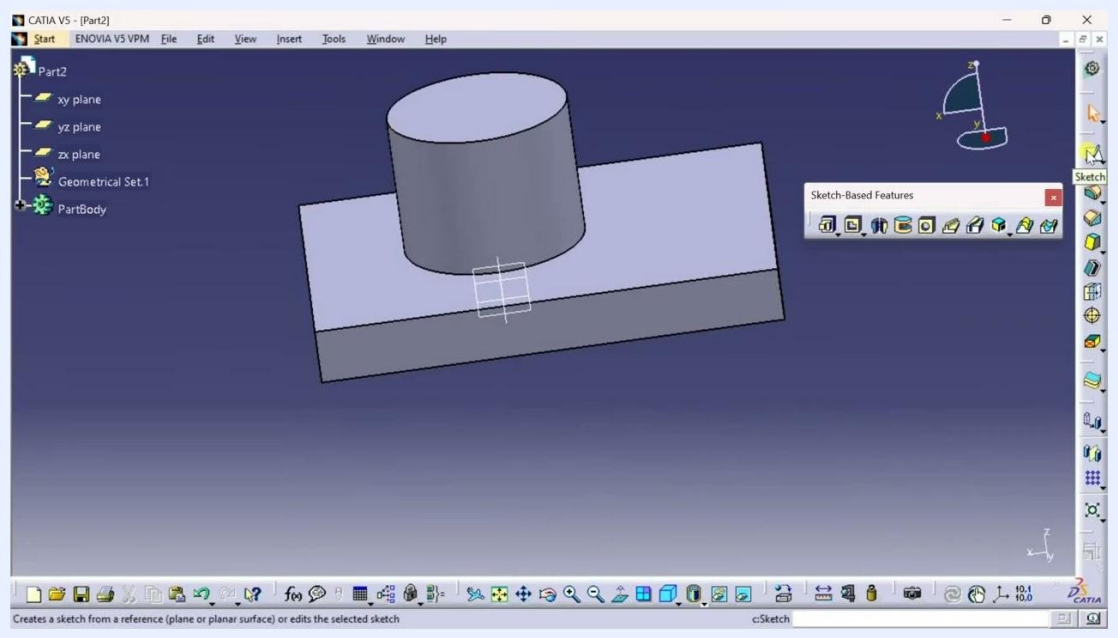
In the side view, position the cursor over the top face of the rectangle and select it. Coordinate values appear for the selected point.



The side view with the cursor selecting the top face of the rectangle and coordinate values shown

Step 28: Project 3D elements as reference lines

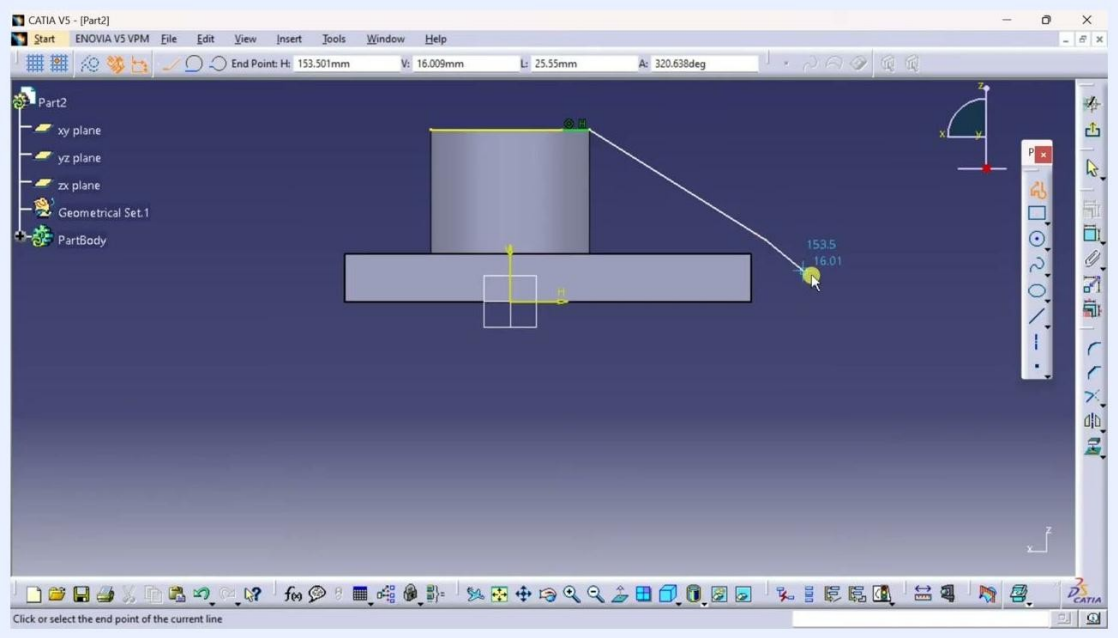
Use the **Project 3D Elements** command to project edges from the 3D model onto the sketch plane. These reference lines help you position new sketch elements accurately.



Projected reference lines on the sketch plane, ready to use as references for the new sketch

Step 29: Draw the stiffener line

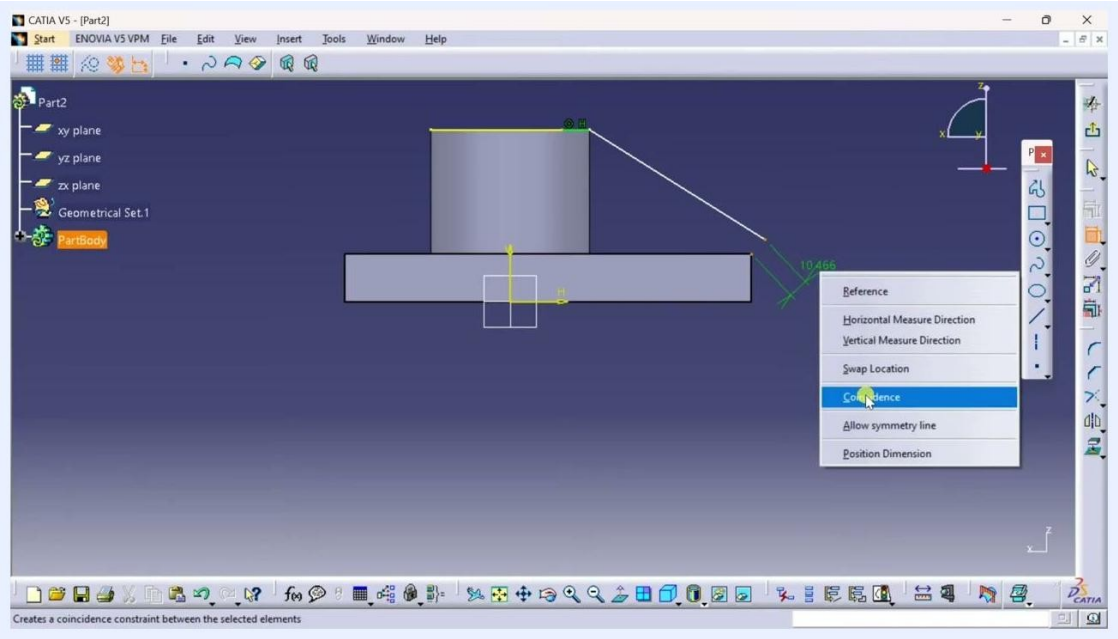
Draw a diagonal line from the top right of the cylinder to the lower right of the rectangle. The line is white, with endpoints highlighted in green and blue. At the lower endpoint, the status bar shows "End Point: H 153.501mm V 16.009mm L 25.55mm A 320.638deg". You are still in the sketcher workbench, and the line is ready to be constrained.



A diagonal line drawn from the top right of the cylinder to the lower right of the rectangle, with endpoints highlighted and coordinate values shown

Step 30: Apply a coincidence constraint

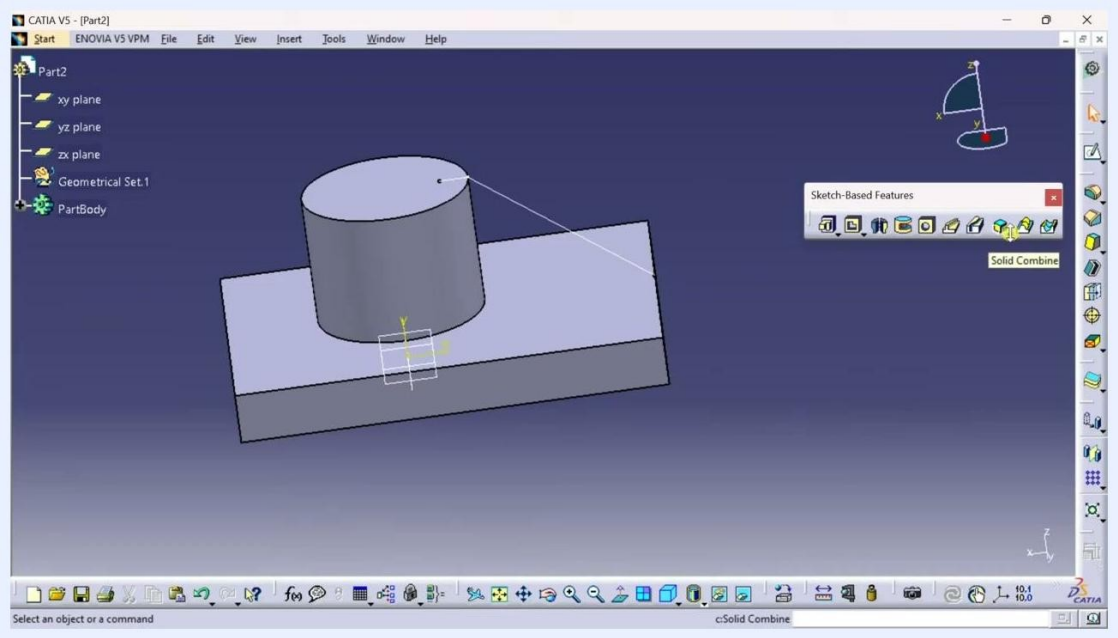
Right-click an endpoint of the line to open the context menu. It lists Reference, Horizontal Measure Direction, Vertical Measure Direction, Swap Location, Coincidence, Allow symmetry line, and Position Dimension. Select **Coincidence** to constrain the endpoint to the selected reference.



The context menu open with the cursor on Coincidence to constrain the line endpoint

Step 31: Convert the line to a construction element

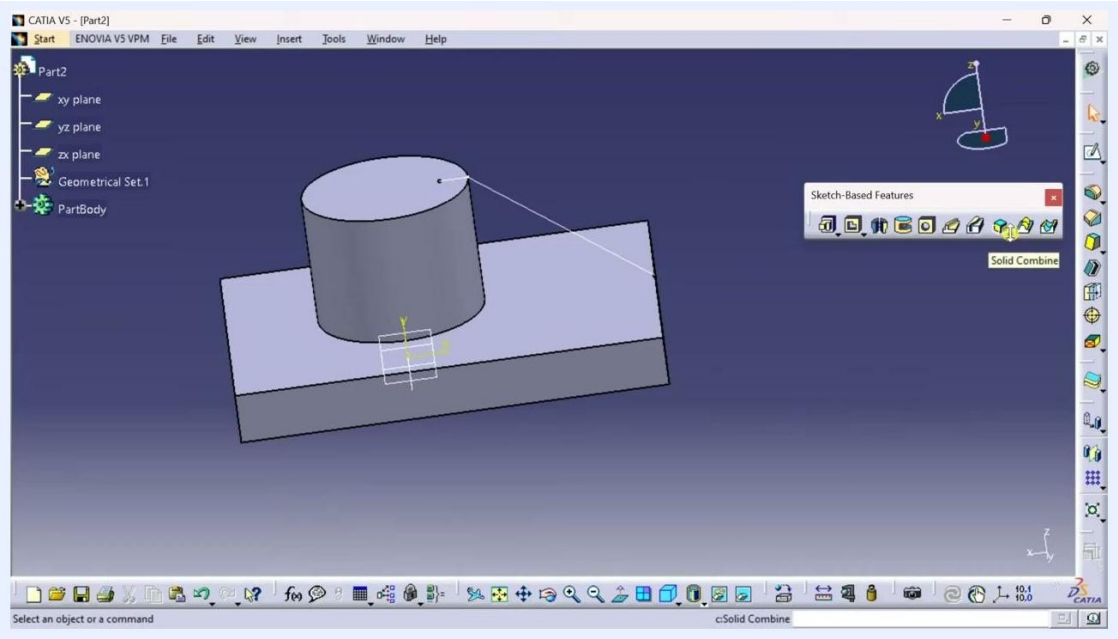
Click the **Construction/Standard Element** button in the top toolbar. The diagonal line turns green, which shows it is now a construction element. The model stays in the side view, with the construction line visible.



The diagonal line shown in green as a construction element, with the Construction/Standard Element button highlighted

Step 32: Activate the Stiffener command

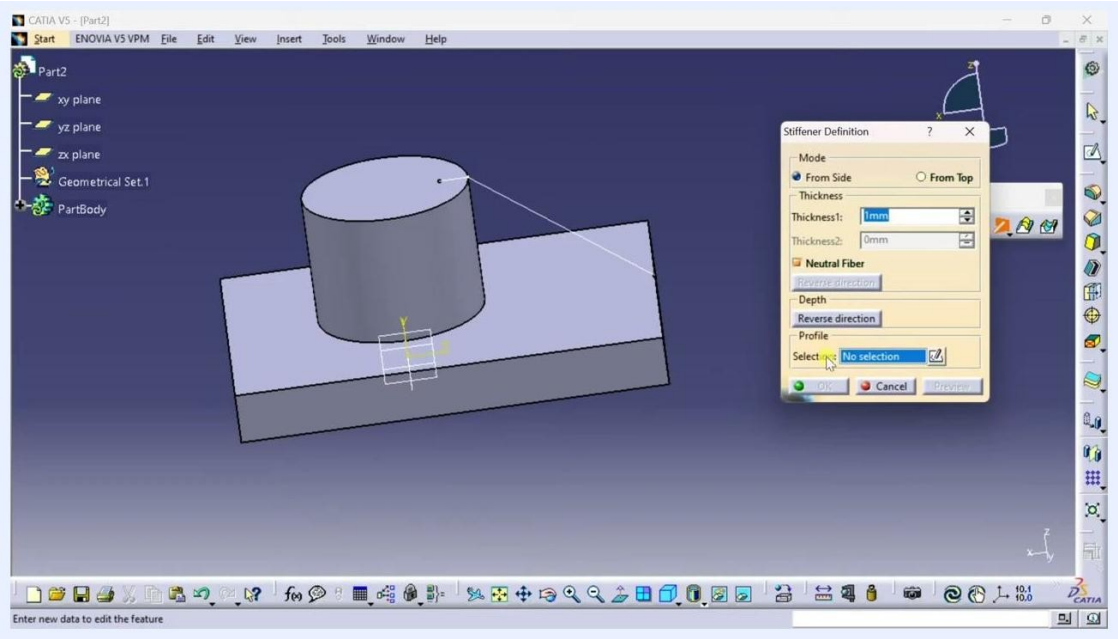
Exit the sketch so the model is shown in isometric view with the construction line visible. In the **Sketch-Based Features** toolbar, click the arrow next to **Solid Combine**, then click the **Stiffener** command. This is the second scenario for how to use stiffener in CATIA V5.



The isometric view with the cursor selecting the Solid Combine (Stiffener) command in the Sketch-Based Features toolbar

Step 33: Select the sketch line as the profile

The **Stiffener Definition** dialog appears on the right. By default, **Mode** is **From Side**, **Thickness1** is **1mm**, **Thickness2** is **0mm**, and **Neutral Fiber** is checked. The **Profile** section shows "No selection". Select the sketch line as the profile.



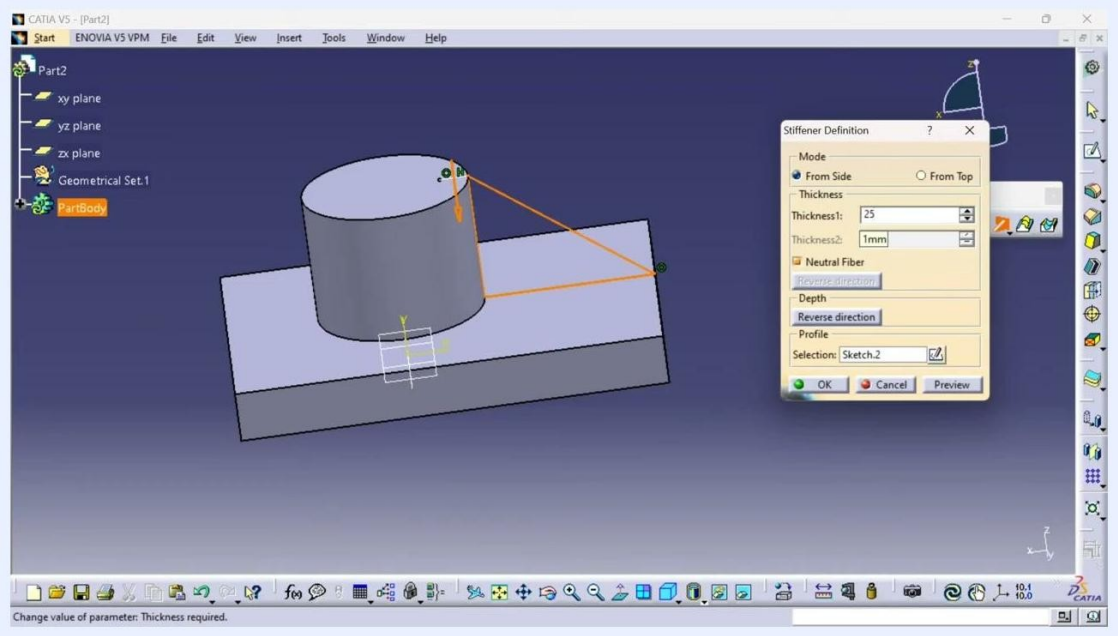
The Stiffener Definition dialog open with No selection in Profile, ready for you to select the sketch line

Step 34: Set the thickness values

With the profile selected, the model shows the cylinder on top of the rectangle and an orange construction line connecting them. An orange arrow indicates the direction of the stiffener. Confirm these settings in the dialog:

- **Mode:** From Side
- **Thickness1:** 25mm
- **Thickness2:** 0mm
- **Neutral Fiber:** Checked
- **Profile:** Sketch.2

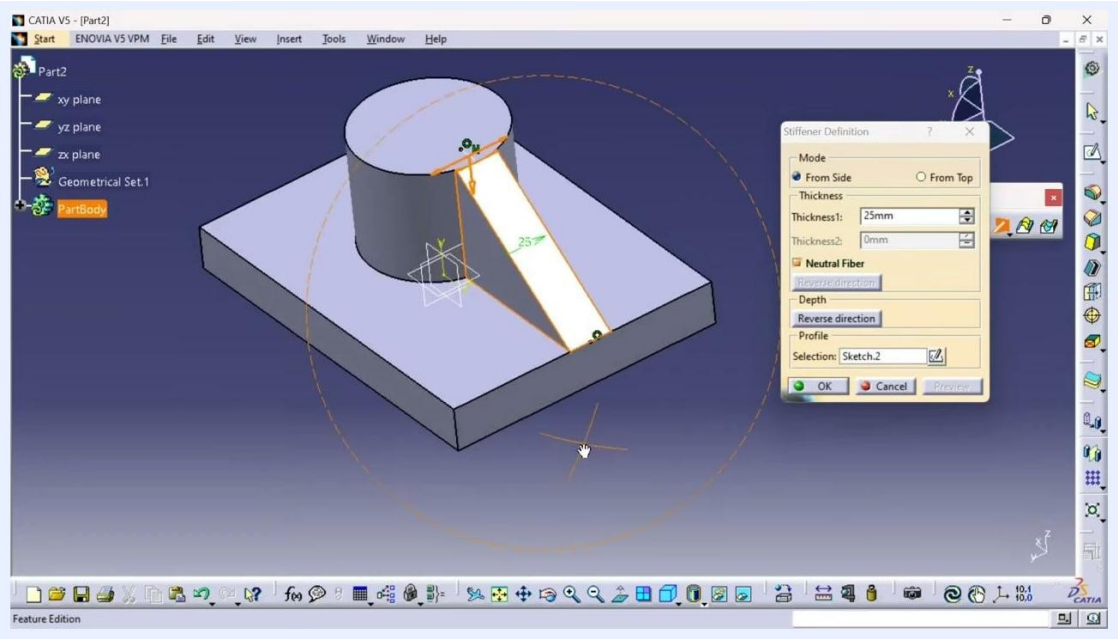
With **Neutral Fiber** checked, the thickness is distributed equally on both sides of the selected sketch line.



The Stiffener Definition dialog with From Side, Thickness1 at 25mm, Neutral Fiber checked, Profile Sketch.2, and the orange construction line and arrow visible

Step 35: Preview and confirm the stiffener

The model now displays the stiffener as a solid feature between the cylinder and the rectangle, following the construction line. The stiffener is highlighted in a light color, and the dimension "25" appears along it to show its thickness. The dialog stays open with your settings. Click **OK** to finish.



The completed stiffener between the cylinder and rectangle, with the Stiffener Definition dialog open and the thickness dimension visible

Summary

You now know how to use stiffener in CATIA V5 for both scenarios. For two flat surfaces, you create a pad, sketch and constrain a line, and apply the Stiffener command with a thickness and Neutral Fiber. For a flat and a round surface, you use Multi-Pad to extrude a circle and a rectangle, project 3D references, draw a construction line, and build the stiffener from it. In both cases, the Neutral Fiber option distributes the stiffener thickness equally on both sides of the profile.