How to use rib in CATIA V5: creating stiffeners in Part Design

This guide shows how to use rib in CATIA V5 with the **Stiffener** feature in the Part Design workbench. You sketch a profile, fully constrain it, and turn it into a stiffener between two plates. You then repeat the process on a cylinder-and-base part, using Multi-Pad and projected edges to build the reference geometry.

Stiffener types

CATIA stiffeners come in two types:

- **Type 1:** Both faces are flat.
- **Type 2:** One face is flat and one face is rounded.

You can use either type in Part Design.

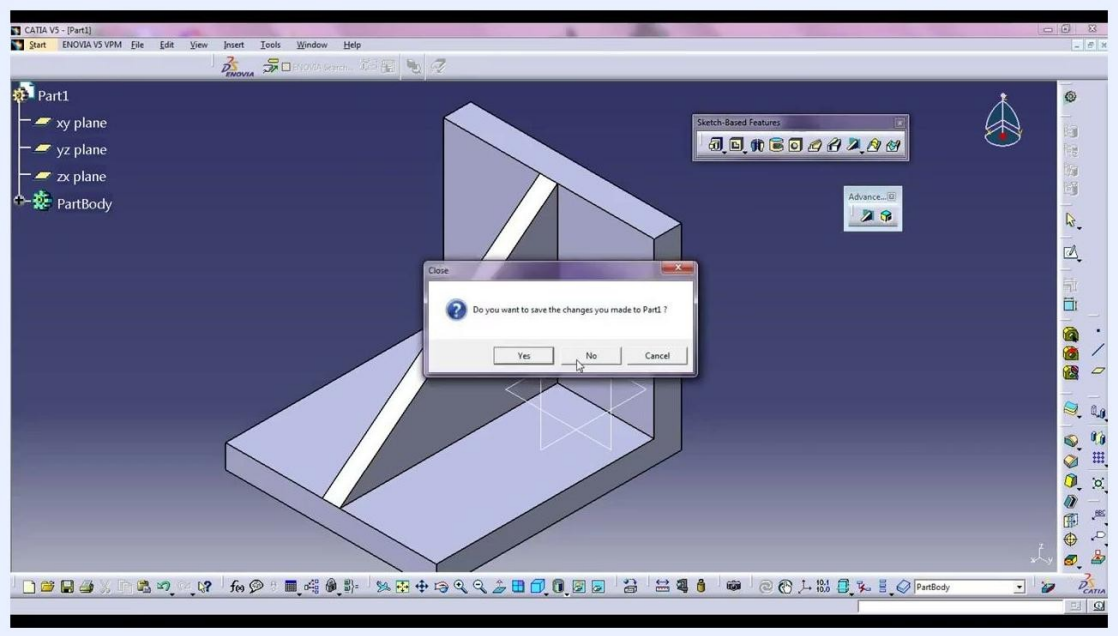
Prerequisites

- CATIA V5 with access to the Part Design workbench.
- A part with two plates to connect, or a base profile that you sketch yourself in the first steps.

Part 1: Create a stiffener between two plates

Step 1. Open the Part Design workbench

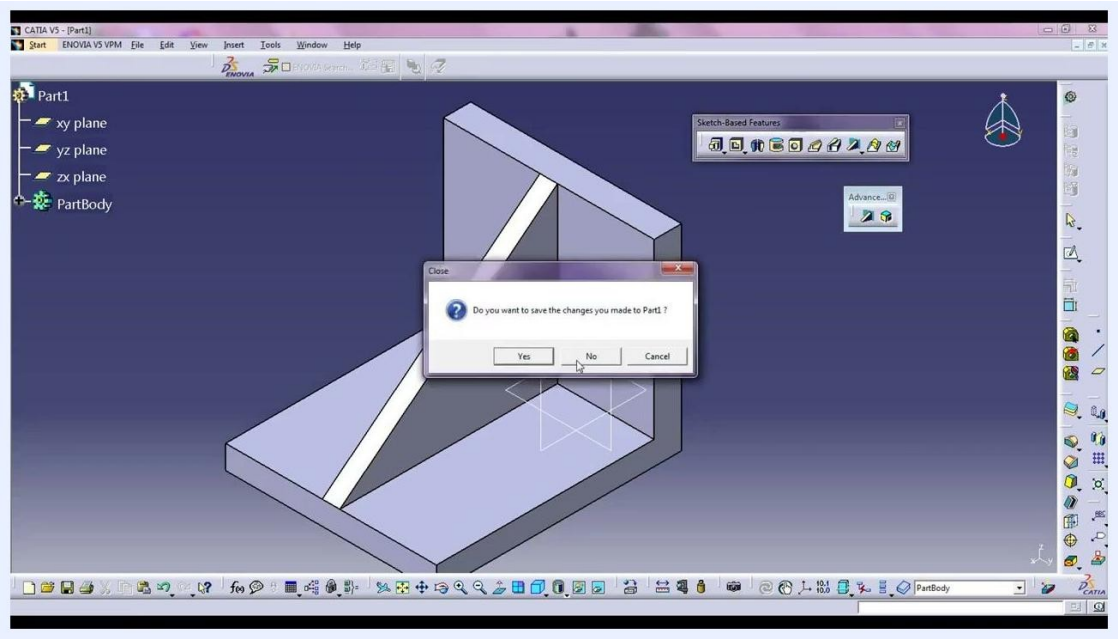
From the CATIA V5 main menu, choose **Start > Mechanical Design > Part Design**.



CATIA V5 main menu with Start > Mechanical Design > Part Design highlighted among the available workbenches

Step 2. Begin sketching the base profile

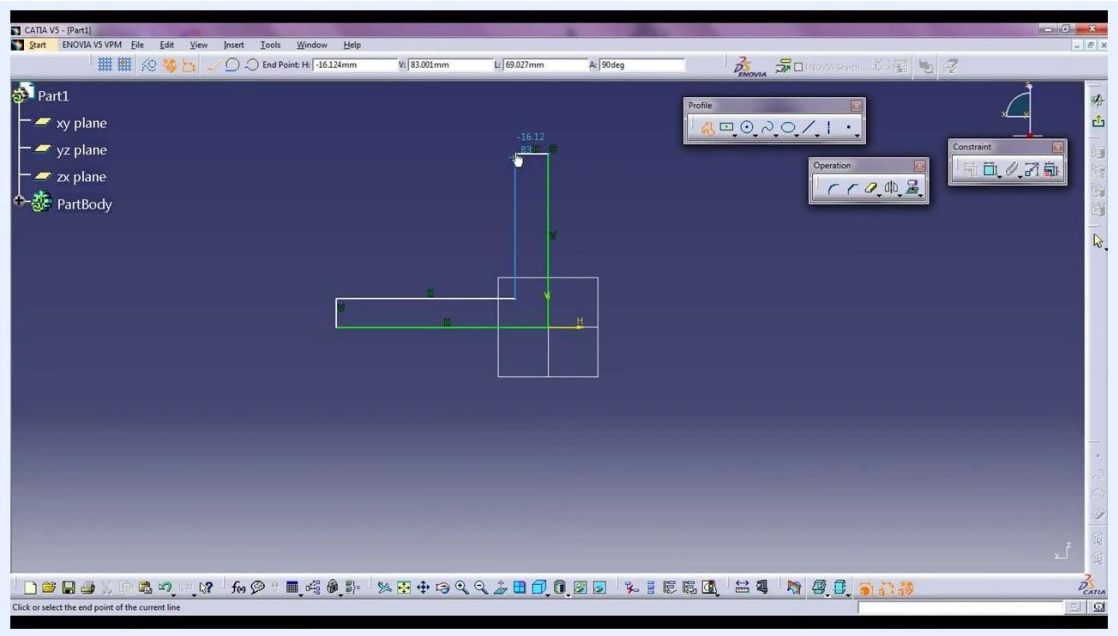
Start a new sketch in the Part Design workbench. The sketcher opens and shows the **Profile**, **Operation**, and **Constraint** toolbars. As you draw a vertical line, the endpoint coordinates appear (for example, V: -1.515mm, L: 86.216mm, A: 268.884deg).



Sketcher interface with a vertical line being drawn, endpoint coordinates displayed, and the toolbars visible

Step 3. Continue the profile without full constraints

Keep sketching the profile by adding horizontal and vertical lines. At this stage the sketch is not fully constrained, so the lines appear green but not every constraint is applied. The endpoint coordinates for the current line might read V: -16.12mm, L: 69.027mm, A: 90deg.



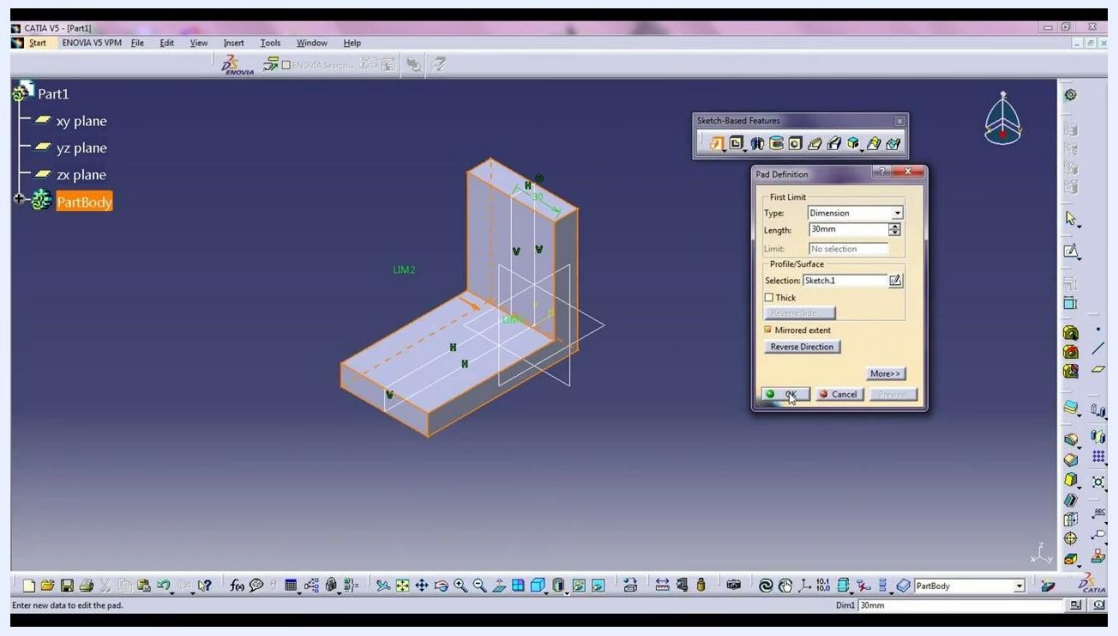
Sketcher interface with a partially constrained profile, green lines, and endpoint coordinates displayed

Step 4. Set the pad dimensions

Exit the sketch and create the pad that forms the plates. In the **Pad Definition** dialog:

- Set the **First Limit** type to **Dimension**.
- Enter the required length (for example, 20mm or 30mm).
- Select the sketch for the pad profile (for example, Sketch1).
- Use the **Mirrored extent** option if your design requires symmetry about a plane.

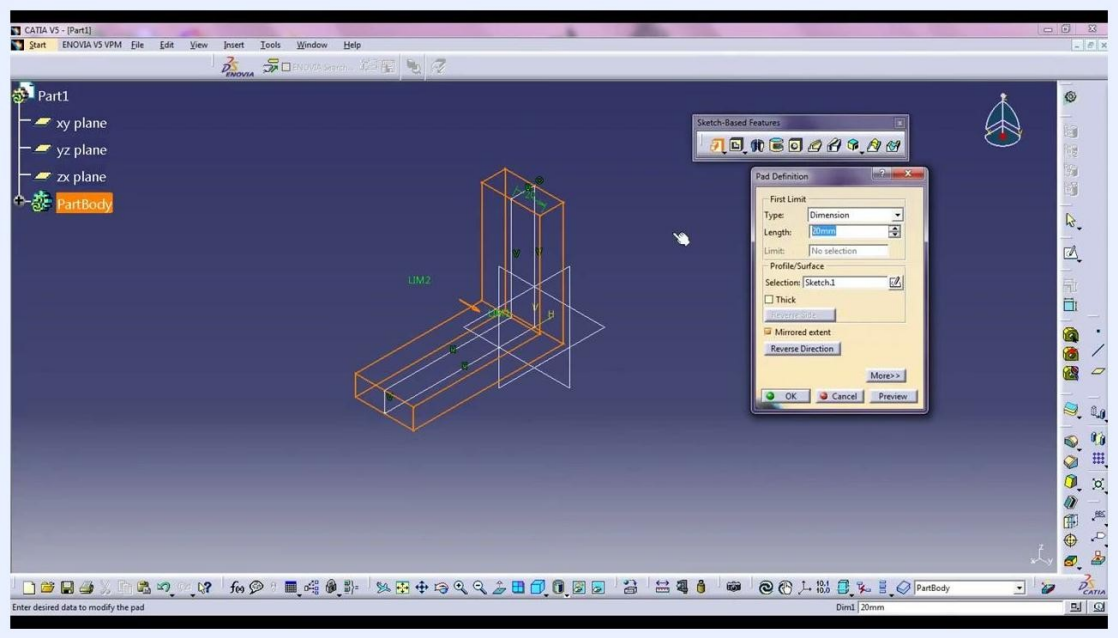
- Click **OK** to create the pad.



Pad Definition dialog with First Limit set to Dimension, Length set to 30mm, and Sketch1 selected; the model shows two perpendicular pads

Step 5. Select the middle plane for the stiffener sketch

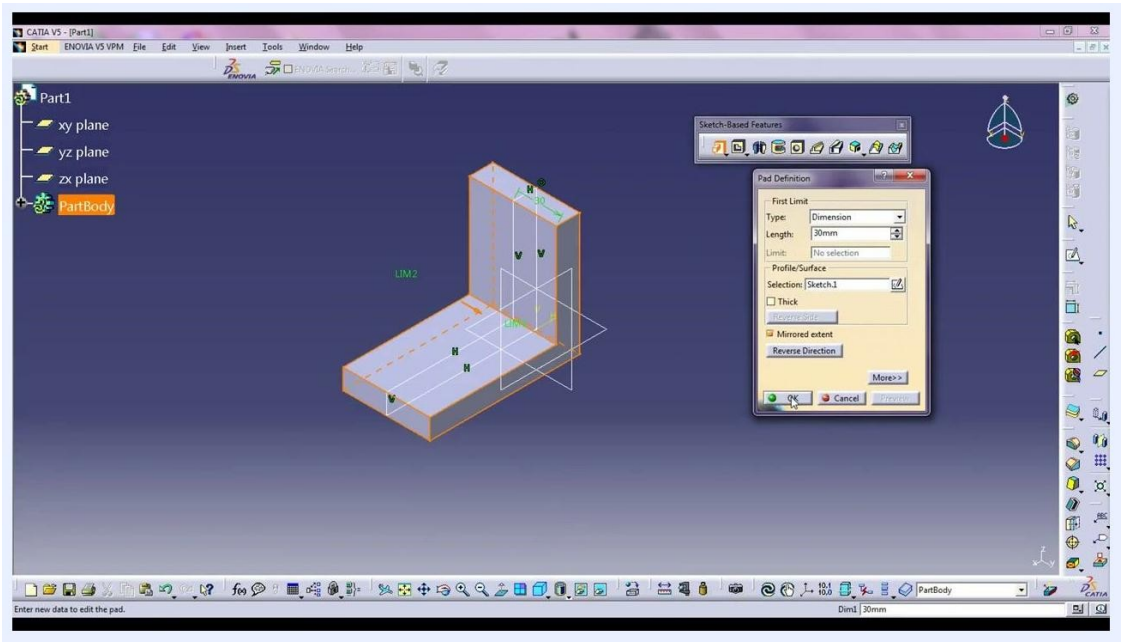
In the tree, select the middle plane where the stiffener will be sketched (for example, the zx plane). Then click the **Sketch** icon to enter the Sketcher workbench.



CATIA V5 interface with the zx plane selected, ready for a new stiffener sketch

Step 6. Draw the stiffener sketch

In the Sketcher, use the **Profile** toolbar to draw a line that connects the two plates where the stiffener will sit. The endpoint coordinates are displayed for precision (for example, End Point H: 109.625mm, V: 16.755mm, A: 220.372deg).

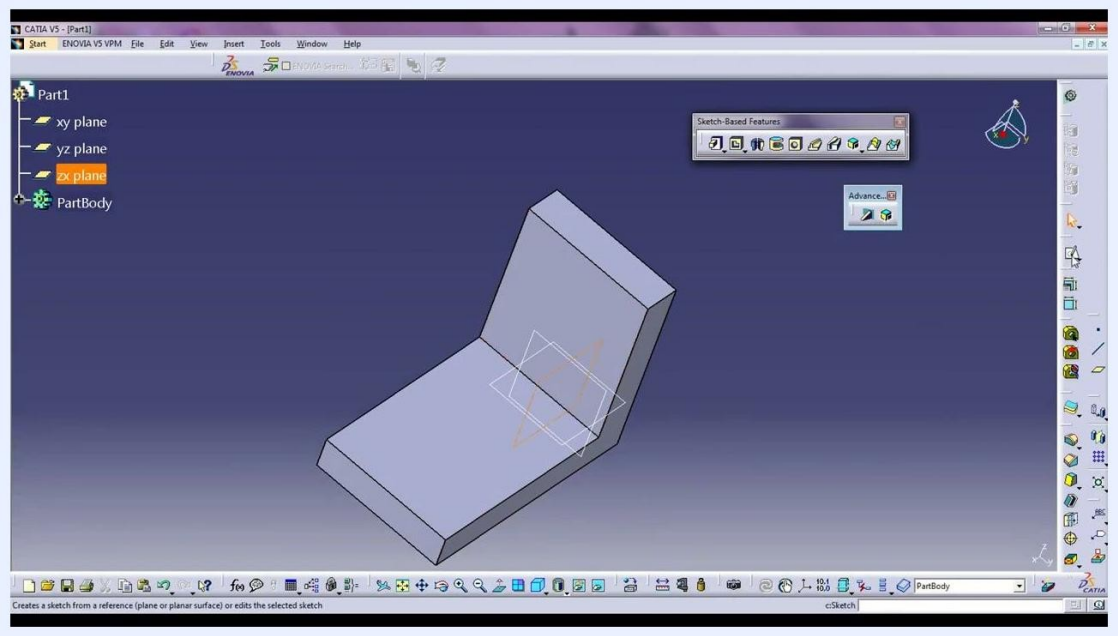


Sketcher interface with a diagonal line being drawn between two plates and endpoint coordinates visible

Step 7. Fully constrain the sketch

Apply geometric and dimensional constraints with the **Constraint** toolbar. Add coincidence, length, and angle constraints as needed. For example, right-click or use the context menu to set a **Coincidence** constraint between the line and the plate edges.

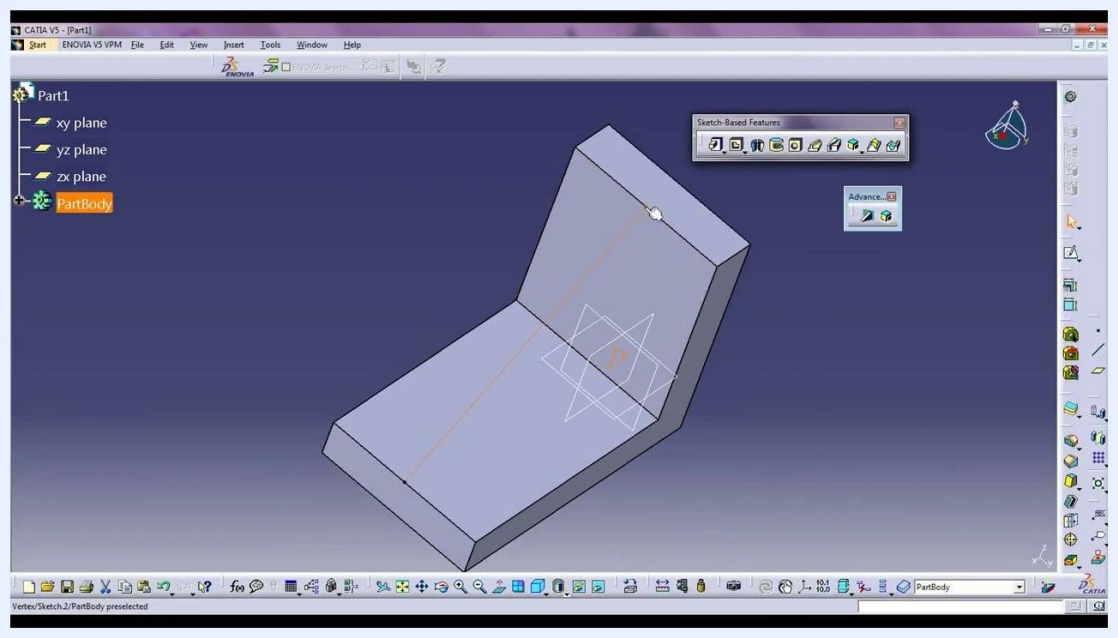
You must fully constrain the sketch. Otherwise, the stiffener feature may not work correctly.



Sketcher interface showing a coincidence constraint being applied to the stiffener sketch

Step 8. Check that the sketch is fully constrained

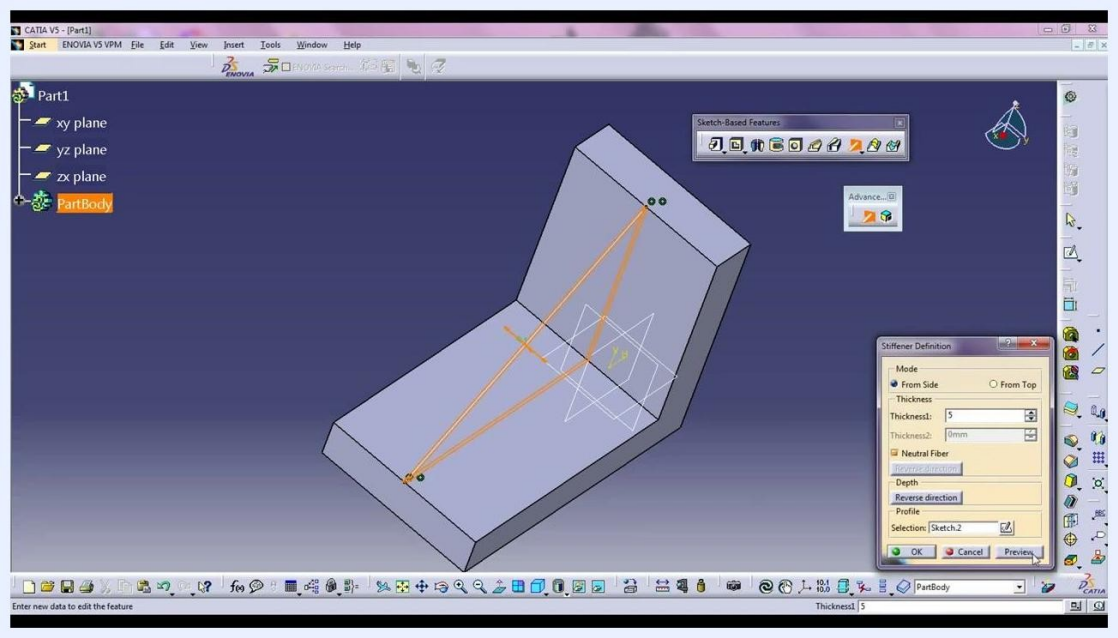
Confirm that all lines in the sketch turn green. This indicates that the sketch is locked in position and dimension.



Sketcher interface with a fully constrained diagonal stiffener sketch and all lines green

Step 9. Exit the Sketcher workbench

Click the **Exit workbench** icon to return to the Part Design workbench. The stiffener sketch is now visible on the model.



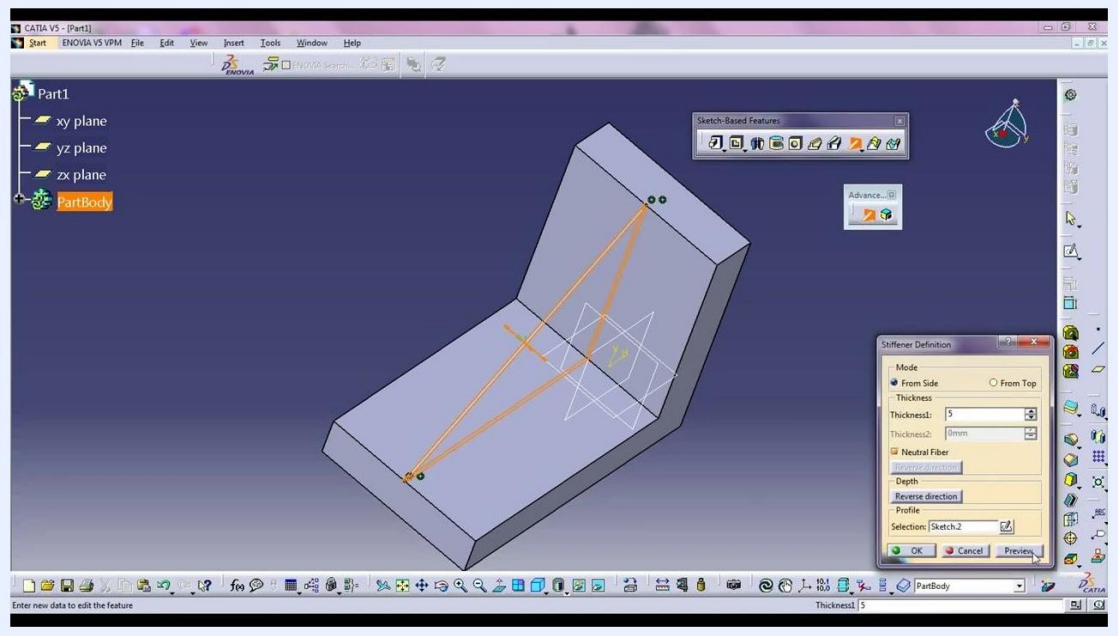
Model with the stiffener sketch visible between two plates, ready for stiffener creation

Step 10. Create the stiffener feature to use a rib in CATIA

Select the stiffener sketch, then click the **Stiffener** icon in the Sketch-Based Features toolbar. In the **Stiffener Definition** dialog:

- Set **Mode** to **From Side**.
- Enter **Thickness1** as 5mm, or your required value.
- Check or uncheck **Neutral Fiber**, depending on whether you want the thickness centered on the sketch or offset to one side.
- Use **Reverse direction** if the stiffener appears on the wrong side.

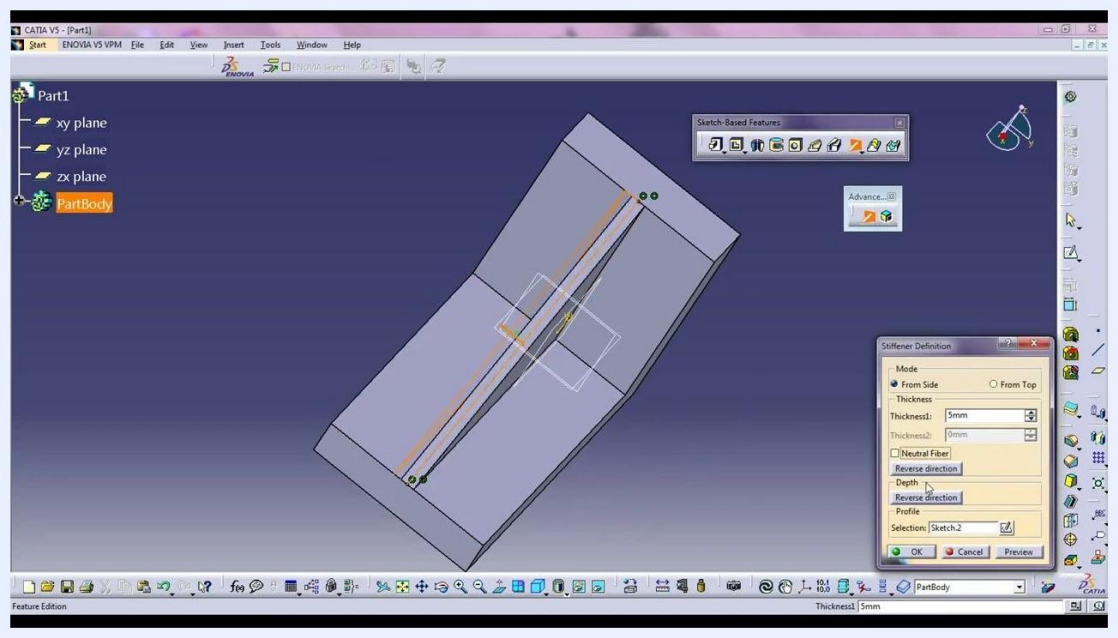
- Click **Preview** to visualize the stiffener.



Stiffener Definition dialog with Thickness1 set to 5mm, the Neutral Fiber option visible, and the stiffener previewed between the plates

Step 11. Adjust the direction and confirm

If the stiffener appears on only one side, click **Reverse direction** to switch sides. When the placement is correct, click **OK** to finalize the feature.

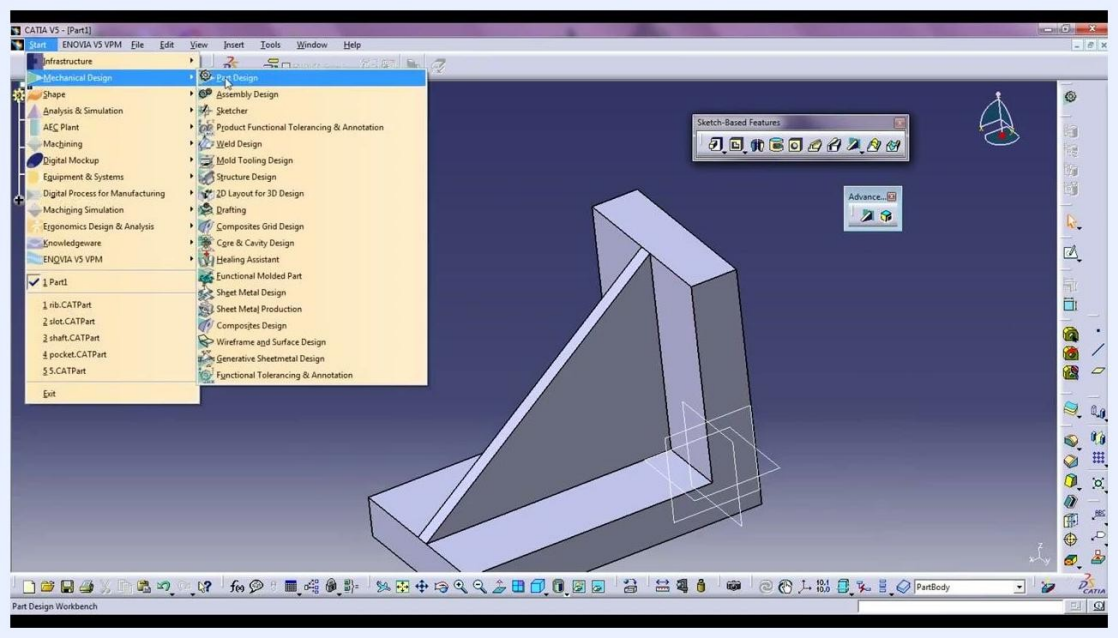


Model showing the stiffener applied between two plates, with the Stiffener Definition dialog open and direction options visible

Part 2: Build a stiffener on a cylinder and base with Multi-Pad and projected edges

Step 12. Remove an unwanted sketch element

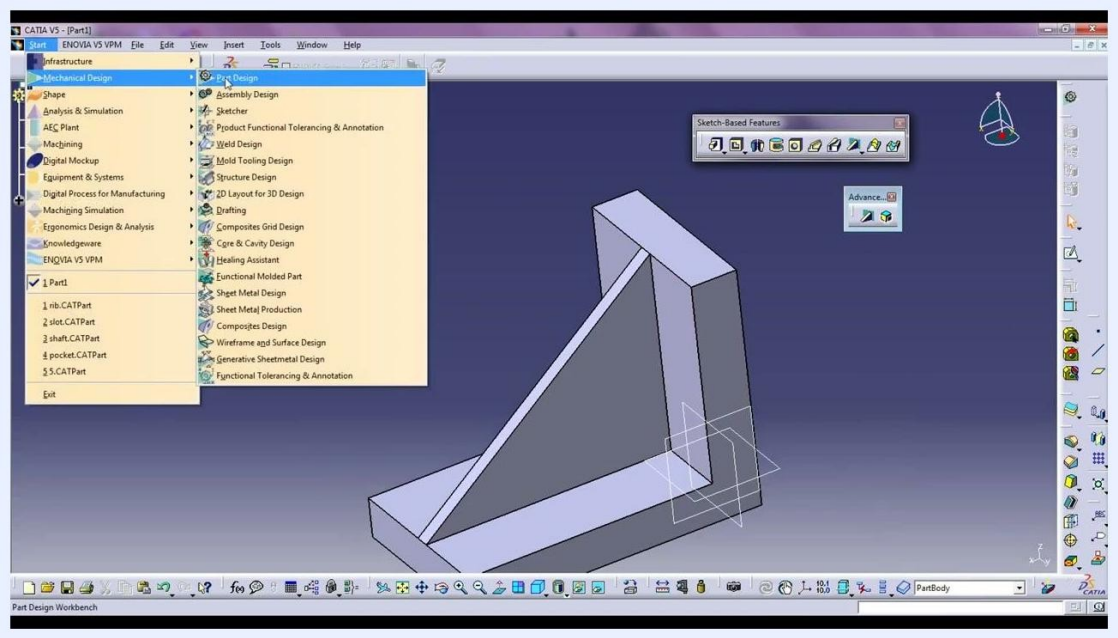
Select the top plane (for example, the xy plane) in the specification tree or on the model. Click the **Sketch** icon to enter the Sketcher workbench. Select the geometry you want to remove, such as the center circle, and delete it.



Sketcher interface showing a rectangle with a centered coordinate system, ready for modification

Step 13. Sketch on the center plane

Make sure you are sketching on the center plane (for example, the zx plane) for proper alignment and reference. Draw or modify the geometry you need, such as a circle at the center of the rectangle.



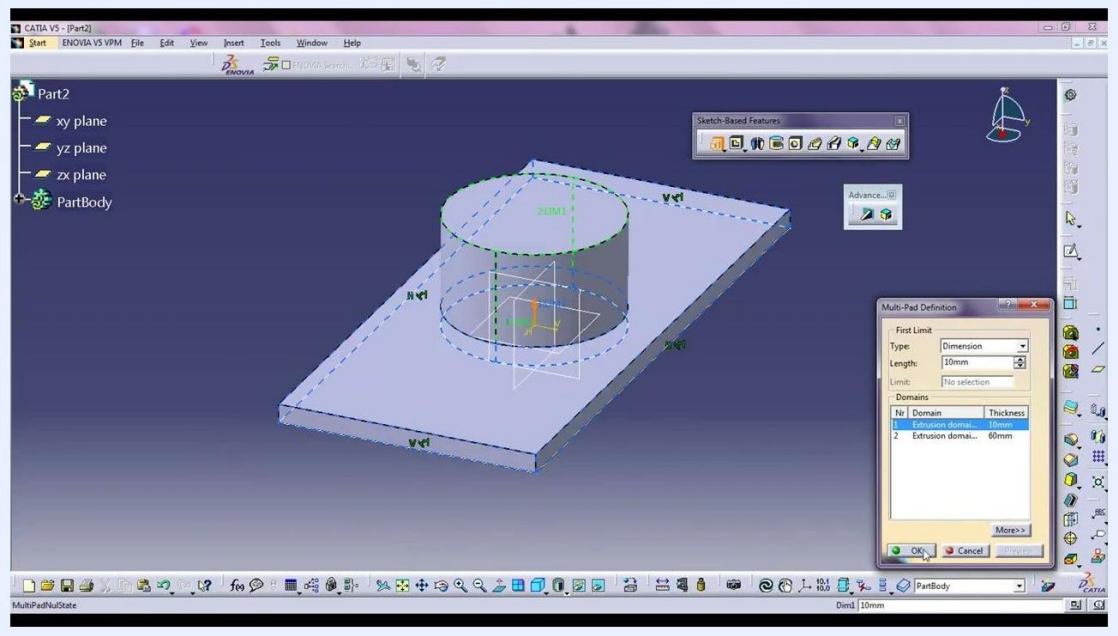
Sketcher interface with a rectangle and a centered circle, using the center plane as the reference

Step 14. Apply the Multi-Pad feature

Exit the Sketcher workbench to return to Part Design. Select the sketch that contains multiple domains (for example, a rectangle and a circle), then click the **Multi-Pad** icon in the Sketch-Based Features toolbar. In the **Multi-Pad Definition** dialog:

- Set the **First Limit** type to **Dimension**.
- Enter Length = 60mm for one domain and Length = 10mm for the other.
- Assign each thickness value to the appropriate domain (the rectangle or the circle).

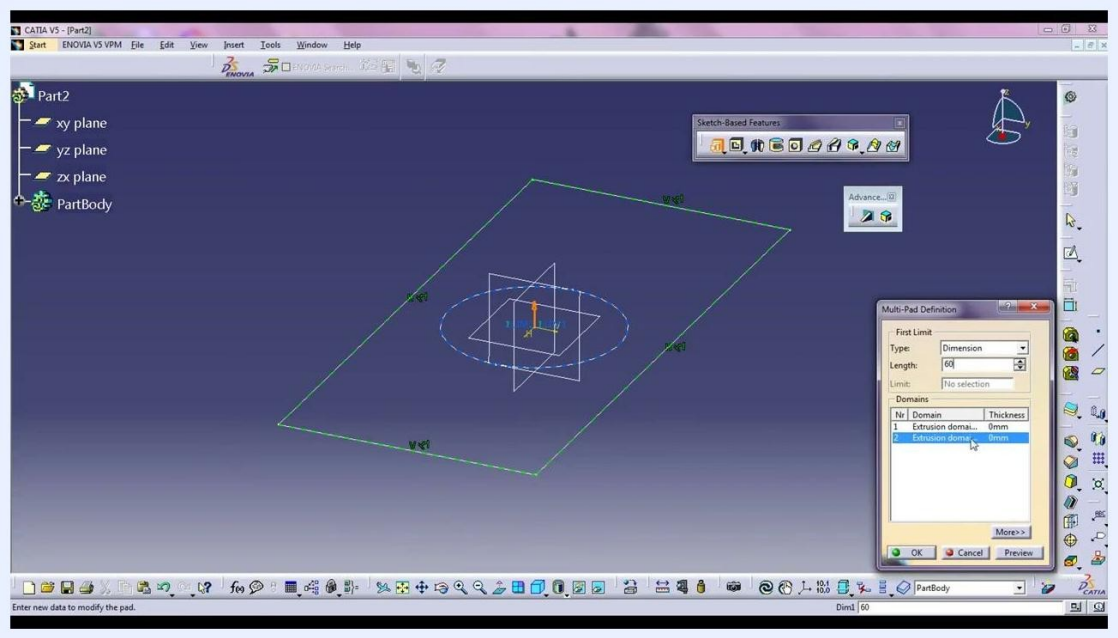
- Click **OK** to create the feature.



Multi-Pad Definition dialog with Length set to 10mm and 60mm for different domains, and the 3D model showing a cylinder on a rectangular base

Step 15. Select the middle plane for the next sketch

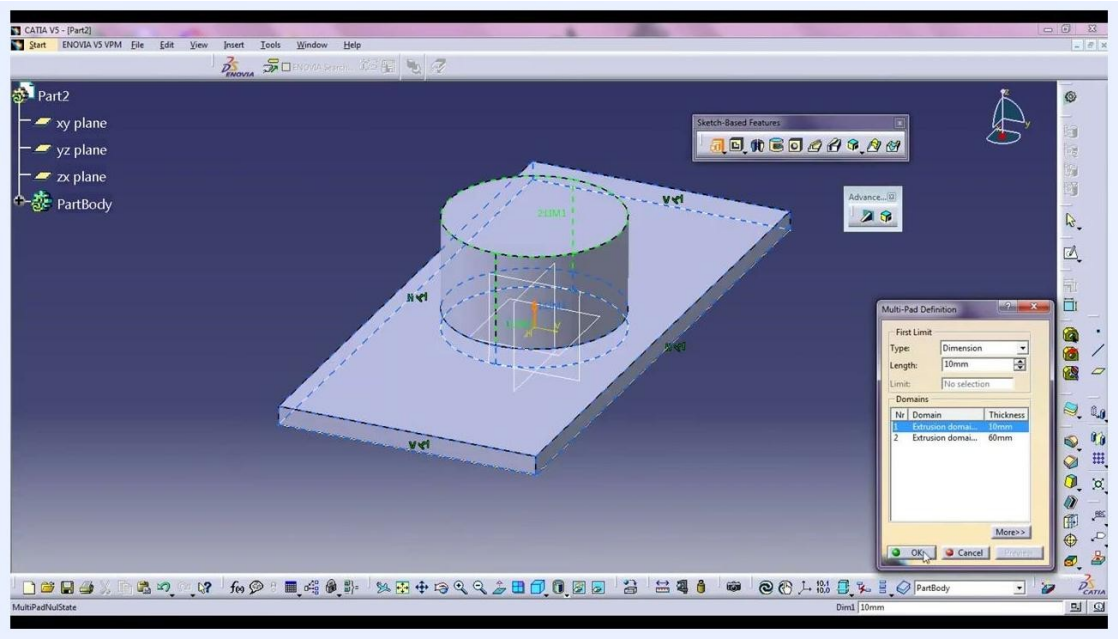
In the specification tree, select the middle plane (for example, the zx plane). Click the **Sketch** icon to create a sketch on this plane. The model displays the new multi-pad feature.



Model with a cylinder on a rectangular base and the zx plane selected for a new sketch

Step 16. Project 3D silhouette edges

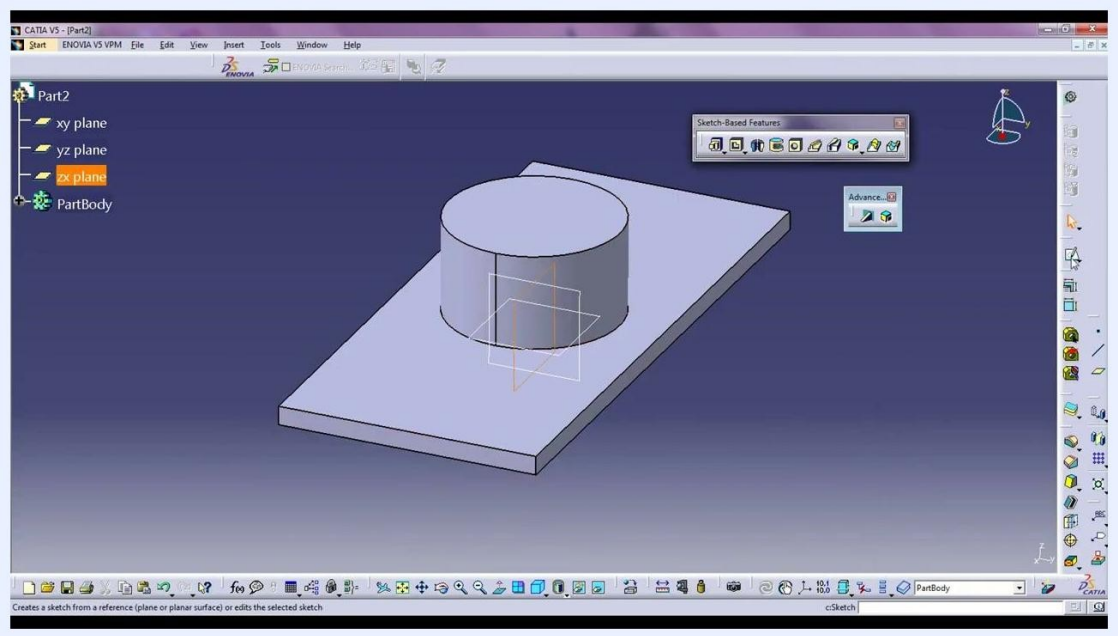
In the Sketcher workbench on the selected plane, use the **Project 3D Silhouette Edges** tool from the **Operation** toolbar. It projects the edges of the 3D geometry onto the sketch plane, where they serve as references for further sketching.



Sketcher interface with the Project 3D Silhouette Edges tool highlighted, showing a side view of the cylinder and base

Step 17. Draw reference lines from the projected edges

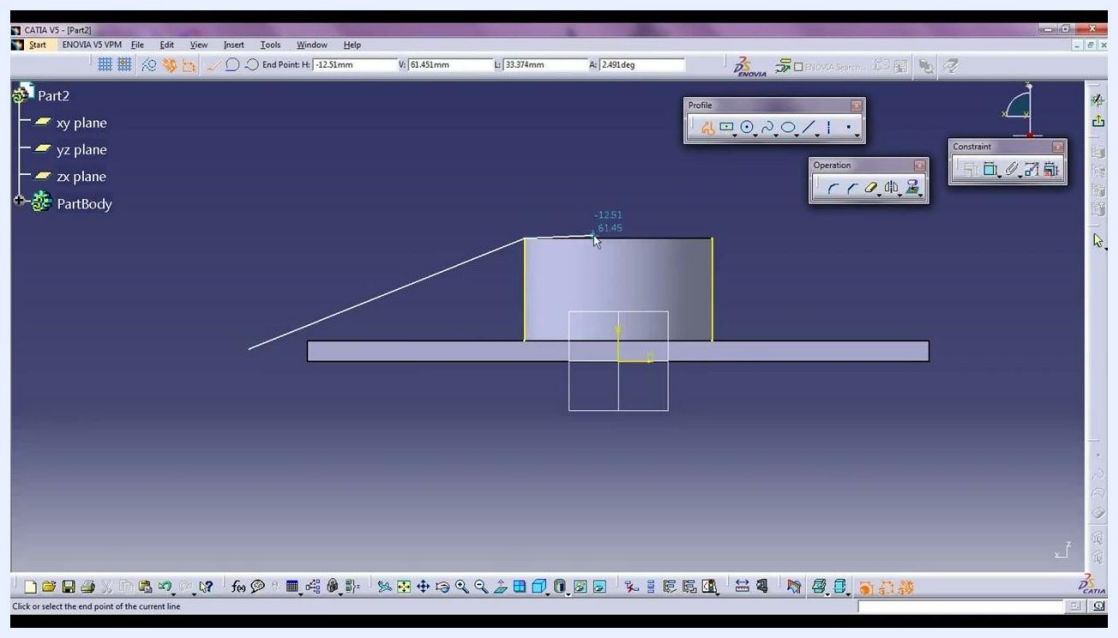
Click the projected edge or point where you want the reference line to start. Draw a line to the location you need (for example, from the top of the cylinder to the front). Adjust the line to match your design intent.



Sketcher interface with a line being drawn from the top of the cylinder to the front, using projected edges as reference

Step 18. Complete the reference geometry

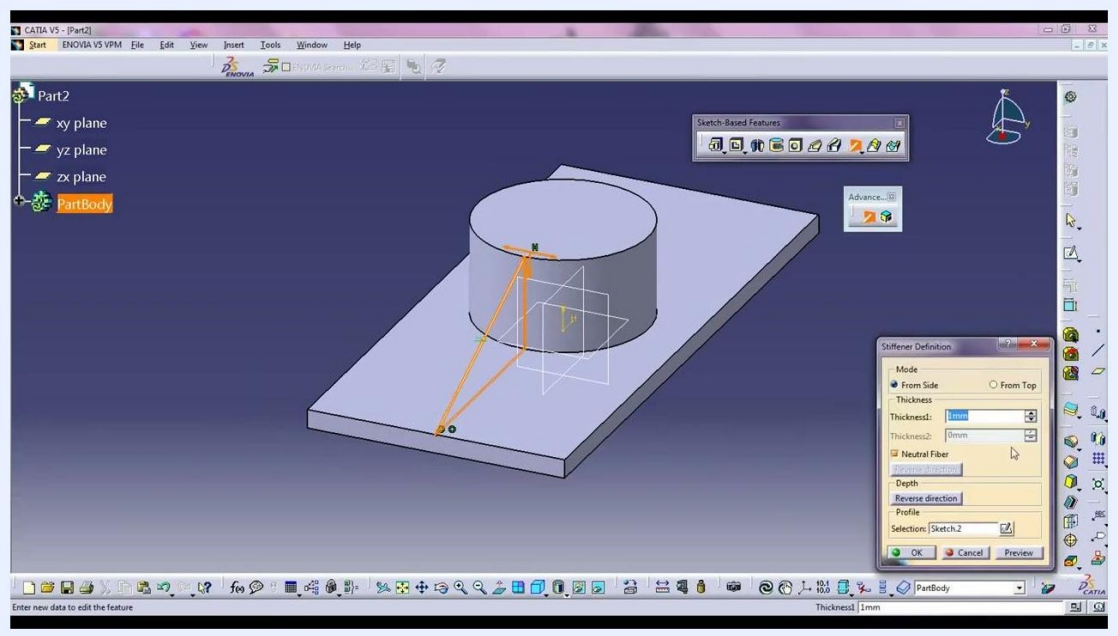
Continue clicking points and moving them as needed. Position the lines according to your design requirements.



Sketcher interface with additional reference lines being drawn and adjusted, showing endpoint coordinates

Step 19. Fully constrain the sketch

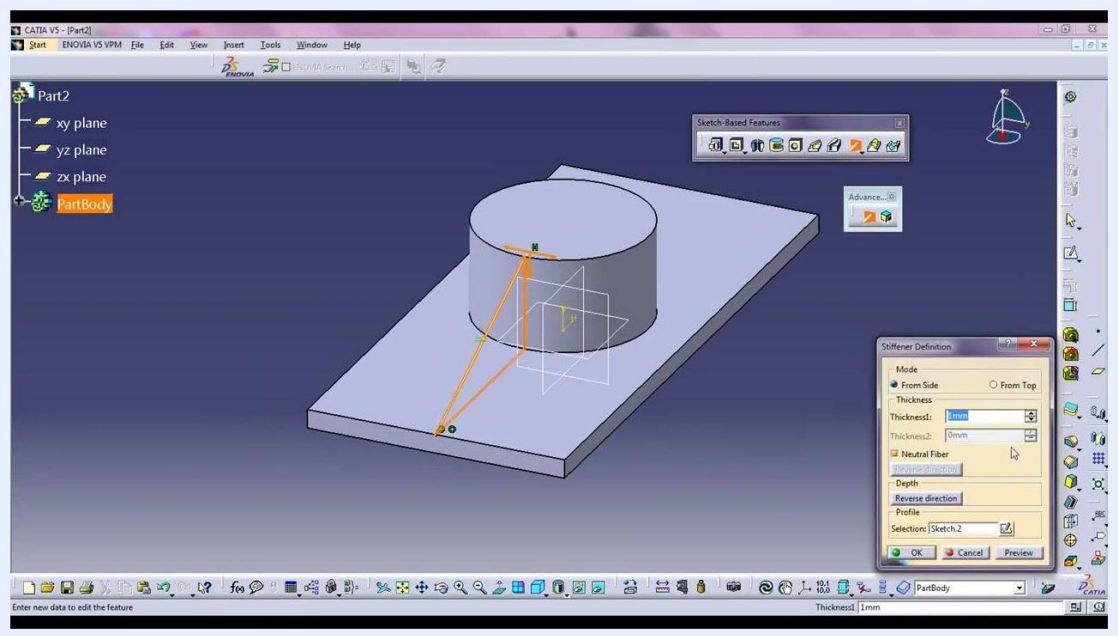
Select the relevant point and line in your sketch and apply the necessary constraints. Confirm that the sketch is fully constrained. All geometry turns green, or the constraint status indicates full constraint.



Sketcher interface showing a side view of a cylinder on a rectangular base, with a slanted reference line and constraint tools visible

Step 20. Convert the shell out edges to construction geometry

Select both shell out edges in your sketch. Change their status to construction geometry with the construction tool in the toolbar. These edges are then used only for reference and do not create solid features.



Sketcher interface with a selected point and line, and PartBody highlighted in the specification tree

Step 21. Exit the Sketcher and open the stiffener tool

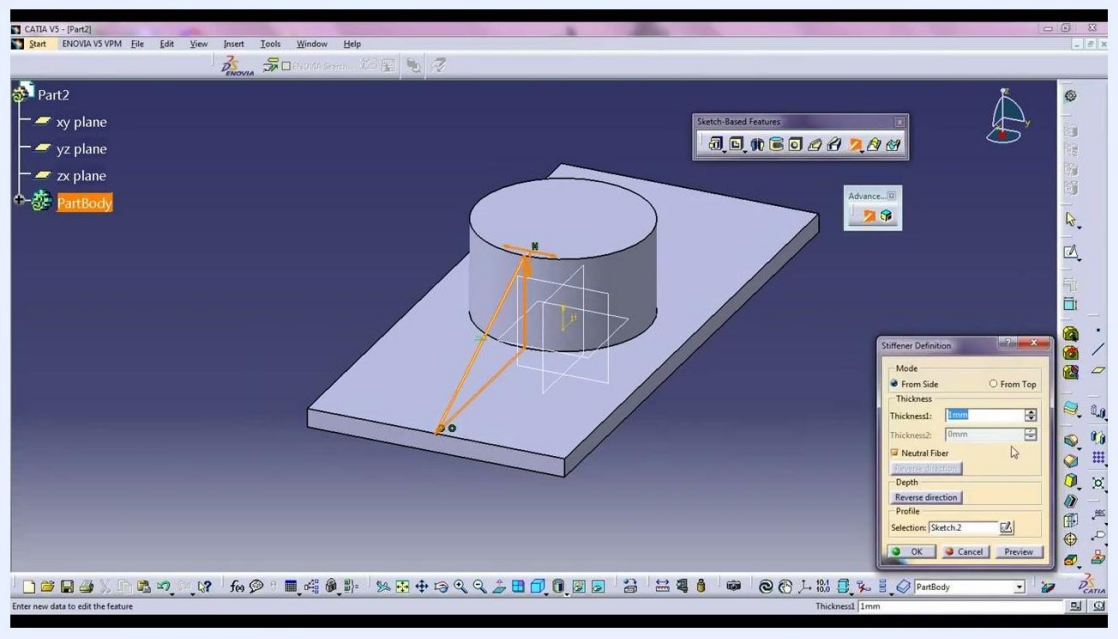
Click **Exit Workbench** to leave the Sketcher and return to Part Design. Then select the **Stiffener** tool from the Sketch-Based Features toolbar. The **Stiffener Definition** dialog appears.

Step 22. Set the thickness to use a rib in CATIA on the cylinder and base

In the **Stiffener Definition** dialog, set the following parameters:

- **Mode:** Select **From Side**.
- **Thickness1:** Enter the thickness you want (for example, `1mm` initially).
- **Thickness2:** Leave it at `0mm` unless you need a second thickness.
- **Neutral Fiber:** Check or uncheck it as your design requires.
- **Profile:** Make sure the correct sketch (for example, `Sketch.2`) is selected.

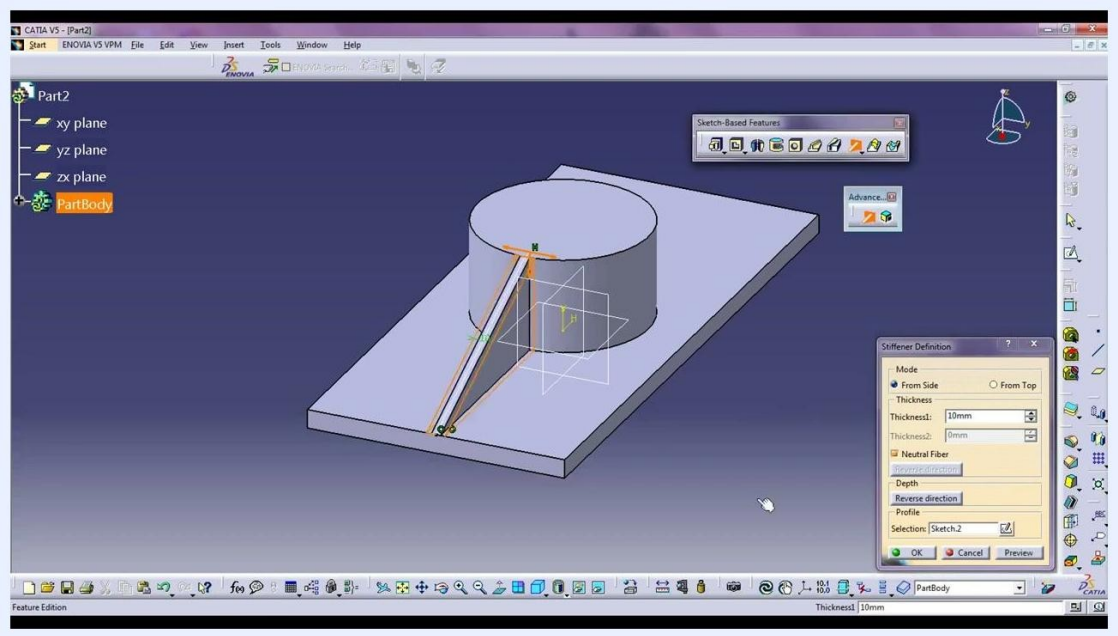
Click **Preview** to visualize the stiffener. When you are satisfied with the thickness and placement, click **OK**. The stiffener now appears as a solid feature connecting the cylinder and the base.



Part Design workbench with the Stiffener Definition dialog open, showing a stiffener profile between the cylinder and the base with Thickness1 set to 1mm

Step 23. Review the final model

Rotate and inspect the model to make sure the stiffener is correctly placed and fully integrated with the cylinder and base. Save your work to preserve the completed part.



Isometric view of the completed part with a cylinder, base, and stiffener

Summary

To use a rib in CATIA V5 successfully, follow these practices:

- Fully constrain your sketches before using them to create features.
- Use **Neutral Fiber** and **Reverse direction** to control stiffener placement.
- Preview the stiffener before you confirm it.
- Sketch on the correct reference plane to maintain alignment.
- Use **Multi-Pad** to extrude multiple domains with different thicknesses in one operation.
- Project 3D edges into your sketch for accurate references.
- Use construction geometry for reference edges that should not create solid features.

What's next

Try both stiffener types (flat-flat and flat-rounded) on your own parts. Adjust the stiffener thickness to suit your design requirements.