How to convert a surface to solid in CATIA V5

This guide shows you how to convert a surface to solid in CATIA V5. You build a multi-section surface from two spline curves, then apply thickness to turn that surface into a solid part. You work in the Generative Shape Design workbench from start to finish.

Prerequisites

- CATIA V5 is installed and open.
- A product is loaded in the main workspace. The window title shows the product name, for example [Product1.2].
- The workspace is empty except for the default axes and the product tree on the left.

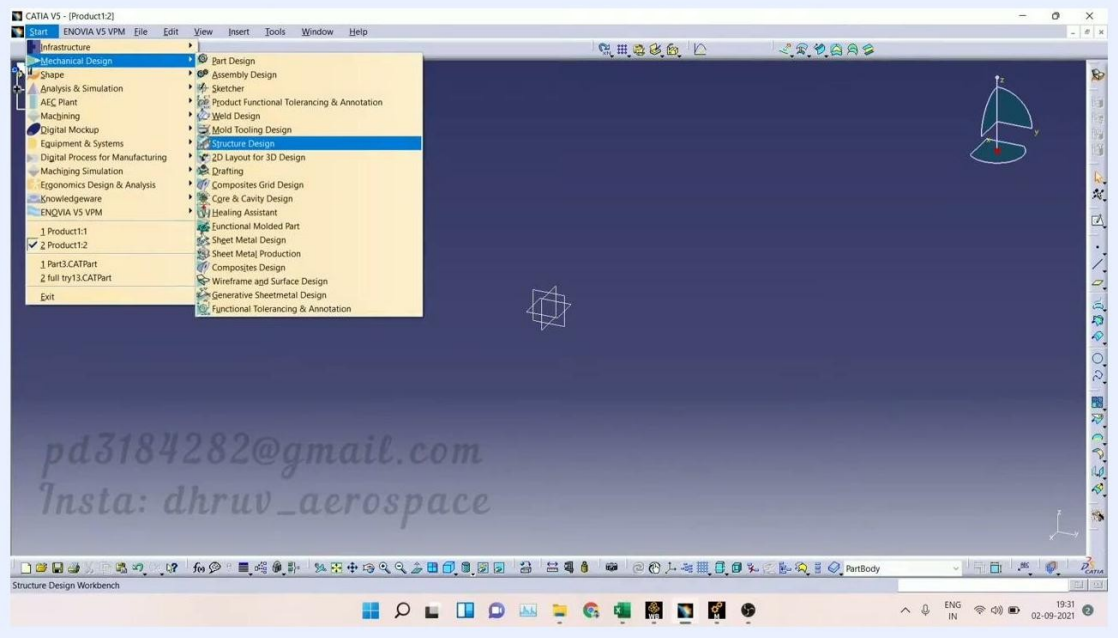
Step 1: Launch CATIA V5 and confirm the product is loaded

1. Open CATIA V5.
2. Check that the main workspace shows the loaded product.

Step 2: Switch to the Generative Shape Design workbench

1. Click the **Start** menu.
2. Hover over **Shape** to expand its submenu.

3. Select **Generative Shape Design**.

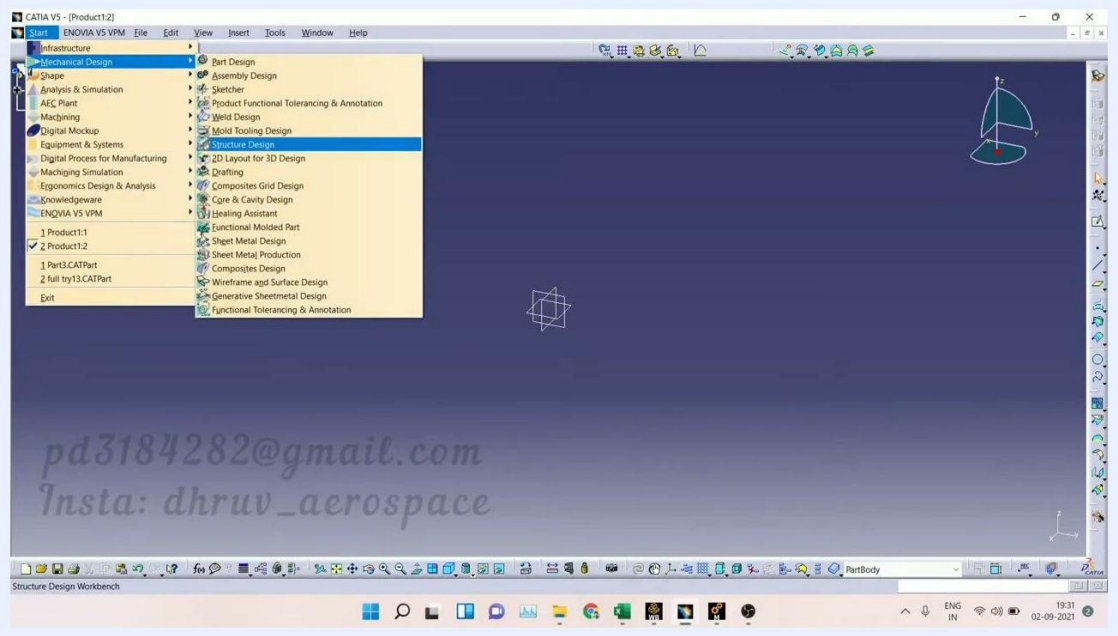


The CATIA V5 Start menu open with Shape > Generative Shape Design highlighted, an empty workspace, and the product tree visible

Step 3: Draw the first spline curve

1. In the Generative Shape Design workbench, select the spline tool.
2. Click to place control points and shape the spline as needed.

The curve appears in the workspace with its control points and coordinate axes. The product tree shows the part structure, for example Part2, xy plane, yz plane, zx plane, and PartBody.

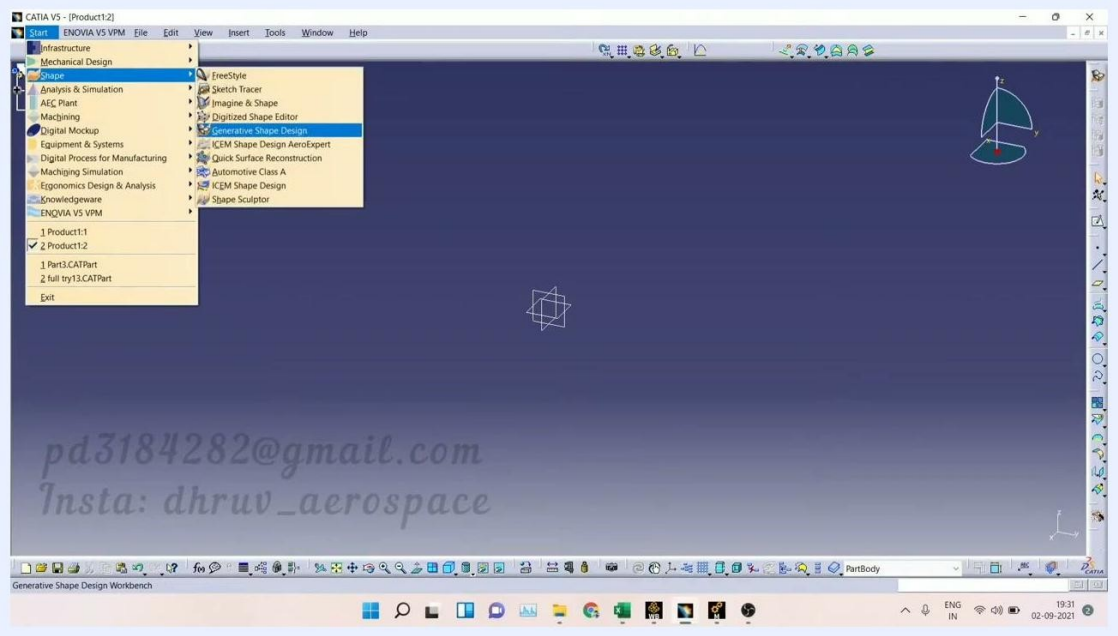


A spline curve drawn in the workspace with control points visible, the product tree on the left, and a grid background

Step 4: Adjust the first spline curve

1. Move the control points to refine the curve shape.

The curve updates dynamically as you adjust the points. PartBody may be highlighted in the product tree as the active part.

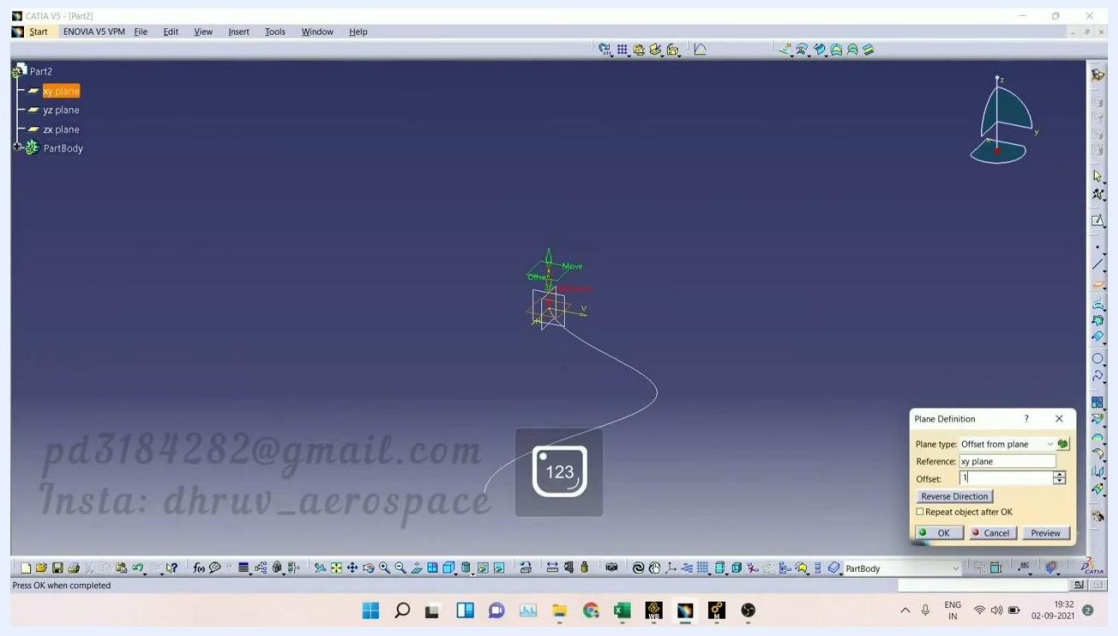


The adjusted spline curve with moved control points and PartBody highlighted in the product tree

Step 5: Create an offset plane

1. Select the xy plane in the product tree.
2. Open the **Plane Definition** dialog.
3. Set **Plane type** to **Offset from plane**.
4. Set **Reference** to xy plane.
5. Enter an **Offset** value, for example 1.

6. Click **OK** to create the new plane.

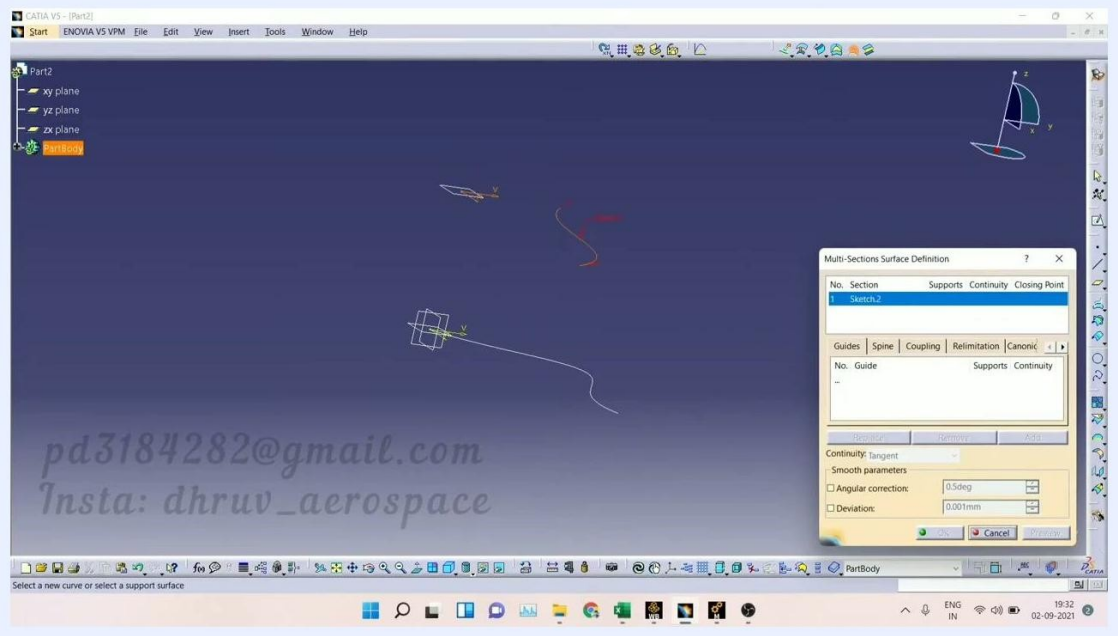


The Plane Definition dialog with Offset from plane selected, xy plane as the reference, and an offset value entered, above the workspace with the spline and axes

Step 6: Draw a second spline curve on the new plane

1. Switch to the new offset plane.
2. Use the spline tool to draw a second curve, similar to the first.
3. Place control points and shape the curve as needed.

The workspace now displays two spline curves on different planes.

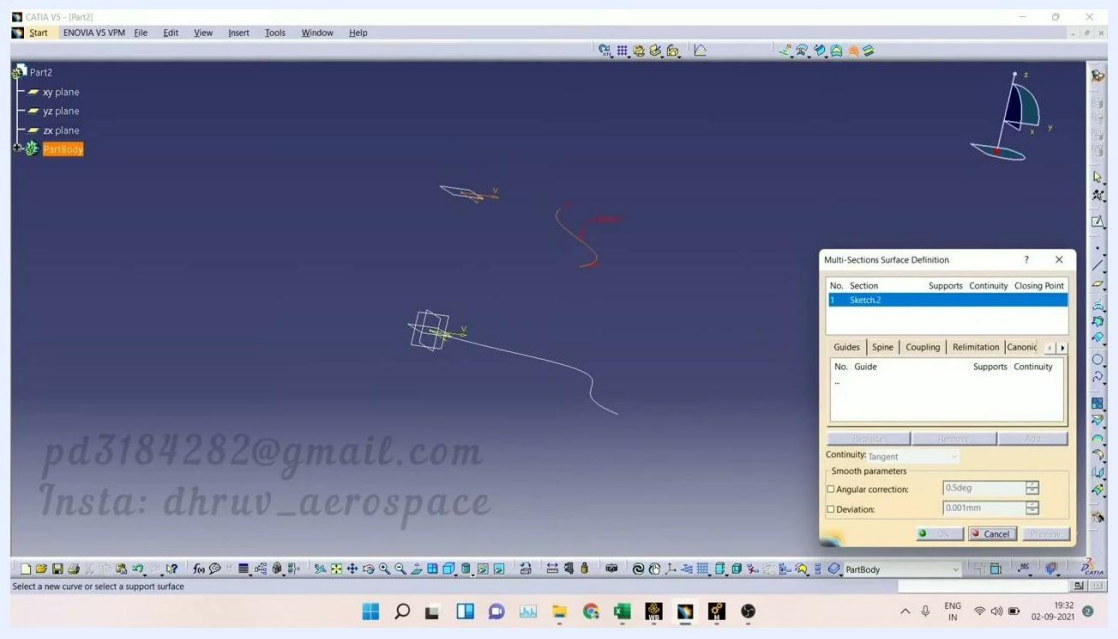


The workspace with two spline curves on different planes, control points visible, and the product tree on the left

Step 7: Refine the second spline curve

1. Adjust the control points of the second spline until you reach the shape you want.

The workspace shows both curves, each with its own control points and coordinate axes. Control point values, for example H: 8.079mm and V: 74.44mm, may appear near the points.



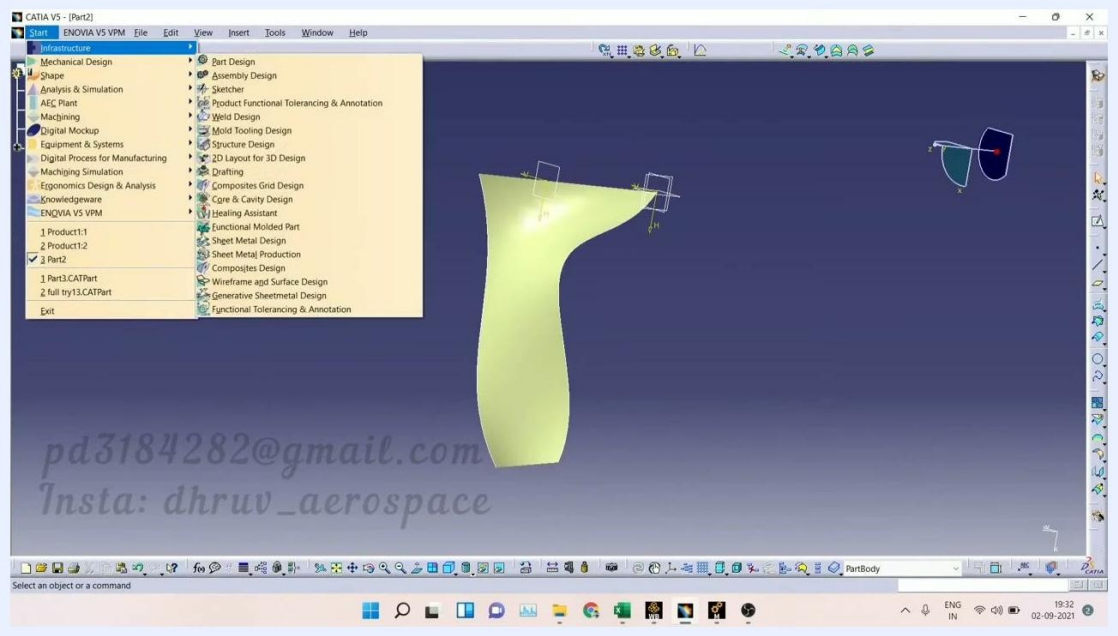
The workspace with two refined spline curves, control points and their values visible, and the product tree on the left

Step 8: Open the Multi-Sections Surface Definition dialog and add the first section

With both splines created, you can build the surface that you later convert to a solid.

1. Open the **Multi-Sections Surface Definition** dialog.
2. Add the first section, for example Sketch.2.
3. Set **Continuity** to **Tangent**.
4. Set **Deviation** to 0.001mm.

The dialog shows the selected section and options for guides, spine, coupling, relimitation, and continuity.

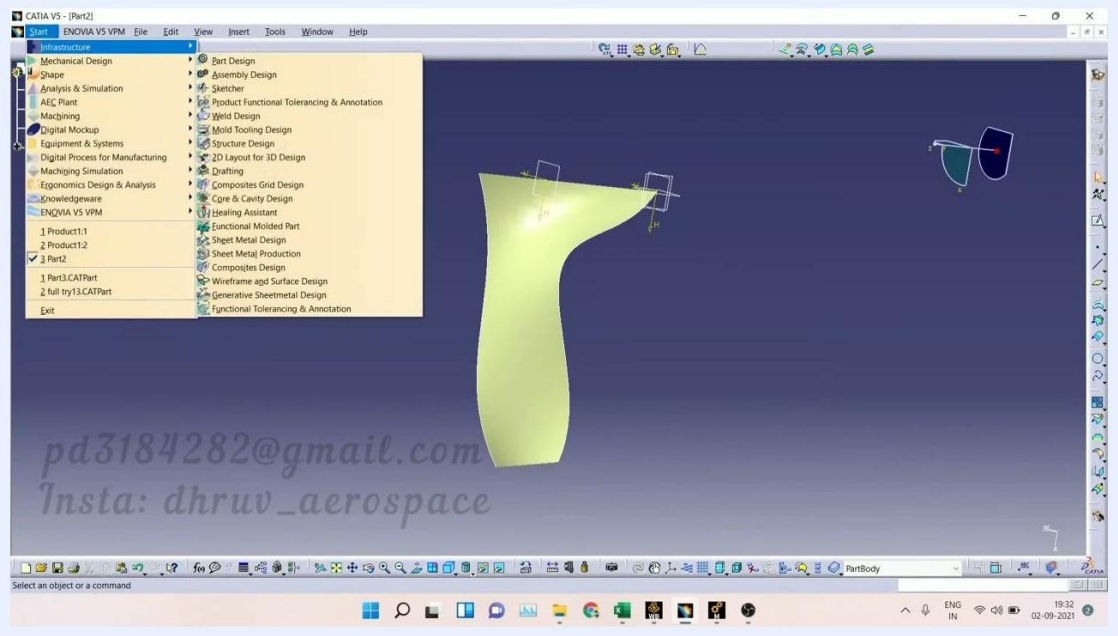


The Multi-Sections Surface Definition dialog with the first section added, above the workspace with two splines and planes

Step 9: Add the second section and generate the surface

1. In the **Multi-Sections Surface Definition** dialog, add the second section, for example Sketch.1.
2. Check that both sections are listed.
3. Confirm that the continuity and deviation settings are correct.

4. Click **OK** to generate the surface.



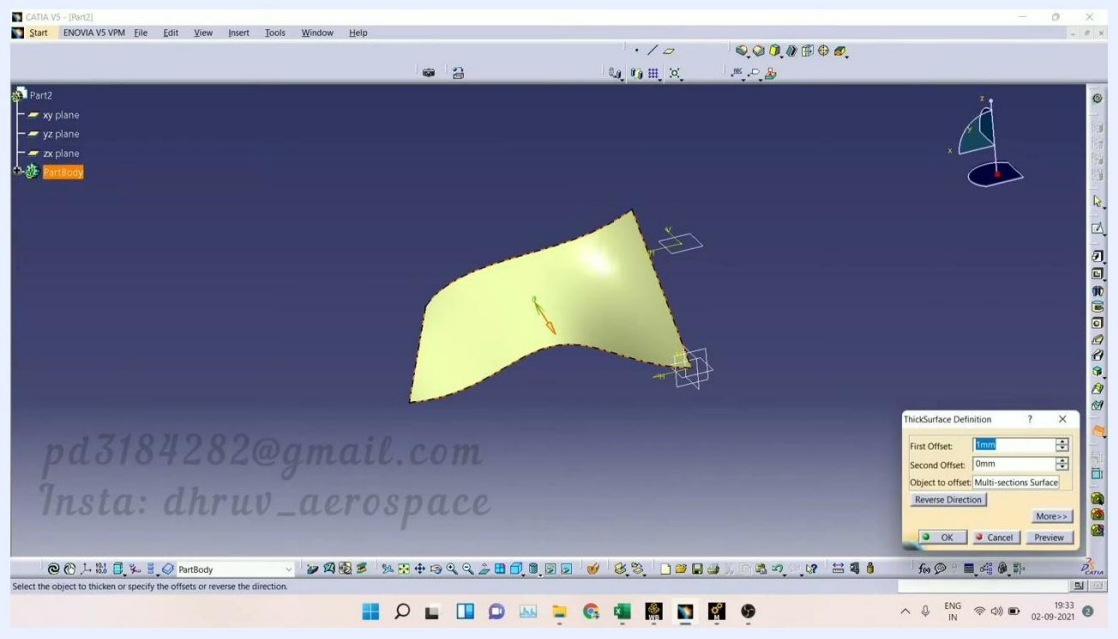
The Multi-Sections Surface Definition dialog with two sections added, above the workspace with two splines and planes

Step 10: Select the multi-section surface

Next, apply thickness to convert the surface to a solid in CATIA.

1. Make sure the multi-section surface is visible and highlighted in the workspace.

The surface appears as a smooth, yellow-shaded patch in the 3D view. The product tree shows Part2 > PartBody with the multi-section surface listed. No dialog boxes are open at this stage.



The multi-section surface highlighted in the workspace with no dialogs open and the product tree visible

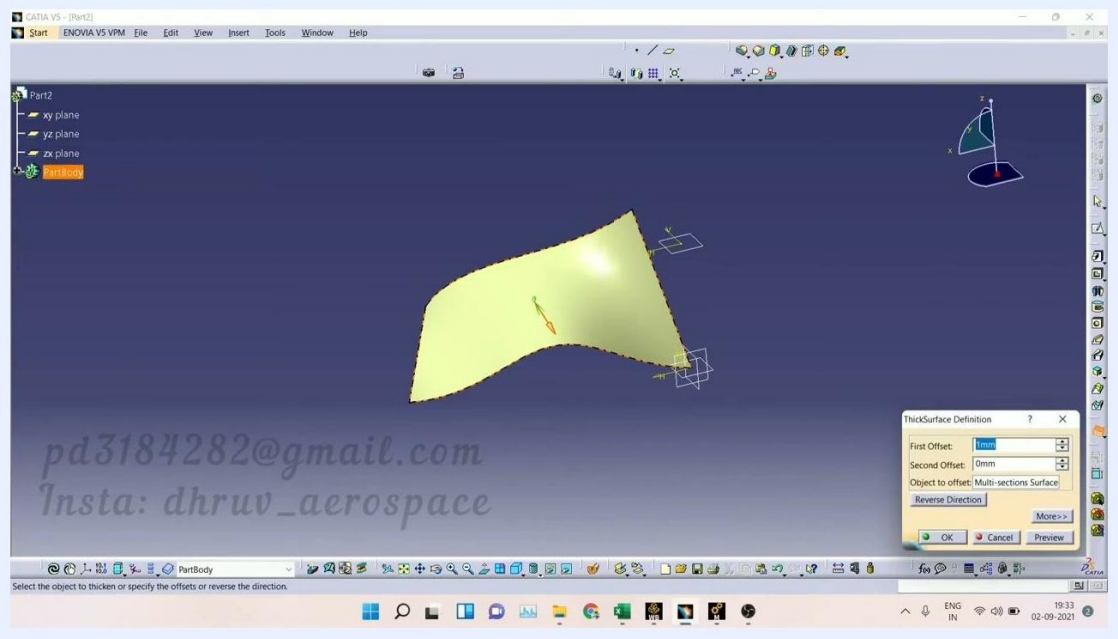
Step 11: Open the ThickSurface Definition dialog

1. Start the thickening operation. You typically do this by selecting the "Thick Surface" tool from the toolbar.

The **ThickSurface Definition** dialog appears in the lower right corner and shows these fields:

- **First Offset:** 1mm
- **Second Offset:** 0mm
- **Object to offset:** Multi-sections Surface

The surface is highlighted in the workspace, and an arrow indicates the direction of the thickness. The dialog also provides **Reverse Direction**, **OK**, **Cancel**, and **Preview**.

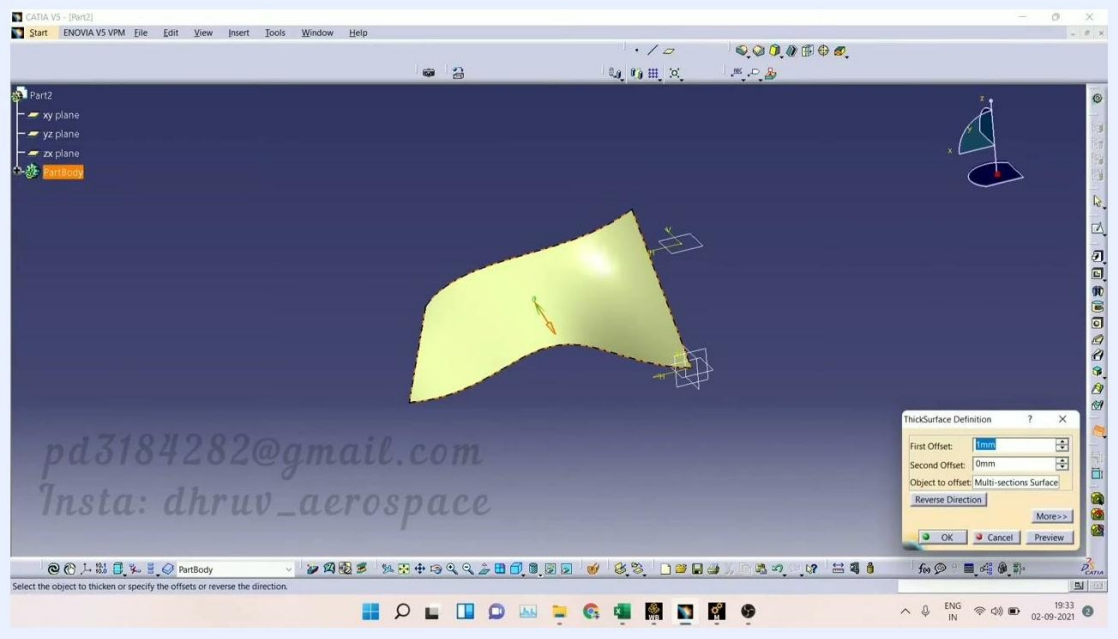


The ThickSurface Definition dialog with First Offset 1mm, Second Offset 0mm, Multi-sections Surface selected, and an arrow showing the thickness direction

Step 12: Preview the thickened surface

1. Click **Preview** in the **ThickSurface Definition** dialog.

The surface updates in the workspace and appears as a solid with visible thickness. The product tree stays visible, and the thickened surface is highlighted.

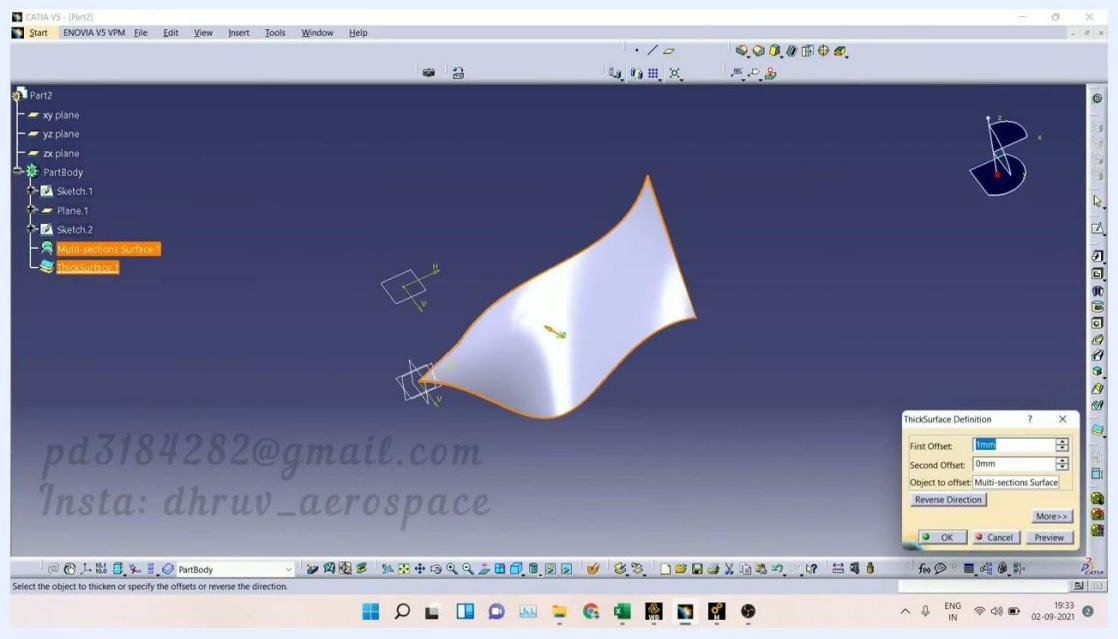


The workspace showing the thickened surface in preview, with the product tree visible and the dialog still open

Step 13: Apply the thickness to finish the surface-to-solid conversion

1. Click **OK** in the **ThickSurface Definition** dialog.

The dialog closes, and the thickened surface remains in the workspace with a shiny, solid appearance. The product tree now lists both Multi-sections Surface.1 and ThickSurface.1 under PartBody, both highlighted.

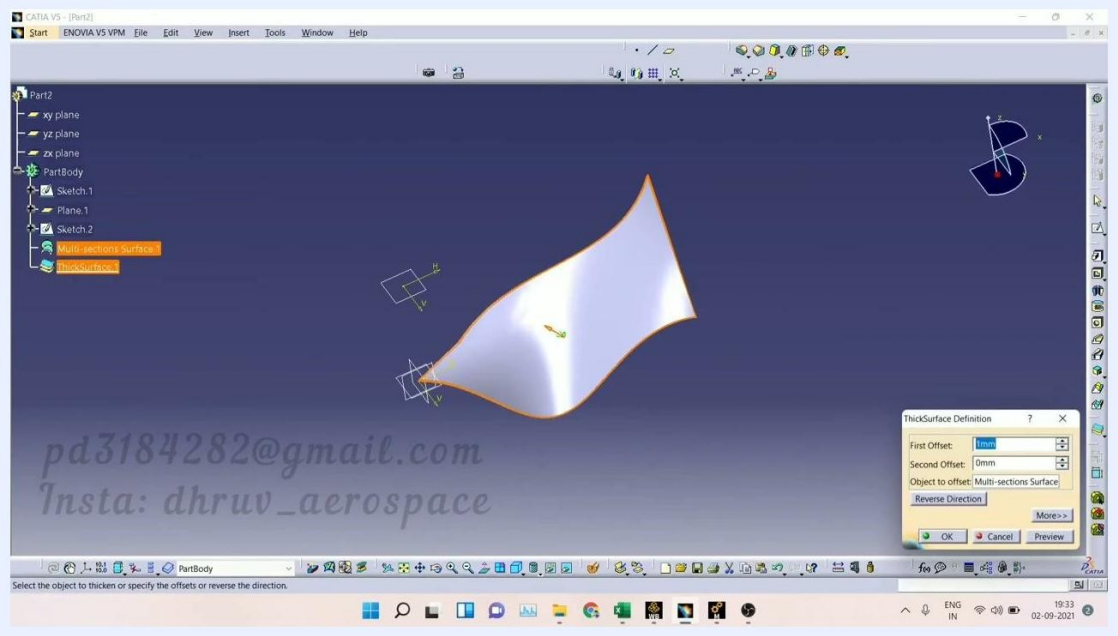


The finished thickened surface in the workspace, with Multi-sections Surface.1 and ThickSurface.1 highlighted in the product tree

Step 14: Inspect the final solid

1. Rotate or pan the view to inspect the thickened solid from different angles.

The workspace shows the completed part, and the product tree confirms the structure. No dialogs are open, and the thickened solid is clearly visible.



The thickened solid fully rendered in the workspace, with the final structure in the product tree and no dialogs open

Summary

You now know how to convert a surface to solid in CATIA V5. You drew two spline curves on offset planes and joined them into a multi-section surface. You then applied thickness with the **ThickSurface Definition** dialog, which turned the surface into a solid part.