How to extrude in CATIA V5: surfaces, dimensions, and "Up-to element" limits

This guide shows you how to extrude in CATIA V5 using the **Extruded Surface** tool. You start with a new part and extrude a curved sketch into a surface. You then extrude a second sketch and set its limit with **Up-to element**, so it ends exactly at the curved surface.

Prerequisites

- CATIA V5 is installed and running.
- You can open a new part document.
- You can draw a spline or curve on a sketch plane.

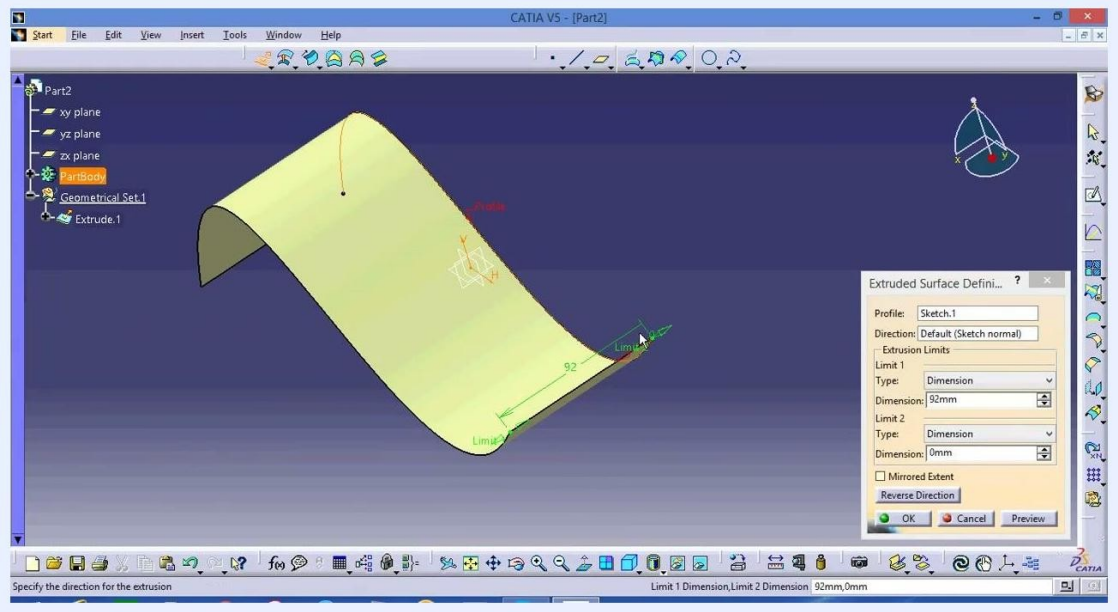
Steps

1. Open a new part and check the workspace

Launch CATIA V5 and open a new part document. Before you start, confirm the workspace shows these elements:

- The tree on the left lists Part2, xy plane, yz plane, zx plane, and PartBody.
- The main grid area is ready for sketching.
- The **User Selection Filter** dialog is open, which lets you filter selectable entities.

No geometry exists yet.



CATIA V5 workspace with grid, coordinate axes, and the User Selection Filter dialog. No geometry is present.

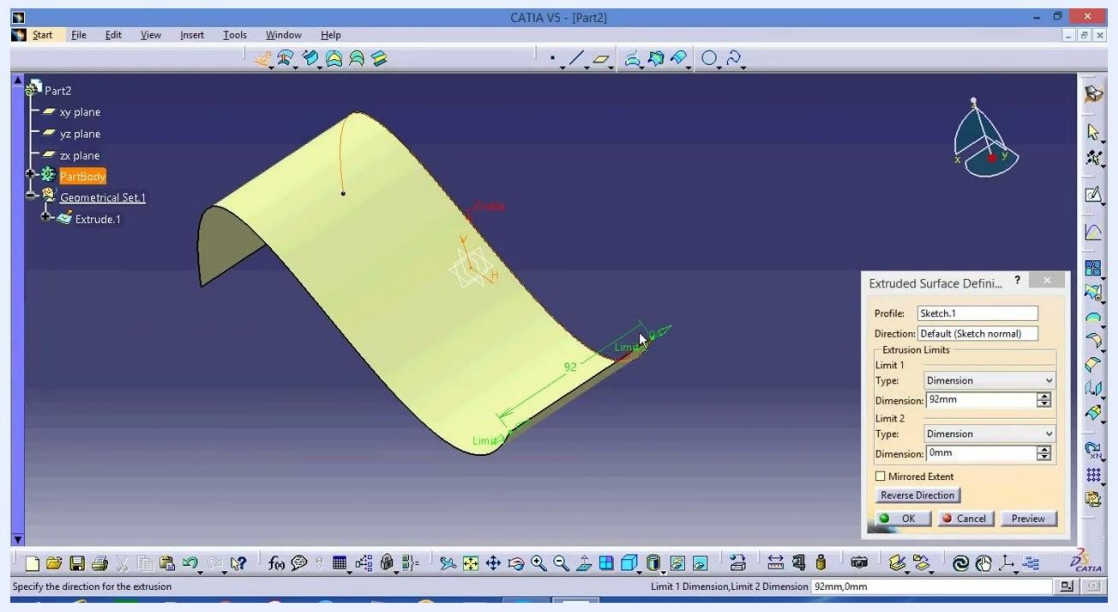
2. Extrude a curved sketch into a surface

Draw a spline or curve on the sketch plane. Then select the **Extruded Surface** tool to generate a surface from the sketch.

The tree now shows Geometrical Set.1 and Extrude.1 under PartBody, and the **Extruded Surface Definition** dialog opens on the right. Set these parameters:

- **Profile:** Sketch.1
- **Direction:** Default (Sketch normal)
- **Limit 1 Type:** Dimension
- **Limit 1 Dimension:** 92 mm
- **Limit 2 Type:** Dimension
- **Limit 2 Dimension:** 0 mm

The preview displays a yellow, curved extruded surface.



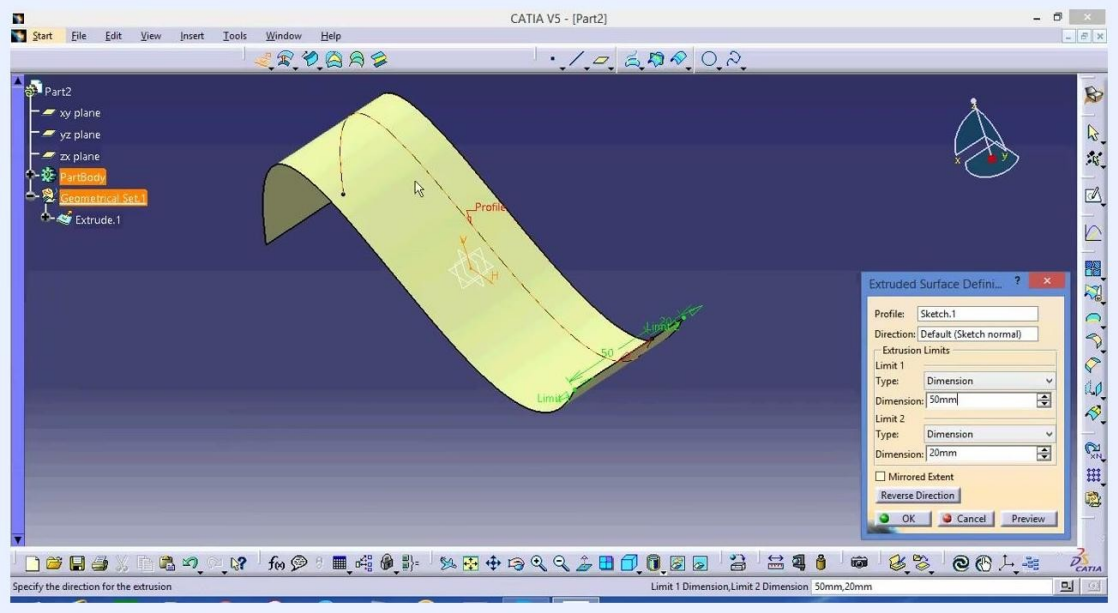
The Extruded Surface Definition dialog with Sketch.1 as the profile, Limit 1 at 92 mm, and Limit 2 at 0 mm, with the curved surface previewed.

3. Adjust the extrusion dimensions

To change the size of the surface, edit the limits in the **Extruded Surface Definition** dialog:

- **Limit 1 Dimension:** 50 mm
- **Limit 2 Dimension:** 20 mm

The preview updates to reflect the new dimensions, and both the surface and its profile remain clearly visible.



The Extruded Surface Definition dialog with Limit 1 at 50 mm and Limit 2 at 20 mm, with the updated curved surface previewed.

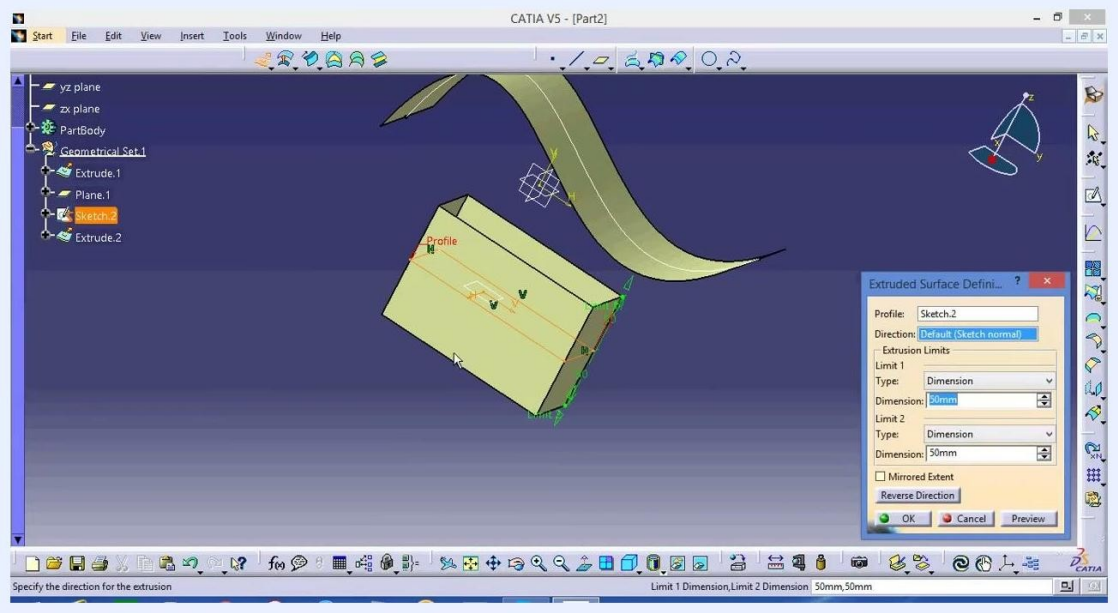
4. Create a second sketch and extrude it

Add a new plane (Plane.1) and create a second sketch (Sketch.2) on it. Then use the **Extruded Surface** tool again to extrude the new sketch.

The tree now shows Plane.1, Sketch.2, and Extrude.2 under Geometrical Set.1. In the **Extruded Surface Definition** dialog, set:

- **Profile:** Sketch.2
- **Direction:** Default (Sketch normal)
- **Limit 1 Type:** Dimension
- **Limit 1 Dimension:** 50 mm
- **Limit 2 Type:** Dimension
- **Limit 2 Dimension:** 50 mm

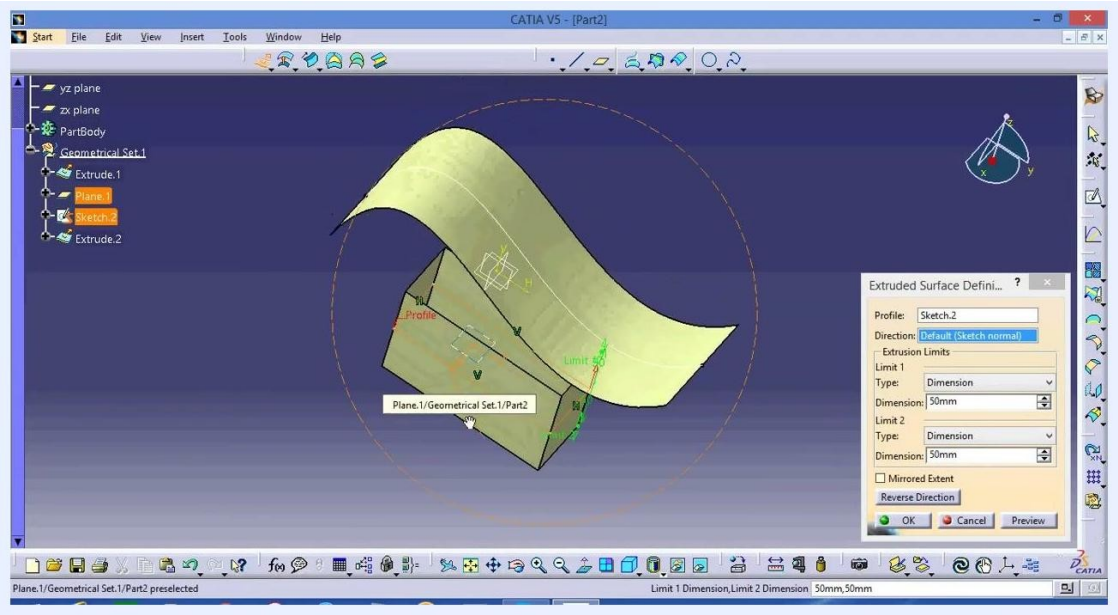
The preview shows a rectangular solid extruded from the new sketch, positioned below the curved surface.



The Extruded Surface Definition dialog for Sketch.2 with both limits at 50 mm, and a rectangular solid previewed below the curved surface.

5. Rotate the view and check the solid's position

Rotate the view to inspect how the curved surface and the extruded solid relate to each other. Select Plane.1 to highlight its position relative to both. The **Extruded Surface Definition** dialog stays open, so you can keep adjusting the extrusion.



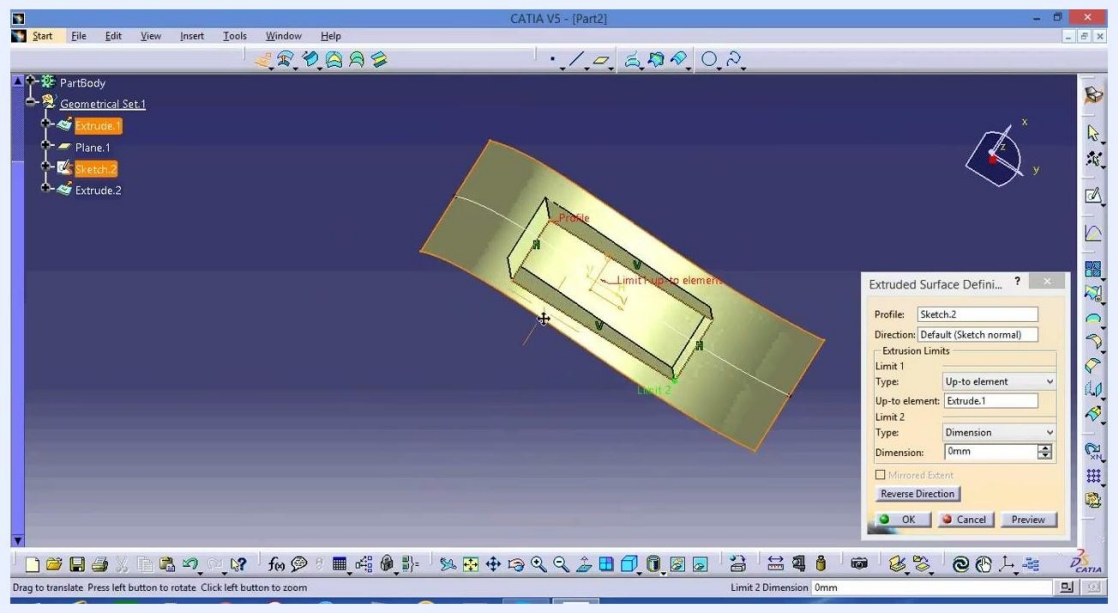
A view of the curved surface and the extruded solid, with Plane.1 highlighted.

6. Extrude up to the curved surface with "Up-to element"

Use an advanced limit option to make the extrusion end precisely at another surface. In the **Extruded Surface Definition** dialog, set:

- **Profile:** Sketch.2
- **Direction:** Default (Sketch normal)
- **Limit 1 Type:** Up-to element
- **Up-to element:** Extrude.1 (select the curved surface you created earlier)
- **Limit 2 Type:** Dimension
- **Limit 2 Dimension:** 0 mm

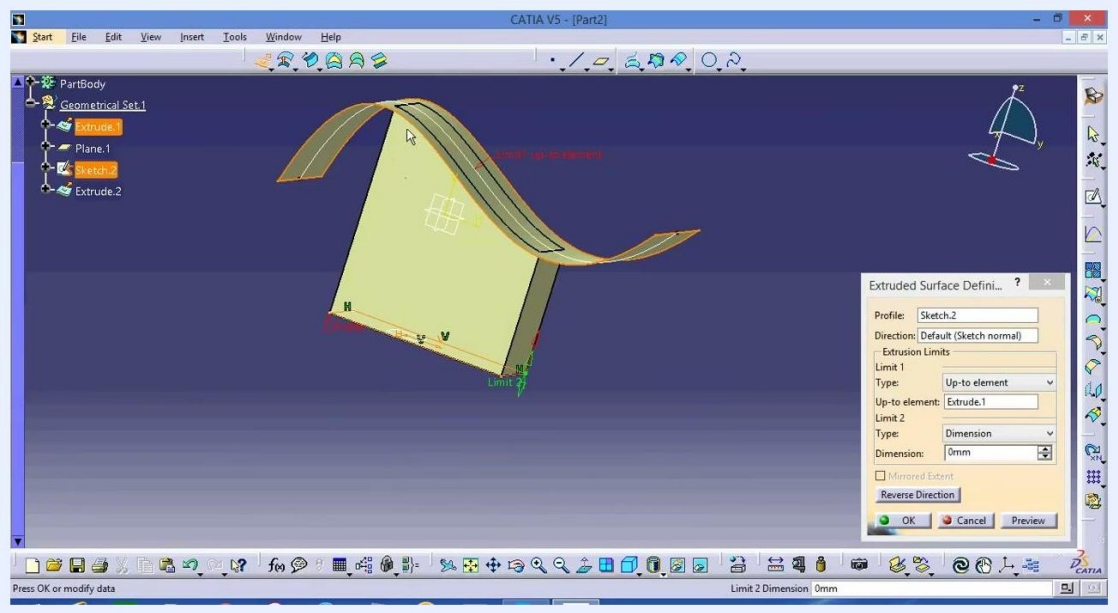
The preview shows the extrusion extending exactly up to Extrude.1, where the solid intersects and ends at the surface. The tree lists Extrude.1, Plane.1, Sketch.2, and Extrude.2 under Geometrical Set.1.



The CATIA V5 workspace showing the extrusion from Sketch.2 with Limit 1 set to Up-to element Extrude.1 and Limit 2 at 0 mm. The solid ends at the curved surface.

7. Rotate the model to inspect the intersection

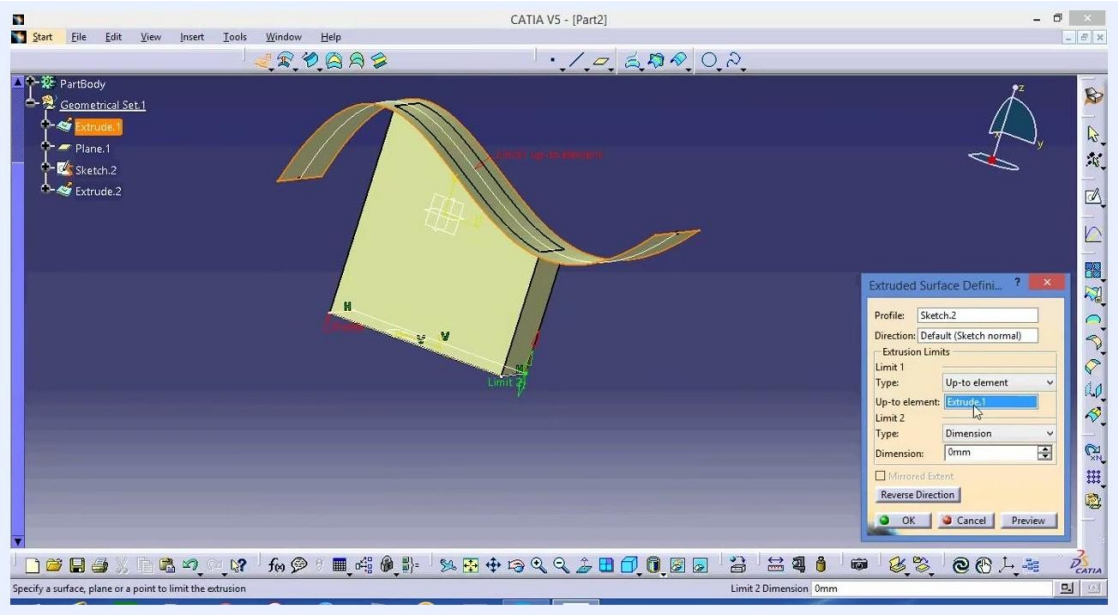
Rotate the 3D view to examine where the extruded solid meets the curved surface. Confirm that the top face of the solid aligns with the curved surface. The red "Limit up-to element" annotation marks this limit.



The model rotated to show the solid intersecting the curved surface, with the Limit up-to element annotation visible.

8. Verify the Limit 1 selection

In the **Extruded Surface Definition** dialog, check that **Up-to element: Extrude.1** is selected for Limit 1. Clicking or highlighting the Extrude.1 entry confirms the selection. The preview stays the same, with the solid ending at the curved surface.



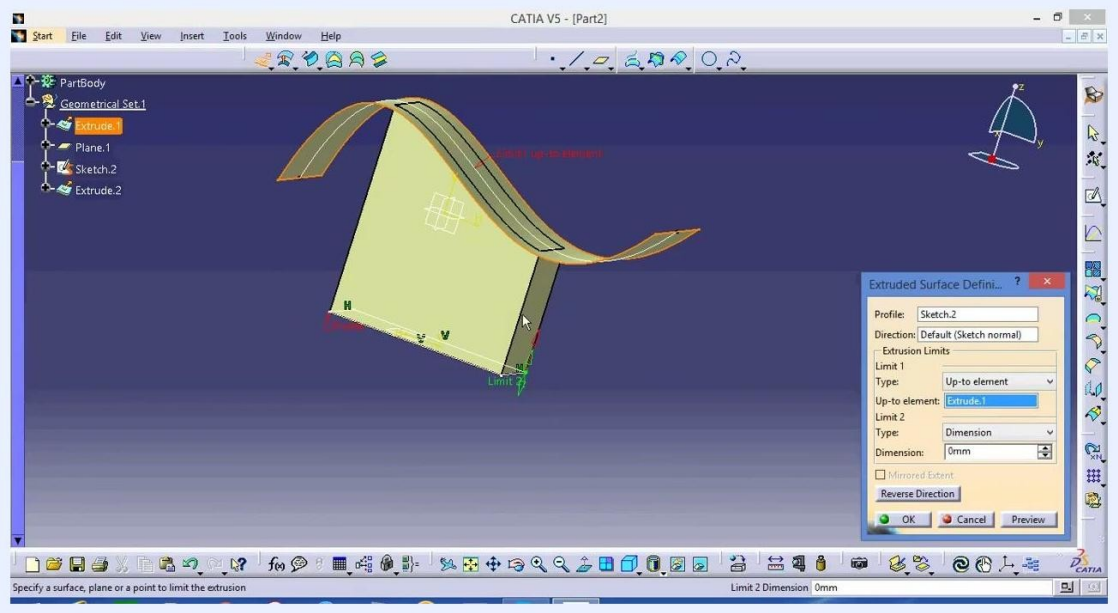
The dialog showing Up-to element Extrude.1 selected for Limit 1, with the solid previewed up to the curved surface.

9. Review the parameters and apply the extrusion

Review all parameters in the **Extruded Surface Definition** dialog:

- **Profile:** Sketch.2
- **Direction:** Default (Sketch normal)
- **Limit 1:** Up-to element (Extrude.1)
- **Limit 2:** Dimension (0 mm)

Click **OK** to apply the extrusion. The workspace now shows the completed solid, bounded by the curved surface and ready for further modeling or analysis.



The finished extrusion, with the solid bounded by the curved surface and all parameters confirmed in the dialog.

Summary

You now know how to extrude in CATIA V5 with the **Extruded Surface** tool. You extruded a curved sketch with dimension limits and adjusted those limits. You then extruded a second sketch and used **Up-to element** so it ends exactly at the existing curved surface.