How to make a pyramid in CATIA: build a 3D part from a sketch base

If you want to learn how to make a pyramid in CATIA V5, you first need the core Part Design workflow: a constrained base sketch, a point in 3D space, a plane, and a solid feature. This guide walks you through that workflow. You sketch a rectangle, dimension it, add a point and a plane, extrude a pad, and cut a pocket. You then repeat the pocket in a circular pattern and measure the finished part.

The example in this guide produces a rectangular pad with a patterned pocket. It does not produce a pyramid.

Prerequisites

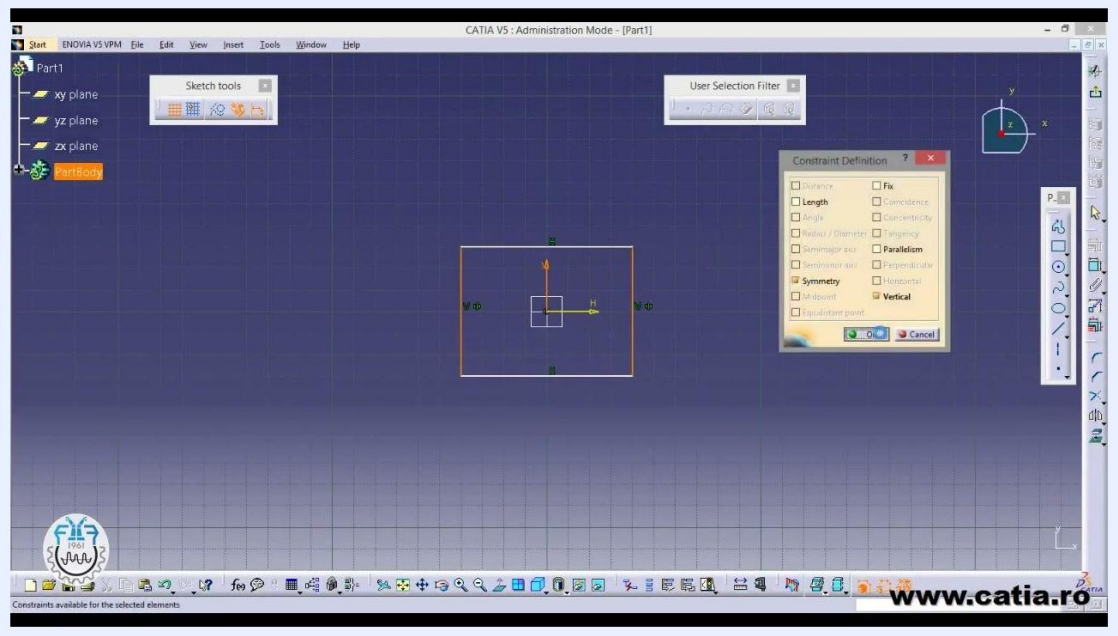
- CATIA V5 installed and ready to open.

Steps

1. Start a new part to begin your pyramid in CATIA

1. Open CATIA V5. The application window shows a starry background and the main menu bar at the top.
2. Click the **Start** menu in the upper left corner.
3. Hover over **Mechanical Design** in the dropdown menu.

4. Select **Part Design** from the extended menu.



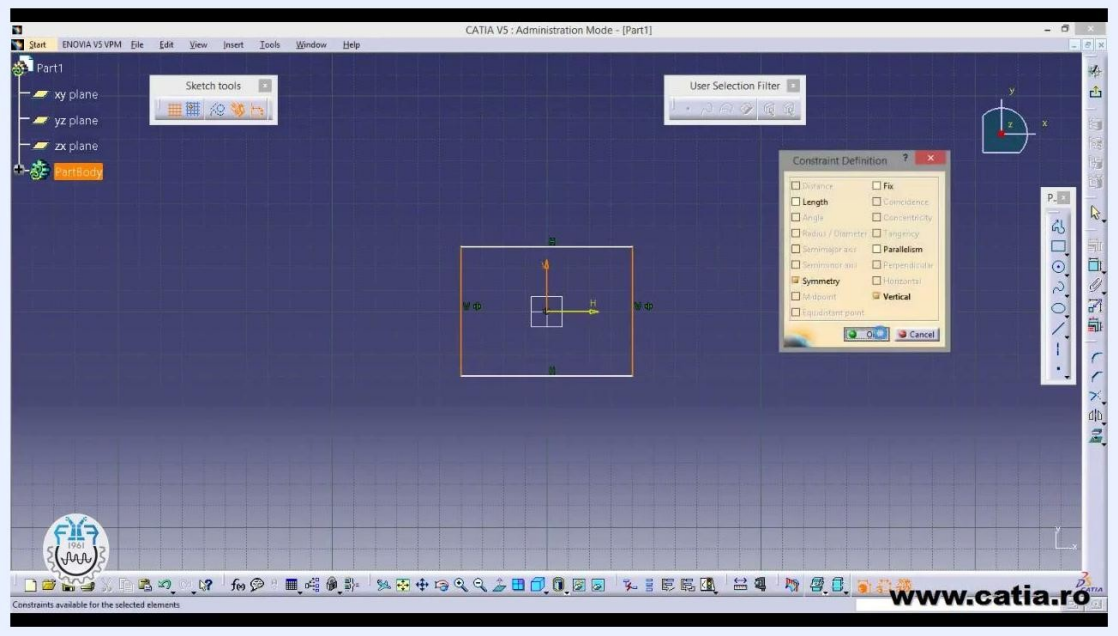
The CATIA V5 Start menu with Mechanical Design > Part Design selected, and a list of recent files below

2. Open the sketch environment

The Part Design workbench opens. The tree on the left shows Part1 with the default planes (xy plane, yz plane, zx plane) and PartBody.

The main area shows a grid with axes and a highlighted rectangle that marks the sketching area. The **Sketch tools** and **User Selection Filter** toolbars are visible. A **Constraint Definition** dialog also appears, where you set geometric constraints for the sketch.

Select the constraints you need, such as **Length**, **Symmetry**, or **Vertical**.

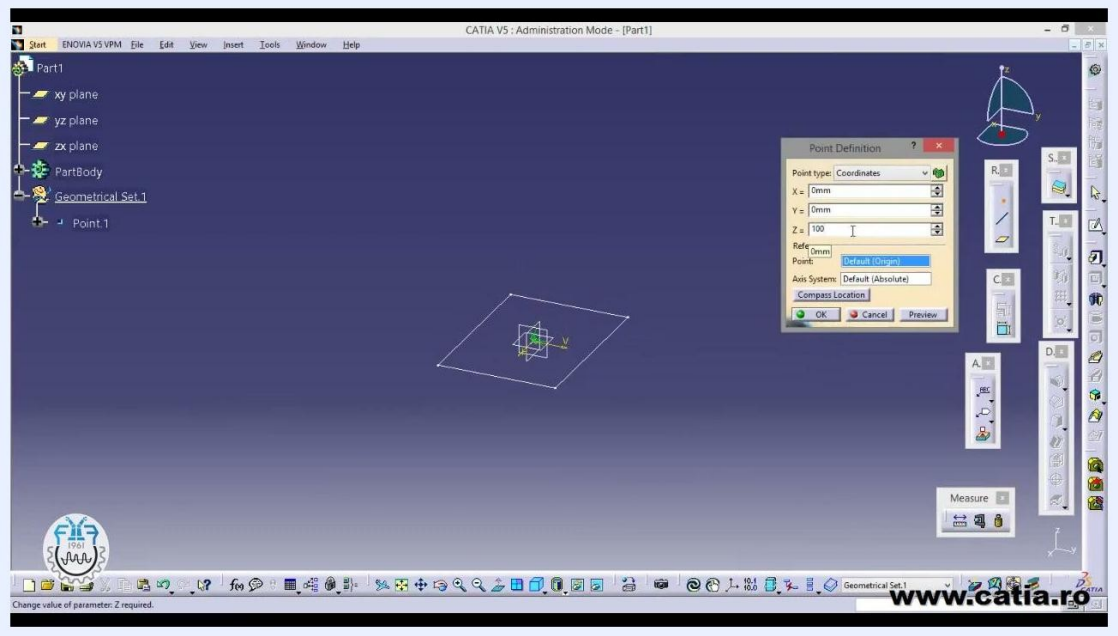


The CATIA V5 sketch environment with a grid, axes, the Constraint Definition dialog, and the Part1 tree

3. Constrain the base rectangle

1. Select the rectangle in the sketch area.
2. In the **Constraint Definition** dialog, check the constraints you want, such as **Length**, **Symmetry**, or **Horizontal**.
3. Click the green **OK** button to apply them.

The sketch lines turn green, which shows the rectangle is fully constrained.

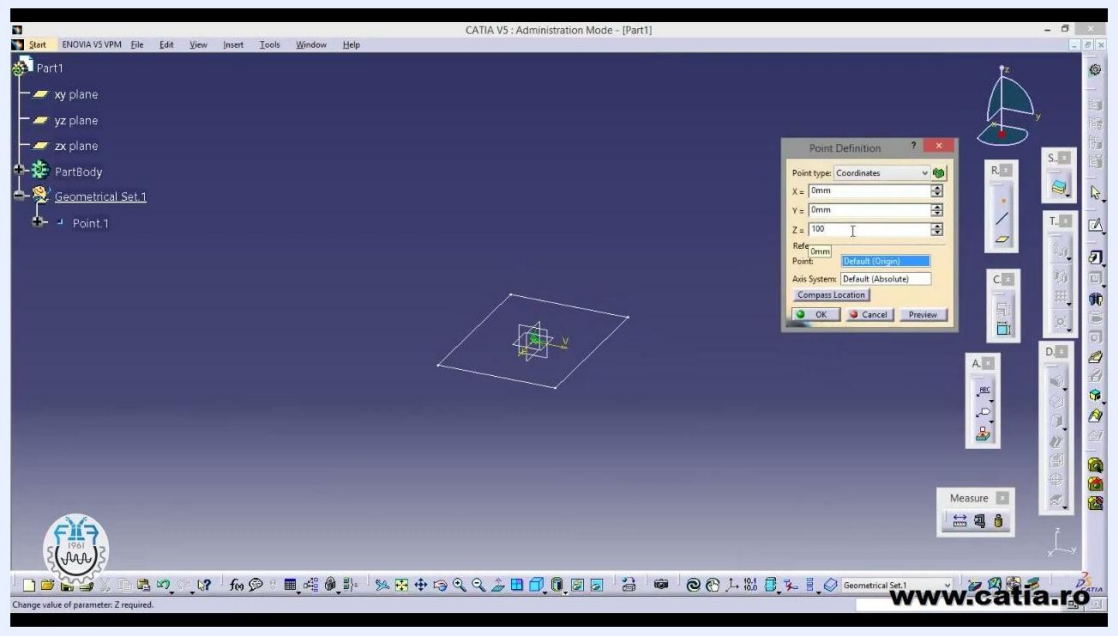


The selected rectangle in the sketch with the Constraint Definition dialog open and constraints being applied

4. Define the sketch dimensions

1. In the new **Constraint Definition** dialog that prompts for a value, enter 83 for the selected constraint, such as the length or width.
2. Click **OK** to confirm.

The rectangle now shows dimension labels, such as 83 and 60.

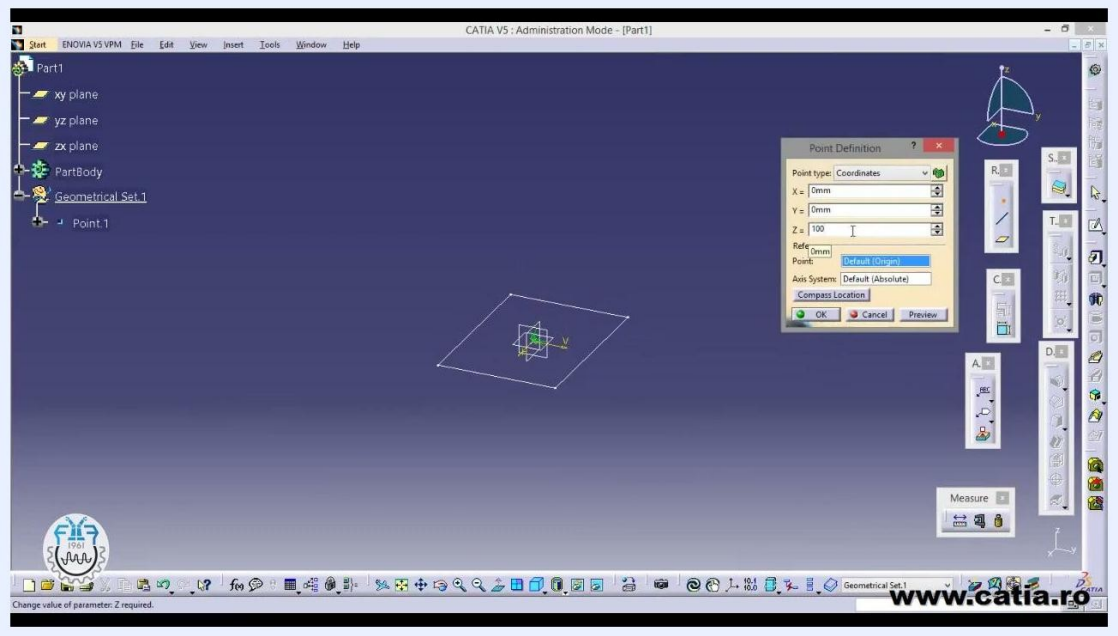


The sketch with dimension labels 83 and 60 and the Constraint Definition dialog for entering a value

5. Create a point in 3D space

1. Switch to a 3D view. The tree now includes Geometrical Set.1 with Point.1.
2. In the **Point Definition** dialog, set **Point type** to **Coordinates**.
3. Enter X = 0mm, Y = 0mm, and Z = 100mm.
4. Make sure **Point** is set to **Default (Origin)** and **Axis System** is set to **Default (Absolute)**.

5. Click **OK** to create the point.

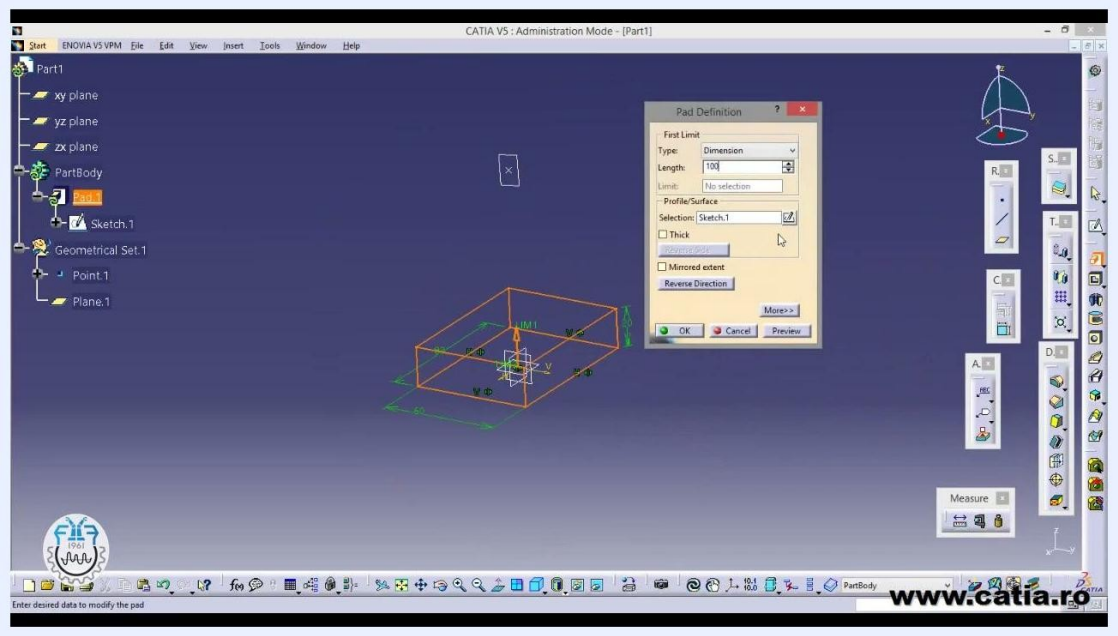


The 3D view with the Point Definition dialog showing X=0mm, Y=0mm, Z=100mm, and Point.1 in the tree

6. Define a plane through points

1. In the **Plane Definition** dialog, set **Plane type** to **Through three points**.
2. For **Point 1**, select Point.1.
3. For **Point 2**, select Sketch.1\Vertex.
4. Leave **Point 3** as **No selection** for now.

The new plane appears in the 3D view and is listed as Plane.1 under Geometrical Set.1 in the tree.

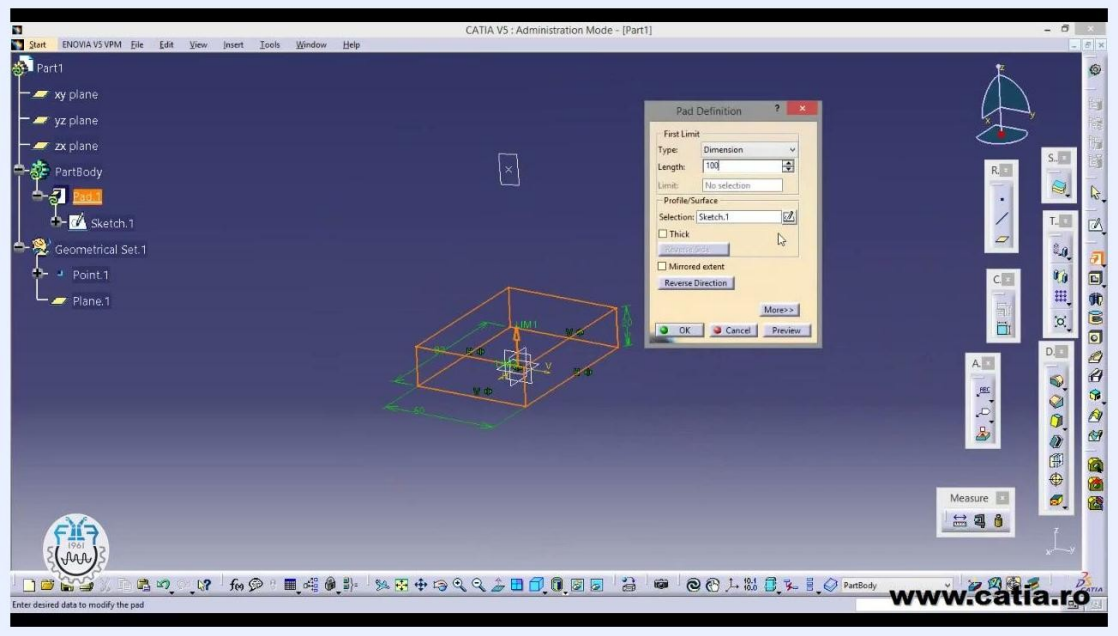


The 3D view with the Plane Definition dialog, Point.1 and Sketch.1\Vertex selected, and Plane.1 in the tree

7. Create the 3D pad

1. In the **Pad Definition** dialog, set **Type** to **Dimension**.
2. Enter Length = 100.
3. Make sure **Selection** is set to Sketch.1.
4. Optionally, check or uncheck **Thick**, **Mirrored extent**, or **Reverse Direction**.
5. Click **OK** to create the pad.

The tree now shows Pad.1 under PartBody.

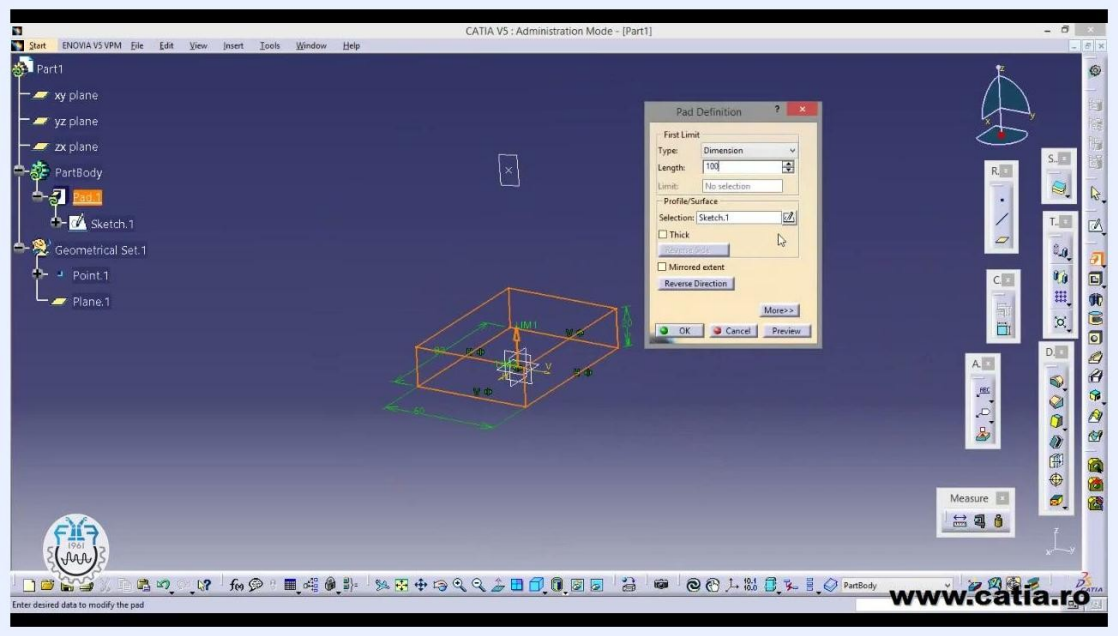


The Pad Definition dialog with Length 100 and Selection Sketch.1, and the 3D pad in the workspace

8. Inspect the pad

The workspace now shows a solid rectangular box. You can rotate it with the mouse to view it from different angles.

The tree shows the structure Part1 > PartBody > Pad.1 > Sketch.1 and Geometrical Set.1 > Point.1, Plane.1.



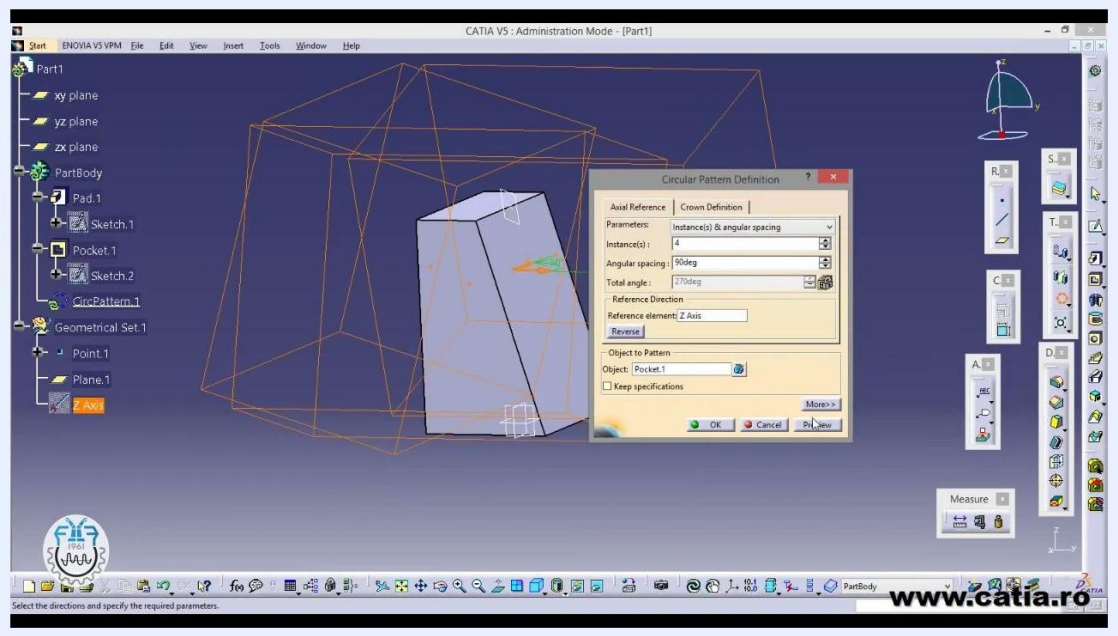
A rotated solid rectangular pad in the 3D view with the tree structure visible

9. Confirm the pocket feature

The **Pocket Definition** dialog shows these settings:

- **Type:** Dimension
- **Depth:** 100mm
- **Selection:** Sketch.2
- **Options: Thick**, **Mirrored extent**, and **Reverse Direction** are unchecked.

The 3D view shows orange outlines as a preview of the pocket, and Pocket.1 is highlighted under PartBody in the tree. Click the green **OK** button to create the pocket.



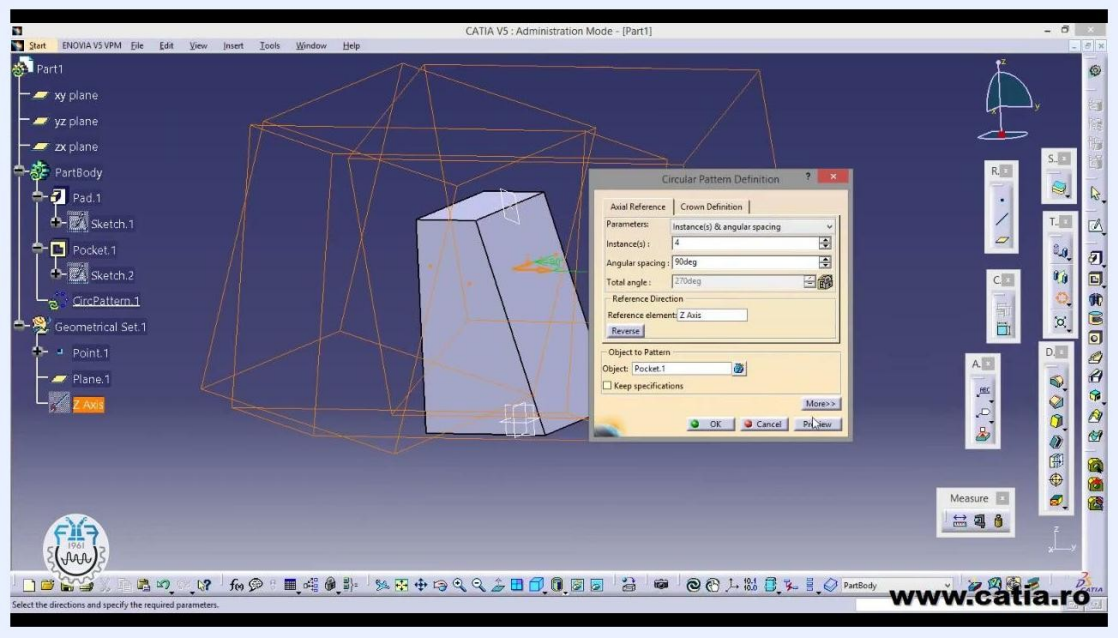
The Pocket Definition dialog with Depth 100mm and Selection Sketch.2, a 3D pocket preview, and Pocket.1 highlighted in the tree

10. Start a circular pattern of the pocket

The **Circular Pattern Definition** dialog lets you duplicate the pocket in a circular array. Set these parameters:

- **Instance(s):** 4
- **Angular spacing:** 90 degrees
- **Total angle:** 270 degrees
- **Object to Pattern:** Pocket.1
- **Keep specifications:** unchecked

For **Reference Direction**, open the dropdown to choose a reference element. The options include X Axis, Y Axis, Z Axis, Create Line, Create Plane, and Create Compass Direction. The tree now shows CircPattern.1 under PartBody.

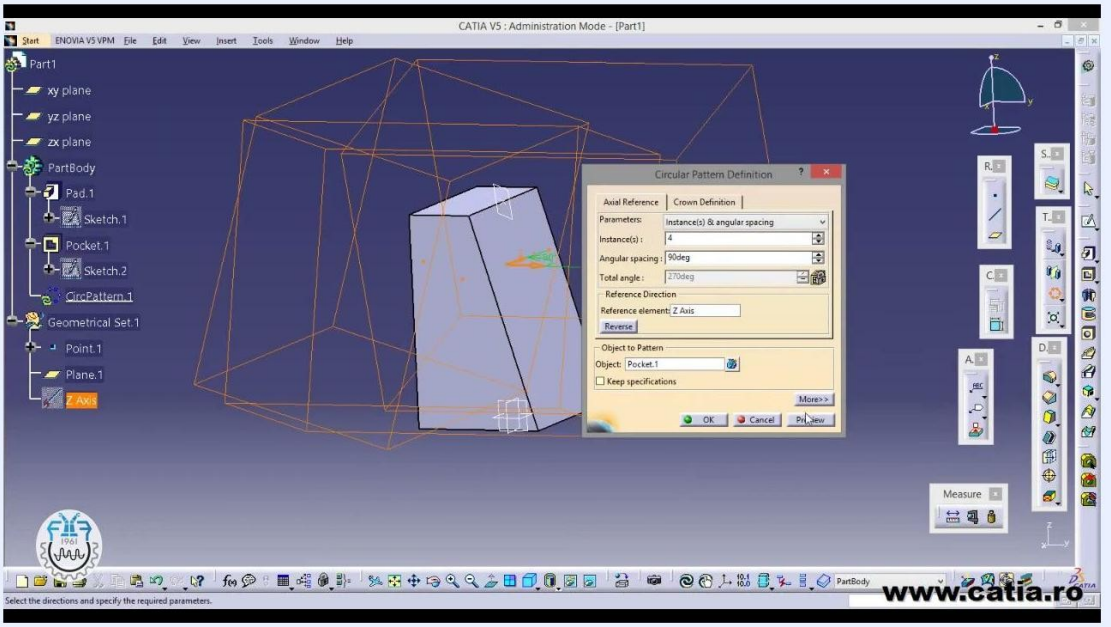


The Circular Pattern Definition dialog with 4 instances, 90 degree spacing, 270 degree total angle, and the reference element dropdown open

11. Set the reference axis

1. With the dialog still open, set **Reference element** to **Z Axis**.
2. Check the orange outlines in the 3D view, which show the previewed pattern positions around the solid. The tree lists Z Axis under Geometrical Set.1.

3. Click **Preview** to visualize the pattern before you confirm it.

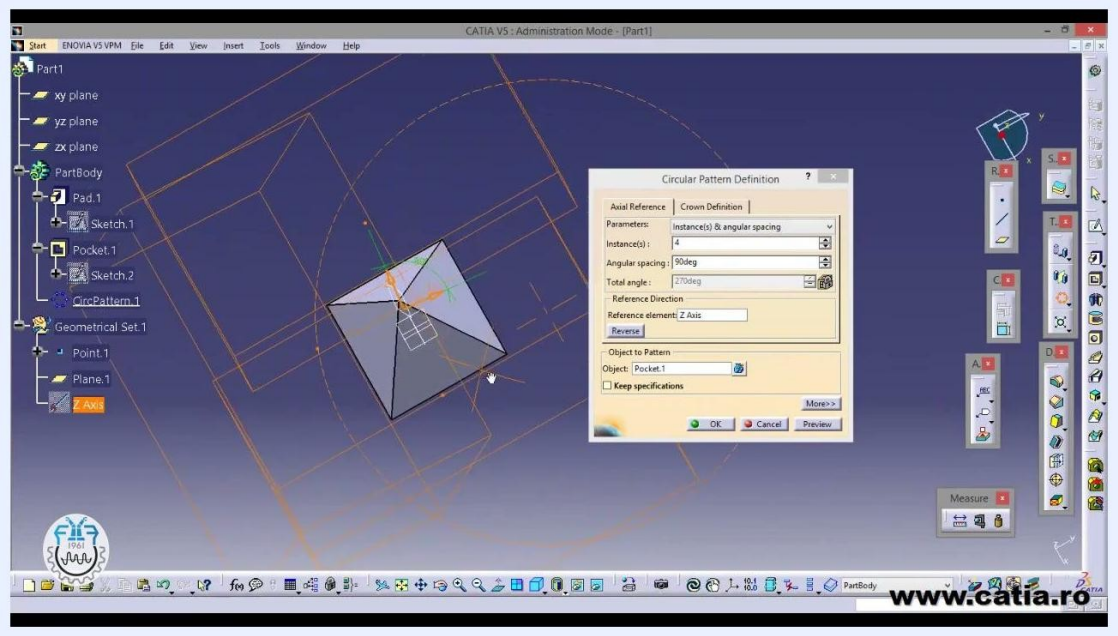


The Circular Pattern Definition dialog with Reference element set to Z Axis and a 3D preview of the pattern

12. Review the circular pattern preview

Rotate the 3D view to a top-down perspective. The dialog still shows 4 instances, 90 degree angular spacing, a 270 degree total angle, Z Axis as the reference element, and Pocket.1 as the object.

The orange outlines show the four patterned pockets distributed around the Z axis. Check that the pattern is correct before you finalize it.



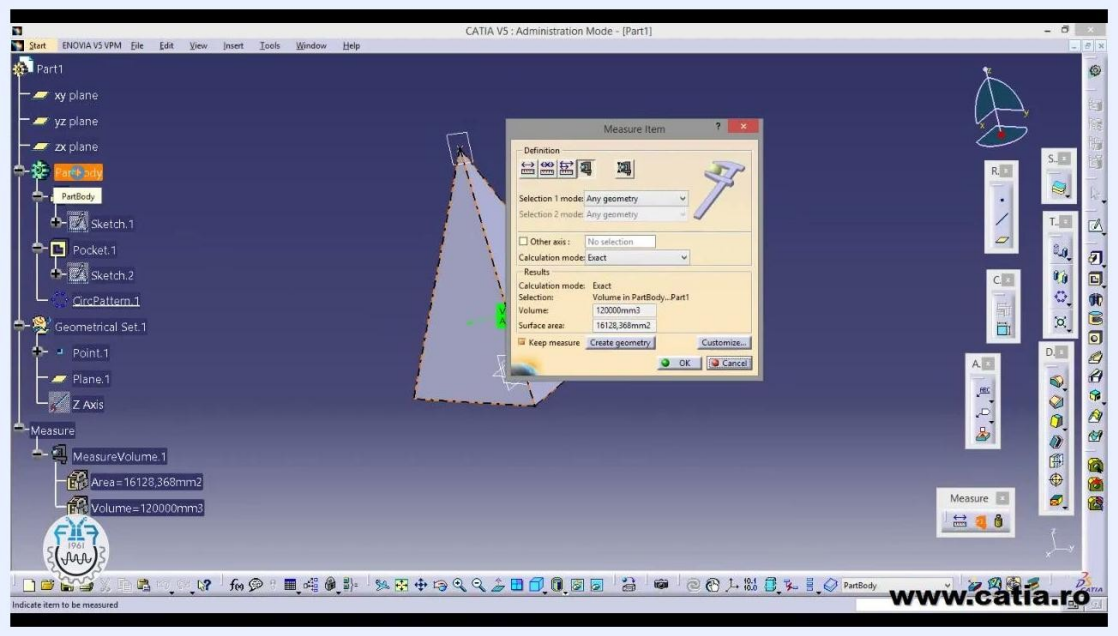
A top-down view with the Circular Pattern Definition dialog and a preview of four pockets around the Z axis

13. Measure the volume and surface area

The **Measure Item** dialog shows the measurement results for the part:

- **Calculation mode:** Exact
- **Selection:** Volume in PartBody...Part1
- **Volume:** 1200000 mm³
- **Surface area:** 16128.368 mm²

The tree now includes a **Measure** section with MeasureVolume.1, which lists the area and volume. Click **OK** to close the dialog.

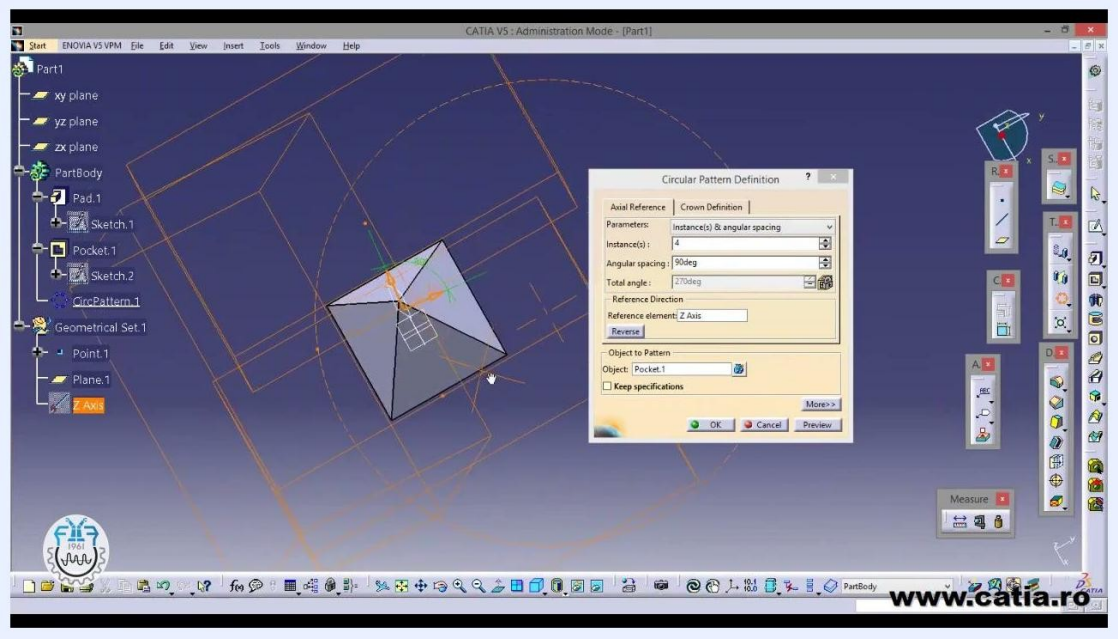


The Measure Item dialog showing Volume 1200000 mm³ and Surface area 16128.368 mm², with MeasureVolume.1 in the tree

14. View the final measurement results

The 3D view shows the part with a green label that reads **Volume = 1200000 mm³** and **Area = 16128.368 mm²**.

The tree confirms the part structure, including the pad, pocket, circular pattern, and measurement results. The workspace shows only the part and the measurement label.



The 3D view with a green label showing Volume 1200000 mm³ and Area 16128.368 mm², and the final part structure in the tree

Summary

You created a 3D part in CATIA V5. You sketched and constrained a rectangular base, added a point and a plane, extruded a pad, and created a pocket. You then patterned the pocket around the Z axis and measured the part's volume and surface area. These are the same building blocks you use when you learn how to make a pyramid in CATIA.

What's next

- Rotate the finished part and compare it with the tree structure.
- Change the circular pattern parameters and use **Preview** to see how the result changes.