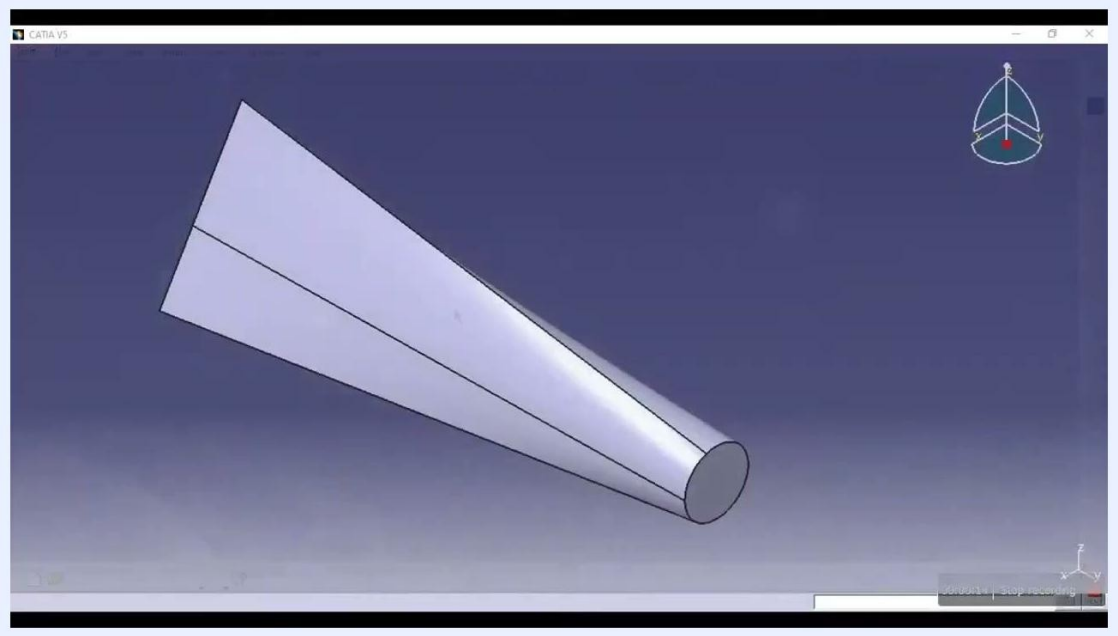
How to use multi-section solid in CATIA V5

This guide shows you how to use multi-section solid in CATIA V5. You start with two circular sections, then move to sections with different profiles, such as a circle and a triangle. You then control the result with coupling modes, a spine curve, and guide curves. By the end, you have built a curved solid and compared it with a reference image.

Prerequisites

- CATIA V5 is installed and open on the main application window.
- You know how to open the Sketcher workbench and draw basic shapes such as circles, lines, and triangles.

When CATIA V5 starts, you see a starry background with the title bar and the standard menu options (**Start**, **File**, **Edit**, **View**, **Insert**, **Tools**, **Window**, **Help**). No part or model is open yet.



CATIA V5 main window with a starry background and the menu bar

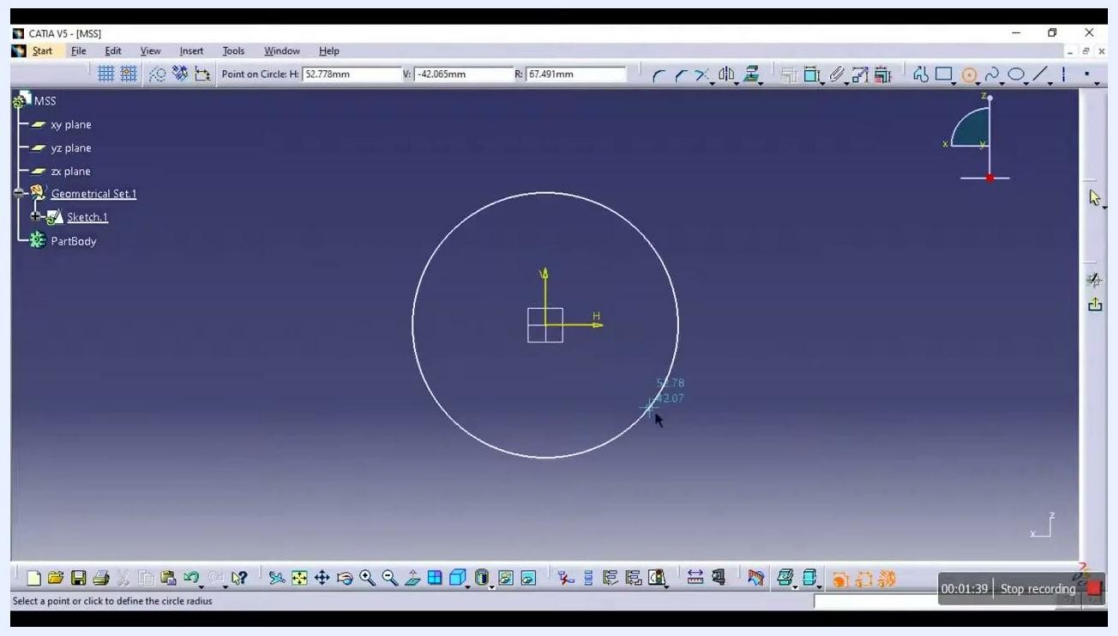
Understand padding and multi-section solid

Step 1. Compare padding with a multi-section solid

Before you build the model, review how the two features differ:

- **Padding:** Between any two sections, the cross-sectional area and the profile stay the same along the whole length.
- **Multi-section solid:** The cross-sectional area and the profile can be completely different from one section to the next.

The comparison shows two 3D models. The "Padding" model is a cylinder with the same cross-section throughout. The "Multi-section solid" model is a tapered or morphing solid with different cross-sections.



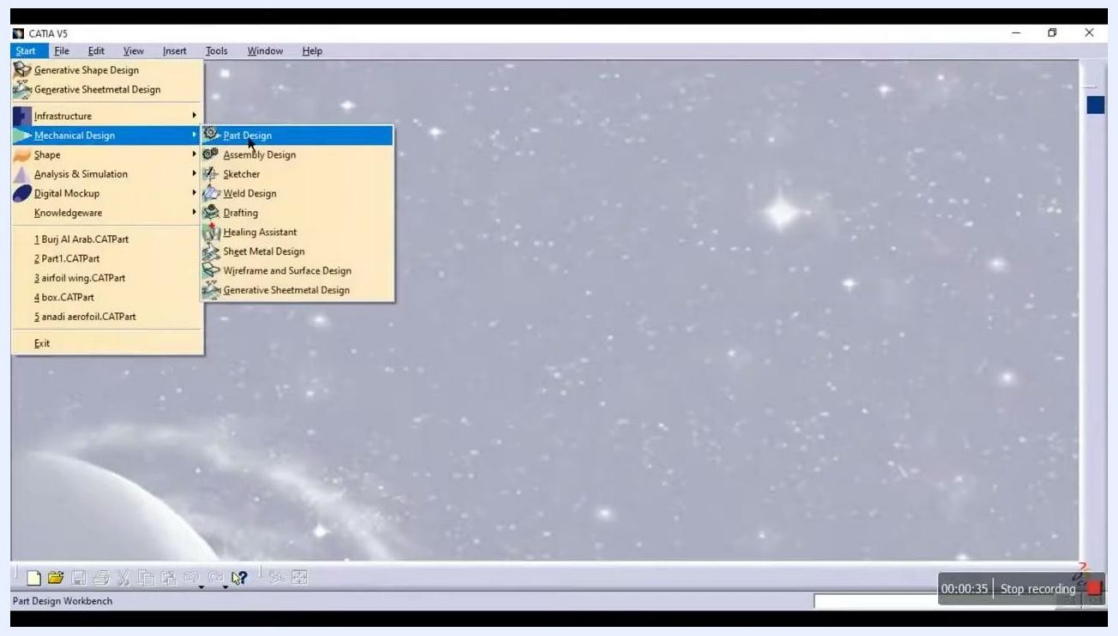
Comparison of a padded solid (same cross-sectional area) and a multi-section solid (different cross-sectional areas) in CATIA V5

Set up the part and the first two sections

Step 2. Open the Part Design workbench

1. Open the **Start** menu.
2. Go to **Mechanical Design**.
3. Select **Part Design** from the list.

The Part Design workbench opens.

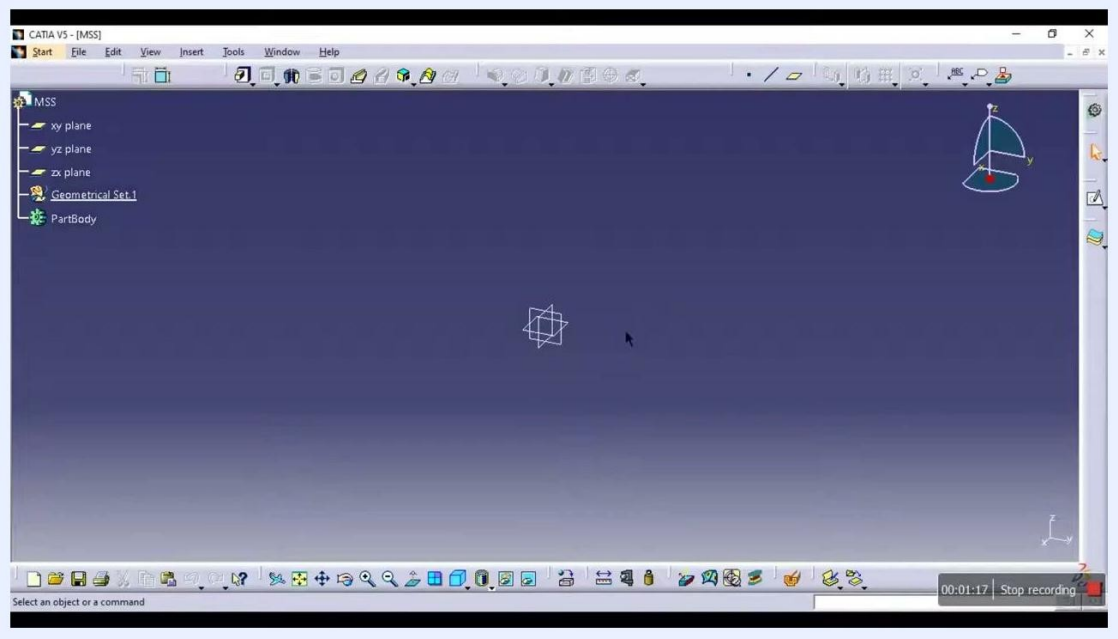


Start menu open with Mechanical Design > Part Design selected in CATIA V5

Step 3. Rename the part and review the workbench

Rename the file **MSS** (Multi Section Solid) so it is easy to identify.

The tree on the left shows the part name (MSS), the standard planes (xy, yz, zx), **Geometrical Set.1**, and **PartBody**. The main workspace is empty and ready for sketching.



Part Design workbench showing the MSS part and the standard planes in the tree

Step 4. Select the ZX plane

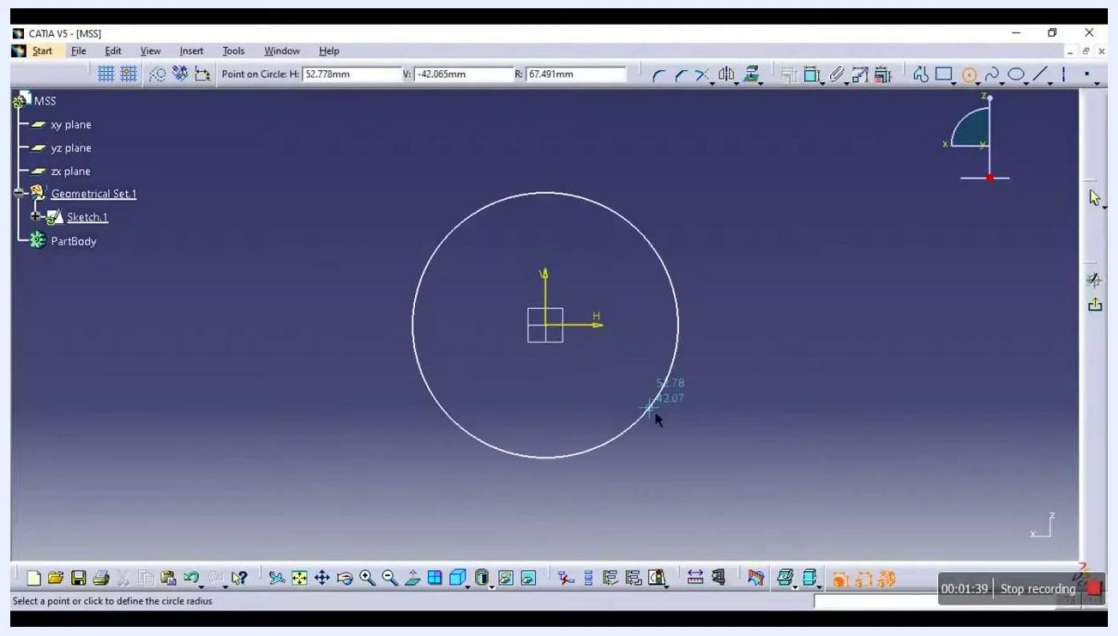
Start with a simple sketch using the default settings.

1. Select the **ZX plane** in the tree on the left.
2. Enter the Sketcher workbench.

Step 5. Draw the first circle

1. Draw a circle on the ZX plane.
2. Set the radius to **60 mm**.

3. Complete the sketch and exit the Sketcher workbench.

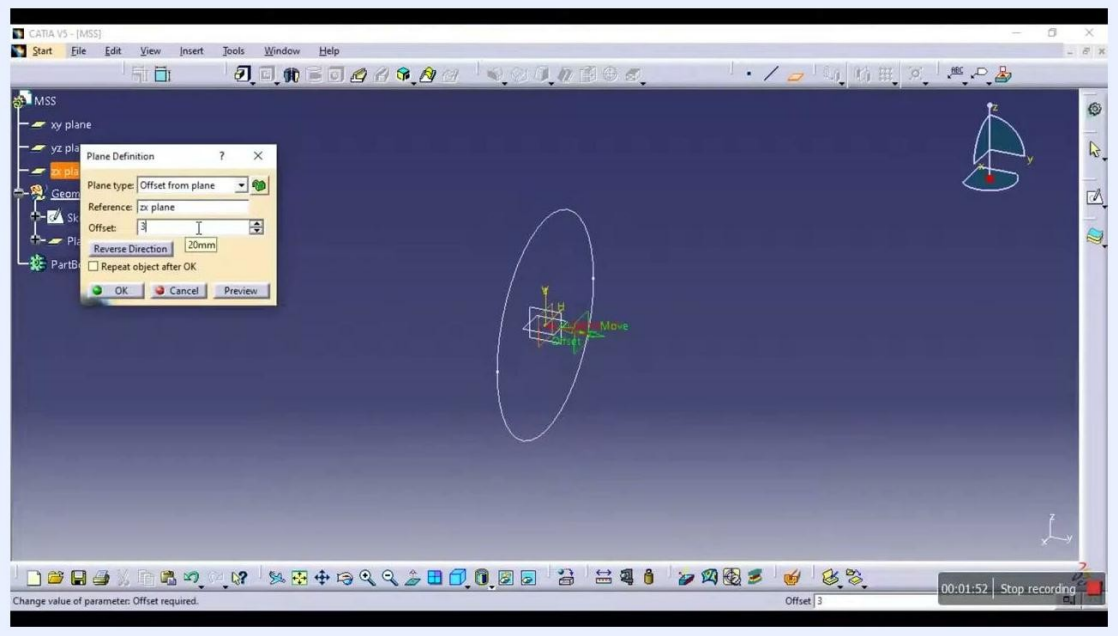


Circle sketch with a 60 mm radius on the ZX plane in CATIA V5

Step 6. Create an offset plane for the second section

1. Open the **Plane Definition** dialog.
2. Set **Plane type** to "Offset from plane".
3. Set **Reference** to "zx plane".
4. Enter **Offset** as **300 mm**. If needed, use "Reverse Direction" to flip the offset.

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

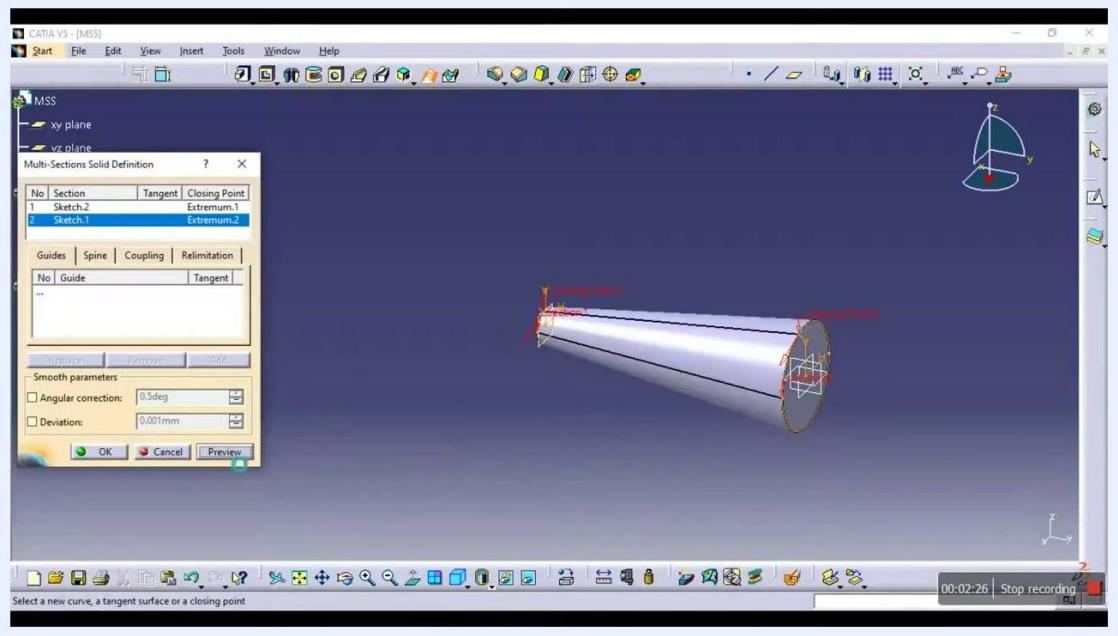


Plane Definition dialog with an offset from the zx plane set to 300 mm

Step 7. Draw a smaller circle on the offset plane

1. Make sure you are working on the new offset plane (Plane.1).
2. Enter the Sketcher workbench.
3. Draw a circle with a **radius of 20 mm**.

4. Complete the sketch and exit the Sketcher workbench.



Sketcher workbench active on the offset plane while drawing a circle with a 20 mm radius

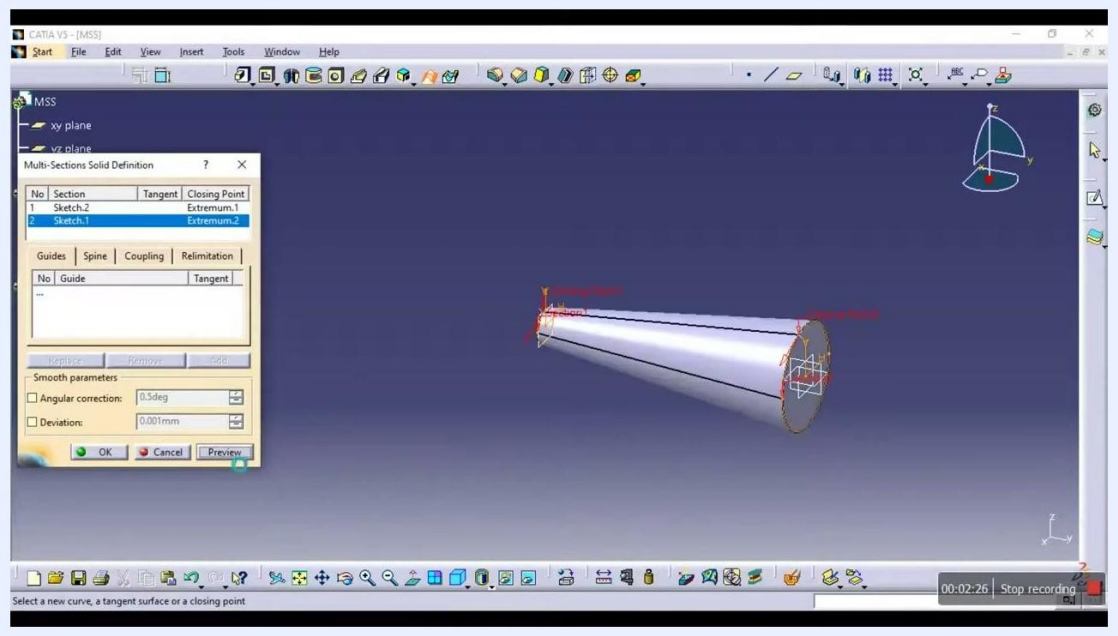
Create a multi-section solid between two circles

Step 8. Use the Multi-Section Solid command

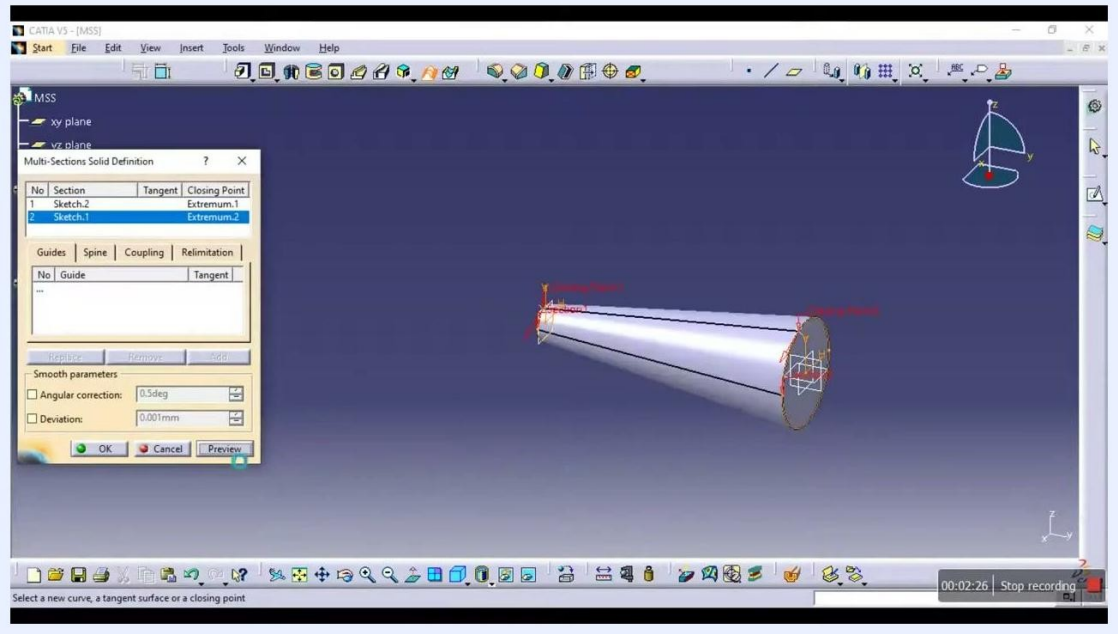
Now use multi-section solid in CATIA V5 with the two circles.

1. Start the **Multi-Section Solid** command.
2. In the Multi-Sections Solid Definition dialog, choose the **first section** as the smaller circle (Sketch.2).
3. Choose the **second section** as the larger circle (Sketch.1).
4. Click **Preview** to visualize the solid.

The solid transitions from a small circle to a large circle. The cross-sectional areas differ, but the profile stays the same (concentric circles).



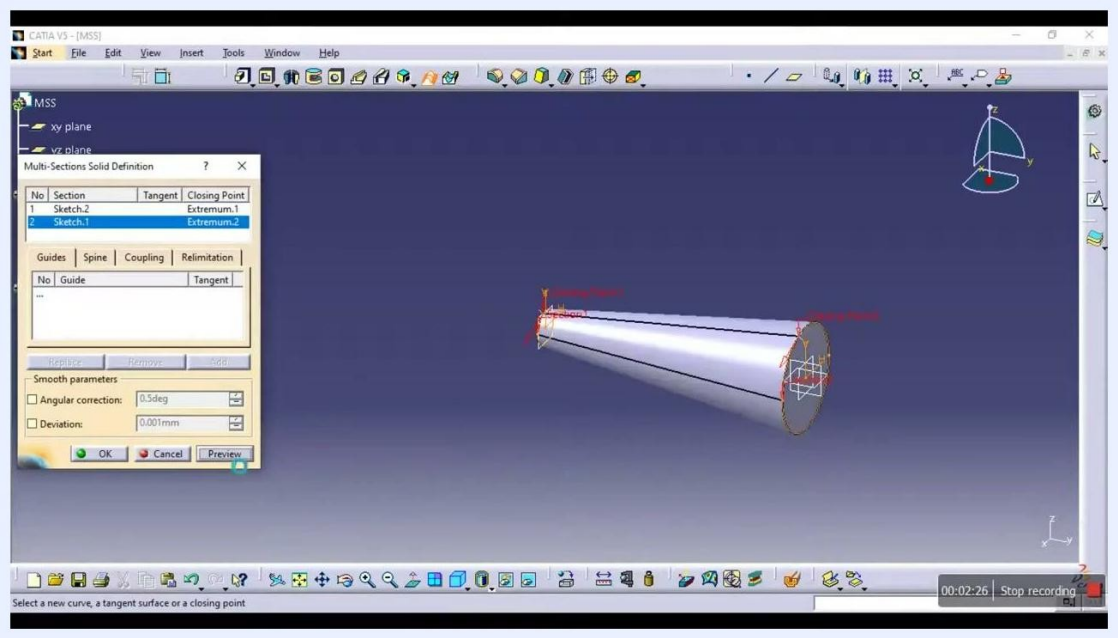
Multi-Sections Solid Definition dialog open with two circles selected as sections



Preview of the multi-section solid showing a tapered shape between two circles

Step 9. Review the generated solid

Rotate and inspect the solid to confirm the transition between the two circular sections. The cross-sectional area changes, but the profile remains circular throughout.



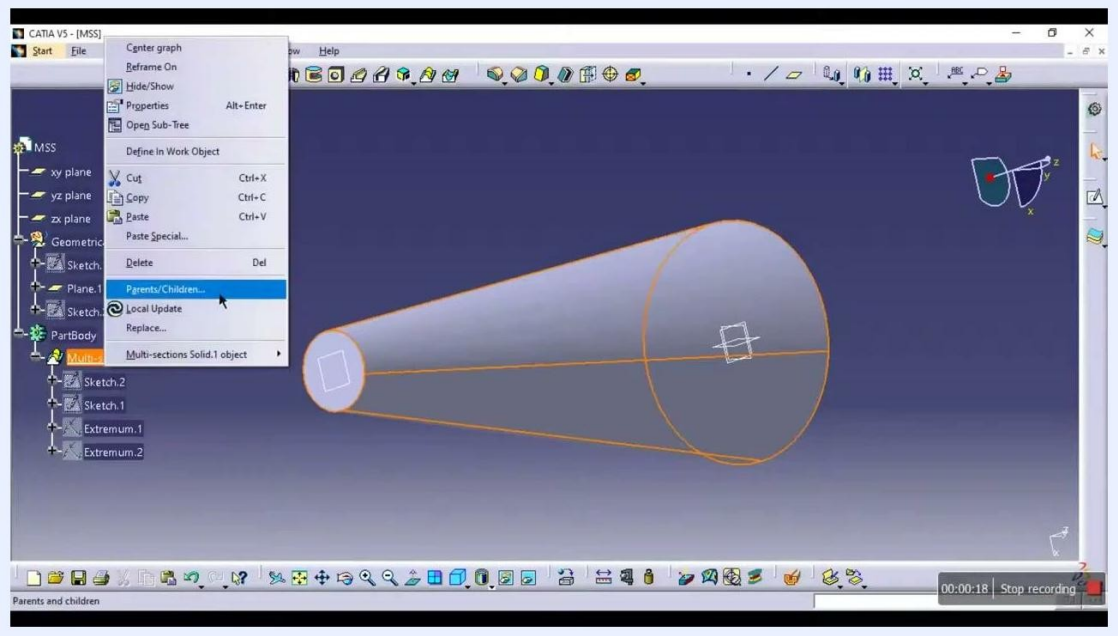
3D view of the multi-section solid showing the transition between two circles

Use different section profiles

Step 10. Delete the first solid

1. In the tree, right-click the **Multi-sections Solid.1** object.
2. Select **Delete**.

3. Confirm the deletion if prompted.

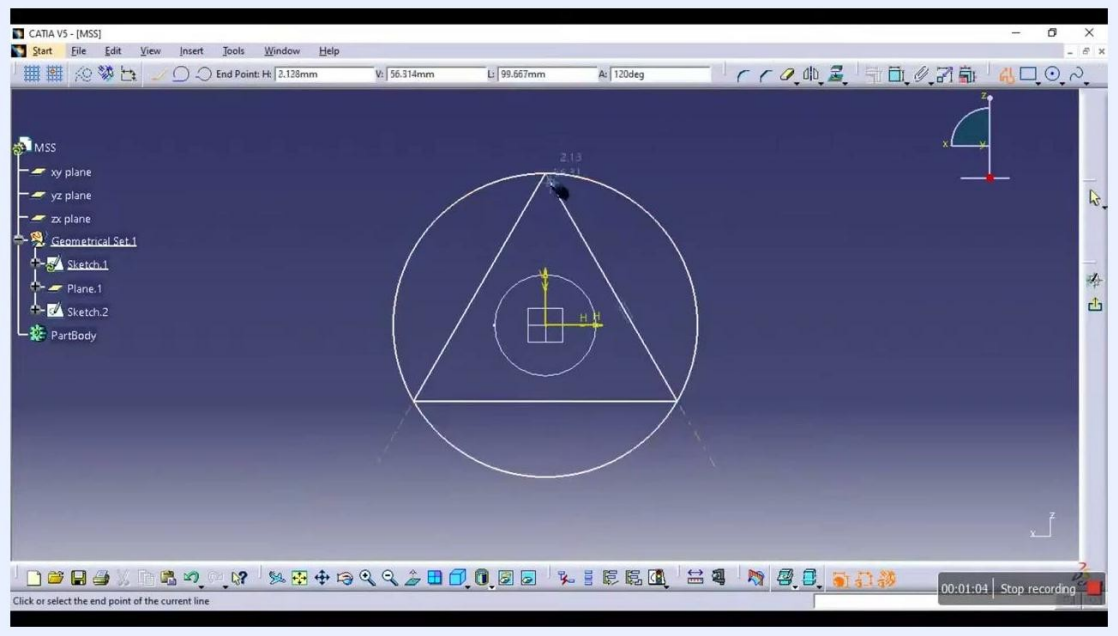


Right-click menu on the Multi-sections Solid.1 object with the Delete option visible

Step 11. Draw a triangle on the base plane

1. Select the **base plane** (the ZX plane or Sketch.1).
2. Enter the Sketcher workbench.
3. Instead of a circle, draw an **equilateral triangle** inside the reference circle.
4. Make sure the triangle is centered and properly dimensioned.

5. Complete the sketch and exit the Sketcher workbench.



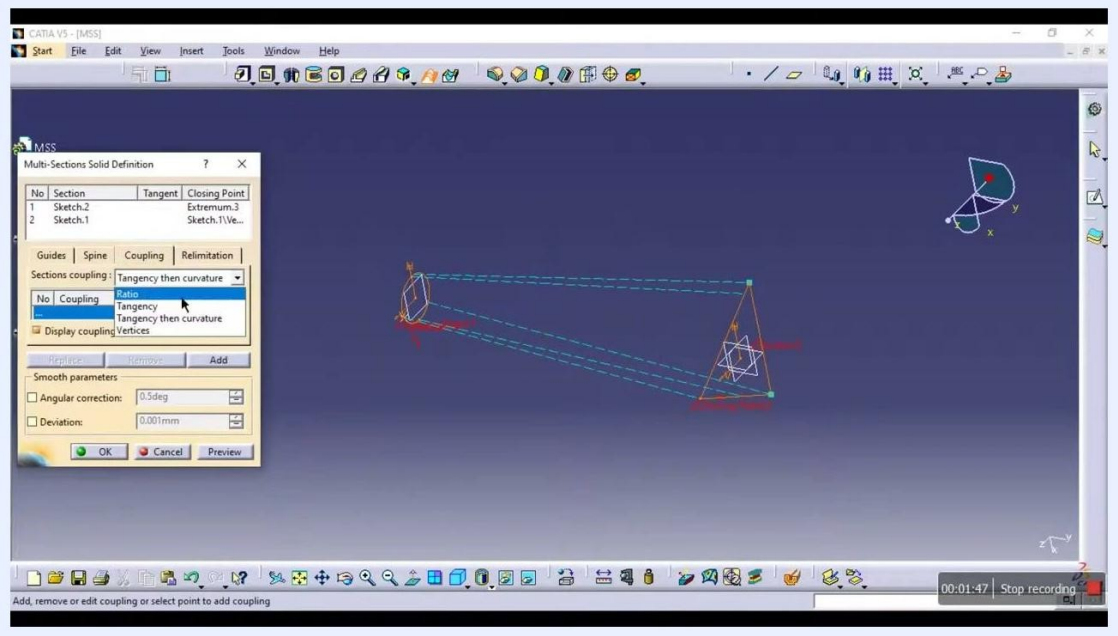
Sketcher workbench active with an equilateral triangle drawn inside a circle on the base plane

You now have sketches for a multi-section solid with different section profiles: a triangle and a circle.

Step 12. Select the circle and triangle as sections

1. Select the **Multi-Section Solid** command from the toolbar.
2. In the dialog, set **Sketch.2** (circle) as the first section.

3. Set **Sketch.1** (triangle) as the second section.



Multi-Sections Solid Definition dialog open with the circle and triangle sketches selected as sections

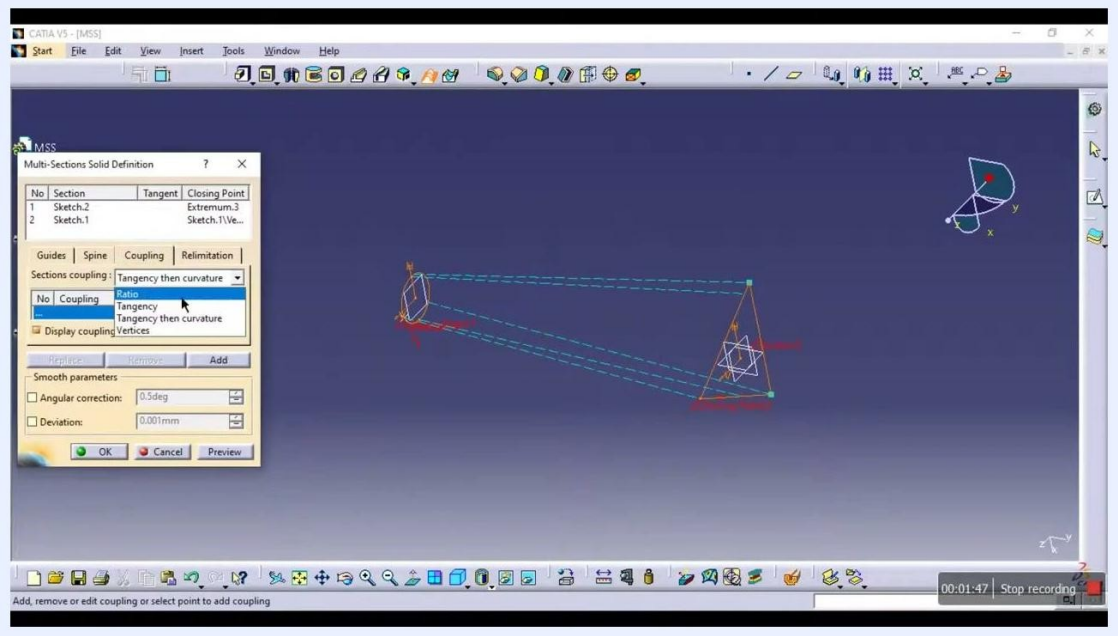
Step 13. Preview the solid and read the error

Click **Preview** in the Multi-Sections Solid Definition dialog. An error appears, indicating that the current coupling mode cannot be applied.

Step 14. Change the coupling mode to Ratio

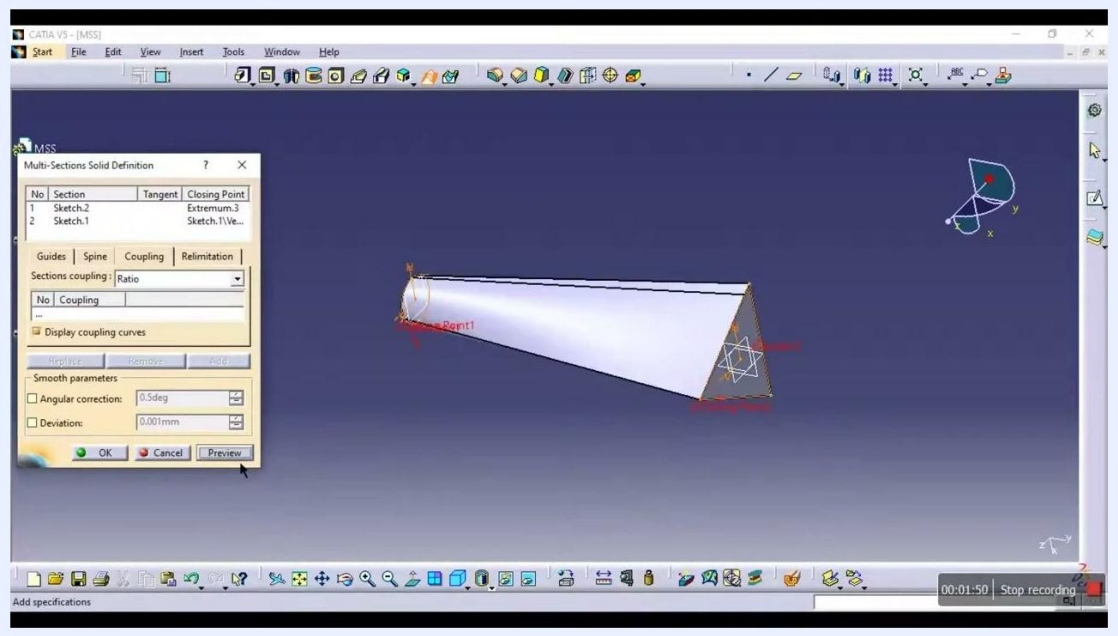
1. Go to the **Coupling** tab in the Multi-Sections Solid Definition dialog.
2. Change the **Sections coupling** option from "Tangency then curvature" to **Ratio**.

3. Click **Preview** again to generate the solid.



Coupling tab open with Ratio selected as the coupling mode between the circle and the triangle

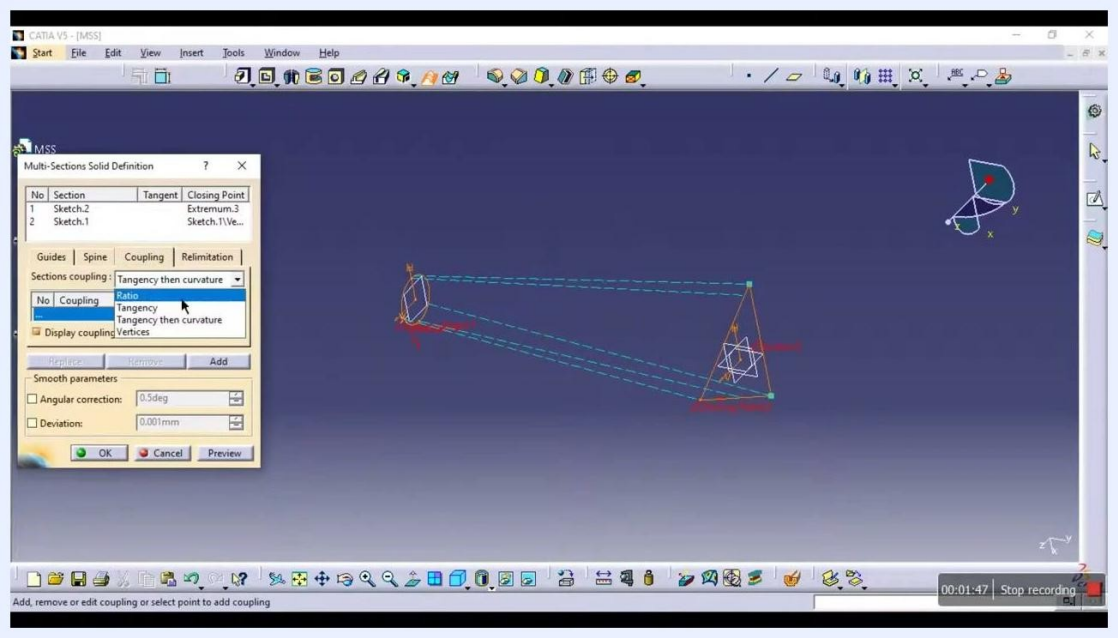
The solid forms and transitions smoothly between the triangle and the circle. The transition creates multiple surfaces and edges, typically four to five, rather than just three.



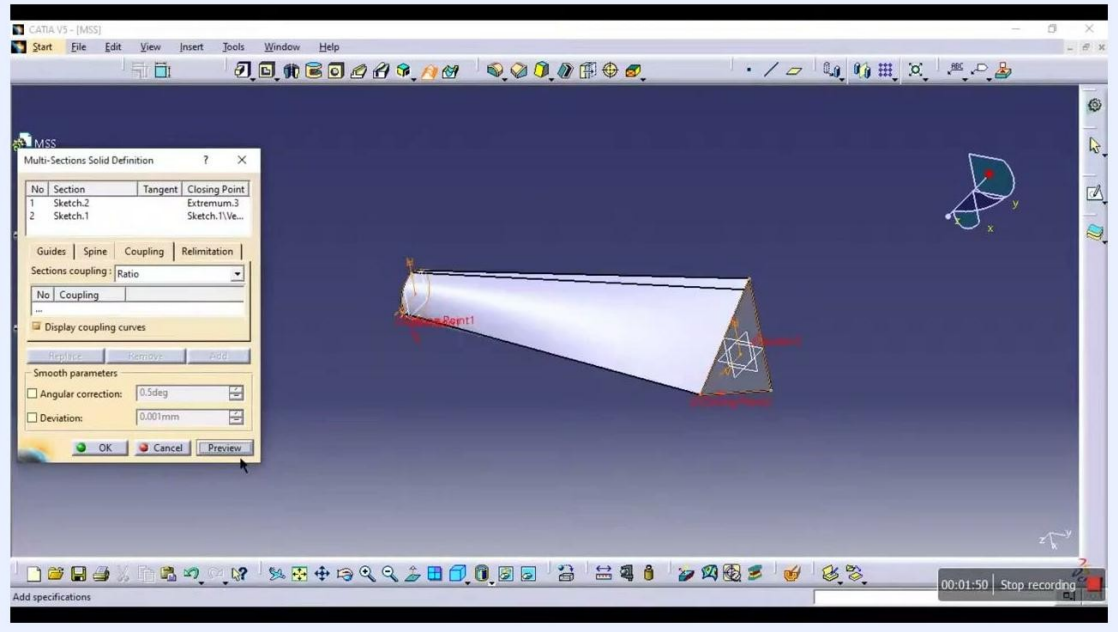
Solid generated with Ratio coupling showing multiple surfaces and edges between the triangle and the circle

Step 15. Inspect the resulting geometry

Rotate and zoom in on the solid to examine the surfaces and edges. The triangle's three edges result in several surfaces connecting to the circle.



Close-up of the multi-section solid highlighting multiple surfaces and edges

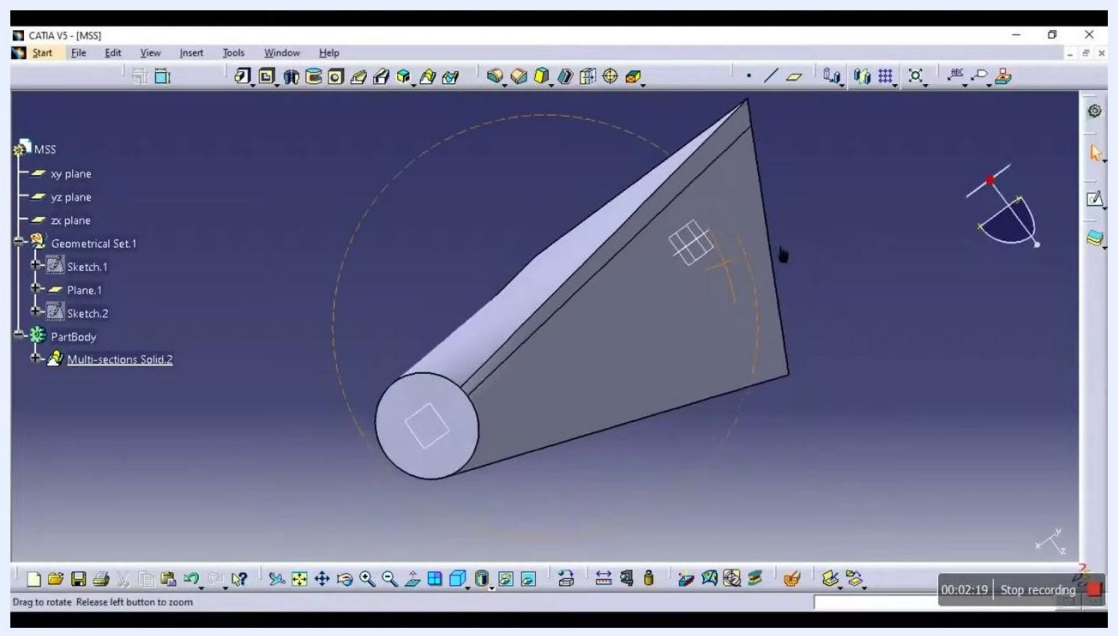


End view of the solid showing the circle and triangle profiles and the connecting surfaces

Get three surfaces with the Vertices coupling mode

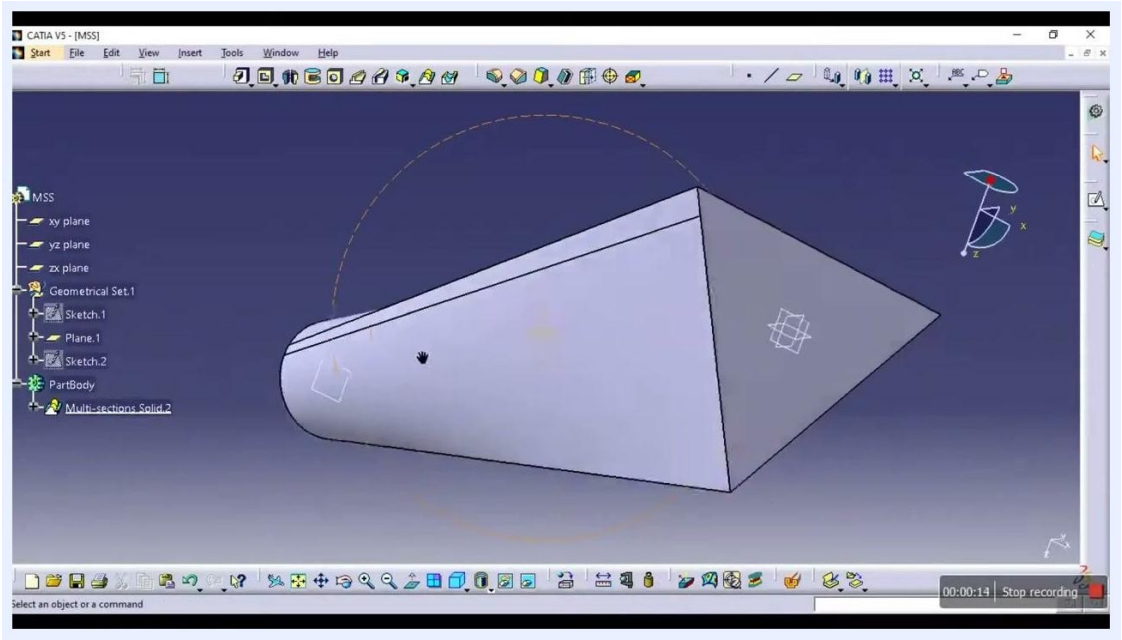
Step 16. Understand why the Vertices coupling mode is needed

To get three connecting surfaces, use the **Vertices** coupling mode. This mode matches each triangle vertex to a specific point on the circle.

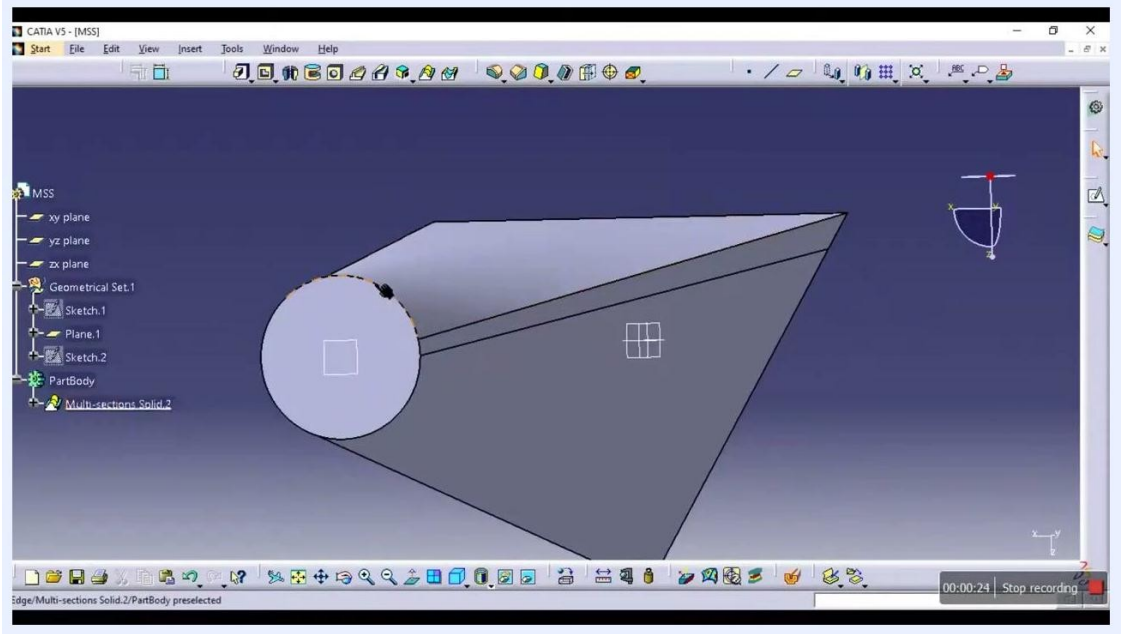


Solid with multiple surfaces, ready for a coupling mode change to get a three-surface result

The triangle has three sides, three edges, and three vertices. The circle has no vertices or edges, which is the root of the coupling issue.



Solid showing the triangle and circle ends, highlighting the difference in vertices and edges

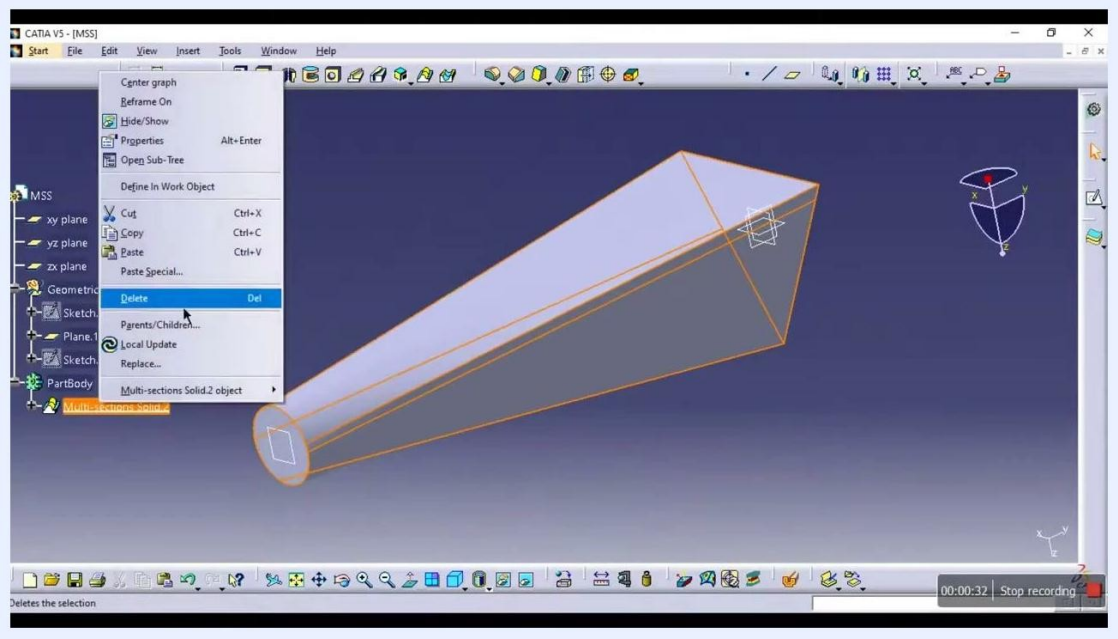


Close-up of the circle end showing no vertices or edges

Step 17. Divide the circle into three equal parts

1. Create or select a sketch that contains a circle.

2. Draw lines from the center of the circle to its circumference, spaced 120° apart, to divide the circle into three equal sectors.
3. Make sure the lines are placed accurately for an equal division.

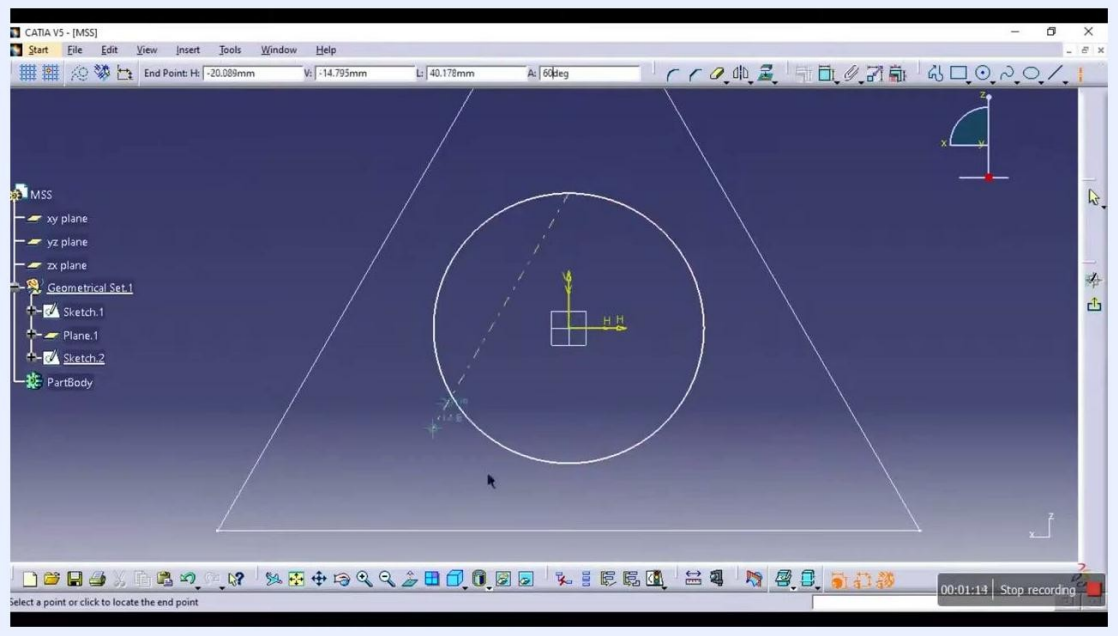


CATIA V5 sketcher showing a triangle inscribed in a circle, with construction lines dividing the circle into three equal parts and Sketch.2 highlighted in the part tree

Step 18. Trim the circle

1. Go to the **Trim** command in the Sketcher workbench.
2. Select "Trim and Keep" to remove unnecessary segments while keeping the required geometry.

3. Trim the circle and lines so that only the three equal sectors remain.



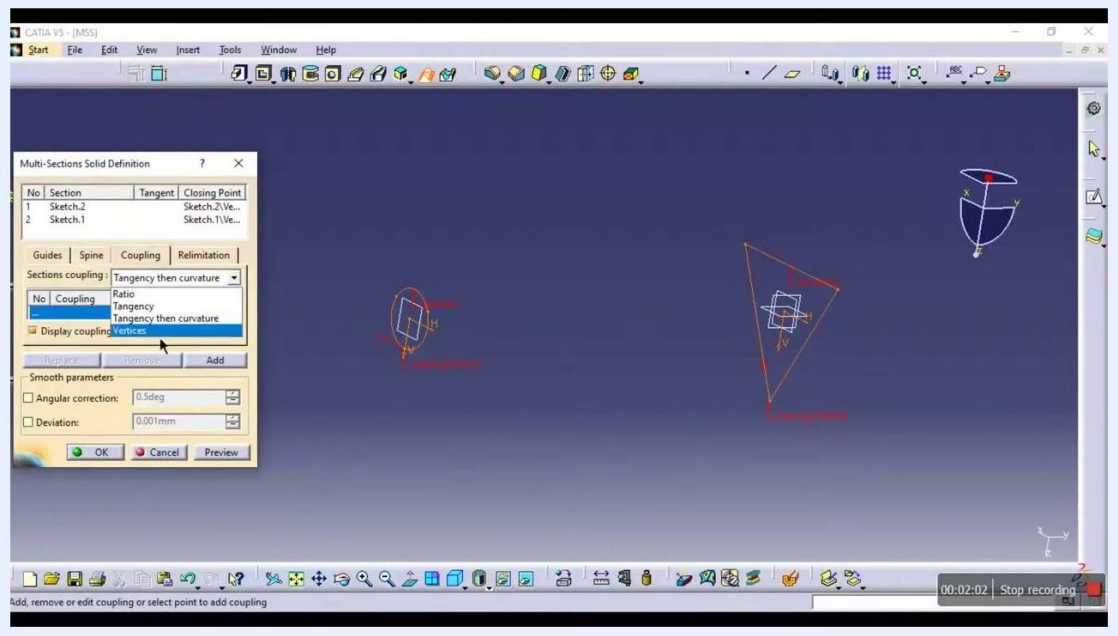
CATIA V5 sketcher with a circle and triangle showing the trim operation in progress, with Sketch.2 highlighted in the part tree

Step 19. Set the Vertices coupling in the Multi-Sections Solid Definition dialog

Make sure you have at least two sketches on parallel planes. One holds the divided circle (for example, Sketch.2). The other holds a different profile (for example, the triangle in Sketch.1).

1. Activate the **Multi-Sections Solid** tool.
2. In the dialog, select the first section (for example, Sketch.2) and the second section (for example, Sketch.1).
3. Go to the **Coupling** tab.
4. Set **Sections coupling** to "Vertices" for precise vertex-to-vertex mapping.

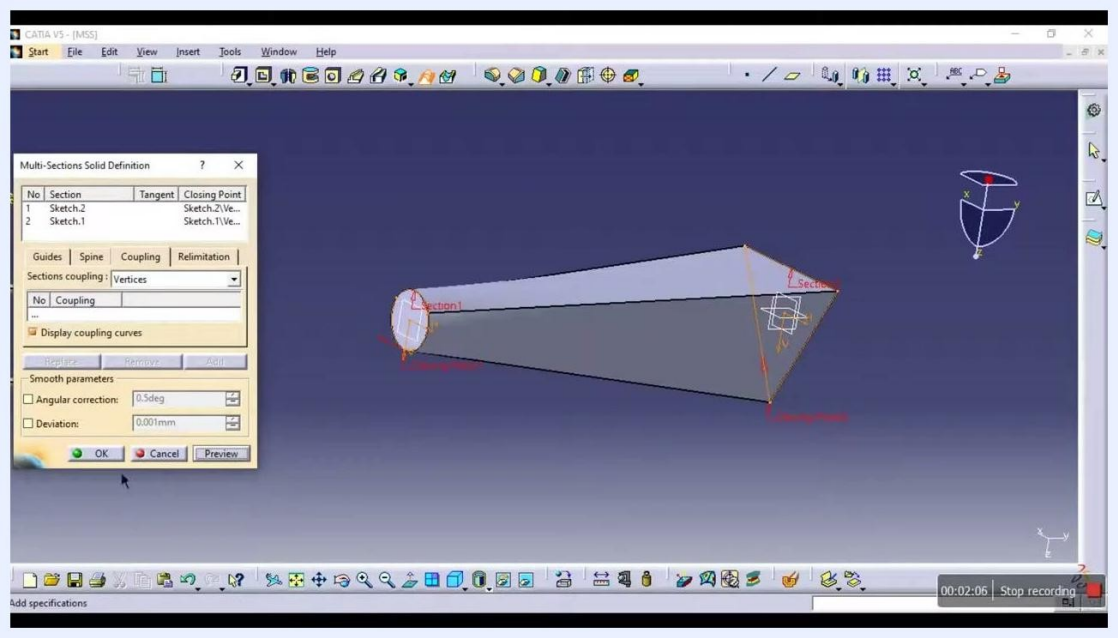
5. Click **Preview** to visualize the resulting solid.



Multi-Sections Solid Definition dialog open with Sections coupling set to Vertices and the circle and triangle sketches selected as sections

Step 20. Confirm and create the solid

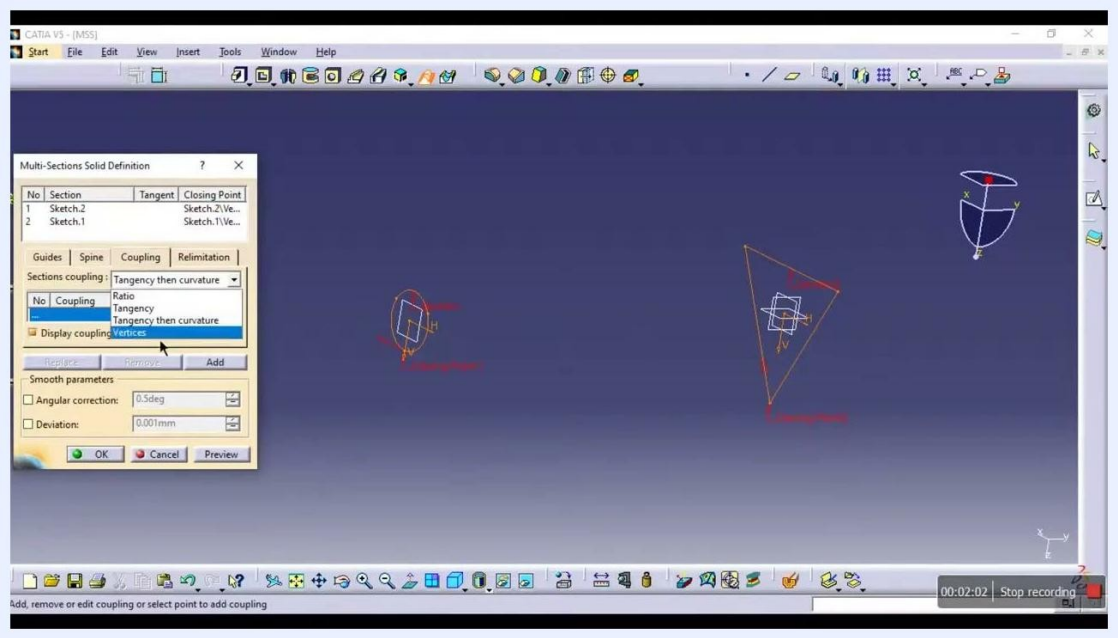
If the preview is correct, click **OK** to generate the multi-section solid. The solid transitions smoothly from the circle to the triangle.



The completed multi-section solid transitioning from a circular to a triangular profile, with the Multi-Sections Solid Definition dialog visible

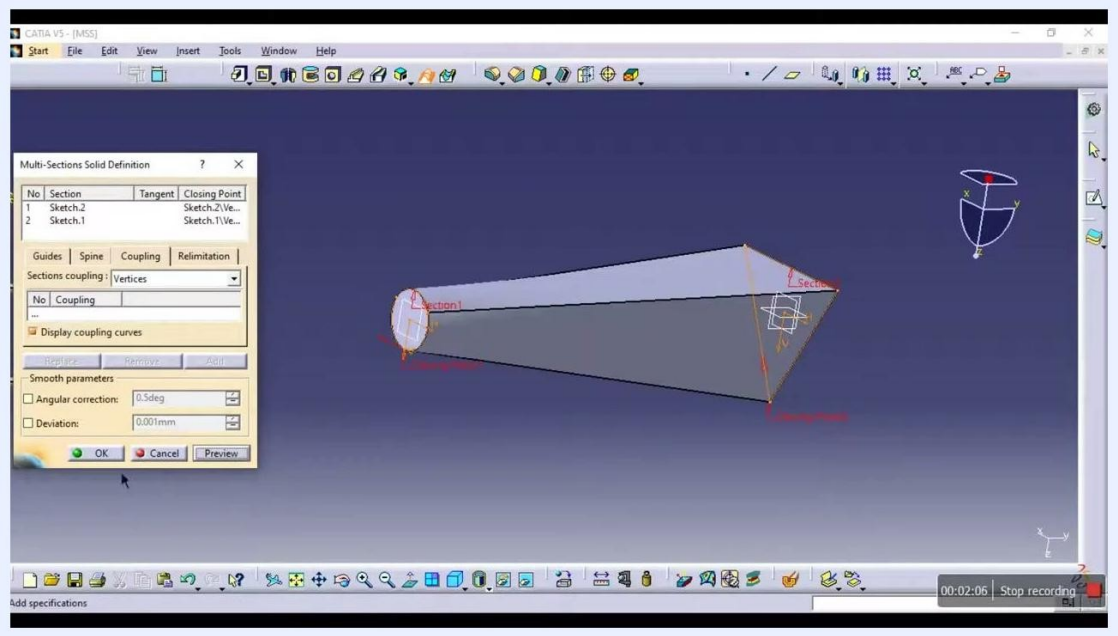
Step 21. Experiment with different end profiles

To create a solid that transitions from a circle to a hexagon, repeat the same steps with a hexagon sketch at one end. Divide the circle into six equal parts (60° apart) for accurate vertex mapping.



A solid transitioning from a circle to a triangle, viewed from the triangle end, with the part tree and 3D model visible

The resulting solid transitions smoothly between the two profiles, for example circle to triangle or circle to hexagon. The process is the same each time. Only the number of divisions and the end profile change.



A solid transitioning from a circle to a polygonal profile, viewed from an angle, with the part tree and 3D model visible

Note:

- Make sure the number of divisions in the circle matches the number of vertices in the end profile for proper coupling.
- Use the "Vertices" coupling mode for precise mapping between corresponding points on each profile.

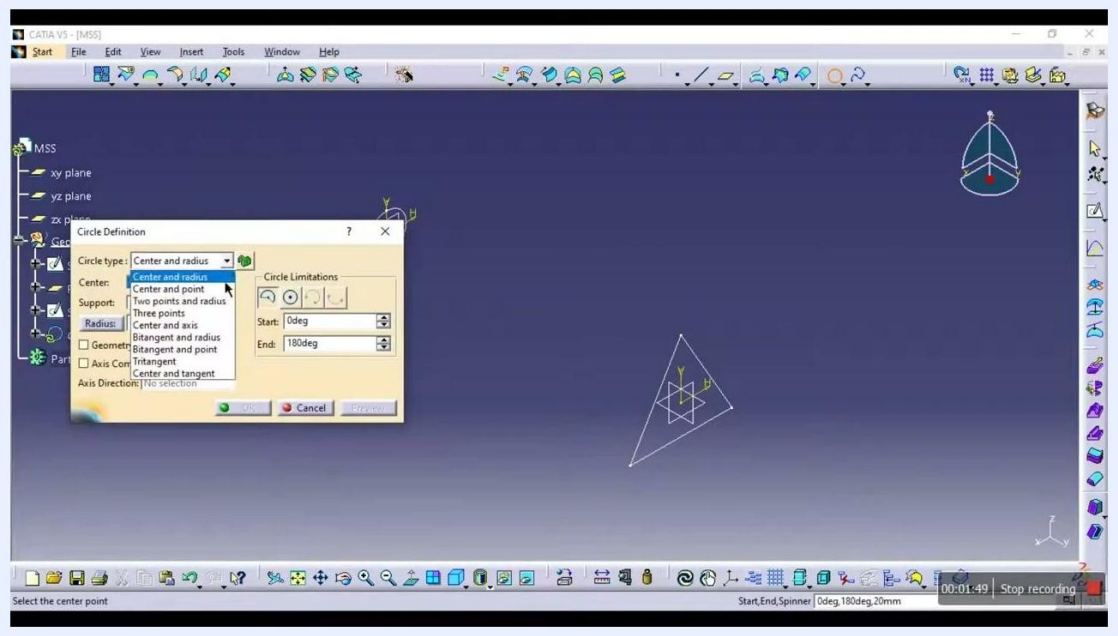
Follow a curved path with a spine

You can also use multi-section solid in CATIA V5 to build a solid that follows a curved path (a spine) rather than a straight line.

Step 22. Start the spine curve

1. Open the **Circle Definition** dialog to create a guiding arc (the spine) between the two section profiles.

2. Select "Center and radius" as the circle type. The dialog offers several circle creation methods.

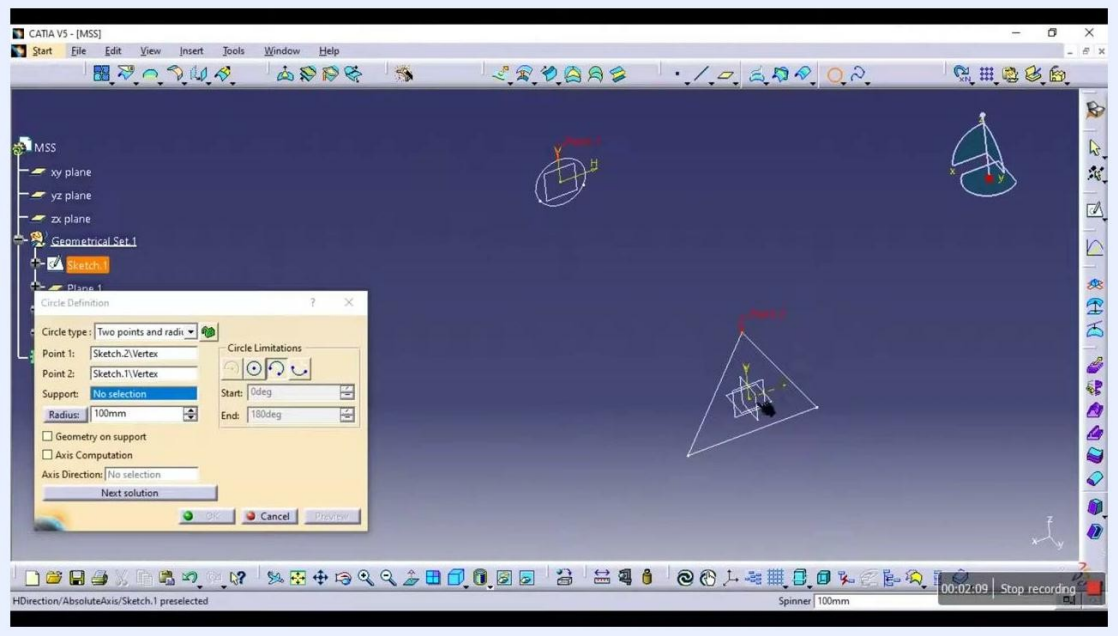


Circle Definition dialog open with Center and radius selected and the two sketches visible in the background

Step 23. Select the points for the spine

1. Change the circle type to "Two points and radius".
2. For **Point 1**, select a vertex on the circle sketch (Sketch.2\Vertex).
3. For **Point 2**, select a corresponding vertex on the triangle sketch (Sketch.1\Vertex).

The dialog now shows both points selected, ready to define the arc.

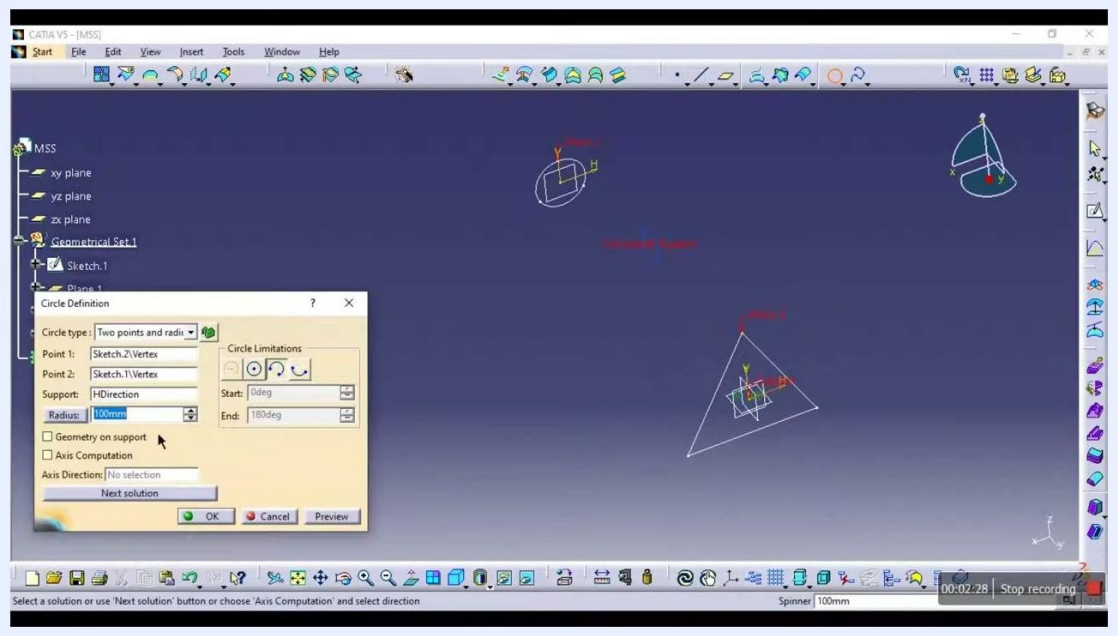


Circle Definition dialog with Two points and radius selected and both points chosen from the sketches

Step 24. Set the support and adjust the radius

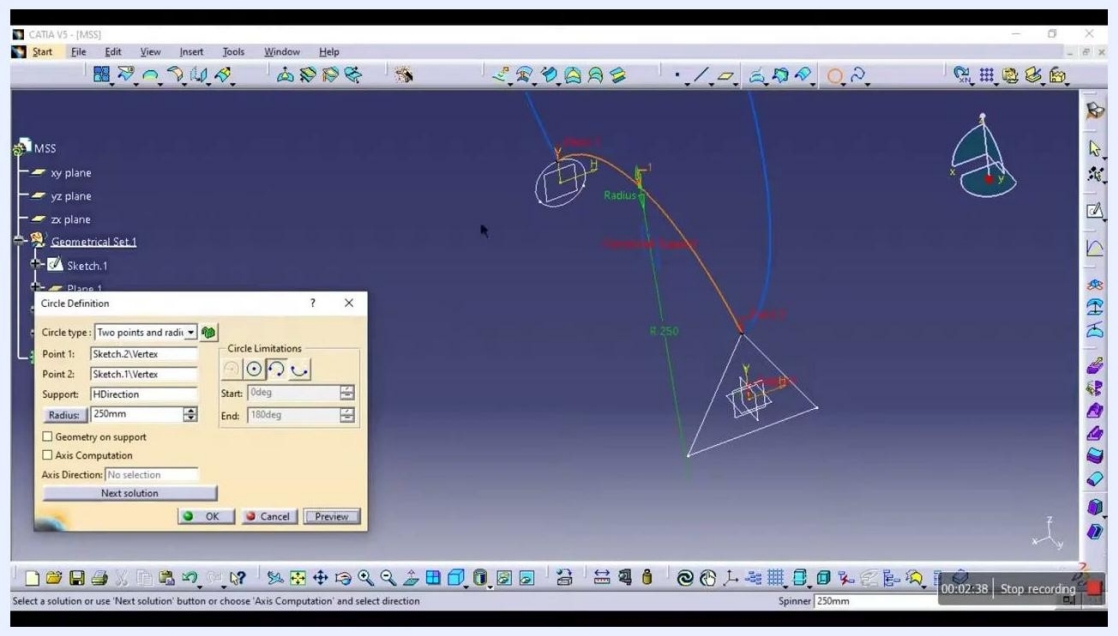
1. Set **Support** to HDirection (horizontal direction).
2. Enter a radius value, for example 100mm.
3. If an error says the radius is too short, increase the radius value.

The dialog and workspace display the selected points and support, with error messages if the radius is insufficient.



Circle Definition dialog with support set to HDirection and a radius of 100mm, with an error message possibly visible

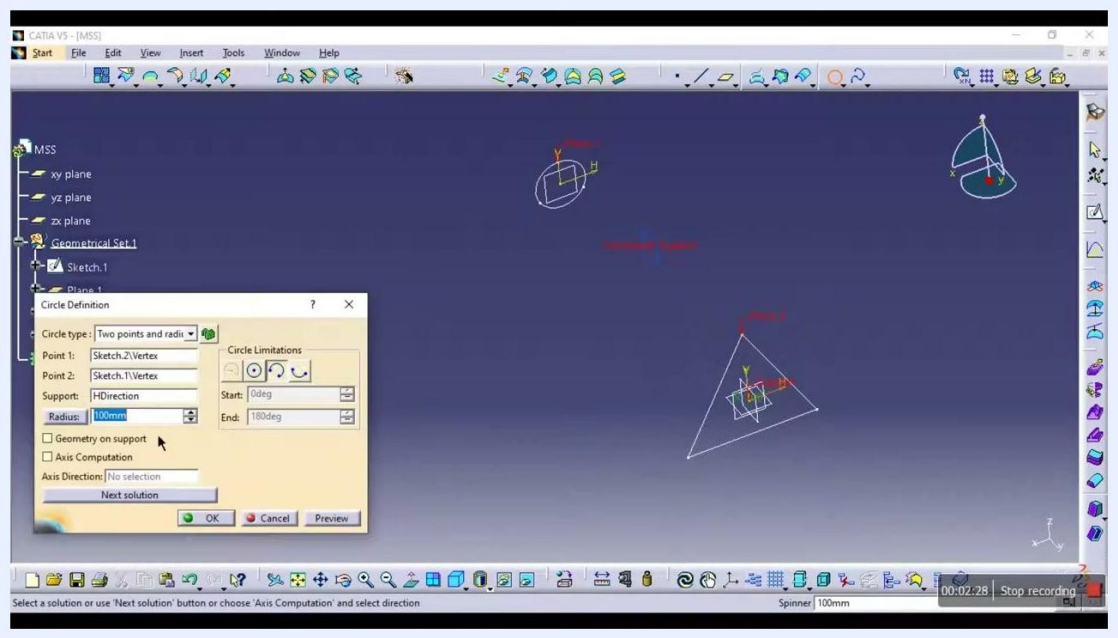
Increase the radius (for example, to 250mm) until the arc is created between the two points. The arc (spine) appears in the workspace, connecting the circle and triangle sketches. The dialog shows the updated radius, and the arc is labeled with its radius and endpoints.



Circle Definition dialog with the radius set to 250mm and the arc (spine) visible connecting the two sketches

Step 25. Switch back to the Part Design workbench

Go to **Start** > **Mechanical Design** > **Part Design**. The workspace shows the two sketches and the new arc (spine), ready for multi-section solid creation.



CATIA V5 menu open showing navigation to Mechanical Design > Part Design, with the arc and sketches visible in the workspace

Note:

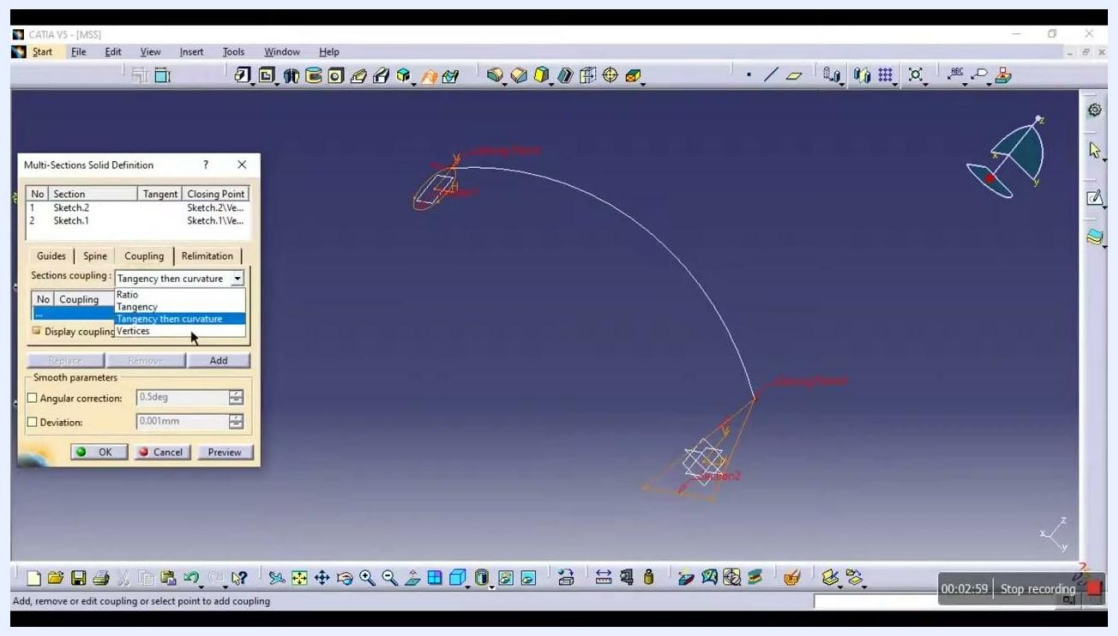
- The spine (arc) guides the transition between the two section profiles, so the solid is curved rather than straight.
- Make sure the arc is properly defined and connects the correct vertices for accurate geometry.

Step 26. Divide the section profiles and set the coupling

Make sure both section profiles (the circle and the triangle) are divided into three equal parts. This division is necessary for accurate coupling between the profiles.

1. Open the **Multi-Sections Solid Definition** dialog and select the **Coupling** tab.
2. In the **Sections coupling** dropdown, select "Tangency then curvature" to control how the sections are blended.

The workspace shows the two section sketches at each end of the arc, with labeled points indicating the divisions.

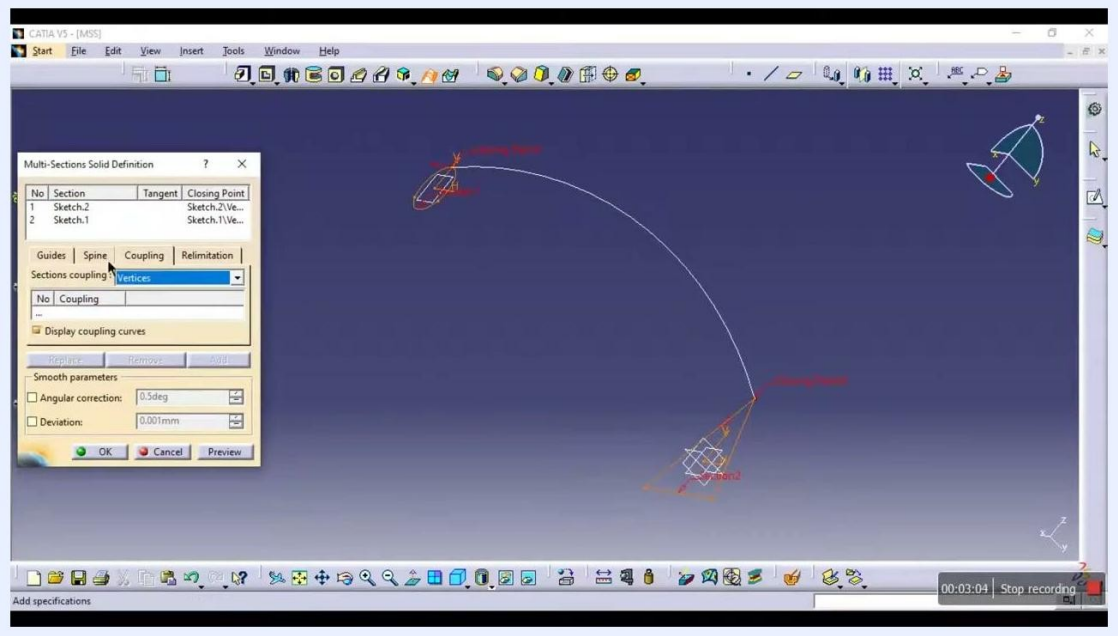


Multi-Sections Solid Definition dialog open showing coupling options and divided section profiles along the arc

Step 27. Select the spine curve

1. Switch to the **Spine** tab in the Multi-Sections Solid Definition dialog.
2. In the **Spine** field, select the arc (for example, Circle.1) as the guiding curve.

3. Click **Preview** to visualize the resulting solid.

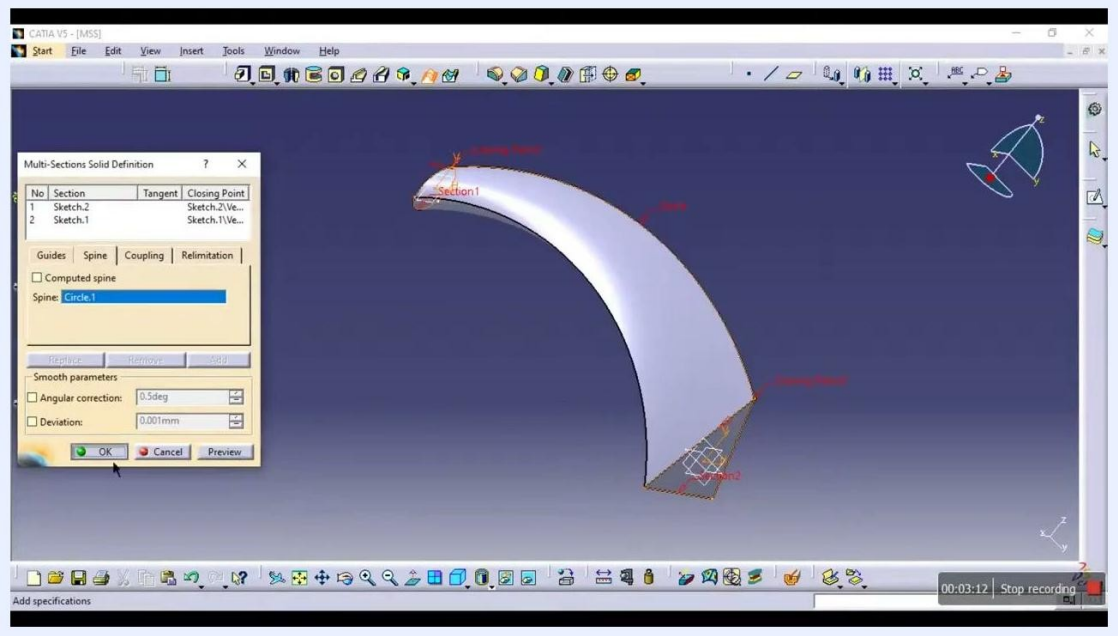


Multi-Sections Solid Definition dialog with the Spine tab active and the arc selected as the spine

Step 28. Preview and confirm the curved solid

After you click **Preview**, the solid appears in the workspace. It transitions from a triangle at one end to a circle at the other, following the curved spine. The dialog stays open and shows the computed spine and smooth parameter settings.

Click **OK** to finalize the curved multi-section solid.



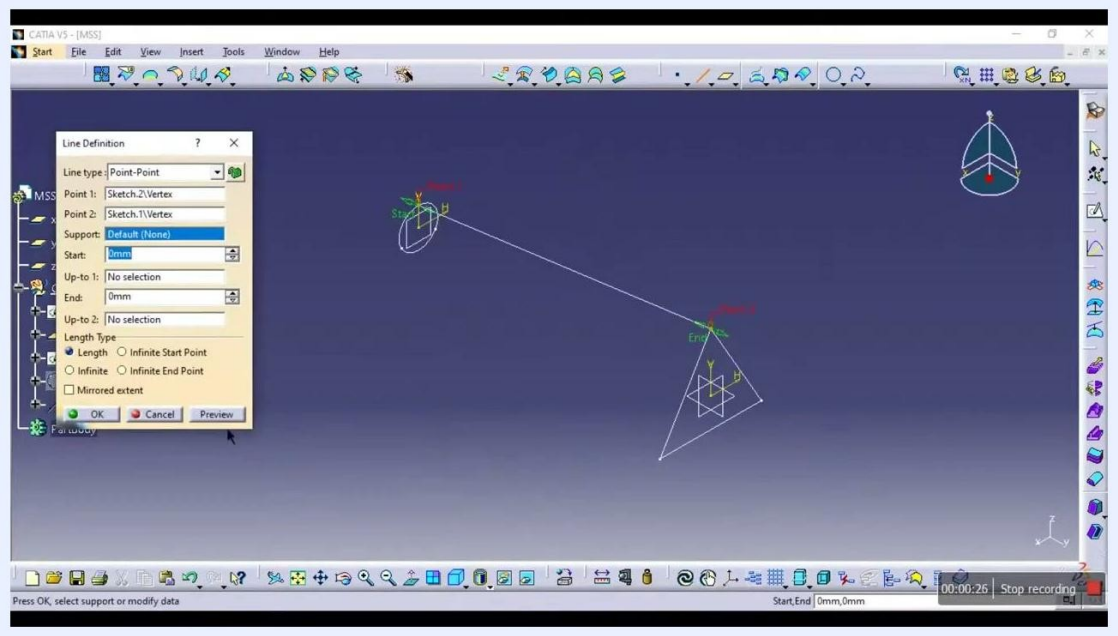
The completed curved multi-section solid with the Multi-Sections Solid Definition dialog confirming the spine selection

Add guide curves and lines

Step 29. Delete the existing multi-section solid

1. In the part tree, right-click Multi-sections Solid.5.
2. In the context menu, select **Delete** to remove the current solid.

The solid is highlighted in orange, and the context menu shows the **Delete** option.

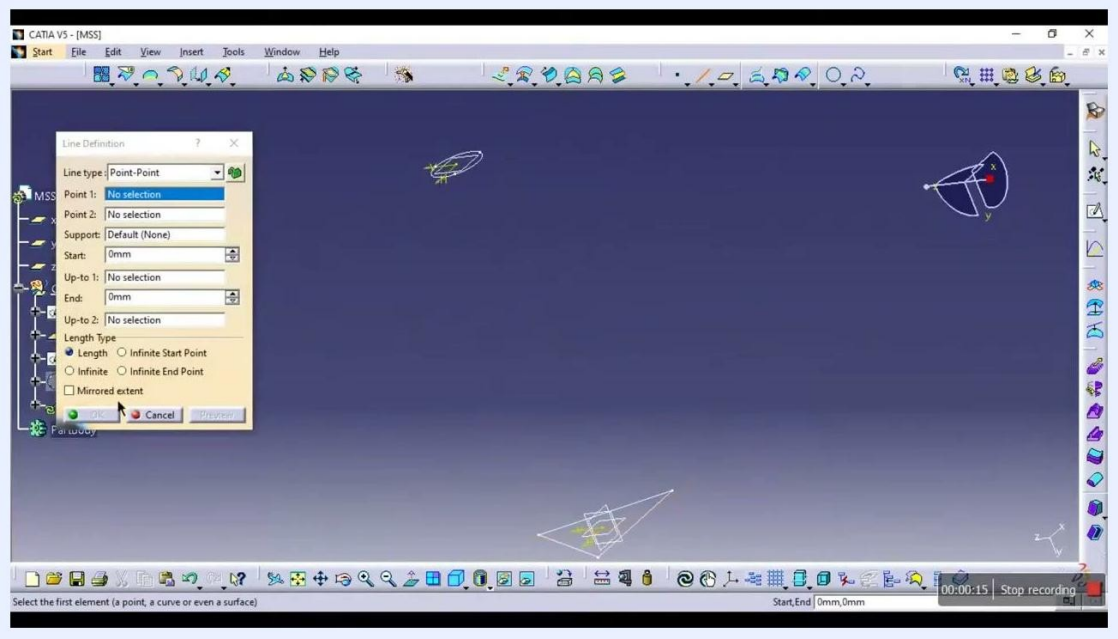


Multi-sections Solid.5 highlighted with the context menu open, ready to delete the solid

Step 30. Start a guide line between the section profiles

1. Open the **Line Definition** dialog to create a new guide line.
2. Set **Line type** to "Point-Point".

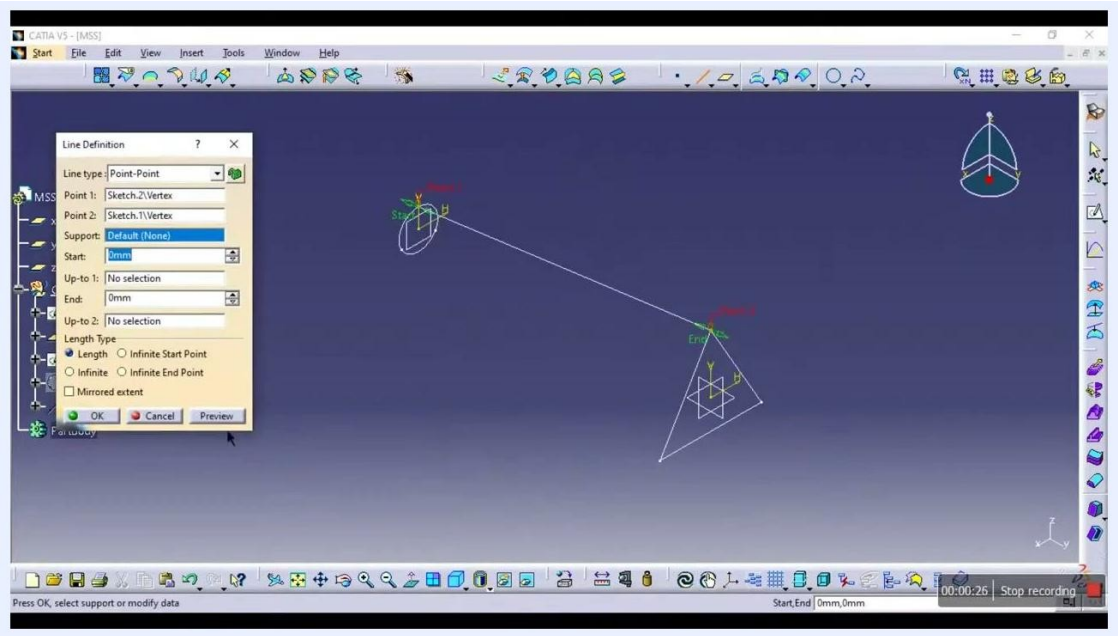
The dialog is ready for you to select the start and end points. The workspace shows the two section profiles (circle and triangle) with no solid present.



Line Definition dialog open, ready to select points for a new guide line

Step 31. Select the line end points

In the **Line Definition** dialog, select Sketch.2\Vertex as **Point 1** and Sketch.1\Vertex as **Point 2**. A straight line connects the selected vertices of the circle and triangle sketches. The dialog confirms the selected points and the default support.



Line Definition dialog showing a straight line between the circle and triangle vertices

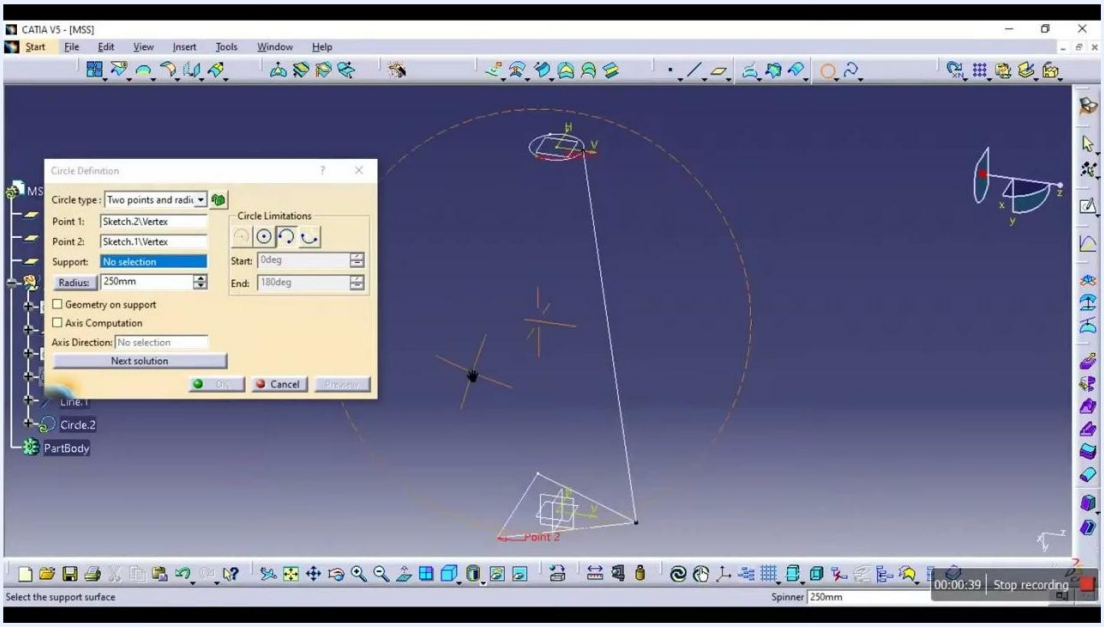
Note:

- Dividing the profiles into equal parts and using the correct coupling method ensures a smooth transition between sections.
- The spine curve determines the overall shape of the solid. Guide curves and lines give you additional control for advanced shapes.
- You can mix straight and curved sections by combining lines and curves as guides.

Step 32. Define a guide circle

1. In CATIA V5, make sure you are in the appropriate workbench for sketching or shape design.
2. Open the **Circle Definition** dialog.
3. Set **Circle type** to "Two points and radius".
4. For **Point 1**, select Sketch.2\Vertex. For **Point 2**, select Sketch.1\Vertex.
5. Leave **Support** as "No selection" for now.

6. Enter 250mm for the **Radius**.
7. Under **Circle Limitations**, set **Start** to 0deg and **End** to 180deg.
8. Do not select "Geometry on support" or "Axis Computation" at this stage. Leave **Axis Direction** as "No selection".
9. Click **Next solution** if needed to preview different circle placements.

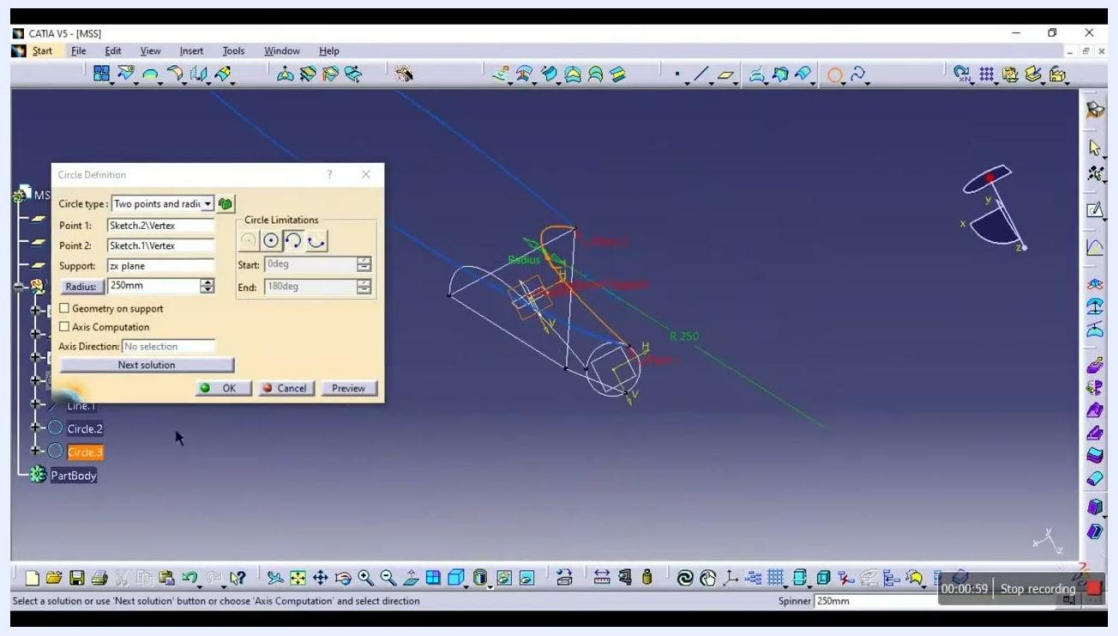


Circle Definition dialog open with parameters set, showing a large construction circle and guide geometry in the CATIA workspace

Step 33. Assign the support plane for the circle

1. In the **Circle Definition** dialog, set **Support** to the zx plane (or the appropriate plane for your geometry).
2. Make sure **Point 1** and **Point 2** remain as previously set.
3. Confirm the **Radius** is still 250mm.

4. Click **Preview** to visualize the circle in the 3D workspace.

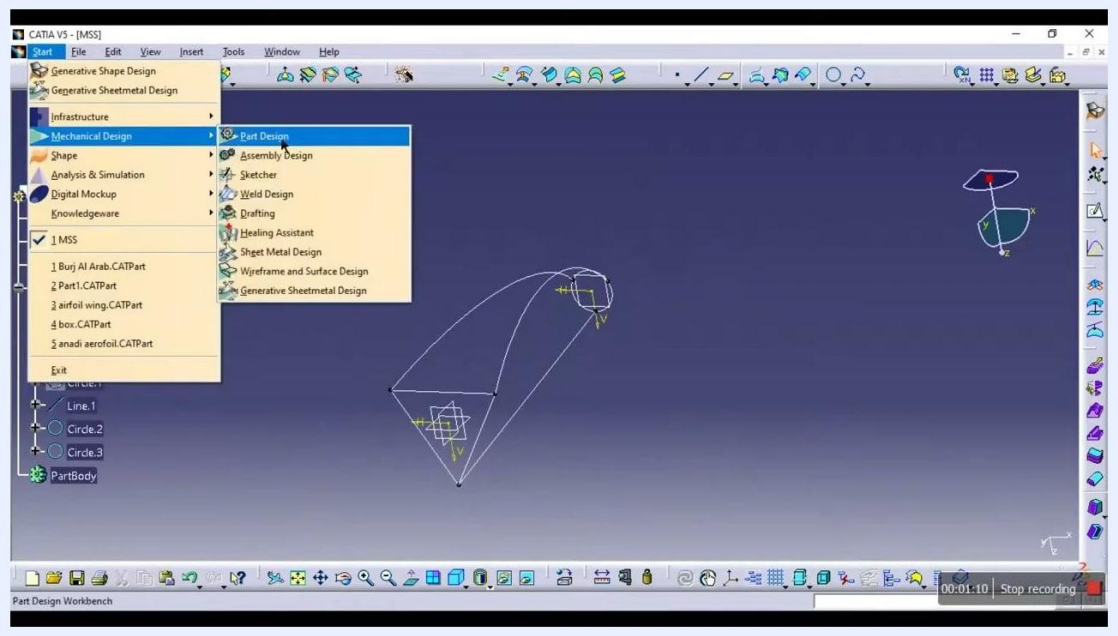


Circle Definition dialog with the support plane selected, showing the circle and guide geometry in 3D

Step 34. Return to the Part Design workbench

1. Click the **Start** menu in the top left.
2. Go to **Mechanical Design** > **Part Design** to switch workbenches.

This prepares the environment for solid modeling based on your sketches and guides.



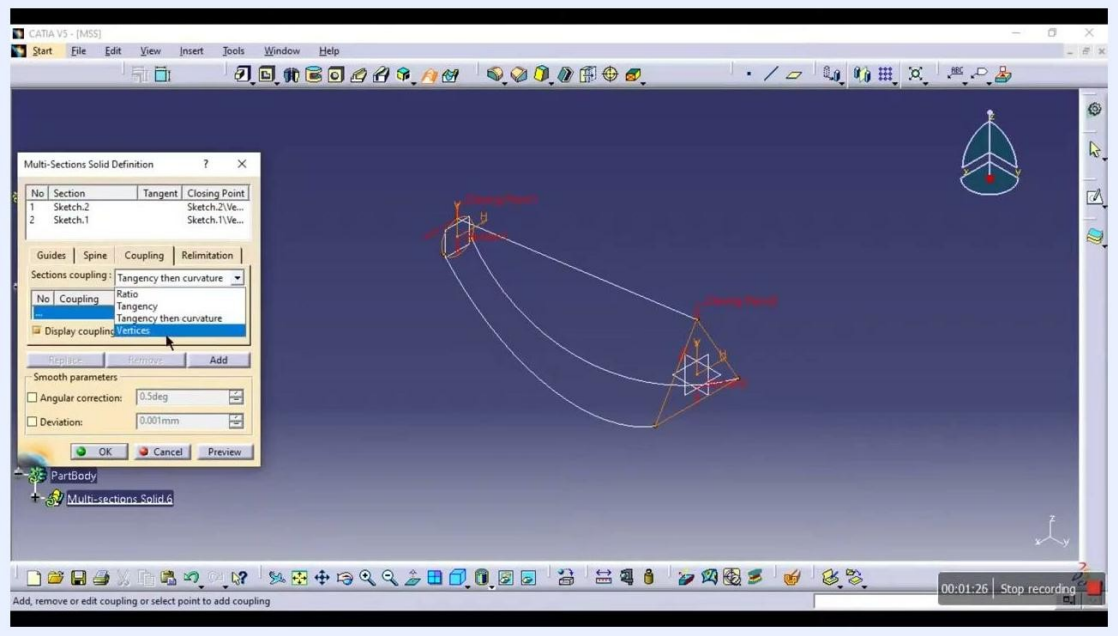
CATIA V5 Start menu open, highlighting navigation to Mechanical Design > Part Design

Create the multi-section solid with guides

Step 35. Define sections, guides, and coupling

1. Open the **Multi-Sections Solid Definition** dialog.
2. On the **Sections** tab, select the first section (Sketch.2) and the second section (Sketch.1).
3. Go to the **Guides** tab. Select Line.1 (the straight line) as the first guide and the relevant curve as the second guide.
4. On the **Coupling** tab, set **Sections coupling** to Tangency then curvature for smooth transitions.
5. Optionally, set **Angular correction** to 0.5deg and **Deviation** to 0.001mm for finer control.

6. Click **Preview** to generate a visual preview of the solid.

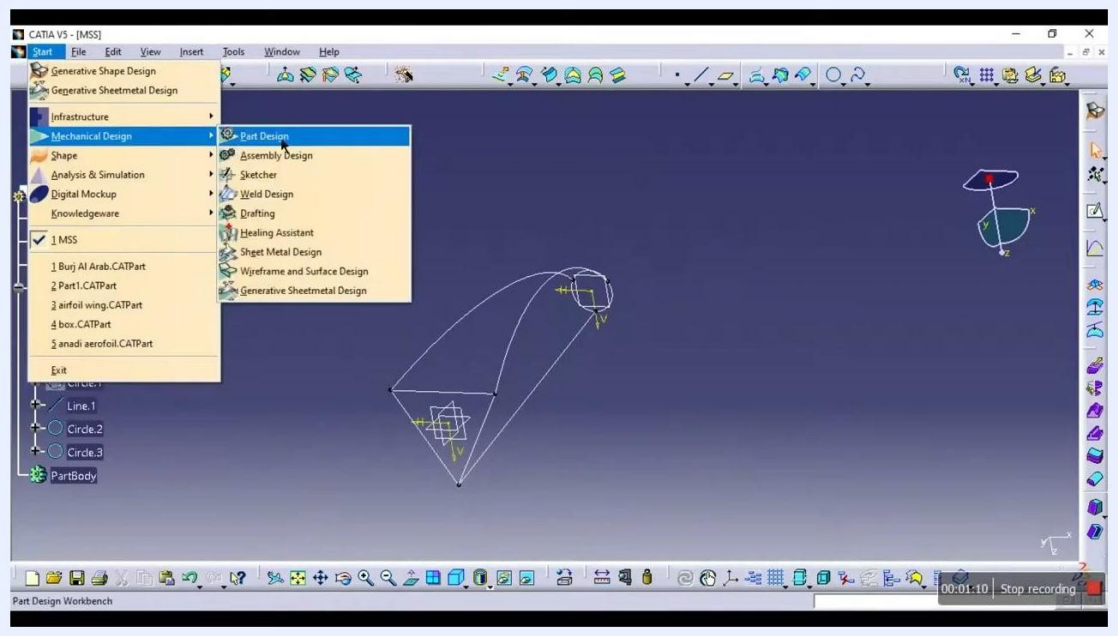


Multi-Sections Solid Definition dialog open with guides and coupling set and a preview of the solid visible

Step 36. Finalize the multi-section solid

1. Review the preview of the generated solid in the workspace.
2. If you are satisfied, click **OK** in the Multi-Sections Solid Definition dialog.

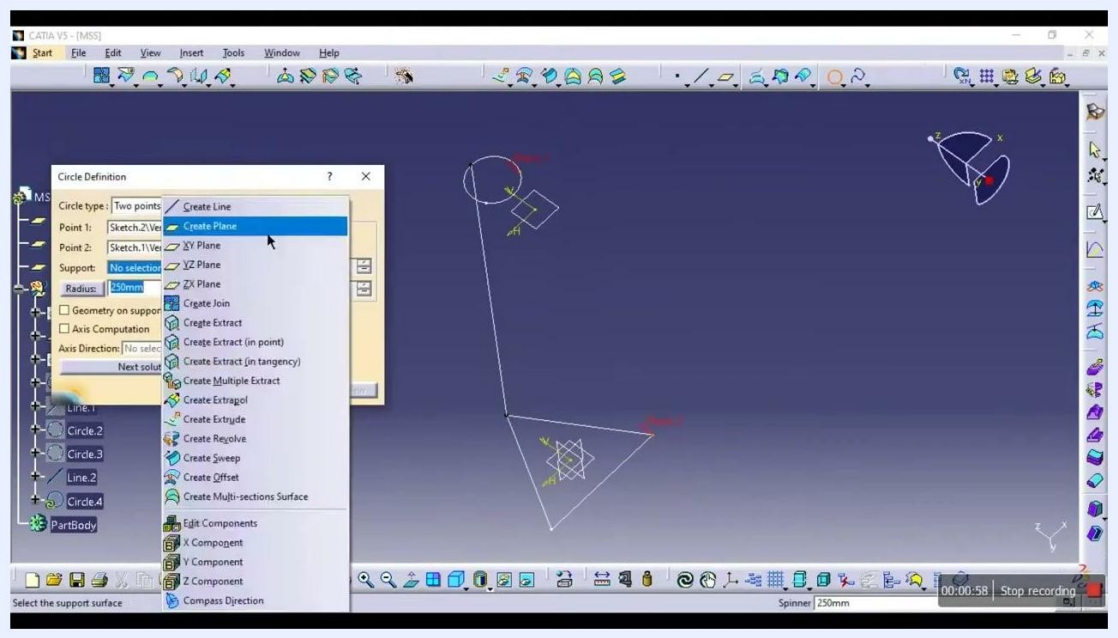
The new solid appears in the model tree as Multi-sections Solid.6 under **PartBody**.



Preview of the completed multi-section solid with guides and sections labeled in the workspace

Step 37. Inspect and rotate the solid

Use the mouse to rotate the new solid and inspect it from different angles. Confirm that the solid follows the intended guide curves and section profiles.



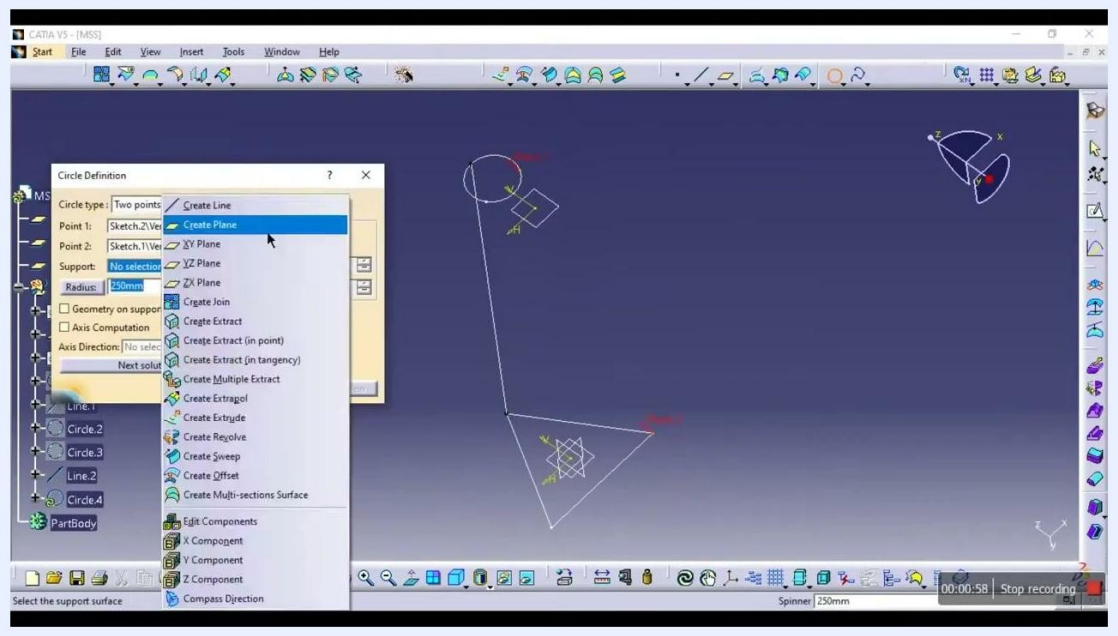
Solid model shown from a different angle with the model tree visible on the left

Step 38. Modify the guide curves for creative design

To experiment with the design, adjust the guide circle:

1. Open the **Circle Definition** dialog again.
2. Select "Create Plane" from the support options if you want to shift the circle or guide to a new location.

3. Adjust the position of the circle, for example by moving it closer to the top vertex of the section.

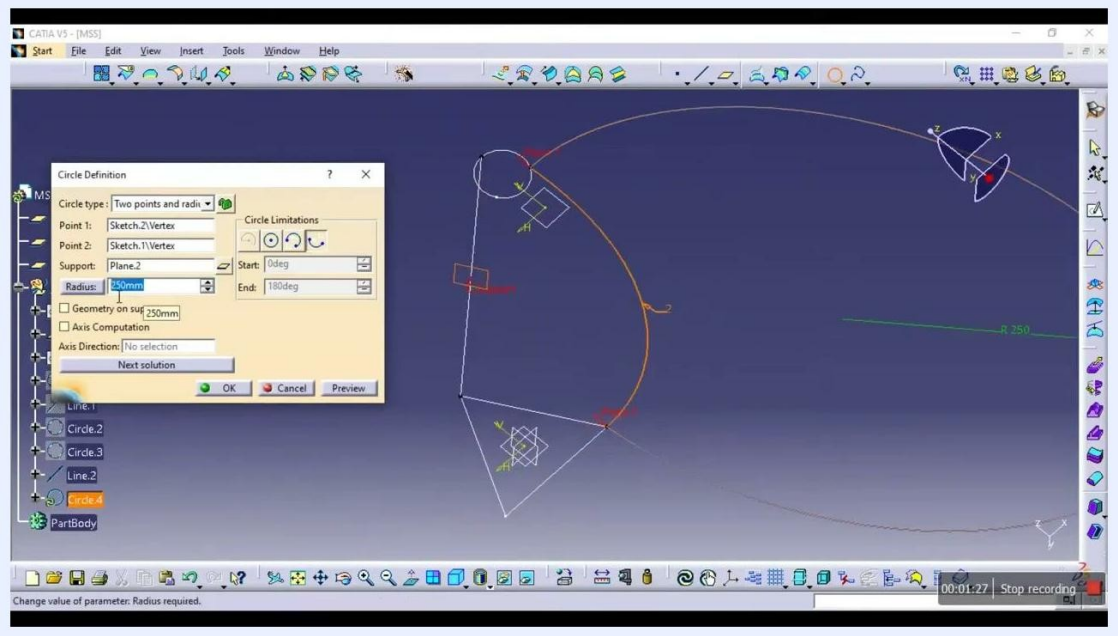


Circle Definition dialog open with support options expanded and Create Plane highlighted

Step 39. Preview and confirm the adjustments

1. Preview the changes in the workspace after you modify the guide curve or circle position.
2. Make sure the new guide placement achieves the effect you want on the solid's shape.

3. Click **OK** to confirm and update the model.



Circle Definition dialog with updated support and geometry showing the new guide curve in the workspace

Review the finished solid

Step 40. Compare the 3D model with the reference image

The CATIA V5 workspace shows your completed multi-section solid on the left and a reference image of a real-world structure (a sail-shaped hotel) on the right. The model tree lists the sketches, planes, circles, and lines used in the design.

Compare the shape and curvature of your 3D model with the reference image. Confirm that the solid closely matches the intended architectural form.



The completed multi-section solid on the left and a reference image of a sail-shaped hotel on the right, with the model tree listing the sketches, planes, circles, and lines used in the design

Step 41. End the modeling session

Save your work if you have not already done so, then conclude your CATIA V5 session.

What's next

Try other end profiles, such as a hexagon. Divide the circle into the same number of parts as the profile has vertices, and use the **Vertices** coupling mode. You can also change the guide circle position to explore different shapes.

Summary

In this guide, you learned how to use multi-section solid in CATIA V5. You built a solid between two circles, then switched to a triangle and a circle. You used the **Ratio** and **Vertices** coupling modes to control the surfaces. You then added a spine curve and guide curves to shape a curved solid and compared it with a reference image.