How to create parameters in CATIA V5 and link them to sketch dimensions

This guide shows how to create parameters in CATIA V5 and use them to drive a sketch and a 3D pad. You start with a rectangle in the Sketcher, then create length parameters with multiple values and a radius parameter in the **Formulas** dialog. You then link each parameter to a sketch dimension and change the values to update the model.

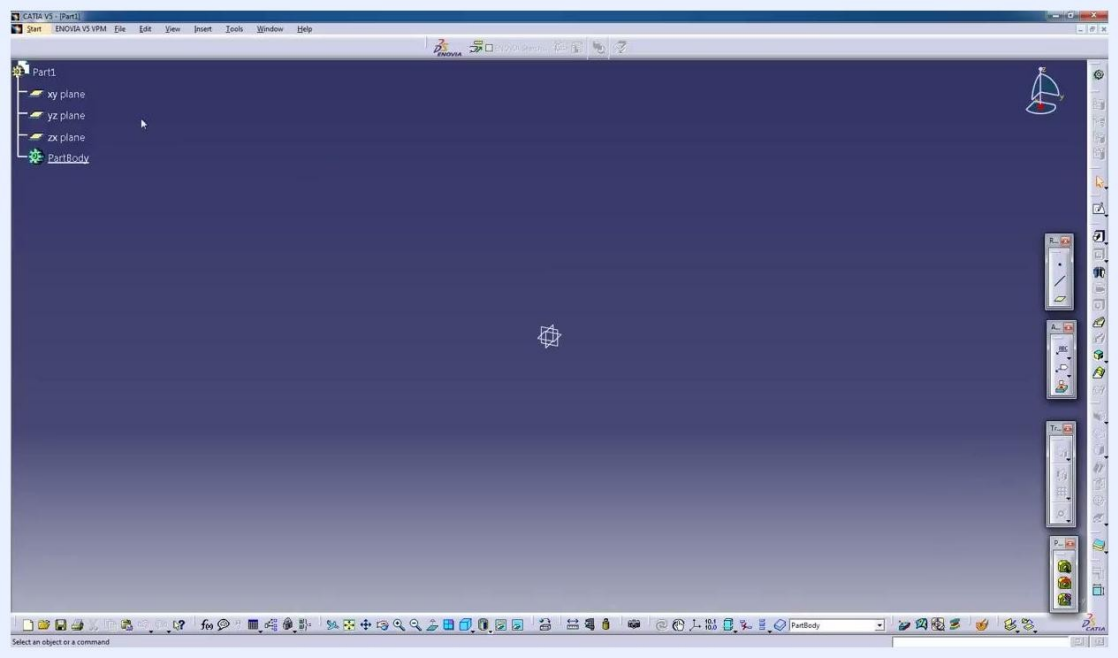
Prerequisites

- CATIA V5 is installed and running.
- You can create a new part and open the Sketcher workbench.

Steps

1. Open a new part

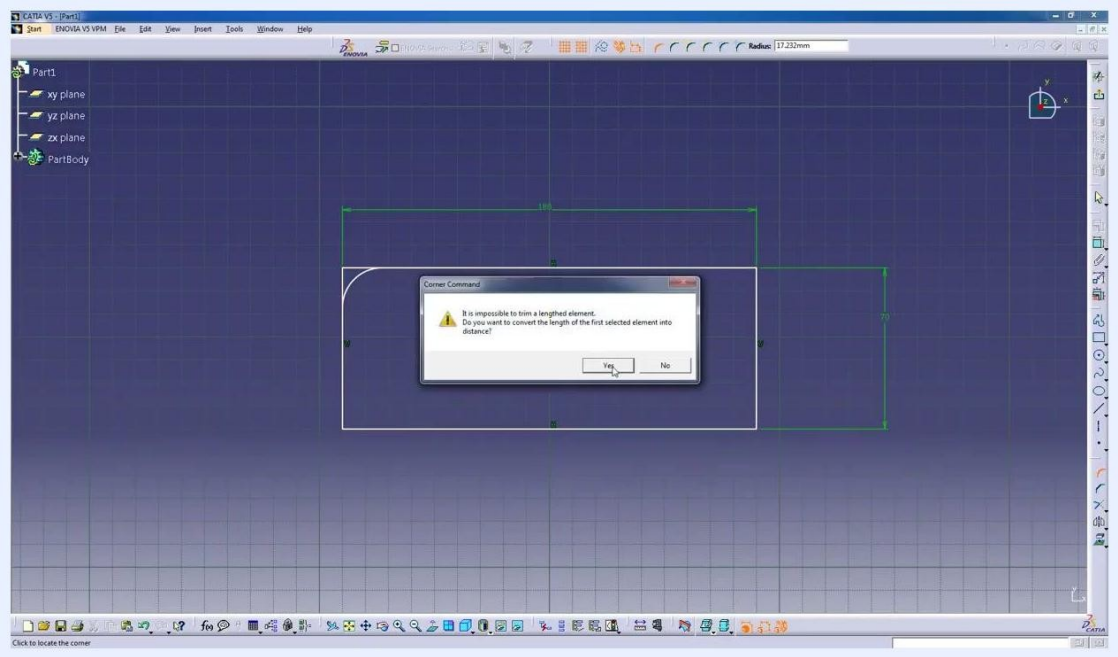
Launch CATIA V5 and create a new part. The tree on the left shows Part1, xy plane, yz plane, zx plane, and PartBody. The main area is empty and ready for sketching.



CATIA V5 initial workspace with Part1 and the planes listed in the tree, an empty main area, and toolbars visible

2. Draw a rectangle in the Sketcher

Enter the Sketcher workbench and select a plane, such as xy plane. Draw a rectangle in the sketch area. The rectangle is centered on the origin, with green constraint points at the corners.



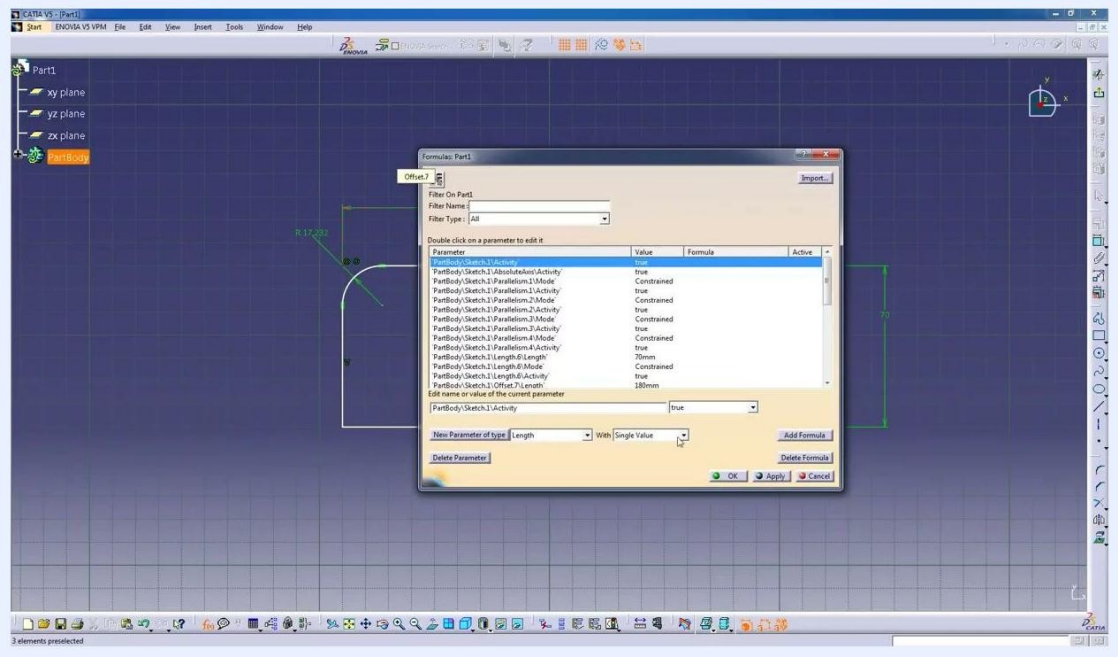
Rectangle drawn in the CATIA Sketcher, centered on the origin with constraint points at the corners

3. Apply dimensional constraints

Add dimensional constraints to the rectangle:

- Set the horizontal length to 180mm.
- Set the vertical height to 70mm.

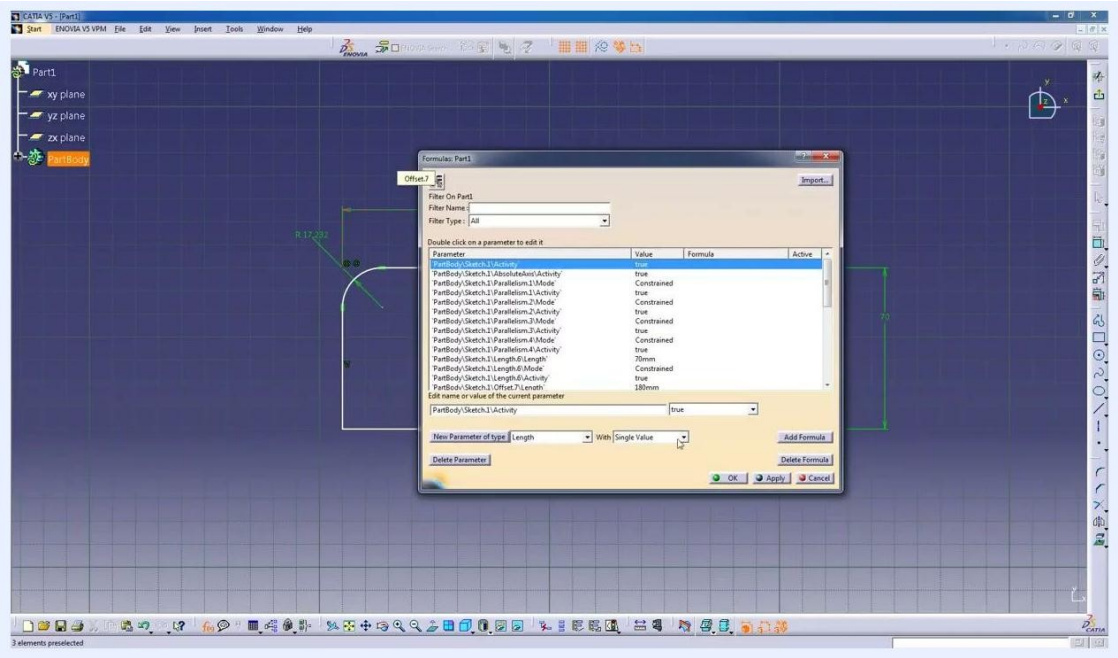
The constraints appear in green above and to the right of the rectangle.



Rectangle with green dimensional constraints of 180mm horizontal and 70mm vertical

4. Add a corner radius and accept the conversion

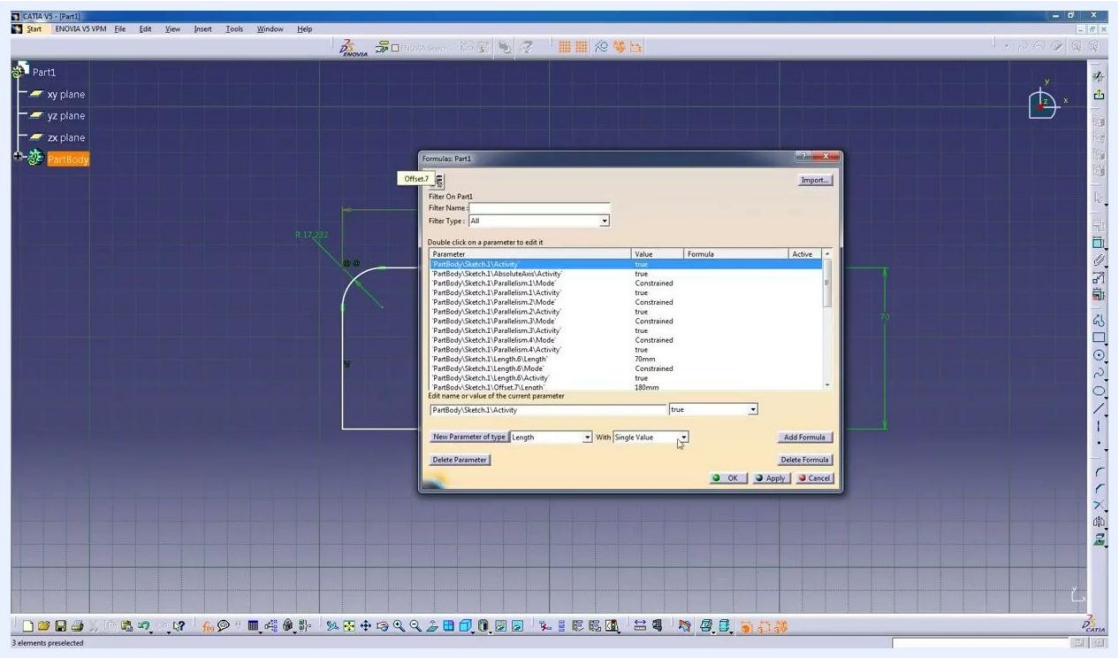
Select the corner tool and try to add a radius to one corner of the rectangle. A warning dialog appears with this message: "It is impossible to trim a lengthed element. Do you want to convert the length of the first selected element into distance?" Click **Yes** to proceed.



Warning dialog about trimming a lengthed element, with Yes and No options and Yes selected

5. Open the Formulas dialog

Open the **Formulas** dialog to manage the parameters of the part. The dialog lists all parameters, including constraints and activities, such as PartBody\Sketch.1\Activity. Double-click a parameter to edit its value or formula.



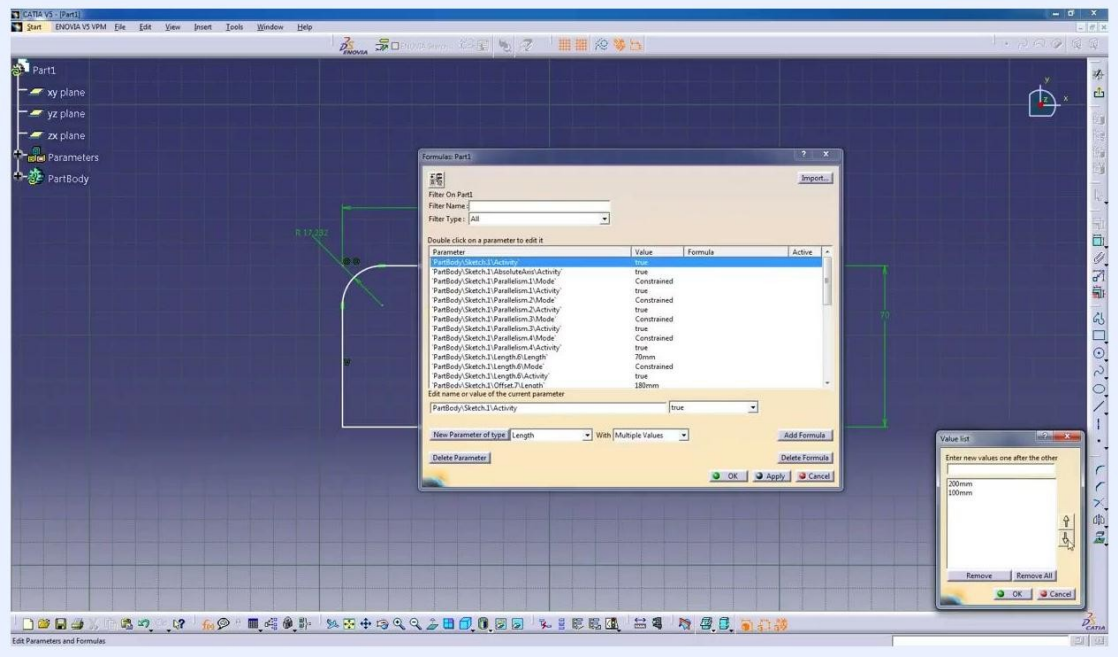
Formulas dialog open, listing parameters such as PartBody\Sketch.1\Activity with their values and formulas

6. Create a parameter with multiple values

This is the first step in how to create parameters in CATIA that offer a choice of values. In the **Formulas** dialog, create a new parameter of type **Length** with **Multiple Values**. A **Value list** dialog appears. Enter these values:

- 200mm
- 100mm

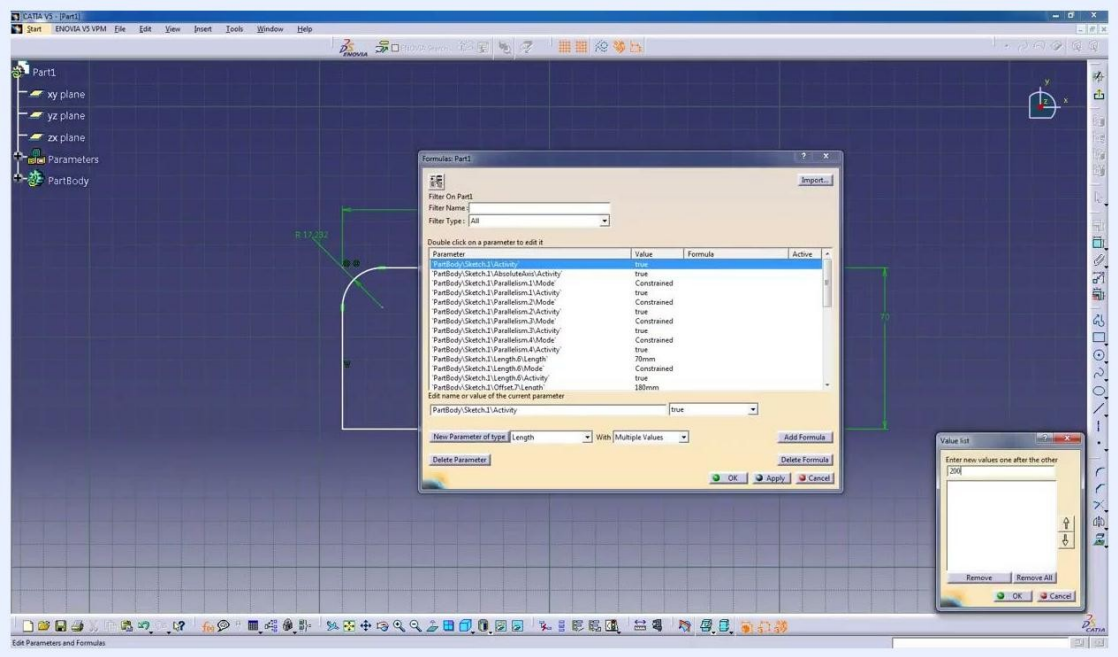
Click **OK** to confirm.



Formulas dialog with the Value list open, showing 200mm and 100mm entered for a new Length parameter

7. Confirm and apply the first parameter

The new parameter, First parameter, appears in the parameter tree with the value 200mm. Click **OK** or **Apply** in the **Formulas** dialog to save your changes.



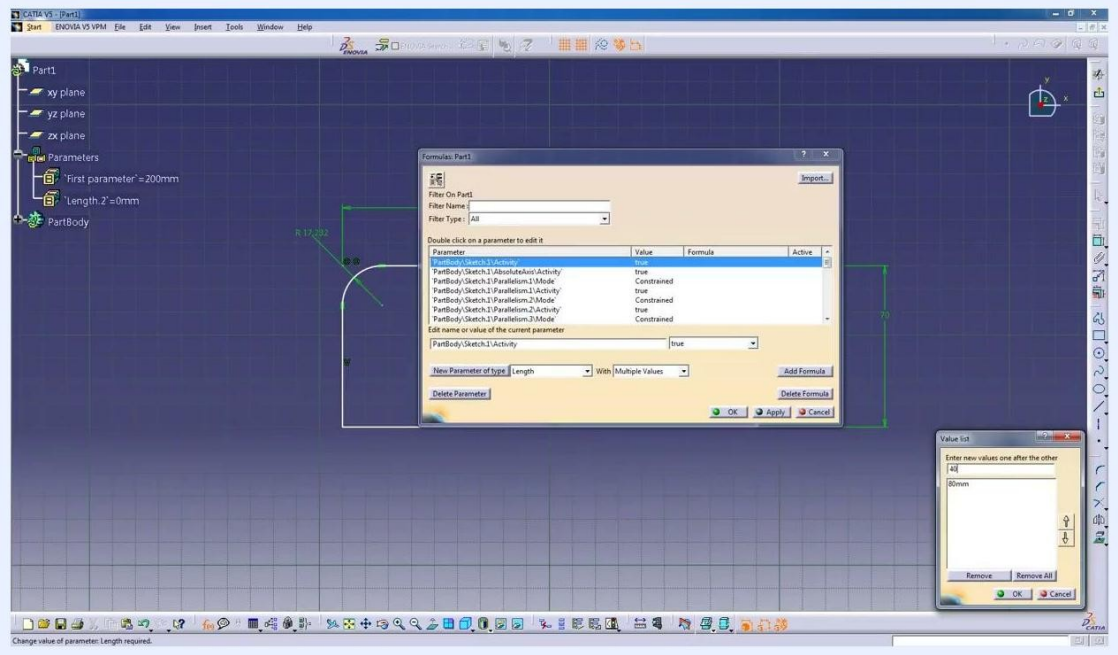
Parameter tree showing First parameter equal to 200mm with the Formulas dialog still open

8. Add a second parameter with multiple values

In the **Formulas** dialog, create another parameter of type **Length** with **Multiple Values**. In the **Value list** dialog, enter these values:

- 40mm
- 80mm

Click **OK** to confirm.



Formulas dialog with the Value list open, showing 40mm and 80mm entered for a second Length parameter

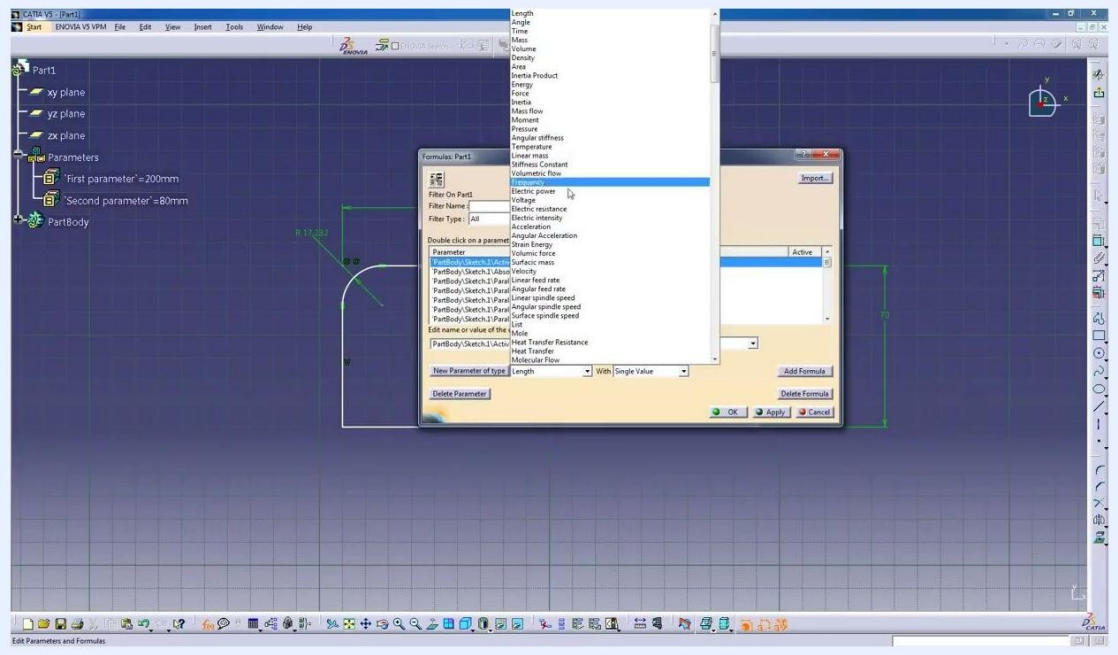
9. Confirm the second parameter

Confirm the creation of the second parameter in the **Formulas** dialog. The parameter is named Second parameter and is set to 80mm. The **Value list** dialog shows 80mm and 40mm as possible values. Click **OK** in the **Value list** dialog.

The parameter tree on the left now shows:

- First parameter = 200mm

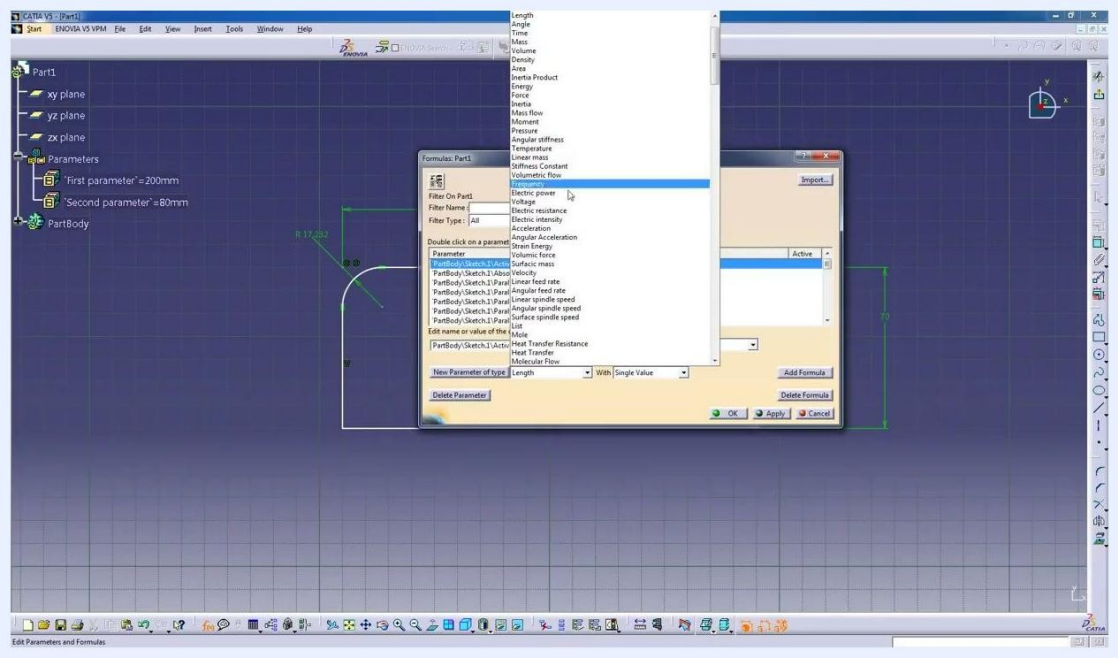
- Second parameter = 80mm



Formulas dialog with Second parameter set to 80mm, the Value list showing 80mm and 40mm, and the parameter tree updated

10. Create a parameter of a different type

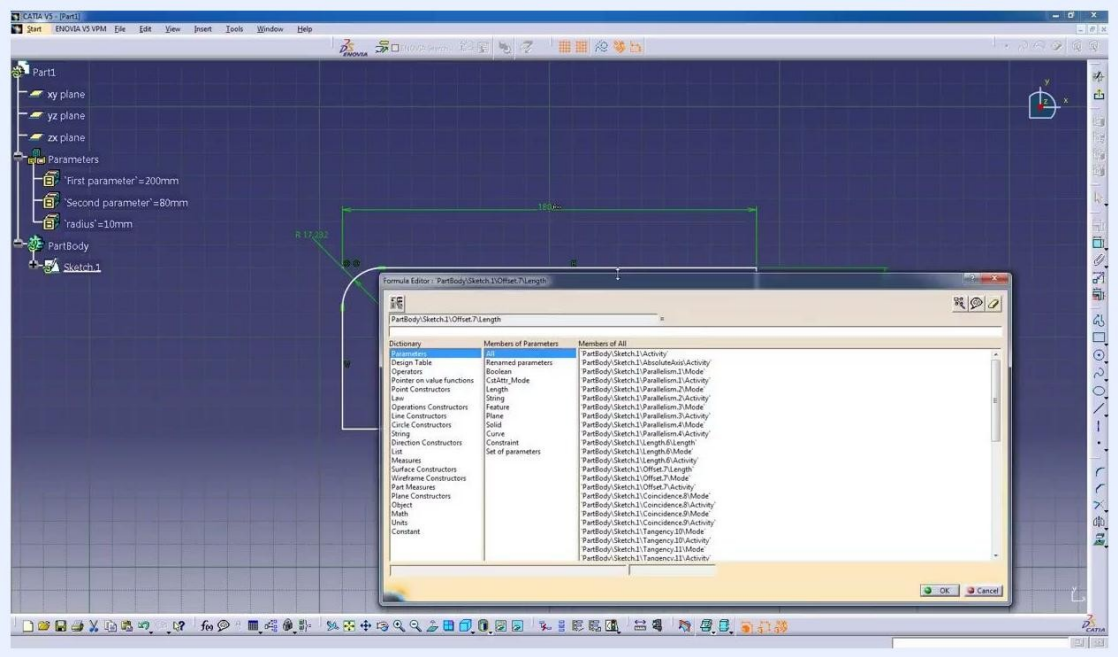
You can create parameters of many physical types. In the **Formulas** dialog, click to create a new parameter and open the drop-down menu of parameter types. The list includes Length, Angle, Area, Volume, Density, Force, Pressure, Temperature, Volumetric flow, Electric power, and more. Scroll the list and highlight Volumetric flow to see the range of available types.



Parameter type drop-down open with Volumetric flow highlighted among many available types

11. Add a radius parameter

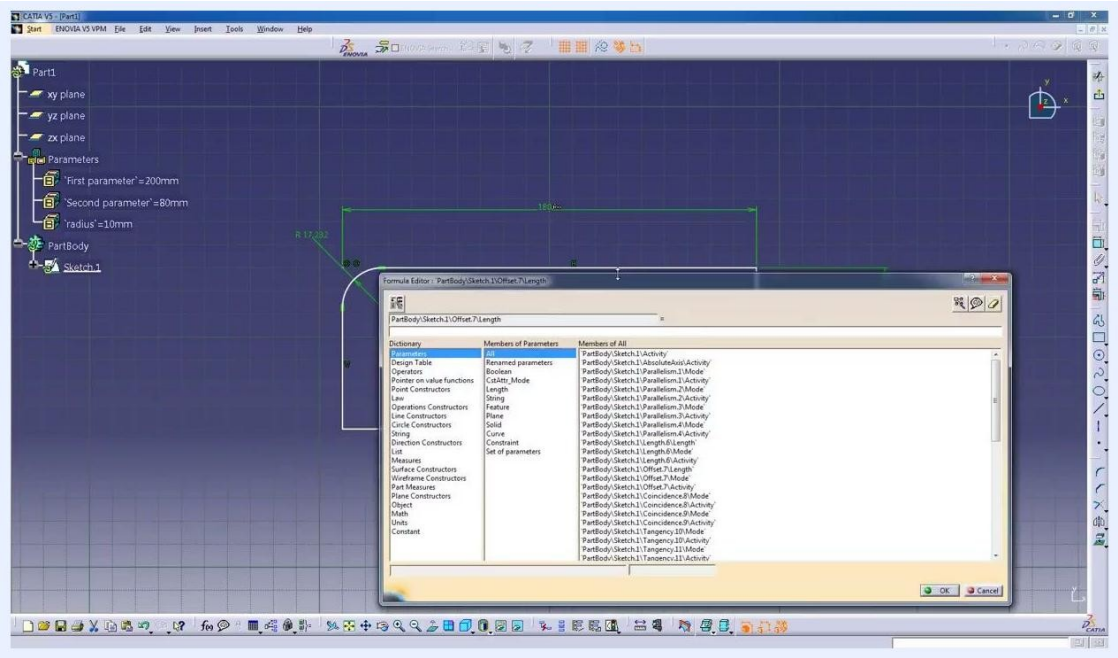
A new parameter named Length.3 appears in the parameter tree with a value of 0mm. In the **Formulas** dialog, rename it to radius and keep its value at 0mm. The parameter tree updates to show radius = 0mm.



Parameter tree showing radius equal to 0mm with the Formulas dialog open and the parameter being edited

12. Edit the radius value

The **Edit Parameter** dialog lets you change the value of radius. Enter 10 in the value field so that the parameter reads radius = 10mm, then click **OK**. The parameter tree shows radius = 10mm, and the sketch displays the rectangle with a rounded corner.



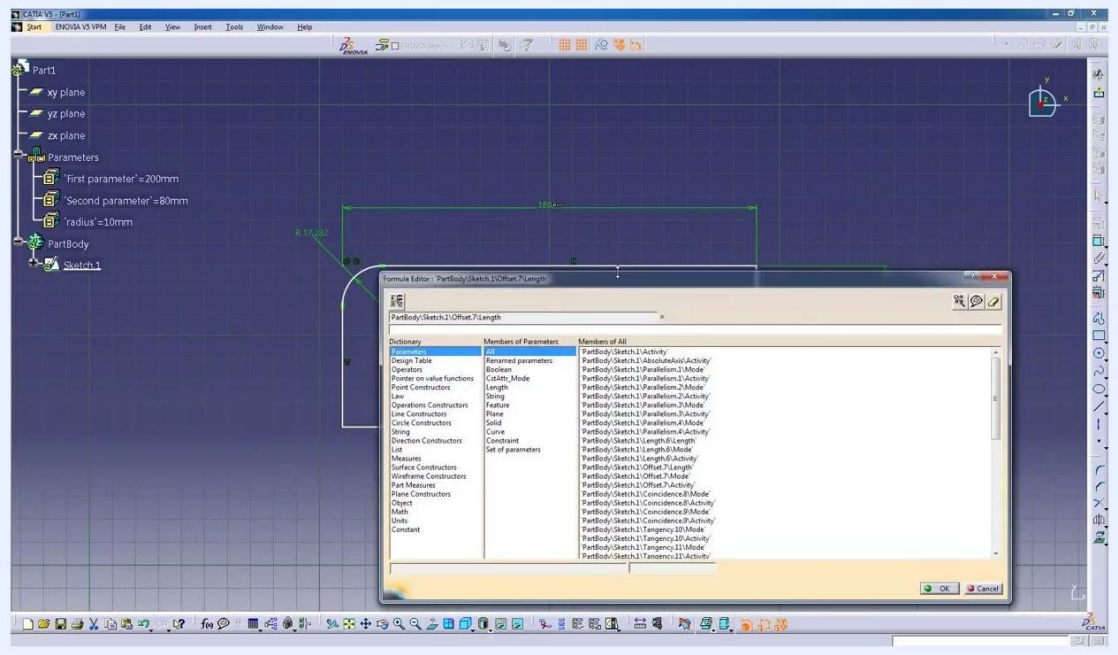
Edit Parameter dialog with radius set to 10, the parameter tree updated, and the sketch showing a rounded corner

13. Open the Formula Editor for a sketch dimension

Open the **Formula Editor** for a sketch dimension, such as PartBody\Sketch.1\Offset.7\Length. The window shows:

- A dictionary of parameter types and constructors on the left.
- Lists of available parameters and members in the center and on the right.
- The formula input field at the top.

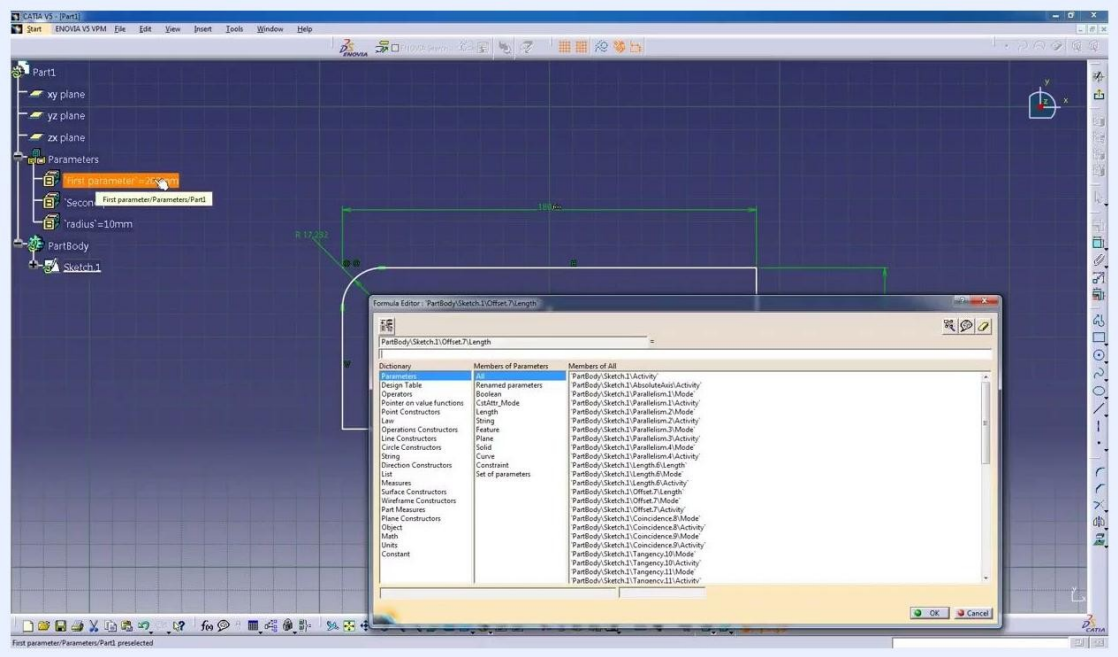
The sketch area remains visible in the background.



Formula Editor open with the dictionary and parameter lists visible and the formula input field at the top

14. Select the first parameter in the Formula Editor

In the **Formula Editor**, select First parameter from the parameter list. The parameter tree highlights First parameter = 200mm. The formula input field is ready to link this parameter to the selected sketch dimension.



Formula Editor open with First parameter highlighted in the parameter tree, ready to link

15. Confirm the automatic update for the first relation

An **Automatic update?** dialog appears with this message: "This relation is synchronous and contains only parameters. Do you want this relation to be updated anyway when global Update is performed as specified in the options?" Click **Yes**. The parameter-driven relation now updates automatically.

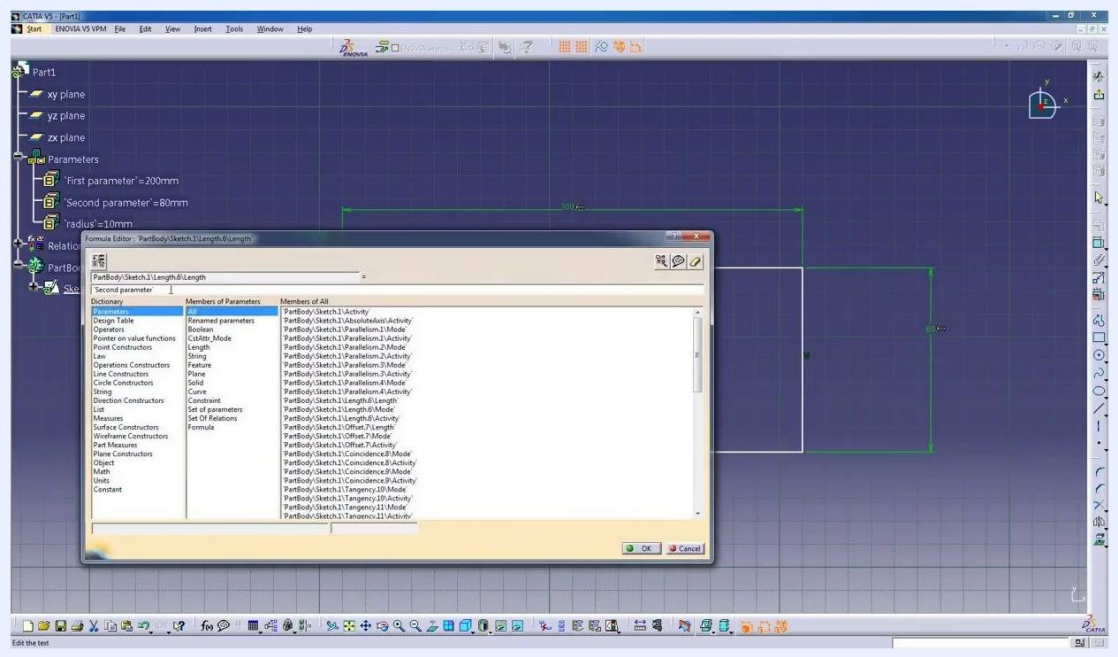
16. Link the second parameter to a sketch dimension

Open the **Formula Editor** for the dimension PartBody\Sketch.1\Length.6\Length. The window shows:

- The formula input field at the top, which displays Second parameter.
- The **Dictionary** pane on the left, with categories such as Parameters, Design Table, and Operators.
- The center pane, which lists Renamed parameters, including Second parameter.

- The right pane, which lists all available members.

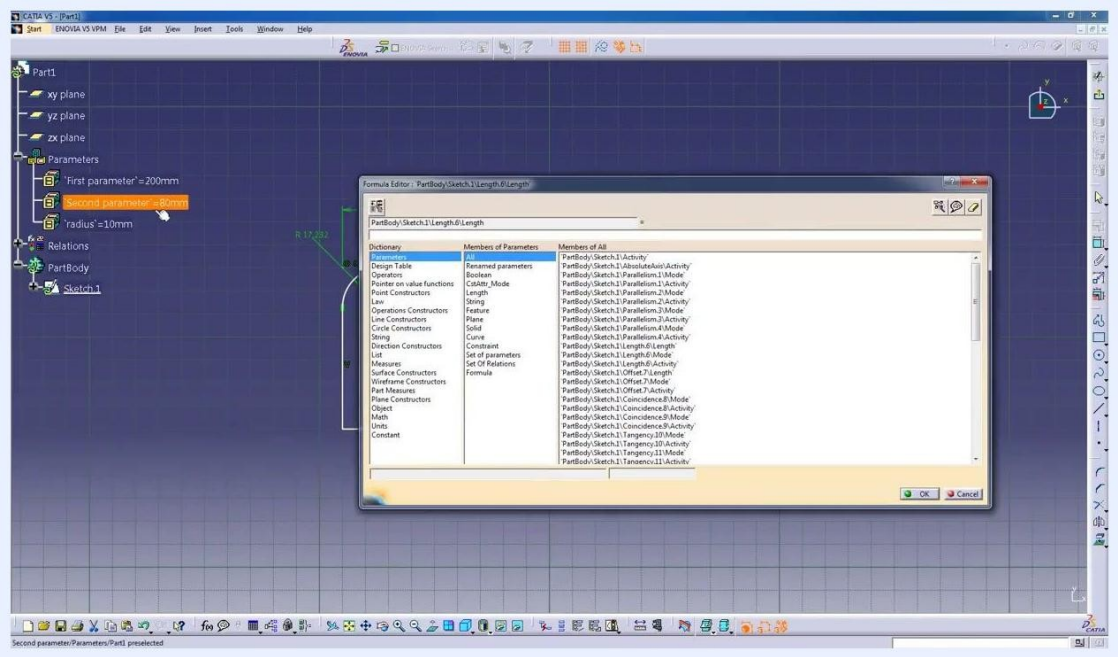
In the background, the sketch shows the rectangle with green dimension lines for 200mm and 80mm. The parameter tree shows First parameter = 200mm, Second parameter = 80mm, and radius = 10mm.



Formula Editor open with Second parameter selected and the parameter tree and sketch visible

17. Confirm the automatic update for the second relation

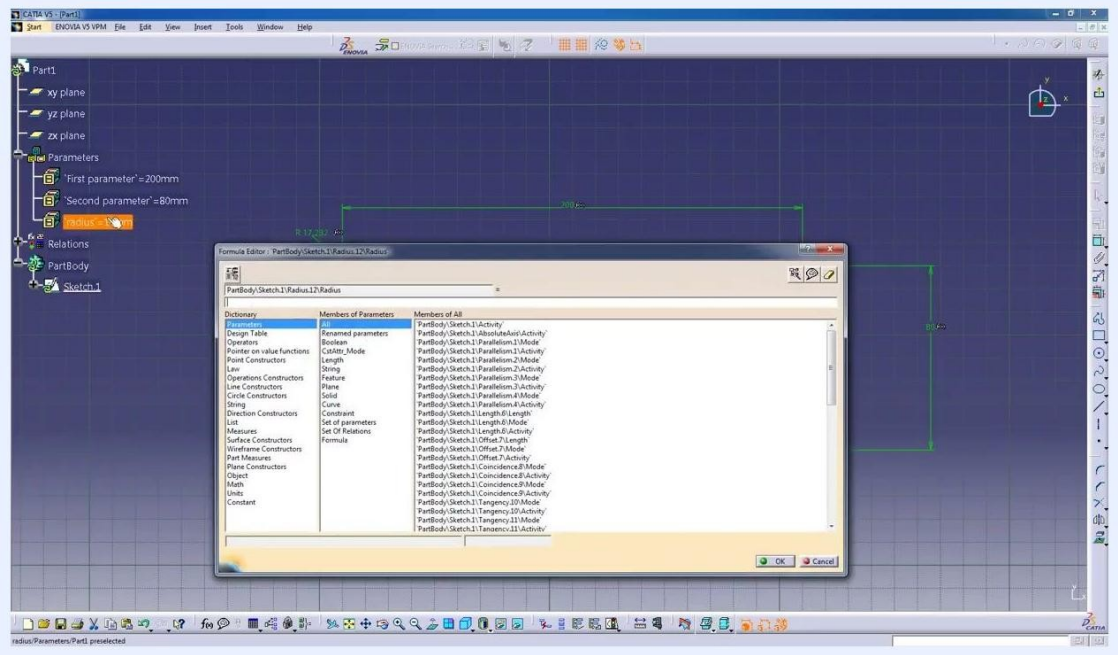
The **Automatic update?** dialog appears again. Click **Yes** to confirm automatic updates for this relation. Both parameters now drive their sketch dimensions and update automatically.



Automatic update dialog with Yes being clicked, and the Formula Editor and parameter tree visible

18. Link the radius parameter to the sketch radius

Open the **Formula Editor** for the radius dimension PartBody\Sketch.1\Radius.12\Radius. The formula input field at the top displays radius, and the center pane highlights radius under Renamed parameters. The parameter tree highlights radius = 10mm, and the sketch shows the rectangle with a rounded corner. This links the radius parameter to the rounded corner.



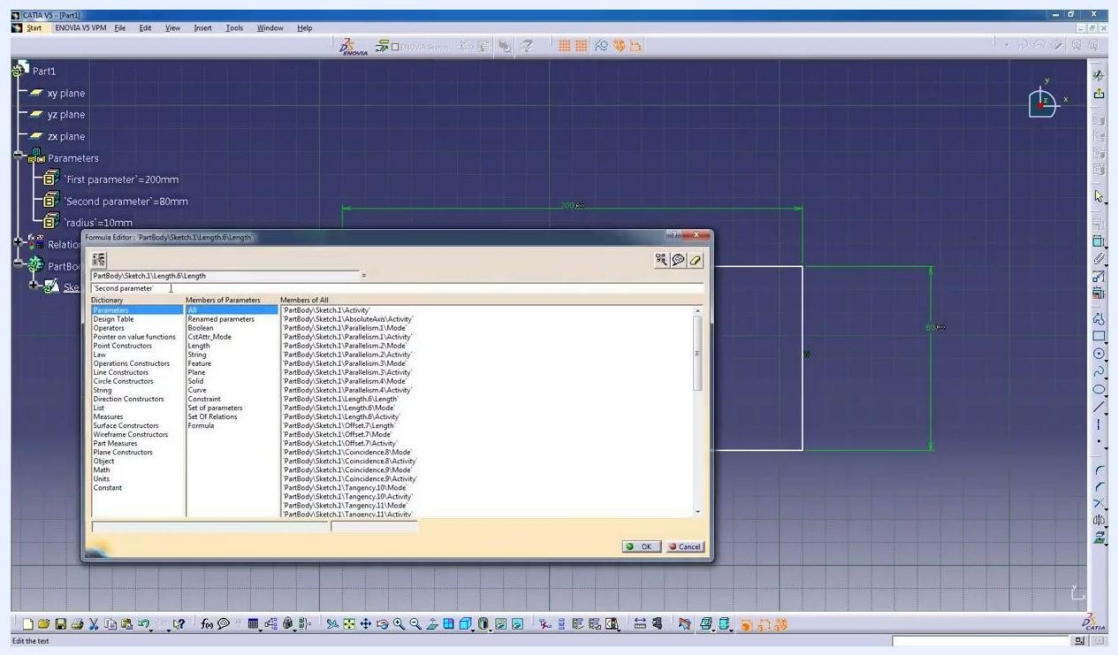
Formula Editor open with radius selected and the parameter tree and sketch visible

19. Create a 3D pad from the parameterized sketch

Switch to the 3D view. The **Pad Definition** dialog appears with these settings:

- **First Limit** is set to Type: Dimension.
- **Length** is set to 20mm.
- **Selection** is set to Sketch.1.
- **Thick**, **Reverse Direction**, and **Mirrored extent** are visible but not selected.

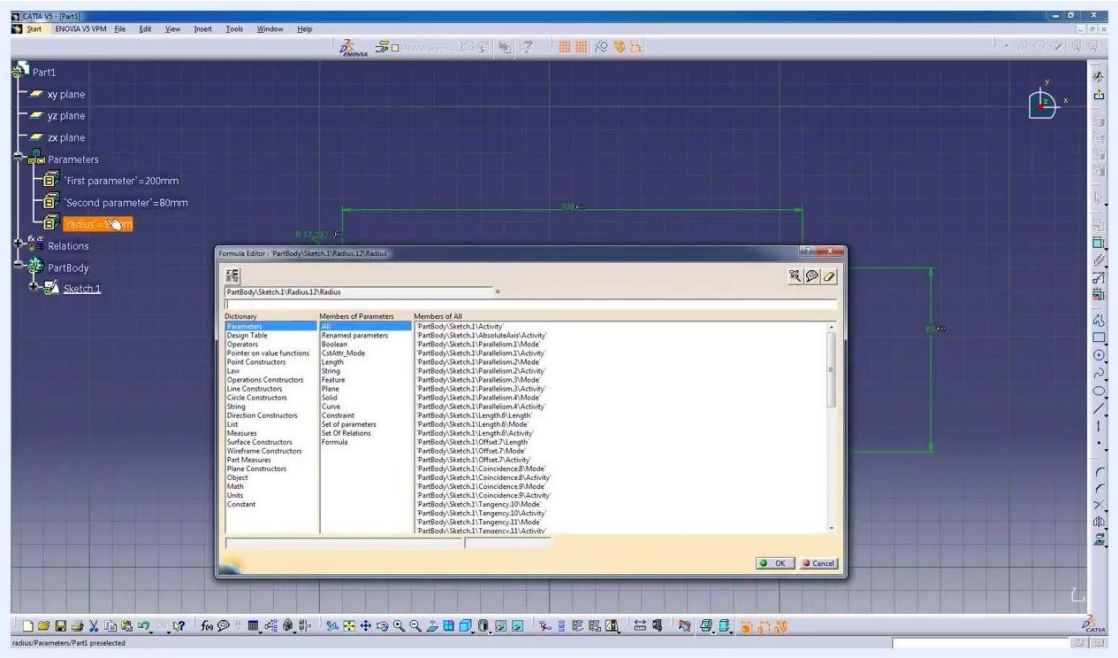
The 3D preview shows the extruded pad. The parameter tree shows all parameters and the new Pad.1 feature under PartBody.



Pad Definition dialog open with the 3D pad previewed and the parameter tree visible

20. Change the first parameter from the drop-down list

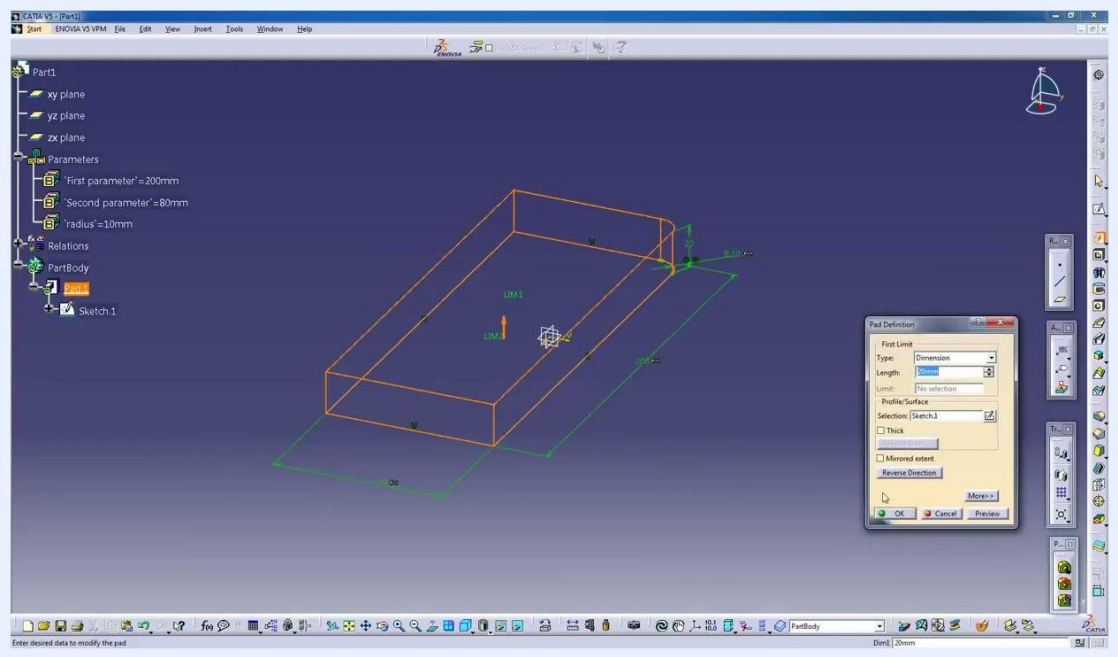
Open the **Edit Parameter** dialog for First parameter. The drop-down menu shows the two values you defined, 200mm and 100mm. Select 100mm. The 3D model updates to the new length, and the parameter tree shows First parameter = 100mm.



Edit Parameter dialog for First parameter with a drop-down listing 200mm and 100mm, and the 3D model updated

21. Edit the radius parameter

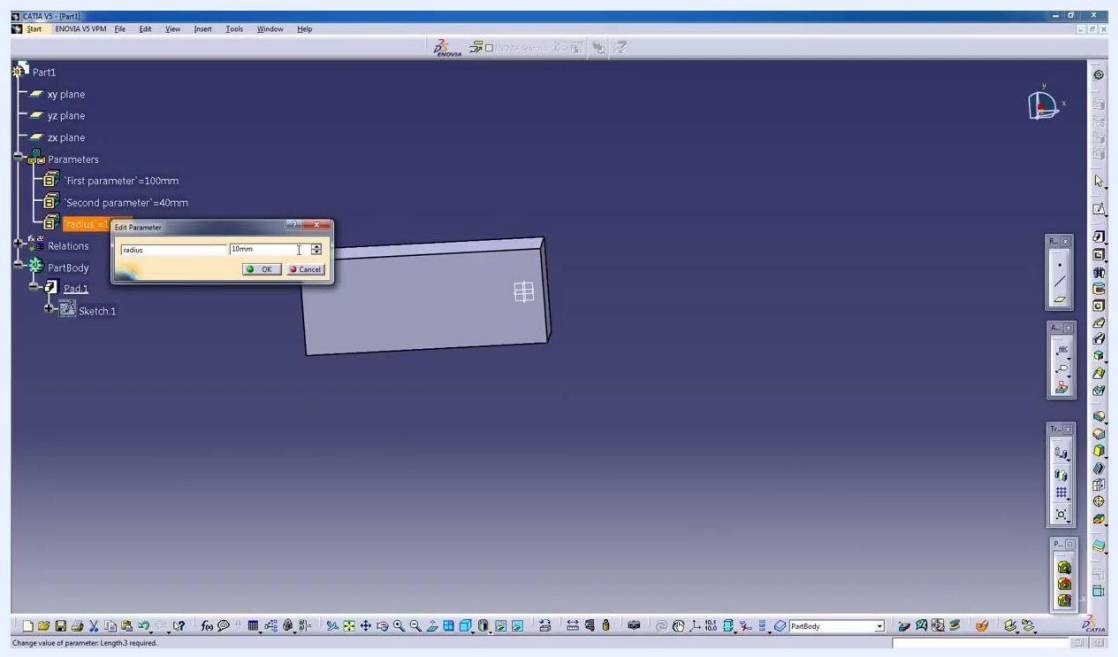
Open the **Edit Parameter** dialog for radius. The value field displays 10mm. Enter a new value or keep the current one. The 3D model updates to reflect the radius, and the parameter tree shows radius = 10mm.



Edit Parameter dialog for radius with the value set to 10mm and the 3D model visible

22. Confirm the radius value

With the **Edit Parameter** dialog for radius still open and the value at 10mm, click **OK** to apply it. The 3D model keeps the current radius, and the parameter tree continues to show radius = 10mm.



Edit Parameter dialog for radius with the value confirmed as 10mm and the 3D model visible

Summary

You now know how to create parameters in CATIA V5 and use them in a design. You created length parameters with multiple values and a radius parameter, linked each one to a sketch dimension with the **Formula Editor**, and confirmed automatic updates. Changing a parameter value in the **Edit Parameter** dialog now updates the geometry of the parameterized sketch and the 3D pad.

What's next

- Use the **Edit Parameter** dialog to switch First parameter or Second parameter between their listed values and watch the pad update.
- Create parameters of other types from the parameter type drop-down in the **Formulas** dialog.