



The following is an archive of Nik's CATIA V5 Tips and Techniques.

To unleash the full functionality of CATIA V5, attend an AscendBridge CATIA Course or request a one day mentoring at your site. Call 1-888-326-TEAM or email catia@ascendbridge.com

#1 Tips and Technique

Q: Do you know how to convert a 2D drawing view (in dwg format) into a 3D Part using Advance Part Modeling options of CATIA V5?

A: You Can use any 2D view with various drawing objects (even in dwg format) to create a 3D Solid.

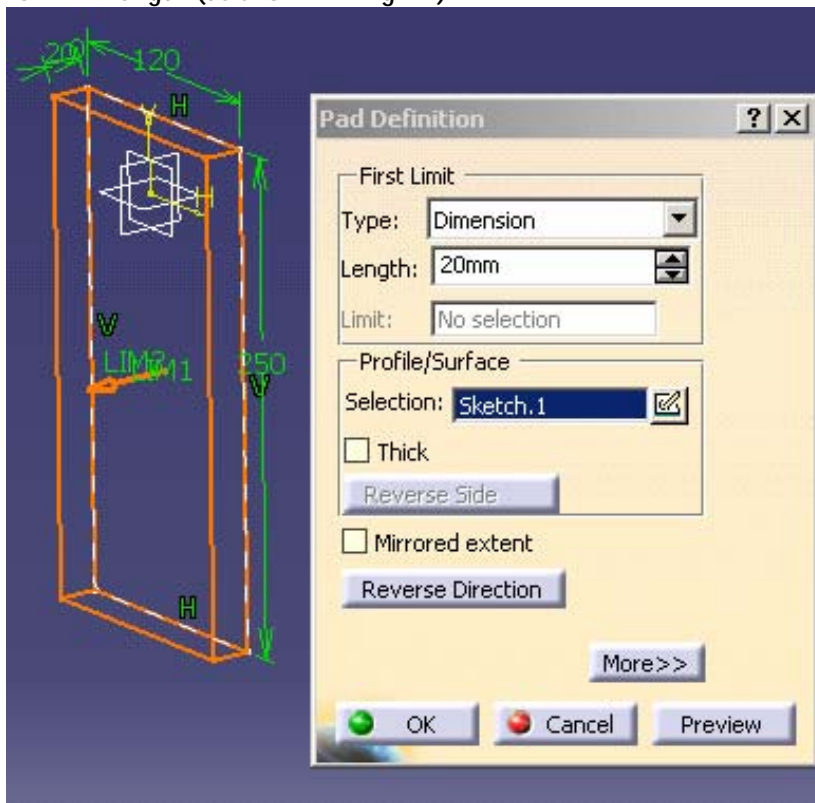
- Copy the 2D view from CATIA drafting screen into Sketcher as sketch.
- As the sketch contains multiple Profiles you can not make a solid feature by simply selecting the given sketch, as a error prompts: Several Open Profiles
- If you select Yes the Feature definition box appears.
- Right click in blue area in front of 'Profile/Surface Selection'
- Click on 'Go to Profile Definition' in Contextual menu
- Profile Definition Dialog Box opens
- Select the Part of Sketch you want to use for that feature.
- You can go on creating other features using same sketch but different sub profiles to make the final 3D Part.

This method also helps in reducing the number of sketches in your Part history tree while modeling complex solids and better management of features using sketches.

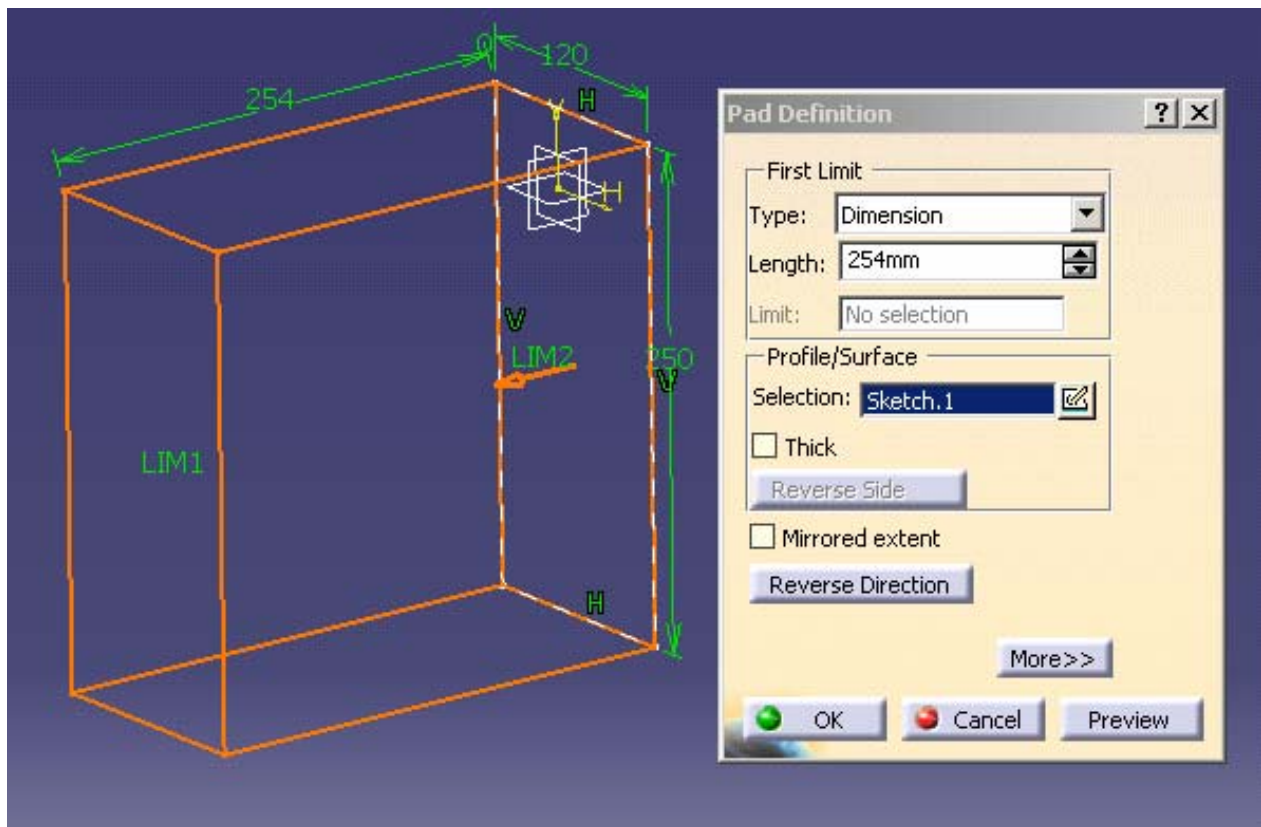
#2 Tip and Technique

Q: Did you know that designers can key in values in combination of units or in formulas in CATIA V5 dialog boxes?

A: You can key in values in any CATIA V5 dialog box in the following formats irrespective of the Standard set units. For example if the length Units are set in mm and you are keying in the value for PAD length (as shown in Fig-1A)



You can key in 10in and the PAD will measure 254mm (as shown in Fig-1B)



Also try to key in $((5\text{in} \times 6)/4) + 9\text{mm} + 500\text{micron}$ and click Preview the PAD will measure 200mm.

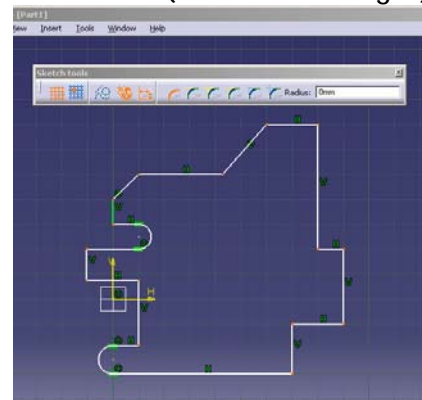
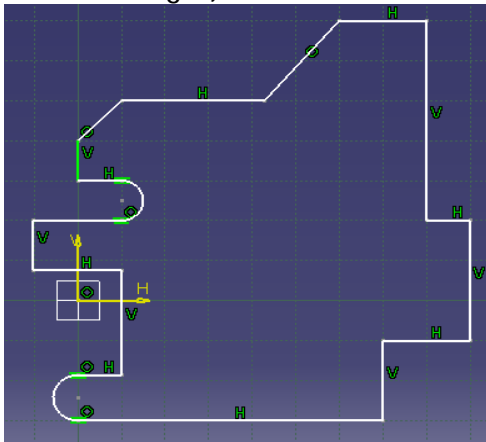
The software automatically computes the entered value (even in the form of complex formula with combined units) equal to the units set in the CATPart and generate features with correct computed measurements.

#4 Tip and Technique

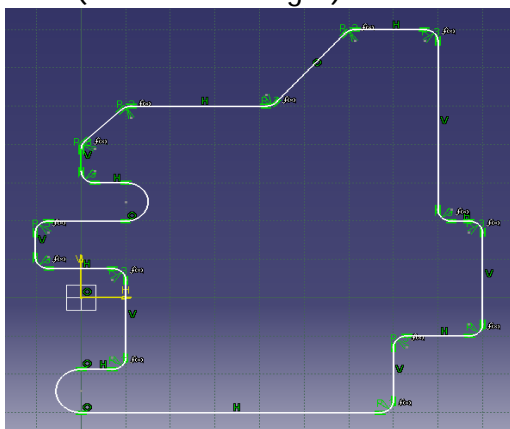
Q: Do you know how to create multiple corners on a complex Profile, of equal radius and related to each other in CATIA V5 Sketcher?

A: You can create multiple corners on selected points on a Profile in one step by following this procedure:

- Draw a required complex Profile in CATIA V5 Sketcher (as shown in Fig-1)
- Multi-select all the points on the Profile where the corners are required and select the Corner icon. Key in Radius value (as shown in Fig-2)



- The corners are created at all the selected points on the profile with given radius. (as shown in Fig-3)
- For modifying the radius of all the corners in one step just double click on first selected corner (without f(x) symbol) and key in the new value all the corners get updated to new value.



The all corners created on the profile with this method are related to the first selected corner with a formula. But if required the formula can be modified or deleted in order to change the radius of any corner independent of the others.