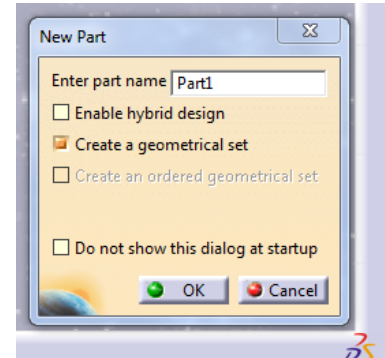# Creating Design Tables in CATIA: Getting on the Right Track
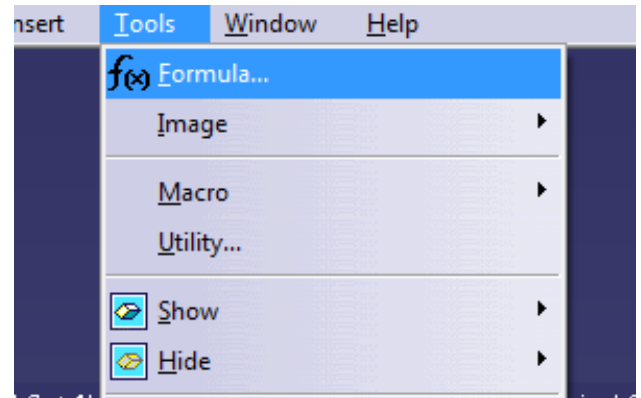
Setting your part:

1. **Workbench Setup:** Open the Sketcher Workbench and when part name dialog box opens, give it a name and deselect "Enable hybrid design" and select "Create a geometrical set". Although this step isn't entirely necessary it allows you to easily distinguish between solids and surfaces/curves.
2. **Relations Tab:** Click on the following setup to make sure the Relations Tab will show up in the Design Tree when it needs to: Tools – Options – Infrastructure – Part Infrastructure – Display- then make sure "Relations" is checked
3. **Part Body:** Right click on "Part Body" in the Design Tree and define as the "In Work Object", since CATIA will auto default to "Geometrical Set.1" being the in work object

Part Building Recommendations:

- **Formula Tips:**
  - When building a part with the intention of using design tables using formulas are essential in the creation of an efficient table.
  - It is easy to see what each parameter is by selecting it; the desired dimension should be highlighted.
  - Within the formula window you can rename parameters that will show up in your design table, although not always necessary, it can make it easier to distinguish parameters.
  - In the list of parameters
    - "Geometrical Set.1\Sketch.1\Length.5\Length" would be the statement that determines a certain dimension within a drawing. Keep in mind that there are units associated with the value.
    - "PartBody\Pocket.1\Activity" would be the statement that determines whether or not this pocket will be activated. Its value is determined by a true or false declaration.
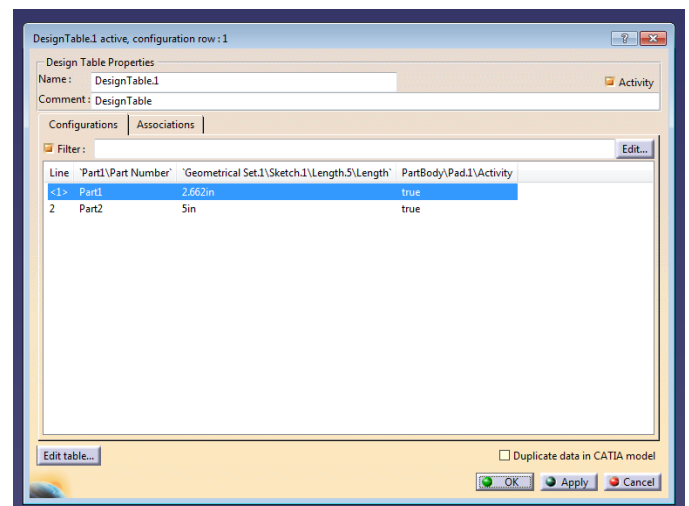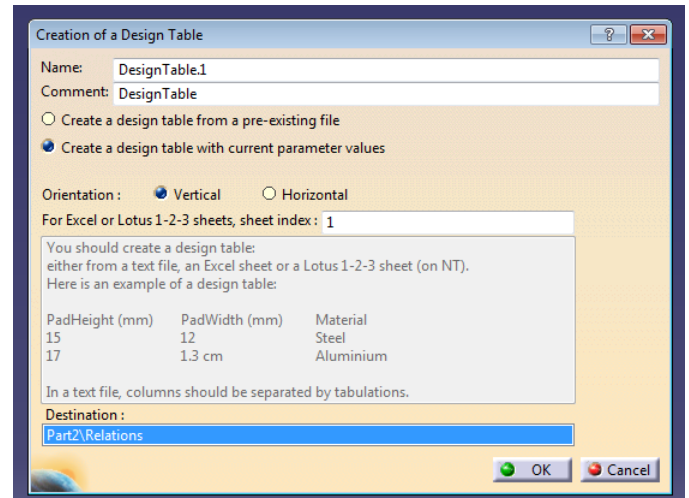
- ▪ "Part2\Part Number" and "Part2\Product Description" can be also constrained within this window.
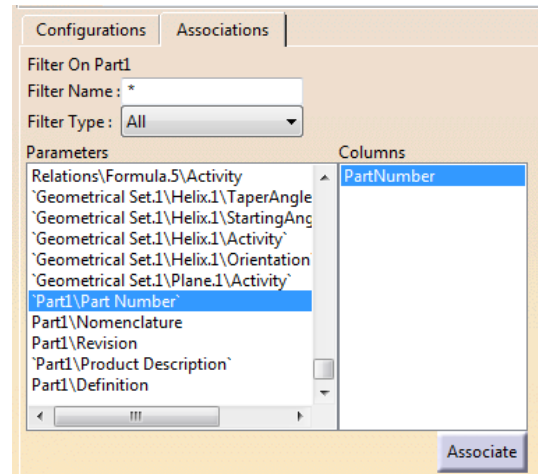
Steps for Creating and Managing a Design Table:

1. Click the Design Table icon on the bottom of the design space.

2. Make sure you select "Create a design table with current parameter values" and also that the orientation is "Vertical".

3. Click OK

4. A new dialog box will open, scroll through the parameters and find the ones that you want to be dictated in your design table.

5. Click OK and your design table will have been created.

6. A new dialog box will open with your design table configuration. This is the main window you will be using to select part configurations and table associations, as noted by the tabs.

7. To edit the table click "Edit Table" in the bottom left hand corner of the dialog box; the design table will open as an Excel spreadsheet.

8. Insert the values in each row to add different design options. Keep in mind that when you modify "Length" objects you need to include the proper dimensions or you will have errors.

9. If this part is going to be made into a catalogue make sure you add a "PartNumber" column and populate it.

10. Once done editing save/close the Excel Worksheet.

11. A "Knowledge Report" dialog box will appear in CATIA, telling you that you have updated the design table Click OK to confirm changes.

12. Now for CATIA to recognize the addition of the "PartNumber" column you will need to select the "Associations" tab. In this tab select the 'Part1/Part Number' in the parameter form and then select "PartNumber" within the column form and then press associate. This will now allow for CATIA to identify the newly created spreadsheet column. Finally select the new association and move it to the top of the list in the "Associations between parameters and columns" form. This window can be handy when you make changes to your excel file independently.

13. Now you should be free to select for different designs from the "Configuration" tab.
14. Click OK finish modifying your design table.

Creating/Using a Catalogue:

1. Once your desired part with design table has been created, save and then click the following: Start – Infrastructure – Catalog Editor

2. Make sure "Chapter.1" is highlighted on the left hand side and select "Add Part Family" by either selecting it through "Insert" or the icon on the right hand side of the window.

3. Give it a name and click "Select Document" and select your part now press "OK"

4. Save the Catalog

5. Now your catalog has been created, you are now free to use it in the Assembly Design Workbench

6. Remember to select a catalog within the Assembly Design Workbench all you need to do is
select the Catalog Editor button on the bottom right-hand of the Design Space.

7. Select your part catalog and now you can create assemblies by combining different parts.