Catia V5 - Parameters and Formulas Tutorial

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I. Tutorial Overview

A. Disclaimer

This document is provided as a free resource, to encourage Catia V5 users in the never-ending quest for new and better skills. It has been provided at the time and expense of the author, with no monetary compensation. The author has no desire to control the end-use of the knowledge received from this document; however, use of any portion of this document for commercial training purposes is strictly forbidden. This tutorial is to remain forever in the realm of free knowledge transfer. Please report any misuse or violation of this free usage policy.

B. Software Requirements

The intended audience of this tutorial, is Dassault Systemes customers using Catia V5, Release 14 or higher, with a minimum recommended P2 configuration, MD2 license, or better. The author is using Catia V5 R14 SP09, and HD2 license.

C. About Parameters and Formulas

These topics are beyond basic Part Design. It is assumed that the reader has mastered basic skills of creating parts in Catia V5. Therefore, no extra attention will be given to the details of steps leading up to the subject matter. This is necessary for clarity and brevity.

Having said that, the purpose of creating Parameters and Formulas, is to allow the user a means to quickly and meaningfully design and edit a part, based on user-defined logic. But what does that mean? Simply put - you, the reader, have the ability to set up relationships that make editing the part easier, not only for yourself, but downstream users! By using a little forethought and ingenuity, the designer (and/or other end-user) can build a part that can be edited on the fly, without ever re-visiting a design function.

This tutorial will explore how to set up parameters and formulas in a methodical way, and to try to steer the reader away from common and painful mistakes en route to the end product. In future tutorials, we will explore ways to integrate the knowledge gained here, into further automation of the design process.

Let's begin.

TIP: The screen shots included in this tutorial may be too small to view when printing. This document is best viewed on the screen, with a zoom ratio of 200%.

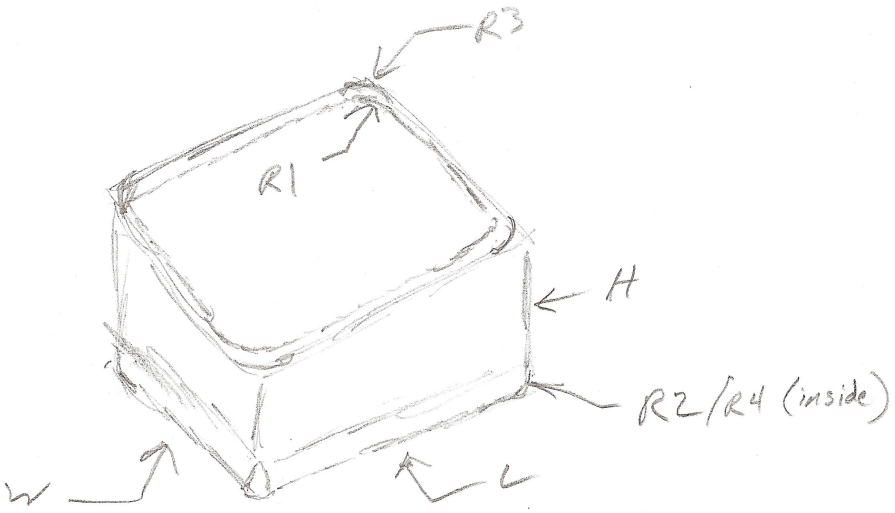
II. The Design Process

The example begins with the selection of a suitable part to be designed. In this case, the example will be based on an open-ended "box" type part, which could represent either a machined, or deep drawn process. We will explore the design requirements, and build the part step by step. It is recommended that the reader follow all steps, and save the resultant part for future tutorials. As always, save your work often.

A. Pre-defining the Part

At this point, the reader may or may not understand anything about parameters and formulas. Thus, it would seem a good time to further simplify the definition of parameters and formulas. Parameters could be thought of as variables and/or constants, (depending on the use) and formulas are the equations. Used in this context, some of the "mystery" of parameters and formulas quickly begins to fade.

We will begin by visualizing, or paper sketching the part we wish to create and control. The design process usually begins with something like this:



From a rudimentary sketch, we will gather all of the information we need to completely lay out the part. This sketch shows that we have several important characteristics to control:

1) Length, width, and height
2) fillet dimensions (inside and out)
3) material thickness

We will also give thought to the 3D model, and its downstream application. Part

numbering also becomes relevant to this design.

To logically determine how to set up relations, we must eliminate redundancy. A good approach would be to think of how this part is produced. We will assume, for the time being, that this is a deep drawn part, regardless of thickness. We know that the part has the 3 basic dimensions of a box – LxWxH – this is a given. And, the part has material thickness. But there are 4 fillets on this part. By studying the sketch, we can deduce that 2 of these fillets are independent, and 2 are dependent. That is, 2 fillets require a defined value, and the other 2 are based on the values of the first 2. (those that required input) To clarify that, the outside fillets are a function of the inside fillets, + the material thickness. This can be one of our formulas!

After just a brief glance of this sketch, it can be determined that the following values are candidates for parameters or formulas in our design process:

1) Part Number (we will discuss this more later)
2) Length
3) Width
4) Height
5) Material Thickness
6) Inside Vertical Fillet
7) Inside Horizontal Fillet
8) Outside Vertical Fillet
9) Outside Horizontal Fillet

Before moving on, you may ask yourself, "why add a parameter, when I can just change a fillet size, or change the dimensions of size in the model?" That's a really good question, and here's the answer: Have you ever opened a model, and had to spend time hunting through operations to figure out which was which? Or have you ever broken a model by modifying the wrong feature, resulting in a frustrating, and seemingly never-ending update cycle? Why not add a parameter, which tells the end-user exactly what he/she is changing? Why not make it clear, and save time and effort for everyone? This point may be lost at this time, but in the course of this example, it shall become abundantly clear.

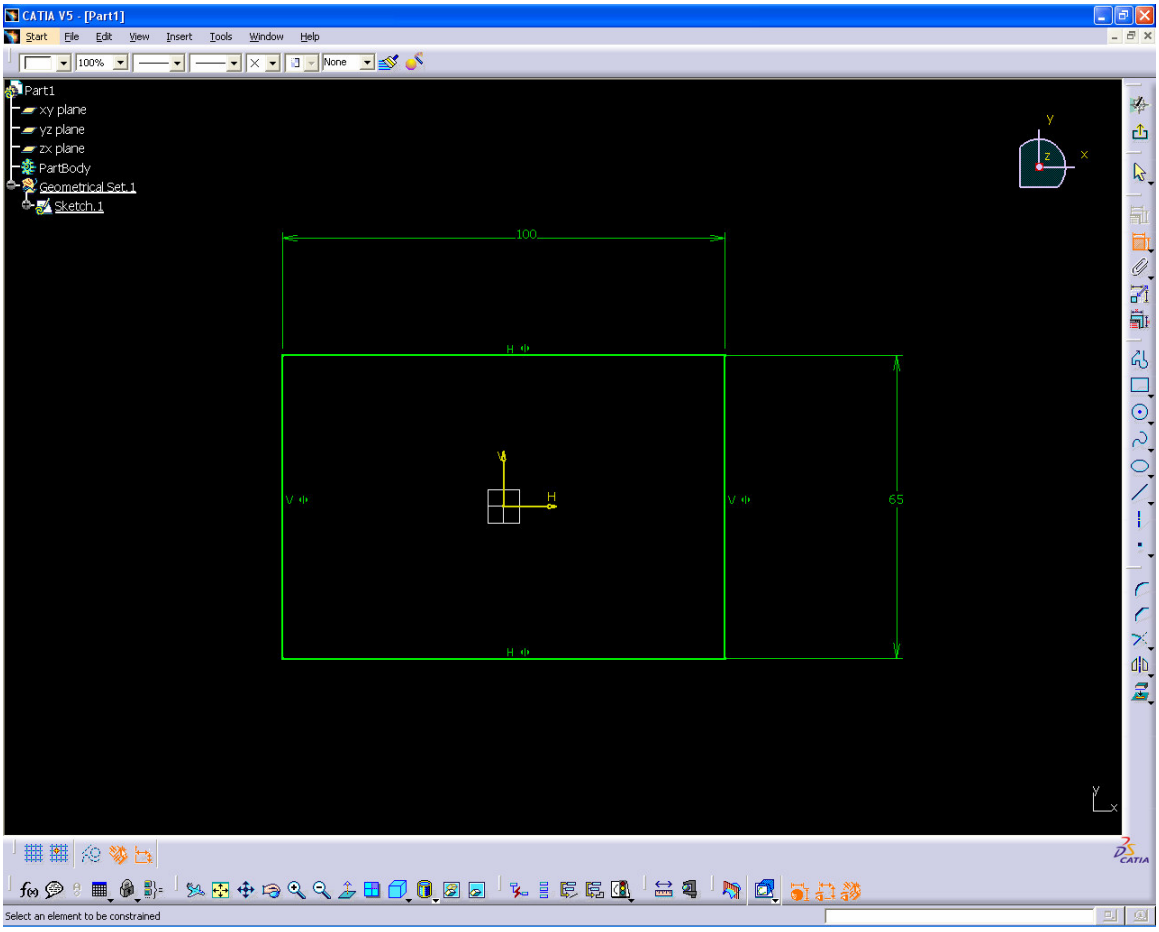
It is time to begin designing the part.

B. Part Layout

Open Catia, and begin with a new CATPart. For now, name the part 1000001.

Start a new sketch, and place it on plane XY. Once inside the new sketch, use the "Rectangle" tool to create a rectangle. We will constrain it around the sketch axis by symmetry, (in both directions) and add distance constraints to iso-constrain the part. The values of the distance constraints are not important at this time.

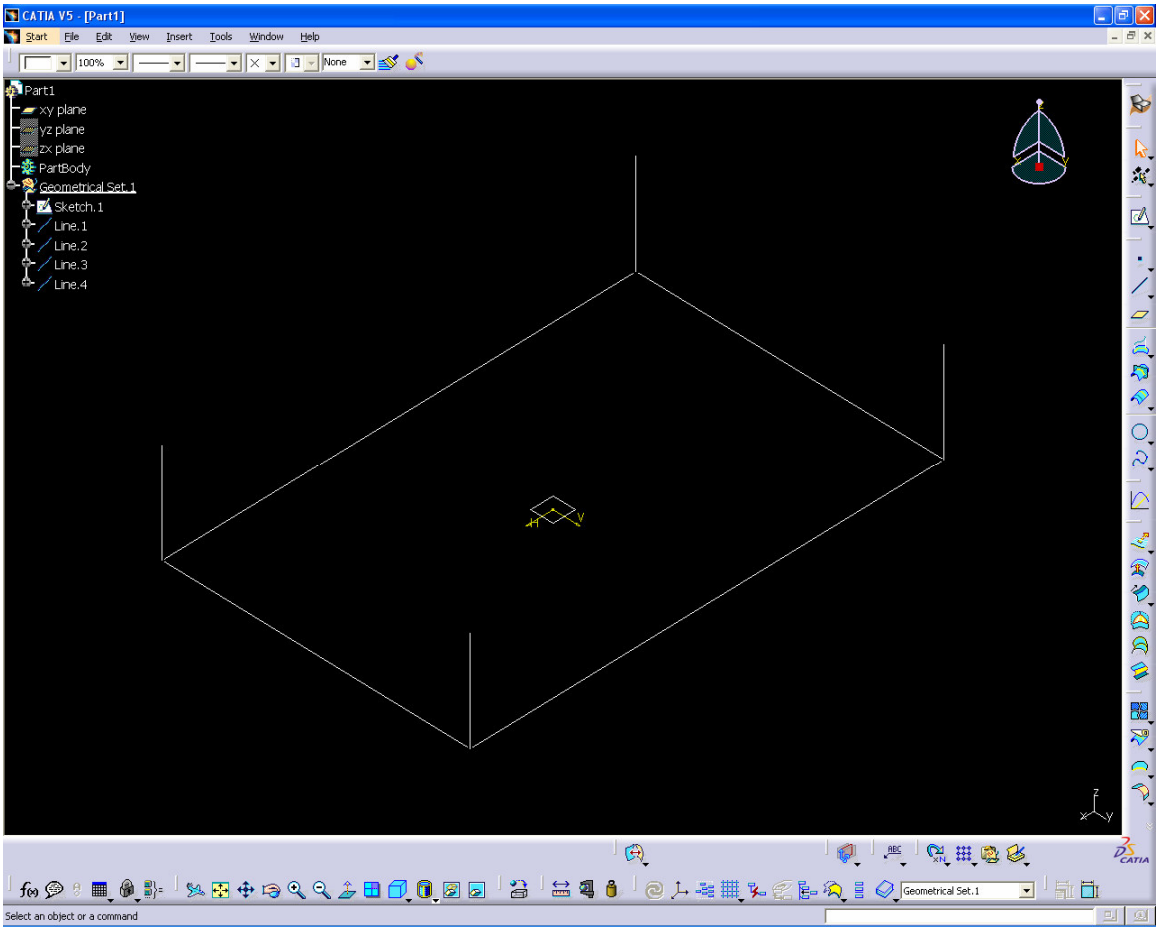
The result will look like this:



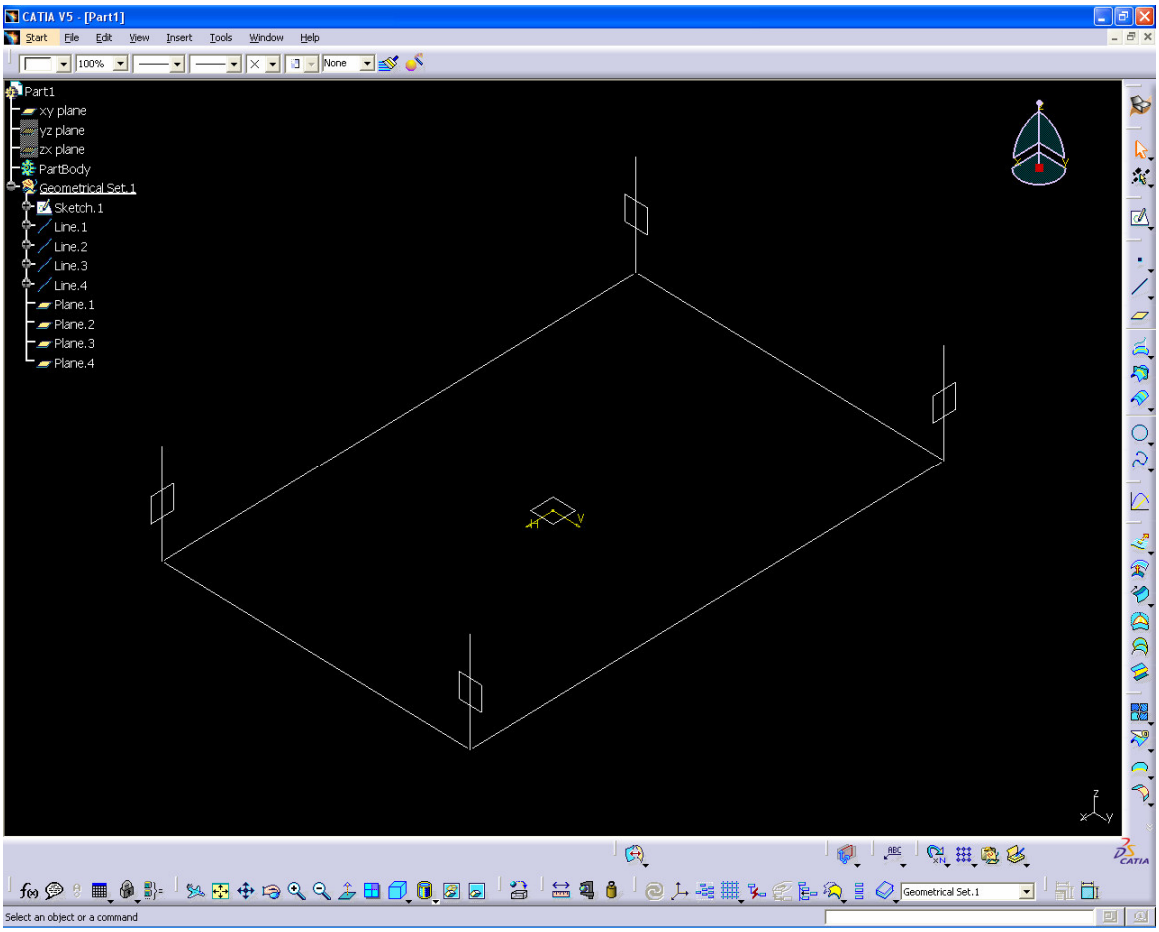
Exit the Sketcher.

NOTE: From this point on, DO NOT use the "Datum element" feature. This example requires history on all created components to function properly.

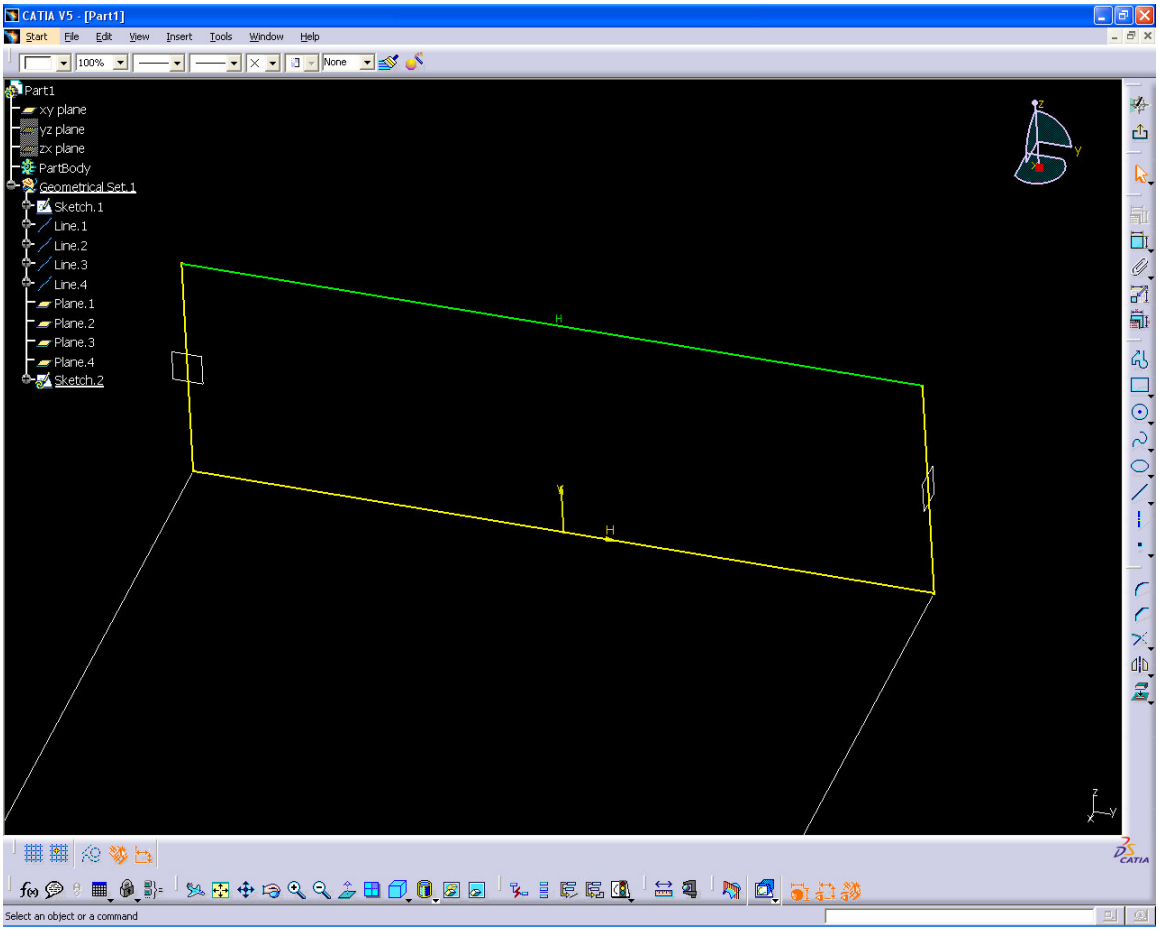
Next, put a line normal to plane XY, at all four vertices of the sketch. The order is not important. The result should look like this:



Now, add 4 planes, using the “through two lines” option. Add them beginning with the line shown at the “12 o’clock” position as the first line, and the “3 o’clock” position as the second line. Continue this process until you get this result:

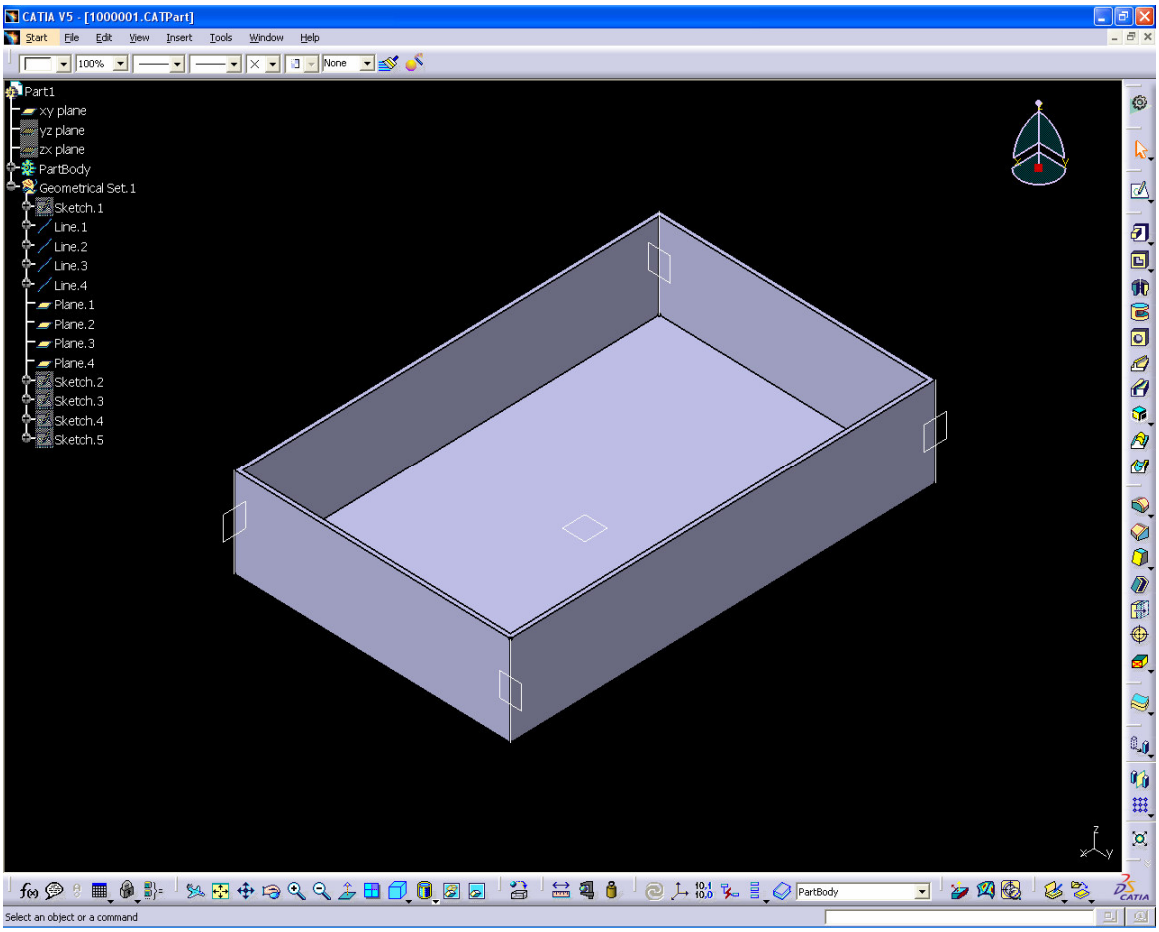


Beginning with the first plane created, make a new sketch. Once inside the sketch, project the 3 edges available – 2 lines, and one edge from Sketch.1. (Do NOT isolate the edges) Once they have been projected, draw a line using the endpoints of the projected lines, to close the box, as shown.

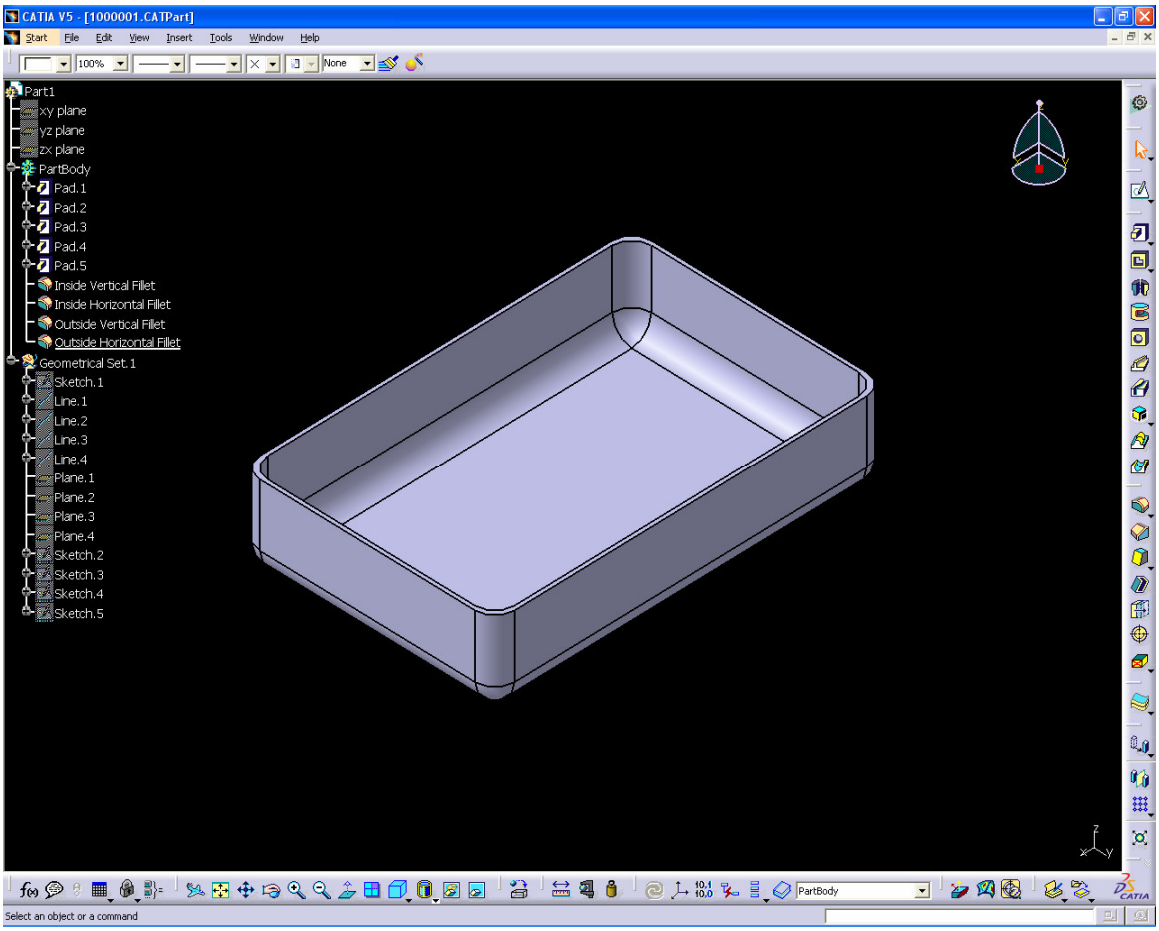


Exit the sketch, and repeat until all sides have been completed.

Enter the “Part Design” workbench, and extrude Sketch.1, with a thickness of 1mm. (direction arrow should be pointing to the “inside” of the box) Do the same on Sketches 2-5, until you get this result:



We will now add a series of fillets. Each fillet will be handled based on its location and orientation. We will add first the “Inside Vertical Fillet”, followed by the “Inside Horizontal Fillet”. Then, the same corresponding fillets will be added to the outside of the part, in the same order. Fillet size is not important, at this point. This example shows the fillet operations renamed in the tree. Your result should look like this:



At this point, the part design is complete. (remember to save the part) Now, we begin the task of creating and assigning parameters.

C. Creating Parameters

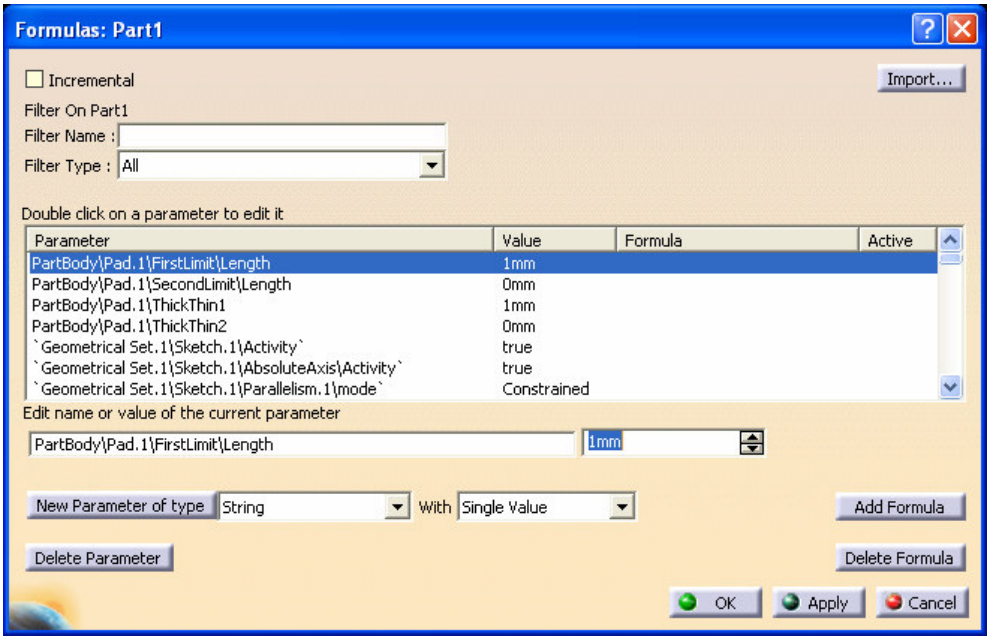
This section covers how to create user-defined parameters, to use in the part previously designed.

Let's revisit our pencil sketch again, and look at the list of features that we decided were important for this part:

1) Part Number (we will discuss this more later)
2) Length
3) Width
4) Height
5) Material Thickness
6) Inside Vertical Fillet
7) Inside Horizontal Fillet
8) Outside Vertical Fillet
9) Outside Horizontal Fillet

Begin by selecting the formula icon: 

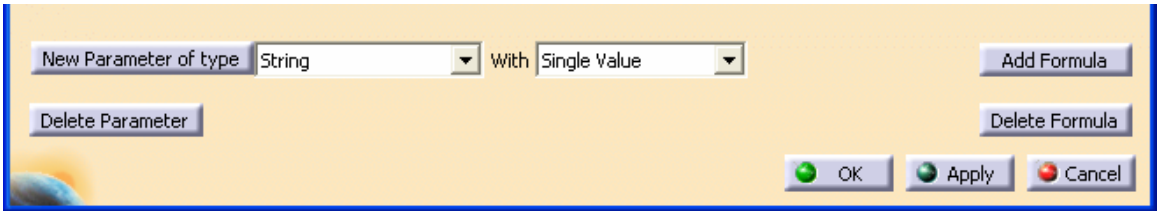
You will get a dialog that looks like this:



You will see a pre-populated list of system defined parameters. We will not deal with these at this time.

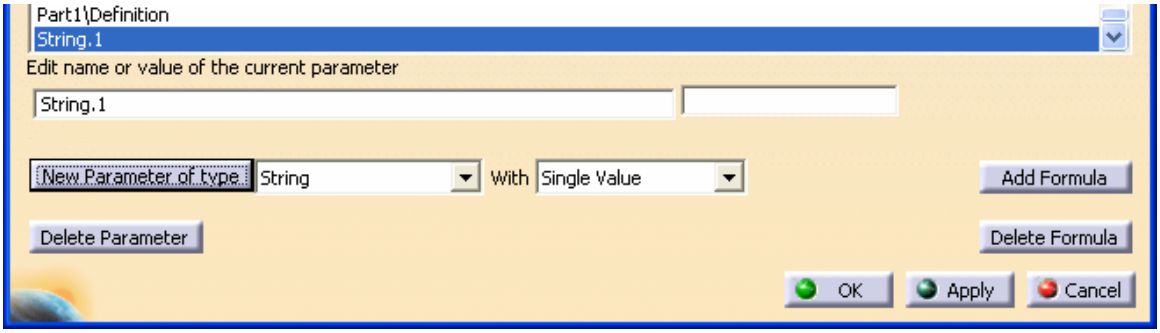
For now, we will begin by selecting the "New Parameter of type" button.

Next to the "New Parameter of type" button, there are two input fields. The first will define the type of parameter that we will create. The second field will determine whether the parameter will consist of a single value, or multiple values. For this tutorial, we will only be discussing parameters with single values.

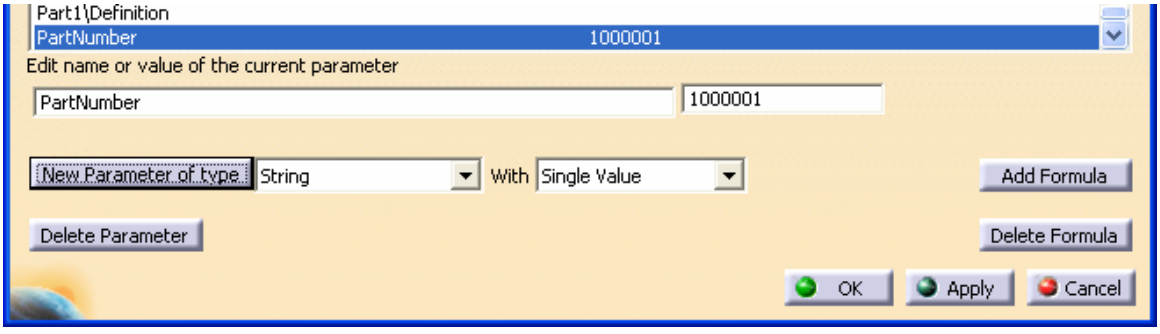


NOTE: There are many different parameter types available. They are too numerous for the scope of this tutorial, and will not be covered here. However, the parameter types used will be explained according to their use.

Set the parameter type to "String" (as shown) and "Single Value". String values are alphanumeric, and are not recognized mathematically. They are used for duplicating text values, or numbers that are not used in a mathematical context.

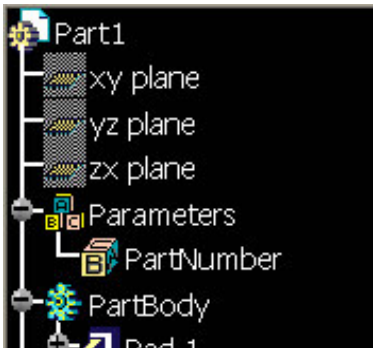


After the new parameter is created, select it in the list, and edit its values. Let's call the first parameter "PartNumber". You should end up with something like the picture below.



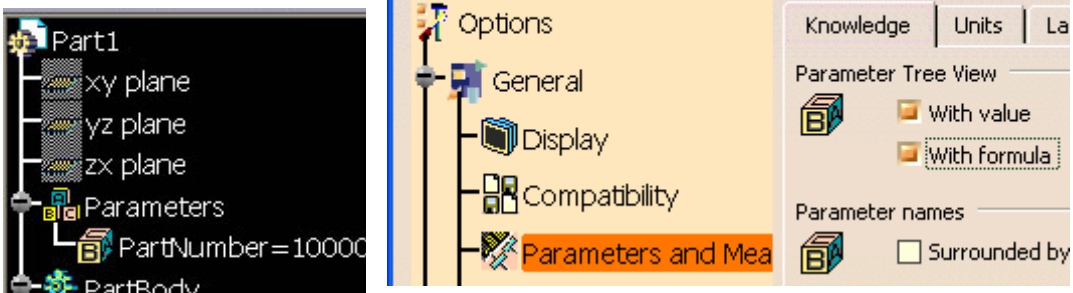
NOTE: "PartNumber" is a namespace used by design tables and catalogs in later tutorials. Copy it exactly as shown, with no spaces between words, and first letter of each word capitalized.

Now that you have finished that, you should have a parameter in the tree, that looks like this:



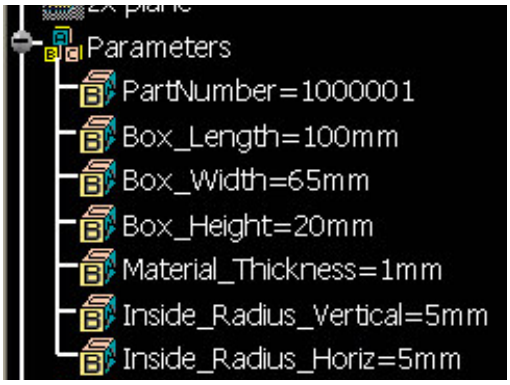
Tip: to view the tree with the parameter and formula values shown, you must first select the option, under **Tools » Options » Parameters and Measures » Knowledge**, and check the boxes labeled "With value" and "With formula".

Before option applied



After option applied

Continue to define parameters for the following requirements. The remaining values are all of the type "Length", as they control a feature of size. (includes radii and diameters)



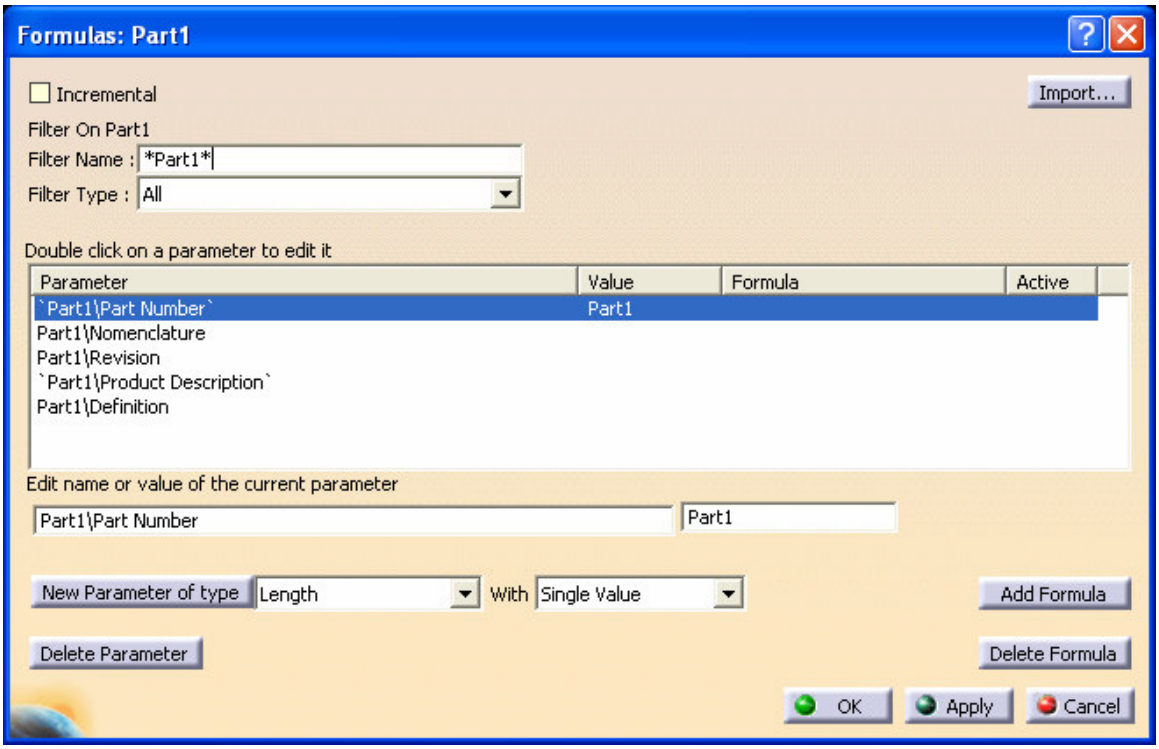
D. Assigning Parameters

Now that the parameters have been created, what do they do? Right now, they are not associated to anything, and in effect, do nothing. In this section, we will explore how to assign parameters to designed part features, and modify parts using the associated parameters.

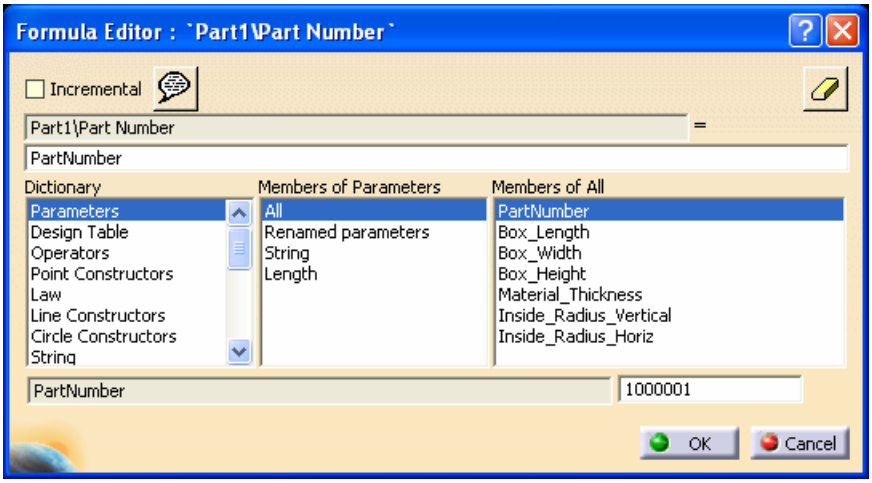
The first step is to figure out where the newly defined parameters need to go. Parameters are assigned using the Formula function. (used in previous steps) If you remember back to the system generated parameter list in the formula dialog, there were several parameter names, with a corresponding path. This is the realm we will be working in. Each of these paths points back to something that we created sometime between the initial file creation, down to the last fillet. We will locate the features that we wish to assign parameters to, and use our parameters to override the design input values.

The formula dialog is a bit intimidating at first. With some experience, it becomes relatively easy to decipher; however, there is a much quicker, and easier way to find a feature, than wading through the list.

Let's go through the parameter list, in order, starting with the part number. Initialize the **Formula** function. When the dialog appears, go to the field called "Filter Name". Here, we can filter data by any criteria. In our case, we have a value "Part1" in the tree, which is where we want to assign the "PartNumber" parameter. (default value - yours may vary) By doing a wildcard search, (*Part1*) we return the following parameters:

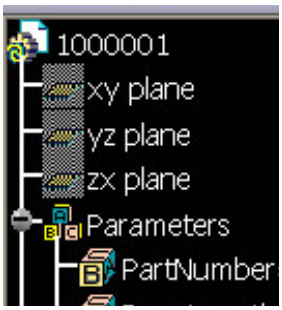


Now that we've located the parameter that we wish to replace with our own user-defined parameter, activate the "Add Formula" option.

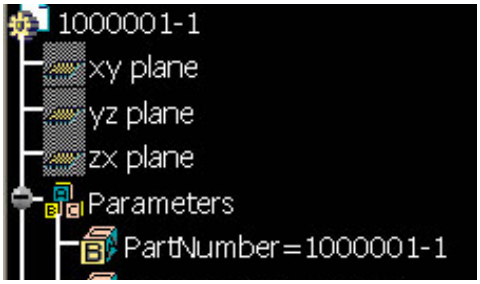


It would be easy enough, at this point, to find the "PartNumber" parameter in the list; however, it is a good practice to get into, to simply select the parameter from the tree, and it will be filled in the formula editor.

Press "OK", and notice the change in the tree:



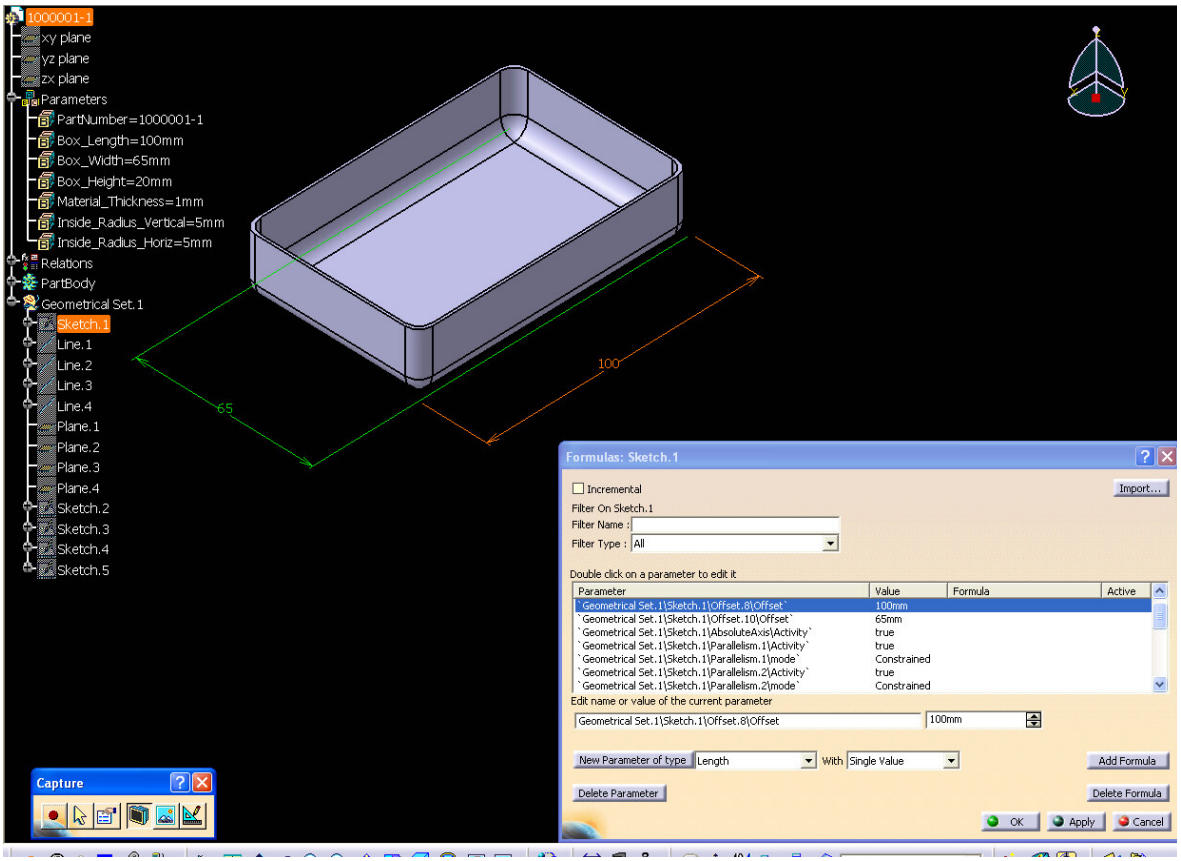
Our part number is now filled out in the tree. Let's change the parameter to see how it works. Double-click the "PartNumber" parameter, and change the value to 1000001-1.



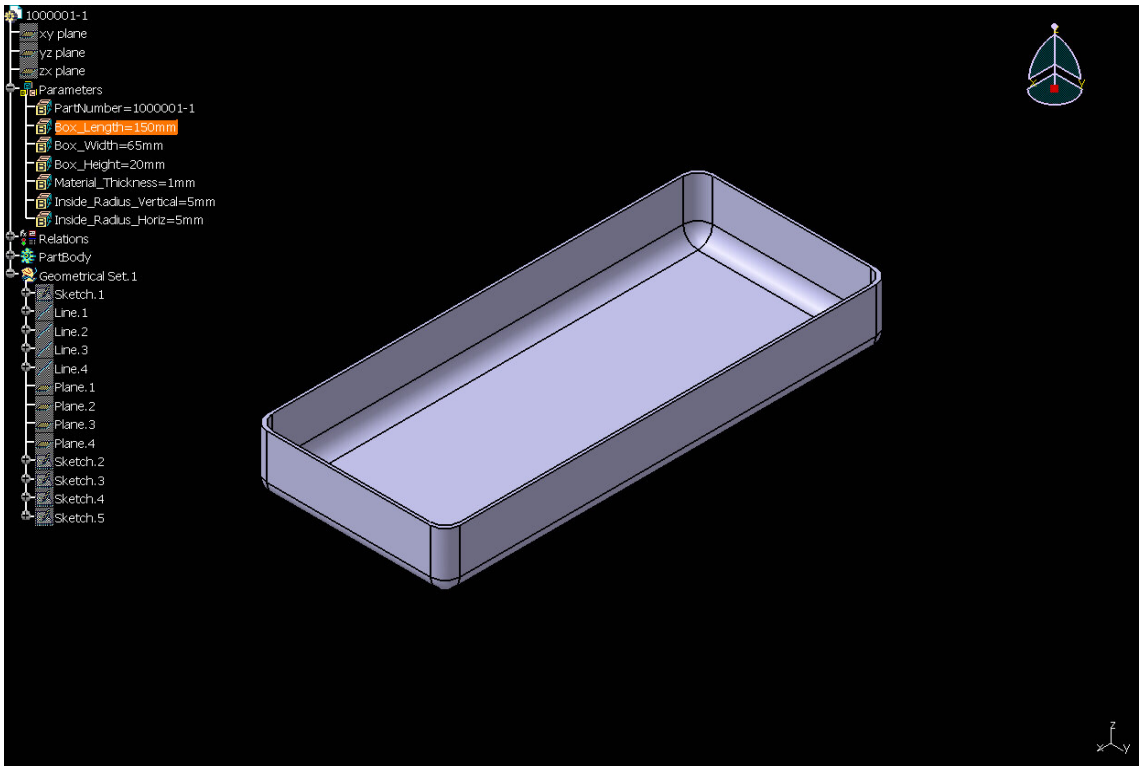
Now, let's move on to the numeric parameters. (Length values)

These are created in the same fashion, but we have a distinct advantage with features of size. Open the **Formula** function again. We will look for the Box_Length feature that we wish to assign our parameter to.

We know that the length is contained in Sketch.1, as this was the footprint of the box. We created lines at the vertices of this sketch. To find this parameter, it is as simple as selecting Sketch.1. In sketch one, our 2 constraints will appear. Select the one you wish to become the Box_Length parameter-controlled length. Next, simply press the "Add Formula" button, and select the "Box_Length" parameter in the tree, and press OK.



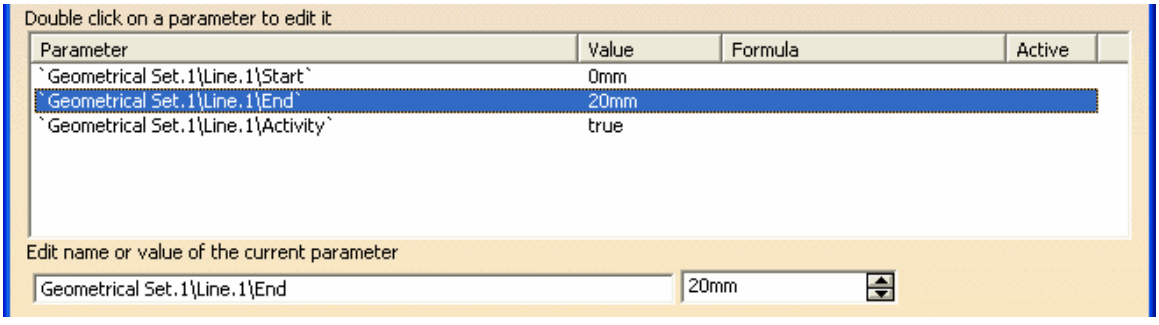
Test the new parameter by changing the number. If the part updates with the new dimension, you have associated the parameter correctly.



Repeat the same steps for the width of the box, and change the Box_Width parameter to verify the successful association.

Now, we will assign the "Box_Height" parameter to the 4 lines that we created from the sketch vertices. Remember, these 4 lines are projected into the sketches used to create the walls, and have parametric history - so they will update based on the current length value of the lines.

Open the **Formula** function. Begin by selecting (in the tree) the first line you created. You will need to select following parameter: (Notice the "Value" column – select the one with the length that you originally defined)

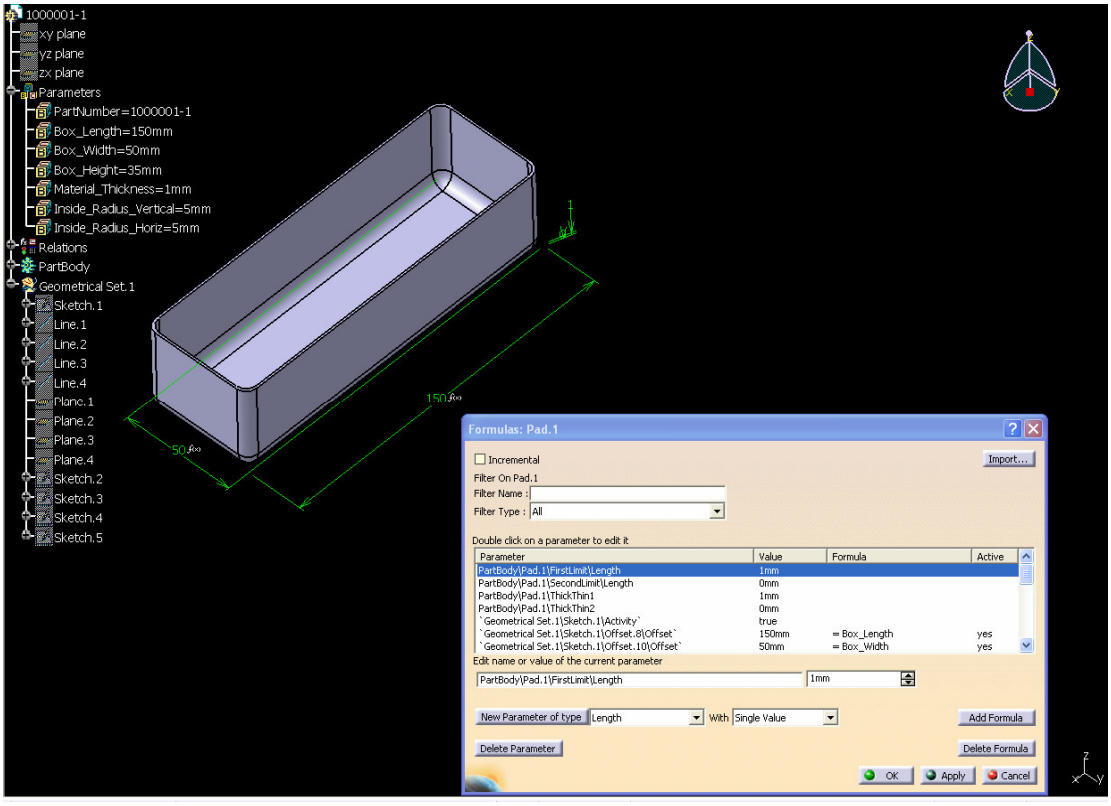


Once this value is selected, we will once again use the "Add Formula" option, this time inserting the "Box_Height" parameter. Press OK, and repeat this step for all 4 lines.

This will update all of the lines simultaneously, thus ensuring a uniform part height.

When you have finished, once again change the parameter value in the tree to view the result.

It's time to assign the material thickness. This time we will call up the **Formula** dialog, and go through the same steps we used on the lines in the previous step. This time, however, we are looking for the length value of the pad offset for each wall of the part. We only offset in one direction, so we will be dealing limit lengths. Select the first Pad operation in the tree, and the following dialog and representations will appear:



We have already mentioned that the limit length is what we are seeking. The value (in this example) is 1mm, and we can select it either in the formula dialog, or by picking the green representation, shown at the right upper right hand corner (3 o'clock) of the part. Once more, using "Add Formula" we can associate the "Material_Thickness" parameter. Press OK. This process must be repeated on all the material walls (Remaining Pad operations) to achieve uniform material thickness. Change the parameter value to verify.

TIP: It is a good idea to have the new parameter set to a different value than the current value of the feature, before association. (i.e. – if line length = 20, set Box_Height to 35) The part will update to the new value, thus confirming success.

The last parameters to assign are the inside fillets. Repeat the steps laid out in the last section, selecting the Inside Vertical Fillet and Inside Horizontal Fillet operations in the tree. Replace the parameter in the Formula dialog which contains the radius value specified at the time of creation, with the appropriate user-defined parameter.

NOTE: Do not modify the outside radii at this time. These features are covered in the next section.

This concludes assigning parameters. We will now put a formula in place, to control the outside radii.

E. Creating Formulas With User Defined Parameters

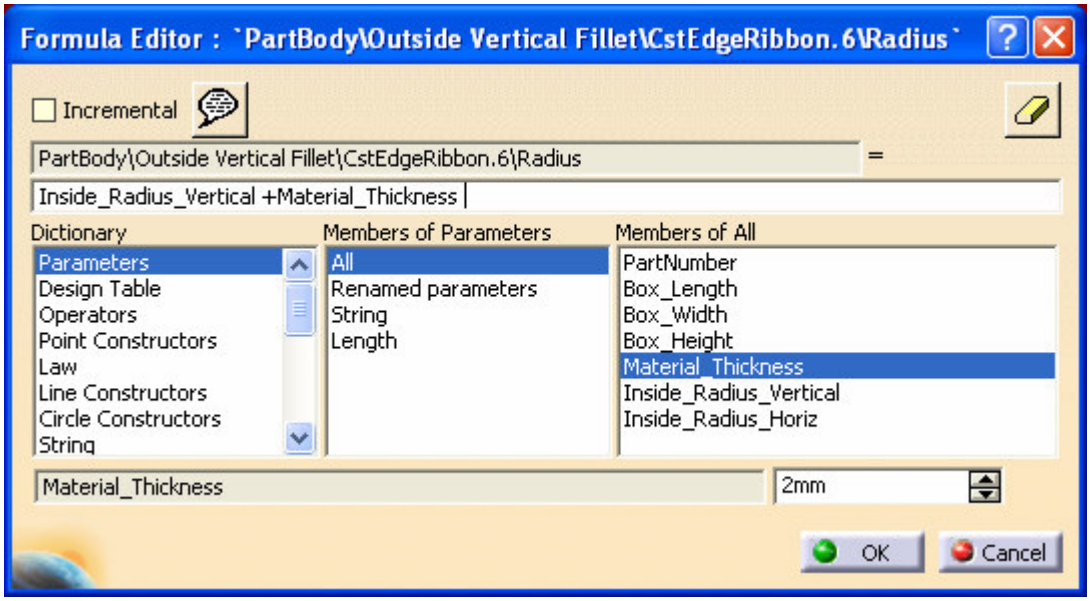
The last step in this lesson is to create formula which controls the radius of the outside fillets, based on the sum of the internal fillet and material thickness values. It is just as easily said as done

We will once more be visiting the **Formula** function. Access the formula command, and begin by selecting the outside vertical fillet in the tree. Next, pick the parameter which contains the radius value. (length) Select the "Add Formula" option.

To create the outside radius value in our formula, select the first independent variable, which is the inside vertical fillet. Do not press OK, at this point. We still have work to do.

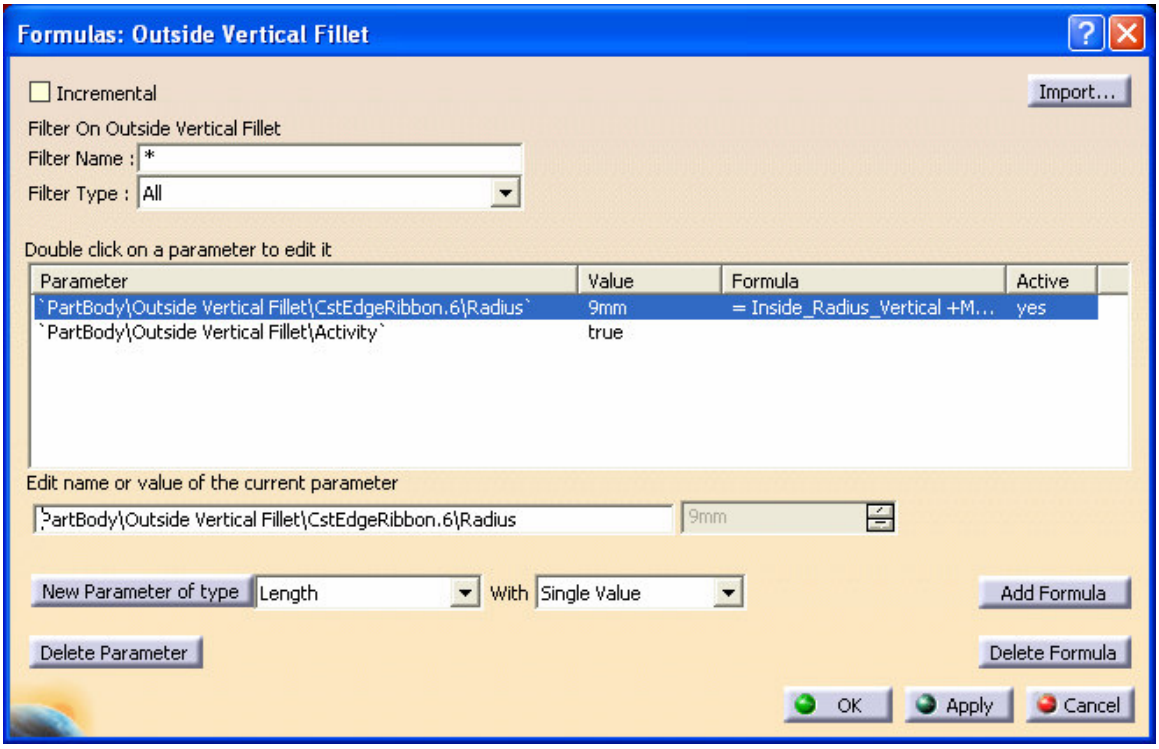
NOTE: Be sure that you base the formula on the correct fillet. The intention is to be able to independently control the fillet size for the 2 defined fillets. Incorrect fillet association – such as inside horizontal fillet, in the outside vertical fillet formula – will obviously cause negative results.

Now, we want to add the value of the material thickness to the fillet value. We will type a "plus" sign, (+) in the formula input field, directly after the parameter value which was populated by our selection. (from the tree) Immediately after typing the "plus" sign, select the Material_Thickness parameter from the tree. Your result should look like this:



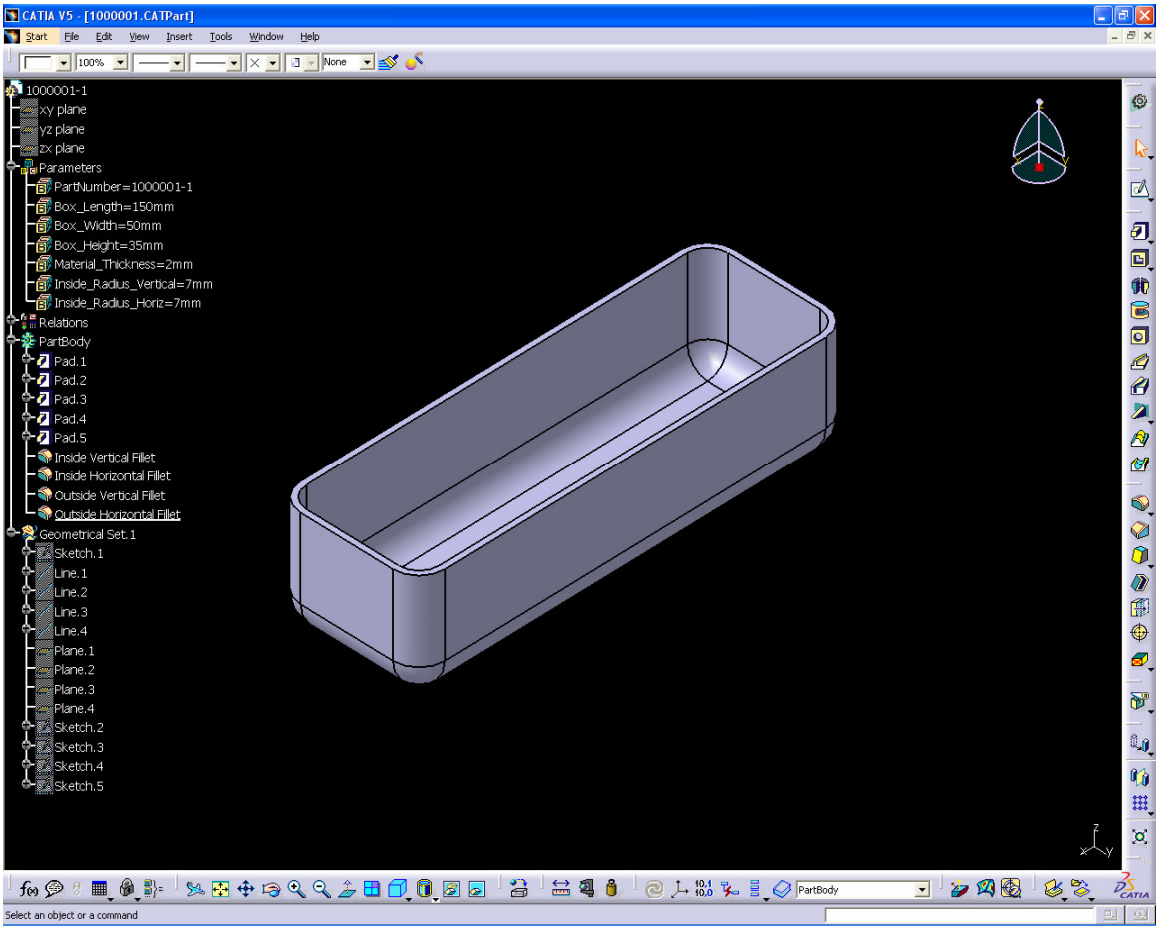
Press OK.

Now, notice the formula field in the Formula dialog. It has been updated with the formula that we just input. It should look like this:



Press OK, and repeat on the horizontal fillet.

Upon completion of this task, we have a part with user-defined parameters, and formulas driven by user-defined parameters. Take some time to analyze the part, and make sure that all relationships are as expected. Change the values, and explore what is, and is not possible with the defined parameters and values.



Congratulations! You have successfully completed this tutorial. Save this part for future reference.

IV. Getting Help

A. Online Documentation

For lingering questions, or related topics not covered in this tutorial, the best source of help is the online documentation, which comes with every release of Catia software, and is updated by service pack. The documentation is quite helpful – the author of this tutorial relies quite heavily on it, in fact. If it is installed locally on the user's machine or server, it can be found by pressing 'F1' on the keyboard. If not installed or pointed on the local machine, contact your Catia administrator for access.

B. Other

Should further questions arise, there are other options available. First, and foremost – when the help documentation fails, or fails to makes sense – check the many available Catia internet forums. Some examples: www.eng-tips.com, http://www.catiav5forum.de/, http://2htts.com/CATBlog. When these options are not appropriate, it may be beneficial to contact a consultant. For Catia design or consulting, visit www.cad-design-engineering.com. (NOTE: the author is associated with this firm, and reserves the right to advertise in this space, in exchange for the time and effort devoted to this document.)

V. Acknowledgements

Special thanks to all who provided their time and support (material and/or other) to this document. It should be noted that the following individuals contributed significantly, and assisted the author (Solid7) in bringing this tutorial to fruition. They deserve the respect and thanks of the Catia User's community at large, and anyone else who might find this document useful. Thank you.

Catiajim

Tsqman

Ferdo

For Questions and Comments, contact Solid7 - the CATBlog administrator.
Last updated on May 9, 2006

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