

MANAGING PARAMETERIZED PARTS WITH VARIABLE CUT DIMENSIONS

Abstract: The design of the custom cut parts, in order to cumulate the lengths of all inserted variations in an assembly is solved, using a case study of a threaded extruded bar, commonly used in mechanical design, and purchased as long bars, of fixed length. Using the facilities offered by the iPart concept in Autodesk Inventor, the main member definition parameters are defined and linked for the automation of design data usage in assemblies. Further, a more complex case study is developed, for multi customized cut dimensions and variable pattern features.

Keywords: custom parametrized parts, CAD, Inventor

1. INTRODUCTION

Similar parts that are coming in variable dimensions, properties or mounting configuration can be designed using the parametrization facilities, usually based on tabular data. Such table-driven iParts are common in parametrized CAD and are largely used in all industries, due to the easy handling of the data and easy iPart family member definition and its insertion into an assembly.

One particular situation is when one parameter, usually the length, doesn't need a particular value, and it rather should be obtained via custom cuts from larger samples or members. In Autodesk Inventor, an iPart table is a normal table, with fixed, predefined data, this problem might appear as not suitable for an iPart solution. The future length of the part should be able to take any value needed, and not just be associated with a number from the table, for that specific variation.

Fixing a custom parameter for an iPart (in this case, the length of a bar), can be done manually, operating directly on the parameters manager or the driving table. Still, this kind of customization can be automated and used further, to define variable lengths for the parts.

2. THE PARAMETRIZATION PROCEDURE

Usually, the iParts with custom cut length are to be defined from basic sketched, 3D developed as extrusions, revolutions or patterns. In either situations, a basic parameter management is required. From the parameters list, any of the values used to define the start-up body can be accessed and edited [1]. In order to obtain and automated operation, a new, custom, numeric parameter will be defined, to retain the length of the bar (or board) – with the name „length”. Any default value can be assigned to this new parameter, one that is associated with the real-life dimensions of the bulk bar can be chosen as reference (Figure 1).

An important observation refers to the ability to export this parameter, to be checked in the export column, in the parameters manager list. Following, in the modelling operation (in this particular case, an extrusion for a threaded bar), the spatial parameter will be not specified as a numeric value, but with the character string

„length”, the name of the custom new parameter defined previously (Figure 2).

Parameter Name	Unit/Typ	Equation	Nominal Value	Tol.	Model Value	Key	
Model Parameters							
d0	mm	12 mm	12,000000	0	12,000000		
d1	mm	length	30,000000	0	30,000000		
d2	deg	0,0 deg	0,000000	0	0,000000		
User Parameters							
length	mm	30 mm	30,000000	0	30,000000		<input checked="" type="checkbox"/>

Fig. 1 Definition of the custom length parameter.

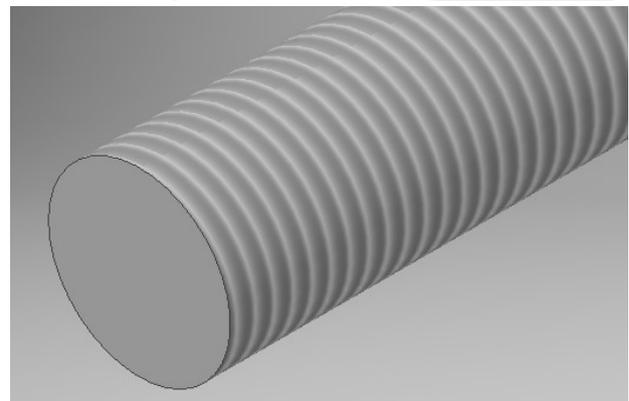
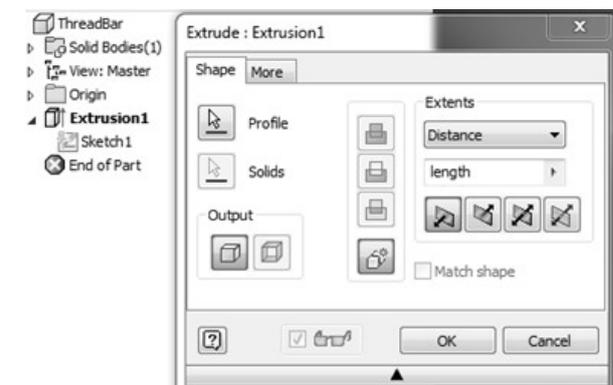


Fig. 2 Customization of parametrized extrusion.

The part is now ready to be transformed into an iPart, which can be easily done from the Manage tab/ create iPart. While, usually, the iParts have different members,

representing variations, described in different rows in the table. The one to be created will have only one member, described by a sole row in the drive table. Special attention should be given to the file name associated with this unique member, due to its specific [1].

Regardless from the dimensions of the future custom-cut part, the part name in any assembly in need for this specific type of component should be the same, as its place and role is the same in the assembled structure. Each time a custom-cut part will be used, Inventor will create a new file, for each new derivative, in the same folder. As consequence, it is a good practice to use the part number in the file name [2], with a basic flag at the end, indicating the basic file which contains the general description of the geometry, and this flag can be a zero „0”, for example, if the part number is ThreadedBar, the file name will be ThreadedBar-0. An incremental number will be added to the end of the file name, so that, each custom variation will have a correspondent file name, differentiated by the final number, as an incremental flag.

In the parameters column of the drive table, the length parameter should be right-click selected from the table header and declared as a custom parameter. The corresponding column will be highlighted, signifying that this parameter will be defined by the user, at the moment of any new variation definition in the assembly. The iPart is created and ready to be used (Figure 3).

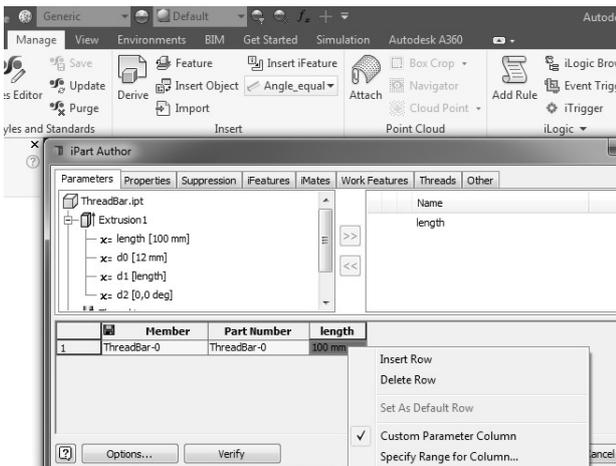


Fig. 3 Linking the data into the member properties.

In order to deepen the automation process of the usage of custom-cut iParts, one more action is recommended, but not mandatory. From the iProperties dialog box, in the Project tab, the Description field should transfer the same information for the plethora of variations that may occur, as the part role in the assembly is the same, so it is advisable to be filled in at this stage. Also, one might want the custom-length parameter to be contained in the variation data and automatically displayed. In the stock number field, an explanatory text can be added, along with the variant custom parameter, in chevrons. The string will contain the equal sign, explanatory short text and the parameter name, as, for example „= custom cut at<length>” (Figure 4). Inventor

checks the current value of the length parameter, and will display in the field that value, followed by the current units. In general, the units can be changed from the Tools tab on ribbon, Document Settings, and it is more efficient to access for this change the property – custom format for the parameter directly right-clicking on the parameter in the Parameters Manager table [3].

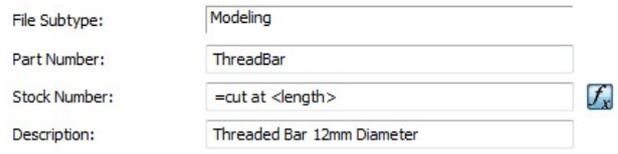


Fig. 4 Linking the data into the member properties.

Before using the new iPart in and assembly, a final intervention on its Bill of Material (BOM) properties, in Document Settings is needed, mainly because, in general, this type of parts are purchased, so, for the sake of the PLM, it should appear as such in the BOM structure, rather than having a „normal” behaviour. Also, the Base Quantity should be changed from „each” to the export parameter „length”, as this is the purchasing criterion (Figure 5). At this moment, when Inventor will quantify the variation of this iPart in an assembly, it will specify the length, which is the important data from the functional point of view.

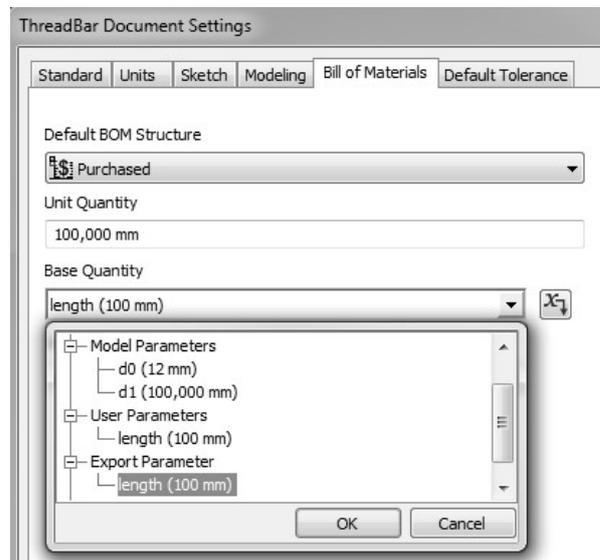


Fig. 5 Adjusting the BOM data.

3. USING THE CUSTOM PARTS IN ASSEMBLIES

In assemblies, the customized cut dimensions parts can be inserted as any iParts, while the drive table will require defining a value for the length parameter and a new file ThreadBar-01 will be created in the same folder as the parent member. File duplication for the variations is excluded, due to the member name algorithm (Figure 6).

The main problem is to cumulate the lengths of all inserted variations, because, for an assembly, it is necessary to know the total bar length to be purchased, quantified from all the members of that particular part.

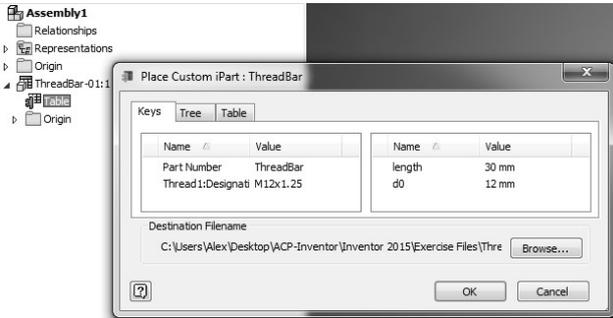


Fig. 6 Table driven variation member inserted in the assembly.

For this particular case study, in the assembly were inserted three variations with the lengths of 250, 300 and 400 mm. Because of the BOM unit defined in the parent member definition file, a BOM view defined in a drawing containing the three members will report the total, cumulated length of the variations, seen as one, single part, which simplifies a lot the assembly structure and the metadata associated (Figure 7).

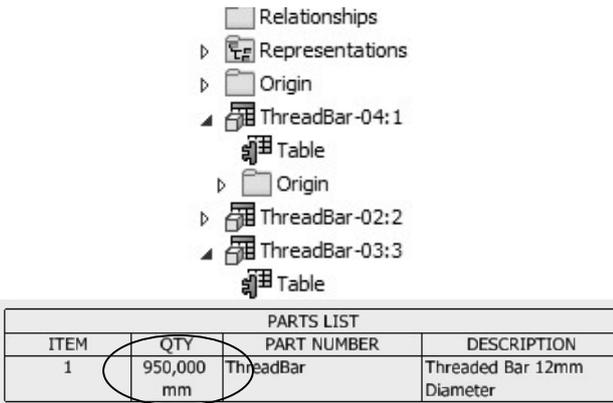


Fig. 7 Total length reported in BOM for all member variations.

It is useful when the value obtained cumulating the lengths of the components as custom cut variations is displayed in units closer to the purchase format, for example in meters. In this case, formatting the BOM column corresponding to the quantity is achieved by opening the dialogue for changing the units and precision, as in Figure 8.

4. TOTAL LENGTH MEMBERS IN FRAMES

The total, cumulated length of the variations can be successfully used in frame assemblies, where, often, the members are identical or just have different length. For the same material and sectional dimensions, it is important to have the total length for purchase. The custom cut part parametrization brings the solution to the total amount in the BOM. The frame represented in

Figure 9 was built from a planar sketch, with basic symmetry constrains generated from a linear pattern. The 20x20x2 square members were placed from ISO standard package available in the application. The ends of the members were trim/ extended with respect the correspondent faces, the mitered at 45°, with no gutter.

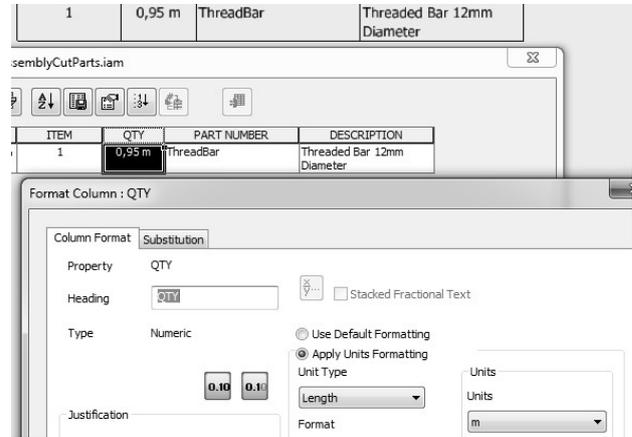


Fig. 8 Format of the length reported in BOM.

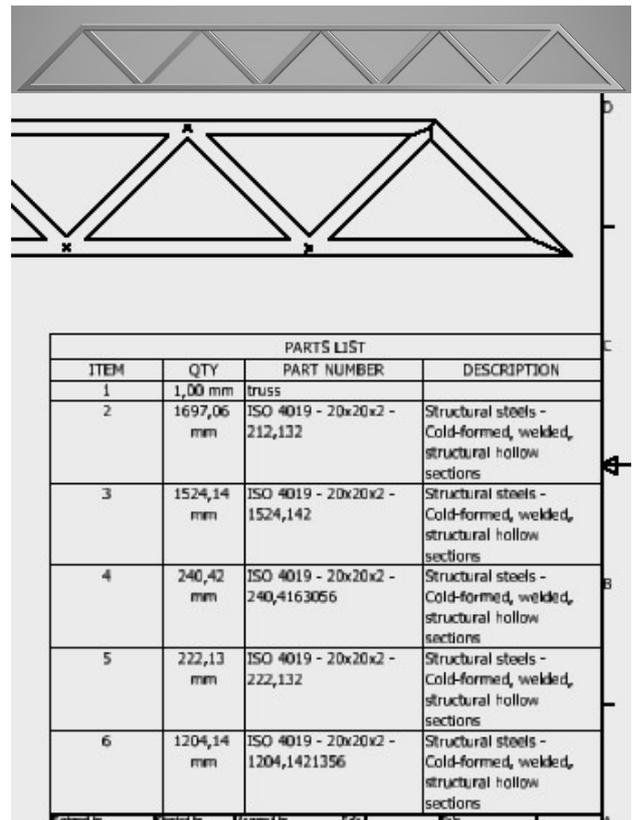


Fig. 9 Total length reported in BOM for frame members.

5. CUSTOM CUT PLASTIC PARTS

Custom cut plastic parts are frequent and automated iParts can respond to the need of accurate reports for dimensions and even areas to be used in assemblies. The challenge is to insert the features the automation process. Conditional custom cuts, where certain repetitive features

should be fully included in the cut length or width, expose the need to include the pattern parameters into the iPart definition.

In this case a special resin grilled plate needs to be cut at variable dimensions, to be used as a virtual component in an assembly, where it's role is to be hot welding material between two surfaces of variable shape and dimensions. The grill needs to be cut in rectangular shapes (Figure 10), and mounted for curing in the assembled position. Both rectangular dimensions will be defined as custom parameters and associated in the iPart table to a generic part name "Grid-00" (Figure 11). When called into an assembly, in the BOM structure, this virtual part will be reported based on the cumulated lengths and widths and will be easily accessible for area calculation.

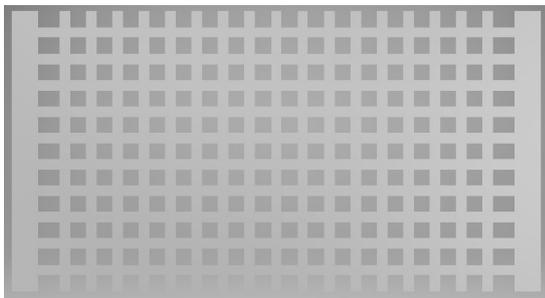


Fig. 10 Custom cut with conditional featuring.

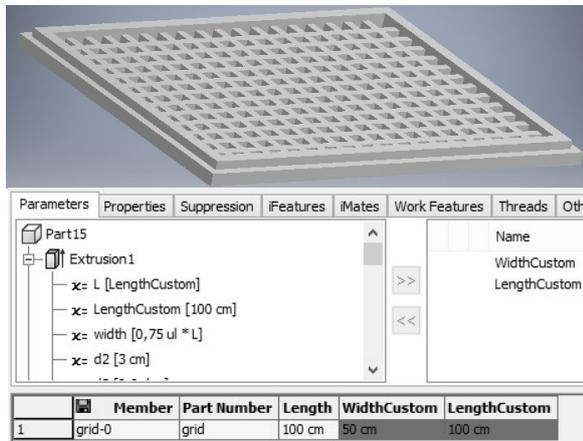


Fig. 11 Plastic iPart with parametrized grid.

The parameters are brought into the BOM with the quantity definition related to the parameters and not to the pieces (each). In order to achieve this, a rule is embedded in the iPart, that will run automatically in response to changes in parameter values, as they will be used as variables (Figure 12).

The logic behind the rule is to check the custom cut lengths and widths and to compare them to an integer of grill feature step, then to display a message with the closest cut dimensions available around the first proposed value.

The user will be informed about his cut length choices and decide which one is more appropriate for the optimization of material usage.

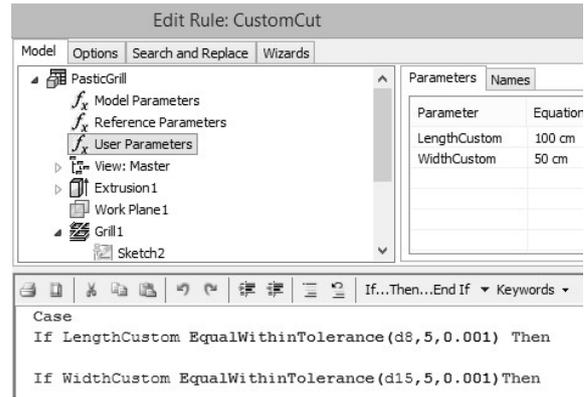


Fig. 12 iRule editor.

5. CONCLUSION

Capturing the design intent in various scenarios is one of the triggers of the automated design workflow. The iLogic concept offers a wide range of possibilities for designing variations and flexible assemblies. This paper presented an efficient automation method, useful for recapitulation in the bill of materials. The purchase becomes easy to handle, as the final reference are specific for each type of bulk (bar, plates) to be cut at precise dimensions.

The design data of an iLogic Inventor part is edited via the parameters manager and the metadata stored in the part properties, so that, at any placement in an assembly, will report the exact, correct custom length, and, even more, the total length to be purchased for all the variants of that particular part in the assembly.

REFERENCES

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