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The Parametrical Design of the Parts from the Same Technological Family

The paper presents a methodology used for rapid design of part solid models for the same technological family. The parameterized models, which allow the rapid generation of a new part with other dimensions / shape, can be obtained by associating variable parameters to the dimensions or to the shape of a part. The methodology was exemplified on a flange type part. The model was design using Autodesk Inventor software and the introduction of the values for the parameters was made in two variants: directly from the Parameters dialog box of the CAD software and from MS Excel file.

Keywords: CAD, parametrical design, technological family

1. The drawing parameters

The parameterized models allow a rapid generation of the parts from the same technological family. This paper presents the methodology of the parameterization applied for a flange part. The flange geometry and the variable dimensions are presented in figure 1. Besides the dimensions was parameterized also the number of the flange holes.

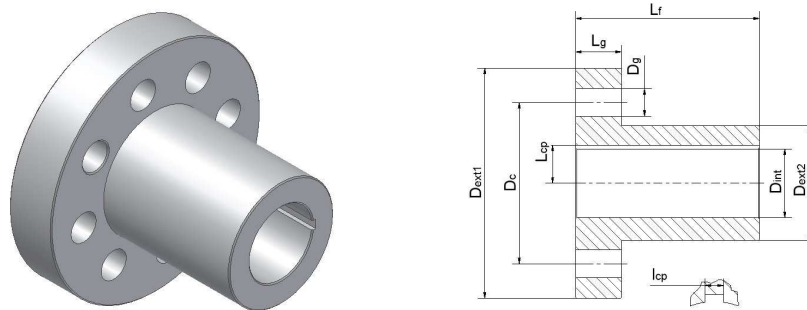


Figure 1. The flange geometry and the parameterized dimensions

The symbols of the parameters were chosen according to the Romanian name of these features. The symbols signification is: D_{ext1} – the outside diameter of the neck flange; D_{ext2} –the second outside diameter; L_f – the length of the flange; L_g – the length of the neck flange; D_{cg} – the diameter of the circle which define the centers of the flange holes; NG – number of the flange holes; D_g - the diameter of the flange holes; D_{int} – the inner diameter; L_{cp} – the dimension from the central axis of the flange to the bottom of the grooved grooved wedge ; l_{cp} - the width of the grooved wedge.

Parameters are used to define the size/shape of features or to control the relative positioning of components within assemblies. It is possible to define three types of parameters:

- *model parameters* (usually noted with $d_1, d_2...$), created automatically by the software when the features are dimensioned;
- *user parameters*, defined by the user in the CAD software;
- *external parameters*, imported from an extern file;

The first step of the application was the insertion of the user parameters for the flange (figure 2) through the *Parameters* window activated by *Parameter* command. It is important to set the correct unit of measurement for each parameter; if is not a correct units correlation between the model parameters and the equation associated to the user parameters (*equation column*) the software will generate a valid model. For unitless parameters (example: the number of holes) the selected value for the *unit column* is *ul*. In the *equation* column can be introduced a nominal value or an equation, using the previously inserted parameters.

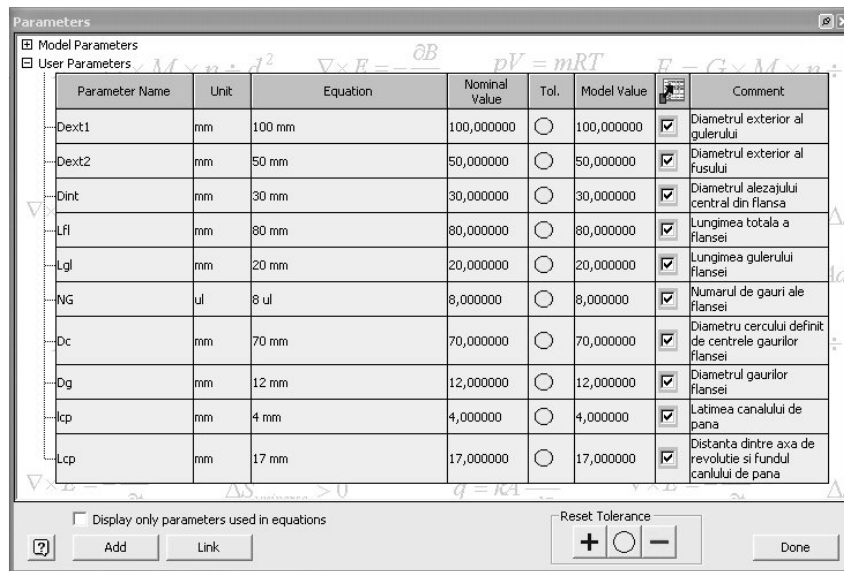


Figure 2. The definition of the user parameters

2. The part model

The flange model conception starts from the sketch presented in the figure 3.

When the sketch is created, the computer associates a *model parameter* to each feature dimension. With the *General dimension* command the authors defined a link between the model parameters and the user parameters.

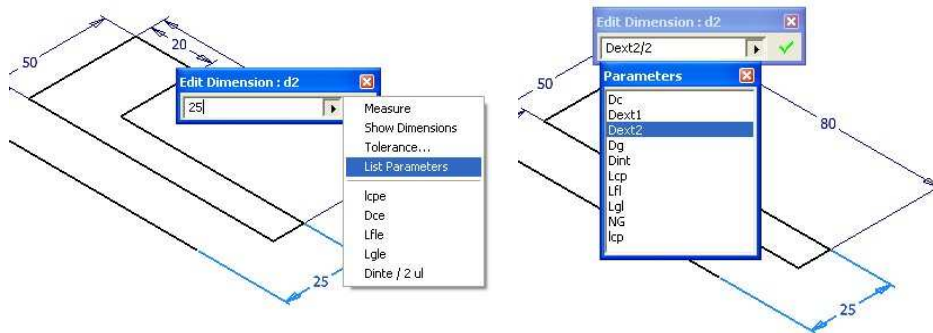


Figure 3. The first sketch and the link between *Model parameters* and *User parameters*

The closed shape will be rotated around the revolution axis (figure 4). The structure of the *Revolve* command allows the definition of one parameter, which is the revolution angle. For all the flanges this parameter has the same value (360 degree) and for this reason the authors didn't define a user parameter associated to the revolution angle.

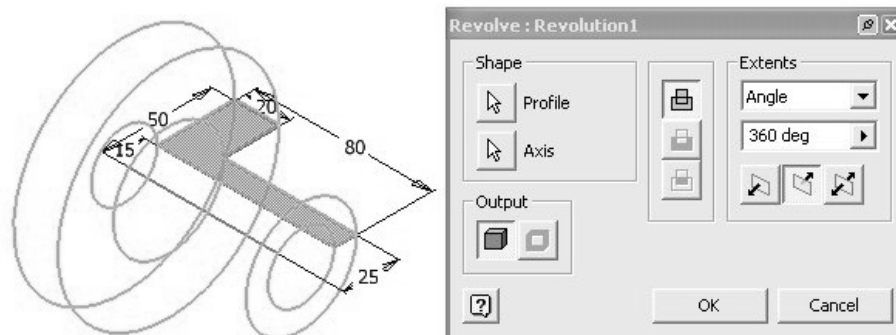


Figure 4. The *Revolve* operation applied on the first sketch

3. The holes number parameterization

The design of the flange holes was made in two steps. Initially was designed one hole with the next sequence of commands: the draw of the circle which define the centers of the flange holes (with *Circle* command), the hole center positioning (with *Point Hole Center* command) and the 3D hole execution (with *Hole* command and D_q parameter).

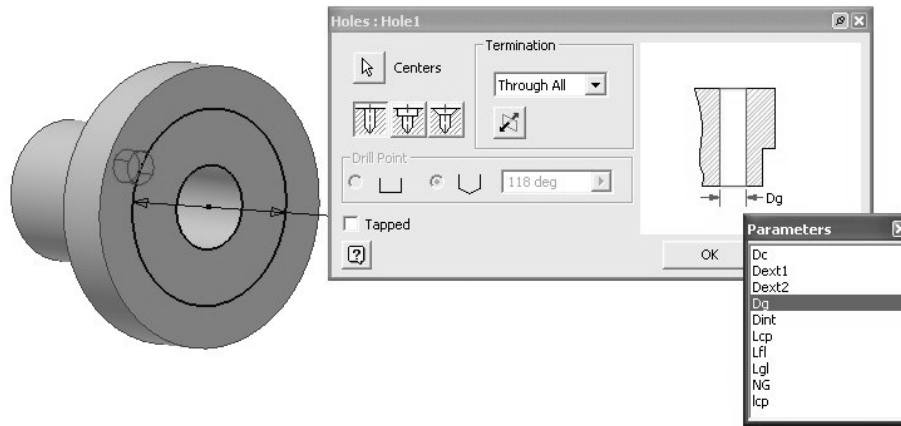


Figure 5. The *Holes* command applied to generate the first hole

In the second step a circular array of holes was generated. The command *Circular Pattern* multiplies an existent feature using two parameters: the rotation angle and the numbers of items. A *user parameter* (the number of flange holes) was used to define the number of items input (figure 6).

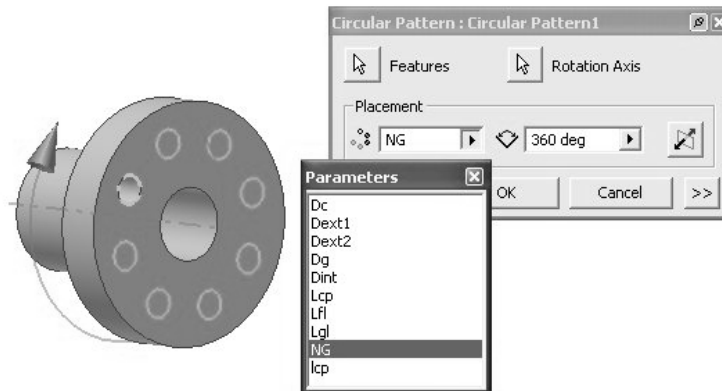


Figure 6. Multiplying the holes using *Circular Pattern* operation

4. The grooved wedge

The grooved wedge profile was defined by a new 2D sketch, where two dimensions have been parameterized: the dimension from the central axis of the flange to the bottom of the grooved wedge (L_{cp}) and the width of the grooved wedge (l_{cp}). The *extrude* operation with *cut* material option was used to create the grooved wedge (figure 7).

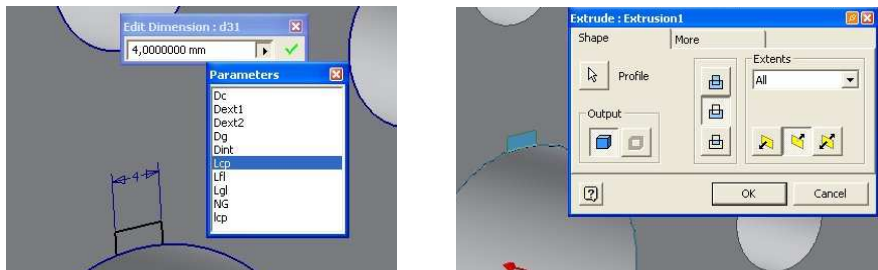


Figure 7. The draw of the grooved wedge

5. The values introduction

In the *Equation* column of the *Parameters* window can be introduced a nominal value or an equation using the previously inserted parameters. The value introduced here may be changed to generate another part of the same technological family, with different shape or dimensions.

If the parameters values are stored in a database file, the designer will transfer this data to Autodesk Inventor via MExcel with *Link* button (figure 8).

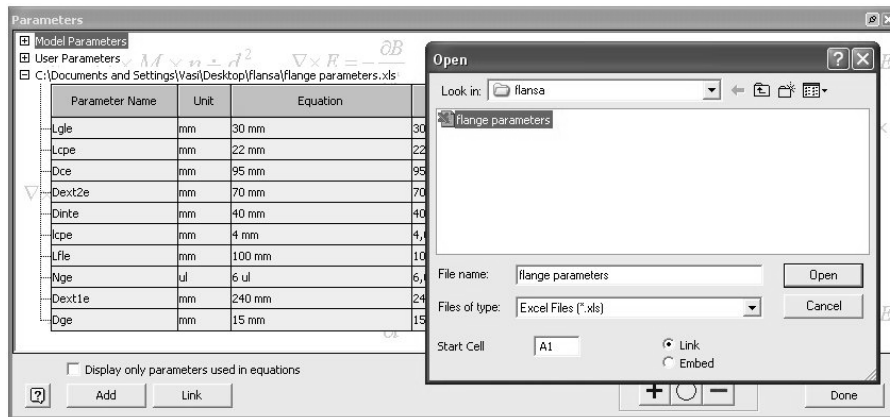


Figure 8. The link with the Excel file

The structure of the MSEXcel file must be the same with the structure of the model parameters box (figure 9).

	A	B	C
1	Dext1e	120 mm	
2	Dext2e	70 mm	
3	Dinte	40 mm	

Model Parameters		
Parameter Name	Unit	Equation
d0	mm	Dext1e / 2 ul
d1	mm	Dinte / 2 ul
d2	mm	Dext2e / 2 ul

Figure 9. The Excel file structure and the equation inserted in Model parameters

6. Conclusions

The 3D models parts of the same technological family can be rapid generated through the parameters (figure 10).

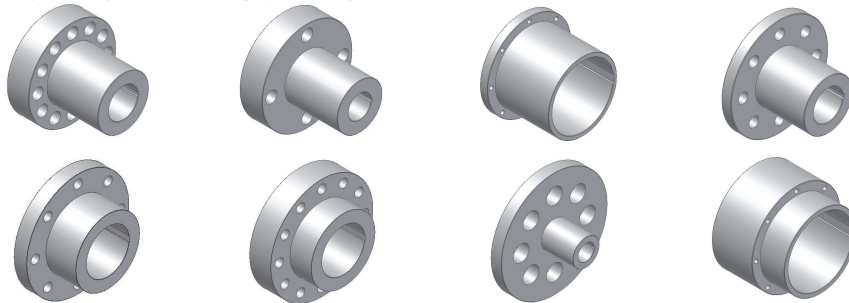


Figure 10. Flanges generated from a base model

References

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