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DESIGN OF AN ELECTRIC MOTOR IN CASE OF THE PRODUCTION OF SMALL AND UNIQUE SERIES

Abstract: After we created in the first article the parameterisation application and the main dimensional parameters, the purpose of this theme is to design the electric motor using CATIA V5 after the engine parts structure. The electric motor components will be designed parameterized to make more easily update when the electric motor functional parameters will be modified. In other words, the purpose of the project presented is reducing the design time and, at the same time, the costs, using the mix between two programs having different structures.

Key words: 3D modelling, parameterized design, CAD, programming, GUI application, cost and time reduction.

INTRODUCTION

During my training in the master program in Engineering Graphics and Design we have highlighted the collaboration between two different programs – CATIA V5 and MATLAB to reduce the design time of a three-phase cage-rotor induction motor. By using this procedure, the product-design team can obtain different projects in a brief time, changing only the performance parameters of electric motor. This method can be used in case of the production of small and unique series.

We talked about the reason why it was used the MATLAB programming language, but also the steps needed in order to create a parameterization function using this programming language [1], [2], [5].

For starters, it was necessary to create a script based on formulas, but also standardized dimensions to achieve the electrical motor's dimensions.

The next step was to create a graphical user interface that can be used in an application without having trouble in using.

Also, the links between the graphical user interface components and the main function have been created.

With these, we were able to create an executable application that can be run by any user.

We have built the structure of the engine parts which will be projected.

Further, there will be presented the next steps in designing the component parts.

We also imported the dimensions of the engine into the Catia design programme in order to achieve a parameterized design.

After designing the engine components, we will modify the engine's functional parameters to get new dimensions, without projecting the parts from the start.

2. THE DESIGN OF ELECTRICAL MOTOR PARTS

The proper design of the electric motor follows after creating the structure of the component parts of the electrical motor and after the connection between the application of the parameterisation function and the structure of CATIA parts have been made.

The components to be designed are (fig. 1):

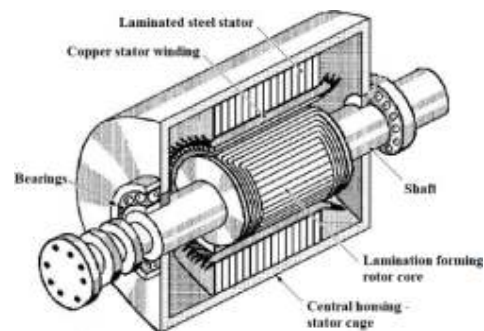


Fig. 1 The electrical motor components [7].

- The rotor assembly which consist of:
 - shaft; lamination forming rotor core; aluminium end rings; aluminium bars; the ventilation propeller [3].
- The stator is composed of:
 - laminated steel stator core; copper stator winding [3].
- The stator case which consist of:
 - the central housing; the front and the rear engine cover – the end shields; the two bearings positioned on the shaft inside the two end shields; the fan guard cover.

For the design of the engine parts we will initially draw the sketch of each piece, being dimensionally constrained by the parameters entered in the table.

The first outline was that of the shaft (fig. 2) built into the body of the first component of the rotor assembly.

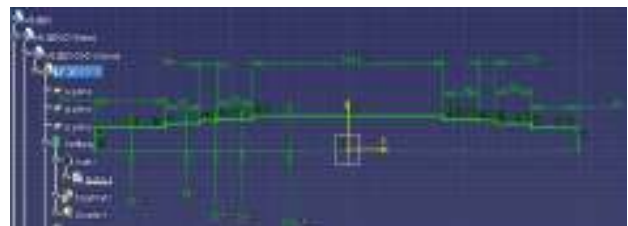


Fig. 2 Shaft sketch.

The sketch was made on the xOz plane because for the 3D dimensioning of this piece, the Shaft tool was used around the Ox axis (fig. 3).

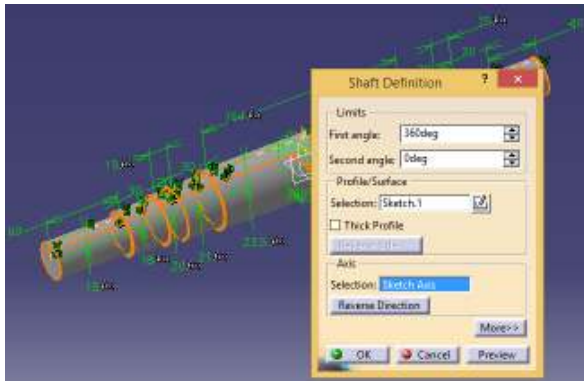


Fig. 3 Shaft 3D design.

After that we applied a 2mm bevel with a 45° angle to the shaft ends faces using the Chamfer tool as can be seen in figure 4.

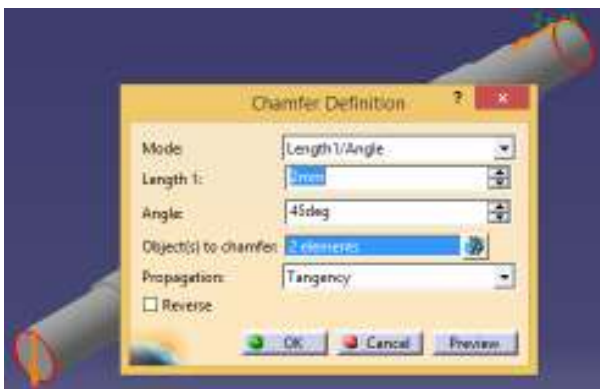


Fig.4 The chamfer applied on shaft ends.

It can be seen in the sketch some dimensions that have a distinctive function sign compared to the other dimensional constraints (fig. 5).

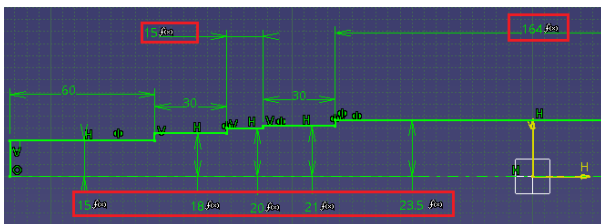


Fig. 5 Parameterized dimensions.

This indicates that dimensional constraints have been made with formulas using the imported data. The constraint was done as it can be seen in figure 7, by right-clicking the mouse on the size value after applying a dimensional constraint from the toolbar of the same name: *Constraint* (fig. 6).

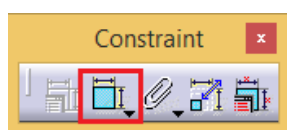


Fig. 6 The Constraint tool.

After completing the first steps as shown in figure 7, a formula editing window will be displayed where we will enter the appropriate dimensions according to the projected piece, by selecting Parameters → Length after which the desired size is selected from the list, as shown in the second step of figure 7.



Fig. 7 Creating a parameterized dimension.

Figure 8 shows the result of the 3D dimensioning of the shaft after applying all dimensional constraints and machining tools.

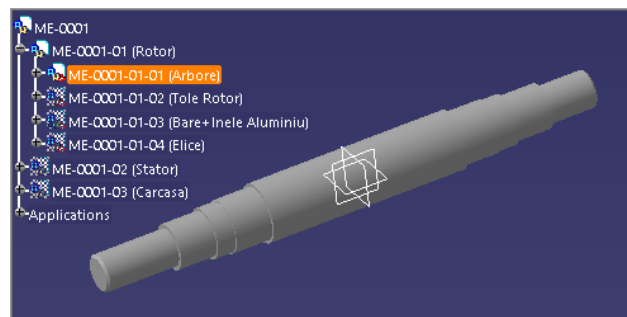


Fig. 8 Shaft 3D design.

The rest of the rotor assembly components are built, parameterized and designed according to the shaft design model, using the necessary tools according to the design plane.

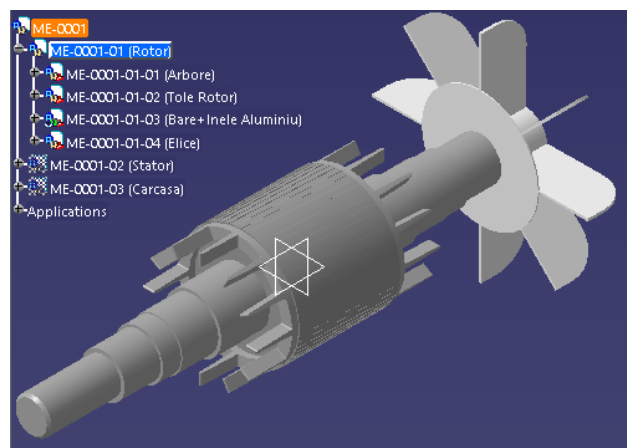


Fig. 9 Rotor assembly 3D design.

The rotor assembly design is shown in figure 9 and consists of the following parts: the shaft - figure 10 a), the lamination forming rotor core - figure 10 b), the aluminium bars shorted by the two aluminium end rings - figure 10 c) and the ventilation propeller - figure 10 d).

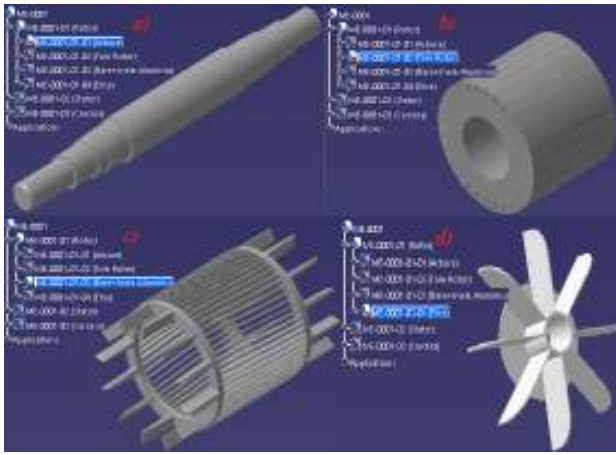


Fig. 10 3D design of Rotor assembly parts.

In figure 11 there are presented the stator assembly components: the laminated steel stator core – figure 11 a) and the copper stator winding - figure 11 b).

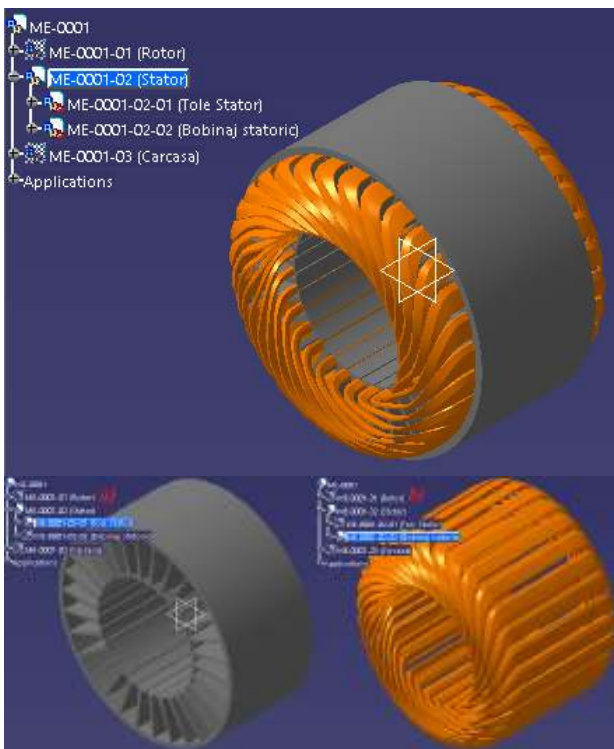


Fig. 11 Stator assembly 3D design.

They were designed, like the rotor assembly design model, using the necessary tools according to the projection plan and the method of drawing sketch.

For example, for the design of the laminated steel stator core, it was used Pad tools - for sizing a laminated plate; the Rectangular Pattern tool with the Ox axis reference direction and a number of instances computed using the formulas from the imported table, equal to the number of laminated plates calculated in the main function were used to create the entire laminated steel stator core, and the Pocket tool was used to design the notches.

For the 3D dimensioning of the copper stator winding, three different "bodies" have been used inside the component part, each of them having a different use (fig. 12).



Fig. 12 Copper stator winding bodies

The first Body called Construction it was used to draw the sketches shown in figure 13, later used in the design of the copper stator winding. This body was designed in order not to clutter a single Body. The piece is a complex one and requires a design made from several parts that will be assembled into a single solid piece.

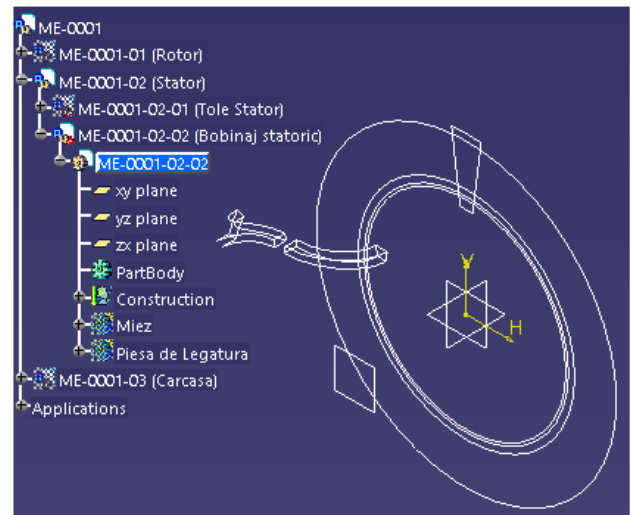


Fig. 13 Copper stator winding sketches.

The main component of the copper stator winding was created inside the Core Body with the help of the first sketch from the Building body - figure 14.

The drawing of this component was made according to the stator grooves model and then it was dimensioned using the Pad tool, with a length of 10% bigger than the length of the laminated steel stator core, being also calculated using formulas. The Thickness function it was applied to the component in order to thin it with a millimeter on both left and right sides.

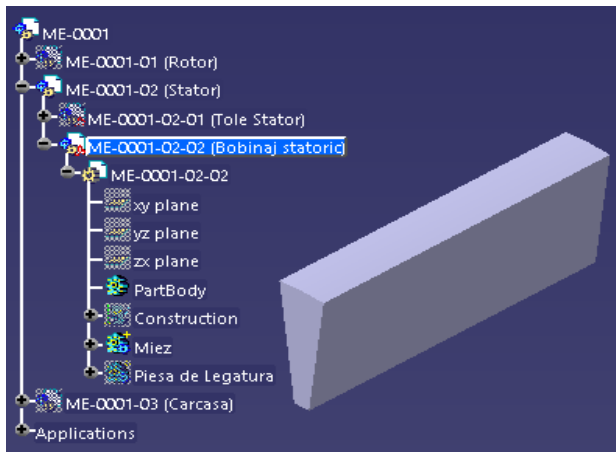


Fig. 14 Copper stator winding sketches.

Inside the last Body, called the Linking Lane, we will create, for the beginning, two other bodies - the first one entitled the Lower Piece and the second one, the Higher Part. To create the lower part, we will copy with a link (fig. 15) the Core Body by creating a solid with the shape and the dimensions of the core and then apply the Split function in order to keep the lower dimensions that we want.



Fig. 15 Copy with link.

The top piece is conceived the same by rotating the solid we created with a calculated angle based on a formula. The solid has to jump to the sixth notch and after that, using the Split tool we will only keep the upper part needed to build the copper stator winding.

The next step is creating the Link by using a copy with a link (fig. 15) of the solid created in the Revolved Core Body based on a formula. It has to jump to the third notch to make the link between the other two previously constructed bodies (fig. 16).

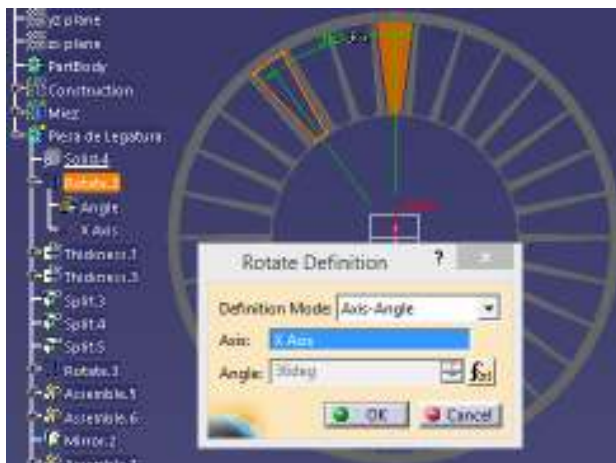


Fig. 16 Connector three by three notch.

We apply Thickness.1 to 21% of the stack length and Thickness.3 in order to thicken it one millimeter on one side.

To design the connector at the end of the copper stator winding, the Split tool that will cut both the part behind the track it will be used (Split.3) after the Plan 1, the upper part (Split.5) and the lower one (Split.4) in order to make the desired piece. Then, it will be rotated around a selected edge of the piece with the 35° angle as in figure 17.



Fig. 17 Copper stator winding connector.

The connection between the Lower Part and the Link Part - figure 18 a), between the Link Part and the Top Part - figure 18 b), will be performed using the Multi-sections Solid tool, using the appropriate curves and the plane surfaces created in the Construction geometrical set.



Fig. 18 Copper stator winding connector.

By using the Assemble tool, they will be constructed as a single solid. Then, we will build the symmetrical duplex against the yOz plane using the Mirror tool.

By using the same Assemble tool, a solid of a copper stator winding will be created – figure.19.



Fig. 19 One copper stator winding.

In order to model the piece and not to overlap after changing the dimensional parameters, we will apply bevels to its edges with different values depending on the thickness of each face, using the Chamfer tool.

After shaping the copper stator winding, we will multiply it by using the Circular Pattern tool around the Ox axis to build the entire stator winding assembly - figure 20 b).

The outer casing assembly is shown in section on the Ox axis in figure 20 below, along with its component parts, named: the center casing - figure 20 a), the front cover - figure 20 b), the back cover – figure 20 c), fan guard - figure 20 d), the two bearings - figure 20 e).

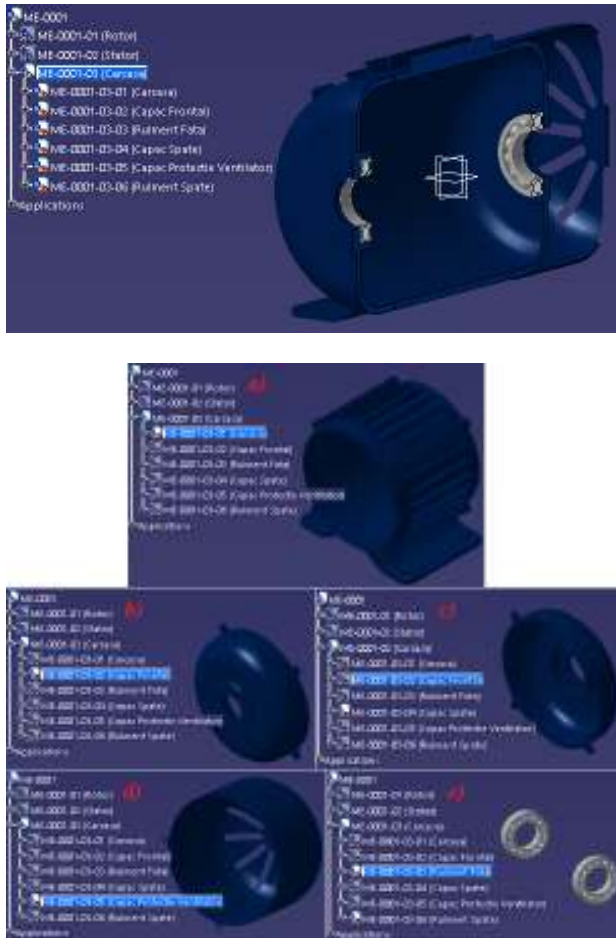


Fig. 20 The stator case (the outer casing assembly).

The central housing and its details were designed using sketches created on the yOz plane and the Pad tool. The housing holes were made using the Pocket tool. The three caps were built on the xOz plane using the Shaft tool, and their details, like those of the central shell, were made on the yOz plane using Pad.

The design of the two bearings was made by building three different bodies: the outer ring, the inner ring and the balls bearing (fig. 21).

The sketches of the outer ring - figure 22 a) of the inner ring - figure 22 b) and the bearing ball - figure 22 c) were made on the xOz plane, where most of the distances and dimensional parameters were determined based on formulas by taking into consideration the position of the bearings on the shaft and the support

inside the outer covers, as well as the dimensions of the other projected parts [4].

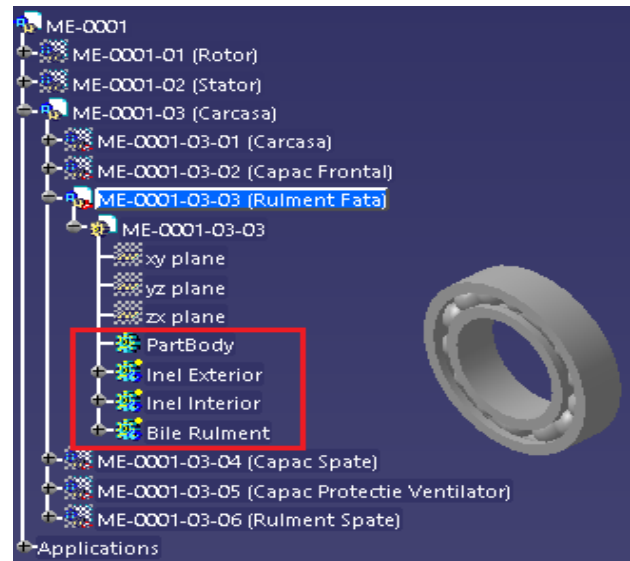


Fig. 21 The bearing bodies.

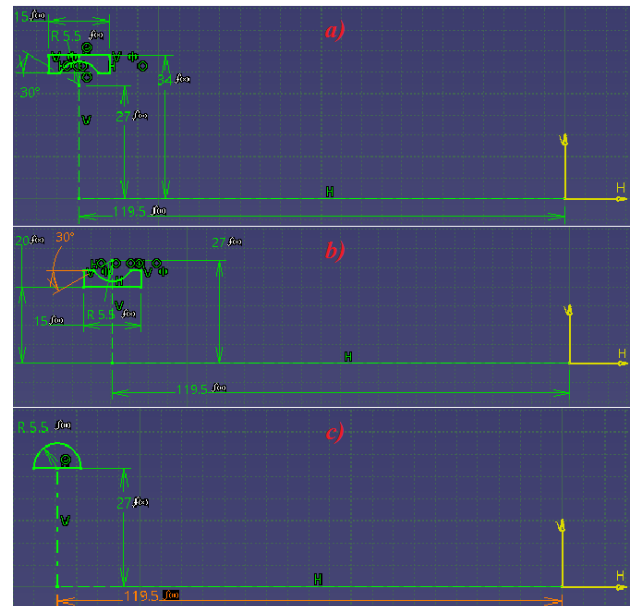


Fig. 22 The bearing sketches.



Fig. 23 The balls bearing.

In order to have a wider view of the components of the electric motor, it was made a section on the Oy axis

of the assembly, listing each component part, as shown in figure 24, and also an enumeration based on the explosion of the electric motor parts - figure 25.

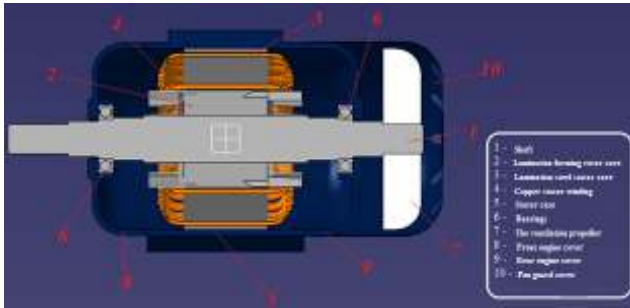


Fig. 24 Section on xOz plane.

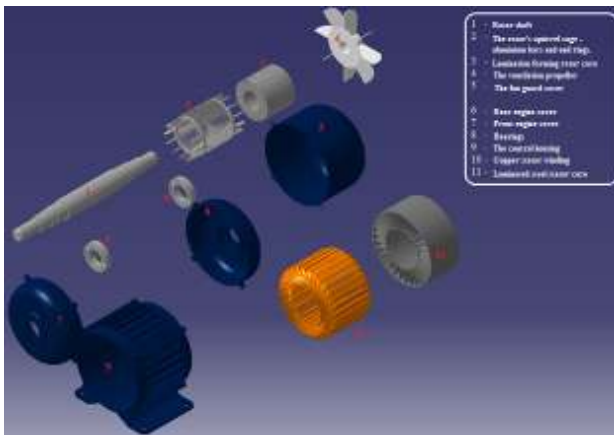


Fig. 25 Parts Explosion view.

3. MODIFYING DIMENSIONAL PARAMETERS

Changing dimensional parameters will be made in the same time with the change of input data by introducing other functional parameters of the electric motor into the parameterization application fields and by applying the Sizing button of the application to run the main function; a confirmation message will appear as in figure 26.



Fig. 26 Confirmation message after application run.

After introducing new input data, other dimensional parameters will be generated and CATIA will display a

window to inform the user that the parameters from the imported table have been modified and the parts whose parameters have been changed will become red, as in figure 27.



Fig. 27 Parameters modification warning.

The modified parts will need an update by using the tool named Update from the toolbar with the same name, and we will wait for the bar which shows the percentage of the Update to charge. The bar is located at the right bottom of the CATIA window - figure 28.



Fig. 28 Update tool

Parametric modification is not always very visible and that's the reason why it was introduced dimensional 3D measurements that change after the Update, as shown in the example in figure 29, for two different sets of input data.



Fig. 29 Parameters update comparison

4. CONCLUSIONS

The purpose for which this work was conceived was the presentation of the way in which a simple and easy to understand the design of an electric motor can be achieved through a succession of well-defined steps.

By mixing two different programs – CATIA V5 and MATLAB the design time of a three-phase cage-rotor induction motor will be reduced considerable.

The theme proposed in this paper can open up new research opportunities in design such as:

- Parameterization of other products required by the profile market;
- Correlation of the design of the product obtained by the proposed method (mixing of the two different programs) with production equipment, such as a numerically controlled machine (CNC). In this way, a product with modified parameters can be launched on the market, in a short time and especially with a smaller effort in the product design stage [6].

In conclusion, the mix of the two programs - CATIA and MATLAB, which has the purpose to 3D dimensioning an electric motor, is practical and easy to achieve using "C" programming knowledge and with which we can resize the engine's 3D design without it is necessary to modify each individual draft, which is why we consider that the method proposed in the presented paper can be extended to any product with "cost-time" benefits that should not be neglected.

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