

FEM STRUCTURAL ANALYSIS OF A FRONT CYLINDRICAL MILLING CUTTER BY USING CATIA

Lecturer PhD. eng. Alin NIOAȚĂ, Faculty of Technical, Medical and Behavioral Sciences, "Constantin Brâncuși" University, nioata.alin@gamil.com

Abstract: With the aid of the finite element analysis method, problems can be studied, the complexity of which is given by the complicated geometric configuration of the bodies, the material inhomogeneities, the anisotropy of the materials, the composite materials, etc. These problems occur frequently in practice at the various stages of development of a product, even when a product already exists, but the problem of improving its characteristics is raised.

KEY WORDS: CATIA, cutting tool, milling cutter, tooth of milling cutter

1. Introduction

CATIA (Computer Aided Three Dimensional Interactive Applications) is a product of Dassault Systemes, one of the most advanced integrated CAD/CAM/CAE platforms based on the latest technologies in the informatic industry.

The CATIA V5 has been available since 1999. At present, CATIA V5 contains over 140 strong applications covering the following areas of assisted engineering:

- hybrid explicit parametric modeling;
- surface modeling, sheet metal;
- modeling of assemblies, optimization of design;
- generation of work drawings;
- design of molds and shapes;
- reverse engineering, rapid prototyping;
- analysis using finite element method;
- kinematic analysis using the virtual prototype;
- simulation of manufacturing processes;
- design of electrical, pipeline, heating, ventilation and air conditioning parts;
- CNC programming for 2/5-axis numerical machine tools.
- translators for converting entities into / from other design environments.

Using the finite element method can solve some types of problems such as:

- problems independent of time:
 - stress and deformation analysis;
 - static analysis of structures;
 - analysis of contact between surfaces;
 - analysis of stresses due to temperature;
- propagation or transition problems:
 - problems of breaking and fissure mechanics under dynamic loads, fatigue behavior, J-integral, fissures, fissure growth;
 - the response of the structures to the aperiodic loads;
 - problems of their own values:
 - natural frequencies and the proper modes of the structures;

2. Modelling of the piece

The work drawing of the cutting tool under discussion below is shown in Figure 1.

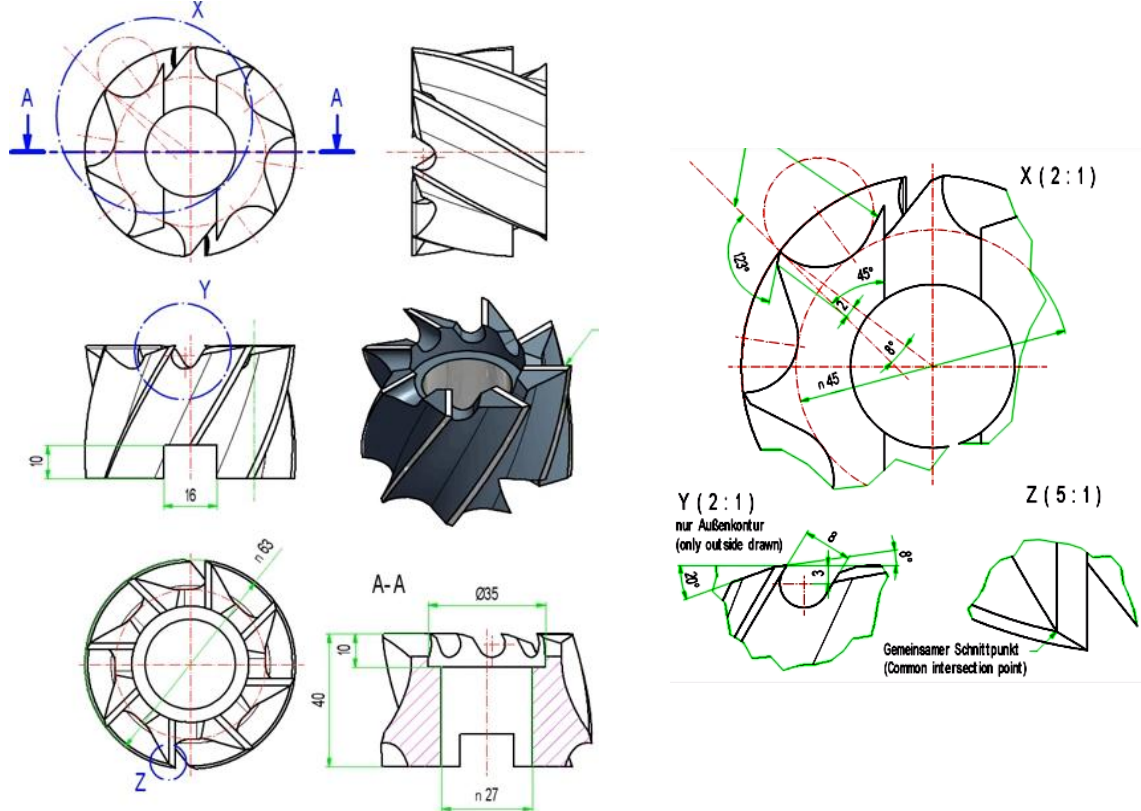


Fig. 1. Front cylindrical milling cutter

At this stage the profile (figure 2) is drawn out of which, by a shaft around the Z axis will generate a cylindrical piece (figure 3). In order to accurately and quickly generate the piece profile, the value fields available in the Sketch Tools toolbar are used.

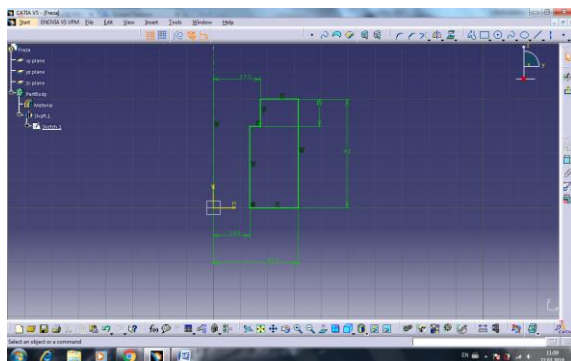


Fig.2. Piece profile

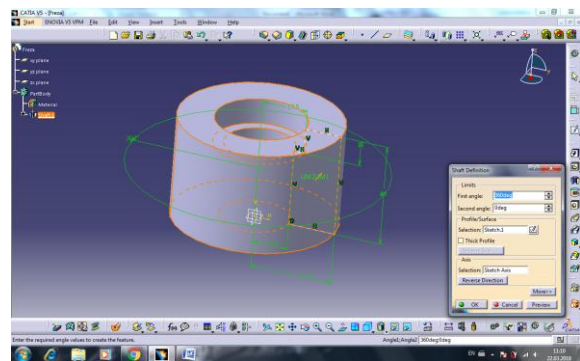


Fig.3. Profile rotation around Z axis

The profile of the tooth root on the bottom of the previously constructed cylinder is drawn by observing the specifications in the drawing of the tool (Figure 4). On the outer surface of the cylinder there is sketched a propeller that copies the angle of

inclination tilt.

Using the Slot command, the tooth hole profile sketched in figure 4 will "move" on the propeller and generate the tooth root on the outer surface of the cylinder(Figure 5).

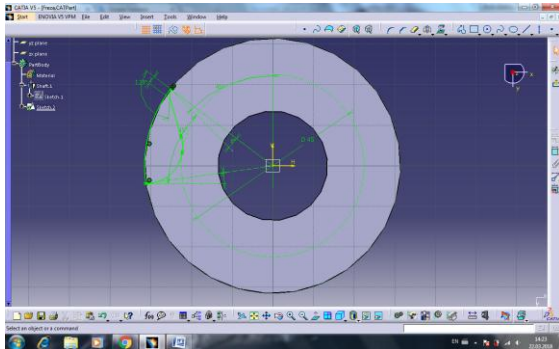


Fig.4. Profile of the tooth root on the cylindrical surface

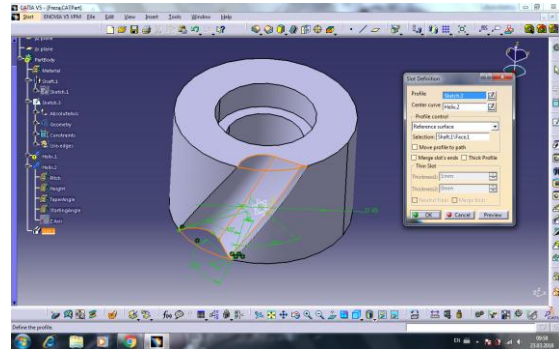


Fig.5. cilindricăModelling of the tooth root on the cylindrical surface

By using the Circular Pattern command, we copy the gap of the tooth 8 times by uniformly distributing` it on the outer surface of the cylinder (Figure 6).

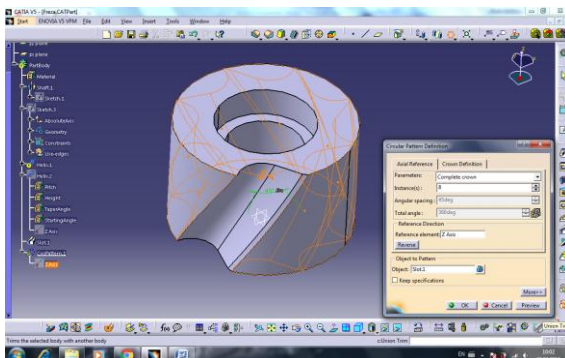
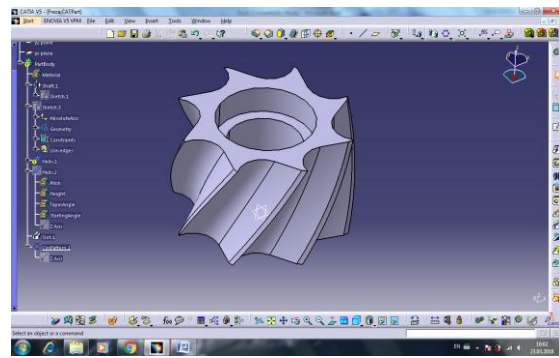


Fig.6.Modelling of teeth on the cylindrical side



Similarly, teeth are modelled on the front of the cutting tool. First, in the YOZ panel, sketch the profile of the tooth root (Figure 7) and then shape the tooth gap by using the Pocket command (Figure 8). Copy the tooth gap on the entire front of the tool circularly (Figure 9).

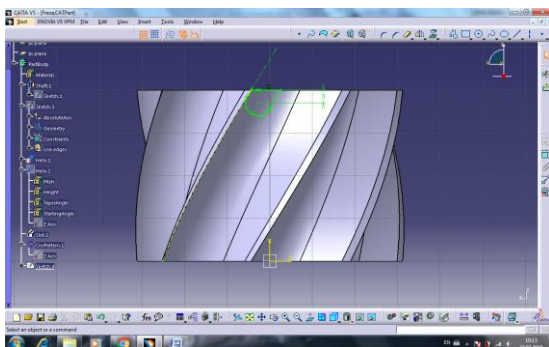


Fig.7. Profile of the teeth root on the front side

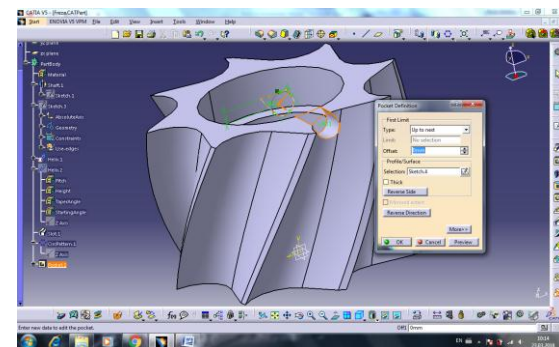


Fig.8. Modelling of the teeth on the front side

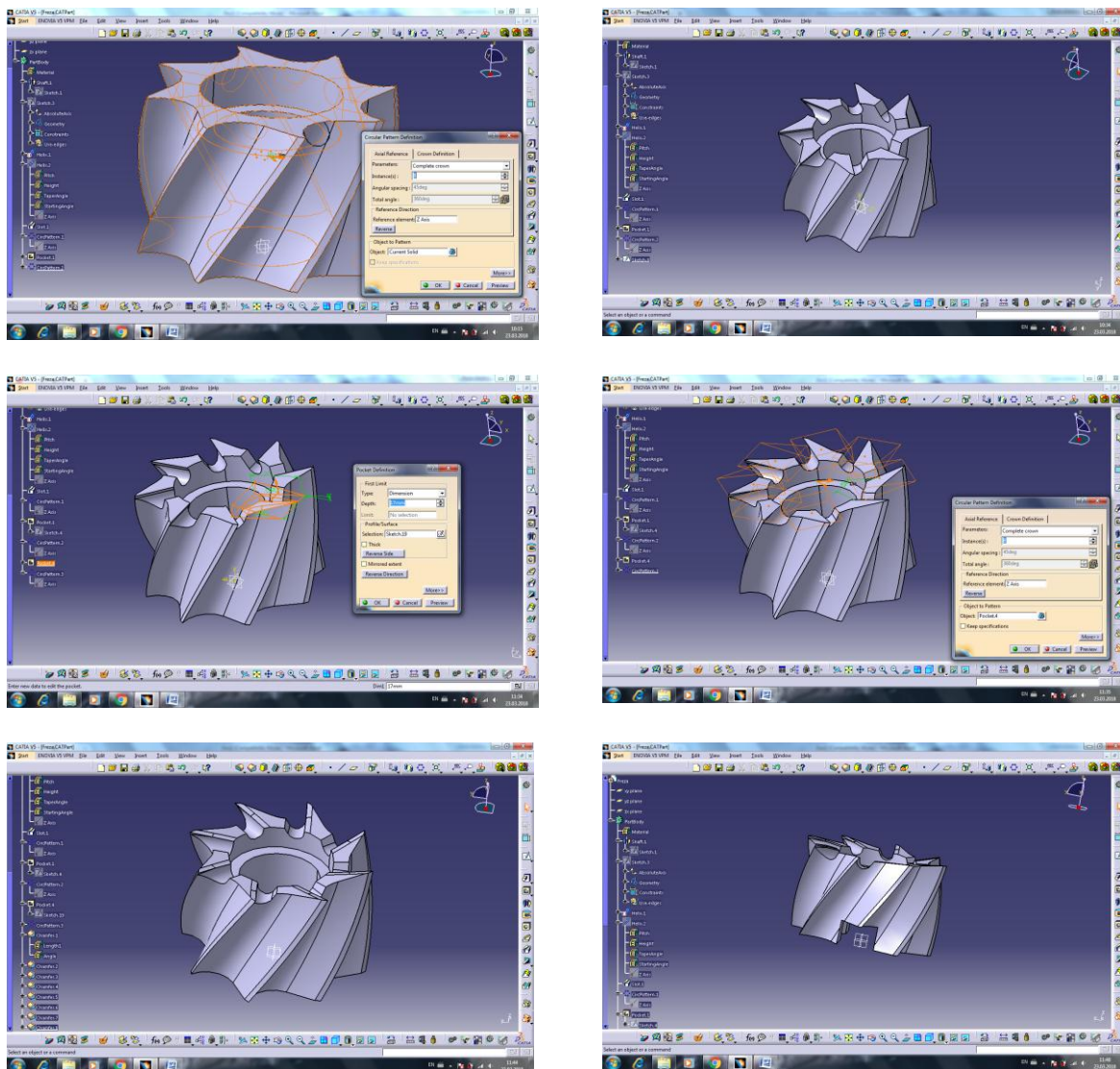


Fig.9. Modelling of teeth on the front side

3. FEM Analysis

After solid modeling in the CATIA Part Design module, the piece is considered to be made of a material (steel) having the following physical and mechanical properties, important during analysis: Young's module ($2 \times 10^{11} \text{ N/mm}^2$), Poisson's coefficient (0.266), density (7860 kg/m^3), coefficient of thermal expansion ($1,17 \times 10^{-5} \text{ }^\circ\text{K}$), permissible strength ($2,5 \times 10^8 \text{ N/m}^2$) (Figure 10).

The CATIA Generative Structural Analysis module is accessed from the Start - Analysis & Simulation menu and the Static Case type is determined, the tree of specifications simultaneously displaying the element with the same name. Although the CATIA program implicitly defines the network of nodes and elements, it is recommended to edit this and determine the size of the element (Size), the maximum tolerance between the discretized model and the real model used in the analysis (Absolute sag), the type of element (Element type), and so on.

Next, a Clamp type restraint (Figure 11) is applied to the gripping surface.

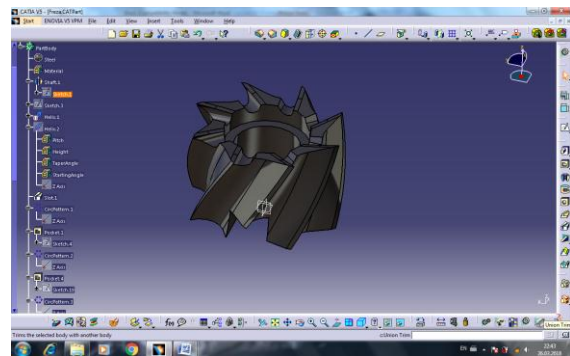
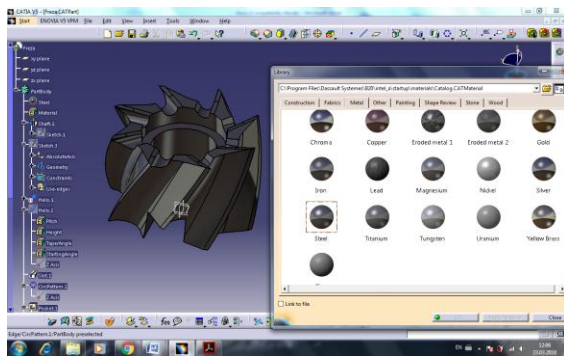


Fig.10. Material assigning

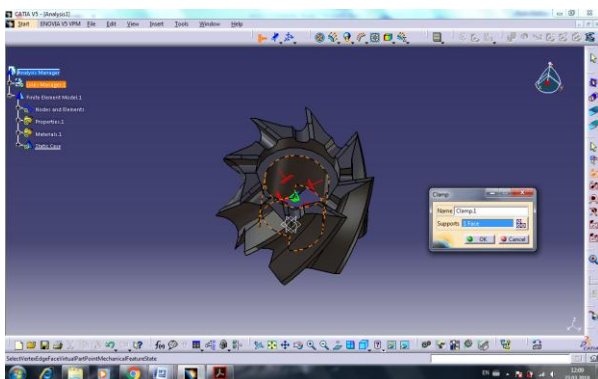


Fig.11. Application of supports

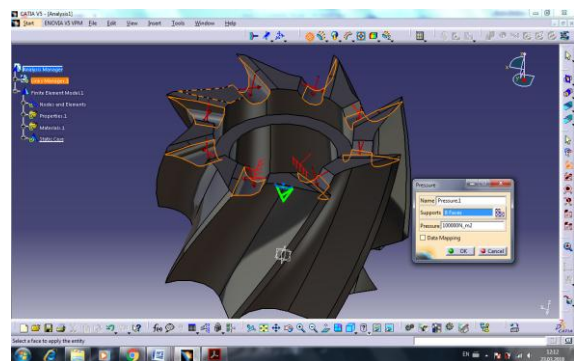


Fig.12. Application of forces

A distributed force (Pressure) is applied to the front teeth release surface, exerting a pressure of $100,000 \text{ N/m}^2$, oriented in the direction of the cutting force. As a result, the Distributed Force 1 element becomes available in the Specification Tree, the force being specified by the arrows on the surface (Figure 12).

Once the restrictions and loading have been established, the actual step of the calculation (analysis) follows. By clicking the Compute icon on the toolbar option All is selected, the first effect of the action being to update the Static Case Solution element (figure 13). Once the calculation is complete, the user has the Image bar tools available to view the results. The specification tree is completed according to the inserted images. Figure 13 shows the image (by using Von Mises Stress, Deformation, Principal Stress, and Precision) corresponding to the calculation of the model and load considered, specifying that the deformations are graphically presented slightly exaggerated to ease the stage of determining the conclusions of the analysis (Figure 14).

In Figure 14, next to this window, is also presented the color palette that accompanies the result-Von Mises image. The lowest stress values are at the bottom of the palette and the highest ones at the top of the palette. The dialog box also contains the explicit values in the Extreme Values area as follows: Min 2.93×10^4 and Max $2.93 \times 10^5 \text{ N/m}^2$. Thus, it is possible to interpret the way the stresses are distributed on the piece according to the colors displayed too.

The blue and light blue colors indicate low stresses (from 52.2 N/m^2 to $2.93 \times 10^4 \text{ N/m}^2$), and yellow to red high-stresses ($2.94 \times 10^5 \text{ N/m}^2$) colors. Considering that the admissible strength of the material is $2.5 \times 10^8 \text{ N/m}^2$, an initial conclusion can be drawn that the piece model will withstand the applied distributed force of 100000N

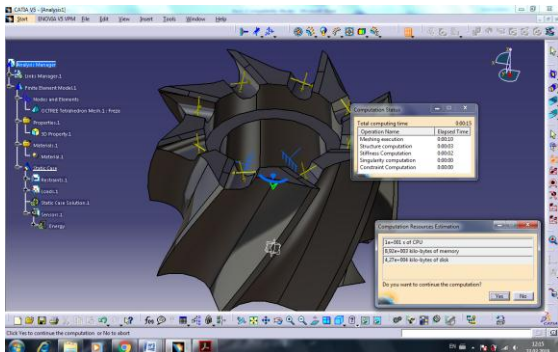


Fig.13. Optimization of the analysis process

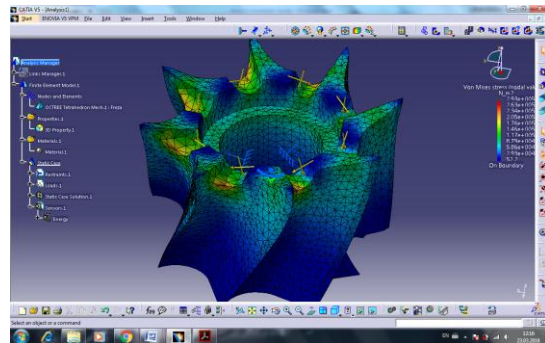


Fig.14. Deformed piece

The process can also be applied to the teeth on the cylindrical side of the cutting tool. A distributed force (Pressure) is applied to the front teeth release surface, exerting a pressure of 250000N/m^2 , oriented in the direction of the cutting force (Figure 15).

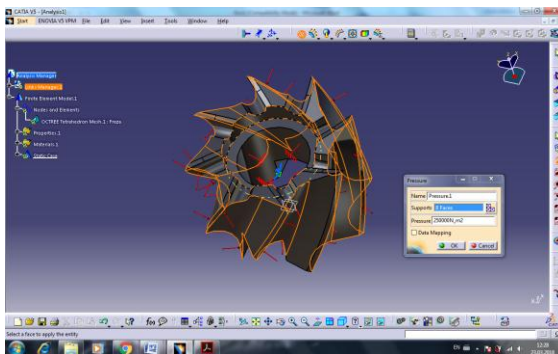


Fig.15. Optimization of the analysis process

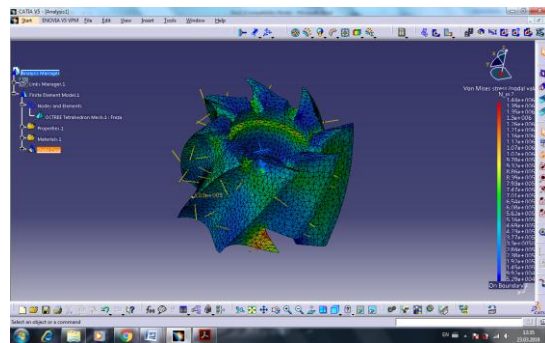


Fig.16. Deformed piece

4. Conclusions

Finite element analysis using the CATIA program is a modern method for studying different contacts, allowing the determination of important parameters for the study of different contacts. In the case of hertz contacts, accurate information on the state of stresses at the contact level is obtained. The punctiform contact analysis by using the classical method (Hertz theory) did not allow such accurate results to be obtained. Von Mises stress determination allows a stress state analysis thus allowing for a correct quantification of this state and the possibility of identifying the deterioration that occurs.

References

- [1] Florin Ciofu - *Hip implant analysis with CATIA*, Reliability and Sustainability Review, Issue 2/2017, ISSN 1844-640X, page 99.
- [2] Ghionea, I.G. - *CATIA v5 - A collection of applications for laboratory activities*, Electronic format, Polytechnic University of Bucharest, 2015;
- [3] Ghionea, I.G. - *CATIA v5 - applications in mechanical engineering*, BREN Publishing House, Bucharest, 2009;
- [4] Ghionea, I.G. - *Assisted Design in CATIA v5 - Theoretical Elements and Applications*, BREN Publishing House, Bucharest, 2014;
- [5] Olariu, V., Bratianu, C. - *Numerical modeling with finite elements*, Technical Publishing House, Bucharest, 1986.