FEM STRUCTURAL ANALYSIS OF A MODULE OF END-MILLING CUTTER USING THE CATIA

Lecturer PhD. eng. Alin NIOAŢĂ, Faculty of Engineering,
”Constantin Brâncuşi” 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, end-milling cutter, forces.

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 [1], [2], [3].

The end-milling cutters are in the form of a cylindrical-frontal tail cutter, the active part of the milling cutter being profiled according to the profile of the gap between the two gear teeth (figure 1) [4].

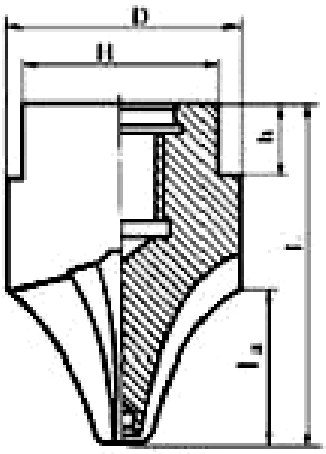


Fig.1 End-milling cutter

The end-milling cutters are in the form of a cylindrical-frontal tail cutter, the active part of the milling cutter being profiled according to the profile of the gap between the two gear teeth (figure 1).

2. Molding of the piece

In this stage, a end-milling cutter with the same construction geometry are considered [4]:

- seat angle $\alpha = 8^o$- 12^o;

- the angle of clearance $\gamma = 5^o$- 8^o;

The 3D modeling of the end-milling cutter to be analyzed with the finite element is shown in Figures 2 [1], [2], [3].

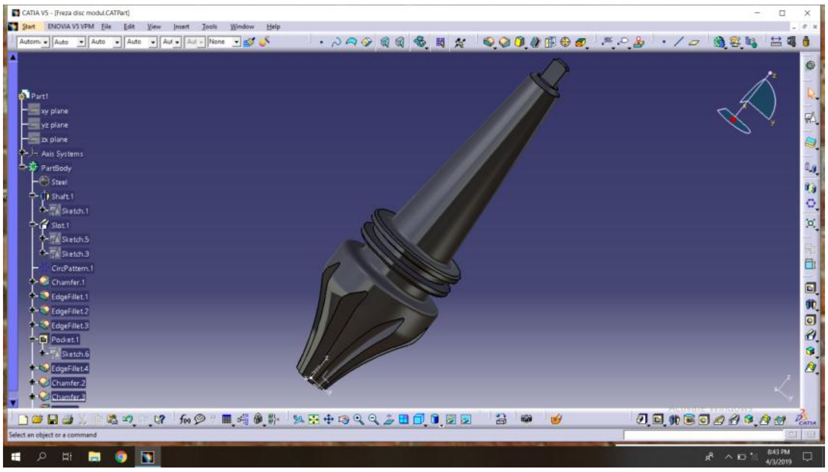


Fig. 2 End-milling cutter

3. Analiza FEM

After solid modelling 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 ($2x10^{11}$N/mm^2), Poisson's coefficient (0.266) , density (7860 kg/m^3), coefficient of thermal expansion ($1,17x10^{-5}$ ^oK), permissible strength ($2,5x10^8$ N/m^2).

The CATIA Generative Structural Analysis module is accessed from the Start - Analysis & Simulation menu and the Static Case type is determined , the specification shaft simultaneously displaying the element bearing the same name . Although the CATIA program defines the network of nodes and elements(discretization), it is recommended to edit this and determine the size of the finite element, 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 [5].

Next, a Clamp-type restriction (Figure 3 for the end-milling cutter) is applied to the clamping surfaces [1], [2], [3].

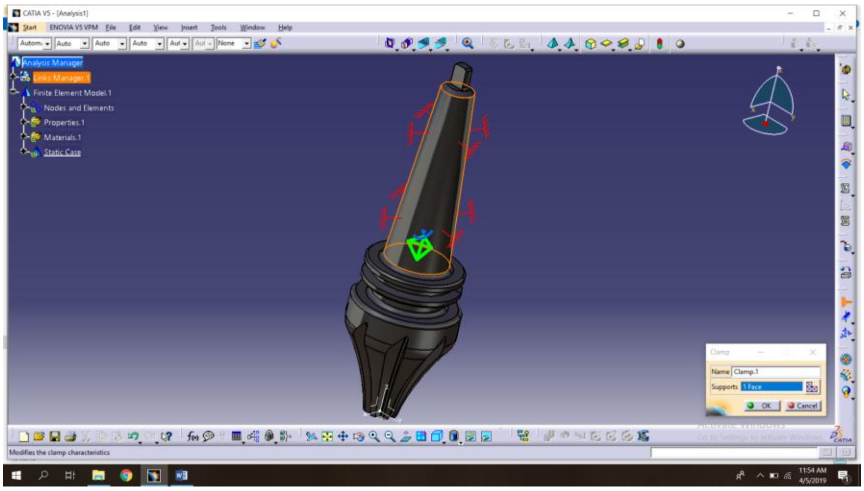
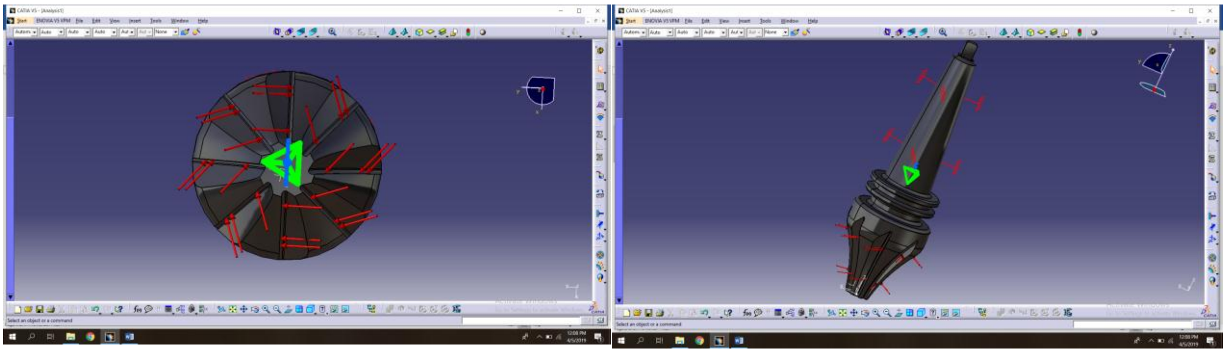


Fig. 3 Applying the supports

A distributed force (Pressure) is applied to the front teeth release surface, exerting a pressure of 100,000 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 (figura 4) [6], [7].



a b

Fig.4. Applying the cutting forces

Once the restrictions and loading have been established, the actual step of the calculation (analysis) follows. By clicking the Compute icon on the toolbar All option is selected, the first effect of the action being to update the Static Case Solution (figura 5) [1], [2], [3], [5].

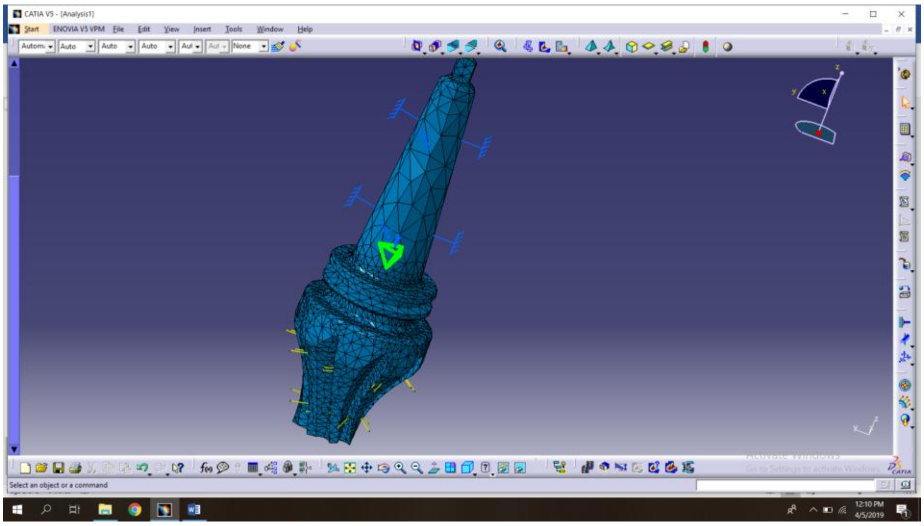
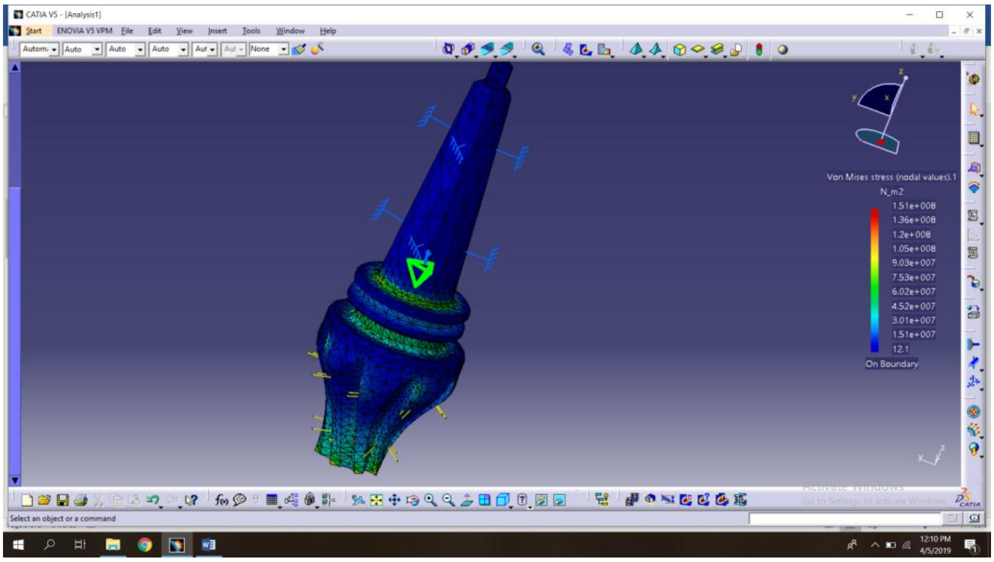
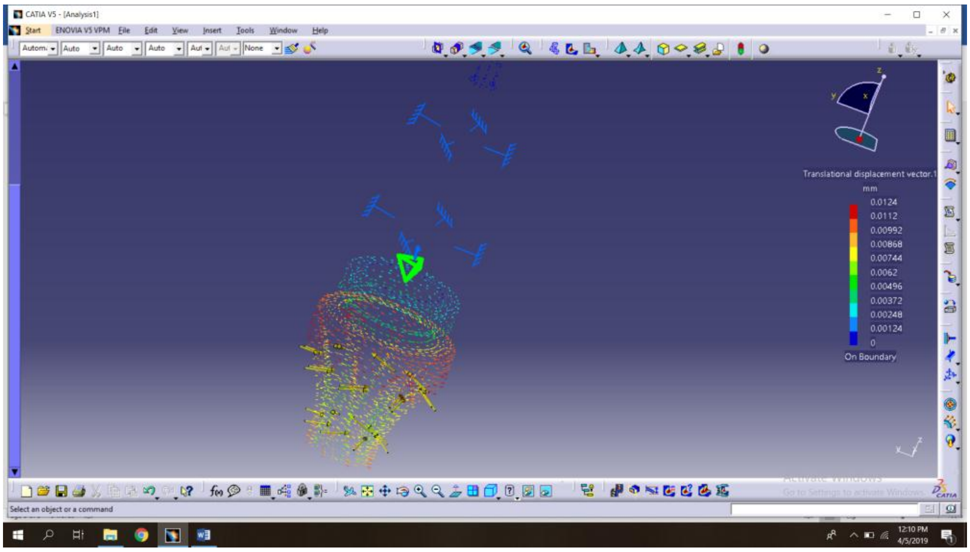


Fig. 5 Optimization of the analysis process

Once the calculation is complete, the user has the Image bar tools available to view the results. The specifications tree is completed according to the inserted images. Figure 6 a,b shows the image (using Von Mises Stress, Deformation, Principal Stress and Precision) corresponding to the calculation of the piece model and load considered, specifying that the deformations are graphically presented slightly overstated in order to ease the stage of determining the conclusions of the analysis.



a



b

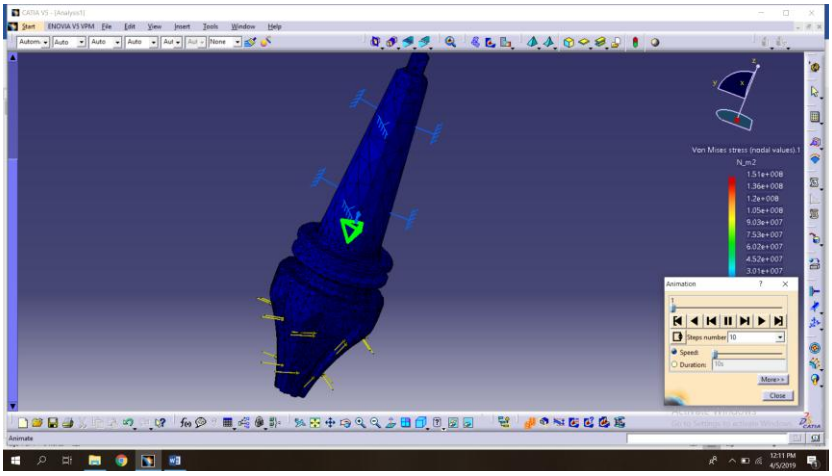
Fig. 6 Deformed piece

In this window is also presented the color palette accompanying the result - Von Mises image. The lowest values of the stresses are at the bottom of the palette and the maximum at the top of the palette. The dialog box also contains explicit values in the Extreme Values area. The blue and light blue colors indicate low stresses, and the yellow to red colors indicate high ones.

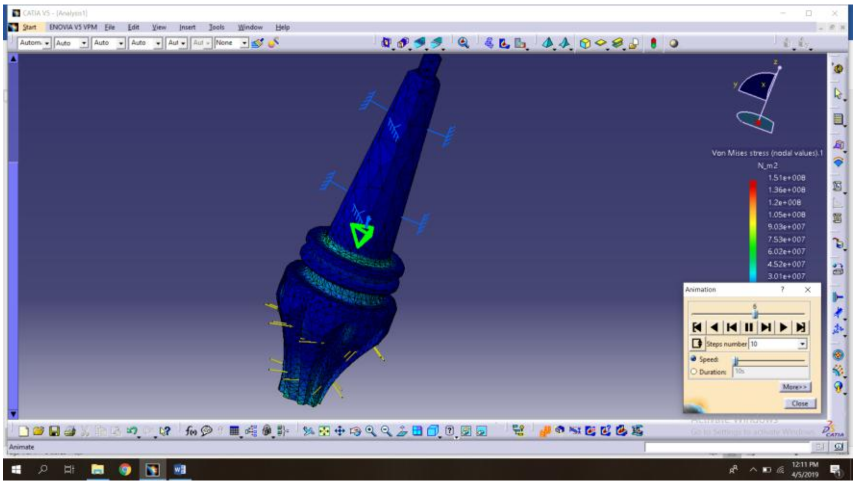
Considering that the admissible material strength is 2,5 x 10^8 N/m^2, it can be concluded that the piece model will withstand the applied forces.

By means of an animation command, the deformations of the end-milling cutter can be viewed clearly, in Figures 7 a, b, c being presented the starting, middle and final moments of the deformations.

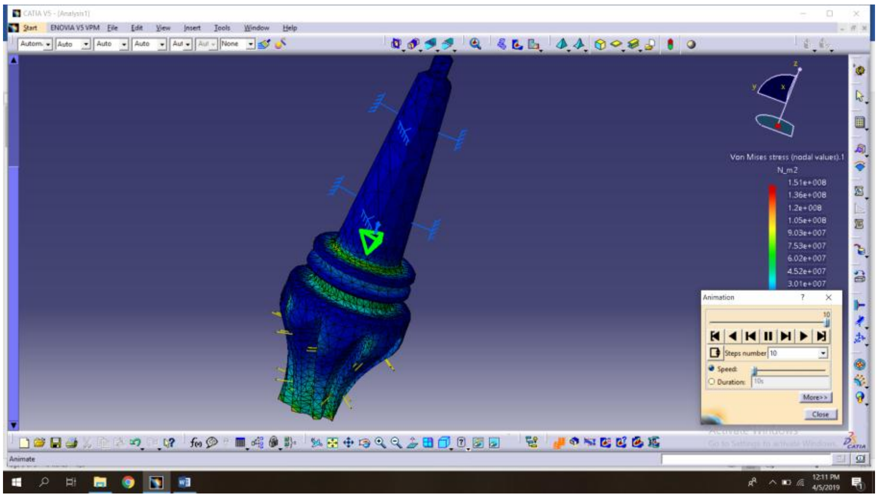
Printr-o comandă de animaţie pot fi vizualizate cursiv etapele deformaţiilor care apar la freza deget modul (figurile 7 a,b,c) fiind reprezentate momentele de început, mijloc şi final ale deformaţiilor.



a



b



c

Fig. 7 Steps of deformation of the enf-milling cutter

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 point contact analysis 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] Ghionea, I.G. – *CATIA v5 – culegere de aplicaţii pentru activităţi de laborator,* format electronic, Universitatea Politehnică Bucureşti, 2015;

[2] Ghionea, I.G. – *CATIA v5 – aplicaţii în inginerie mecanică*, Editura BREN, Bucureşti, 2009;

[3] Ghionea, I.G. – *Proiectare asistată în CATIA v5 – elemente teoretice şi aplicaţii*, Editura BREN, Bucureşti, 2014;

[4] Nioaţă, A. – *Proiectarea sculelor aşchietoare*, Editura ”Academica Brâncuşi”, Târgu-Jiu, 2010.

[5] Ciofu, F. - *Hip implant analysis with CATIA*, Revista Fiabilitate şi Durabilitate, Nr.2/2017, ISSN 1844-640X, pag.99.

[6] Mihut, N. - *Method in calculating own vibration frequencies of open sections bars with thin* walls - MODTECH INTERNATIONAL CONFERENCE - MODERN TECHNOLOGIES IN INDUSTRIAL ENGINEERING IV, PTS 1-7, Book Series: IOP Conference Series-Materials Science and Engineering, Volume: 145, Article Number: 042029, DOI: 10.1088/1757-899X/145/4/042029,Published: 2016

[7] Pasăre M. - *Rezistenţa Materialelor, vol. 1*, Editura Sitech, Craiova, ISBN 978-973-746-621-1, 212 pg., 2007.