

A PRACTICAL APPROACH IN THE FINITE ELEMENT METHOD STUDY OF A MECHANICAL PART

Ghionea Ionuț

*Lecturer, University POLITEHNICA of Bucharest, Faculty of Engineering
and Management of Technologic Systems, e-mail: ionut76@hotmail.com*

Abstract: *The paper presents a practical application methodology in FEM analysis for the mechanical parts design. The procedure uses the CATIA CAD-CAE environment, where the 3D model preparation is simple and fast. The necessary calculations for the analyzed part are made using the finite element method. The purpose of this approach is to determine the resulting stresses under a loading case. Also, in the paper are presented many theoretical aspects in the FEM field of interest.*

Key words: *CATIA, finite element method, finite element analysis, Von Mises stresses, displacements*

1. INTRODUCTION

The finite element method (FEM) is one of the most used methods that are available in our days for different calculations in the field of engineering. This method and the programs based on it become fundamental components in the computer aided design systems. They are indispensable in all engineering activities where high performance is required.

The main purpose of this paper is to present a practical application using the finite element analysis to elaborate a correct, adequate and efficient calculus model to improve the mechanical components design.

The topics in the paper vary from theoretical aspects to a practical problem of the finite element modeling and analysis, with many explications, helping the reader to understand the problems and to draw clear and convincing conclusions.

2. THE FINITE ELEMENT METHOD – BASIC CONCEPTS

One of the major advantages in the finite element method is the simplicity of its basic concepts. It is very important that the FEM user learn and correctly understand these concepts, because they include certain hypotheses, simplifications and generalizations.

In order to obtain a higher efficiency, the concept of structure is used in a more general and simpler way than usually. In FEM, a structure means an ensemble of bars, plates, shells or solids.

Using a CAD-CAE system it is possible to replace the real external forces by efforts (forces, accelerations, torsion moments, masses etc.) to which they are statically equivalent, but this equivalence is not allowed in the theory of elasticity.

To perform a finite element analysis, the user must develop a calculus model of the analyzed structure. These models are only approximate mathematical models of the structure. There are no algorithms and general methods for developing a unique model that approximates, with a known error, the real structure. Generally, several models, all of them correct, but with different performances, can be elaborated for the same structure. The development of a model is based on the intuition, experience and imagination of the user. The model should efficiently synthesize all the available information about the analyzed structure.

This model consists of lines, planes or curved surfaces and volumes, created in a 3D CAD environment. In this stage of development, the model is continuous, with an infinite number of points, as the real structure. The main goal of FEM is to obtain the finite element mesh, transforming the continuous structure into a discrete model, with a finite number of points.

This operation is done using a mesh for the model, which is correct from an engineering point of view, the knowledge of stresses and displacements in a certain number of points inside the structure is normally enough to characterize the mechanical behaviour of the structure.

The finite element method defines these unknowns only in the nodes of the model and calculates their values in these points. That's why the meshing process must be performed in such a way as to have a number of nodes large enough in the areas of great interest in order to achieve a satisfactory approximation for the geometry of the structure and for the boundary and loading conditions. Thus, the mesh of the structure has a major importance in the FEM analysis.

The points defined in the mesh are called nodes. The primary unknowns of FEM are defined in nodes, and their values are the analysis results. These unknowns identified in the nodes can be displacements (displacement model) or stresses (stress model).

For the displacement model, it can be admitted that the deformed shape of the structure under a certain loading case, is defined by the displacements of all the nodes with respect to the initial node net. Each node may have a maximum of six components of the displacement, called nodal displacements in a coordinate system: three linear displacements and three rotations. Some nodes are constrained and thus their displacements are zero or known by imposed values, so they should not be calculated anymore.

The meshing process divides the model into a certain number of quadrilateral or triangular fragments, called finite elements. These elements are assembled together in

common nodes, also called vertices. The FEM study the finite element as a single piece in interaction with the other elements only in nodes. Thus, the study of the real structure is replaced with the study of the ensemble of finite elements obtained by meshing, in an idealization of the real structure.

For better results, the process should be adequate to the purpose of the analysis, implying the respect for some important rules regarding the meshing process and the elaboration of the model, and, also, to use adequate finite elements. The dimensions of these elements can be as much as small, but should be always finite, they cannot tend to zero.

Unfortunately, a general finite element with universal use wasn't yet conceived. In this way, the finite element must be designed in all details: geometrical, mechanical, mathematical etc., a very hard task, but most of modern CAD-CAE systems are automatically resolving it with a little intervention from the user.

The mesh of a structure can include elements defined for different types of analysis, as: linear elastic, nonlinear, heat transfer, fluid mechanics, electromagnetism etc.

In the finite element method practice, the role of the material's characteristics is very important. Thus, the material attached to the finite element can be homogeneous, isotropic or with a certain anisotropy.

Each finite element is an ensemble of conditions and hypotheses and should be used with care and only after a complete study of the environment where is functioning the real structure: loadings, stress type, interaction with other elements etc.

3. ADVANTAGES, DISAVANTAGES OF THE FINITE ELEMENT METHOD

As advantages of the FEM can be counted:

- General use: this is a numerical method used for solving problems in mechanics of deformable bodies, fluid mechanics, heat transfer, biomechanics etc. The loading case can be static, dynamic, nonlinear, time dependent.
- Liteness: to solve a problem with this method, the user is free to elaborate the calculus model to meet his requirements. After this elaboration, often it is very simple to modify it, the goal being the obtaining an enhanced variant of the initial model.
- Simplicity of the basic concepts: The method doesn't require the user to possess good knowledge of mathematics, just an engineering background. The basic concepts, presented above, are easily to understand for a correct use.
- Existence of the FEM computer programs: the use of the finite element method requires a huge amount of numerical calculations, impossible to be solved by man. For this reason, many companies implemented in their CAD-CAE systems procedures for FEM

calculations. Also, there are some FEM specialized programs, offering the user a good percent of stability and safety.

- Pre- and post processing facilities: The use of the computer programs reduce the time of work, enable automatic meshing and verification of the model. The results obtained from the FEM can be presented as images, tables, listings, drawings, diagrams, animations etc.

The FEM has few disadvantages, like:

- The method is approximate: the analysis is not performed on a real structure, but on a model of it. All the results (stresses, strains, displacements etc.) are approximated and the user cannot precisely estimate the difference between the obtained results and the real ones because it appears a percent error, also difficult to know.

- The calculus model is subjective and arbitrary: the user is free to elaborate the model without restrictions, but he is not sure about the correctness of the model, or about the efficiency of the analysis that uses it. Sometimes, it is a good practice to validate the results by experiments on real structures.

- The FEM programs are complex and expensive: the available FEM programs are very complex, with many options and results offering capabilities, that's why their price is high.

4. THE PRACTICAL APPLICATION

In this application it will be presented a FEM analysis of a support type part, having the draft design shown in figure 1:

In both modeling and analysis processes it is used the CATIA program. Thus, after the solid modeling in the CATIA Part Design module, the user applies on the part a material (steel), having the following physical characteristics: Young modulus ($2 \times 10^{11} \text{ N/m}^2$), Poisson ratio (0,266), density (7860 kg/m^3) and Yield strength ($2,5 \times 10^8 \text{ N/m}^2$).

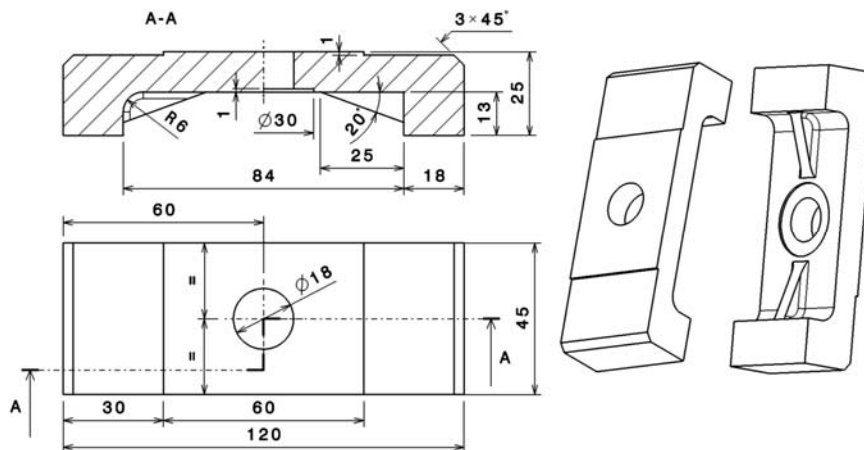


Fig. 1. Support type part

After that, in the CATIA Generative Structural Analysis, the user establish a Static Case type analysis and defines the node and finite elements net, in a process called discretization. In this process it is highly recommended to edit the default net parameters, defining the new finite element size (5 mm) and type (linear), the absolute sag (1,25 mm) between the model and the real part etc. To do this, the user must double-click the subelement “OCTREE Tetrahedron Mesh”, found in the specification tree, as shown in figure 2:



Fig. 2. The discretisation process of the part

In the next stage, the part will be constrained, using the Clamp constrained, applied on the two support surfaces, shown in figure 3. The specification tree is completed with the subelement "Clamp.1". The dialog window contains in the "Supports" field two faces, marked on the part model.



Fig. 3. The application of the Clamp constraints

On the part’s superior and functional surface it is applied a 3000 N distributed force, perpendicularly oriented on the surface, in the opposite direction of Z axis. Thus, in the specification tree becomes available the subelement "Distributed Force.1", the force is symbolized by four arrows on the surface, it’s value and orientation, the coordinate system where it was created, all these parameters can be specified in the corresponding fields of the dialog window from the figure 4:

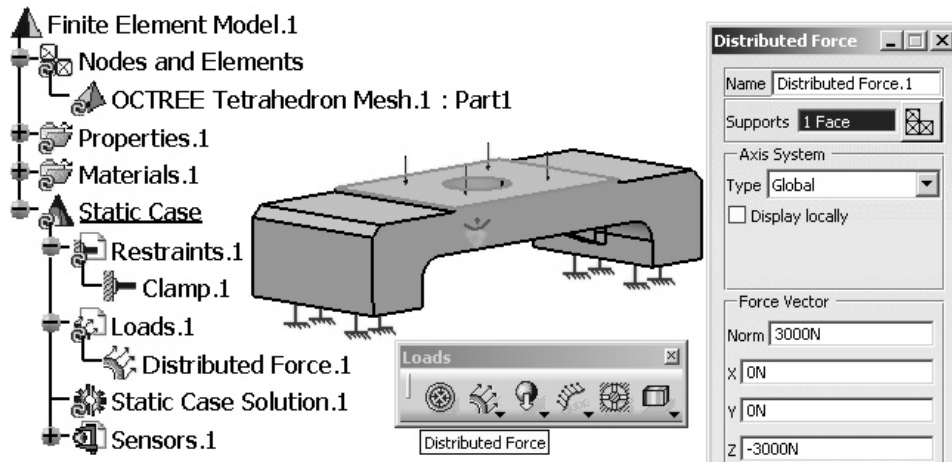


Fig. 4. The distributed force application

After the constraints and loading case establishment, the next stage is the analysis calculus, launched by pressing the “*Compute*” icon. As a result, the dialog window with the same name is opened (figure 5), where the user selects the “*All*” option, followed by the update of the subelement “*Static Case Solution*”.

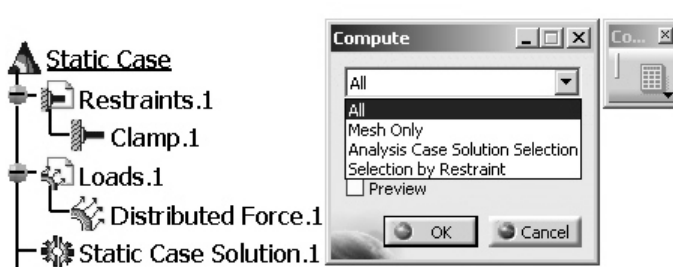


Fig. 5. Launching the calculus stage

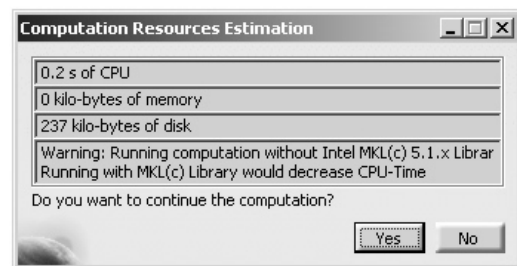


Fig. 6. Calculus resource estimation

If the user unchecks the “*Preview*” option, the analysis time is shortened by not displaying the information window “*Computation Resources Estimation*” (figure 6). However, this window has an important role in very complex analysis processes, because it offers informations about the calculus time and the necessary space on disk.

Once the calculus is finished, the user has at his disposal some tools from the “*Image*” toolbar to visualize the results. The specification tree is completed with different subelements depending of the inserted images. By default, the last added is the active one, by deactivating the previous others. In figure 7 is shown the specification tree containing a list of three images (and also results) and their icons, as part of the subelement “*Static Case Solution*”.

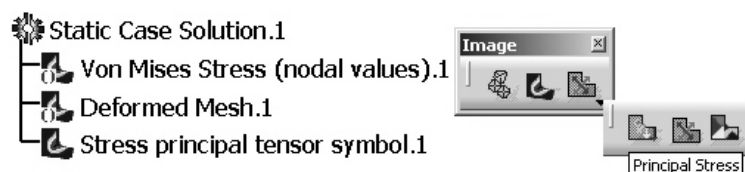


Fig. 7. The Image toolbar and the list from the specification tree

In the following figures (8 ... 11) are presented four image-results, corresponding to the calculus applied on the analyzed mechanical part. It is important to precise that the deformations are graphically represented in an exaggerated way to simplify the drawing conclusions stage.

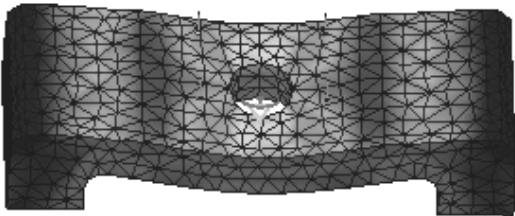


Fig. 8. The *Von Mises Stress* result

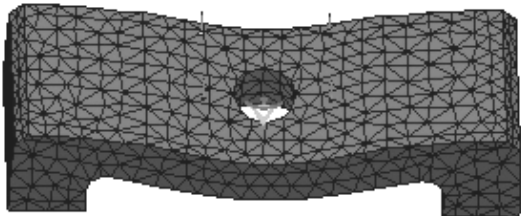


Fig. 9. The *Deformed Mesh* result

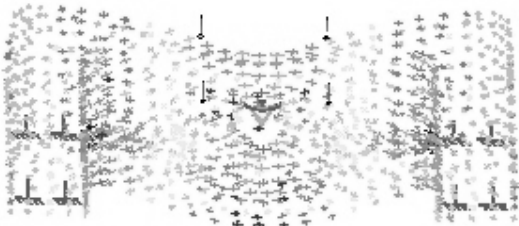


Fig. 10. The *Stress Principal Tensor* result

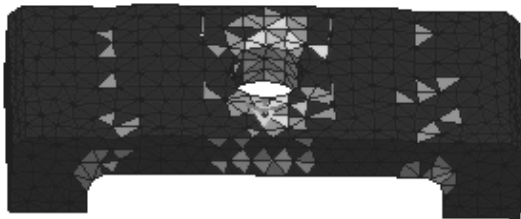


Fig. 11. The *Estimated Error* result

To find the maximum and minimum tensions (stresses) appeared after the analysis, the user must activate the "*Von Mises Stress*" result and, from the "*Analysis Tools*" toolbar, applying the "*Information*" tool, the program opens an information window with the same name, shown in figure 12.

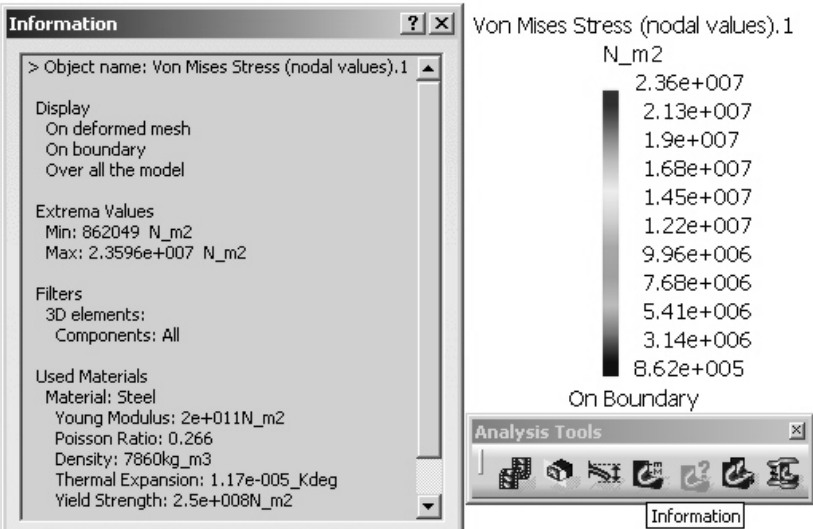


Fig. 12. The extreme values displaying for stresses

In the above figure, near the information window is presented the stresses values palette for the "*Von Mises Stress*" result. The smallest values for the stresses are in the lower

section of that palette, and the highest ones in the upper section. Also, these two extreme values are shown in the information window.

After the analysis ends, the user should compare the yield strength value for the chosen material with the maximum value resulted in the "*Von Mises Stress*" result. Thus, that last value is almost ten times smaller ($2,36 \times 10^7 \text{ N/m}^2$ vs $2,5 \times 10^8 \text{ N/m}^2$), so a conclusion can be drawn: the mechanical part rezists to the considered loading case.

5. CONCLUSION

Using the values resulted from the analysis, the user finds out if the part can support the loading force. In this case it does and no geometrical changes should be done, but there are situations which impose some modifications in the shape of the part body.

The finite element method is a useful analysis tool for engineers and researchers who can study the behaviour of a structure (distinct parts or assemblies) in a virtual way, without building it.

6. REFERENCES

1. Constantinescu, I.N., Sorohan, Șt., Pastramă, Șt. (2006). "*The practice of finite element modeling and analysis*", Editura Printech, ISBN 978-973-718-511-2, București.
2. Cook, R. (1995). "*Finite element modeling for stress analysis*". Editura John Wiley & Sons Inc., New York.
3. Ispas, C., Ghionea, I. (2003). "*The management of computer aided design in the conception of products*". International Conference on Economic Engineering and Manufacturing Systems ICEEMS, Transilvania University of Brașov, ISBN 973-635-216-1.
4. Ghionea, I., Anania, D. (2002). "*The management of the computer aided design using virtual prototype*". Romanian Journal of Technical Sciences. Applied Mechanics, Tome 47. Proceedings of the ICMaS 2002, Editura Academiei Române, ISBN 973-27-0932-4, București.
5. Ghionea, I. (2004). "*Module de proiectare asistată în CATIA V5 cu aplicații în construcția de mașini*". Editura BREN, ISBN 973-648-317-7, București.
6. Ghionea, I. (2004). "*Utilisation of knowledge-based parametric modeling in engineering design*". Romanian Journal of Technical Sciences. Applied Mechanics, Tome 49. Proceedings of the ICMaS 2004, Editura Academiei Române, ISBN 973-27-1102-7, București.
7. Ghionea, I. (2002). "*Business Process Reengineering*". Tehnică și Tehnologie, anul 2 (nr.4). Editura Sincron Edit, ISSN 1453 8423, București.