Proceedings of the 16th International Conference on Manufacturing Systems – ICMaS
ISSN 1842-3183
University POLITEHNICA of Bucharest, Machine and Manufacturing Systems Department
Bucharest, Romania

CONSIDERATIONS ABOUT THE METHODOLOGY AND RESULTS FOR THE FINITE ELEMENT ANALYSIS OF A MECHANICAL ASSEMBLY

Ionuț GHIONEA

Abstract: *The paper presents some considerations and an application methodology in FEM analysis for a mechanical assembly. The procedure uses the CATIA CAD - CAE environment, where the 3D model preparation is simple and fast. The necessary calculations for the analyzed components 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 explanations, helping the reader to understand the problems and to draw clear and convincing conclusions.

2. THE FINITE ELEMENT METHOD - 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 [1].

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 user's intuition, experience and imagination. The model should efficiently synthetize all the available information about the analyzed structure.

A 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 [1].

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 admited 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 [4].

The mesh of a structure can include elements defined for different types of analysis, as: linear elastic, nonlinear, heat transfer, fluid mechanics, electro-magnetism 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. THE PRACTICAL METHODOLOGY

This article presents some finite elements analysis results for a fixture device with circular eccentric gear (cam) and two arms bridle. The cam's working profile is a circular arc. The fixture device is represented in figure 1 by an isometric view, showing the working position with a piece [5].

The functional role of this device is to fix a wedge type piece on two principal diad abutments (bolts) for planar surfaces.

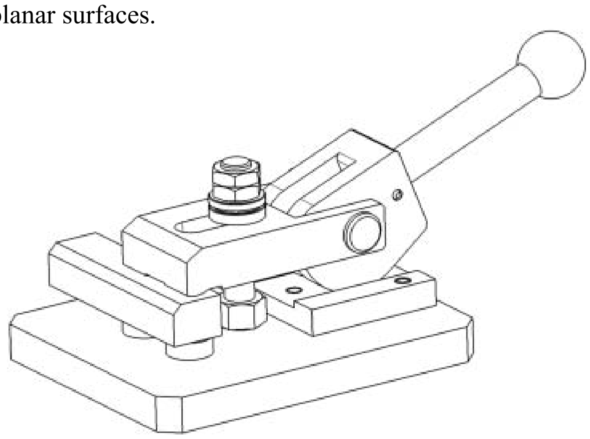

Fig. 1. The fixture device in isometric view.

The considered assambly is composed by these elements: bed plate, abutment's bolts with flat cylindrical head, M16 nut, M16 dowel screw, M10 cylindrical socketed screws, cylindrical pins, guide, swell pin, holder with spherical head, cam, splint pin, M16 narrow nuts, washer with spherical locating surface, washer with conical pocket, attachment flange for gripping cams, piece [3].

To fix the piece on the device, the user can manual action the cam using the holder. The eccentric gear remains clamped under the influence of the friction forces and of those normals after the removing the force applied on the holder. During the rotation of the cam, it's active (working) surface is in contact with the guide mounted on the bed plate. One end of the bridle is articulated to the cam using the swell pin and the other one is free and applies the fixing force. The bridle is mounted on the dowel screw, being limited in it's vertical moving by two washers, fixed at the upper end of the dowel screw with two nuts. By their form, these washers allows a self placing on the superior surface of the bridle, together with the cam's rotation, when the tightening end of the bridle is in contact with the piece, which is relying on the abutment's bolts [5].

To simplify the fixture device, the FEM calculus and the explanations, the coiled spring was not represented, but it could assure the lifting and sustaining of the bridle when the eccentric gear is unbend and the piece is removed. During the fixture, over the main elements of the device are applied some bending, tension or compression stresses.

The scope of the finite elements analysis for the presented device is to identify the zones with high stresses and their values. The article does not consider the stresses resulted from the technological process when the piece is machined, supports' and tightening elements' rigidity.

For this analysis it is used the module Generative Structural Analysis from the CATIA software package. Also, for the 3D modelling of the device and it's assembly should be used two other modules: Part Design and Assembly Design [7]. A very important step in the FEM analysis process is the way which the device's components are assembled using numerous constraints of the following types: coincidence, surface contact, linear contact, offset and fix. The establishment of these constraints must be done after the functional role of each device's component and it's position within it. Thus, in figure 2 is presented a fragment from the assembly constraints list used for this device.

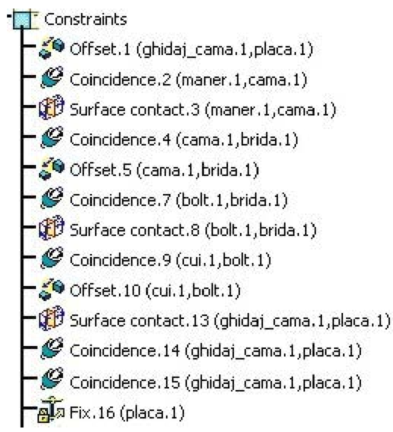

Fig. 2. Fragment of the assembly constraints list.

Fig. 3. Node network discretization.

As an example, there are coincidence constraints between the axes of the swell pin and of the cam's hole, between the axes of the dowel screw and it's corresponding nuts, linear contact constraints between the active surface of the cam and the guide's planar surface, between the tightening curved surface from the bridle's end and the piece's planar surface. Also, as a further example, there is a surface contact constraint between the supporting surface of each abutment's bolts and the bed plate etc. Any other constraints should be imposed for a complete definition of the fixture device.

Each component receives a material (steel) applied on it, having the following properties: Young modulus (2×10^{11} N/m^2), Poisson ratio (0.266), density (7860 kg/m^3), thermal expansion (1.17×10^{-5} K) and the yield strength (2.5×10^{8} N/m^2) [7].

Using the module Generative Structural Analysis, a node network discretization is done for each component, establishing the dimension, the type of each finite element and the tolerance between the real model and the discretized one etc. Figure 3 shows some of these settings for the cam. The finite element type is choosen as Linear because the assembly is composed by a relative major number of components, and a Parabolic element type could extend too much the calculus time [4, 7].

Using the assembly constraints, in the next step there are established the physical constraints, necessary in the simulation of the tensions' transmitions process, generated by the application of a force on the holder. Thus, the physical constraints are choosen after the assembly constraints and they are of these following types: Fastened Connection Property, Pressure Fitting Connection Property and Contact Connection Property.

In figure 4 it can be observed a fragment from the list of these physical constraints, along with some of their symbols positioned on the respective components. The whole list is much bigger because of the number of components and of the assembly constraints between them.

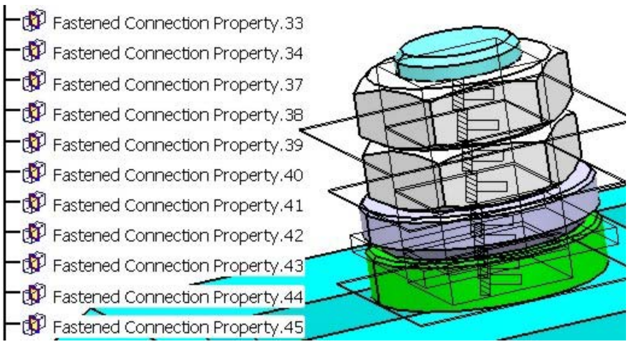

Fig. 4. Fragment of the physical constraints list and symbols.

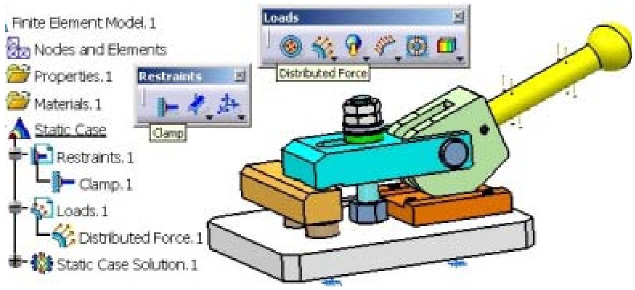

Fig. 5. Application of the fixing restraint and the loading force.

As an example, the coincidence assembly constraints between the swell pin's axis and the axis of the cam's hole become physical constraints of type Fastened Connection Property. The coincidence assembly constraints between the abutment's bolt's axes and the corresponding axes drilled in the bed plate become physical constraints of type Pressure Fitting Connection Property. Also, the surface contact assembly constraint between the guide and the bed plate become a physical constraint of type Contact Connection Property [3, 7].

The next step in this application consist in adding a fixing restraint (Clamp), positioned on the base surface of the bed plate. Also, in this step too, there is established the applied load on the holder, a force (Distributed Force) of 600 N value. This force has the device's working direction (holder-cam-bridle), tightening the piece. The specification tree, positioned on the left side of the CATIA interface, will show the subelements "Clamp.1" and "Distributed Force.1" (figure 5).

After these preliminary stages (choosing the discretization values and imposing of constraints and loads) follows the launching of the finite elements analysis process.

The procedure is very specific to the CATIA software and implies some calculus actions, computer resources and time [1, 3, 4]. In the final of this process, the specification tree is completed with the subelement "Static Case Solution", which may contain different solutions depending of the used instruments: Deformation, Von Mises Stress, Displacement, Principal Stress and Precision. To determine the tensions inducted in the assembly components by the applied loading force, it can be used the Von Mises Stress results. Also, using the Image Extrema instrument, there can be highlighted (located) the minimum and maximum tension values, at a global or at a local level. The specification tree from the figure 6 presents the subelements "Von Mises Stress" and "Extrema" which shows on the device's 3D assembly model some indicators containing the type of the extreme values. Also, in the figure it can be observed the node network discretization [7].

From the colours and values palette, which correspond to the Von Mises model representation, it can be observed that the maximum tension in the device's assembly has the value of 2.63×10^{7} N/m^2, located on the holder, in the joint area between it's end and the cam. For the first analysis it was obtained an error percent of 45.81%, but this result is too inaccurate, so an assembly node network discretization refinement is necessary, followed by another analysis process, applying the instrument New Adaptivity Entity [7].

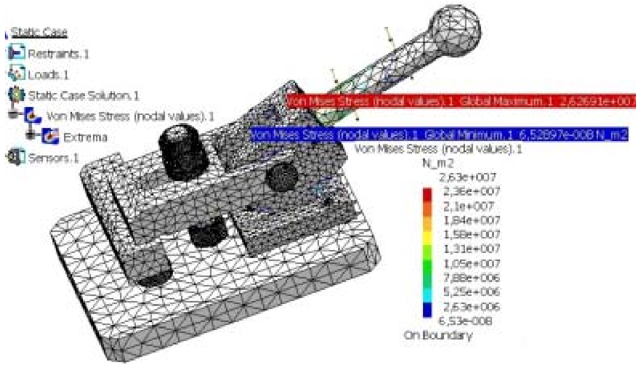

Fig. 6. Representation of the Von Mises results and the localization of the extreme tensions.

The user may impose and wants to obtain a 20% error in three iterations, but in the final of this second analysis step, the error percent is decreased only to 26.57%, but, also, a maximum tension increase on the holder, in the same area of joint: 5.18×10^7 N/m^2.

If these values (error percent and tension) are not satisfactory, although the maximum tension value is smaller than the yield strength for the choosen material (steel), another node network discretization refinement can be made, using the finite element type as Parabolic, mainly to decrease the error percent.

Taking into consideration the functional role of the assembly components, it is imposed the analysis of those which are submited to wear and fatigue, the most important being the cam, the bridle and the dowel screw.

Thus, using the CATIA facility to display the tension for each component, in the figures 7, 8 and 9 are presented the extreme tensions in colours and values palettes for those three components.

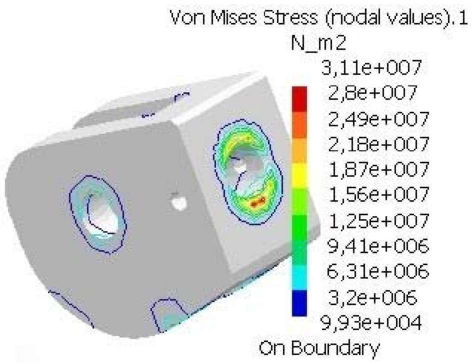

Fig. 7. Tension representation and values for cam.

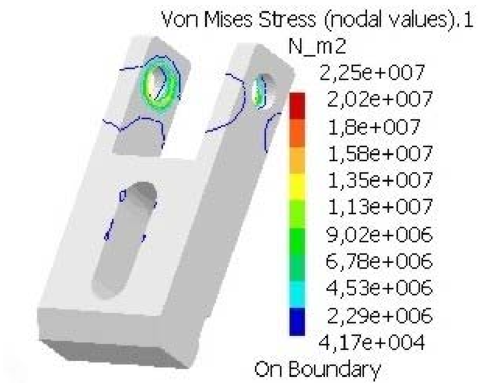

Fig. 8. Tension representation and values for bridle.

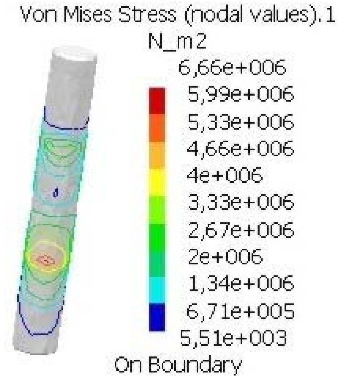

Fig. 9. Tension representation and values for dowel screw.

4. CONCLUSIONS

For the cam, the most solicited areas are: the assembly hole with the holder (3.11×10^7 N/m^2), the assembly hole with the swell pin (1.1×10^7 N/m^2) and the contact surface with the guide.

For the bridle, the most solicited areas are: the surfaces of the assembly hole with the swell pin (2.25×10^7 N/m^2) and the planar contact surface with the conical pocket washer (3.35×10^6 N/m^2). On the contact area with the piece, the bridle presents tensions of 1.1×10^5 N/m^2.

The obtained results for the dowel screw shows that the most solicited areas are at the upper end, in the proximity of the assemblying zone with the nuts (6.66×10^6 N/m^2).

Also, the other end (the assembly zone with the bed plate) is solicited, the tensions value is 6.17×10^6 N/m^2. Thus, the tensions' distribution along the dowel screw indicate that it is draughted and bended.

REFERENCES

[1] Constantinescu, N. I., Sorohan, Şt., Pastramă, Şt. (2006). *The practice of finite element modeling and analysis*. Editura PRINTECH, ISBN 978-973-718-511-2, Bucureşti.

[2] 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, pp. 529-532, ISBN 973-27-0932-4, Bucureşti.

[3] Ghionea, I. (2007). *Proiectare asistată în CATIA v5. Elemente teoretice şi aplicaţii*. Editura BREN, ISBN 978-973-648-654-8, Bucureşti.

[4] Ghionea, I. (2007). *A practical approach in the finite element method study of a mechanical part*. Scientific Bulletin, Serie C, Volume XXI, Fascicle: Mechanics, Tribology, Machine Manufacturing Technology, pp. 251-258, ISSN-1224-3264, North University of Baia Mare, may 2007, Baia Mare.

[5] Tache, V., Ungureanu, I., Stroe, C. (1985). *Elemente de proiectare a dispozitivelor pentru maşini-unelte*. Editura Tehnică, Bucureşti.

[6] Vlase, A., (1996). *Tehnologia construcţiilor de maşini*. Editura Tehnică, ISBN 973-31-0777-8, Bucureşti.

[7] ***, *CATIA V5R15*. (2005). *Documentaţie de firmă*. Dassault Systemes.

Author: Drd. ing. Ionuţ GHIONEA, lecturer, University POLITEHNICA of Bucharest, Production Engineering Department, Bucharest, Romania, e-mail: ionut76@hotmail.com