THE ANNALS OF “DUNAREA DE JOS” UNIVERSITY OF GALATI
FASCICLE IX. METALLURGY AND MATERIALS SCIENCE
N°. 4 – 2014, ISSN 1453 – 083X

ANALYSIS FREQUENCIES OF A VIBRATION-TYPE STRUCTURE HALL

Ionel PETREA
“Dunarea de Jos” University of Galati, Romania
e-mail: ipetrea@ugal.ro

ABSTRACT

Its main purpose was to identify natural vibration frequencies for a warehouse type structure in order to avoid resonance phenomena of the structure. The analysis is performed using the finite element analysis module of the CATIA system.

KEYWORDS: Von Mises stress, displacement field, restrictions, loads

1. Introduction

The finite element method (FEM) is one of the best methods of doing calculations and simulations in engineering.

CATIA systems contains Generative Structural Analysis modules, which is a suitable for advanced finite element analysis.

The steps in a finite element analysis are as follows:

- three-dimensional modeling of components assembled using CATIA Part Design and CATIA sketcher modules;
- insert all component parts using CATIA Assembly Design module;
- application of a material to the assembly components;
- accessing CATIA Generative Structural Analysis module and determination of the type of analysis;
- definition of nodes and elements (a process called meshing) and it’s editing;
- setting restrictions;
- adding loads for the model;
- performing analysis calculations;
- visualization and interpretation of the results.

Hall-type structures are characterized by rigidity, being made generally of elements made of steel or wood. The identification of natural vibration frequencies is particularly important in order to avoid, through the design process, the frequent earthquakes, thereby avoiding the appearance of resonance structure in case of earthquakes, a phenomenon that can lead to partial or total destruction of the hall.

The Hall type structure in Figure 1 is composed of parts made of wood, hardened and joined together by elements made of steel.

The analysis is shown in Figure 2. To analyze the natural vibration frequencies a rigid connection between elements is considered and the foundation is modeled through a restriction that requires the cancellation of 6 degrees of freedom possible embedding surface.

The analyzed subassembly consists of elements made from oak (longitudinal elastic modulus E = 1.32 x 10 N/mm^2 and density ρ = 753 kg/m^3) and assembled reinforced by elements made of steel (modulus of elasticity E = 2.1 x 10^5 N/mm^2 and density ρ = 7860 kg/m^3).

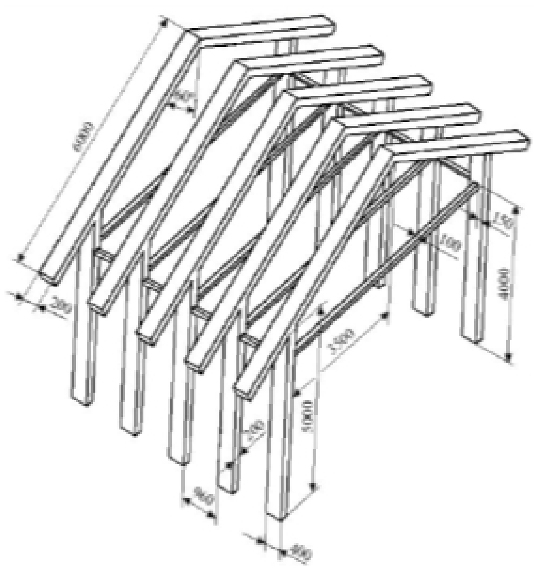


Fig. 1. *Hall type structure*

2. Geometric modeling

The geometric design of the main structure (wood) is obtained in the Sketcher module (Figure 2).

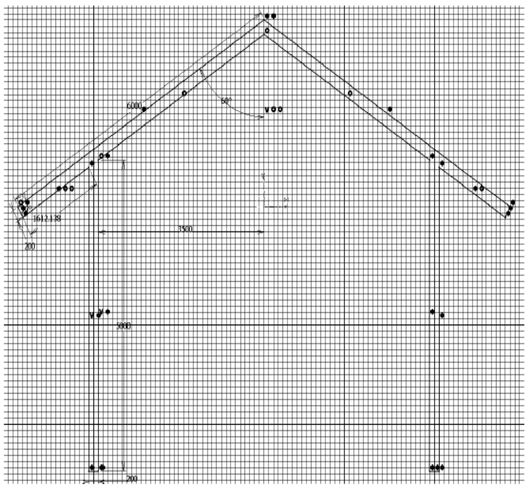


Fig. 2. *Outline of the main structure*

The main structure is obtained by extruding a 400 mm (thickness structure) profile previously created (Figure 4).

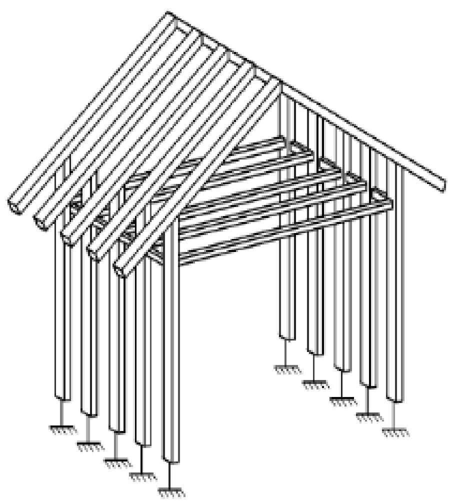


Fig. 3. *The finite element analysis*

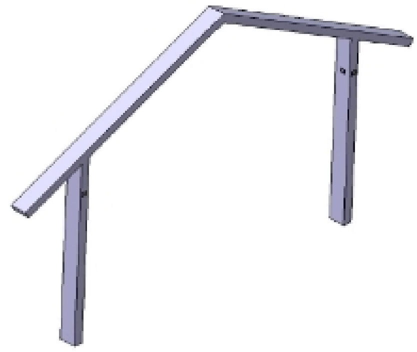


Fig. 4. *Main structure*

The element geometric relationship between structures made of wood is obtained in the Sketcher and Part Design module (Figure 5). The geometric design of the reinforcing element is shown in Figure 6.

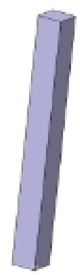


Fig. 5. *Connection element*

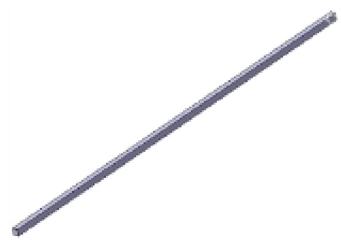


Fig. 6. *Reinforcing element*

The insertion of the component parts is made in the CATIA Assembly Design module (Insert/Existing Component) (Figure 6).

The assembly is created using geometric constraints between components (Contact Constraint and Coincidence Constraint).

Fig. 7. *Assembly structure*

Application of material using Apply Material icon. Entering the values of material characteristics required for the finite element analysis is performed using the CATIA environment library materials, to choose from:

- Metallic material group steels (Steel), which changes the values of the elastic modulus (Young's modulus) and Poisson coefficient, taking into account the values given as input.

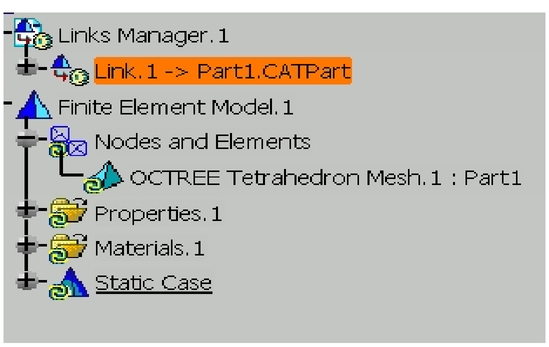
- Material ranging from wood (Wood), Group oak wood (Oak), for which the modulus of elasticity (Young’s modulus) and Poisson coefficient are accepted.

3. Finite element analysis

Accessing CATIA Generative Structural Analysis module and setting the type of frequency analysis allows for the analysis of the vibration modes of the structure in terms of imposing rigid type connections between components and considering a database connection to cancel the 6 degrees of freedom associated with the possible points.

The definition of nodes and elements (a process called meshing) determines the size of finite element (Size), the maximum tolerance between discretized model and real model used in the analysis (Absolute sag) type element (element type), etc. For this, a double click is executed on “Mesh tetrahedron OCTREE” sub-item found in the specification tree.

Figure 7 presents the specification tree and the dialog with the same name, which contains the finite element size (30 mm), minimum tolerance (6 mm) and the type of the element as parabolic.



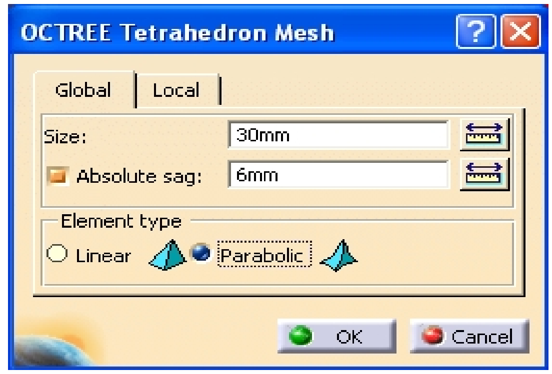


Fig. 8. Discretization of the part model

Restrictions definitions

The connections between elements is achieved through a connection type Rigid Connection, applying the geometric constraints generated module assembly, faces common in contact.

The contact imposed based model is defined by canceling the 6 degrees of freedom associated potential surfaces Contact element wood foundation (Clamp) (Figure 8).

The effective step analysis calculation is done using the *“Compute”* icon.

The display and interpretation of results is achieved, with the help of *Image* on the toolbar. In Figure 9 the specification tree is exemplified, containing a list of three images and their icons.

Figure 10 displays the corresponding results Von Mises stress (values of a scalar field energy density obtained from the volume of strain).

The numerical values of these stresses are displayed along with the Von Mises stress. Also, the nodes location is displayed where minimum and maximum values of these tensions are located.

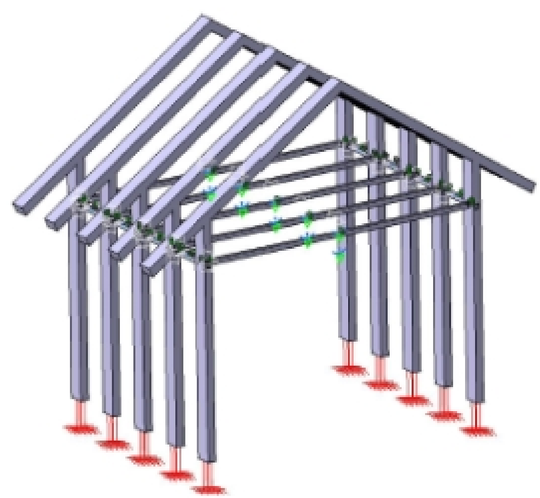


Fig. 9. Restrictions definitions

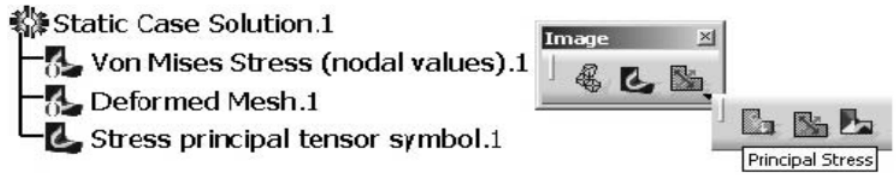


Fig. 10. *Tools of image display and the corresponding menu tree*

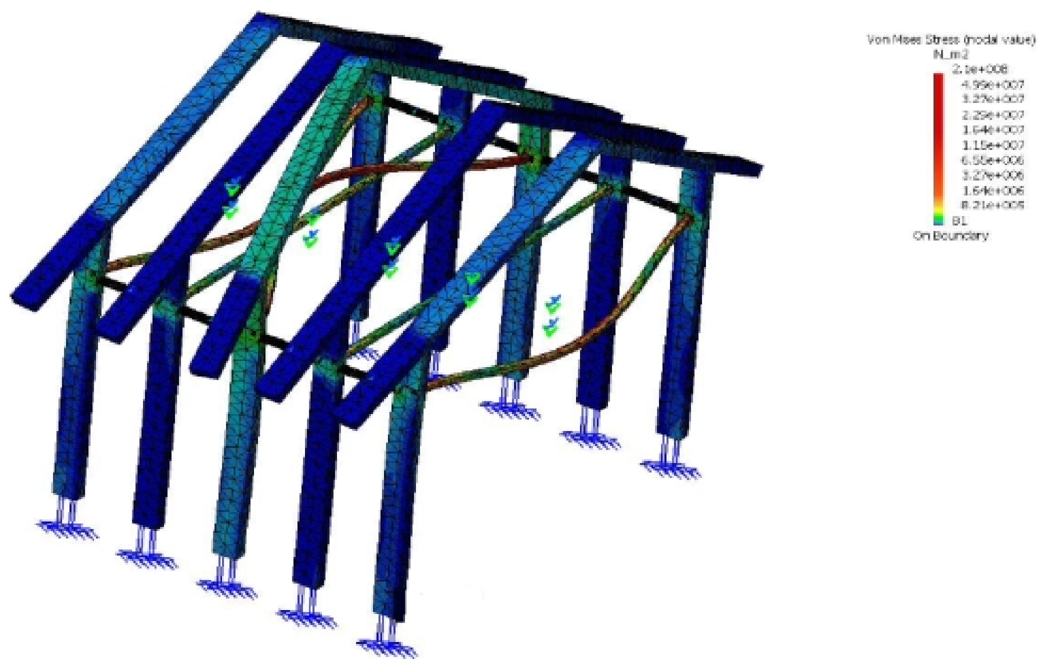


Fig. 11. *Result of Von Mises stress and indication of extreme stress*

4. Conclusions

The analysis of displacement and stress fields shows that the analyzed subassembly is a rigid structure with high peak voltages stiffeners (maximum Von Mises equivalent stress is 210 MPa). To avoid structural or technological damage, adopt constructive measures (increase cross section of stiffeners and/or use of materials with superior mechanical properties).

Identification of natural vibration frequencies is useful in order to avoid the phenomenon of resonance.

References

[1]. Ionuţ Gabriel Ghionea, *Proiectare asistata în CATIA V5 – Elemente teoretice şiaplicaţii*, Editura BREN, Bucureşti, 2007.
[2]. Mihai Tiberiu Lates, *Metoda elementelor finite. Aplicaţii,* Editura Universitatii Transilvania din *Brasov*, 2008.