THE ANNALS OF “DUNAREA DE JOS” UNIVERSITY OF GALATI.
FASCICLE IX. METALLURGY AND MATERIALS SCIENCE
N⁰. 4 – 2013, ISSN 1453 – 083X

# STATIC ANALYSIS OF THE STRUCTURE TYPE TANK THERMALLY

**Ionel PETREA**
“Dunărea de Jos” University of Galaţi, 111, Domnească Street, 800201, Galaţi, Romania
email: ipetrea@ugal.ro

## ABSTRACT

*This paper consider finite element analysis for structural analysis of a tank under thermal loadings. Geometric modeling is done in CATIA and for finite element analysis the author used FEA module in Catia. The aim is to determine von Mises stress field, deformed mesh and thermal loadings.*

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

## 1. Introduction

Finite element method (FEM - Finite Element Method) is one of the best methods of performing calculations and simulations in engineering.

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

The steps in 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 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.

A thermal field in the reservoir structures leads to tension and the tension distribution is useful in the design product phase. Distribution and value of tensions are very important in assembly design of the structure of the tank.

Figure 1 shows the schematic of the mechanical structure of the boiler, composed of a reservoir and heating, for heating of water. The assembly of the tank and heating system is made a specific fitting element.

The application aims to determine maximum values of Von Misses stress, to displacement produced by thermal field acting on the system heating temperature at T = 100°C and the gravitational acceleration g = 9.81m/s$^2$. In this respect, connection with the modeling of the reservoir is achieved through a restriction that requires cancellation of the 6 degrees of freedom.

Analyzed structure is made of steel OL37, with the following mechanical characteristics: longitudinal modulus E = 2.1 x 10$^5$N/mm$^2$, *Poisson ratio* γ = 0.3 and density ρ=7800kg/m$^3$.

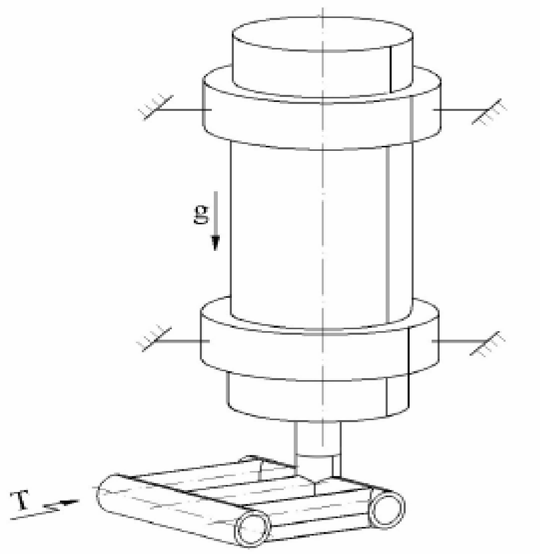


***Fig.1*** *Boiler 3d model*

THE ANNALS OF “DUNAREA DE JOS” UNIVERSITY OF GALATI.
FASCICLE IX. METALLURGY AND MATERIALS SCIENCE
N$^{0}$. 4 – 2013, ISSN 1453 – 083X

## 2. Geometric modeling

The ensemble consists of two components: the reservoir and heating structure. Using CATIA Part Design and CATIA Sketcher modules the authors designed the two components (Figure 2 and Figure 3).

***Fig. 2.*** *Reservoir*

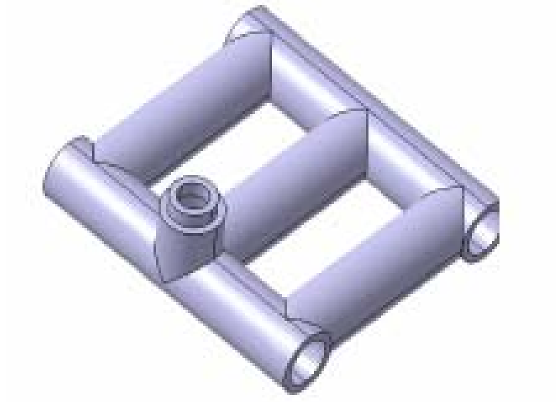

***Fig. 3.*** *Heating structure*

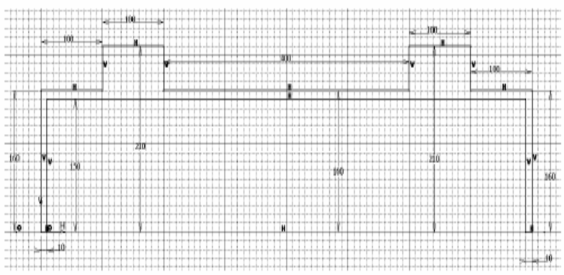

***Fig. 4.*** *Sketch of the reservoir*

Sketch of reference reservoir is done in sketcher module, as seen in Figure 4. Sketch of reference heating structure is presented in Figure 5.

Inserting the component parts are made in CATIA Assembly Design module (Insert / Existing Component) - Figure 6.

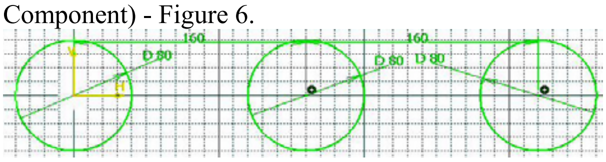

***Fig. 5.*** *Sketch of reference heating structure*

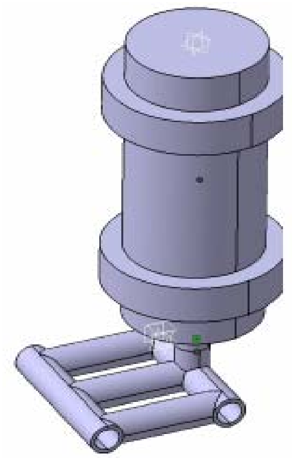

***Fig. 6.*** *Assembly structure*

The assembly is created using geometric constraints between components. *(Contact Constraint and Coincidence Constraint)* Application of material using Apply Material icon. The part is considered carbon steel or alloy steel material, with the following physical properties: thermal expansion coefficient ($1.17\times10^{-5}$K) and admissible strength ($2.5\times10^{8}$ N/m$^{2}$).

## 3. Finite element analysis

First step is to access CATIA Generative Structural Analysis module and we set the type of static analysis (static loading), specification tree simultaneously displaying element with the same name and admissible strength.

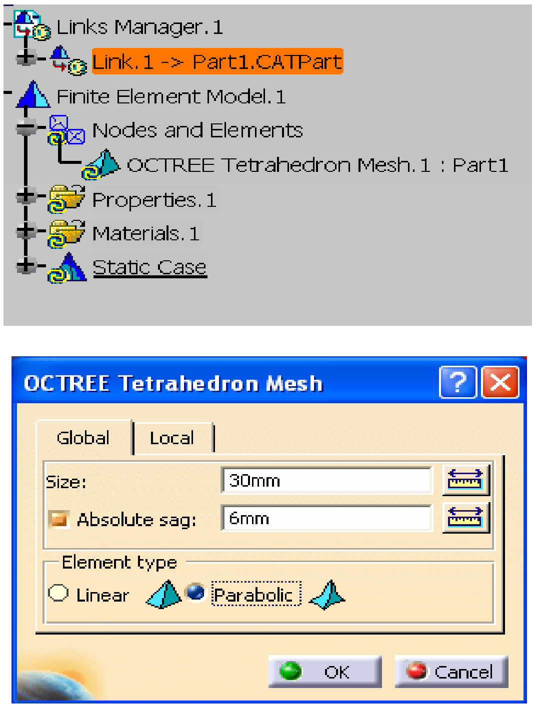


***Fig. 7.*** *Discretization of the part model*

Definition of nodes and elements (a process called meshing). It is determined 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, it is executed double click on "Mesh tetrahedron OCTREE" sub-item found in the specification tree.

Figure 7 presents the specification tree and 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.

***Restrictions definitions***

Modeling the interface between the inlet of the hot water tank and heating pipe structure is achieved by selecting the type geometric constraints coincidence of axes of the two tubes (Pressure Fitting Connection).

Contact condition imposed to the model presumes cancellation of the six possible degrees of freedom associated with the side surfaces of the tank).

***Stress definitions,*** using toolbar Load.

Loading is modeled as an acceleration gravity acting on the elements structural components (select the two elements of the sequence, Vector Acceleration: X= 9.81m/s$^2$, Y= 0N, Z= 0N) and form a thermal field acting on the structure (Temperature Field, Temperature = 373deg) (Figure 8).

***Effective step analysis calculation,*** using “*Compute*” icon.

***The display and interpretation of results,*** with the help on the toolbar *Image*. In Figure 9 is exemplified specification tree, containing a list of three images and their icons. Deformed state of the model is visualized using Deformation command (Figure 10).

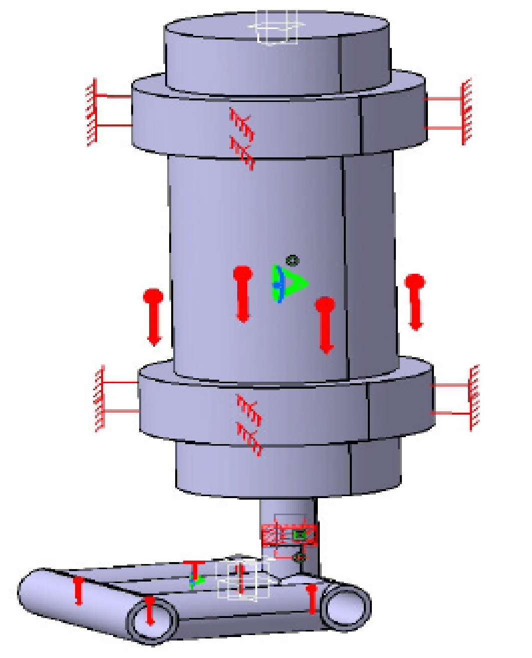

***Fig. 8.*** *Restrictions and stress definitions*

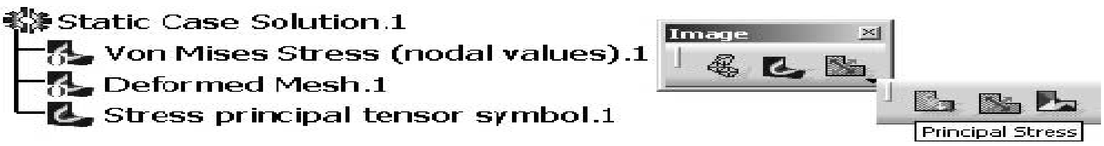


***Fig. 9.*** *The tools of image display and the corresponding menu tree*

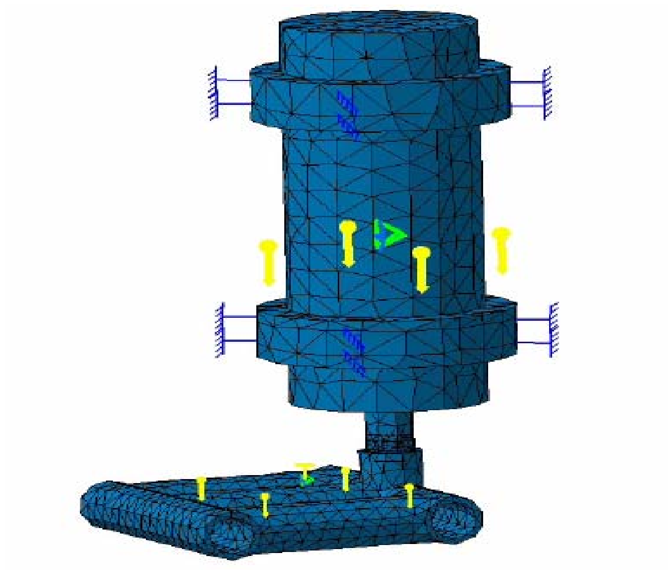

***Fig. 10.*** *Deformed state of the model*

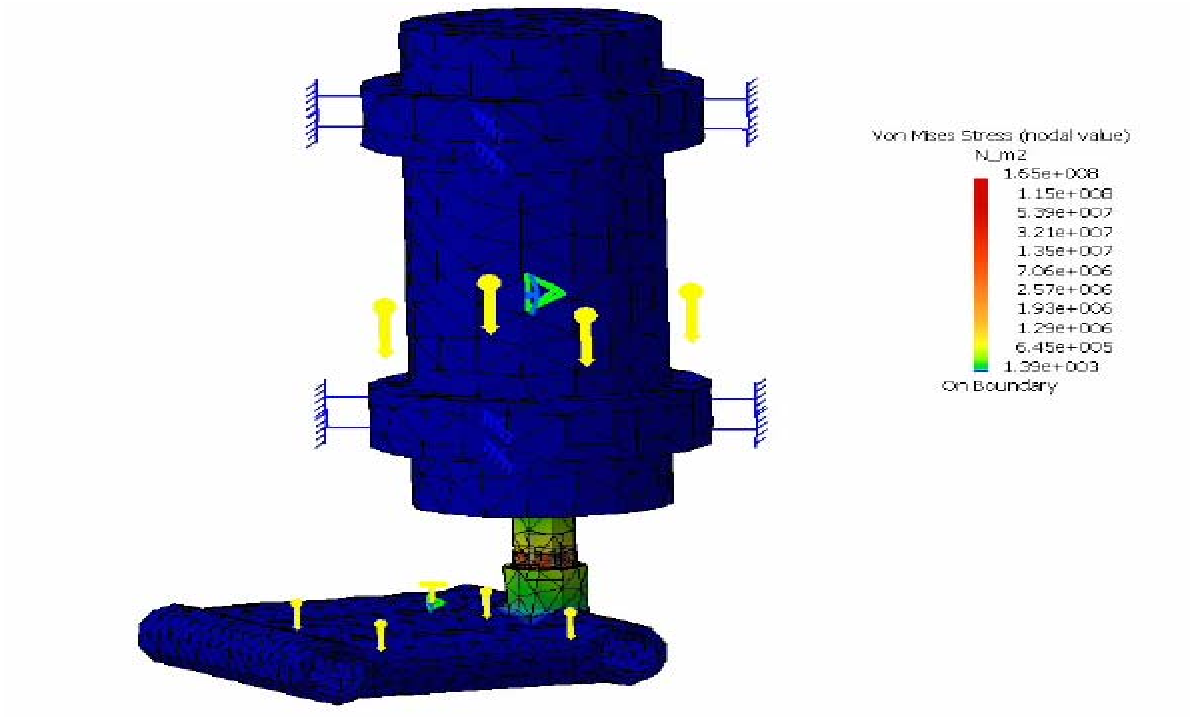


***Fig. 11.*** *Distribution of Von Misses stress*

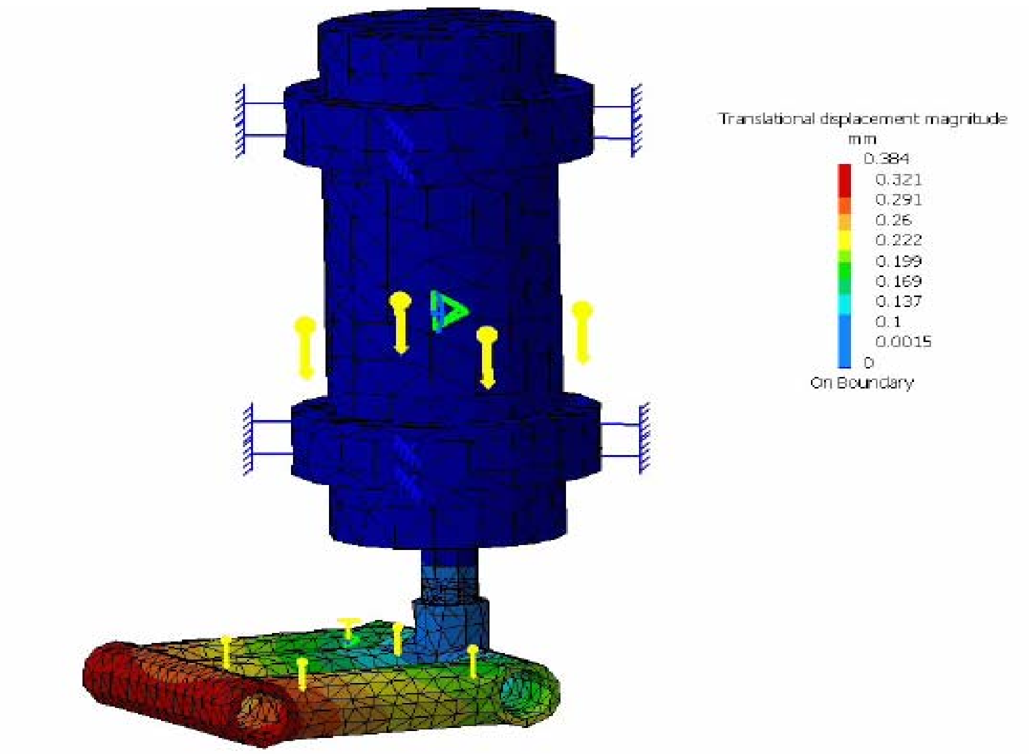


***Fig. 12.*** *Displacement field*

Figure 11 displays the corresponding results Von Misses stress (values of a scalar field energy density obtained from the level of strain). The numerical values of these stresses are displayed along with the Von Misses stress. Also is displayed the nodes location where are located minimum and maximum values of these tensions. To view field trips (in mm) of network nodes as a result of the imposition of conditions and loading restrictions on the model it is used Image tool bar Displacement. Figure 12 shows the color palette and offset values for the three-way junction for a node.

## 4. Conclusions

The analysis of displacement field has shown that the maximum displacements are small (0.384 mm). Maximum Von Misses equivalent stress is found in the assembly and has value of 165MPa.

To reduce these tensions there are constructive and / or manufacturing measures to be taken into account, such as increasing area of assembly or the use of materials with superior mechanical properties).

## References

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