THE ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE IX. METALLURGY AND MATERIALS SCIENCE
N°. 4 - 2025, ISSN 2668-4748; e-ISSN 2668-4756
Article DOI: https://doi.org/10.35219/mms.2025.4.15

STATIC ANALYSIS OF A THERMAL SUPPORT

Ionel PETREA, Marian-Iulian NEACȘU
"Dunarea de Jos" University of Galati, Romania
e-mail: ionel.petrea@ugal.ro, marian.neacsu@ugal.ro

ABSTRACT

In this paper, a simulation is performed using CATIA software and the finite element method, regarding the static analysis of a thermally stressed support. The simulations were performed for several temperatures: 20 °C, 100 °C, 200 °C, and 300 °C, in order to observe the behavior of the support made of different materials at these temperatures.

KEYWORDS: modeling, simulation, support, constraints, thermal stress

1. Introduction

Engineering activity, in general, has as its outcome the creation of technical objects, which materialize as a result of complex production processes. The main stages of the creation of a technical product are: defining a general concept of the product, creating the technical project, establishing the manufacturing technology, creating the experimental model and product approval, and the actual manufacturing [1].

In the design of these structures, an integrated CAD/CAM/CAE software may be used, which incorporates the finite element analysis module FEM [2].

Support-type structures have the role of supporting components, being made of metal profiles. Thermal fields appear in these structures during their operation or during heat treatment processes. Stresses and deformations due to thermal fields are important parameters that must be taken into account in the product design stage.

CATIA (Computer Aided Three Dimensional Interactive Applications), a product of the Dassault Systems company, is one of the most widely used integrated CAD/CAM/CAE systems worldwide, with applications in various fields, including: the machine-building industry, aeronautics, naval, automotive, robotics, agricultural machinery, chemical and food industries, and many others.

The Finite Element Method (FEM) is one of the best existing methods for performing various calculations and simulations in the field of engineering. This method has become a basic component of modern computer-aided design systems.

Analyses performed using FEM are indispensable in all high-performance engineering activities.

FEM involves dividing the part into finite elements, a process called discretization. The result is a network of nodes and elements, called a mesh. At the nodes of the mesh, the program calculates the stresses and deformations that occur in the part, depending on the demands to which it is subjected. Essentially, the system of equilibrium equations between external forces and the stresses that occur is numerically solved.

Calculations performed using FEM represent a very important stage of the design, but can generally be performed only after clarifying other aspects, such as: the beneficiary's requirements, imposed costs, delivery times, available materials and technologies, product durability, production volume, ecological requirements, etc. [3, 5].

Thus, for a given product, some restrictions can be taken into account: the number and maximum value of static and/or dynamic loads, maximum values of deformations, different safety coefficients (for buckling, rupture or fatigue), minimum sensitivity to imperfections in execution, assembly or operation, vibration frequencies, deformation speed in stationary plastic flow, product life, weight, material and its moments of inertia, rigidity under different stresses, static and/or dynamic stability, behavior under different simultaneous loads, etc.

It should be noted that in the CAD-FEM-CAM sequence there is an iterative process of design-calculation-execution. In this process, synthesis and analysis operations of the prototype and the model for finite element calculation are successively performed.

At each iteration of the process, improvements are made to the prototype or computational model until the desired performance is achieved.

THE ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE IX. METALLURGY AND MATERIALS SCIENCE
N°. 4 - 2025, ISSN 2668-4748; e-ISSN 2668-4756
Article DOI: https://doi.org/10.35219/mms.2025.4.15

The finite element analysis of a structural model is, in fact, a numerical verification process. Thus, for a given, dimensionally defined geometry, for a given load and clearly specified support conditions (restrictions), the values of displacements, stresses, support reactions, natural frequencies, etc. are obtained [5, 6].

2. Finite Element Analysis Steps

One of the major advantages of the finite element analysis method is the simplicity of the basic concepts. It is very important for the user of a FEM program to understand and correctly apply these concepts because they include certain assumptions, simplifications and generalizations.

Finite element analysis involves going through the following steps [3]:

I. Geometric modeling of the part or component assembly. The Sketcher and Part Design modules, as well as the Assembly Design module, are used.

II. Applying a material, using the Apply Material icon.

The CATIA material library has a wide variety of materials.

In the practice of finite element analysis, the model material can be homogeneous and isotropic or anisotropic in nature. Also, the elastic and physical constants of the material may depend on temperature and/or stress.

Among the mechanical properties of the applied material, the most important is the admissible resistance, Yield Strength, expressed in MPa. The maximum stresses that occur in the part, calculated by the program, must not exceed the allowable resistance. Otherwise, it is necessary to return to the design stage and apply another material or resize the part.

III. Accessing the finite element analysis module, which includes the following steps:

1. Discretization of the part. By default, in CATIA, the part is discretized (divided into finite elements), but the designer has the possibility of modifying the size and type of the finite elements. Thus, from the command tree, access Finite Element Model/Nodes and Elements/Octree Tetrahedron Mesh, Figure 1.

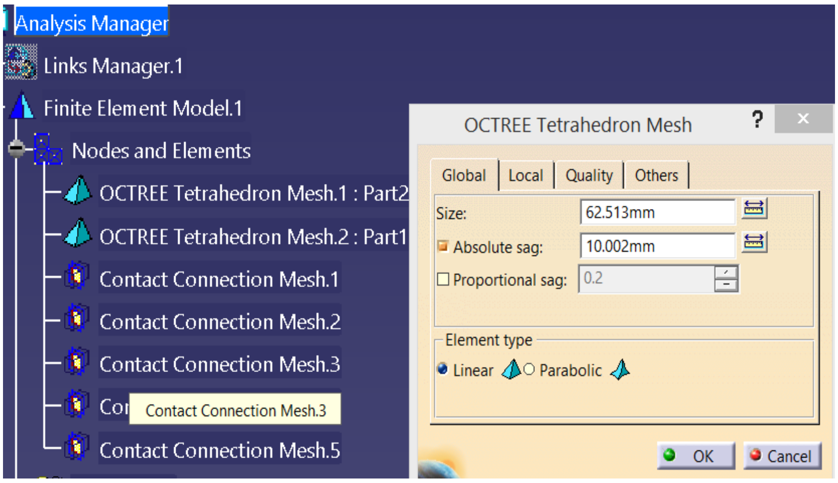

***Fig. 1.** Determining the dimensions of finite elements*

2. Specifying restraints and constraints, using the tools on the Restraints toolbar.

Constraints represent an interface between the analysed model and the subassembly or assembly of which it is a part. If the model is not correctly constrained, the finite element analysis will give significant errors. Some examples of constraints include:

Clamp - constrains all degrees of freedom for a curve or surface.

Surface Slider - allows points on a deformable surface to slide along a rigid surface.

Slider - creates a prismatic joint constraint.

Sliding Pivot - creates a cylindrical joint constraint.

Pivot - are conical joint constraints.

User defined - creates generic constraints, allowing any combination of nodal degrees of freedom to be set.

3. Defining the loads to which the part is subjected, using the Loads toolbar.

The Loads tools add various loads to models that are analysed with FEM. Examples of loads include:

Pressure - this load, expressed in MPa, creates a uniform pressure, applied to a certain surface.

N°. 4 - 2025, ISSN 2668-4748; e-ISSN 2668-4756
Article DOI: https://doi.org/10.35219/mms.2025.4.15

Distributed Force - distributed forces are, in fact, systems of forces statically equivalent to a resultant of a real force.

Moment - moments are systems of forces equivalent to a resultant torque (Nxm).

Bearing Load - simulates a contact load applied to surfaces of revolution.

Acceleration - represents concentrated loads of the acceleration field type (N/kg or m/s^2).

Line Force Density - represents concentrated loads (N/m) of the linear traction force field type.

Temperature Field - creates temperature fields, in degrees Kelvin (K) on the bodies of the analysed models.

4. The calculation stage, using the Adaptivity and Compute toolbars.

The Compute tool triggers the finite element analysis process, but only for the fully constrained and loaded model. It can perform static analysis, frequency analysis and free frequency analysis (Figure 2).

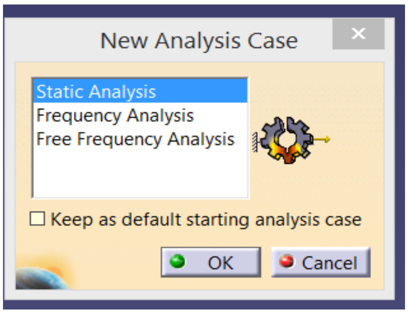

Fig. 2. *Types of analysis with FEM*

The calculator specifies the resources required for the calculation (Figure 3).

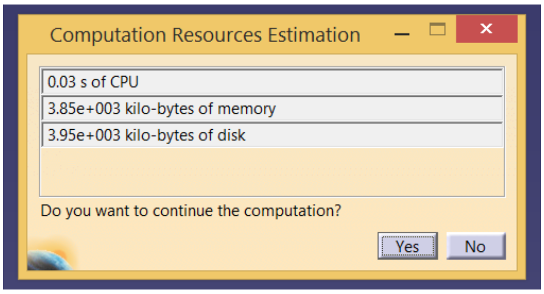

Fig. 3. *Estimated resources for calculation*

With the help of the New Adaptivity Entity tool, the user can refine the network of nodes and elements (mesh), so as to obtain the required precision of the mesh.

In industrial practice, it is considered that the model is correctly solved if it meets an error percentage of approximately 10%.

5. Processing the results - using the tools on the Image bar

3. Static analysis of a thermally stressed support

Thermal fields appear in structures during their operation or during various heat treatment processes. Stresses and deformations due to thermal fields are important parameters that must be taken into account during the product design stage.

The design of mechanical structures subjected to thermal fields and/or mechanical loads requires solving problems by also taking into account the influence of the thermal field on their resistance and deformation, which can be decisive in certain cases.

Figure 4 shows the diagram of a mechanical support-type structure, used to support parts subjected to heat treatments. The model can also be used to cool continuously cast slabs.

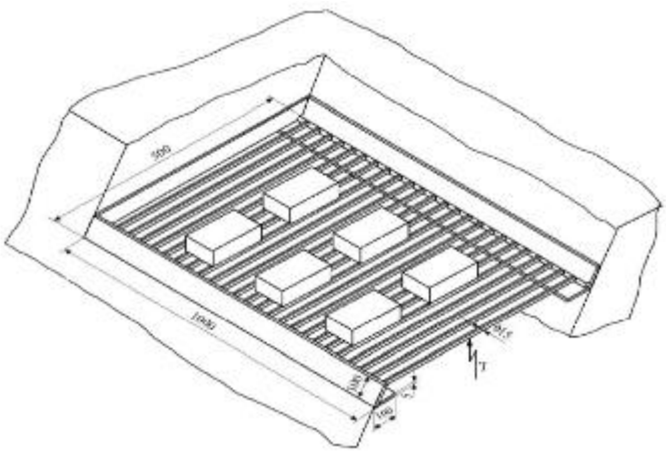

Fig. 4. *Mechanical support structure*

THE ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE IX. METALLURGY AND MATERIALS SCIENCE
N°. 4 - 2025, ISSN 2668-4748; e-ISSN 2668-4756
Article DOI: https://doi.org/10.35219/mms.2025.4.15

The application aims to determine the maximum values of the von Mises equivalent stress and, respectively, the displacement, produced by the thermal field acting on the support system at a temperature of t = 300 °C and by the weight of the parts subjected to heat treatments, G = 2000 N.

The analysed structure is made of OL37 steel, with the following mechanical characteristics:

E = 2.1 · 105 N/mm^2, where E is the longitudinal modulus of elasticity;

ρ = 7800 kg/m^3 represents the density; Transverse shrinkage coefficient (Poisson) = 0.3.

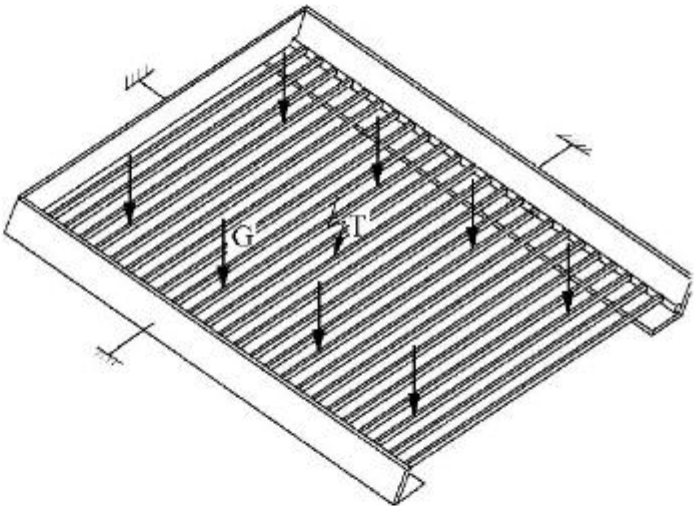

Fig. 5. *The analysis model*

4. Finite element modeling

The main condition for a model to be analysed using the finite element method is that the user assigns a material for it that corresponds to its functional role.

To generate the finite element model, the commands Start ⇒ Analysis & Simulation ⇒ Generative Structural Analysis ⇒ New Analysis Case ⇒ Static Analysis ⇒ OK are followed, which enables the static analysis of the assembly under the conditions of imposed constraints and time-independent loading conditions.

For the component elements of the mechanism, the finite element size and the maximum permissible deviation for geometric modeling sag are chosen according to the figure (the menu being activated by double-clicking on OCTREE Tetrahedron Mesh.1: the part-type elements are successively selected from the specification tree).

Dimensions of 30 mm are imposed for the finite elements, with a maximum deviation of 5 mm, using linear elements (Figure 6).

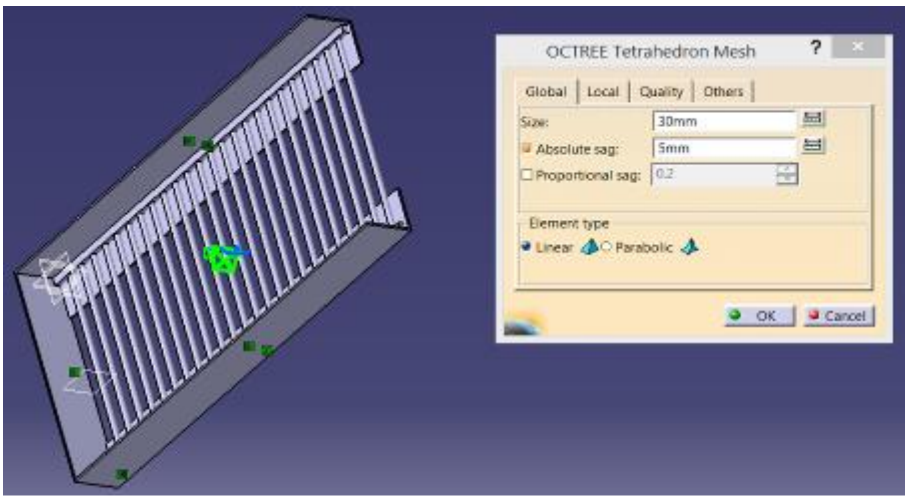

Fig. 6. *Determining the dimensions of finite elements*

THE ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE IX. METALLURGY AND MATERIALS SCIENCE
N°. 4 - 2025, ISSN 2668-4748; e-ISSN 2668-4756
Article DOI: https://doi.org/10.35219/mms.2025.4.15

4.1. Constraint Modeling

In general, constraints can be applied to a single element (Length, Fix, Horizontal, Vertical) or between two elements (Distance, Angle, Coincidence, Parallelism, Perpendicular). Often, constraints are applied to selections consisting of several sketch elements.

A constrained profile does not allow its dimensions to be modified except by editing the component constraints individually. Editing the value of a constraint is done by double-clicking the mouse on the respective constraint, resulting in the appearance of the dialog window (Constraint Definition), which allows the modification of the respective value.

Modeling the connections between the support surface and the frame is done by selecting the geometric contact connections using the Contact Connection command between the frame and the support surface (Figure 7).

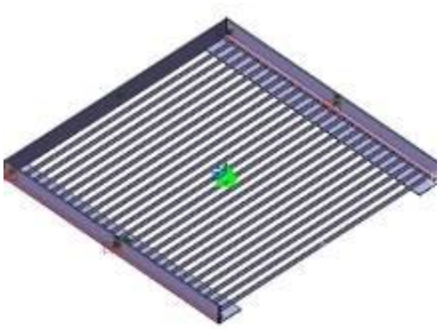

Fig. 7. *Modeling the connections between the support surface and the frame*

The connection with the base imposed on the side surfaces of the frame is defined by cancelling the 6 degrees of freedom associated with the surfaces: (Clamp), Clamp Name: Clamp.1, Supports: two faces selecting the side surfaces of the frame, OK (Figure 8).

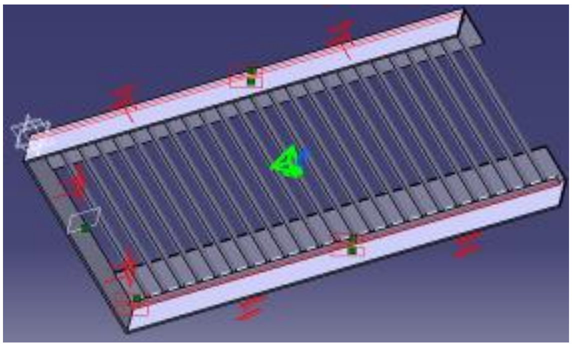

Fig. 8. *Applying restrictions*

4.2. Load Modeling

The loads are modeled in the form of a distributed force acting on the frame (Distributed Force), with Supports: 24 faces, selected as the upper faces of the frame profiles; Force vector: X = 0 N, Y = –2000 N, Z = 0 N, and in the form of a thermal field acting on the support (Temperature Field), applied by selecting the frame; temperature: 573 K, OK (300 °C = 573 K) (Figure 9).

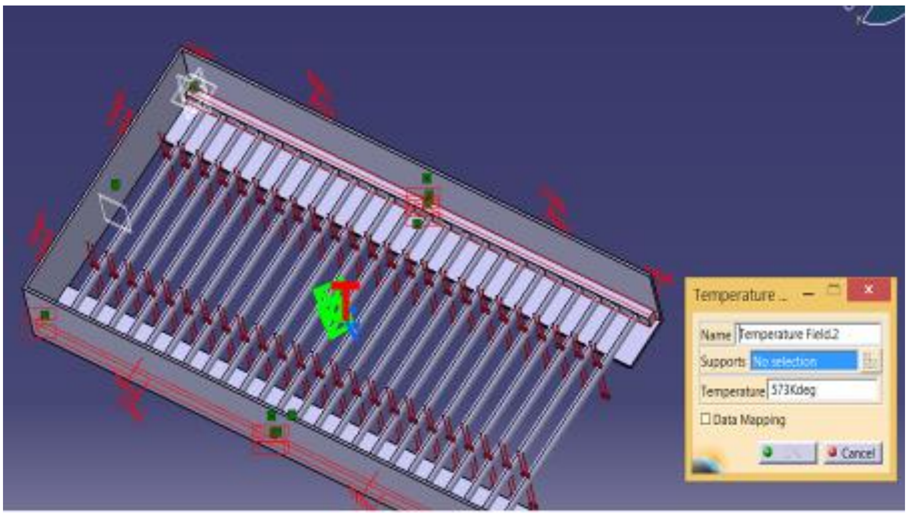

Fig. 9. *Application requests*

4.3. Model Checking

In the model checking stage, information is obtained about the correctness of the generated model: (Model Checker), OK; the green LED is lit and is accompanied by a message confirming the correctness of the model preparation.

Fig. 10. *Model check*

4.4. Model Solving

The model is solved automatically by the software: (Compute) ⇒ Compute ↓All (Figure 11).

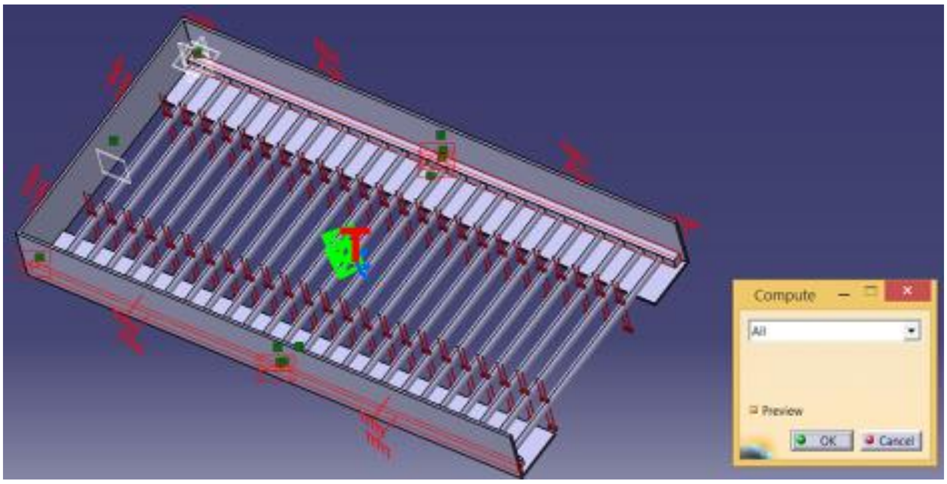

Fig. 11. *Calculation step*

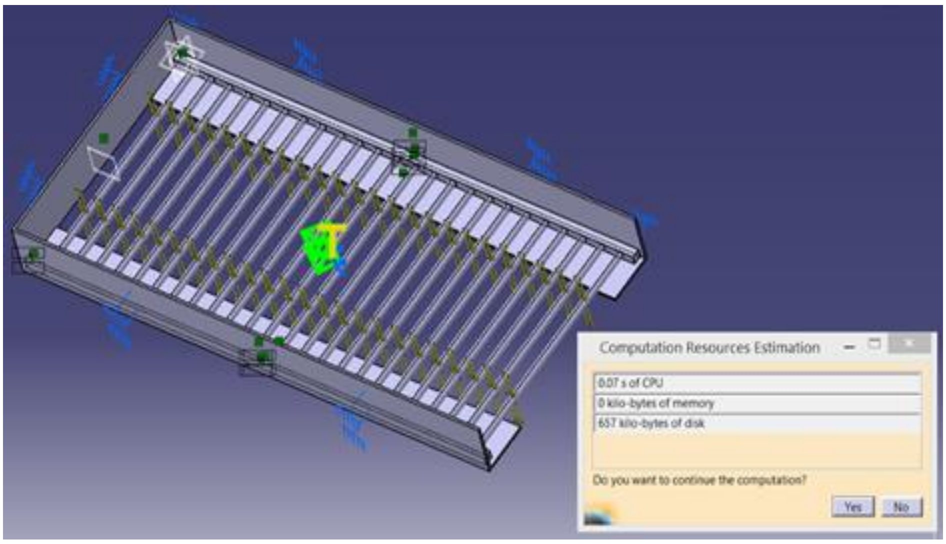

Fig. 12. *Computing resources*

N°. 4 - 2025, ISSN 2668-4748; e-ISSN 2668-4756
Article DOI: https://doi.org/10.35219/mms.2025.4.15

5. Processing the results

CATIA provides the results of the finite element analysis in the form of images. There are five such results:

The deformed state of the model is visualized by activating the Deformation command.

The scale factor is modified by activating the Deformation Scale Factor icon, Figure 13.

Von Mises equivalent stresses are obtained using Stress Von Mises command, Figure 14.

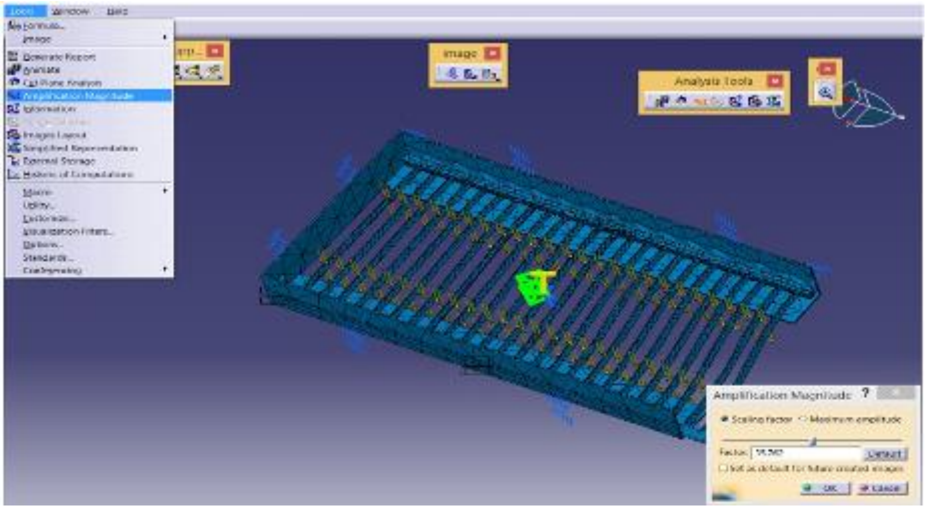

Fig. 13. *Deformed state of the model*

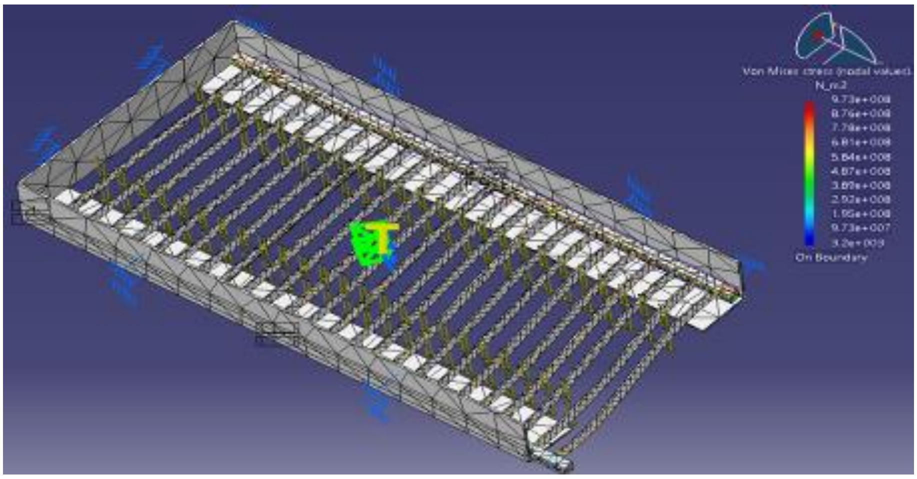

Fig. 14. *Von Mises equivalent stresses*

The maximum of the Von Mises equivalent stresses is found in the support elements and has a value of approximately 973 MPa.

To reduce these stresses, constructive and/or technological measures are adopted, such as increasing the support profile or using materials with superior mechanical properties.

The displacement field is visualized by the command (Displacement), Figure 15, where the maximum displacement reaches 3.38 mm.

The principal stress tensor is presented in Figure 16.

The maximum principal stresses are approximately 329 MPa. These stresses are lower than the von Mises equivalents. However, in the design of the support, the von Mises equivalents must be taken into account, as these stresses are correlated with the yield strength of the steel.

Calculation accuracy and the information window are presented in Figure 17.

The estimated global error rate is about 21%, a value that is unacceptable in the design and simulation process.

To reduce these errors, the simulation is repeated after mesh refinement, considering that the

THE ANNALS OF “DUNAREA DE JOS” UNIVERSITY OF GALATI
FASCICLE IX. METALLURGY AND MATERIALS SCIENCE
N°. 4 - 2025, ISSN 2668-4748; e-ISSN 2668-4756
Article DOI: https://doi.org/10.35219/mms.2025.4.15

repetition of the analysis in order to reduce the global error value is done by applying the New Adaptivity Entity tool.

Thus, in the "Global Adaptivity" dialog window, in the "Supports" field, the "OCTREE Tetraltedron Mesh" element is selected, then in the "Objective Error (%)" field, the target value of 13% is entered.

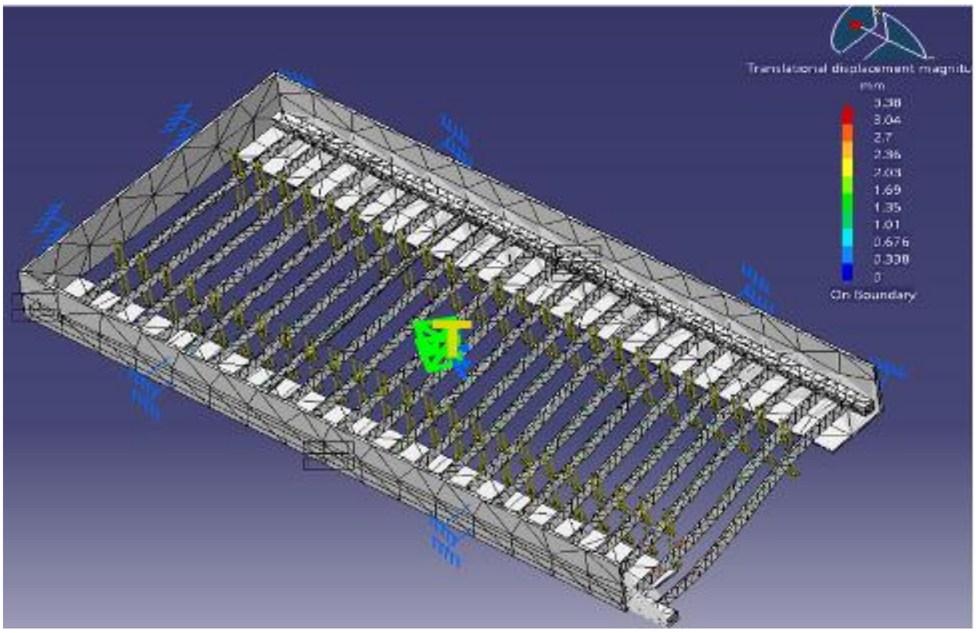

Fig. 15. *The field of displacements*

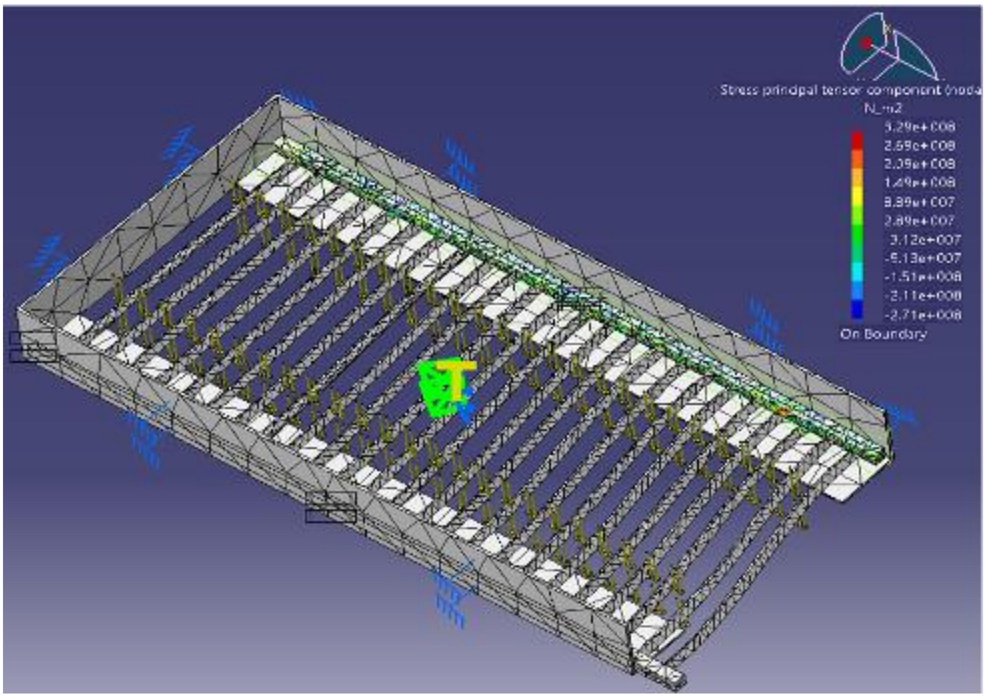

Fig. 16. *Principal stress tensor*

THE ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE IX. METALLURGY AND MATERIALS SCIENCE
N°. 4 - 2025, ISSN 2668-4748; e-ISSN 2668-4756
Article DOI: https://doi.org/10.35219/mms.2025.4.15

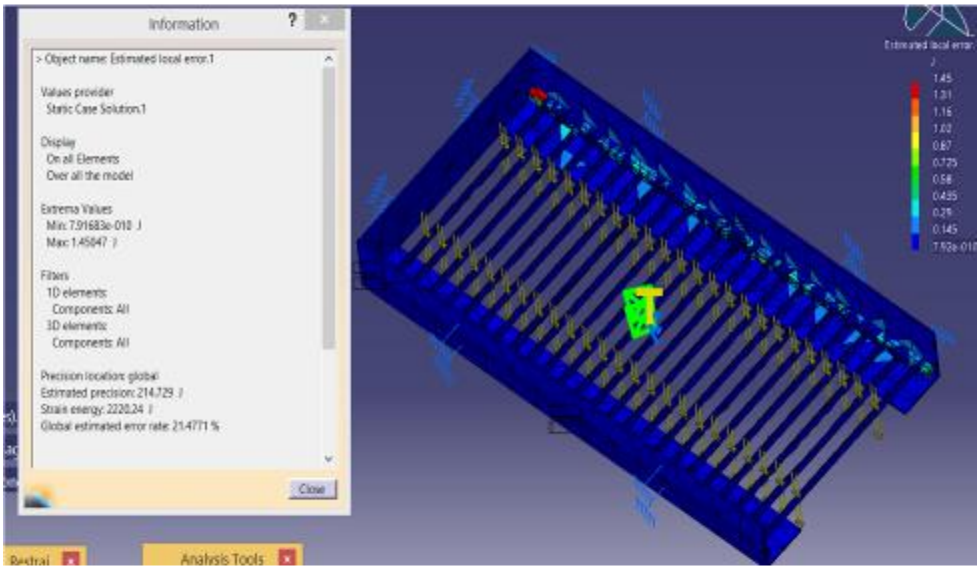

Fig. 17. Estimated calculation error rate

6. Conclusions

The simulations were performed for several temperatures: 20 °C, 100 °C, 200 °C and 300 °C.

The values for the von Mises equivalent stresses, principal stresses and displacements are presented in Table 2.

Table 2. *Stress and strain values*

Temperature	20 °C	100 °C	200 °C	300 °C
Von Mises stress - MPa	8	278	626	973
Stress principal tensor component - MPa	6.2	94	212	329
Translational displacement magnitude - mm	0.04	0.96	2.17	3.38

Table 3. *Yield strength for structural steels*

Steel	OL37	OL44	OL50	OL60
Yield strength $R_{p0,2}$	240	260	290	330

Thus, with an applied force of 2000 N, the maximum stress is 973 MPa at a temperature of 300 °C. These values indicate that none of the steels are suitable for manufacturing the support. Larger dimensions and thicknesses of the support, or a steel with superior mechanical properties, are required.

When simulating with an applied force of 500 N, at a temperature of 300 °C, the maximum stress is 220 MPa. From these values, it follows that all four steels can be used to make the support.

References

[1]. **Ghionea I.**, *CATIA Knowlegle Advisor. Utilizarea regulilor, formulelor şi reacţiilor pentru parametrizarea completă a unui motor hidraulic linear*, Tehnică şi Tehnologie, nr. 3, Editura Tehnic Media, Bucureşti, 2003.

[2]. **Ghionea I.**, *Module de proiectare asistată în CATIA V5 cu aplicaţii în construcţia de maşini*, Editura BREN, Bucureşti, 2004.

[3]. **Marin C., *et al.***, *Modelarea cu elemente finite a structurilor mecanice*, Editura Academiei Române, Bucureşti, 2002.

[4]. **Tache V., *et al.***, *Elemente de proiectare a dispozitivelor pentru maşini-unelte*. Editura Tehnică, Bucureşti, 1985.

[5]. **Trebaol G.**, *Designing parametric spur gears with Catia V5*, 2007.

[6]. *******, *CATIA V5R15*, Documentaţie de firmă. Dassault System, 2005.