International Journal of Modern Manufacturing Technologies
ISSN 2067–3604, Vol. XVI, No. 3 / 2024
https://doi.org/ 10.54684/ijmmt.2024.16.3.84

ASPECTS REGARDING THE MODELING AND THE FINITE ELEMENT ANALYSIS FOR THE STRUCTURAL ELEMENTS OF A SCAFFOLD

Maria Aurora Potirniche [1], Gigel-Florin Căpățână[2]

[1,2]"Dunărea de Jos" University of Galați, Engineering and Agronomy Faculty of Brăila, Research Center for Mechanics of the Machines and Technological Equipments
Calea Călărașilor 29, 810017 Brăila, România

Corresponding author: Maria Aurora Potîrniche, aurora.potirniche@ugal.ro

Abstract: Under the general name of are understood the constructions composed of pipes, joining parts and their accessories, with which auxiliary constructions are executed that serve to execute the final ones, after which they are dismantled and the material is stored until a new use. The consecrated material for these constructions was wood. Along with the advance taken by the final civil and industrial constructions, the need to replace wood with metal material appeared. The use of metal brought a series of advantages and the achievements with the help of metallic materials proved remarkable. At the moment, the basic elements of these constructions are mild steel pipes, assembled with joints pieces and removable accessories, thus constituting a metal structure that can be used in the most varied forms. In this way, unlike the definitive metal constructions in which the sections of the constituent elements each correspond exactly to the effort it must withstand, at the temporary constructions the problem is solved by an easy-to-manipulate material, with elements that can be changed between them. Within the frame of this work, it is wanted to address two aspects, namely: the general behavior of a temporary auxiliary construction made of steel pipes (scaffold type) under the action of functional loads; the analysis of a joining element from the scaffold componency, an element that allows the assembly of two pipes in orthogonal directions. The approach to the two aspects mentioned was possible using tools made available by a specialized software of finite element analysis namely Femap.

Key words: demountable constructions, metal pipes, scaffold, functional loads, analysis of a joining element, specialized software of finite element analysis.

1. INTRODUCTION

Dismountable constructions made of metal pipes are constructions composed of pipes, connecting parts and their accessories, with the help of which auxiliary constructions are executed that serve to execute the final ones. After they have fulfilled their functional role, the auxiliary constructions are dismantled and the material is stored until a new use [1], [2].

For a long time, the material used for these constructions was wood. The momentum taken by the final civil and industrial constructions on the one hand, and, on the other hand, the use of wood in other areas of great industrial importance led to the need to replace wood with metal material.

The use of metal material brought a number of advantages:

- lowering the cost of assembly and disassembly work;
- almost complete recovery of the metal material;
- the possibility of creating larger gaps between the elements of the metal structure, which is important when these gaps must be used, for example in structures over watercourses or traffic arteries;
- creating a homogeneous structure with smaller deformations;
- the possibility of creating multiple and varied types of structures.

The achievements with the help of metallic materials proved remarkable [3].

The basic elements of auxiliary constructions are mild steel pipes, assembled with joints and removable accessories. In this way, a metallic structure is formed that can be used in the most varied forms. A particular advantage is the material of the temporary structure which is easy to handle, with elements that can be changed between them [4].

The choice of metal pipes for the construction of temporary structures is based on technical and practical criteria. From a technical point of view, the pipe section has the same moment of inertia in relation to any diameter. Following a comparative analysis between the pipe-type section and other sections, it was concluded that the pipe is therefore 2.7 times more resistant than the L profile with the equivalent section, which is an important advantage. The pipe is also 1.8 times lighter than the L profile [5], [6].

From a practical point of view, the pipe is smooth, has no corners that constitute rust deposits, is easy to store, easy to transport, clean and paint. When assembling it does not present any difficulty, because, due to the symmetry in relation to its longitudinal axis, it does not require any complicated operation, being able to be put into operation anyway with respect to this axis. In this way, time is saved and assembly mistakes cannot be made. This is all the more important, as pipe structures must be assembled and disassembled many times and these operations are often done by unskilled persons [7], [8].

2. MATERIALS AND METHODS

Within this subchapter is wanted to address two aspects:

- the general behavior of a temporary auxiliary construction made of pipes (scaffold type) under the action of functional loads;
- analysis of a joining element from the scaffolding, an element that allows the assembly of two pipes in orthogonal directions.

a) The general behavior of a temporary auxiliary construction made of pipes (scaffold type) under the action of functional loads

The dimensions of a temporary auxiliary construction made of pipes are determined by its destination and the loads it will have to bear, as well as by constructive considerations, taking into account that the material used is a metal pipe. The scaffolding used for the case study was created in the Femap finite element analysis software and its construction took into account the prescriptions in force regarding the dimensions of such resistance structures. A first dimension that had to be chosen was the height of the scaffolding. A scaffold with a total height of 7.5 m was modeled, with a floor height of 1.5 m and two work platforms located at 3 m and 6 m from the ground. The prescriptions recommend a height of a working platform of 3 m, because it saves material, but the delays that occur both when assembling the scaffolding and when dismantling it must be taken into account, due to the additional devices that must be inserted in order to workers to get from one floor to the next floor above.

The second dimension to choose was the distance between the axes of the pillars both in the longitudinal sense, called the span opening, and in orthogonal sense, called the distance between the construction planes. A total dimension in the longitudinal plane of 8 m was chosen, the distance between the axes of the pillars being 1 m, and for the distance between the construction planes a distance of 1 m was chosen. To determine these dimensions, a practical rule was taken into account which requires that the mild steel pipes used for these constructions should have a slenderness of less than 350, to reduce the danger of their damage during current handling, loading and unloading from trucks or wagons [9], [10], [11].

Within the preprocessing module in Femap, a homogeneous and isotropic material was defined from which its structure and mechanical characteristics will be made.

The type of finite elements suitable for the construction of a structure made of pipes was chosen, namely one-dimensional finite elements of the *beam* type. By choosing the *beam* finite element type, a rigid connection type was imposed between the nodes of the structure. When setting the beam type property, a diameter ϕ 60x2 was chosen for the pipes of the structure.

The nodes resting on the ground were supported using spatial joints, and the structure was loaded with a functional load of 300 kg (corresponding to a force of 3000 N) at the level of each work platform.

An observation would be that only the metal construction made of pipes was studied, not taking into account the fact that, at the level of the work floors, on the metal construction the work platforms made of wooden planks are placed.

b) Analysis of a joining element of the scaffolding

The joint parts have the role of connecting or join two pipes. There are also constructions of joint parts that simultaneously join several pipes, such as spherical joints.

The nodes are formed by pairs of pipes with crossed axes that make angles of 90^0 between them, two by two, and orthogonal connections are used to connect them [2].

According to the number of pipes that meet in a node, the nodes are single, double, triple or multiple.

Simple nodes are those where two pipes assembled with an orthogonal connection or stop-connection meet.

Double nodes are those in which three pipes are joined, being formed by two simple nodes. The joints are made with orthogonal connections [3].

Triple nodes are those in which four pipes join, being formed by three simple nodes.

Joining more than four pipes results in multiple nodes, such as concurrent pipe nodes.

The orthogonal connections intervene most often in demountable constructions, having a primary role and determining their strength and stability.

The design variant adopted for the connection depends on the production volume:

- for small manufacturing volumes or one-off production, it is possible to use the machining through cutting of the entire piece from the full semi-finished product;
- for large production volumes, a cast semi-finished product can be made, on which subsequent processing is applied by cutting to obtain the final shape and dimensions.

For the case study in this work, an orthogonal connection type clamping element was chosen that allows the assembly of two pipes in orthogonal directions [9], [10], [11]. The modeling was done in the Inventor assisted design software and is presented in fig. 1.

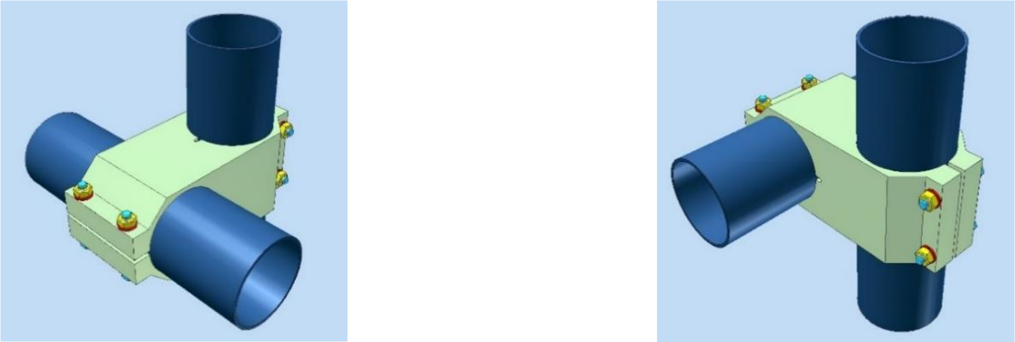


Fig. 1. Orthogonal connection for assembling two pipes

The connection was designed to join two pipes with the diameter ϕ 60x2 and has the dimensions 180x66x66. The geometry of the connection was changed at the ends by removing some parts of the material. This was done in order to consume less material, to ensure an aesthetic appearance, but also for good maneuverability at the level of the tightening screws. An elastic channel was also made on the opposite side of the screw-tightening area, which allows the material to deform when the screws are tightened. The elastic channel was made with a diameter of 3 mm.

The joint has been imported into the Femap finite element analysis software for analysis.

A material for the resistance structure was declared, isotropic and homogeneous, as in the previous case of the general structure formed by pipes.

For the choice of the type of finite elements, it was taken into account that the connection was modeled as a solid and a discretization with tetrahedral three-dimensional finite elements was chosen.

The model was supported using embedding on the entire surface around which the finite elements were rotated in the previous step (a square-shaped surface 66x66).

The loading with unitary force was done on the circumferences of the circles practiced for inserting the screws (only on the part on the outer surface), so that the forces have a compression effect, simulating the closing of the connection and the tightening of the pipes.

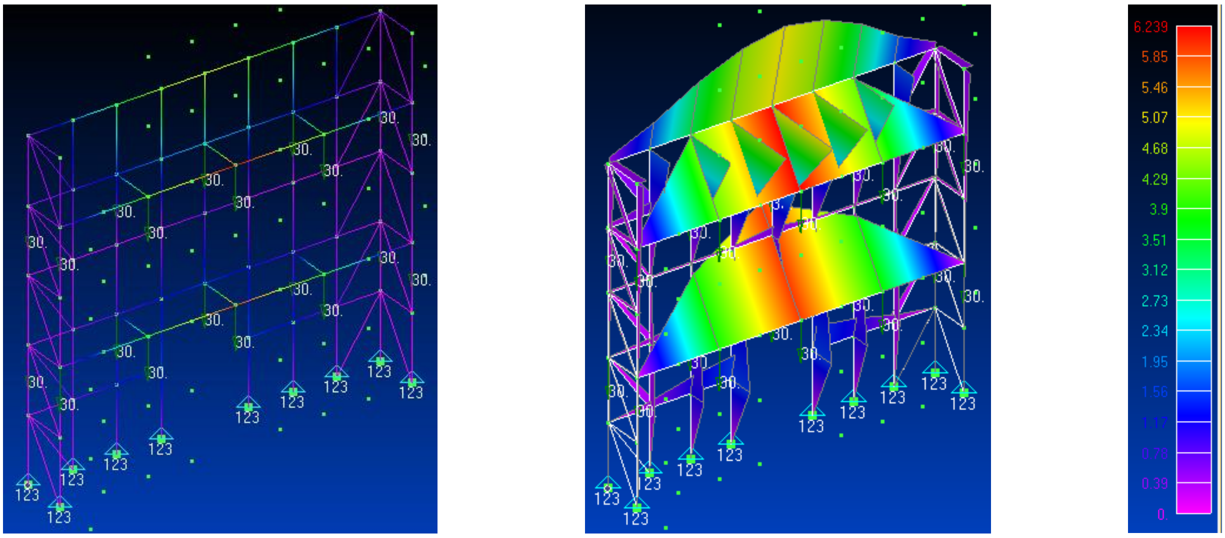
3. RESULTS AND DISCUSSION

The analysis and interpretation of the results for the scaffolding requires the establishment of the areas of the model of the resistance structure where the maximum values of the interest characteristics are produced, as well as the mode of their variation on the whole of the analyzed model, or on a subdomain thereof.

Solving the problem in the present case considered obtaining results that show where the dangerous section occurs in terms of bending stress at the level of the entire structure, what is the maximum displacement of a node of the structure and what is its value, viewing the diagrams of bending moment on each of the pipes of the structure.

From the data made available to the user by the finite element analysis software Femap, some concrete values were chosen that reflect the response of the structure to the action of functional loads:

- the maximum displacement of a node of the structure occurs in the plane of symmetry of the scaffolding, at the level of a node of the structure located on the outer edge of the working platform located at a height of 6 m and has a value of 6.239 mm (figure 2); the variation graph of displacements at the nodes for the scaffold type structure is shown in figure 3;
- the maximum bending moment occurs on an element located in the plane of symmetry of the structure, at the level of the work platform located at a height of 3 m and has a value of 13532 Nmm (figure 4); the variation graph of the bending moment for the scaffold type structure is shown in figure 5;
- the maximum bending stress occurs on an element located in the plane of symmetry of the structure, at the level of both work platforms and has a value of 4.785 N/mm^2 (figure 6); the variation graph of the bending stress for the scaffold type structure is shown in figure 7.



(a) (b) (c)

Fig. 2. Representation in Femap of displacements at nodes
(a) representation of displacements on the contour; (b) representation of displacements as a diagram; (c) the displacements values at nodes [mm]

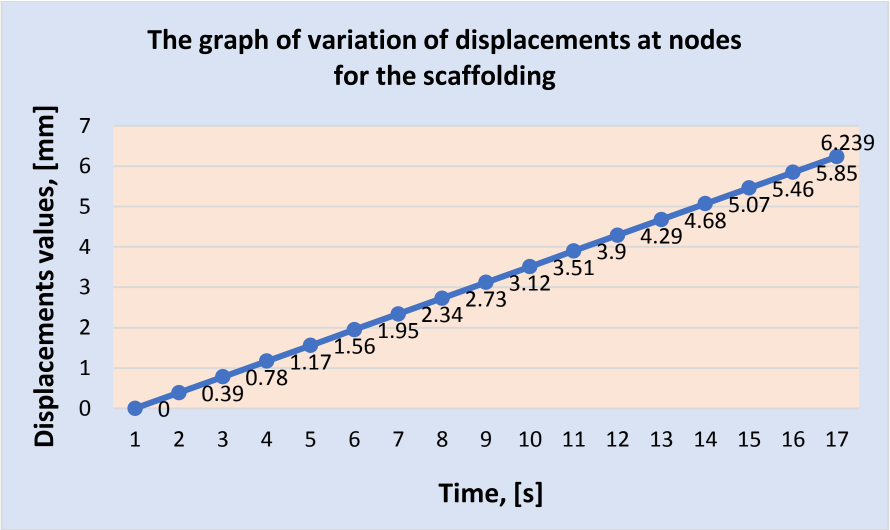
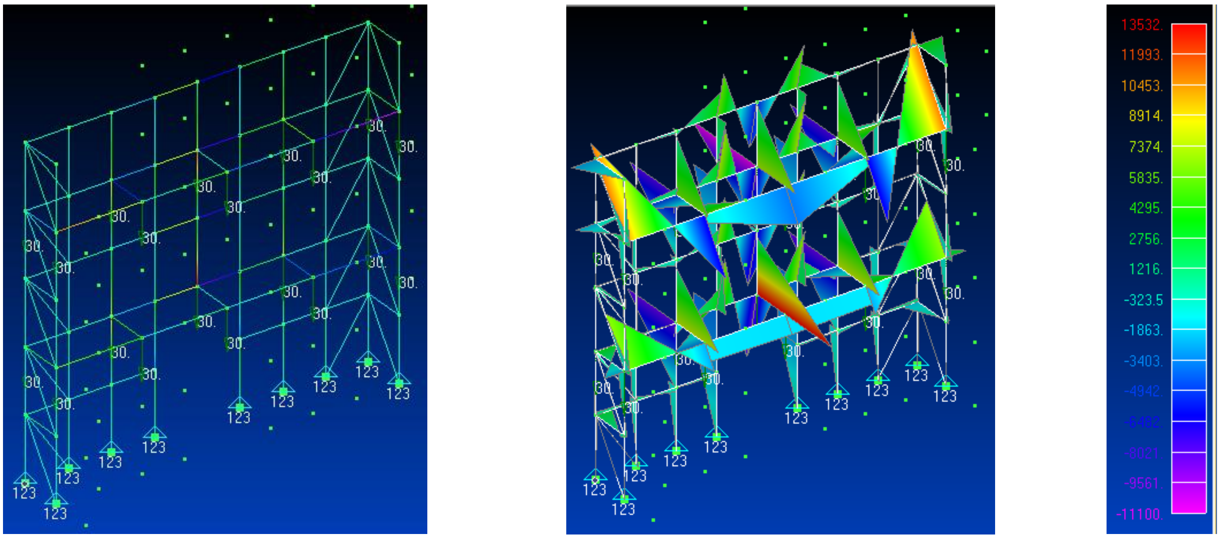


Fig. 3. The variation graph of displacements at the nodes for the scaffold type structure



(a) (b) (c)

Fig. 4. Representation in Femap of the bending moment
(a) representation of the bending moment on the contour;(b) representation of the bending moment diagram; (c) the bending moment values [Nmm]

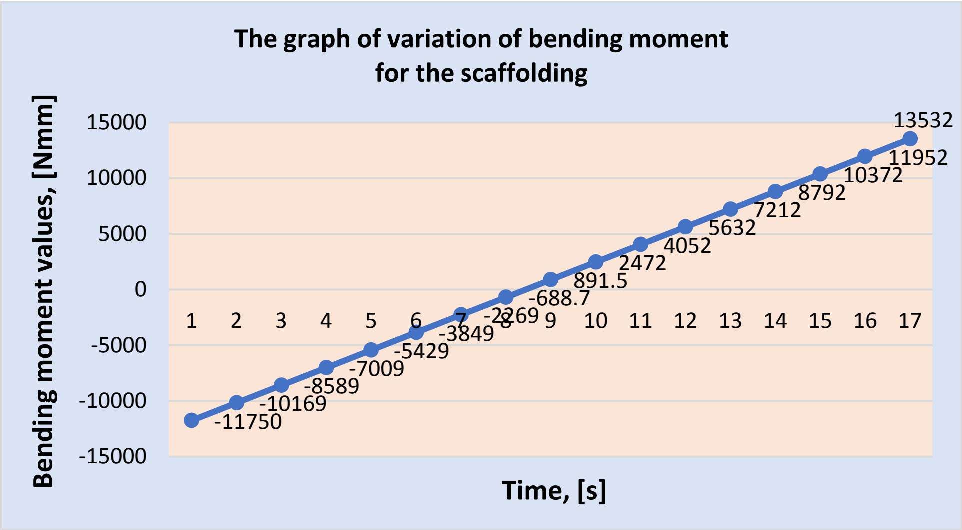
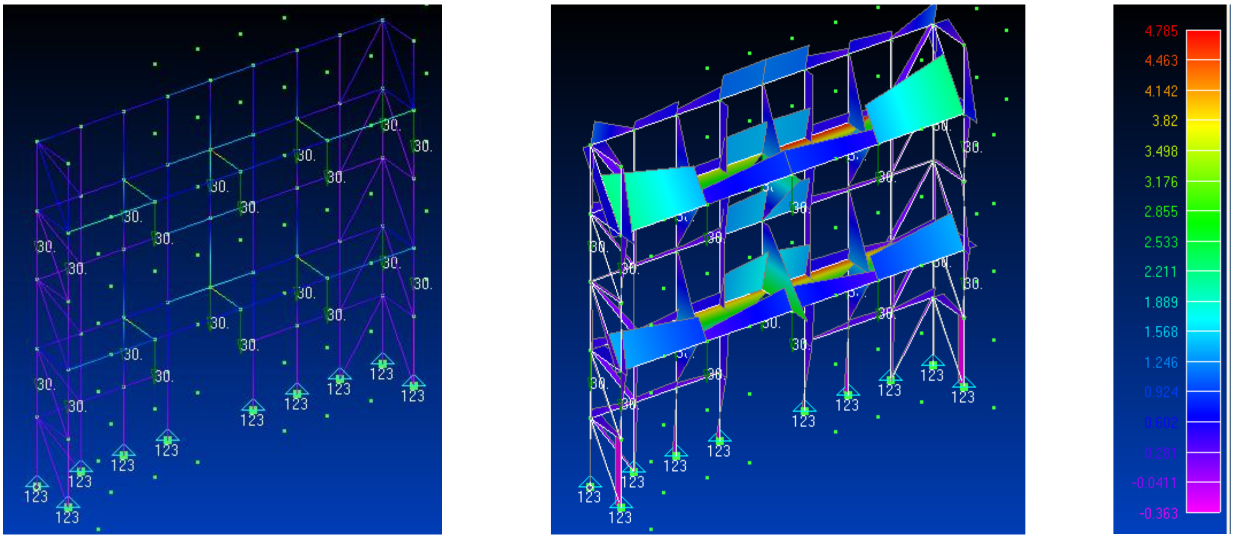


Fig. 5. The variation graph of bending moment for the scaffold type structure



(a) (b) (c)

Fig. 6. Representation in Femap of the bending stress
(a) representation of the bending stress on the contour
(b) representation of the bending stress diagram; (c) the bending stress values [N/mm^2]

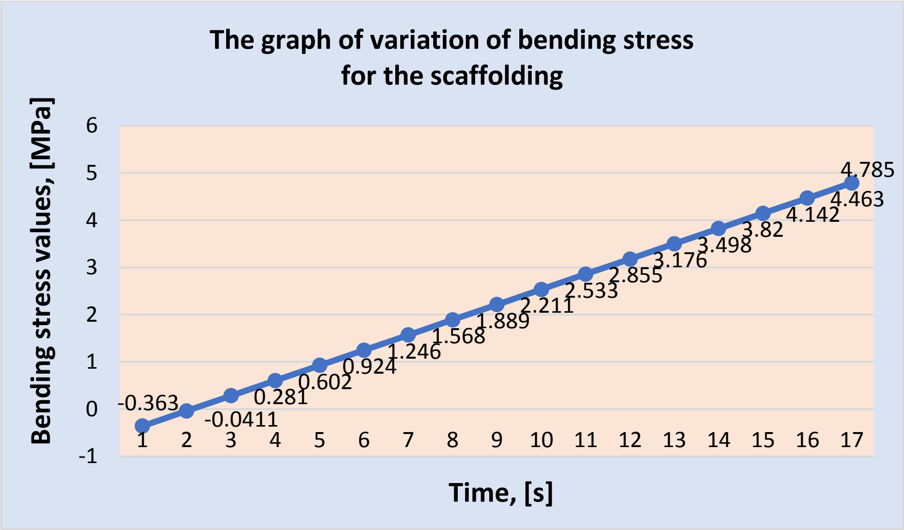


Fig. 7. The variation graph of bending stress for the scaffold type structure

Regarding the results obtained in the Femap software for the orthogonal connection, in the following, some concrete values obtained for displacements at the nodes and for stresses on the finite elements will be presented:

- the maximum displacement of a node of the structure occurs under force, next to the circles practiced for inserting the screws and has a value of 0.144 mm (figure 8); the variation graph of displacements at the nodes for the orthogonal connection is shown in figure 9;

- the maximum stress, calculated with the help of a theory of resistance suitable for the tenacious material used to make the structure (in the present case, the 5th theory of resistance is used, the Hubber – Henkey – von Mises theory), appears near the ends of the holes practiced for the introduction of pipes and has a value of 5.473 daN/mm2 (figure 10); the variation graph of the bending stress for the orthogonal connection is shown in fig. 11.

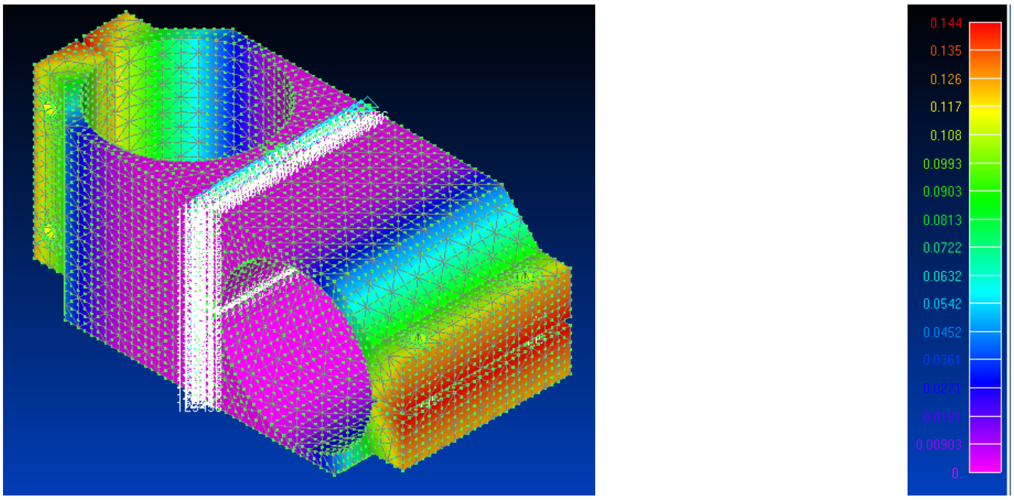


Fig. 8. Representation in Femap of displacements at nodes on the contour and their values [mm]

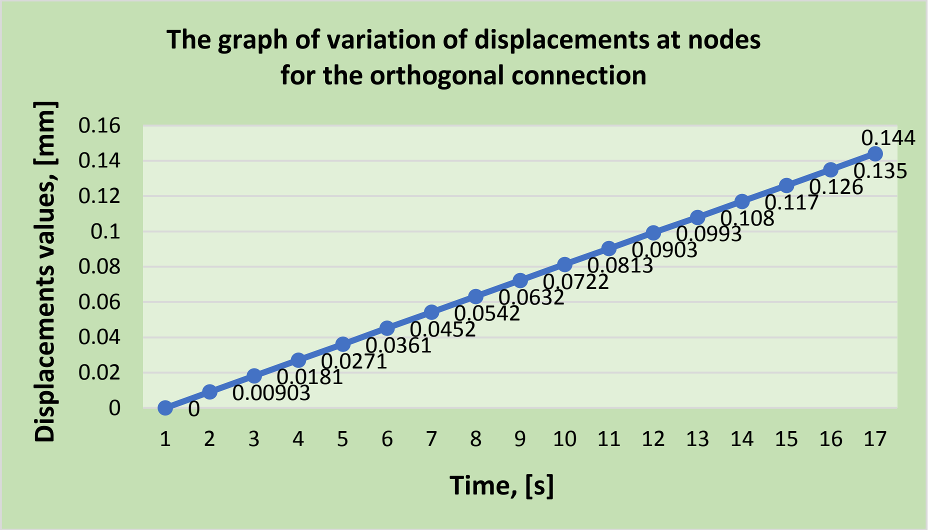


Fig. 9. The variation graph of displacements at the nodes for the orthogonal connection

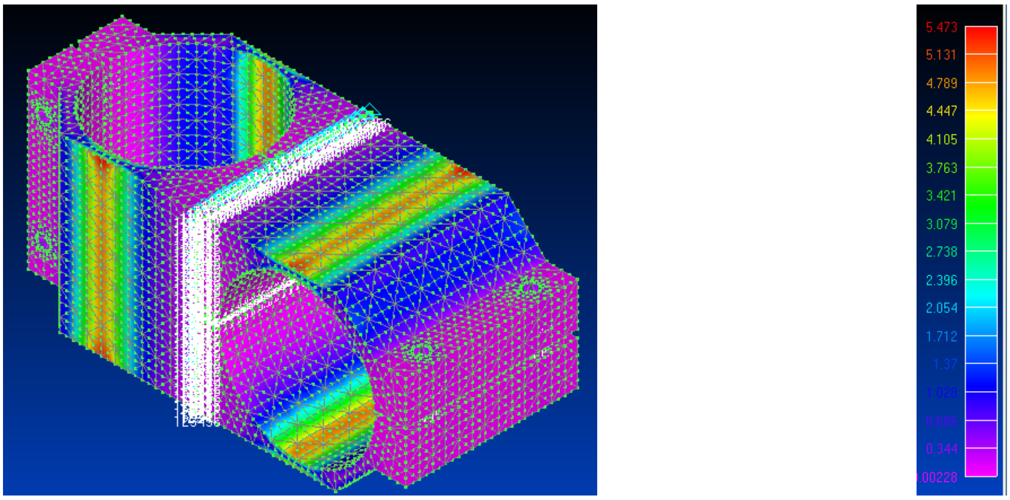


Fig. 10. Representation in Femap of bending stress on the contour and his values [N/mm^2]

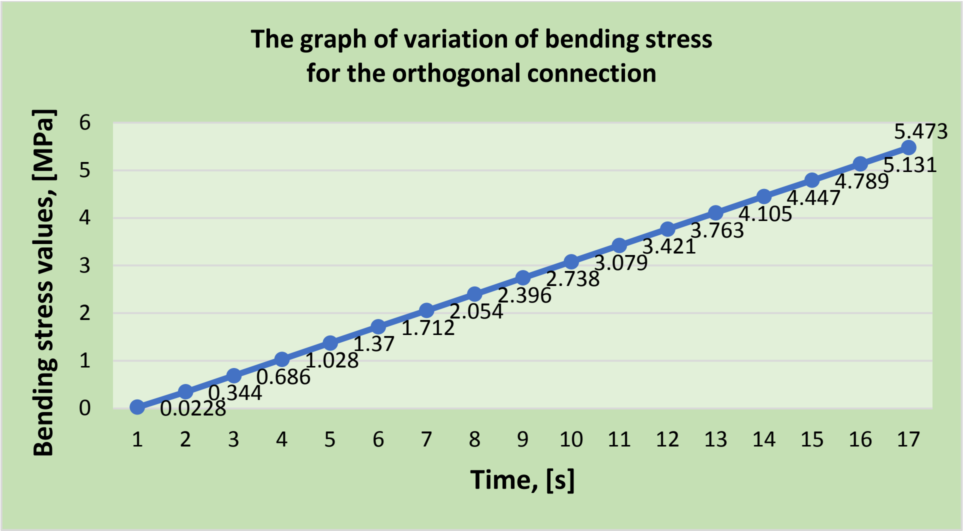


Fig. 11. The variation graph of the bending stress for the orthogonal connection

Interesting in the case of modeling solids and their analysis is the tool provided by the finite element analysis software called *Advanced Post* with the help of which it is possible to visualize, using planes implicitly defined from the software or user-defined, interest parameters such as:

- bending stresses on surfaces that move inside the solid (*Dynamic Cutting Plane*) (figure 12);
- the areas (surfaces) in the solid that present the same stress (*Dynamic Isosurfaces*) (figure 13);

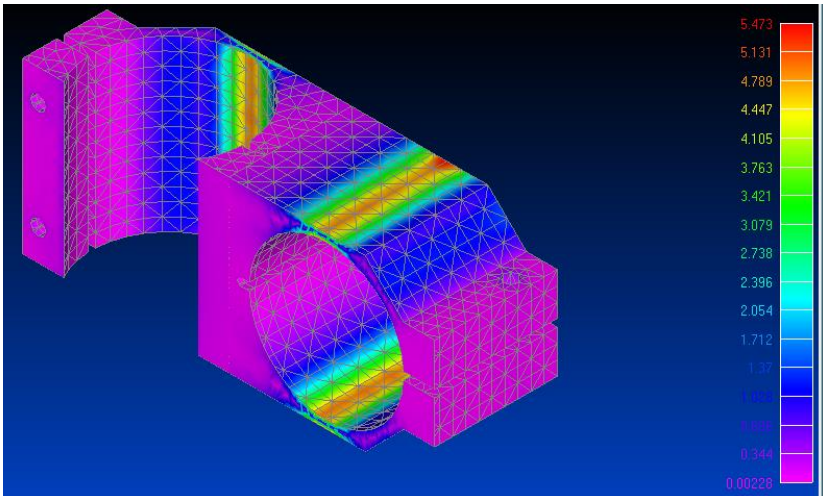


Fig. 12. Representation in Femap of the bending stress in a plane inside the part, using the *Dynamic Cutting Plane* command [N/mm^2]

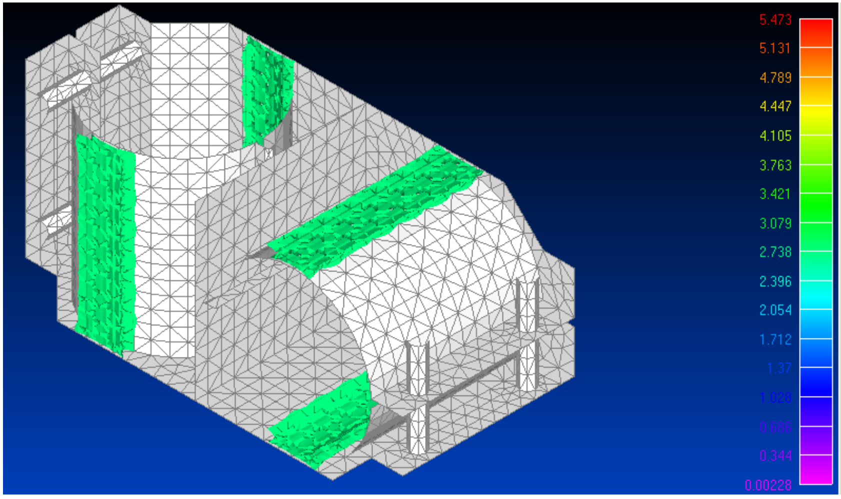


Fig. 13. Representation in Femap of surfaces that have the same bending stress values, using the *Dynamic Isosurfaces* command [N/mm^2]

- displacements at nodes in a plane inside the solid (*Dynamic Cutting Plane*) (figure 14);
- the areas (surfaces) in the solid that present the same displacements at the nodes (*Dynamic Isosurfaces*) (figure 15);

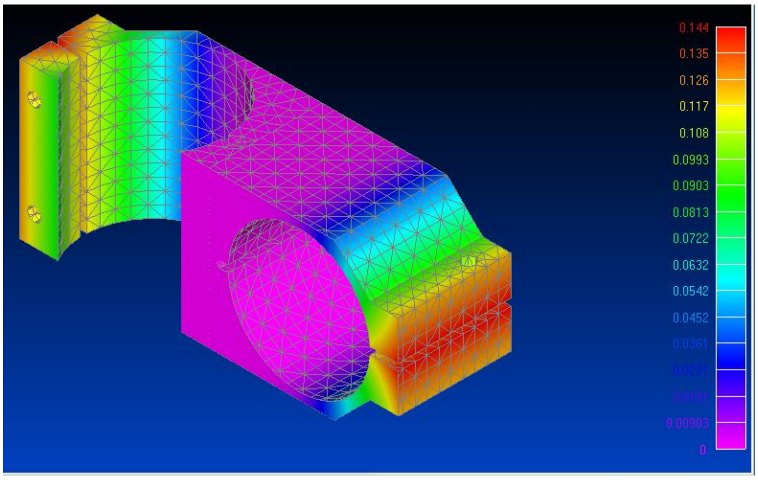


Fig. 14. Representation in Femap of the displacements at nodes in a plane inside the part, using the *Dynamic Cutting Plane* command [mm]

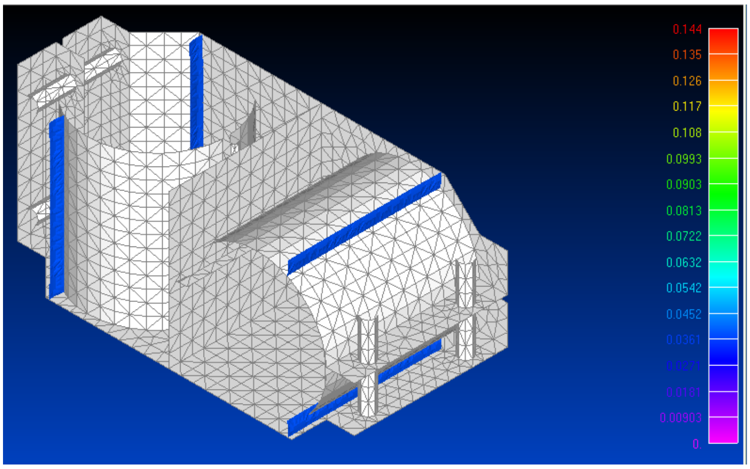


Fig. 15. Representation in Femap of surfaces that have the same displacements at nodes values, using the *Dynamic Isosurfaces* command [mm]

- potential deformation energy stored by the structure on surfaces that move inside the solid (*Dynamic Cutting Plane*) (figure 16);

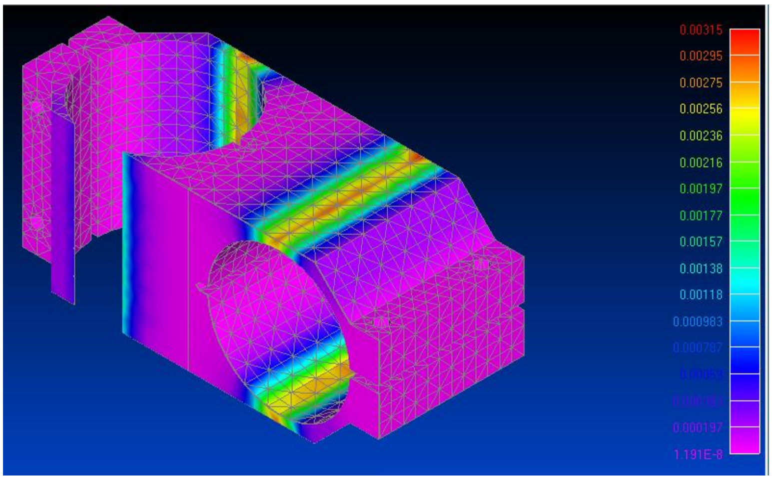


Fig. 16. Representation in Femap of the potential deformation energy stored by the structure in a plane inside the part, using the *Dynamic Cutting Plane* command [J]

- the areas (surfaces) in the solid that present the same potential deformation energy (*Dynamic Isosurfaces*) (figure 17).

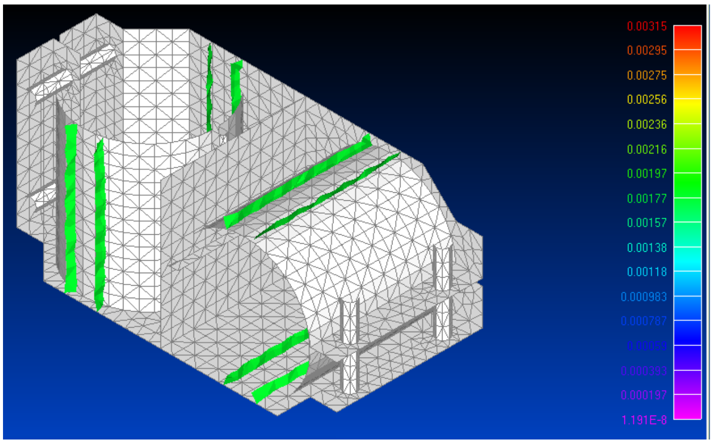


Fig. 17. Representation in Femap of surfaces that have the same potential deformation energy values, using the *Dynamic Isosurfaces* command [J]

The variation graph of the potential deformation energy is shown in figure 18.

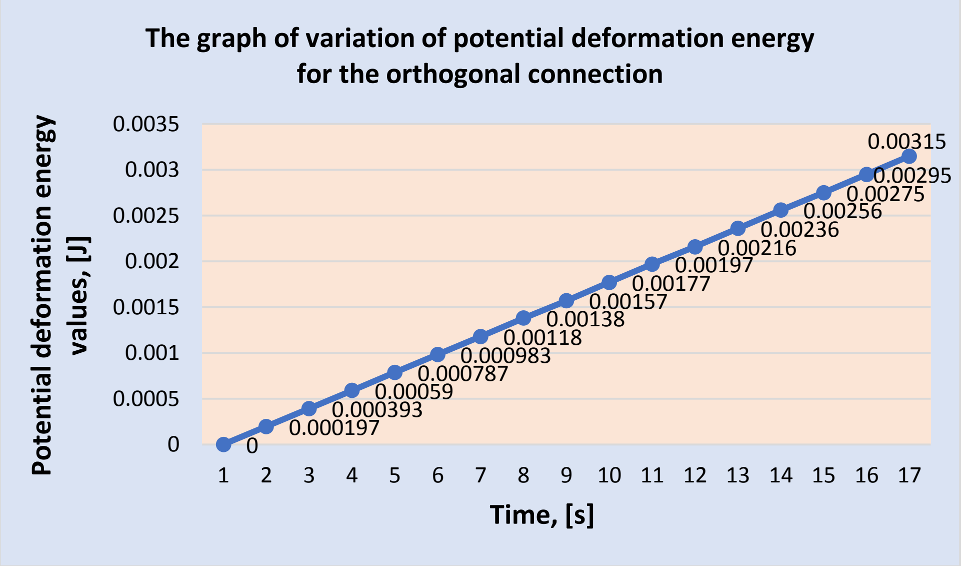


Fig. 18. The variation graph of the potential deformation energy for the orthogonal connection

The visualization of the bending stresses inside the solid using the *Dynamic Cutting Plane* command is very important for specialists in the domain because it is possible to know exactly the dangerous section or sections, where the stress takes maximum values and measures can be taken to reduce the stress value.

It is also interesting to visualize the potential deformation energy accumulated by the structure and its values. If the structure is stressed within elastic limits, the specialists in the field can know the value of the potential deformation energy that the structure releases, after removing the stress, to return to its original shape and dimensions.

Based on the potential deformation energy, the deformations of the structure can be calculated and measures can be taken to design an optimal variant for the respective resistance structure.

4. CONCLUSIONS

The scaffolding used for the case study in the paper is not a fictitious structure, but a real one, made based on the prescriptions in force regarding the dimensions and functioning of such resistance structures.

For the construction of the scaffolding, a circular pipe type section was chosen following a comparative study between several types of equivalent sections. The circular pipe presents important advantages both from the point of view of material consumption and from the point of view of resistance, which is why it is the most suitable for the realization of metal constructions.

For the orthogonal connection in the work, the connection with which the pipes are joined at the nodes, a model was chosen with a geometry that is easy to make and also easy to modify depending on the diameter of the pipes used. In this way, the obtained geometric constructions are varied and easily adapt to the environment and working conditions. The constructive solution chosen for the connection allows for easy assembly or disassembly (just by tightening or loosening 4 screws) and makes a rigid and safe joint.

By using the chosen orthogonal connection, a high clamping force is ensured, and the structure where this joining element is used can take on large loads.

A great advantage is the fact that inserting the pipe into the connection is done directly and easily.

The connection was subjected to stresses through a uniformly distributed load along the circumference of the 4 holes made for the screws; as the resultant of the distributed loads, a unitary force was used, used in order to find the maximum stresses from the orthogonal connection. This calculation method takes into account the maximum stresses and the admissible resistance of the material and leads to the identification of a factor for multiplying the applied load. In this way, the maximum load that can be applied to the connection, called the capable load, can be determined.

The calculation carried out in the Femap finite element analysis software greatly reduced the user's workload because, by simply changing some parameters, various models can be obtained that can be quickly analyzed. The analysis of several models offers the possibility to choose the optimal variant of the structure required by the user.

The optimal variant can be made according to different criteria imposed by the user and by the practical domain (material consumption criterion, resistance criterion, rigidity criterion, aesthetic criterion).

In the model chosen for the scaffolding in the current work, an optimization has already been achieved both from the point of view of material consumption and from the point of view of the structure's resistance.

In the model chosen for the orthogonal connection, the geometry at the ends was changed by removing a quantity of material and an elasticity channel was practiced, on the opposite side of the tightening zone with screws, which allows deformations when the screws are tightened. Under these conditions, less material was consumed for the orthogonal connection, an aesthetic aspect was ensured, but also a good maneuverability at the level of the tightening screws. In conclusion, an optimization of the connection was carried out taking into account all the criteria presented above.

Funding: This paper has received no external funding.
Conflicts of Interest: There is no conflict of interest.

REFERENCES

1. Muscă, G.D., (2020). *Mașini de ridicat și transportat - Note de curs, Partea I,* Galaţi University Press, ISBN 978-606-696-191-2, ISBN 978-606-696-192-9, 127 pages, Galați, pp. 80-87, pp. 94-100.
2. Deutsch, I., (1980). *Rezistenţa materialelor*, EDP Publishing House, Bucharest, pp. 306-316, pp. 466-474.
3. Dumitrache, P., (2006). *Controlul parametrilor definitorii ai reţelelor de discretizare cu elemente finite, având dimensiunile variabile în progresie aritmetică*, Buletinul celui de-al XII - lea Simpozion Naţional de Utilaje de Construcţii, SINUC, ISSN-1842-6352, Bucharest.
4. Muscă (Anghelache), D.G., Năstac, S.M., (2021). *Parametric modelling and finite elements study for the boom of a truck crane*, IOP Conference Series: Materials Science and Engineering, 1182, Article Number: 012051, doi:10.1088/1757-899X/1182/1/012050.
5. Vlase, S., Negrean, I., Marin, M., Năstac, S., (2020). *Kane's method-based simulation and modeling robots with elastic elements, using finite element method*, Mathematics 8 (5), 805, https://doi.org/10.3390/math8050805,
6. Muscă (Anghelache), G.D., (2018). *Static and dynamic stability for floating cranes*, IOP Conference Series - Materials Science and Engineering, 400, Article Number: 082015, DOI: 10.1088/1757-899X/400/8/082015, Published: 2018, WOS:000461147400195,
7. Niculescu, M., Irimia, C., Roșca, I.C., Grovu, M., Guiman, M.V., (2017). *Structural dynamic applications using principal component analysis method*, Proceedings of CONAT 2016 International Congress of Automotive and Transport Engineering, 90-99, Springer International Publishing.
8. Grovu, M., Irimia, C., Roșca, I.C., Husar, C., Guiman, M.V., (2018). *Simulation Environment for Fatigue Analysis of Car Wheel Rim under Radial Fatigue Test*, Proceedings of CONAT 2016 International Congress of Automotive and Transport Engineering, RoJAE, 24(4), available at: https://www.researchgate.net/publication/350657785_Radial_Fatigue_Analysis_of_Automotive_Wheel_RimISO_3006
9. *** Product catalog of *Schmiedag Mannesmann*, Germany, available at: https://www.gmh-gruppe.de/standorte/schmiedag-gmbh/
10. *** Product catalog *ASL Scaffolding Tech*, United Kingdom, available at: https://www.attleboroughscaffoldhire.co.uk/?gad_source=1&gclid=CjwKCAiAyJS7BhBiEiwAyS9uNaWW_M2QP1vPGsnfQzdYL2DSi233TXWAkzqMshSQ_7k9ROVfHzs4XxoCXAMQAvD_BwE
11. *** Product catalog *Innocenti*, Italy, available at: "Innocenti". Archivio Storico (in Italian). Milano: Fondazione Fiera. Archived from the original on 2024-05-15.

Received: February 21th, 2024 / Accepted: December 15th, 2024 / Paper available online: December 20th, 2024 © International Journal of Modern Manufacturing Technologies.