THE ANNALS OF “DUNĂREA DE JOS” UNIVERSITY OF GALAŢI
FASCICLE V, TECHNOLOGIES IN MACHINE BUILDING,
ISSN 1221- 4566, 2018
https://doi.org/10.35219/tmb.2018.1.02

APPLICATION OF TOPOLOGY OPTIMIZATION ON A STIFFENING STRUCTURE OF A TANK FOR WATER STORAGE USING FINITE ELEMENT METHOD

Ionel GAVRILESCU, Doina BOAZU

Department of Mechanical Engineering, “Dunărea de Jos” University of Galaţi,
ionel.gavrilescu@ugal.ro

ABSTRACT

The purpose of the present work is to use topology optimization to propose a design for a stiffening structure of a tank for water storage. The structure under topology optimizations is a stiffening structure of a tank made of steel sheet and having the dimensions: 12m length, 2.5m width and 3m height.Some topology optimization on 3D and 2D models of the tank and its stiffening structure were performed for the purpose of qualitatively determining the mass distribution in transverse and longitudinal sections.The conclusions of these analyzes were used for size optimization calculations and for the establishment of a proposed structure consisting of plates for tank and beams for the stiffening and bearing structure. The allowable stress in the size optimization of the structure made of plates and beams was of 100MPa. In all these calculations no buckling was taken into account.
These type of calculations can be used for determination of stiffening structure when casting concrete into a metallic coffer.

KEYWORDS: topology optimization, finite element method

1.INTRODUCTION

Topology optimization combines the physical aspects of a problem with finite element method to propose an optimal shape for a blank structure having specific boundary conditions [1].

This shape optimization method allows introduction of predefined holes and cavities in the blank structure.

The results of topology optimization are quantified in great saving in weight or in improvement of structural stiffness.

A great advantage of this method is that the optimized complex piece can be related with its technological process.

Topology optimization is an easy-to-use module in the ANSYS WORKBENCH program starting with version 18.

Prior to launching topology optimization, a static structural analysis is performed.

2.3D TOPOLOGY OPTIMIZATION

Because of double symmetry of the structure, only a quarter of a structure has taking into account by imposing the two symmetry plane.

The structure was considered simply supported and can be extended outward by a distance of 50cm (see Fig. 1). So the 3D model of the blank structure is the tank having the thick of the wall of 50cm. The load is hydrostatic pressure applied on the inner faces of the tank (see the Fig. 2). In the mesh, the length of side of solid element was of 10cm. The topology optimization is applied after a static structural calculus was performed.

The purpose of 3D topology optimization is to minimize compliance with 20% weight retention.

The topology density as a result is represented in the Fig. 3.

As a conclusion of this 3D calculation, the walls of the tank can be modeled with plates of the same thickness and the stiffening structure can be modeled of beams.

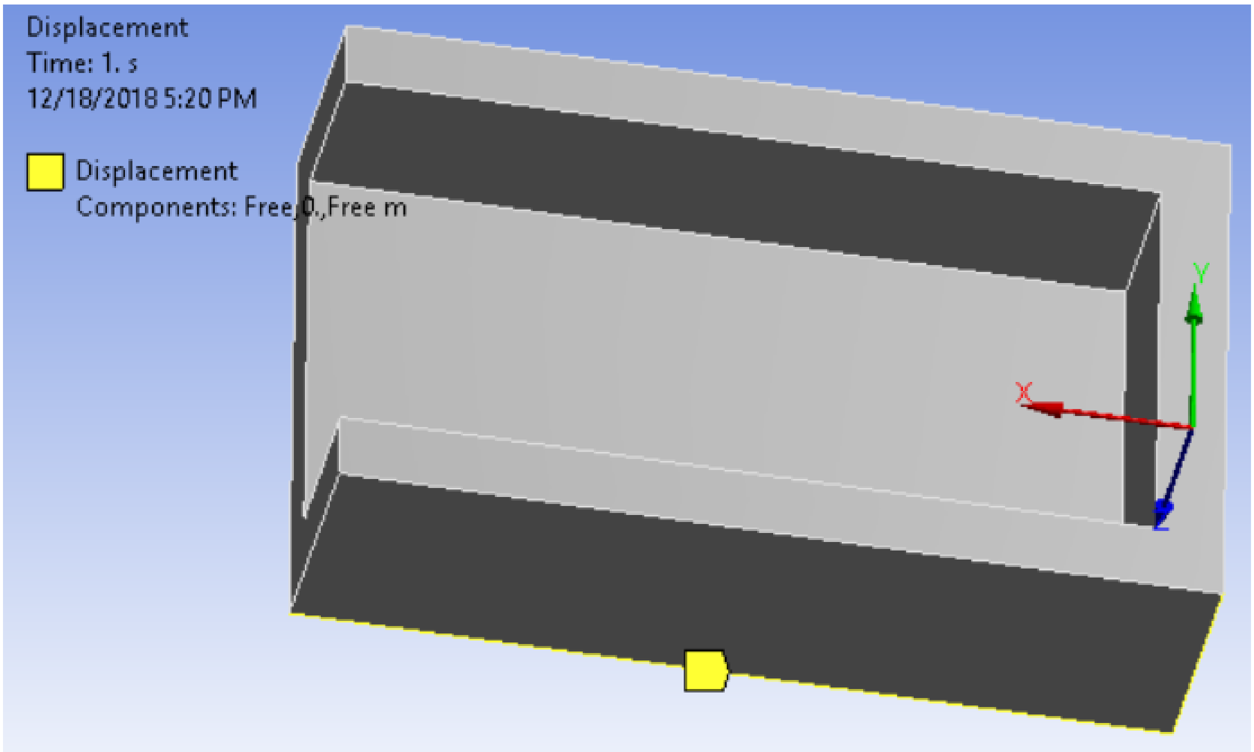

Fig. 1.*Boundary conditions – simply supported of the blank structure*

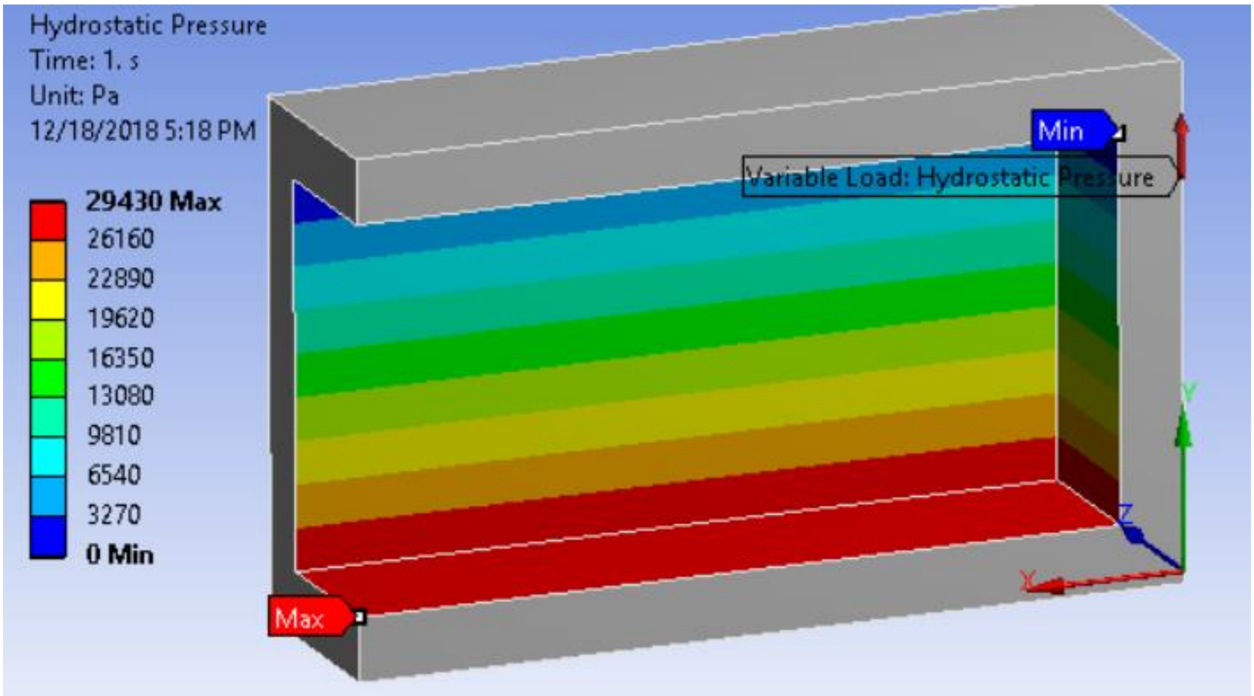

Fig. 2.*Hydrostatic pressure as load*

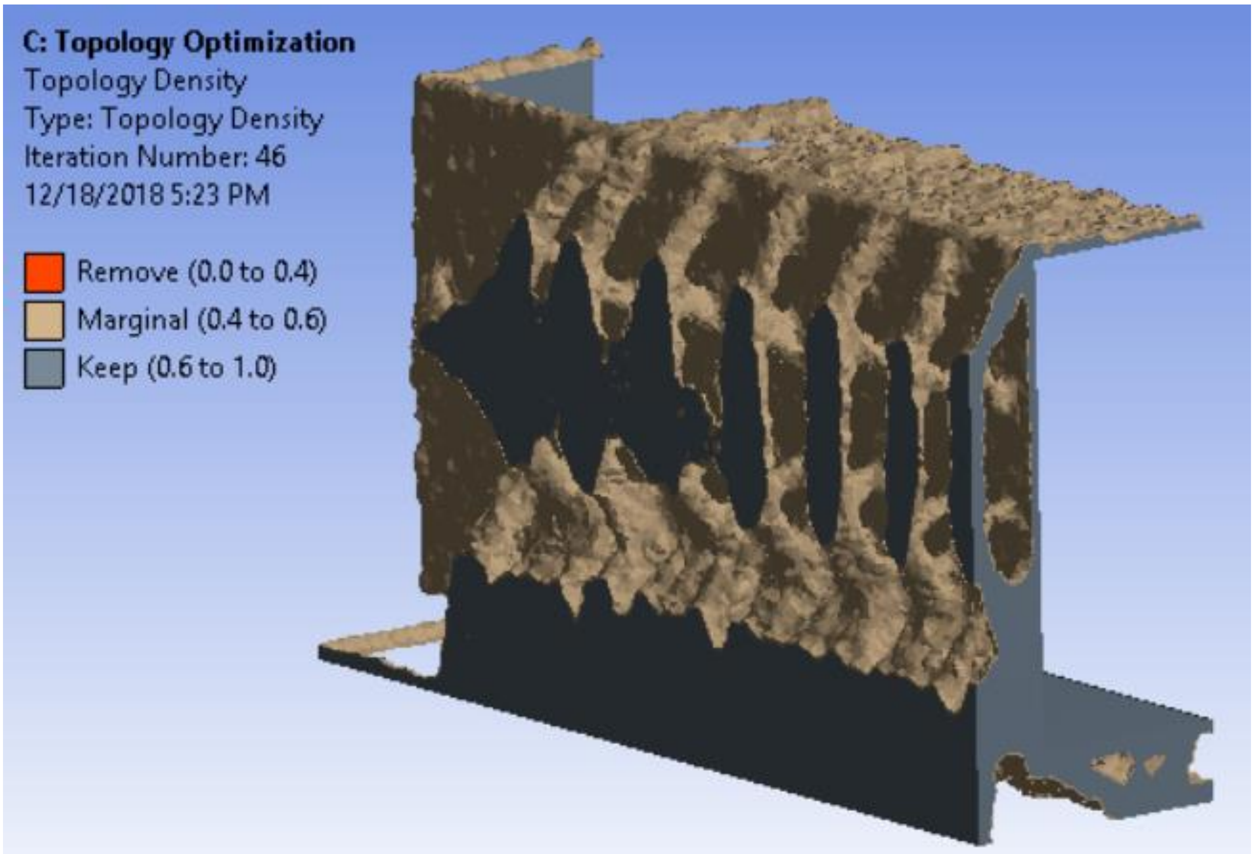

Fig. 3.*The topology density of the structure*

3.1 TOPOLOGY OPTIMIZATION OF A CROSS SECTION OF THE TANK

The 2D behavior of the model is considered plane strain and the element size is of 1cm.

The supportstructure of the tank is considered extended outward by a distance of 50cm from the walls of the tank (Fig. 4). The model has 98523 nodes and 32376 quadrilateral elements.

The load is hydrostatic pressure applied on the inner faces of the tank (see the Fig. 5).

The stiffening structure is considered simply supported (see Fig. 6).

The purpose of 2D topology optimization is to minimize compliance with 20% weight retention.

The topology density as a result is represented in the Fig. 7 (the element size is of 1cm).

If the element size is of 5cm the topology density is modified as it is represented in the Fig. 8.The meshing with the element size of 5cm provides a clearer picture of the stiffening structure (as beams position and their dimensions).

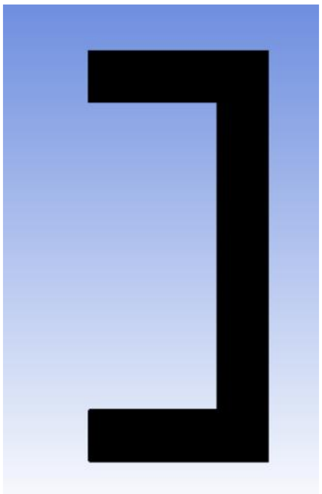

Fig. 4.*2D model – a cross section of the support blank structure of the tank*

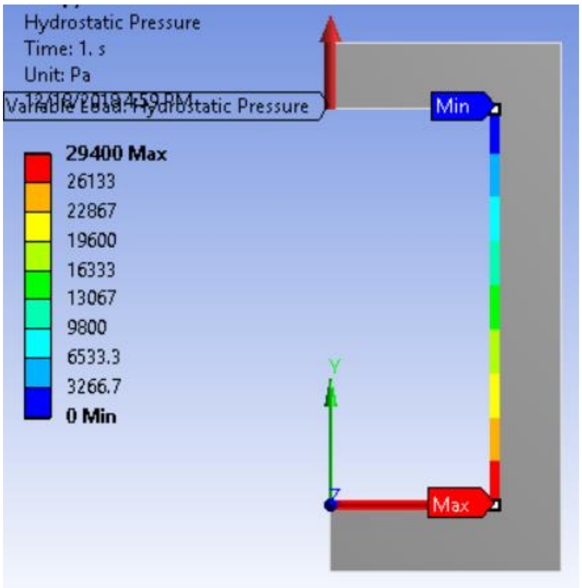

Fig. 5.*Hydrostatic pressure applied on the inner sides of the tank*

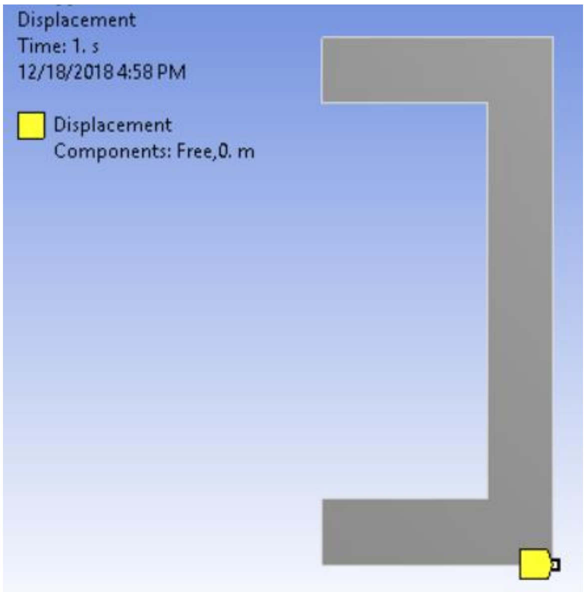

Fig. 6.*Simply supported of the structure of 2D model*

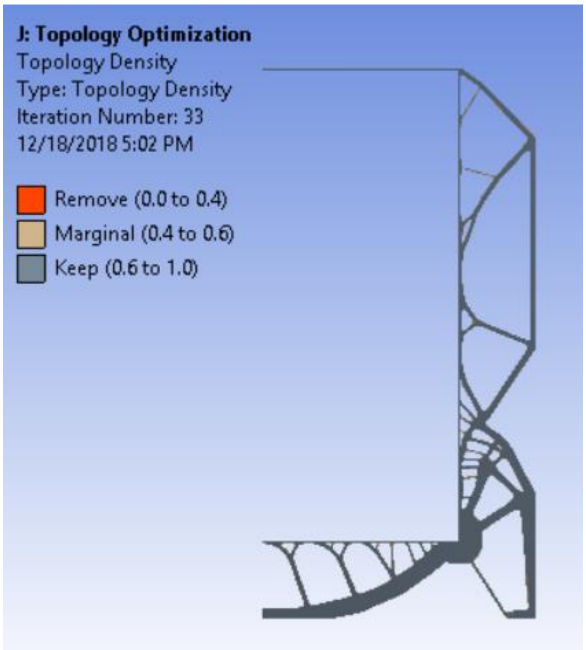

Fig. 7.*The 2D topology density of a cross section of stiffening structure(the element size is of 1cm)*

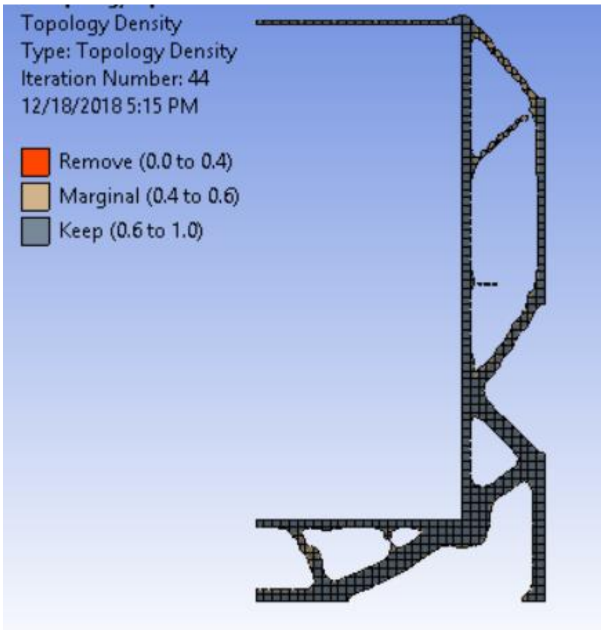

Fig. 8.*The 2D topology density of a cross section of stiffening structure (the element size is of 5cm)*

3.2 TOPOLOGY OPTIMIZATION OF A LONGITUDINAL SECTION OF THE STIFFENING STRUCTURE

A longitudinal section at a third of the height of the tank is subjectedto a topology optimization calculus by minimizing compliance with 20% weight retention.

The model has 4981 nodes and 1550 elements (the size of quadrilateral element is of 5cm).

The load is a constant pressure of 16300Pa (see Fig. 9).

The purpose of 2D topology optimization is to minimize compliance with 20% weight retention, and the topology density is represented in the Fig. 10.

The results of Fig. 10 determine the position and dimensions of the beams in this very important longitudinal section.

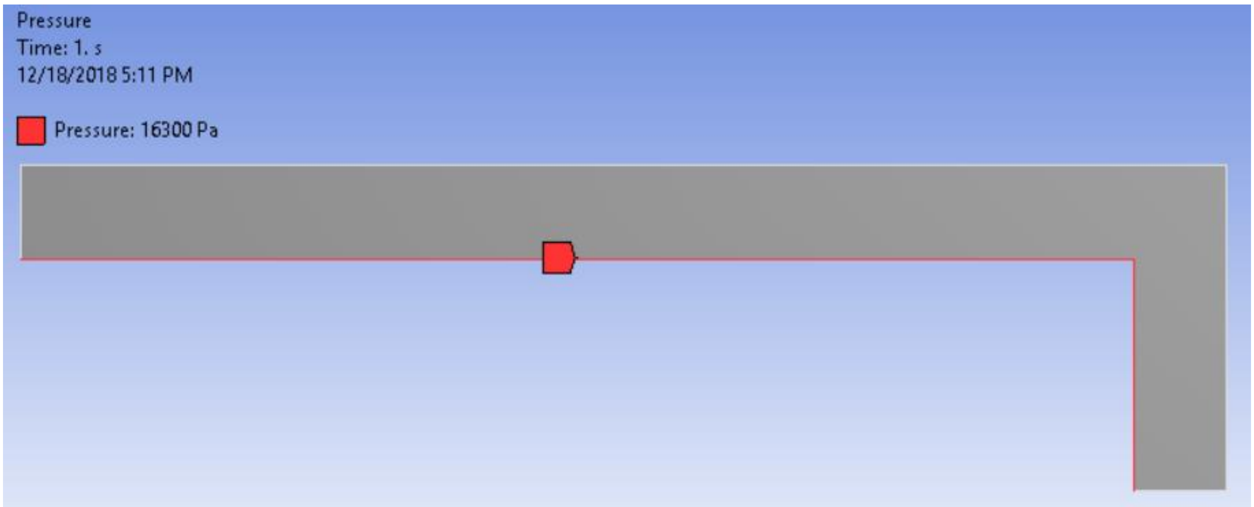

Fig. 9.*Model 2D – a longitudinal section of the stiffening structure of the tank*

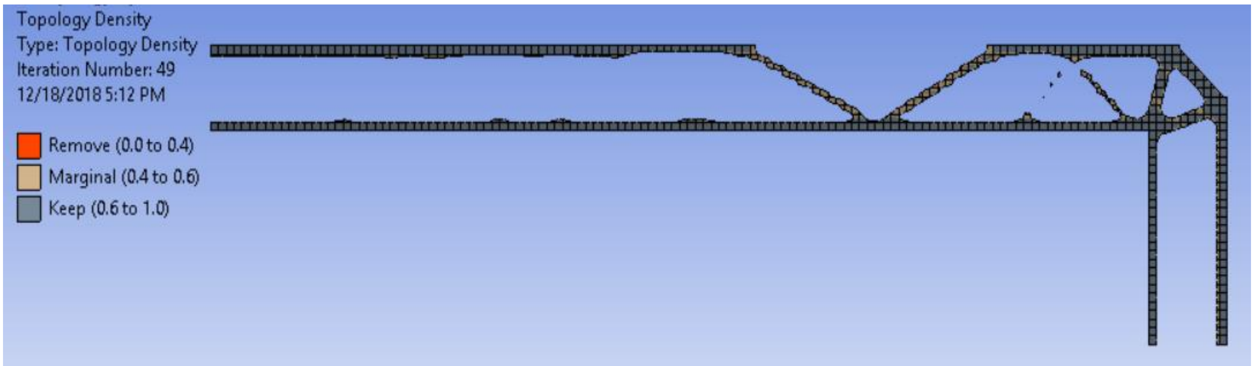

Fig. 10.*Topology density for a longitudinal section of the stiffening structure of the tank (the size of the quadrilateral element is 5cm)*

4.3D SIZING OPTIMIZATION OF THE STIFFENING STRUCTURE OF THE TANK (MODELING WITH PLATES AND BEAMS)

The starting pointin 3D sizing optimization analysis was the 2D topology optimization results with the element size of 5cm; the dimensions of the beam elements have been estimated. The tank is considered supported on transverse beam frames arranged at a step of 50 cm. It was considered that 50cm is a convenient distance because in the plates (the tank walls) the equivalent stresses are not greater than 0.78Pa. The thickness of the tank walls is considered to be of 6mm. It has been observed from the calculation that the stiffness of the beam frame has a low influence on plate stresses. Stressesin the plates are mainly determined by the length of the gap between two frames (50cm) and the inner hydrostatic pressure in the tank.

The geometry of the model is represented in the Fig. 11. The tank was modeled with plate elements and the stiffening structure of the tank was modeled with beam elements. The mesh of the model has the element size of 7cm (see Fig. 12). The model has 621 nodes and 429 elements.

The boundary conditions applied on the model are represented in Fig. 13 (hydrostatic pressure) and in Fig. 14 (supports).

The distribution of maximum combined stress from coupling the bending stress with axial stress is shown in the Fig. 15.

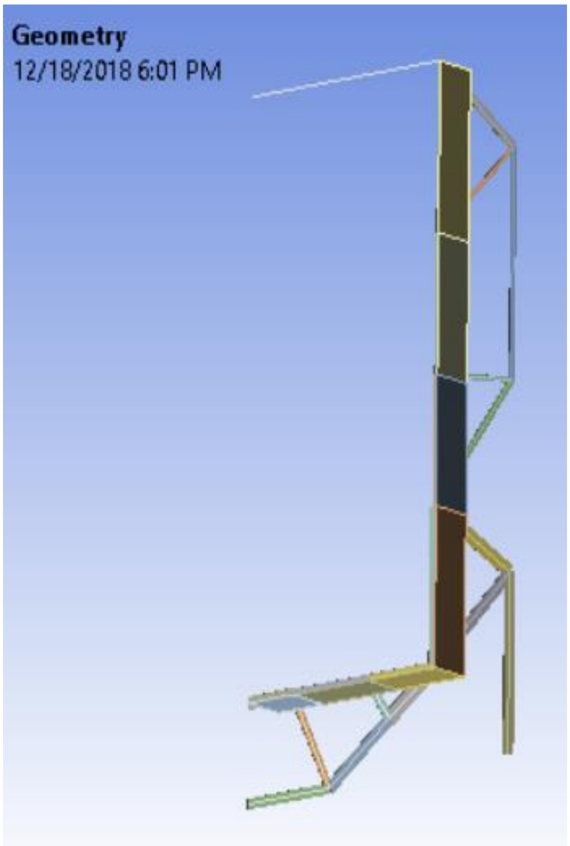

Fig. 11. *3D model of a longitudinal section of tank*

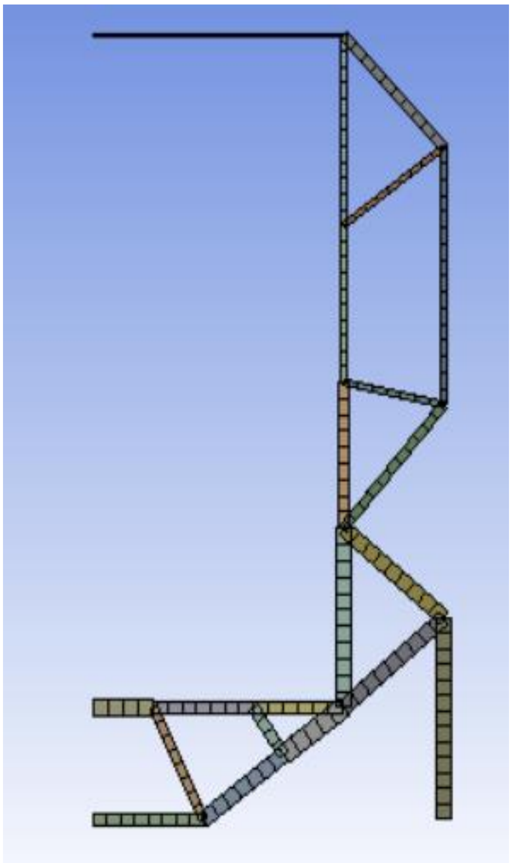

Fig. 12. *The mesh of model*

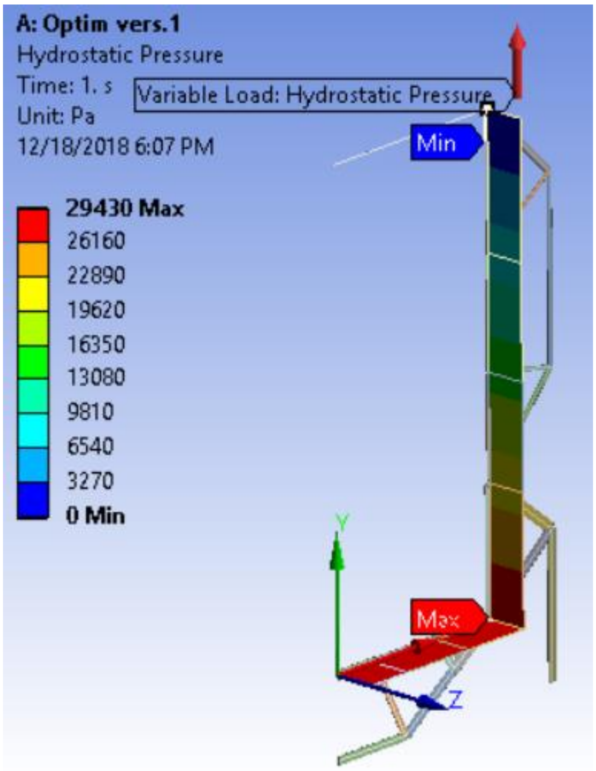

Fig. 13. *Hydrostaticpressureapplied on theinnerfaces of theplates*

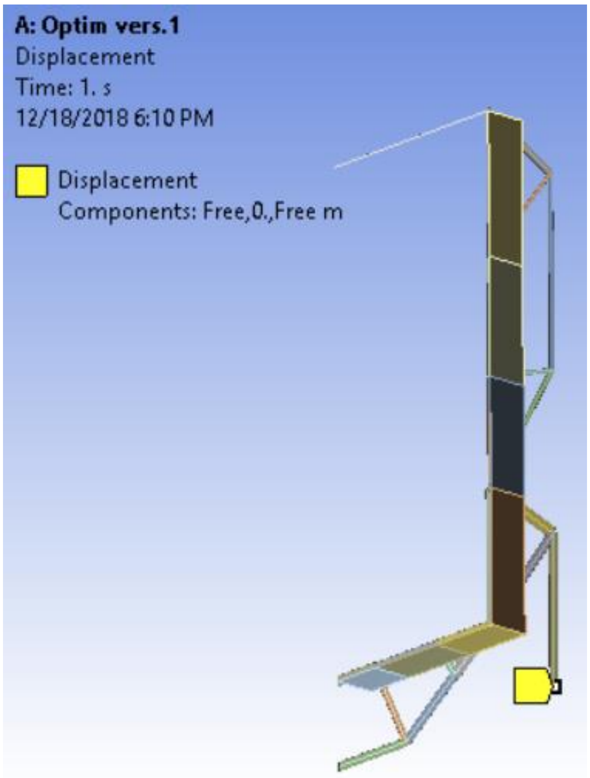

Fig. 14. *The support of the model*

In the Fig. 15, beams are shown after optimization. In the optimization process it was considered that the heights of the sections were variable and were determined so that the total mass of the structure was minimal without exceeding the admissible stress of 1e8Pa. To achieve the above results, the NLPQL optimization algorithm was applied [2], [3]. It is an optimization algorithm based on the use of gradients (first-order derivatives) of the functions that are part of the optimization process. It is one of the best performing optimization algorithms in this category. Unfortunately, in calculating the finite element, gradients cannot be calculated and as a result they are estimated by differences.

In the case of *n* optimization variables, the gradient calculation assumes *n* + 1 or 2*n* + 1 gradients, according to the method. In this issue there are 21 optimization variables, and as a result for the gradient calculation, at least 22 finite element analyzes are required. In solving any optimization problem a lot of gradient calculations is necessary and the computational effort may be large. When solving the above problem, more than 600 finite element analyzes were performed without achieving (default) convergence. By simply visual verification of the stress distribution in the Figure 15 it can be noticed that the solution is not optimal. Apart from the large computational effort, there is a problem of principle.The functions for which the gradient is calculated in this problem are not derivable and consequently the theoretical gradient is not defined. For example, the maximum stress is calculated on a section at a point. The point is not the same as changing the parameter values because the position of the maximum stress can change. In addition, the section on which the maximum stress appears on the structure is not the same when changing the parameters. Despite these negative aspects, the mass of the structure was significantly reduced by optimizing from 93kg to 76kg. The attempt to continue using the same method (NLPQL) to make a new optimization starting from the previous solution failed. Probably the solution obtained is, at least numerically, a local optimal point. It is tried to get away from this solution using the genetic algorithm.

After approximately 300 finite element analyzes, no progress has been observed and the application of this method has been dropped.

An algorithm based on the beam of equal strength has been tried. This algorithm proved successful. Starting from the 76kg in previous solution, we reached a 65kg of the structure after only 9 finite element analyzes. According to this simple engineering algorithm, simple calculations are made at each beam to modify the section iteratively. Briefly the algorithm assumes:

- data at one point are: ***h1*** section height,
  s1 maximum combined stress on the beam;
- should determine ***h2*** the correct section height that leads to its allowable stress on the beam.
- it is assumed, approximately, that the moment on the beam does not change when the section height changes;
- in the above assumptions ***s1*** would be calculated with ***s1 = c / h1 ^ 2*** where ***c*** is a constant; similar
  sa = c / h2 ^ 2
- eliminatethe constant and get ***h2 = h1 * sqrt (s1 / sa)***
- to avoid the oscillations of the iterative values, relax the size ***h2*** as follows: note ***dh = h2-h1*** and calculate
  h2r = h1 + omega * dh; we used ***omega = 0.8*** and did not modify it because the algorithm is very fast converging.

The solution obtained using the proposed algorithm is shown in the Fig. 16.

It is noticeable that the beams are very thin and as a result there is a risk of buckling. In order to reduce the length of the beams, it was considered that the bearing structure could extend beyond the tank by no more than 30cm.The optimization was done with the same proposed algorithm.

After 12 iterations and the same number of finite element analyzes, the 68kg solution was obtained for structure mass.

The solution obtained using the proposed algorithm as the maximum combined stress is shown in the Fig. 17.

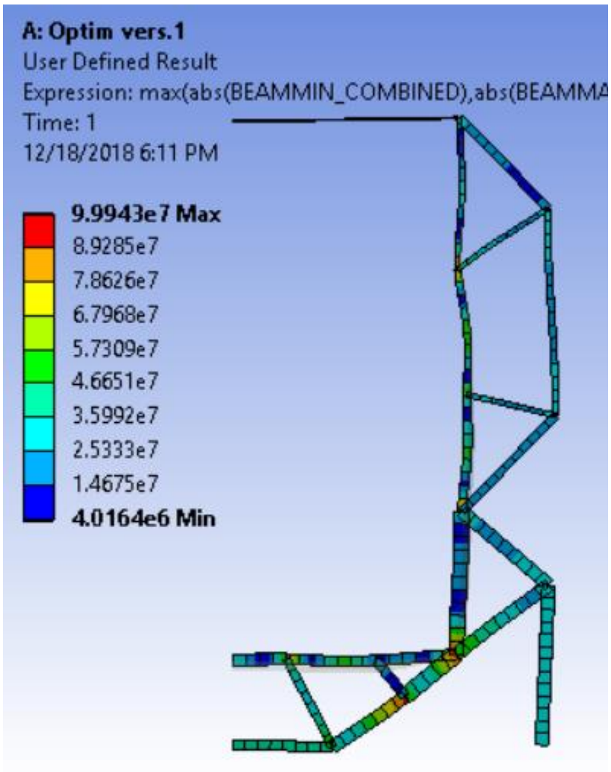

Fig. 15.*The maximum combinedstress in beams*

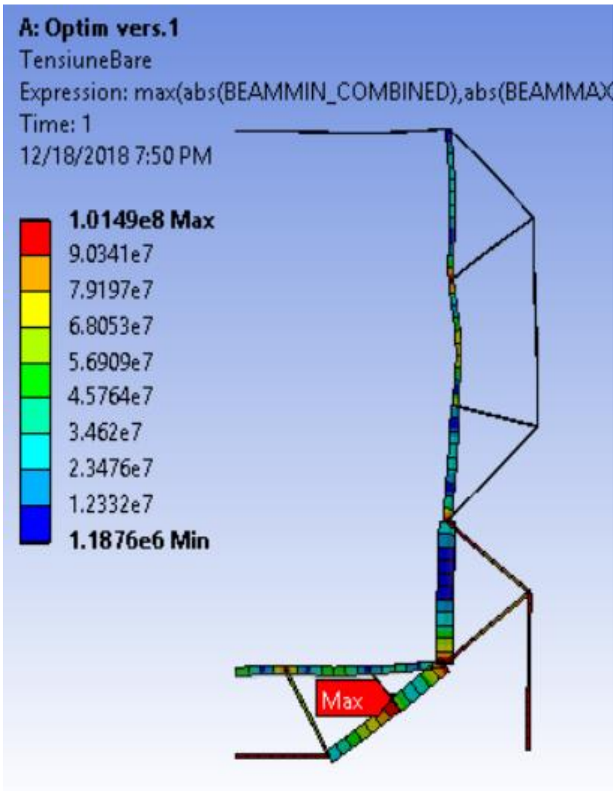

Fig. 16.*The maximum combined stress distribution with the proposed algorithm (the stiffness structure extended 50cm)*

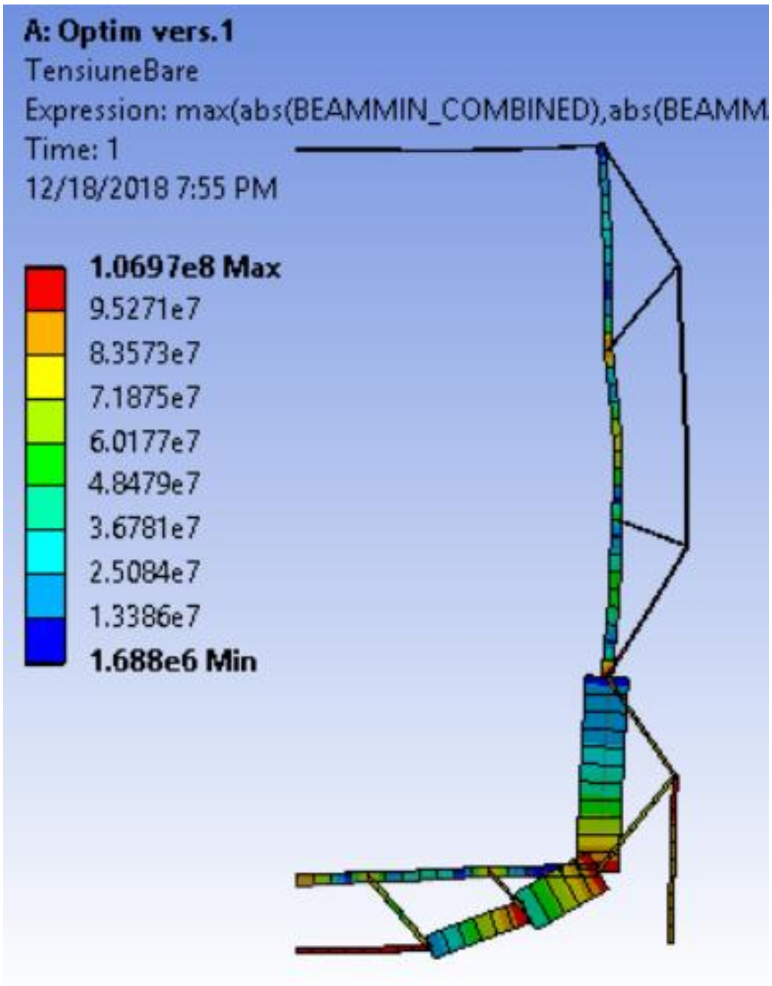

Fig. 17.*The maximum combined stress distribution with the proposed algorithm (the bearing structure extended 30cm)*

In relation to the masses of the structure presented above, we consider that 50kg represents the mass of the tables and the remaining mass is half the weight of the beams. To take into account the symmetry conditions, beams have only half the section. For example, the mass of the 68kg structure corresponds to (68-50) * 2 = 36kg the mass of a transverse frame.

5. VERIFICATION CALCULUS OF THE PROPOSED MODEL OF THE TANK AND ITS STIFFENING STRUCTURE IN 3D MODELING USING PLATES AND BEAMS

The tank has the dimensions 12mx2.5mx3m, where 3m is the height.

The structure was calculated on the quarter and the necessary symmetry conditions were imposed.

Frame beams, transverse and longitudinal, had the initial dimensions resulting from the optimization applied to the structure with a single transverse frame. Initial stresses in this calculation were about 15% higher than in the single frame calculation. The algorithm described in the singleframe calculation was applied and after 4 iterations (equal with the number of finite element analyzes) the optimized structure shown in the Fig. 21was obtained.

The distance between the beam frames was 50cm, sufficient for the equivalent stresses in the plates to not exceed 0.7e8 Pa.

The beam elements have a rectangular cross section with a width of 1cm and the height resulting from the optimization ranging from 0.5 to 18 cm. All frames, transversal and longitudinal, are identical.

The model with plates and beams has 11330 nodes and 9523 elements. The mesh on beam structure is represented in Fig. 17 and the mesh on plates is represented in Fig. 18.

The applied load is the hydrostatic pressure on the tank walls as in the Fig. 19 and in Fig. 20 the supports of the stiffening structure are represented.

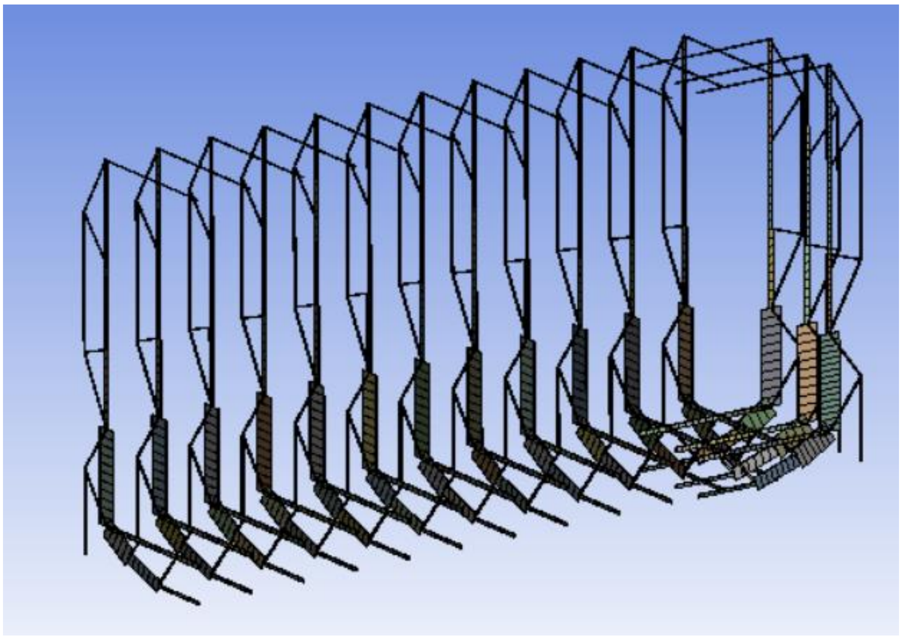

Fig. 17.*The mesh on thestiffness beam structure of the tank-themeshsizeis of 7cm*

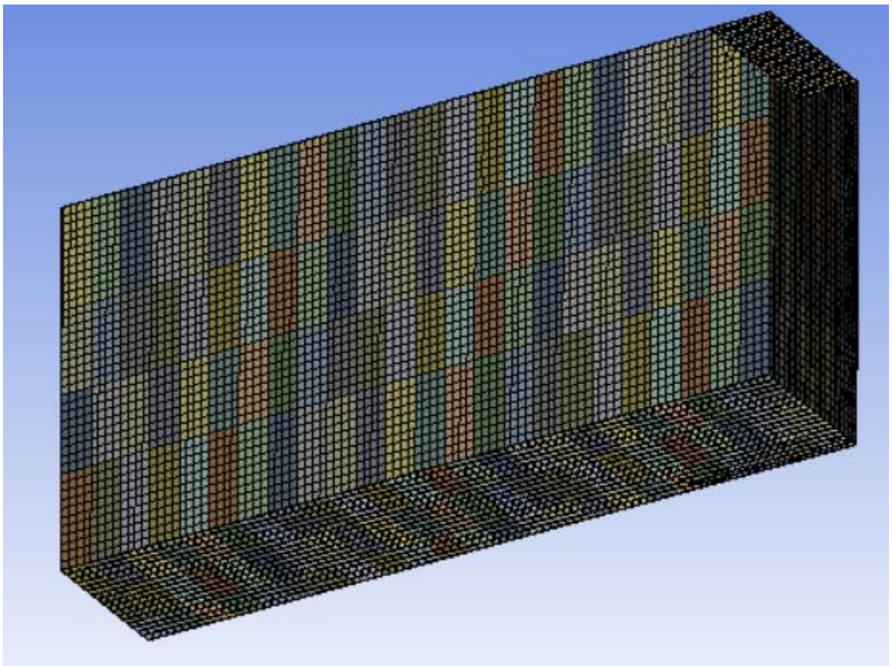

Fig. 18.*The mesh on the plate walls of the tank (the thickness of platesis of 6mm)-* *themeshsizeis of 7cm*

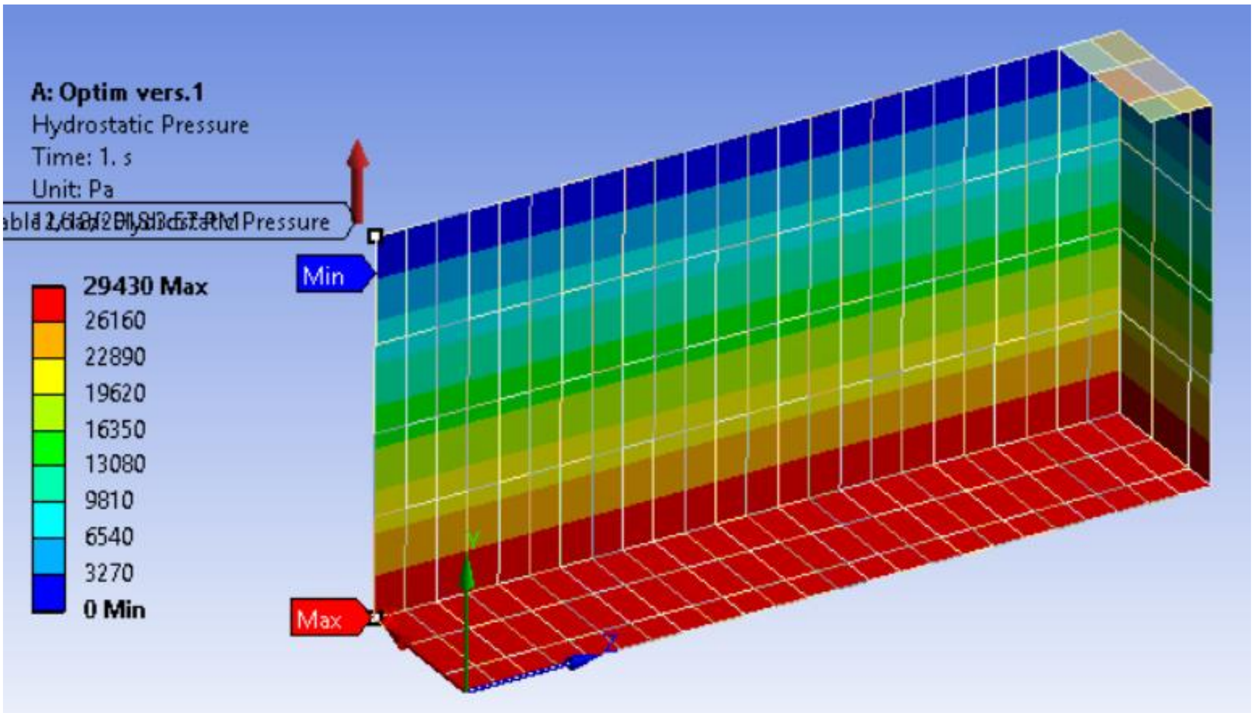

Fig. 19.*Hydrostatic pressure applied on the inner faces of the tank*

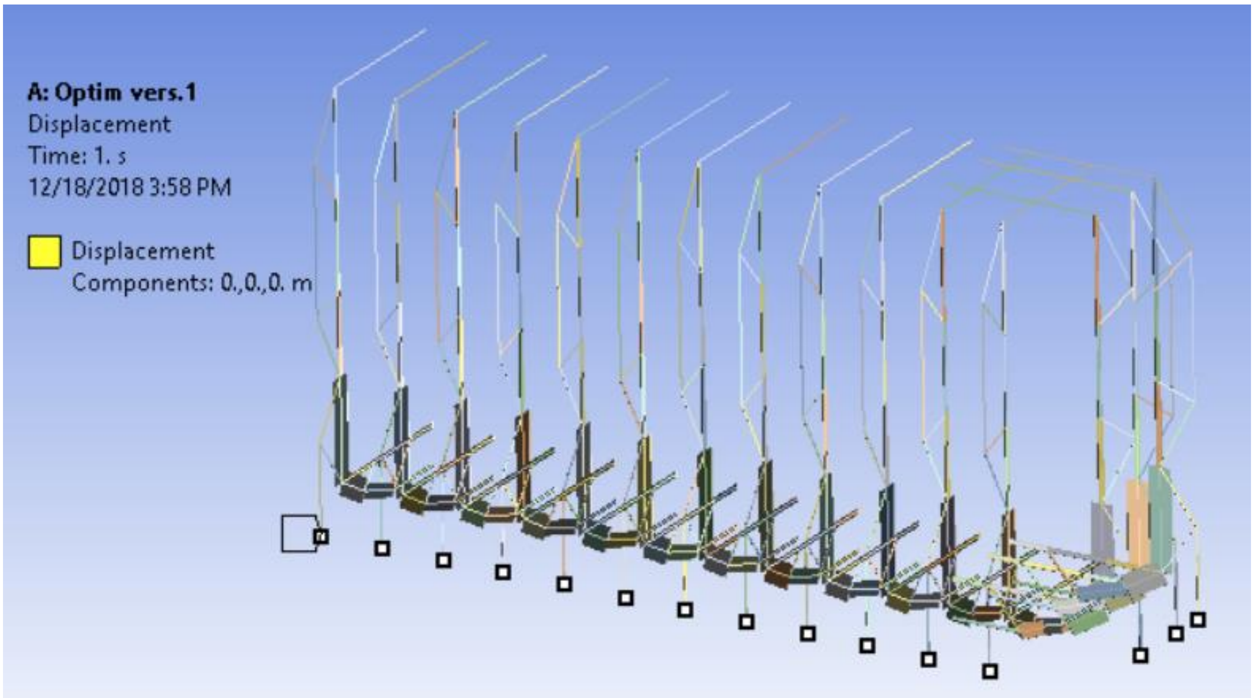

Fig. 20.*The supports of stiffness structure*

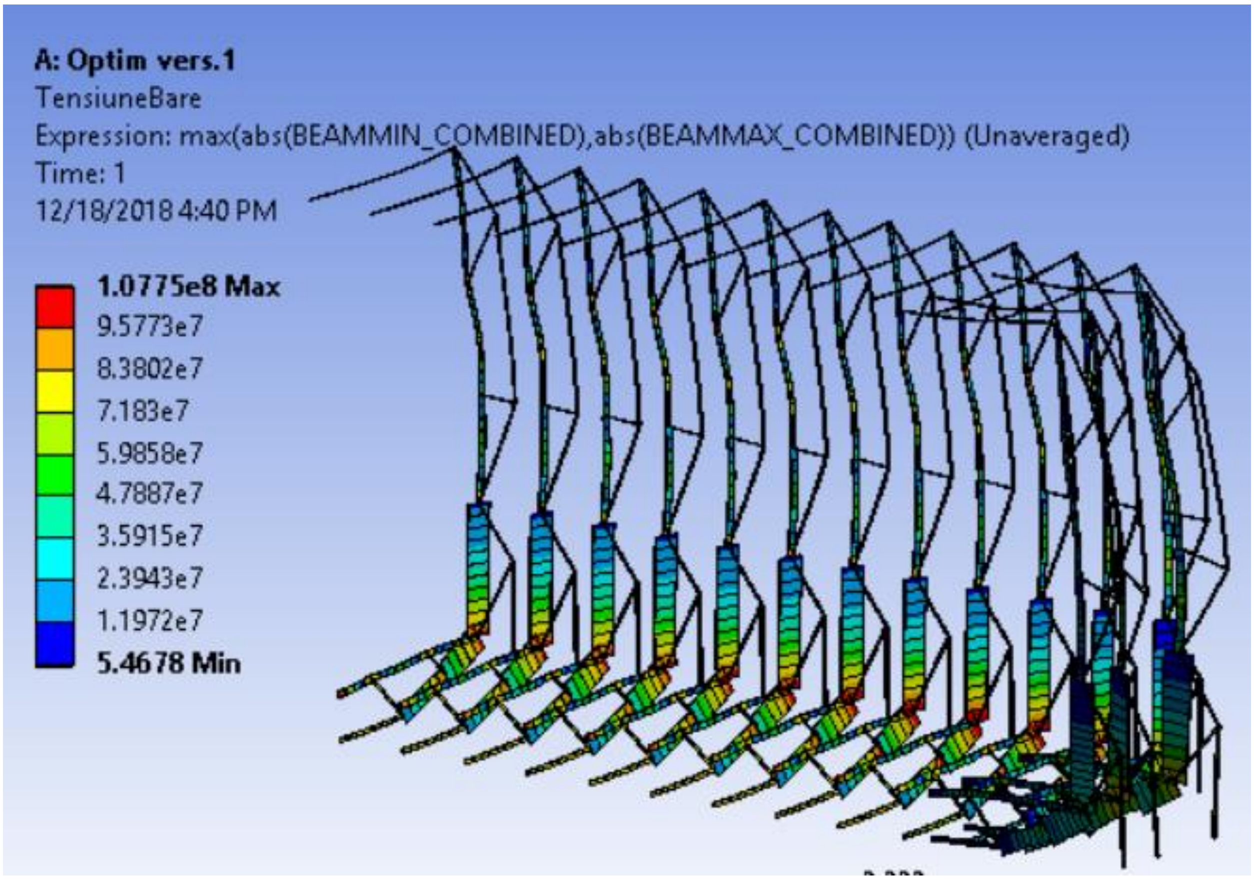

Fig. 21.*The maximum combined stress distribution in the beam elements of the stiffening structure of the tank (an amplification scale factor of 46 times is used to represent shape deformationof the stiffening structure of the tank)*

5. CONCLUSIONS

The beam frames are optimized so that each maximum combined stress is close to the 100MPa the maximum allowable stress. The danger of buckling was also not taken into consideration because in this preliminary project it was only wanted to determine how the massis distributed. In the case of a final project, another type of section will be used, e.g. the square pipe and the potential danger of loss of stability will be avoided.

REFERENCES

[1] ***, ANSYS WORKBENCH documentation;

[2] **Frédéric, J., Bonnans, J.C.G., Lemaréchal, C., Sagastizábal, C.A.**, *Numerical Optimization: Theoretical and Practical Aspects*, Springer, 2006;

[3] **Kirsch, U.**, *Structural optimization: fundamentals and applications*, Springer-Verlag, 1993.