

# NUMERICAL SIMULATION OF THE STRESS AND DEFORMATION ZONE ON THE CABIN'S FRAMEWORK OF A MOBILE EQUIPMENT

*D. Nedelcu, T. Mănescu*  
*"Eftimie Murgu" University Reșița*

## INTRODUCTION

The machines with human operator are important equipments for technological process of the material manipulation and processing. One of the important function of these equipments is the safety function of the machine-human system as a response to the events which can damage the resistance of the structure. The paper present a study with finite elements on the cabin's framework of a mobile equipment: numerical simulation of the stress and deformation zone obtained for external loads applied on the structure.

## 1. NUMERICAL SIMULATION TOOLS

Experimental study of resistance structures involve high costs, due to:

- high costs of the experimental test stands and of the recording apparatus data;
- supplementary costs for design and execution of the resistance structures if are not accorded with standards;
- supplementary costs for personal involved in experimental process.

Numerical simulation on the computer offer some advantages comparative with experimental methods, like:

- eliminate the costs corresponding to the physical execution of the components, of the equipments and stands;
- the possibility to repeat the numerical simulation for various conditions:
  - obtain important conclusions useful for resistance structures design;
  - save huge information volumes which can be used thereafter;
  - easy modifications of the virtual components with lower costs comparative with the real components.

For numerical simulation of resistance structures, the geometry must be drawn in 2D or 3D space with a CAD (computer-aided design) software, followed by finite element method

analyse (FEM). For the present paper Autodesk Inventor software was used for drawn of the 3D geometry and the Cosmos Design Star software for FEM analyse.

### 1.1. Autodesk Inventor software

Autodesk Inventor is a software for 3D modelling of the parts and assembly [1]. The design of a components is made in 3D space and the result is the computer equivalent of the real components.

Multiple components can be assembled together to obtain a assembly, which can be integrated in another assembly, and so it can be generate complex assembly with many components.

The software can generate very quickly paper drawing with views, projections, breaks, details, dimensions, hole tables, parts components; the software maintain a bidirectional correspondence between 3D model and paper drawing, so a modification in 3D model will generate modification in paper drawing.

In the old fashion of components design process, the result is a paper drawing with views and sections and the reader of the drawings must compose in his mind the real shape of the components. With Inventor the process is inversed: the result is the 3D model and also the paper drawing, so the reader can see on the computer screen a virtual copy of the real component.

### 1.1. Cosmos Design Star software

COSMOS Design STAR [2], [3], [4] can perform the following types of analyses:

- linear stress analysis
- natural frequencies and mode shape calculations
- critical buckling load estimation
- linear and nonlinear steady state and transient heat transfer analysis
- nonlinear stress analysis.

After building the model in a CAD system and generating the proper file, the model will be imported into Design Star, where

studies will be created, solve them, and visualize the results.

All geometry construction and editing must be done in CAD system. It is not possible to modify the geometry in Design Star, except the possibility to suppress some part feature, which will not be considered at the moment of running analysis. But it is possible to maintain a direct link between CAD software and Design Star, so if the geometry is modified in CAD software, loads and restraints will be applied to the new geometry in Design Star software. This option is valid only for Solid Works, Solid Edge și Autodesk Inventor.

## 2. THE GEOMETRY

For numerical simulation were considered the following versions for the geometry, created with Autodesk Inventor fig. 1 și fig. 2:

- Cabin 1 – Cylindrical tube  $\Phi 80 \times 5$  mm;
- Cabin 2 – Quadratic tube  $\square 80 \times 80 \times 5$  mm.

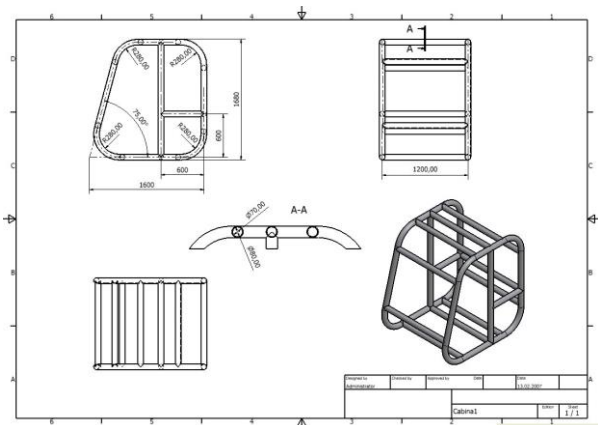


Figure 1. The geometry of the cabin no. 1

The difference between the two geometries is made by the shape (cylindrical or quadratic) of the section of cabin's framework.

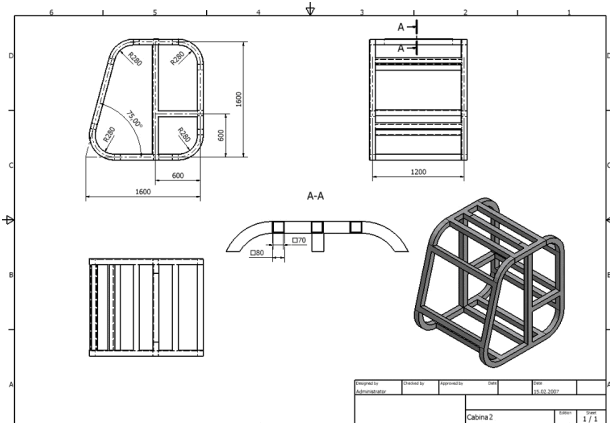


Figure 2. The geometry of the cabin no. 2

## 3. BOUNDARY CONDITIONS

When loads are applied to a body, the body deforms and the effect of loads is transmitted throughout the body. The external loads induce internal forces and reactions to render the body into a state of equilibrium.

Linear static analysis calculates displacements, strains, stresses, and reaction forces under the effect of applied loads and is based on the following hypothesis:

- static assumption - all loads are applied slowly and gradually until they reach their full magnitudes; after reaching their full magnitudes, loads remain constant;

- linearity assumption - the relationship between loads and induced responses is linear; for example, if you double the loads, the response of the model (displacements, strains, and stresses), will also double; this assumption is valuable if all materials in the model comply with Hooke's Law (stress is directly proportional to strain), the induced displacements are small enough to ignore the change in stiffness caused by loading and boundary conditions do not vary during the application of loads (loads must be constant in magnitude, direction, and distribution).

The internal forces in a body vary from one point to the other. Across any small internal plane area, loads are exerted by the part of the body on one side of the area upon the part on the other side. Stress denotes the intensity of these internal forces (force per unit area).

For a given geometry it is possible to generate multiple design studies. Design Studies offer the possibility to perform various types of studies on the model. Each study represents a what-if scenario and is defined by:

- analysis type and options;
- material(s);
- a set of loads and restraints;
- a mesh.

The steps for linear stress analysis are:

- define a design study;
- select the material for the model;
- define applied loads and restraints;
- mesh the model;
- running analysis;
- plotting results.

The material for cabin was selected from Cosmos library with the following properties: the Modulus of Elasticity  $E=2.1 \times 10^5$  N/mm<sup>2</sup>, Poisson's Ratio 0.28, density 7850 kg/m<sup>3</sup>, yield strength  $\sigma_c = 230$  MPa and fracture strength  $\sigma_r = 380$  MPa.

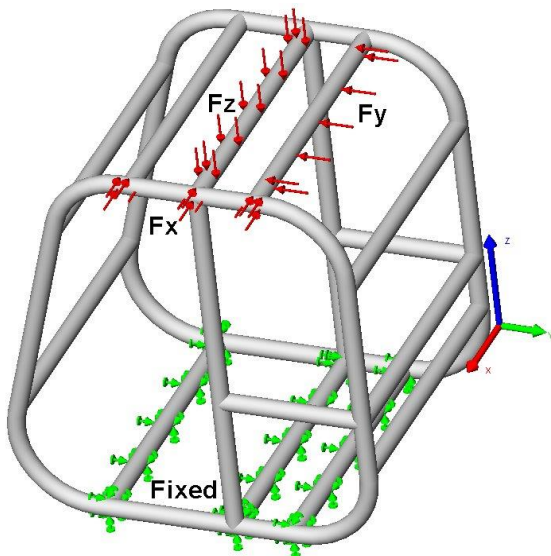
Numerical simulation was made for 4 loads, table no. 1, figure no. 3 and 4, with values calculated according standard recommendations.

**Table 1.** Loads applied on the cabin's framework.

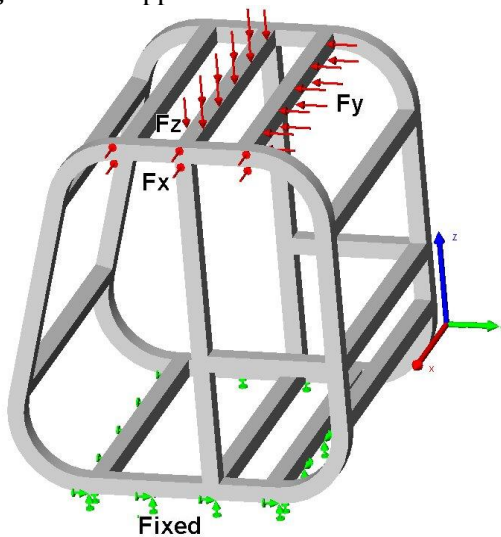
Type Load	Expression	Value [N]
Lateral	$F_x=6 * M$	14400
Back	$F_y=4,8 * M$	11520
Vertical	$F_z= 19,61 * M$	47064
Simultaneous	$F_x + F_y + F_z$	

where  $M = 2400$  Kg is the equipment mass.

The framework will be fixed on the bottom; the fixed condition consist on 0 values for X,Y and Z translation, so  $U_x=0$  ,  $U_y=0$  and  $U_z=0$  conditions will be applied, figure no. 3 and 4.



**Figure.** Load applied on cabin no. 1



**Figure 4.** Load applied on cabin no. 2

## 4. THE MESH

Finite element analysis provides a reliable numerical technique for analyzing engineering designs. The process starts with the creation of a geometric model. Then, the program subdivides the model into small pieces of simple shapes (elements) connected at common points (nodes). Finite element analysis programs look at the model as a network of discrete interconnected elements. FEA assumes that the behavior of each element varies in particular known fashions for various conditions. The finite element method predicts the behavior of the model by manipulating the information obtained from all the elements making up the model.

Meshing is a very crucial step in design analysis. The automatic mesher generates a mesh based on a global element size and local mesh control specifications. Mesh control lets you specify different sizes of elements near vertices, edges, and faces. The program estimates a global element size for the model taking into consideration its volume, surface area, and other geometric details. The size of the generated mesh (number of nodes and elements) depends on the geometry and dimensions of the model as well as the specified element order (draft or high quality), element size, mesh tolerance, mesh control, and contact specifications.

For the framework cabin it is possible to generate the following types of elements:

- Tetrahedral solid elements for meshing solid parts in a solid study.
- Triangular shell elements for meshing shell study.

Because the thickness of the framework wall is small a solid mesh will generate a huge number of finite elements, which require a large computer memory and a long running time. Shell elements are naturally suitable for modeling thin parts (sheet metals). So, for this reasons the shell mesh was chosen. For a shell study, it is possible to select one of two options:

- draft quality mesh: the automatic mesher generates linear triangular shell elements, defined by 3 corner nodes connected by 3 straight edges.
- high quality mesh: the automatic mesher generates parabolic triangular shell elements, defined by 3 corner nodes, 3 mid-side nodes, and 3 parabolic edges.

Every shell node has 6 degrees of freedom, which are the translations and rotations for the three orthogonal axes.

Every shell has one top face and one bottom face, which must be aligned, because the results are in general not identically on the two faces.



The cabin's geometry no. 1 was meshed in 11828 finite elements with 23626 nodes, figure no. 5, and the cabin's geometry no. 2 was meshed in 9498 finite elements with 18966 nodes, figure no. 6.

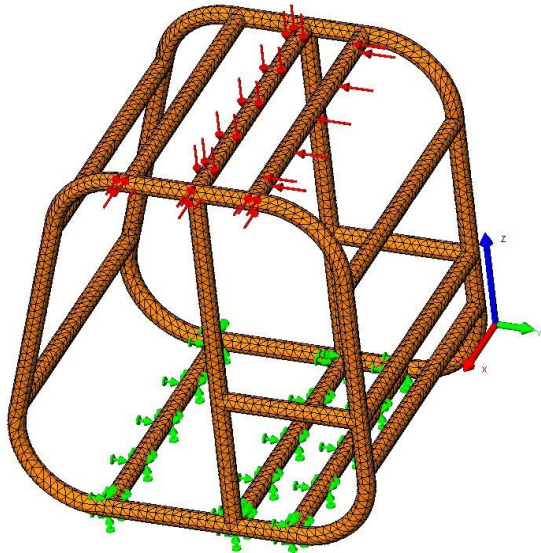


Figure 5. Mesh of the cabin no. 1

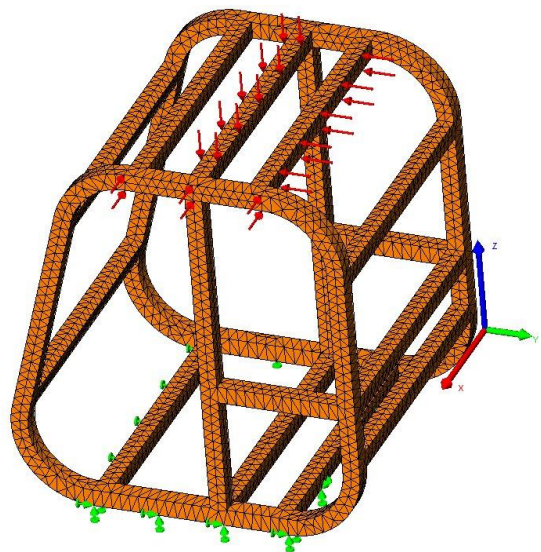


Figure 6. Mesh of the cabin no. 2

### 5. RESULTS FOR CABIN NO. 1

The study results for cabin no. 1 are presented in table no. 2, from where the following conclusions result: for lateral load  $F_z=47064$  N the VonMises stress value is greater than the yield strength  $\sigma_c$  with 14%, into the plastic domain, fig. 7, and for simultaneous loads corresponding to the three directions OX, OY, OZ, the VonMises stress value is greater than the yield strength  $\sigma_c$  with 52,74% on the bottom face of shell mesh, into the plastic domain, fig. 8.

Table 2. Results for cabin no. 1.

Cabin no. 1				
Load case	Loads N	$\sigma_{\text{VonMises}}$		$\Delta$ mm
		Top MPa	Bottom MPa	
Lateral	$F_x=14400$	209.0	189.0	6.51
Back	$F_y=11520$	136.6	151.6	5.36
Vertical	$F_z=47064$	236.1	262.2	4.06
All	$F_x+F_y+F_z$	324.5	351.3	9.06

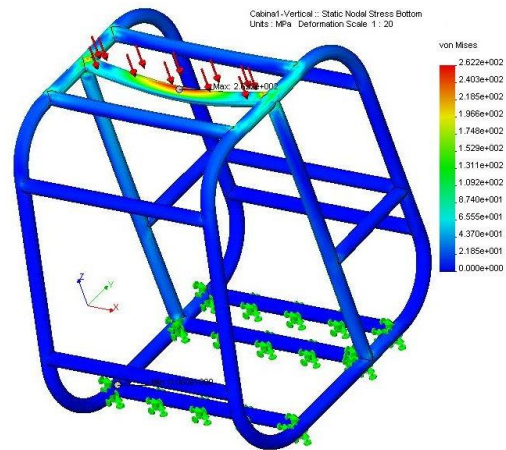


Figure 7. Cabin no. 1 ; Fz ; Stress

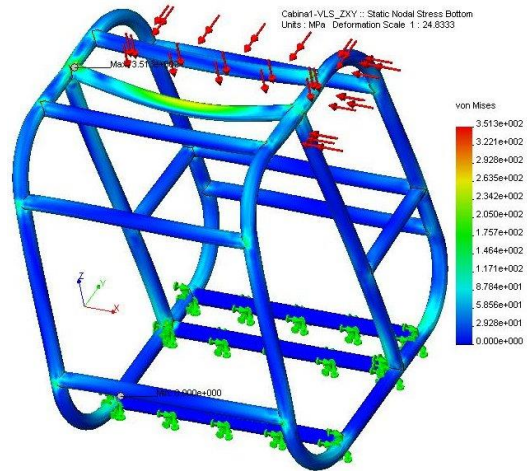


Figure 8. Cabin no. 1 ; Fx+Fy+Fz ; Stress

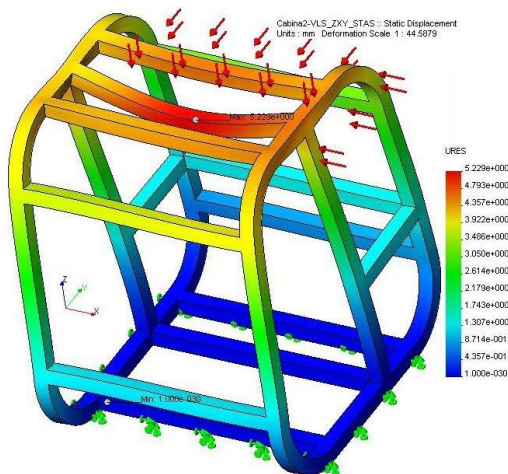
### 6. RESULTS FOR CABIN NO. 2

The study results for cabin no. 2 are presented in table no. 3, from where the following conclusions result: for all individual load cases, the VonMises stress values are smaller than the yield strength  $\sigma_c$  and displacements  $\Delta$  were reduced with 50% comparative with the same cabin no. 1 values; for simultaneous loads corresponding to the three directions OX, OY, OZ, the maximal displacement value is 5.23 mm, fig. 9, and the VonMises stress

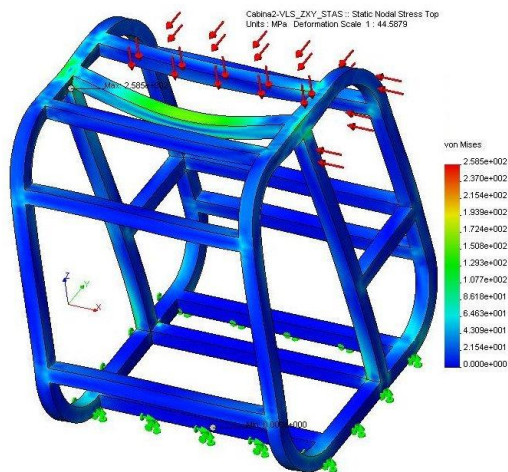
value is greater than the yield strength  $\sigma_c$  with 12,4% on the top face of shell mesh, into the plastic domain, fig. 10.

**Table 3.** Results for cabin no. 2.

Cabin no. 2				
Load case	Loads	$\sigma_{\text{VonMises}}$		$\Delta$
		Top	Bottom	
	N	MPa	MPa	mm
Lateral	$F_x=14400$	92.9	95.9	3.47
Back	$F_y=11520$	115.1	91.7	3.11
Vertical	$F_z=47064$	183.8	193.7	2.92
All	$F_x+F_y+F_z$	258.5	238.9	5.23



**Figure 9.** Cabin no.2 ;  $F_x+F_y+F_z$  ; Displacement

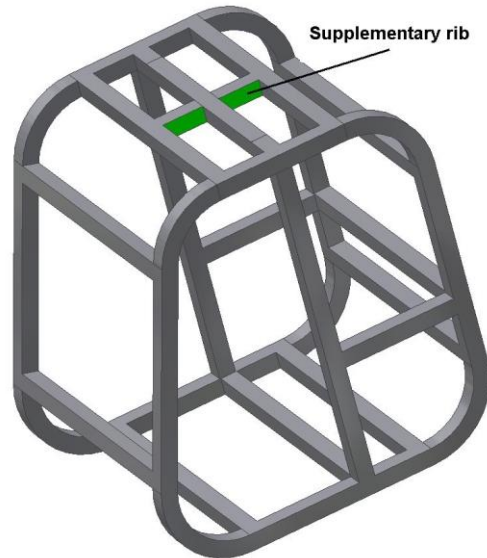


**Figure 10.** Cabin no. 2;  $F_x+F_y+F_z$  ; Stress

### 7. RESULTS FOR CABIN NO. 2 WITH MODIFIED GEOMETRY

For cabin no. 2 the maximal VonMises stress value is greater than the yield strength  $\sigma_c$  into the plastic domain, which impose geometrical modifications, with the goals of decrease the value

of VonMises values under the limit of the yield strength  $\sigma_c$  and the structure to work in elastic domain. For this reasons, the geometry was modified with a supplementary rib on the superior zone of the framework cabin no. 2, fig. 11.



**Figure 11.** Modified cabin no. 2

The modified cabin's geometry no. 2 was meshed in 9596 finite elements with 19158 nodes,

The study results for modified cabin no. 2 are presented in table no. 4, from where the following conclusions result: for individual lateral and back loads the result are similar with the result for unmodified cabin no. 2; for simultaneous loads corresponding to the three directions OX, OY, OZ, the maximal displacement value is reduced at 4.55 mm, fig. 12, and the VonMises maximal stress value is reduced at 168.8 MPa on the bottom face of shell mesh, fig. 13.

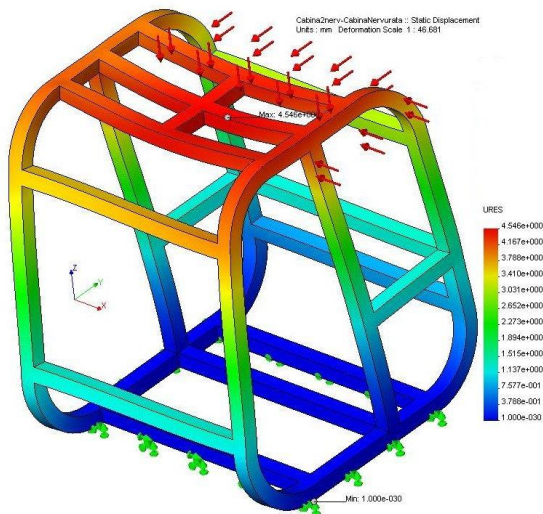
**Table 4.** Results for modified cabin no. 2.

Cabin no. 1 with rib				
Load case	Loads	$\sigma_{\text{VonMises}}$		$\Delta$
		Top	Bottom	
	N	MPa	MPa	mm
Lateral	$F_x=14400$	85.3	79.1	3.43
Back	$F_y=11520$	114.5	91.28	2.77
Vertical	$F_z=47064$	82.2	89.2	1.45
All	$F_x+F_y+F_z$	154.0	168.8	4.55

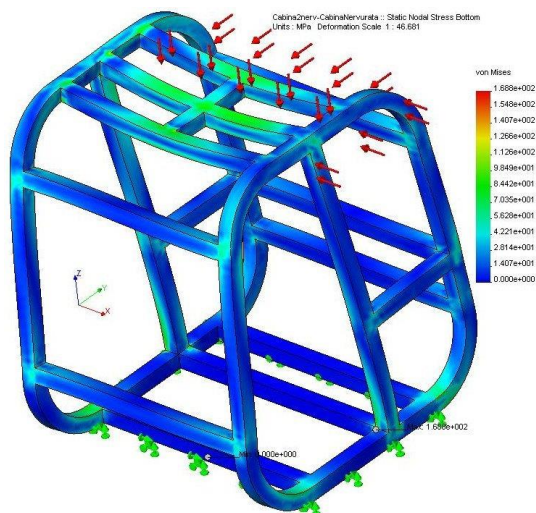
The comparative values form table no. 3 and 4, show that the rib has a significance influence for vertical load case, which decrease the maximal values for stress and displacement with 50%; this effect is also signaled for simultaneous loads, so the



maximal stress is decreased under the limit of the yield strength  $\sigma_c$  into the elastic domain.



**Figure 12.** Modified cabin no. 2;  $F_x+F_y+F_z$ ; Displacement



**Figure 13.** Modified cabin no.;  $F_x+F_y+F_z$ ; Stress

displacement values and send the work resistance domain from the plastic to elastic.

Together, the Autodesk Inventor and finite element Cosmos Design Star software offer to the engineer the necessary tools for optimize the resistance structures, by computer numerical simulation, and the possibility to select optimal solution for specified criteria.

### Bibliography

1. **Nedelcu, D.** *Modelare parametrică prin Autodesk Inventor*, Editura „Orizonturi Universitare” Timișoara, ISBN 973-638-116-1, Mai, 2004.
2. **Mănescu, T.Ș., Nedelcu D.**, *Analiză structurală prin metoda elementului finit*, Editura „Orizonturi Universitare” Timișoara, ISBN 973-638-217-6, Octombrie, 2005.
3. **Nedelcu D, Mănescu, T.Ș, Câmpian C. V.**, *Finite Element Through COSMOS M/Design STAR*, FME Transactions Volume 32, Number 1, 2004, University of Belgrade, Faculty of Mechanical Engineering YU ISSN 1451-2092, 2004.
4. \*\*\*\*\*, *COSMOS Design STAR Basic User's Guide*.

## 8. CONCLUSIONS

The paper analysed the resistance structure of a framework for mobile equipment, for many transversal framework section, to reduce the maximal stress value under the value of the yield strength  $\sigma_c$ . The optimal solution was selected on computer numerical simulations.

For the same loads applied to framework, the quadratic section  $\square 80 \times 80$  is better than cylindrical section  $\Phi 80$ , for resistance point view.

To reduce the maximal VonMises value the geometry of cabin no. 2 was modified by adding a supplementary rib, which decrease the stress and