The 4th International Conference
″Advanced Composite Materials Engineering ″
COMAT 2012
18- 20 October 2012, Brasov, Romania

A COMPARATIVE STUDY FOR TWO TYPES OF FRAME USED FOR DUMMY NECK IMPACT TESTING DEVICE

Ciunel Stefanita[1], Popa Dragos-Laurentiu[2], Campian Ovidiu Vasile[3]
[1] University of Craiova, Craiova, ROMANIA, ciunel_stefanita@yahoo.com
[2] University of Craiova, Craiova, ROMANIA, popadragoslaurentiu@yahoo.com
[3] Transilvania University of Brasov, Brasov, ROMANIA, c.ovidiu@unitbv.ro

Abstract: *The paper presents a complete study made on two mechanical structures for an impact testing device. The two structures were designed in Solid Works using main software modules. First, was made a FEA analysis on the two types of frames using a static loading, but the results were not conclusive. In that case, was made a dynamic analysis. The results were organized and analyzed. In the final, the paper presents important conclusion including the reasons for the choosing of the best type of framework.*
Keywords: *neck testing system, frontal impact, dummy, dynamic simulation, FEA analysis*

1. INTRODUCTION

According to Traffic Police statistics, the majority of road accidents in Romania in 29% irregular and inappropriate speed is due to road conditions. Among the victims (dead and wounded) of road traffic accidents, the first place is car drivers (64%), the second pedestrians (26%), followed by cycling (4%) and carters (1%).
The head-neck test device designed by our research team was a complete system used for the calibration and testing of the neck component for the Hybrid III family, BioSID, EuroSID-1 used for impact tests. The neck can be tested in both the flexion and extension modes. The neck performance specifications for these tests are velocity at impact, pendulum acceleration, total rotation of the head/neck system, moment and force about the occipital condyle. According to the above mentioned devices we tried to study the first time in a virtually finalized with the practical implementation of a device for testing in a laboratory studying cranio-cervical complex behavior in situations similar to real ones [1], [2], [3], [4], [7], [8], [9]. The paper presents a complete study made on two mechanical structures for an impact testing device**.**

2. THE STATIC VIRTUAL TESTING OF THE TWO STRUCTURES [5], [6]

To realize a dummy neck device for testing were analyzed two structures shown in Figure 1.

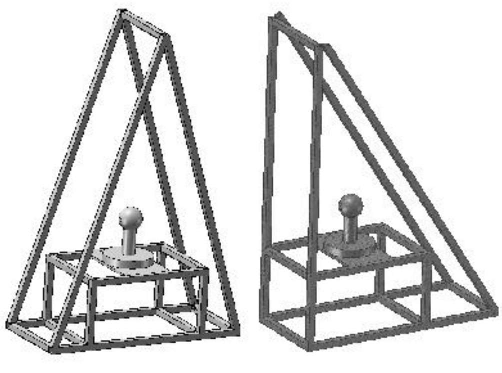

Figure 1: Two structures of framework for the testing device.

The main dimensions of the two structures are shown in Figure 2.

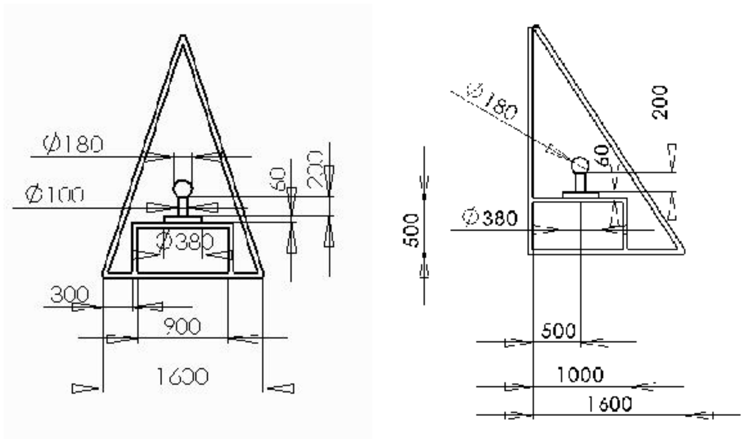

Figure 2: The main dimension of the two structures

The stand is made of metallic rectangular 50x50 mm pipe. Initially, to chose the optimal shape, did not joining elements (welding, threaded components etc.).
These three-dimensional structures were generated in a parametric modeling program and were subjected to virtual testing began on static checking and comparing results.
For static testing of the two structures was used finite element method and to simulate the load we follow the next steps:
- was calculated load (force) similar to hitting the table stand sphere to another sphere of mass 10 kg, with acceleration of about 30G:

$$F= m*a \approx 10*30*10=3000 \text{ N} \quad (1)$$

- constrains were set for both the fixed-type structures (Figure 3).

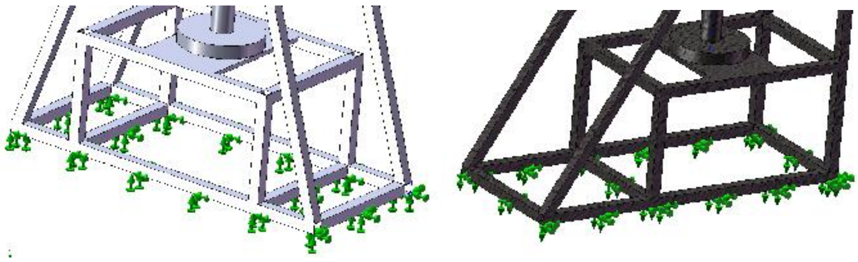

Figure 3: Fixed constrains for the bases of the two structures.

- attached to a force of 3000N on the sphere field of stands with normal direction (Figure 4).

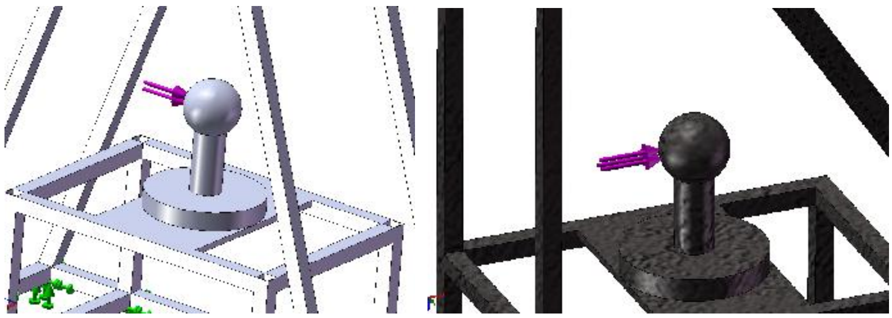

Figure 4: Establishing the position, direction and size of force applied to the systems of the two structures.

- to the nexy stage was startes the dividing process into finite elements using “tetrahedron” type. Thus, the structure of “isosceles triangle” (shown in Figure 5) to obtain a structure with following characteristics:

-average dimension of the solid finite element: 28,07 mm
- average tolerance of the solid finite element: 1,4 mm
- high quality of dividing operation
- total numer of the nodes of finite elements: 82365
- total number of of the solid finite elements: 42368
- the precent of the distorting elements 0%

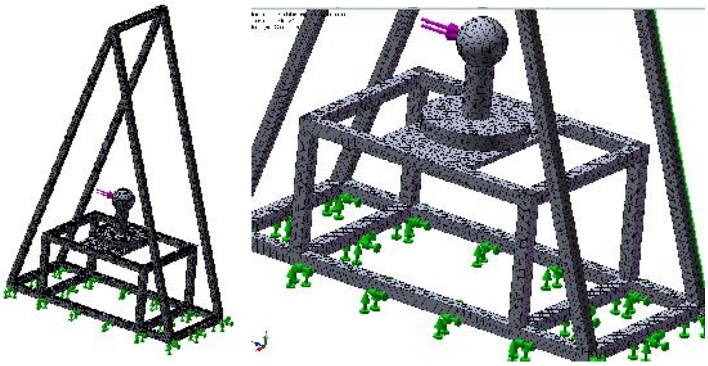

Figure 5: Solid finite element division of the structure of “isosceles triangle”

For the structure of "right triangle" (shown in Figure 6) was obtained a structure of finite volume elements (solid) with the following characteristics:

-average dimension of the solid finite element 30 mm
- average tolerance of the solid finite element 1,5 mm
- high quality of dividing operation
- total numer of the nodes of finite elements 58774
- total number of of the solid finite elements 30318
- the precent of the distorting elements 0%

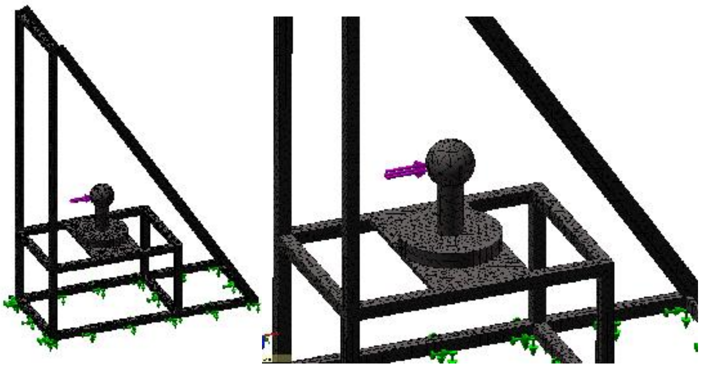

Figure 6: Solid finite element division of the structure of "right triangle".

- in the final stage were obtained simulation results of static loading consisting of tension maps, displacements and strains. For comparison, these results were presented in Figures 7, 8 and 9. To observe correct distortions and deformation modes was used scale of 250 to 1.

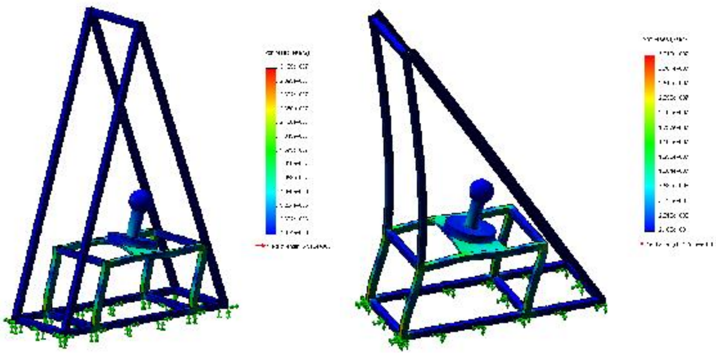

Figure 7: The compared maps of von Mises stress for the two structures.

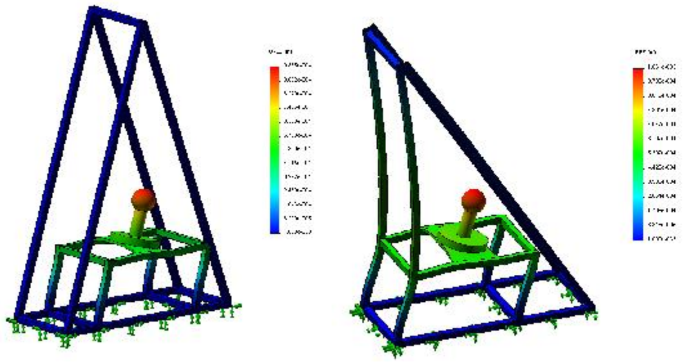

Figure 8: The compared maps of displacement for the two structures.

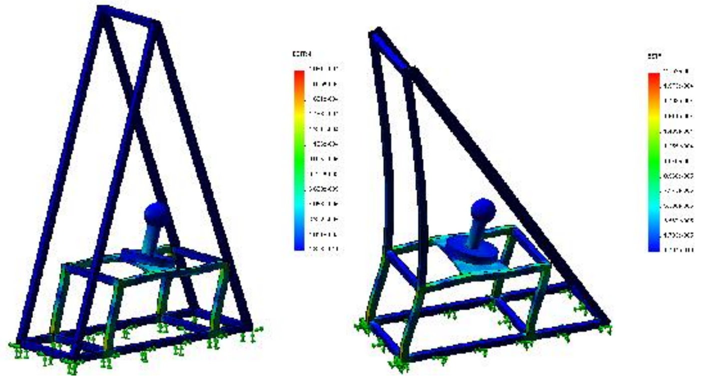

Figure 9: The compared maps of strain for the two structures.

The analysis of these results requires a comparative organizational values of maximum stress, displacements and strains. These data are extracted from the maps above and are summarized in Table 1.

Table 1: Comparative data results

	Isosceles triangle structure	Right triangle structure	Difference	Difference [%]
Stress	3,159x10^7 Pa	3,012x10^7 Pa	0.147 x10^7 Pa	4,6%
Displacement	0,987 mm	1,061 mm	-0,074	-7,4%
Strain	1,981	2,152	-0,171	-8,6%

Analyzing the results obtained after running the application can draw the following conclusions:
- Comparing the resulting maps and data summarized in Table 1 it is found that the values are significantly close, maximum stress is smaller displacements and deformations higher for the structural variant of "right triangle";
- In terms of layout stress, strains and displacements is found that if the structure type "isosceles triangle" are taken by table stand (unfavorable), while the structure of the type "right triangle" are taken by the frame (favorable);
- Maximum values and simulation results for the static load testing are very close is needed an additional test to chose the optimal structure.

3. THE DYNAMIC VIRTUAL TESTING OF THE TWO STRUCTURES [5], [6]

To test the two structures in dynamic regime, models have been imported under a software of kinematics and dynamics analysis. In this environment, the two structures were hitting tested with a sphere with mass 10 kg, and acceleration of 30G. In figure 10 are presented the models of the two structures. Virtual test system has following characteristics:
- The system consists of three models: a sphere weighing 10 kg, stand structure and a base (soil);
- The structure is single-seated on the ground, friction between these components with coefficient of 0.17 (steel-steel);
- the sphere is driven by a virtual actuator for 0.2 seconds acceleration of 30G, then the sphere is free to hit area of the stand component;
- Observation and testing period is 0.8 sec.

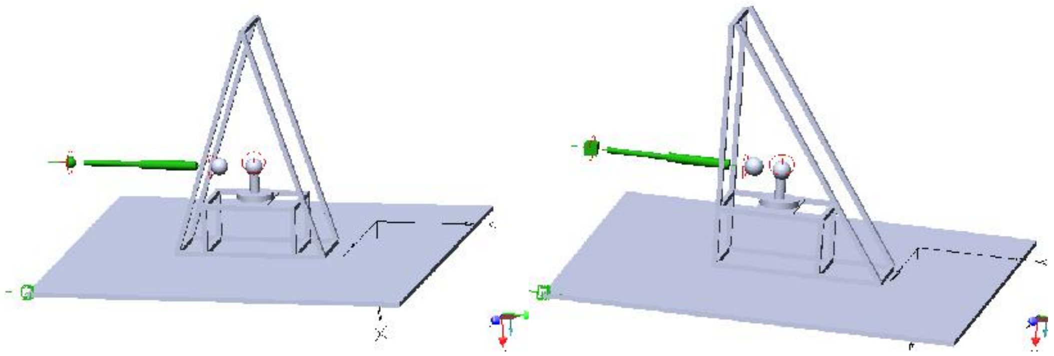

Figure 10: The two structures supposed to the dynamic testing.

A first result is the simulation movie. For the structure "isosceles triangle" in Figure 11 were presented three main frames.

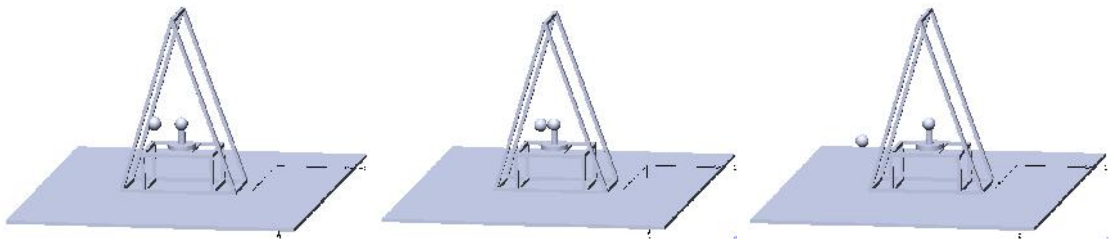

Figure 11: Main frames of the dynamic test aplied to the "isosceles triangle" structure.

For the "right triangle" structure in Figure 12 are shown three main frames.

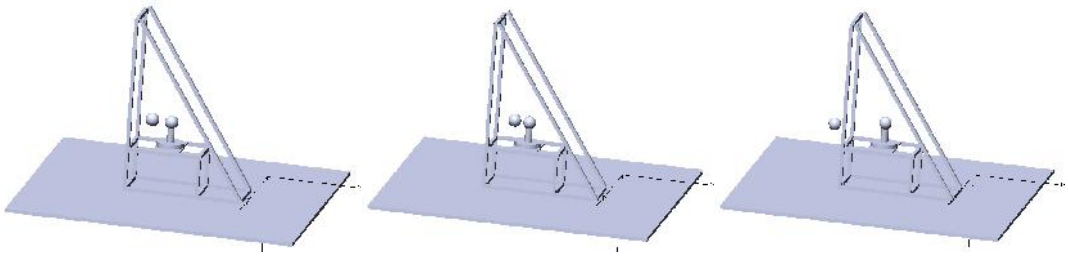

Figure 12: Main frames of the dynamic test applied to the "right triangle" structure.

Kinematics and dynamics simulation environment allows to determine all kinematic and dynamic parameters of system components studied. To analyze and compare the stability of the two structures in terms of dynamics, diagrams were automatically generated for speed and acceleration structure analysis. In Figures 13 are velocity and acceleration charts for "isosceles triangle" structure.

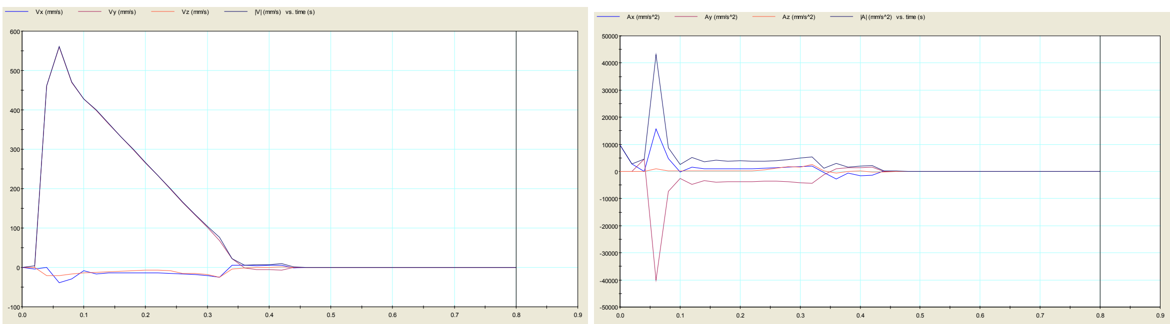

Figure 13: The velocity and acceleration components for the "isosceles triangle".structure.

In Figures 14 are presented velocity and acceleration charts for "right triangle" structure.

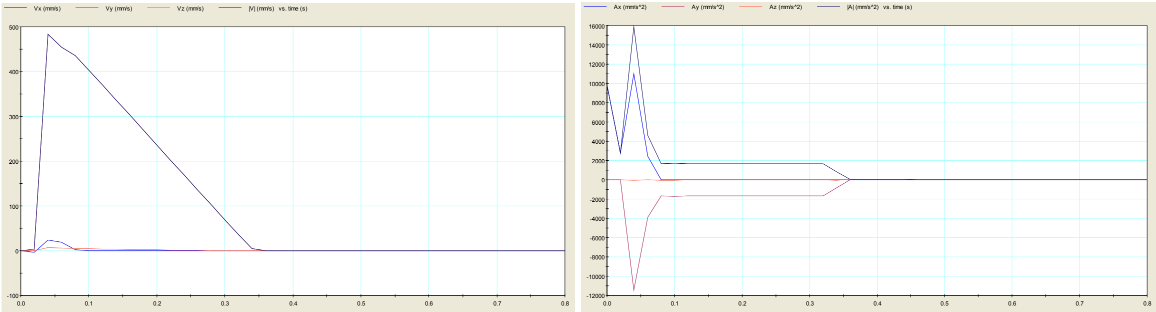

Figure 14: The velocity and acceleration components for the "right triangle" structure.

4. CONCLUSIONS

The analysis of the diagrams presented in Figures 13-14 which describe the dynamic behavior of the two structures supposed to the virtual test, can draw the following conclusions:
- for the "isosceles triangle" structure, the speed ball immediately after impact is 0.559m/s and acceleration at the same time is 43.1 m/s;
- for the "right triangle “structure" speed ball immediately after impact is 0.483m/s and acceleration at the same time is 15.9 m/s;
- after impact "isosceles triangle structure " is moving to 274 mm, and the "right triangle" structure moves by 80 mm.

Studying static and dynamic simulation results, it can be concluded that the "right triangle" structure is more stable than "isosceles triangle" structure and can be chosen to develop a test device for the dummy neck model.

REFERENCES

[1] S Georg Christian Kamm, Novel Apparatus for Evaluation of Head and Neck Injury, Texas Tech University, 2003.
[2] FMVSS 208, Adult Occupant Protection in Frontal Impact.
[3] ISO/TR 7861:2003 -Road vehicles -- Injury risk curves for evaluation of occupant protection in frontal impact.
[4] Lansade Celine – Analyse cinematique tridimensionelle du rachis cervicsl sain et pathologique in vivo, Laboratoire de BioMecanique, Arts et Metiers ParisTech, 25 juin, 2009.
[5] SolidWorks – User Guide and Tutorials, 1995-2012.
[6] VisualNastran – User Guide and Tutorials – 2001-2004.
[7] Withnall et al. - U.S. Patent, No. 6871525B2/03.29.2005, Method and apparatus for testing football Hemlet.
[8] http://www.humanaeticsatd.com
[9] http://www.dummymodels.com