ANALELE UNIVERSITĂȚII
"EFTIMIE MURGU" REȘIȚA
ANUL XVIII, NR. 1, 2011, ISSN 1453 - 7397

Dan Alexandru Micu, Daniel Mihail Iozsa

Analysis of the Rollover Behavior of the Bus Bodies

When performing calculations using ANSYS software have been achieved so far, only a part of the requirements imposed by regulation on how to conduct computerized mathematical modeling, the paper presented only a first set of results in this area, following research should be conducted detail.

***Keywords**: analysis, bus, structural, rollover*

1. Introduction

European regulation "ECE-R66" entitled "Resistance of the superstructure of Oversized Vehicles for Passenger Transportation" is able to prevent catastrophic rollover accidents to ensure the safety of bus passengers [6]. It applies to single-deck rigid or articulated designed and constructed for the carriage of more then 22 passangers, whether seated or standing, in addition to the driver and crew. The purpose of this regulation is to ensure that the vehicle superstructure has sufficient strength so that the residual space during and after the rollover, like luggage, is intruding into the residual space and no part of the residual space projects outside the deformed structure.

The rollover test is a lateral tilting test specified as follows: the complete vehicle is positioned on a tilting platform, with the suspension locked, and tilts slowly to the unstable equilibrium position. Trying to start rolling in this unstable position of the vehicle with zero angular velocity, rotation axis passing through the point of contact wheel - ground. Car overturns into a ditch, having a horizontal, dry and smooth concrete ground surface with a nominal depth of 800 mm [6]. The rollover test shall be carried out on side of the vehicle that is more dangerous with respect to the residual space.

Computer simulation of a rollover test on a complete vehicle is an approval method. It allows manufacturers to test design and safety features virtually in the crash scenario until they obtain the safest and optimum design, thus saving time and money in developing expensive costly prototypes [2].

2. Creating models using nonlinear finite element

In order to calculate the resistance of a body structure, ANSYS was used - a computer software specialized in the analysis of structures that use the finite element method.

To make such a calculation the completion of specific stages is required: the designing of the geometrical model; of the finite element model; making the actual calculations and, finally, results interpretation.

Only a segment of the bus body was designed. This segment is composed of a caisson bearing elements made of steel truss type and superstructure (lwalls and roof side) made of stainless steel. The segment is fitted to the bottom with front and rear of the bus modules.

The complete geometric model as well as the finite element model contain the space of survival. In this way it will be easier to observe whether during the process the structure enters this space.

The structure is made of rolled bars of a rectangular, square or L-shaped profile. The entire geometrical model was composed of surfaces, while trying to keep a high accuracy with the shape and junction of the projected bars' profiles.

Figure 1. Show the geometrical model developed through the use of ANSYS software.

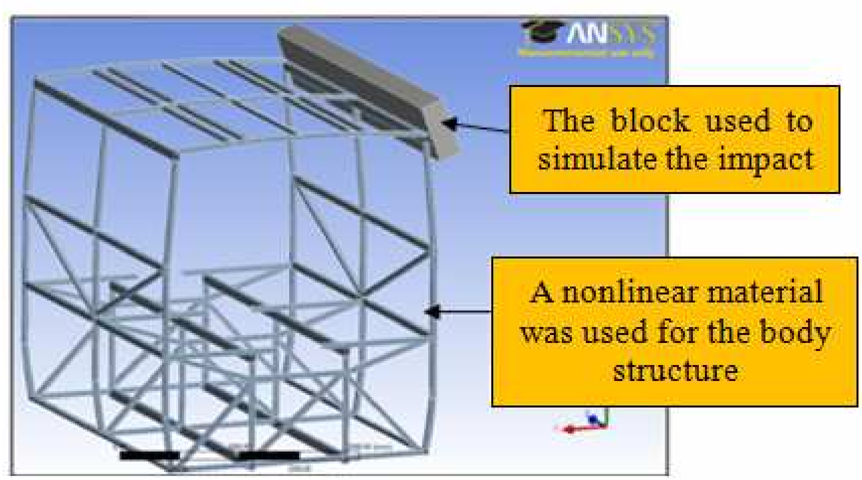

Figure 1. Geometric model constructed through the use of software.

The finite element model is based on the geometric model by finite element mesh geometry. The mesh only used shell elements.

The method used is the rollover test simulation by hitting with a steel block on the edge of the structure on the direction in which it would hit the wall horizontally when the rollover takes place. The structure will be blocked at the basis of the chassis frame.

The initial potential energy of the pendulum must be equal to the total energy of impact indicated in relation 2.1, according to ECE Regulation 66. This could easily determine, by calculation, according to the pendulum mass (depending on volume and density of its material) and the length of the pendulum, the height h, to which the pendulum should be initially raised, respectively the drift angle of the pendulum (Figure 2.).

Collision with the pendulum method is a method specified by the above mentioned regulation as an experimental analysis method of rollover.

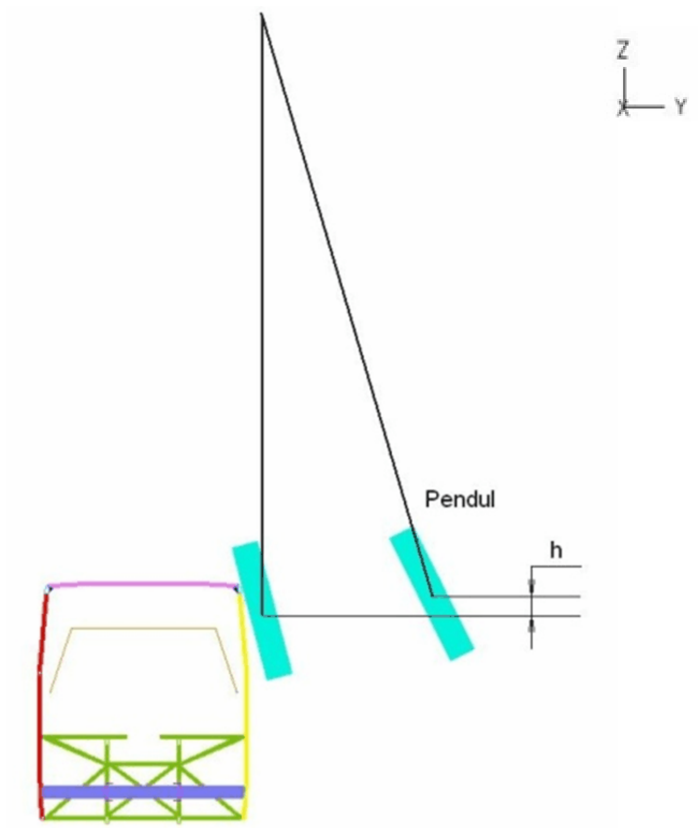

Figure 2. Striking geometric model of the pendulum.

The total energy of impact is established by determining Δh via a graphical method:

$$E = 0.75 M g \Delta h \quad (1)$$

where:

E - total energy of impact;

M - structure's mass under test, test bus segment length being approximately equal to a quarter of its total length;

g - gravitational acceleration;

Δh - distance from the center of gravity position of the structure when it is tested on the bench until its final position after capsize or vertical displacement of the center of gravity during the rollover;

In order to determine the travel distance of the center of gravity, the graphic method specified in Regulation ECE R 66 is used. Thus, the displacement of the center of gravity is equal to the difference between the position of center of gravity when the structure is tilted bench position limit equilibrium and center of gravity position when the plane reaches the lower trench structure.

Determination of center of gravity position was necessary in order to determine the inclination to which the structure topples over the CG_2 position. Its position was determined by underpinning it on four points on a lateral side and determining the reaction forces.

Using the results on the position of the gravity center, graphical methods were used to determine the inclination of the structure in relation to the inclined plane, at early impact, and the position of the structure at the moment of the impact (CG_3), in order to determine the inclination of the pendulum and the height to which the pendulum needs to be initially raised.

Pendulum inclination was measured by using the specifications in Regulation R-66, and is equivalent to the angle at which a quasi-static force is applied on the structure.

Using the energy conservation law, by equating the kinetic energy of the block, at the moment of the impact, with the total energy impact, the speed value that needs to be actioned on the block can be identified. Thus, the equality becomes:

$$Mg = 0{,}75 \frac{m \cdot v^2}{2} \quad (2)$$

where:

m - mass unit, determined by density and volume as follows:

$$m = \rho \cdot V = \rho \cdot L_b \cdot l_b \cdot h_b \quad (3)$$

ρ - density of steel used for the block;
L_b, l_b, h_b - dimensions (length, width, height) that define the block;
v - speed of the impact block, [m / s];

3. Interpretation of results

Ansys software allows drawing diagrams that portray the evolution of a particular parameter. Thus, Figure 3 shows the time evolutions of strain, and the values corresponding to this paragraph are presented in Figure 4.

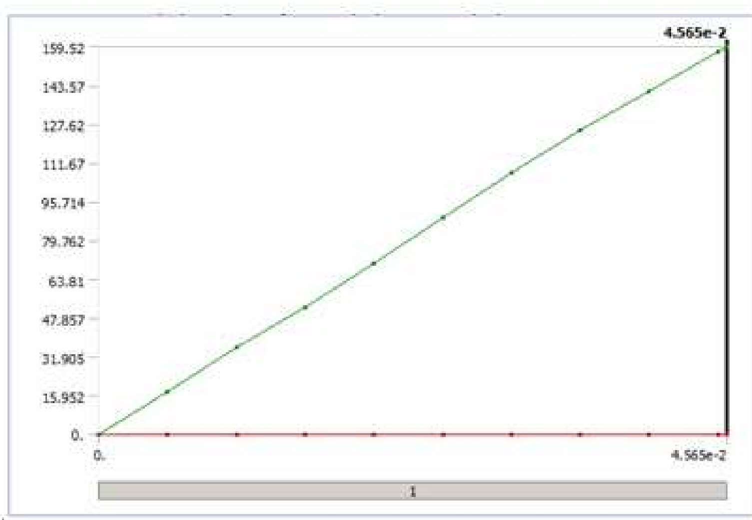

Figure 3. Total deformation in time.

For this study case, observations show that the strain has a linear evolution in time, at least until moment 0.046 s. Due to the very long duration (2-3 days) required by the process. The processing times of very long duration (2-3 days) were an impediment to achieving a completion of results.

TABLE 51
Model (A4) > Explicit Dynamics (A5) > Solution (A6) > Total Deformation

Time [s]	Minimum [mm]	Maximum [mm]
0.		0.
5.e-003		17.203
1.e-002		35.814
1.5e-002		52.368
2.e-002		70.504
2.5e-002	0.	89.289
3.e-002		107.49
3.5e-002		125.
4.e-002		141.29
4.5e-002		157.39
4.565e-002		159.52

Figure 4. Total deformation.

Figure 5 presents the evolution over time of equivalent stress and values are presented in Figure 6.

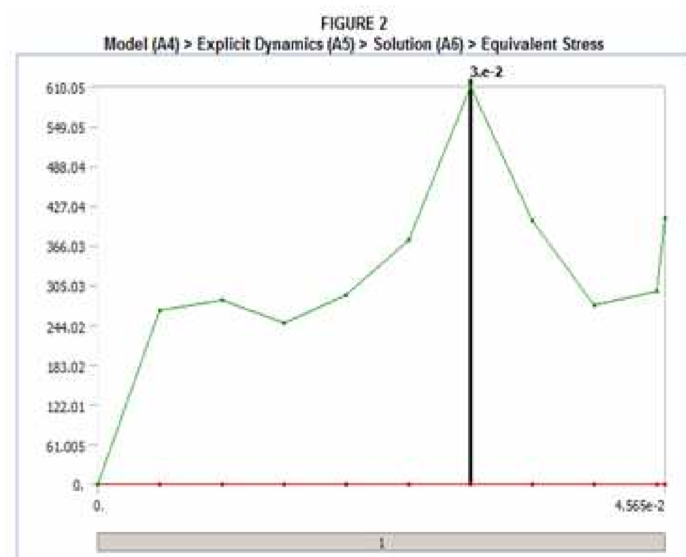

Figure 5. Equivalent stress evolution in time.

TABLE 52

Model (A4) > Explicit Dynamics (A5) > Solution (A6) > Equivalent Stress

Time [s]	Minimum [MPa]	Maximum [MPa]
0.		0.
5.e-003		266.68
1.e-002		283.23
1.5e-002		247.6
2.e-002		290.06
2.5e-002	0.	375.13
3.e-002		610.05
3.5e-002		404.2
4.e-002		274.96
4.5e-002		296.42
4.565e-002		407.41

Figure 6. Stress - time equivalent.

Figure 7 shows the time evolution of relative plastic strain mesh with elements of maximum 50 mm length.

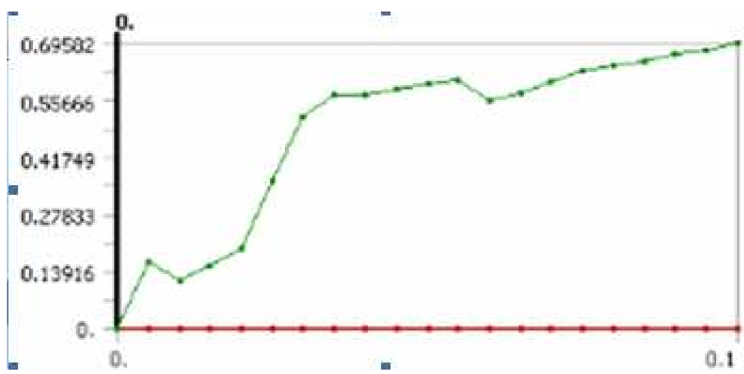

Figure 7. Plastic strain variation diagram.

It is noted that at the time of impact, the structure is subjected to initial relative plastic strain that has an increasing tendency value , with few negative variations. At the point of maximum plastic strain, a continuing growth trend is recorded. Therefore, further study needs to be conducted in order to identify the maximum deformation value of the structure.

Figure 8 shows the maximum deformation recorded in relation to survival space. It can be seen that the bus structure does not enter the bus residual space, under the performed test conditions.

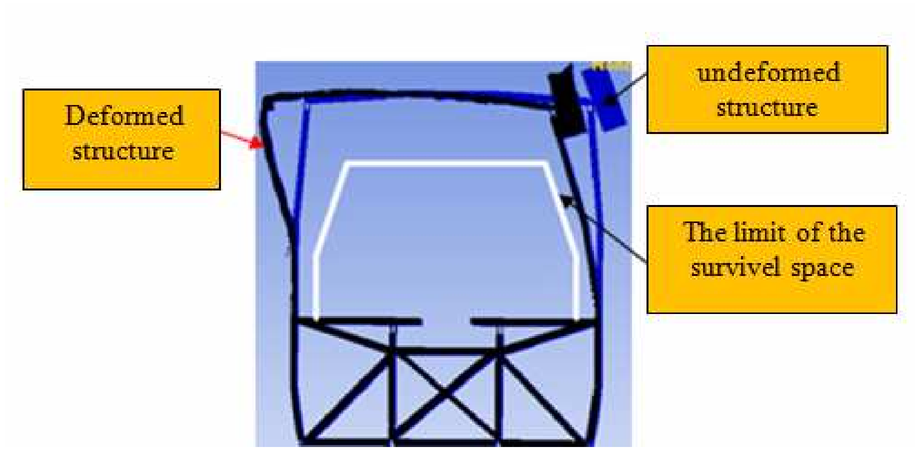

Figure 8. Highlighting the maximum deformation recorded in relation to the survival space.

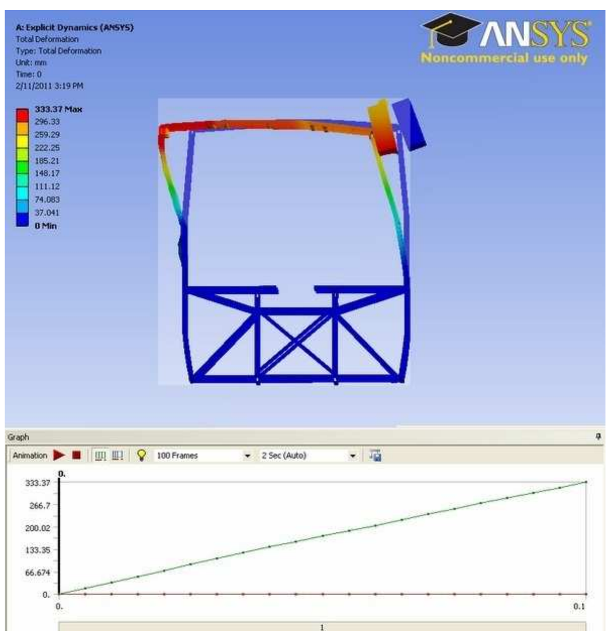

Figure 9. Regional deformations of the structure.

Maximum deformation is presented through the deformed structure overlayed on the undeformed one, Figure 9. Also presented underneath, is a graph of the strain variation in time.

4. Conclusions

The resistance was calculated for a tourist bus superstructure, with a stainless steel body shell and a middle chassis profile made out of carbon steel, designed and built by Grivita S.A. Industrial Company, Bucharest. Calculations have sought a verification of the requirements on the bus superstructure strength, to be in accordance with Directive 2001/85/EC (ECE Regulation No. 66-A) in order for this type of vehicle to be granted approval. Superstructure resistance check was made by calculation, analyzing the overturning a section of the superstructure, in accordance with the regulation. The segment that has been verified, the percentage of total weight assigned to this segment and the distribution of these weights on the segment structure, were established by the manufacturer.

When performing calculations using ANSYS software, only a part of the imposed requirements, on how to conduct computerized mathematical modeling, have been achieved so far. Thus, the paper represents only a first set of results in this area, following that further research shall be conducted in detail.

References

[1] *F2006V226- On Furthering The Understanding of Vehicle Rollover Through State of the Art Test and Simulation*;

[2] *Rollover and Roof crush analysis of low-floor mass transit bus*, Department of Mechanical Engineering, Graduate School of Wichita State University, December 2006;

[3] Martinez L, Aparicio F, ș.a., *Improving occupat safety in coach rollover*, INSIA, Polytechnic University of Madrid, Spain, 2003;

[4] Meghan Elisabeth Henty, *Virtual Simulation of a Pickup Truck Rollover Test Using the Nonlinear Finite Element Cod PAM-CRASH*, Master Thesis, The Pennsylvania State University, May 2003;

[5] M Mao, E.C. Chirwa, W. Wang, *Assessment of vehicle roof crush test protocols using FE models: inverted drop tests versus updated FMVSS No. 216*, The University of Bolton, UK, 2006;

[6] *Regulamentul ECE R 66*;

Addresses:

- As.Drd.Eng. Dan Alexandru Micu, "University POLITEHNICA of Bucharest", Splaiul Independentei, nr. 313, 60042, Bucharest, dan.alexandru.micu85@gmail.com
- Conf. Dr. Eng Mihail Daniel Iozsa, "University POLITEHNICA of Bucharest", Splaiul Independentei, nr. 313, 60042, Bucharest, daniel_iozsa@yahoo.com