DESIGN AND STUDY OF DOOR COMPONENTS FOR A TWO-SEATER ELECTRIC VEHICLE IN SIDE IMPACT CONDITIONS

Panagiotis Bazios[*], Polychronis Spanoudakis, Nikolaos Tsourveloudis

School of Production Engineering and Management, Technical University of Crete, Greece

KEYWORDS –
FEM modelling, side impact, crashworthiness, ANSA, μETA,

ABSTRACT –

Crash simulation is extensively used by the automobile industry for the development of vehicles with regard to their passive safety – known as crashworthiness. This paper presents the use of Finite Element Method (FEM) modeling under side impact conditions (Euro NCAP) and design optimization of various components for a prototype two-seater electric vehicle, investigated with the aid of ANSA and μETA software. Considering an existing chassis, doors, side panels, door hinges and impact bars, are designed and evaluated targeting: 1)simplicity in manufacturing, 2)lower weight depending on use of materials and 3)security provided to occupants. The mentioned structural parts were modeled and analyzed given the restrictions of materials mechanical and physical properties. Moreover, the extensive use of impact analysis targeted on reducing the number of experiments, thus minimizing cost and development time of the components. Specifically, the effect of geometry modelling (parameters and elements) on simulation time was considered and used to provide adequate results in reduced time. Thus, the capabilities offered by ANSA in meshing of different geometries of the vehicle, giving us the ability to export realistic and reliable results are discussed and presented. Final results present the effect of impact tests towards components redesign, leading to different component thickness control and evaluation of stress distribution.

TECHNICAL PAPER -

1. INTRODUCTION

The automotive industry today is confronted with perhaps some of the most significant technical challenges in its history. In most regions of the world, regulations and legislation are being enacted to enforce higher fuel economy and lower tailpipe emissions. In all cases, the trend is upward, and in some regions, the impending regulations are forcing a substantially greater emphasis on improving fuel economy.
Crashworthiness and occupant safety is another area affecting vehicle design where regulations and legislation are becoming more strict. The National Highway Traffic Safety Administration in the U.S. is in the process of enacting changes to its safety rating program that will be the most significant since the rating program began in 1979. Similar changes are underway in Europe and other parts of the world [1].
These challenges, along with quickly changing customer preferences, and the ever-increasing market demand for better vehicle performance and reliability require automotive companies to rethink past approaches and to find better ways to design better vehicles at lower costs in shorter time. Simulation provides a strong opportunity and means to bring about some of the necessary cost efficiencies, while at the same time providing the platform to drive innovative product development. Moreover, simulation methods continue to advance in sophistication as well as the level of fidelity and accuracy they can provide. This ongoing evolution is partly fueled by the relentless geometric growth in the capacity and availability of high performance computing. In this paper the use of such simulation methods is used to design the doors of a new two-seated electric vehicle, through series of side impact crash tests. Occupant protection and crash worthiness are the key objectives in order to result to

the best possible product design that will meet the aforementioned requirements and additionally: Construction simplicity, ease of processing and placement of materials. Furthermore, with the use of ANSA sophisticated tools it becomes easy to control and overcome modelling and meshing problems reducing mesh size, complexity and simulation time.

2. DESIGN REQUIREMENTS

As already mentioned, the goal of this project is the design of door components that meet the following requirements:

• Construction simplicity
• Ease of manufacturing and placement of materials
• High rigidity and safety

According to the above, it was decided to use conventional type doors. The assembly of the door to the cabin of the vehicle consists of five pieces in total (Figure 1):

1. Side panel
2. The inner skin of the door
3. The outer skin of the door
4. The impact bars
5. Hinges

The vehicle chassis which forms the passenger cabin has been already defined and designed in a previous project [2]. Therefore, this was a first restriction in door components design. At first a side panel was defined, introducing the necessary geometrical specifications of the door components mount to the chassis. Then the design moved on to the inner and outer door panels and finally to impact bars placement. The set design target was to achieve reduced component complexity and low weight in order to reduce manufacturing cost and provide low vehicle energy consumption.

Figure 1 - Door components exploded view, as attached to vehicle chassis.

3. MODELLING

Shell elements were used for all components during FEM modelling. The type of analysis applied was dynamic structural and the objectives were to increase the stiffness and reduce weight by controlling the thickness of every component. Von Mises theory was used as the stress calculation function. At first, an approximation model with the use of laminate tool was

made. The thickness of the outer panel was chosen at 1.5mm, for the side panel 2mm, for the inner panel 1.2mm and for the impact bars 1mm. The material properties used in the model are presented in Tables 1, 2.

The materials used for almost all door components, except the impact bars, is Aluminum 6063-T6 and Steel 304 respectively. This choice was made accounting deflection and energy absorbed during impact conditions for different materials (Magnesium, Aluminum, Steel) as shown at Figures 2, 3 [3]. Magnesium was rejected due to its high cost, and thus specific, easy to find, aluminum and steel alloys where chosen for the parts already mentioned.

Table 1. Aluminum 6063-T6	
Density (x 1000 kg/m3)	2,7
Elastic modulus/ Young (GPa)	68,9
Poisson Ratio	0,33
Yield Strength (MPa)	241

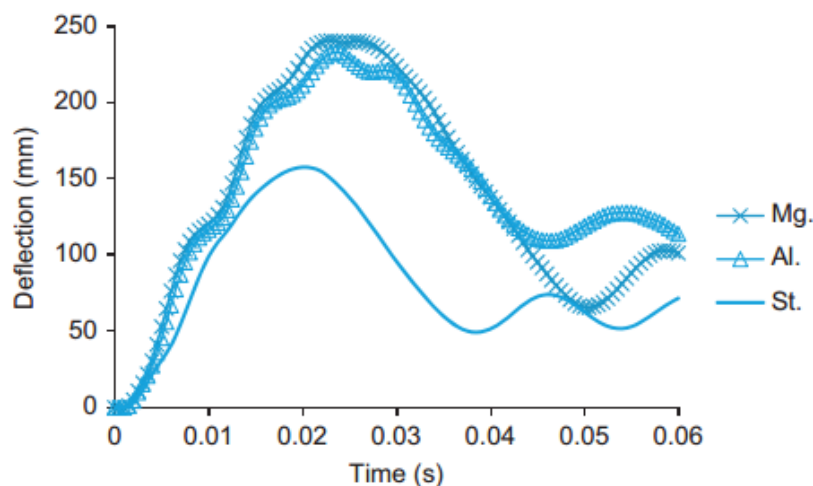| Table 2. Steel 304	
Density (x 1000 kg/m3)	7,8
Elastic modulus/ Young (GPa)	210
Poisson Ratio	0,33
Yield Strength (MPa)	215

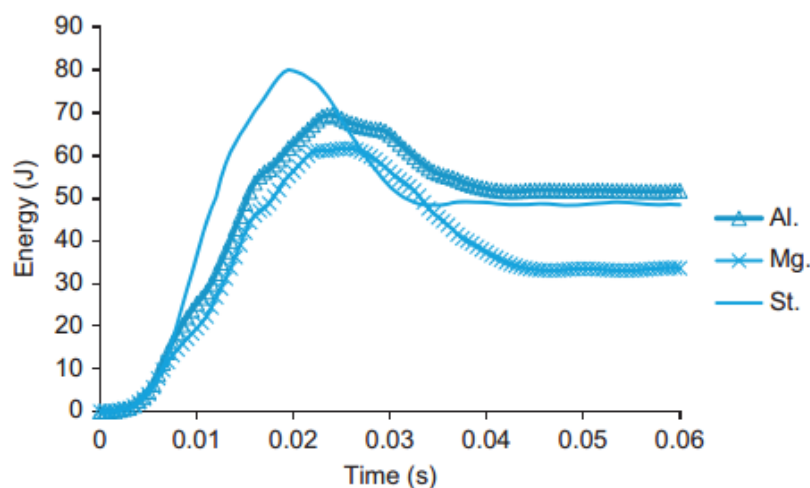Figure 2 - Diagram of deformation-time in side impact conditions

Figure 3 - Time-absorbed energy in side impact conditions

During the modelling process, ANSA is used as pre-processor. The finite element model of the vehicle is composed of *205253 quads* and *3131 second order trias* elements. Every component was assigned with different properties as shown at Figure 4.

Several ANSA functions and abilities where used in order to overcome geometry problems and crash test modeling parameterization using LS-DYNA. Specifically, the model consists of parts with complex geometries, which are in contact with each other. This causes a penetration of surfaces, which we have to face with. The *MIDDLE SKIN* function is able to solve the above problem producing high quality mesh and assigning nodal thickness [5]. Also, *Batch Mesh* gave us the opportunity to mesh the different geometries of the vehicle efficiently and quickly. Finally, the *ADDED MASS* function helped in reducing the elements of secondary parts (passengers, wheels, etc.) by replacing them with point masses at the center of gravity [4]. The above mentioned are just some of the sophisticated tools provided to users using ANSA as a leading CAE software for crash simulations.
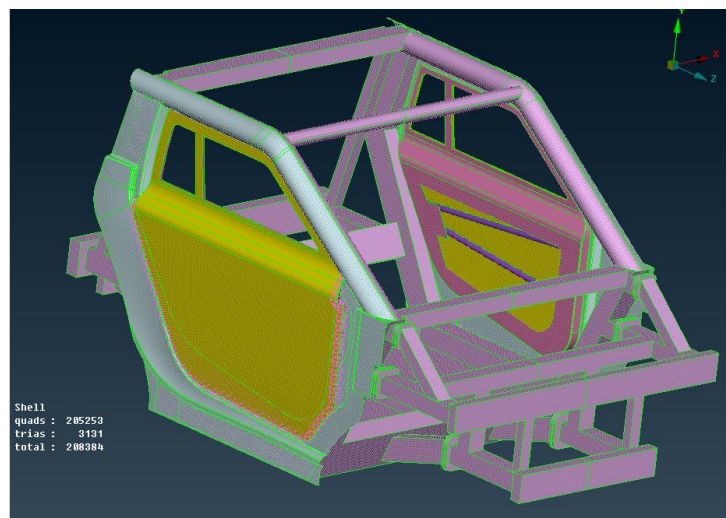
Figure 4 - Created finite element model in ANSA

4. EXPERIMENTAL SETUP

Car side impacts are the second most common crash configuration responsible for fatal and serious injuries. Side impact conditions are well defined according to EURO NCAP documentation [6]. Specifically, it is defined as the case where a stationary car is hit on the driver's door at 56Km/h by a deformable barrier mounted on a trolley. This assessment accounts the side intrusion-deformation as well as the data from a crash test dummy.

In our case the car body is subjected to a side impact analysis with a speed of 56 km/hour, and the complete model is modeled in ANSA (Figure 5). The objective is to find the optimal thickness of the door components, in order to minimize impact energy and the intrusion of the door to the cabin. The optimal thickness of door components is not uniform, but it is needed to increase the thickness in areas where needed. In order to reduce simulation time, just the front part of the impacting vehicle is used, by replacing the rest of the vehicle model with point masses, using the *Added Mass* function. Simulation time is reduced by minimizing grid elements.
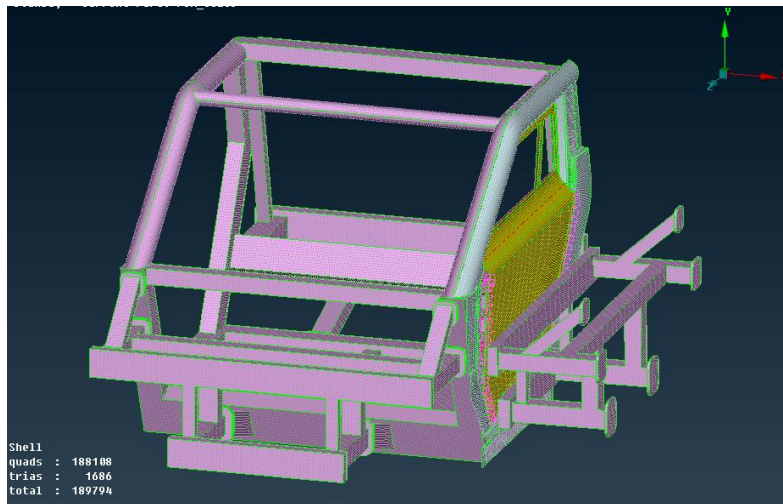
Figure 5 - Created finite element model with impactor in ANSA

5. RESULTS

After several simulation and redesign phase continuous improvements, a final solution was reached, ensuring high component stiffness in order to avoid door intrusion and provide reduced impact stresses. At first, the side panel thickness was set at 2mm, the inner and outer panel at 1.5mm and the impact bars with a thickness of 3mm. At the figure following (Figure 6), various side impact simulations are presented, at different component thickness, as displayed in μETA [7].
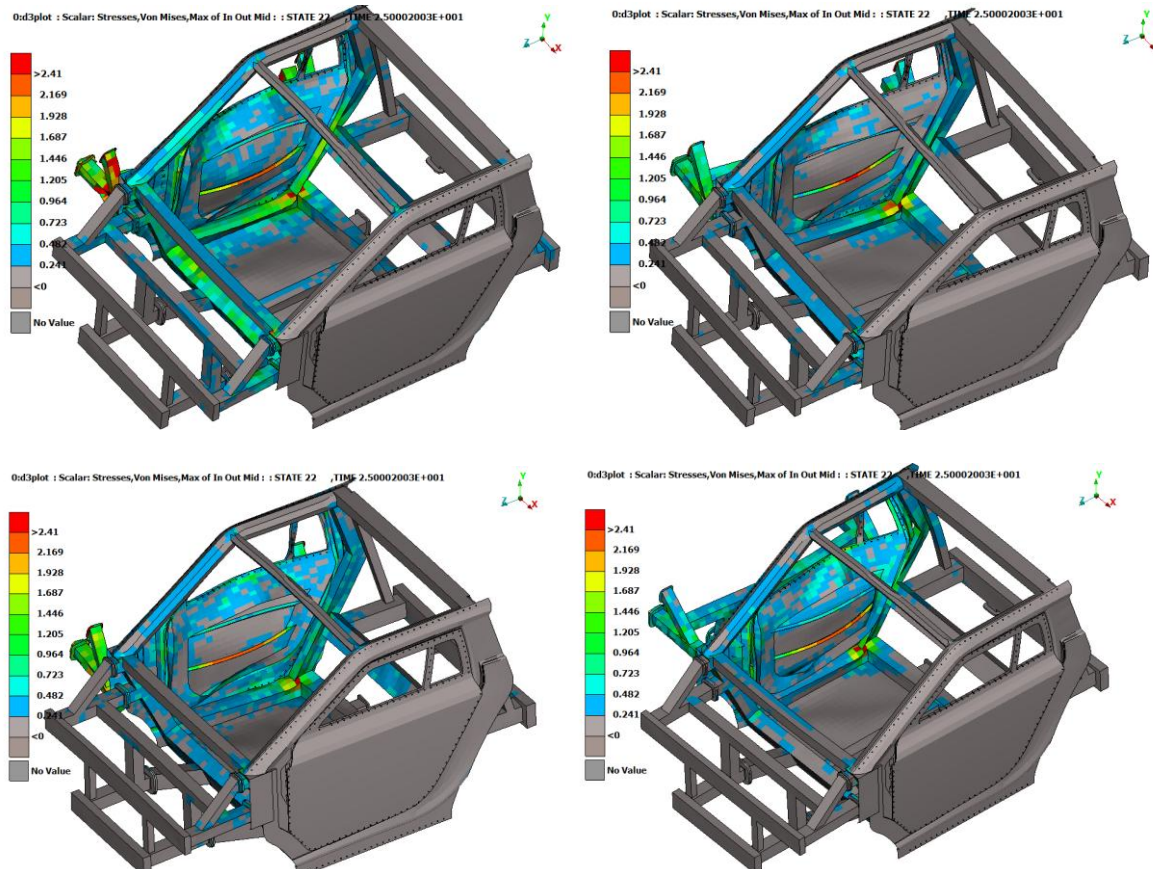

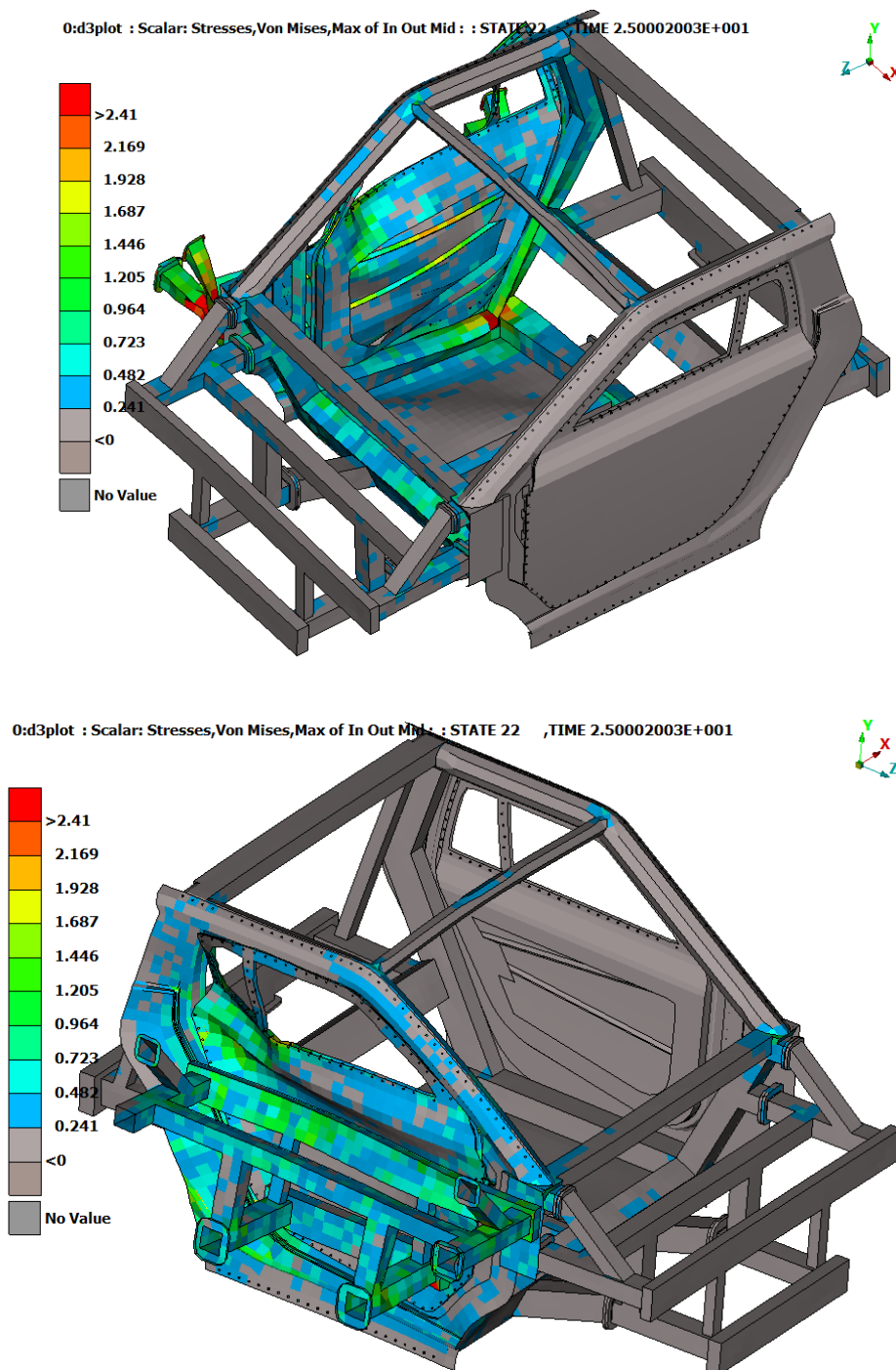Figure 6 – Side impact simulation trials

Figure 7 – Final side impact stress display.

In the final best case, all door components result to stresses lower than the material yield limit (Figure 7). It can be observed, that the maximum stresses are applied at the impact bars with a max of 192 MPa (below the steel yield strength which is found 215 MPa). Also, it is shown that deformation in the area of the door is extensive but within the permissible limits. The aluminum sheets low thickness results to door high deformation while mainly the impact bars absorb the most of the crash energy. All this leads to the conclusion that the main component for impact energy absorption is the impact bars.

6. CONCLUSIONS

This paper presents the use of Finite Element Method (FEM) modeling and crash test simulations in side impact conditions according to Euro NCAP regulations and standards. The design optimization of various components for a prototype two-seater electric vehicle, is investigated with the aid of ANSA and μETA software. Based on an existing chassis, the door components (side panels, door hinges and impact bars), are designed and evaluated targeting: 1)Construction simplicity, 2)Ease of manufacturing and placement of materials, 3)High rigidity and safety.

The materials used for almost all door components, except the impact bars, are Aluminum 6063-T6 and Steel 304 respectively. The type of analysis applied was dynamic structural and the objectives were to increase the stiffness and reduce weight by controlling the thickness of every component. Several ANSA functions and abilities where used in order to overcome geometry problems and crash test modeling parameterization and achieve short time simulations.

Simulations provide insight regarding door component deformations and stress development. Results show, that deformations in the area of the door are extensive but within the permissible material limits. Specifically, the aluminum sheets low thickness results to inner and outer door panel high deformation, while mainly the impact bars absorb the most of the crash energy. It is found that optimal design of the bars and door panels as also the selection of their thickness, control the peak tensile stresses below material's yield stress.

REFERENCES

(1) 2020 Roadmap - European New Car Assessment Program, Euro NCAP, 2015.
(2) T. Katsaros, "Design and study of electric vehicle's chassis in environment CAD/CAE", Diploma thesis, Technical University of Crete, Greece, 2013.
(3) Ali Ghadianlou, Shahrir Bin Abdullah, "Crashworthiness design of vehicle side door beams under low-speed pole side impacts",Thin-Walled Structures, Vol. 67, pp. 25-33, 2013.
(4) ANSA version 12.1.5 User's Guide, BETA CAE Systems S.A., 2008.
(5) From solid to shell, ANSA v.13.x Tutorials, BETA CAE Systems S.A.
(6) Side Impact Mobile Deformable Barrier Test Protocol – Version 7.0.1, EuroNCAP, 2015.
(7) μETA PostProcessor version 6.2.0. User's Guide, BETA CAE Systems S.A., 2008.