DESIGN AND DEVOLOPMENT OF AN AUTOMOBILE PART

TOBB
UNIVERSITY OF
ECONOMICS AND TECHNOLOGY

RENAULT

Department of Mechanical Engineering, Ankara, 2011
Senior Students : Cemre TAHMAZ, Cansu GÜRATEŞ, Taylan SUMAN, Ahmet ÇATAK
Supervisors : Assoc. Prof. Mehmet Ali Güler, Dr. Çiğdem Zeytin Derin
Co-Supervisors: Fırat Özer, Recep Muhammet Görgülüarslan

SUMMARY

In an automobile, there are lots of parts which prevent the car to take damage during a crash. The middle of the chassis of the automobile is the most important part since; the chassis protects the automobile from being crushed. In the middle of the chassis, there are three components; tunnel and two reinforcements. These reinforcements are required to improve strength of the tunnel. The tunnel is assembled with the reinforcements by spot welds. However, this method is time consuming and expensive. In this study, these three components are designed to be manufactured in one piece (by using CATIA and ANSA) for OYAK-RENAULT. As a criterion, designs compensated same geometry with baseline. In addition, new materials which provide at least the same strength are chosen. Then, static, modal and side impact tests were carried out in order to compare the final design with the baseline. Finite element models prepeared in ANSA. ANSYS was used for static and modal analysis. Side impact tests were done using LS-DYNA

PROBLEM DEFINITION

•The reinforcements are manufactured in Romania. Thus, there are some logistic path between Romania and Turkey. (indirect cost)

•In the factory, the transportation of the tunnel and the reinforcements costs too much. (indirect cost)

•To assemble tunnel and reinforcements, there are lots of spot welds which increase the manufacturing cost because of factors such as electric and labor. (direct cost)

Thus, to manufacture three components in one piece will reduce manufacturing cost of the automobile.

STATIC AND MODAL ANALYSES

Static and modal analyses are performed in ANSYS for both new designs and baseline design. Best design is selected due stress distribution observed at the end of analyses (Figure 1). For 377.42 Hz frequency maximum 69 mm displacement is observed.

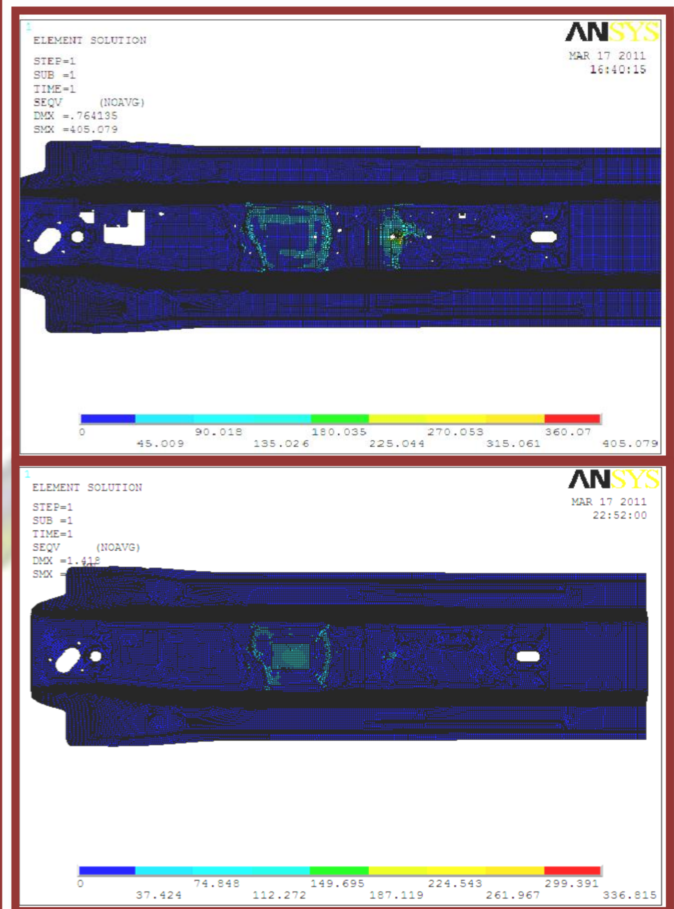

Figure 1. Comparement of The Design and Baseline, Statik Analysis

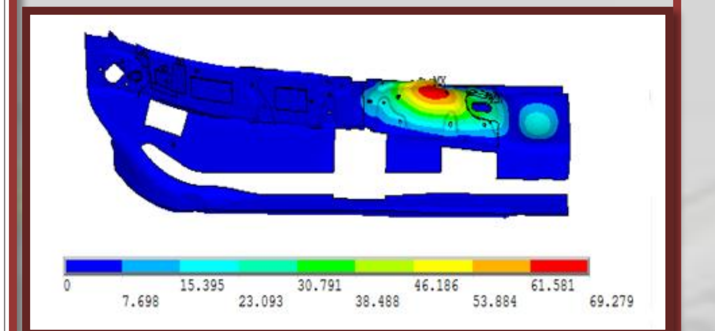

Figure 2. Modal Analyses The Design, Frequency 377.42

CRASH ANALYSES

The crash analyses are realized for the best design and baseline in accordance with ECE R95 standards (Figure 3). Maximum 500 MPa stress occurred on the design (Figure 4).

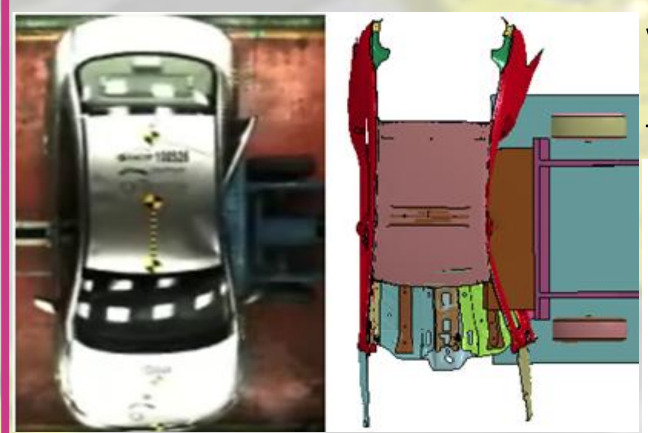

Figure 3.Comparencement of the Crash Test With Crash Simulation

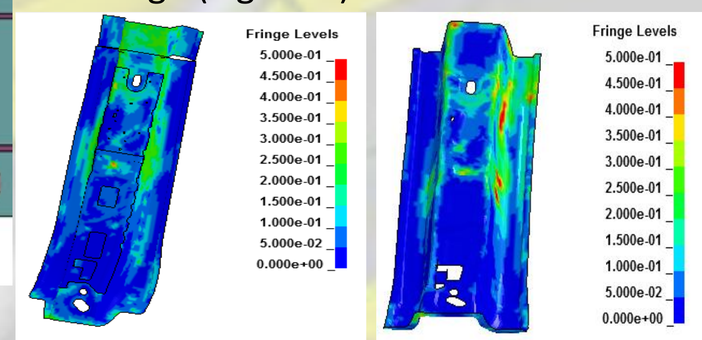

Figure 4. Crash Stress Results of the Baseline and Design

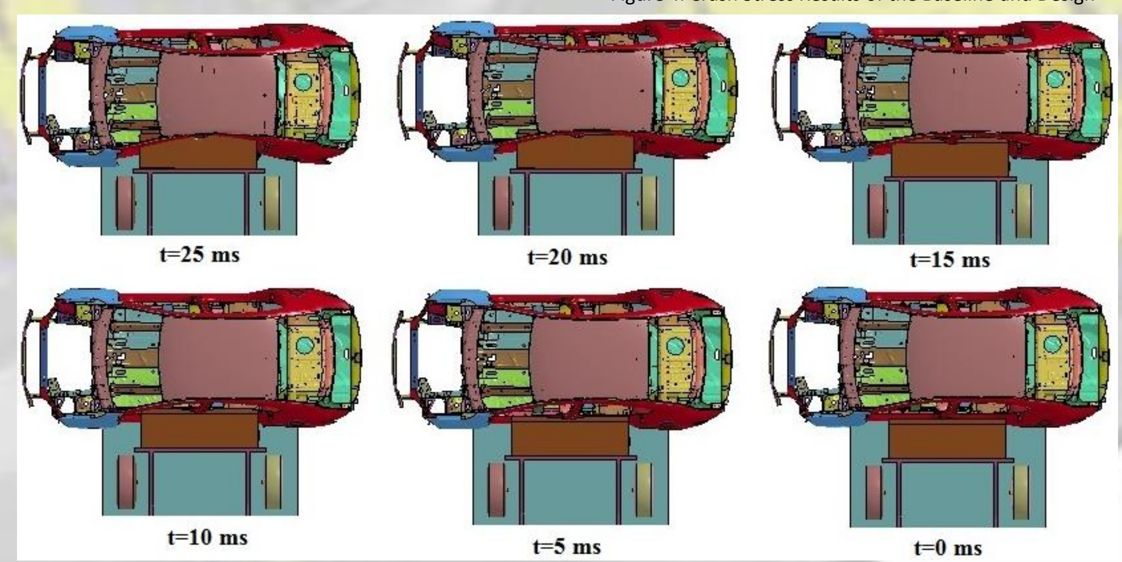

Figure 5. Side impact crash simulation results in LS-DYNA

FUTURE WORK

Study on formability of the new component with hot forming may be studied since Renault desires to invest on hot forming technology. The prototype of the new design will be manufactured either with cold or hot forming. Feasibility of new design is a new research subject. The mass production of the design would begin if all these studies were completed.

CONCLUSION

New designs should be in compliance with existing components thus; have to compensate the same geometry with the baseline design. Because of the geometry limitations, the thickness was selected 0.9 mm.

Since 405 MPa maximum stress observed on the design at the static analyses, material is selected from literature and RENAULT archives with 620 MPa yield strength for the design in order to compensate baseline strength.

At the end of crash analysis, stress results of the design and the baseline were similar. New design succeeded in the crash test. The forming simulation consists of gravity, closing, forming and springback simulation and resulted in a successful manner. No cracks or severe thinning occurred during forming simulation. From Figure 8 one may conclude best springback simulation obtained with mesh size between 0.625 mm and 1.25 mm. Forming material against the anisotropy direction may result non-uniform stress distribution (Figure 10).

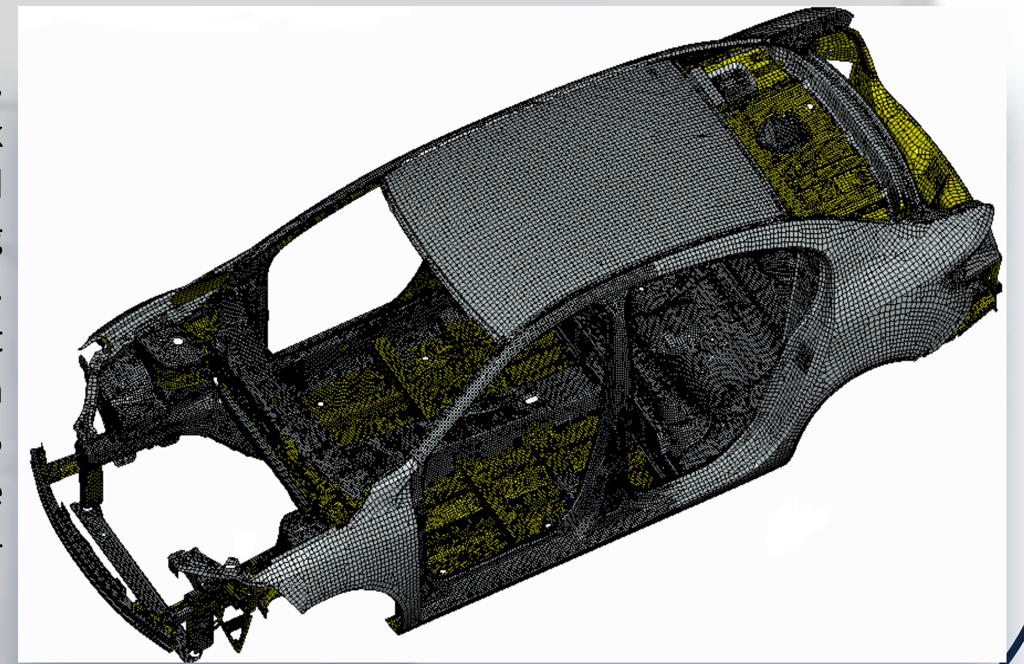

Figure 11. FEM of the automobile

FORMING SIMULATION

The tool geometries of the baseline design were generated in ANSA according to data obtained from OYAK-RENAULT (Figure 6). Finite element models of these tool components were also prepared in accordance with the forming simulation. Due to the long computation time of the real tool, It had been decided to perform the same metal forming simulation procedure on U-channel tool. Simulation parameters had been set in LS-Prepost deck. The effect of element size (Figure 8) on springback behavior and effect of rolling direction (Figure 9 and 10) on forming are investigated.

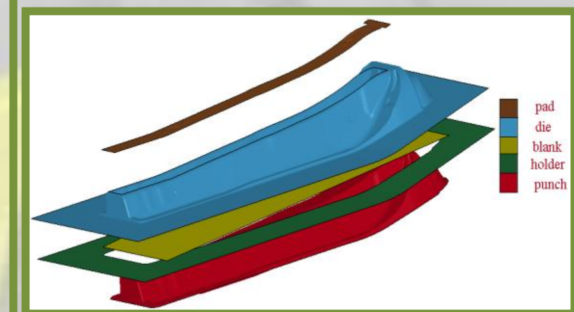

Figure 6. Tool Geometry of New Design

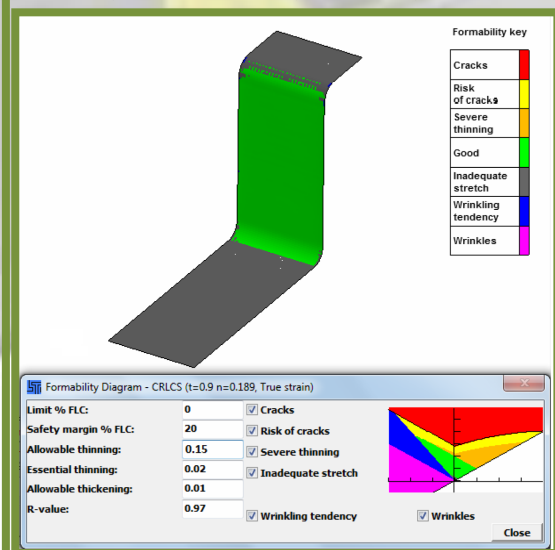

Figure 7. Formability Of New Design

Formability of U-Channel resulted in a seamless manner. (Figure 7)

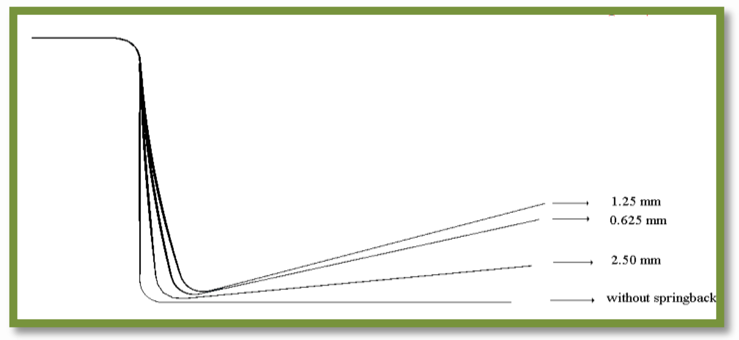

Figure 8. Dependence of Springback on Mesh Size

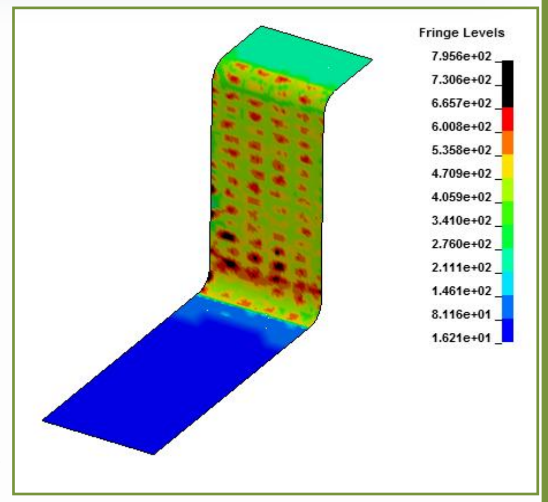

Figure 9. Anisotropy in X Direction

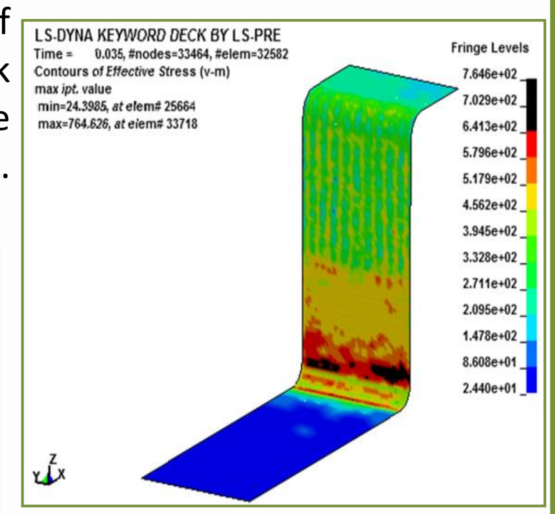

Figure 10. Anisotropy in Y Direction

Very special thanks to research Assistants Fırat Özer and Recep Muhammet Görgülüarslan for their contributions to this study and also to our parents for their presence.