Vehicle B Pillar Strength and Metal Forming Analysis Against Roof Crush Load using Finite Element Methodology

Lakshya Pangaonkar[1], Aabhas Tiwari[2], Anil Lashkari[3], Ashish Solanki[4]

[1,2,3,4]Indore Institute of Computer Aided Engineering, Indore, India

---***---

Abstract - *In United States every year, more than 250,000 rollover accidents happen, killing more than 10,000 individuals, as researched by the National Highway Traffic Safety Administration. It has been observed that rollover accidents are a worst of all other accidents. When we deeply analyzed the BIW (body in white) structure, we found that B pillar plays the most important role in preventing this type of accidents. The purpose of this paper is to increase the strength of the occupant compartment in rollover crashes with the help of B pillar and also perform the sheet metal forming simulation for the same.*

*Key Words***: Rollover, Motor Vehicle Safety, Computer Aided Engineering, Body in White**

1. INTRODUCTION

The necessities for rollover safety of an automobile are defined in Federal Motor Vehicle Safety Standard (FMVSS) 216. The motivation behind the standard is to reduce injury and wounds coming about because of the breakdown of the top into the passenger's compartment in a rollover accident. A force is executed quasi-statically to the side of the forward edge of the vehicle's top structure through a vast force plate with suitable dimension according to NHTSA (National Highway Traffic Safety Administration). Exact and productive limited component demonstrating of the top pound safety test can encourage the outline of more secure automobiles and additionally lessen advancement and testing expenses. Safety standard FMVSS 216 was made to build insurance from Rollover of a car in accidents. FMVSS No. 216 looks to diminish passing and genuine wounds coming about because of the top being smashed and pushed into the tenant compartment when the top strikes the ground amid rollover crashes.

1.1 Methodology

Firstly we modeled a finite model of our car, then we applied the boundary conditions for roof crush simulation after checking the strength of structure which we performed on the sheet metal forming simulation for B pillar structure.

1.2 Finite Element Model of a Car

The full vehicle finite element model is modeled in HYPERMESH preprocessor. The only body in white of a car has been used excluding interior and exterior trims because it has a negligible effect on the overall roof crush resistance response. In our methodology, we have used linear steel as a rigid material with card image MAT 20 in LS DYNA and properties with suitable stress-strain value has been executed for the force plate. Laminated glass has been used for the front and rear mirror and side window as well and MAT 24 for all deformable parts with suitable stress-strain values.

Table 1: Thickness of individual components

COMPONENT	THICKNESS
Pillar A	1.404 mm
Pillar B	1.198 mm
Pillar C	0.834 mm
Side rail	0.754 mm
Roof	0.750 mm

1.3 The Force Plate

A rigid plate was modeled with the dimension (1829*762) according to NHTSA and targeted over the roof is put longitudinally at an edge of 5 degrees to the level towards the front of the vehicle pivot at a detachable point of 25 degrees beneath the flat. According to IIHS (insurance institute of highway safety), the plate should bear a load of an unloaded vehicle up to 4 times. This type of plate will come under the good plate rating. Force will be applied according to Gross vehicle weight rating (GVWR) and the plate displacement should not exceed over 127mm for the safety purpose.
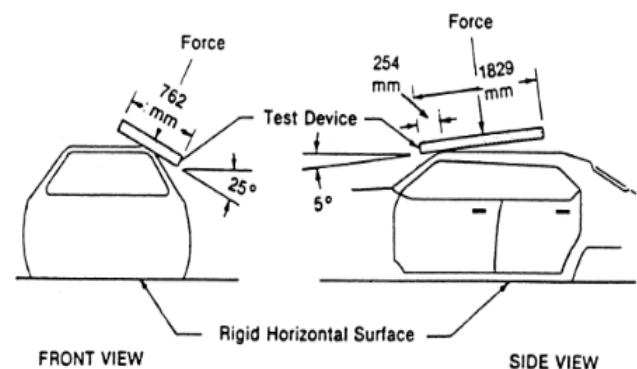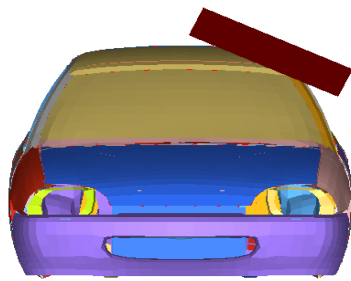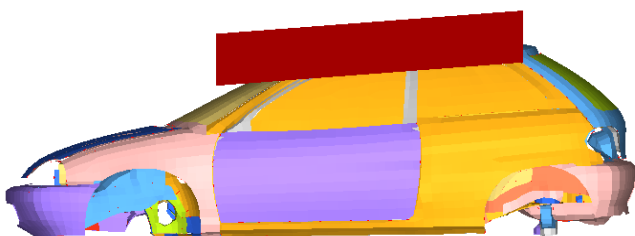

Fig.1: Force plate

1.4 Model Set Up

Placed the body frame of the vehicle on a rigid horizontal surface and fixed the vehicle rigidly in position by the use of SPC (single point constraints) and assuming all the windows are closed and all the doors are locked. Then apply force in a downward direction perpendicular to the lower surface of the test device at a rate of not more than 13 millimetres per second until reaching a force in newtons of 1\1\2 times the unloaded vehicle weight of the tested vehicle, measured in kilograms and multiplied by 9.8 or 22,240 newtons, whichever is less and then find out desired results of our simulation. Many iterations have been performed by varying the thickness of sheet metal and pillars. Then the resultant displacement has been measured over LS- DYNA PREPOST until it does not come less than 127mm which is desirable for the safety point of view.

(a)

(b)

Fig.2 (a) (b) Model Setup of force plate and car
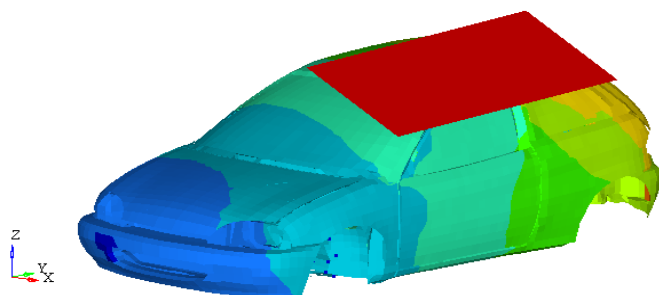
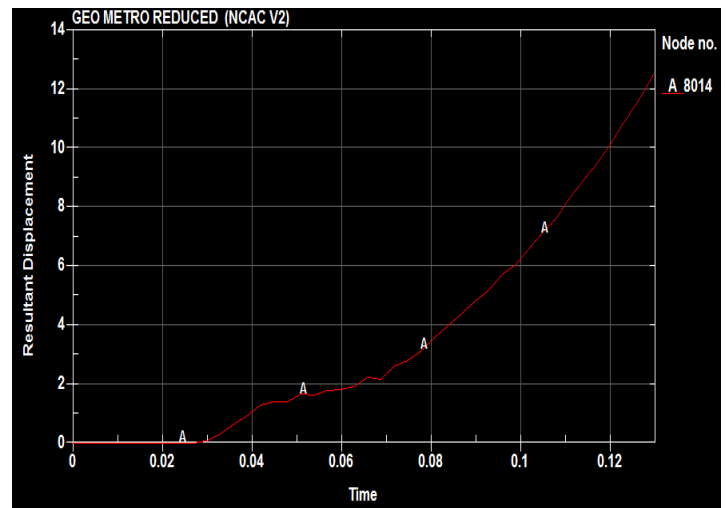## 1.5 Model Simulations

Fig.3: Rollover simulation of car

Fig.4: Resultant Displacement curve

After analyzing the result we have observed the maximum displacement on B pillar is 12.89 mm, which is below 127mm. On the other hand, in the direction of weight optimization, the first approach was to reduce the thickness of the B pillar and then perform the same test and observe the results again.

1.6 Modification

Table 2. Thickness of individual components

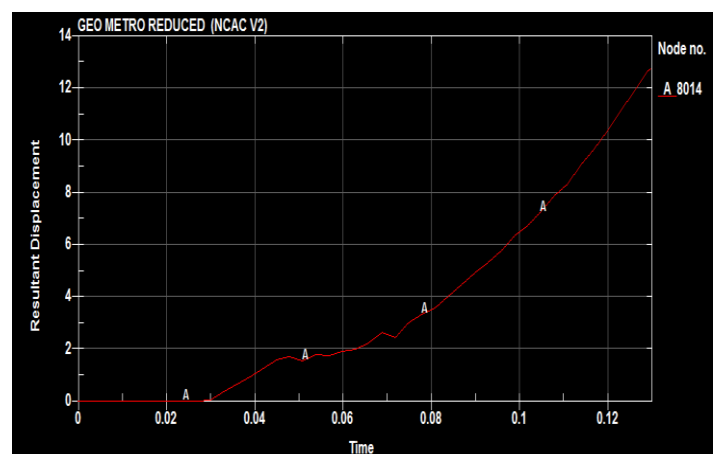| COMPONENT	THICKNESS
Pillar A	1.404 mm
Pillar B	0.800 mm
Pillar C	0.834 mm
Side rail	0.754 mm
Roof	0.750 mm

Fig.5: Resultant Displacement curve

After the second observation, we can see that there is no such variation in the results, so we can use 0.8mm thickness in place of 1.198 mm thickness for the B pillar.

1.7 METAL FORMING

After successfully analyzing the strength of structure we need to know our designs are valid in respect of manufacturing process, and for that we need to perform a metal forming analysis using HYPERWORKS tool called HYPERFORM. Majority of automobile components are made by the stamping process. In this the material is deformed plastically to take the shape of a die with the help of metal forming simulation. We can identify the problematic areas and assume the real like solutions in computers without any expensive experiments. Metal forming simulation is used at the stage of tool design. In this study, we predict the cause of failure in respect of formability and thinning.

Sheet metal forming simulation also known as forming simulation, is a specific application of non-linear finite element analysis technology that calculates the process of sheet metal stamping, predicts common defects such as splits, wrinkles, spring back and material thinning.. The technology has many benefits in the manufacturing industry, especially the automotive industry, where lead time, cost and lean manufacturing are critical to the success of a company. Sheet metal forming involves forming and cutting operations performed on metal sheets, strips, and coils. The surface area-to-volume ratio of the starting metal is relatively high. Tools include punch, die which are used to deform the sheets.

1.8 Finite Element Modelling

We will take a finite element model of B pillar from our existing car model and import this it into the HYPERFORM - Radioss_one_step tool.
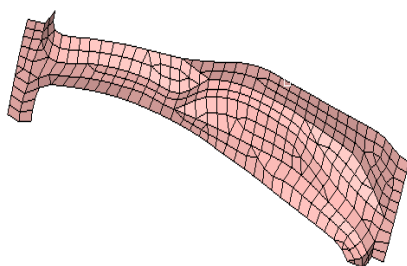

Fig.6: Finite Element Model of B pillar

1.9 Simulate the Metal Forming Process

1.10 Orient the Part

For the stamping simulation, it is assumed that the undeformed blank where the part is made lies along the XY

plane and the die travels along the negative Z axis to stamp the part into the required shape.

1.11 Material Collector
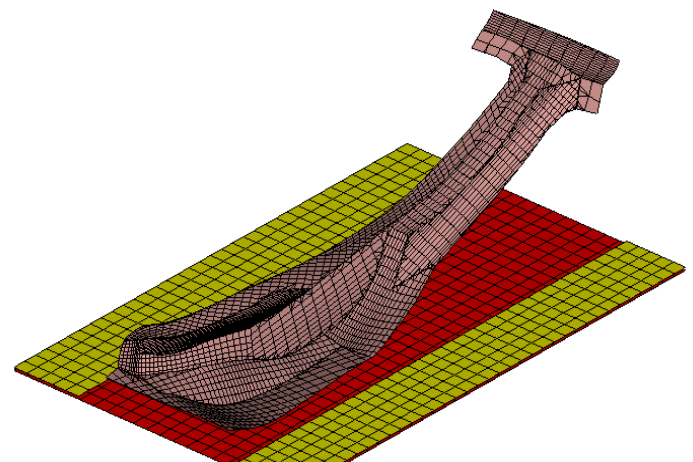

1.12 Boundary Condition

Fig.7: Boundary Condition

1.13 RESULTS

Simulation makes it possible to detect errors and problems, such as wrinkles or splits in parts, on the computer at an early stage by using two parameters called thinning and formability.

Thinning

The results show that the maximum % thinning is 23 % at the corner shown in red. This is usually the value where splits in sheet metal start to occur from too much thinning of the material. After changing the thickness (0.8 mm) we observed that the percentage thinning is 21% and we need to reduce it more in the third iteration. We changed some shape of the geometry and observed the % of thinning decreased up to 16%, which was good for us to continue.
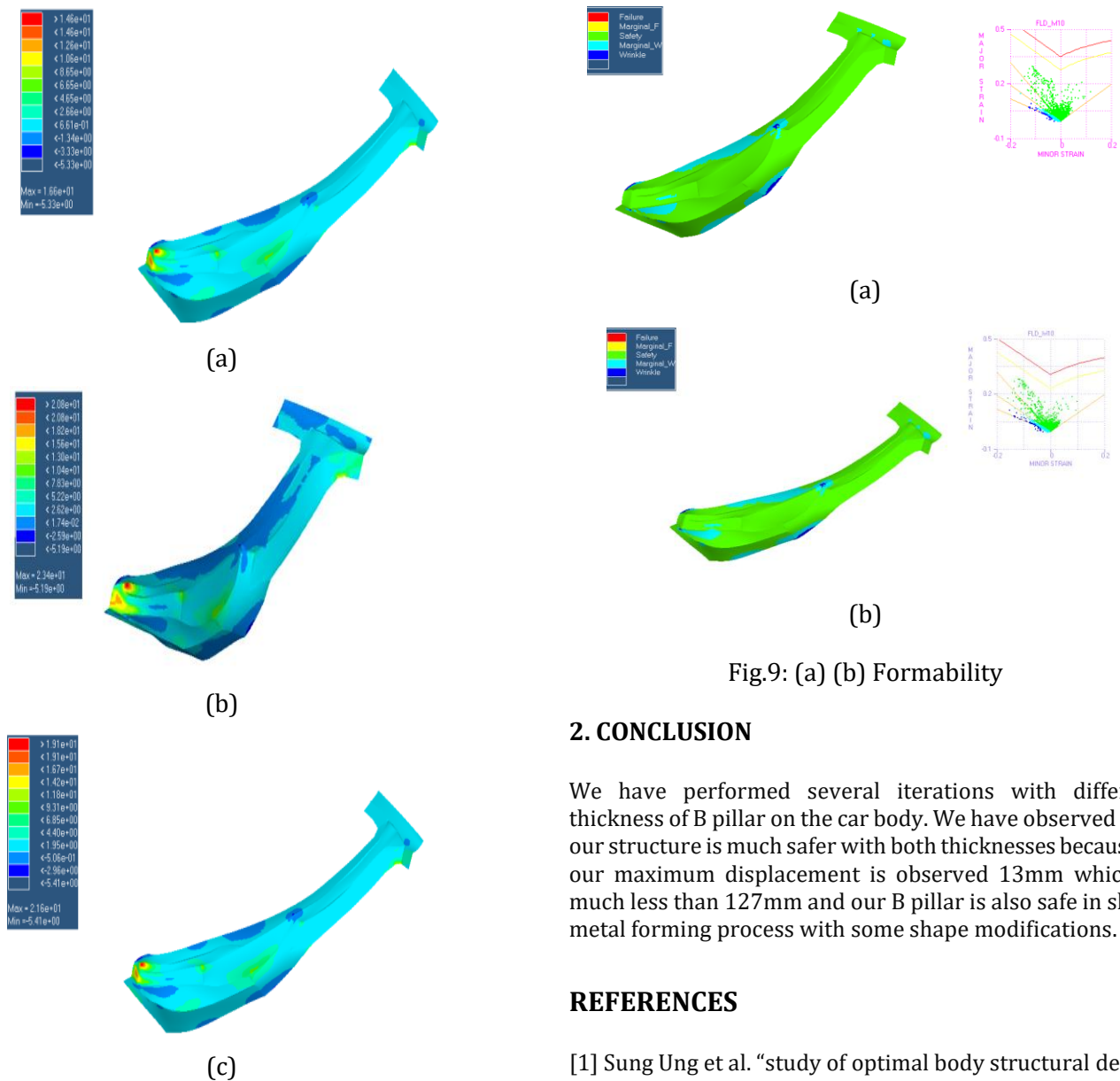
(a)

(b)

(c)

Fig.8: (a) (b) (c) Thinning percentage

Formability

Formability tells you whether or not the part that is being analyzed will have any problems during the manufacturing process. There are no red Failure zones shown on the meshed part and all of the corresponding points from each element that are plotted on the FLD curve in the upper right corner of the screen are below the yellow marginal line. This indicates that if the material used to make the part has properties that are close to the ones that were specified for the analysis, then the part should not see any splits during the manufacturing process. If any points are plotted above the yellow marginal line it indicates that splits may occur and if they fall above the red line then splits are predicted to occur and the part should be redesigned.

(a)

(b)

Fig.9: (a) (b) Formability

2. CONCLUSION

We have performed several iterations with different thickness of B pillar on the car body. We have observed that our structure is much safer with both thicknesses because of our maximum displacement is observed 13mm which is much less than 127mm and our B pillar is also safe in sheet metal forming process with some shape modifications.

REFERENCES

[1] Sung Ung et al. "study of optimal body structural design for coupe type vehicle in rollover events", paper number 13-0173

[2] Sudhir S. Shenoy (2006) "Energy absorption of a car roof reinforced with a grid stiffened composite panel in the event of a rollover", " Department of mechanical engineering and the faculty of Graduate school of Wichita state university"

[3] Allison E. Louden et al. "NHTSA research on improved restraints in rollovers", "National Highway Traffic Safety Administration, USA", paper number 11-0213.

[4] Robert Lange et al. "Rollover injury science and rollover crash typology", "24th International Technical Conference at the United States", paper number 11-0116

[5] Mingzhi Mao et al. "Vehicle roof crush modeling and validation", "5th European LS- Dyna users conference"

[6] Code of federal regulations TP-216-05 "Roof Crush Resistance" National Highway Traffic Safety Administration U.S. Department of Transportation, November 16, 2006

[7] Formability Analysis for Trapezoidal Cup Forming Using Hyper Form, HTC 2011

[8] Development of simple tutorial for metal forming using advanced computer-aided engineering tools. Raghu Echempati, Kettering University, Andy Fox, Kettering University, AC 2010-1011