ISSN 2229-5518

Analysis of automotive structural support to resist intrusion due to rollover impact

Venkateswaran M[1], Dr. Shibu G[2]

M.E Computer Aided Design, Design Division,
Department of Mechanical Engineering,
CEG Campus, Anna University, Chennai-25
Final year student[1], Senior Grade Assistant Professor[2]
jmvenkates@gmail.com

Abstract-Car safety became an issue almost immediately after the invention of the Automobile. To protect occupants from direct impact, the passenger compartment and structure should be safe at crash. Rollover crash is one of the most fatal crash scenarios that lead to death of people across the globe. Vehicle damage often includes deformation of the roof and its supporting structures. Head and neck injuries are common, and associated with roof deformation. Every year about 33,000 people lose their life. National Highway Traffic Safety Agency (NHTSA) formed a regulation to improve vehicle support structure to meet the safety of the car at rollover roof crush. This project is aimed to improve the automotive structural support to resist the intrusion due to rollover impact in order to meet the Federal Motor Vehicle Safety standards 216 (FMVSS 216).In this work, intrusion of structural support is evaluated as per FMVSS 216 standard load conditions. The structural support design will be improved through iterations to meet the upgraded regulations.

Keywords- vehicle safety, FMVSS, rollover

1. INTRODUCTIONS

The structural support is located between the front and rear doors of the vehicle. It does not only house electrical wiring and connection spots for the passenger seat belts, but it provides structural support for the vehicle in case of a side collision or rollover of the vehicle. Structural support location in vehicle model is shown in figure 3. It is called as Center pillar.

There are two panel available in that structural support. They structural inner panel and outer panel. These two panels are joined together using spot welds. Then these assembly is placed in between of body side inner panel and Center pillar interior trim. Structural inner panel is used to hold the grab handles and seat belt mountings

2. METHODOLOGY

The study starts with getting the required cad model from different responsible Design & Review engineers. Getting the thickness information about the cad model. Extracting the mid surface of the each sheet metal parts. Resolving the issues at mid surface extraction

- Identify required boundary conditions of our analysis. Gathering material properties of each cad model. Feed those information in cad mesh model and run the setup in appropriate solver tool.
- Interpret the results and find out intrusion of the structural support. Get intrusion displacement value and compare with FMVSS standard requirements.
- If the intrusion not meeting the specified FMVSS standard threshold limit. Investigate the model and improve the part to meet the requirement. Repeat the analysis to get desired value. Methodology's Flow chart is shown in figure 1.

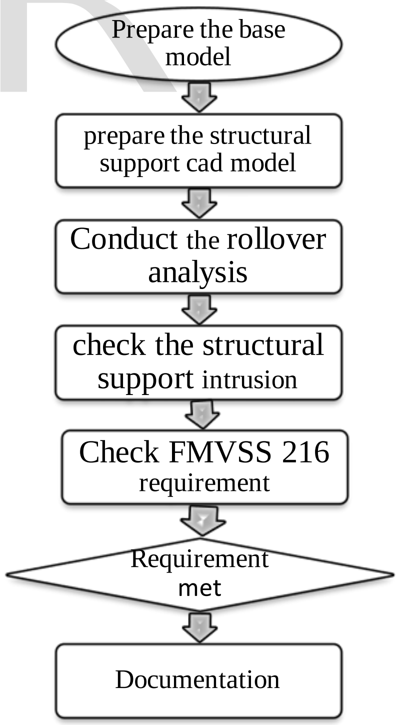


Figure 1: Methodology

ISSN 2229-5518

Our testing implementation must meet certain requirements stated by the United States Government. These standards are presented by FMVSS 216, which we found on the National Highway Traffic Safety Administration (NHTSA) website. The current standard states that the Analysis must apply a load 2.5 times the curb weight of the vehicle, with a maximum allowable Displacement of 127 mm. The platen must also be placed at a 25° roll angle, as well as a 5° pitch angle. The Visual representation of FMVSS 216 is shown in figure 2.

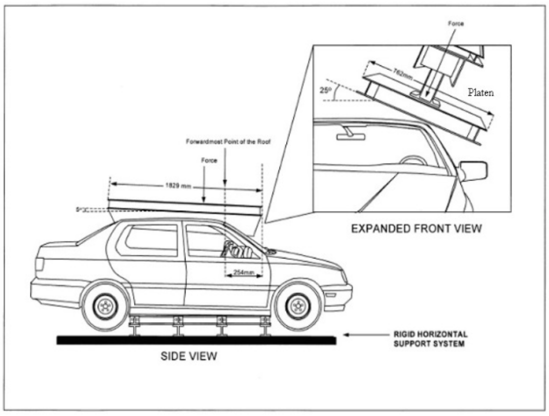


Figure 2: FMVSS 216 Vehicle Setup

4. FE MODEL

Computational vehicle models need to capture the deformation and interaction of vehicle parts and subsystems occurring during impact. The accuracy with which the crash behaviour of a vehicle is simulated depends on the quality of the computer aided design (CAD) data and its meshing. CAD geometry should be accurate in shape and size to resemble the actual vehicle. The FEM mesh should be dense enough to ensure computational convergence and to keep the computational time reasonably low. The methodology adopted for the preparation of a finite element model of Compact Sports utility model.Hypermesh is used to mesh the model. First all CAD models generated in software's like CATIA were converted into IGES format. This model is loaded in hyper mesh, Firstly mid-surfaces were extracted. Then geometry cleaning was done by using options like "geom clean-up" and "defeature" to modify the geometry data and prepare it for meshing operations. This process involved deletion of holes and curvatures of a very small radius (less than 5 mm), which have less structural significance. Figure3 shows mesh model of the vehicle.

These are required parts to resolve the intrusion issue at rollover analysis.mid-surface
Are extracted and meshing done for below listed parts. The cad model to this listed parts are

3. FMVSS REQUIREMENT

1. Structural support outer panel(center pillar)
2. Structural support inner panel
3. Body side outer panel
4. Roof panel
5. Rocker panel
6. roof rail

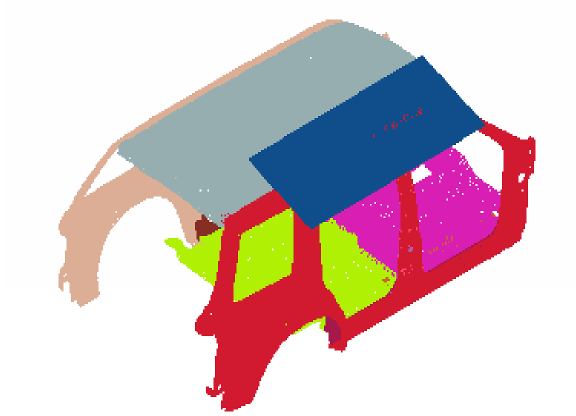


Figure 3: Vehicle Mesh Model

5. LOAD APPLIED AT SWR 2.5 (FMVSS 216-UPGRADED)

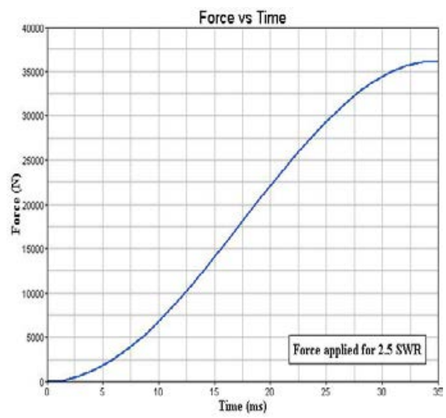
As per the upgraded FMVSS 216 standard, the vehicle should withstand 2.5 times of load of its own unloaded vehicle weight. The deformation is should not be more than 127 mm.The platen is loaded into the vehicle's roof as per Quasi static load curve shown in figure 5.

Applied load = 2.5 X UNVWX X 9.81

= 2.5 X 1475 X 9.81

= 36174.375 Kg-m/S^2

≈ 36175 N



ISSN 2229-5518

Figure 4: Load curve

The structural support is displaced to 144.715 mm which is more than the FMVSS standard (127mm). Due to this intrusion, this model got failed. So that, the structural support needs to improve to reduce the intrusion. So that, safety of the passengers increased and which helps to meet the regulations. Intrusion for structural support for 2.5 SWR is shown in figure 6.

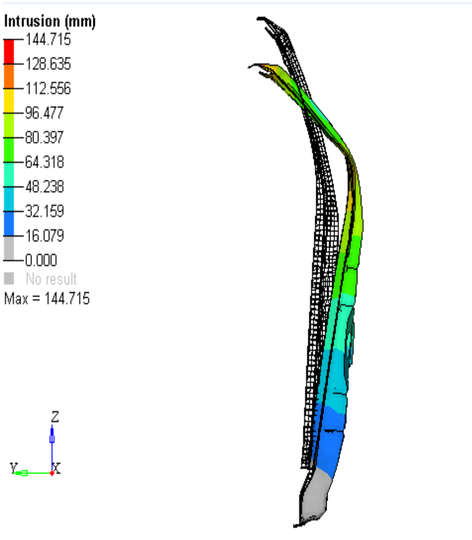


Figure 6: Structural support intrusion at base model.

6. SOLUTIONS W/O CM DESIGN

MODEL PREPARATION FOR THICKNESS ITERATION

Element penetration may occur in cases where there are thickness increment is specified part and it may penetrate with surrounding parts. We can identify and locate such Intersection errors of Geometry or FE-Model mesh (Line, Shell and Solid elements It is also possible that although there are no intersecting geometries, there is element penetration due to shell element thickness at locations where parts are in close proximity (e.g. flanges). Correction of such Thickness Penetration, for both Geometry and FE-Model mesh can be done to evaluate the analysis.

All penetrations in the vehicle FE model is identified and fixed to conduct analysis.

A. STRUCTURAL SUPPORT OUTER PANEL THICKNESS ITERATIONS

Iteration:

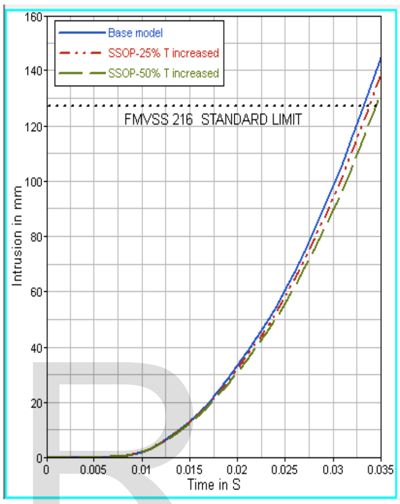
1. SSOP thickness is increased to 25% and SSIP thickness is constant as per base model
2. SSOP thickness is increased to 50% and SSIP thickness is constant as per base model

The intrusion values were shown in figure 7 and table 1

S.NO	***Thickness of SSOP (mm)***	***Thickness Increment (%)***	***Intrusion of the model (mm)***
1	0.9	25	138.52
2	0.9	50	132.05

Figure 7: Intrusion measurement for SSOP thickness increment

B. STRUCTURAL SUPPORT INNER PANEL THICKNESS ITERATIONS



Iteration:

S.NO	***Thickness of SSIP (mm)***	***Thickness Increment (%)***	***Weight of SSIP (Kg)***	***Intrusion of the model (mm)***
1	1.1	25	2.76	138.54
2	1.1	50	2.76	132.93

1. SSIP thickness is increased to 25% and SSOP thickness is constant as per base model
2. SSIP thickness is increased to 50% and SSOP thickness is constant as per base model

These iteration results were shown in figure 8.

C. STRUCTURAL SUPPORT INNER AND OUTER PANEL THICKNESS ITERATIONS

Iteration:

1. SSIP thickness is increased to 50% and SSOP thickness is increased to 25%
2. SSIP thickness is increased to 50% and SSOP thickness is increased to 50%

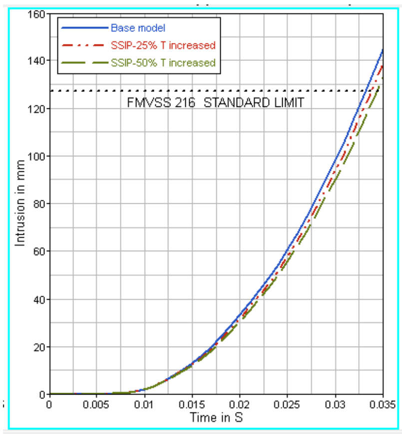


Figure 8: Intrusion measurement for SSIP thickness increment

These thickness iteration were shown in Figure 9.

S.NO	***Thickness of SSOP(SSIP) (mm)***	***Thickness Increment SSOP-(SSIP) (%)***	***Intrusion of the model (mm)***
1	0.9 (1.1)	25-(50)	126.05
2	0.9 (1.1)	50-(50)	119.21

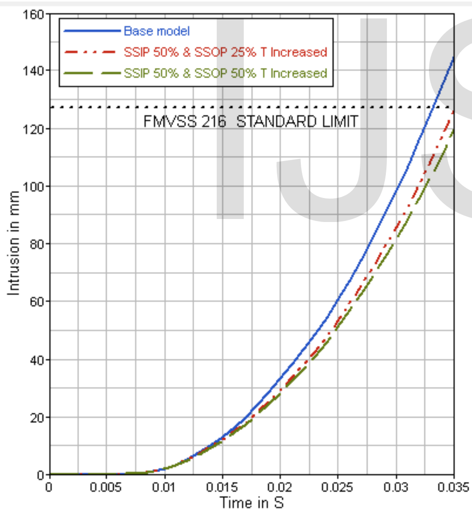


Figure 9: Intrusion measurement for SSIP/OP thickness increment

7. SOLUTION WITH CM DESIGN

As per thickness iteration studies, Structural support inner panel and structural support outer panel thickness is increased to 50% from its base model and vehicle structural intrusion is measured. Structural Support Outer Panel (SSOP) weight increased from base model is 1.78 Kg and Structural Support Inner Panel (SSIP) weight increased from base model is 1.38 Kg. Countermeasure design is the best way to reduce the intrusion compared than thickness iterations. After doing the space analysis in that support structures, Boundary of the counter measure is identified and conceptual brackets are created based on space is available. Figure 10 shows a bracket and this has been positioned in between of the structural support inner panel & outer panel. The total weight of this bracket is 1.77 kg. This bracket positioned at both sides of vehicle.so, that weight is increased in model to 3.54 kg.

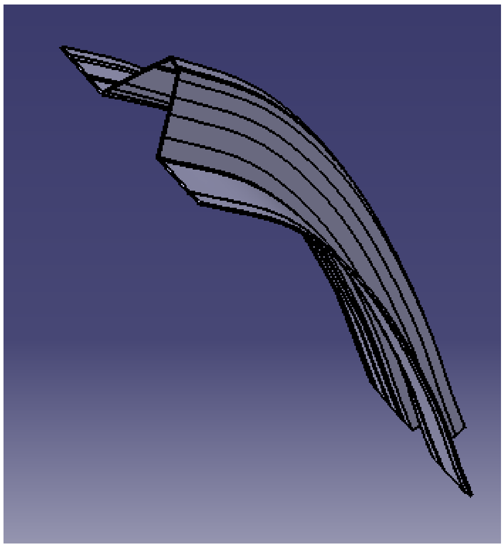


Figure 10: Counter Measure Bracket Design.

After positioned this bracket in vehicle model. Analysis can be done with this model. Figure 11 shows the intrusion is restricted to 71.84 mm.

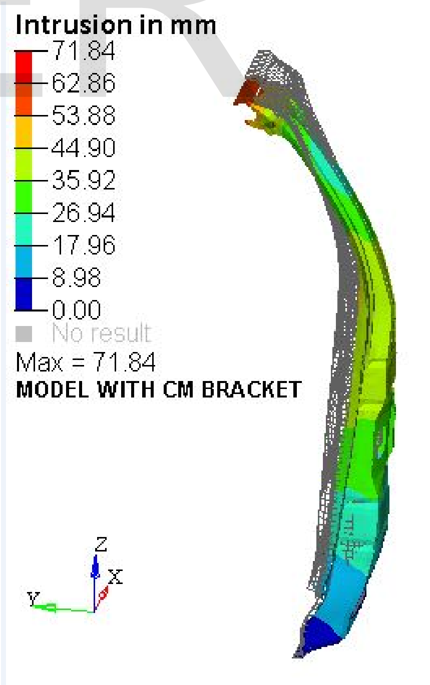


Figure 11: Structural support intrusion at model with CM design.

7. CONCLUSION

As per result comparison between old and upgraded standards. The structural support intrusion was 144.72 mm. as per Federal motor vehicle safety standards 216, the structural support intrusion should within a limit of 127 mm. due to thickness iterations on structural support outer and inner panel, This has been reduced to 119.71 mm. hence, this model has been met the FMVSS 216 standards. Structural intrusion is restricted to 71.84 mm with this CM design. The intrusion is restricted to 50.3% when compared with base model by just adding a bracket with weight of 1.77 kg at both side of vehicle.

8. REFERENCES

1. http://www.nhtsa.gov/
2. http://www.globalncap.org/u-s-ncap/
3. http://www.euroncap.com/tests.aspx
4. Pankaj S. Deshmukh, "rollover and roof crush analysis of low-floor mass transit bus", Dr. Babasaheb Ambedkar Marathwada University, 2006.
5. Fern Gatilao, Gerald Roesser and Brad Reaume, "An Overview of FMVSS 216a - Roof Crush Resistance Testing", MGA Research Corp, 2010.
6. Sameer Gupta, "evaluate structural foam alternatives in A-pillar and bumper designs", LS DYNA Conference, 2009.
7. N. Chase and R. Sidhu, "Optimization of a B-Pillar Undergoing Roof Crush Using HEEDS COMPOSE", Red Cedar Technology, Inc. 2012.
8. Practical finite element analysis, Finite to infinite, 2008.