INTERNATIONAL JOURNAL OF ADVANCE RESEARCH AND DEVELOPMENT

IJARnD

(Volume3, Issue7)

Available online at: www.ijarnd.com

Structural analysis and optimization of BIW A- pillar used in automobiles

Kishor S. Naik[1], Patil A. R.[2]

[1]Student, Sahyadri Valley College of Engineering & Technology, Rajuri, Maharashtra

[2]Professor, Sahyadri Valley College of Engineering & Technology, Rajuri, Maharashtra

ABSTRACT

Pillars are among the key components of automobiles. Pillars failure is one of the most common structural imperfection and causes overall structure failure. Since during operation Pillars are subject to various static and dynamic loads which alter the vehicle performance, however, increasing the durability of Pillar in automobiles is necessary. In this paper, a Structural Analysis is performed on Pillars of the automobile to obtain its characteristic by subjecting the Boundary conditions. The Analysis has been carried out in two phases i.e. modeling and structural analysis. Firstly Pillars of automotive are modeled with the help of CATIA and imported in ANSYS, in structural Analysis FEA of A-Pillar has been carried out with adopting different case study such as different material parameter etc. Results showed that both materials were almost identical and show good agreement with the experimental data which is stated in the literature.

Keywords— *A-pillar, Dent, Deformation, Quality feel, Composite reinforced*

1. INTRODUCTION

The A-pillar is an important load carrying component of any automobile body. It is a primary support structure for the roof and is typically a thin-walled, spot-welded, closed-section structure made from high strength steels. Automobile with major panels welded together is called BODY IN WHITE or BLUE BUCK. This consists of a number of panels; one among them is A-PILLAR. This is a structural member as the sides of the windshield on which doors will be mounted. The paper deals with the modal analysis of an A-pillar, for the given dimensions. The dimensions of the A-pillar are taken from the drawing or references whichever is available. The 3D model is prepared using the CATIAV5 software. Meshing is done in Hypermesh and structural analysis is carried out on A-pillar to determine the natural frequencies and mode shapes of a structure. Post-processing is done using ANSYS software. The A-pillar design's acceptance is done from the results obtained in analysis on composite reinforced A-pillars. Re-analysis results will show the different results from which best design is selected. It is then fabricated and validated experimentally.

2. LITERATURE SURVEY

Ying Yang et, al. Mode calculation and testing of a car body in white. In this paper, the modal analysis of a BIW is achieved both with finite element method and experimental test. The finite element model is established by considering the special characteristics of welding points because the boundary conditions will change the modes sensitively. Comparing with the calculated modes based on the FEM to those of the tested of the BIW, it is shown that the natural frequencies and vibration shapes correspond to each other. These results will provide the basis for improving and optimizing the design of a car body.

J.W.L.H. Maas, Development and validation of a vibration model for a complete vehicle, this report focuses on the development and validation of a multi-body model of a complete car. In this report, a vibration model was built for a complete vehicle. The first step towards this model is to build a model for the Body in White to validate the torsion and bending properties of the vehicle body. The model consists of three masses which are connected to each other with two degrees of freedom. This connection allows bending and torsion in the body and the stiffness and damping for these connections are assumed to be linear. In the modal analysis laboratory, the model validation is performed using measurements.

Sameer Gupta, This study examined the viability of using structural foam designs as a lightweight alternative in B-pillar and bumper designs. The B-pillar analysis is done for side impact performance Typical stamped steel structures were used as baselines and the rear bumper was analyzed for low-speed impact performance with respect to intrusion and energy absorption. Structural foam designs were evaluated using dissembles and iterated until the almost same performance was noticed. Dissembled results display the final design iteration of both the B-pillar and rear bumper achieved performance which is equal to the baseline with the benefit of reduced weight.

Wang Youmin et, al. To obtain the best injection molding process parameters and mold design for the car B-pillar skid plate, the process parameters of injection molding CAE needs to be optimized. After determining the parts quality standards, we complete the range and analyze the results. But this method can only identify the influence trend of parameters on the test results; it can't get the best process parameters. This modified simplex method is now used as optimization method by this paper. We transform the function optimization into the injection molding CAE process optimization. This paper uses the modified simplex method as optimization method, transforms the function optimization into the CAE process optimization, optimizes for the injection molding process parameters. We get the best parts injection molding process parameters:

3. OBJECTIVE

- Development of Hybrid design of Aluminum and composite material BIW A-Pillar and an iterative approach of material and topology optimization for better mechanical strength properties using FEA tools and validating through experimental results.
- To compare existing design results with optimized design results.
- To Validate FEA results through testing of A-Pillar.

4. PROBLEM STATEMENT

BIW A-pillar is a prominent part in vehicle body which contributes in car structural strength, crashworthiness, durability etc. The failure may occur if the stresses are not within a permissible limit. This may cause damage to the attaching structures like doors, interiors, etc. Therefore for safe operation, some measures will be taken to increase the strength and durability of the component. In this project, measures will be taken to increase the strength and durability of the optimized model than the existing model.

5. METHODOLOGY

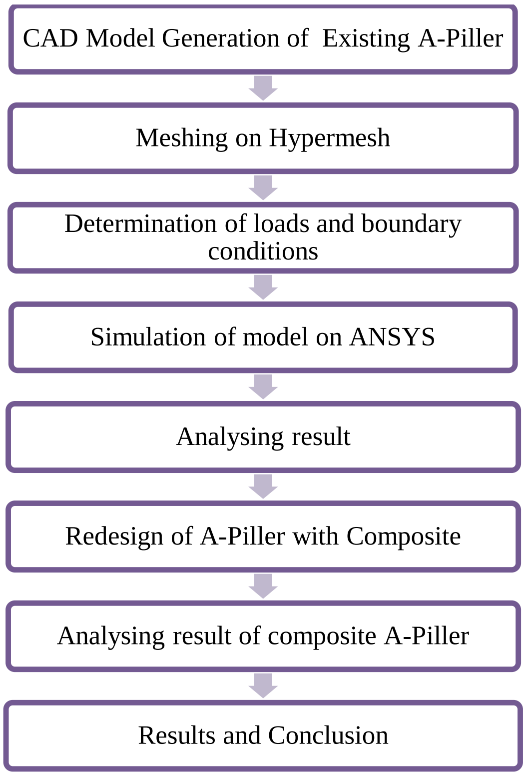

Fig. 1: Flowchart

A-Pillar of existing sedan car is designed in CATIAV5 then a meshing of the model is done on Hypermesh. Determination of loads and boundary conditions applied load on three different potions. Using ANSYS16 Simulation of a model for Dent and Quality feel at different points is done. After analyzing the results of existing A-Pillar redesigned for optimization and existing material with composite reinforcement is done on CATIAV5 then followed the same process to analyses optimized composite A-Pillar at three different points. Then the results where compared and concluded.

6. DESIGN CONSIDERATION

In modern passenger vehicles, the A-Pillar is an important structural safety component. A Pillar is carrying to a large load in order to minimize the deformation of the occupant compartment generally the larger the cross-section the more load the A-pillar can transfer. However, the A-pillars, in general, more or less reduce the forward vision angles for the driver. Therefore the width and strength of the A-Pillar are important vehicle safety parameters. The strength and size requirements on the A-Pillar are in contradiction. In an A-pillar design in which the cross-section is folded and expands when needed the conflicting requirements can be combined into one component.

7. DESIGN AND ANALYSIS

The chapter Design and Analysis of BIW A-pillar of dissertation includes design and analysis of a BIW A-pillar of a sedan car. Dimensions of the existing BIW A-pillar have been selected from references and CAD model of an AS-pillar have been prepared in CATIA V5. The finite element analysis is carried out by using Hypermesh and ANSYS as post-processor.

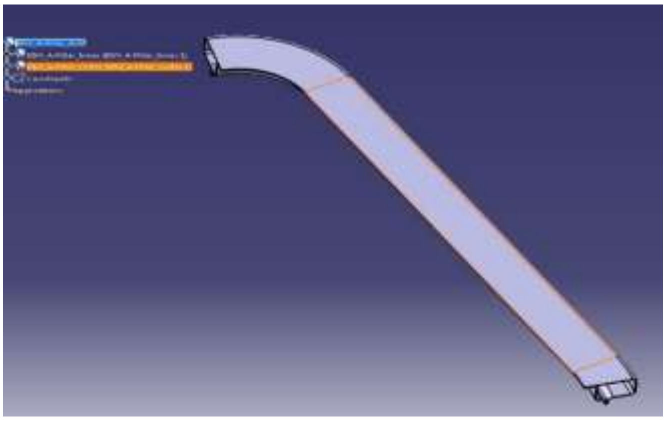

Fig. 2: BIW A-Pillar CAD model

7.1 A general procedure for finite element analysis

Certain methods in formulating a finite element analysis of a physical problem are common to all such analyses, whether structural, heat transfer, fluid flow, or any other problem. These steps are embodied in commercial finite element software packages (some are mentioned in the following paragraphs) and are implicitly incorporated in this text, although we do not necessarily refer to the steps explicitly in the following chapters. The steps are described as follows.

Preprocessing

- Define the geometric domain of the problem.
- Define the element type(s) to be used.
- Define the material properties of the elements.
- Define the element connectivity's (mesh the model).
- Define the physical constraints (boundary conditions).
- Define the loadings.
- The preprocessing (model definition) step is crucial.

Analysis: Analysis is done by selecting the appropriate solver and carrying out the operations in various stages to obtain a solution. Particularly analysis is carried out in three stages by performing various operations in software.

Meshing:

Table 1: Meshing Details

Element Type	Shell 63 (2D element)
Number of Nodes	5435
Number of Elements	5419

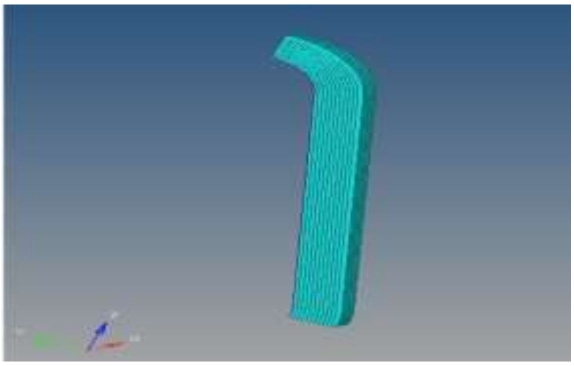

Fig. 3: Meshing of BIW A-pillar in Hypermesh (Shell 63-2D Mesh)

Initially, the IGS file is imported to the meshing software like Hypermesh. The CAD data of the BIW A-pillar is imported and the surfaces were created and meshed with shell element (Shell 63) (2D Mesh Element Type). Depending upon the dimensions of the A-pillar the mesh element size will be selected.

Table 2: Material properties Alloy Steel

Property	Value
Young's modulus, E	2.110^9 Mpa
Poisson's Ratio, ν	0.2
Density, ρ	7.9 x 10^{-9} tonne/mm^3
Yield Strength	**520MPa**

7.2 Finite Element Analysis of A- Pillar

To perform FEA of existing A-Pillar, continuum (A-Pillar) is discretized into a finite number of elements through meshing process and then boundary conditions are applied to the system. Fixed support and forces are applied shown in the figure below. A- Pillar acts mainly as a supporting member for roofing there will be no any continuous working load acts on it but the quality and strength of A-pillar matters a lot in accidents especially like roll over or crash of vehicles. Keeping this view in mind A-pillar is analyzed for sudden random dent loads and Quality feel tests at three equidistance points

Dent at point 1

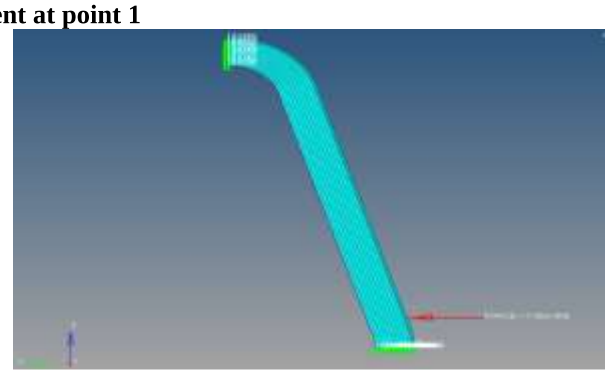

Fig. 4: Meshed model with applied boundary conditions

8. RESULT FOR STRESS AND DEFORMATION

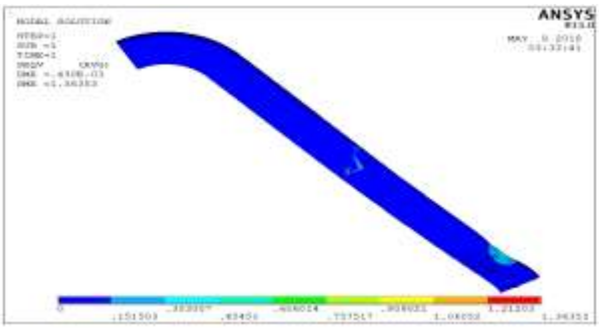

Fig 5: von-mises stress for Dent at point 1

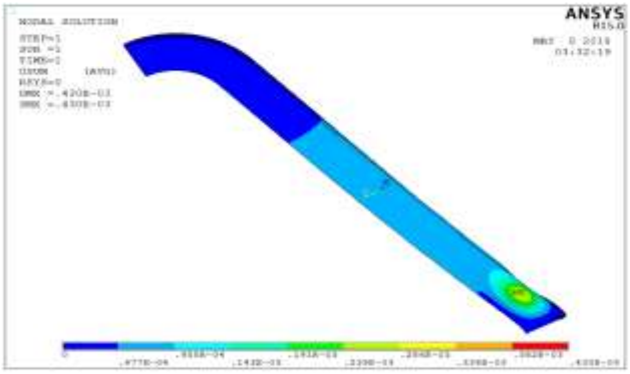

Fig 6: Displacement result for Dent at point 1

From above figure 5 & 6, it is observed that Dent at point 1 Maximum Stress of **1.36 Mpa** and it can be seen that the deformation is 0.0004 mm.

Dent at point 2

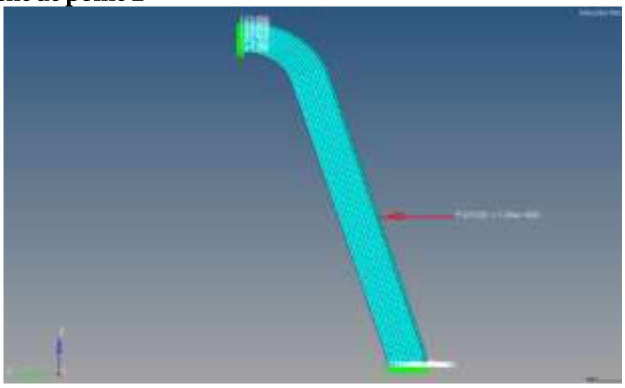

Fig. 7: Meshed model with applied boundary conditions

Result for Stress

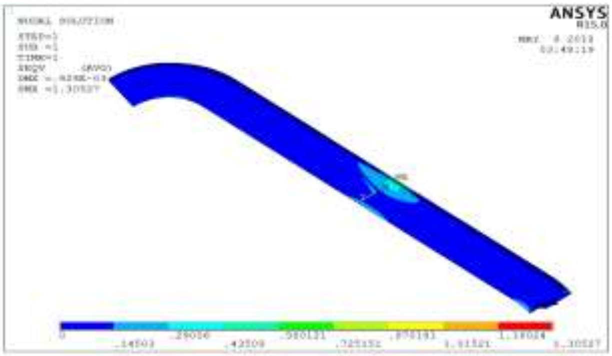

Fig. 8: von-mises stress for Dent at point 2

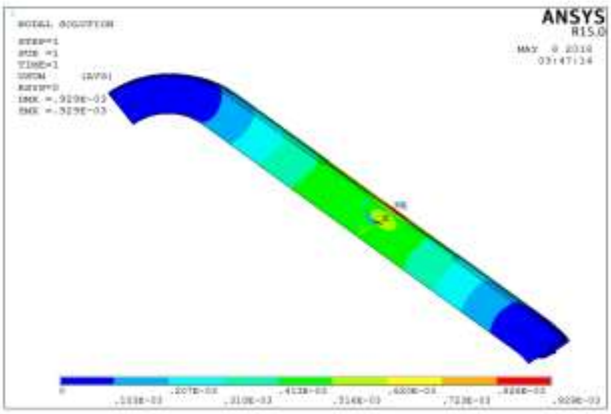

Fig. 9: Displacement result for Dent at point 2

From figure 8 and 9. It is observed that Dent at point 2 Maximum Stress of **1.30 Mpa**, and it can be seen that the deformation is 0.0009 mm.

Dent at point 3

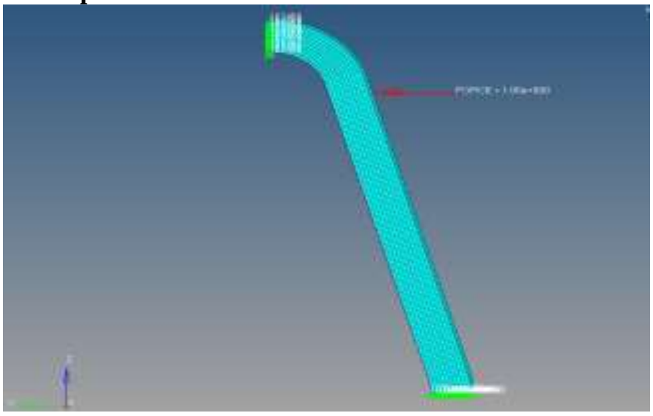

Fig. 10: Meshed model with applied boundary conditions

Result for Stress

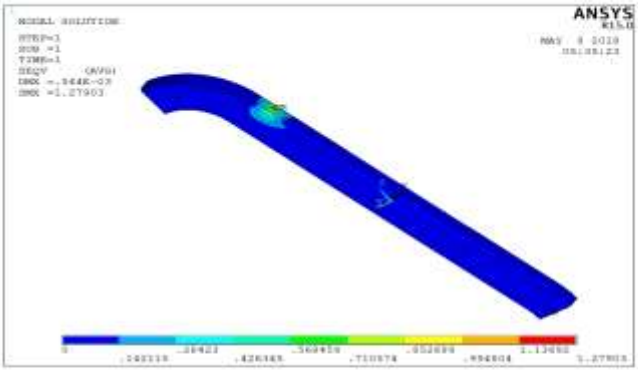

Fig. 11: von-mises stress for Dent at point 3

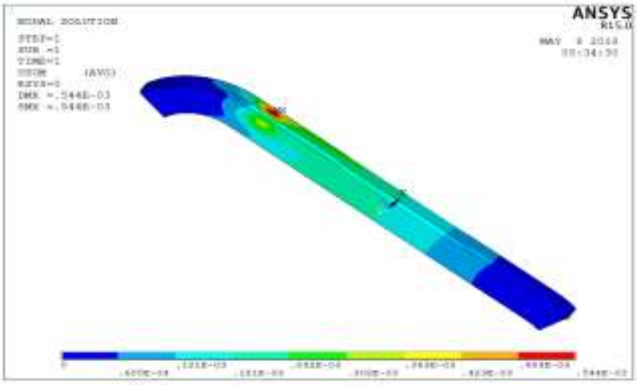

Fig. 12: Displacement result for Dent at point 3

From figure 11 & 12, it is observed that Dent at point 3 Maximum Stress of **1.27 Mpa,** and it can be seen that the deformation is 0.0005 mm,

Quality Feel at points
Result for Stress

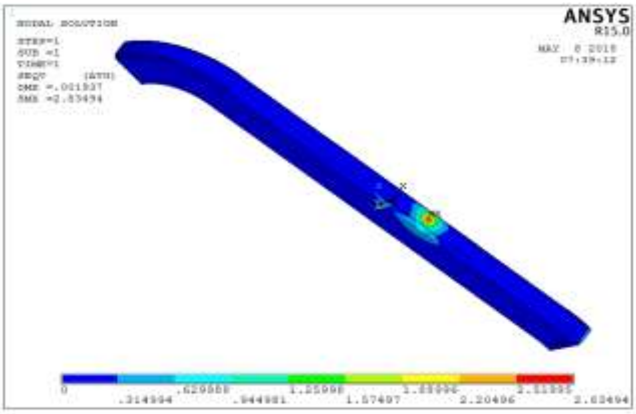

Fig. 13: von-mises stress for Quality Feel at point 2

Table 3: Quality feel at points result for stress and deformation

Quality Feel at		
Point 1	Maximum Stress	**2.78 Mpa**
	Deformation	**0.0012 mm**
Point 2	Maximum Stress	**2.83 Mpa**
	Deformation	**0.0019 mm**
Point 3	Maximum Stress	**2.78 Mpa**
	Deformation	**0.0012 mm**

By observing above results for considered dent and quality feel loading conditions existing steel A-pillar is so stronger and stiffer with a structural point so as for the scope of research we can go for material optimization by replacing existing steel material with lightweight and structurally stable composite materials.

9. OPTIMIZATION OF BIW A- PILLAR

To have lighter, less cost and may have better strength too Optimization methods were developed. Many optimization types, methods, software technique, and tools are available due to the revolution of the high-speed computing and software development. There are four disciplines for the optimization process.

a. **Topology optimization:** During the optimization process, the optimum material layout is set according to the design space and loading case.
b. **Shape optimization:** this optimization gives the optimum fillets and the optimum outer dimensions.
c. **Size optimization:** the goal of applying this optimization process is to reach the optimum thickness of the component.
d. **Topography:** it is an advanced form of shape optimization, in which a design region is defined and a pattern of shape variable will generate the reinforcement.

Weight reduction is done using reinforcement optimization technique. The weight reduction is done by reducing the thickness of existing A-pillar and replacing reduced thickness with composite glass fiber through topology optimization by meeting the strength, safety factor targets and the corresponding weight reduction is analyzed.

9.1 Finite element analysis of composite reinforced A-Pillar (Steel reinforced with glass fiber)

Here regular 1.5mm thick walled steel A-pillar is made to 0.75 mm and the remaining 0.75 mm is reinforced with a composite material (Glass fiber) as part of R & D keeping view for a better life, functionality, weight reduction, increased strength and load carrying capacity. Considering same boundary conditions as above.

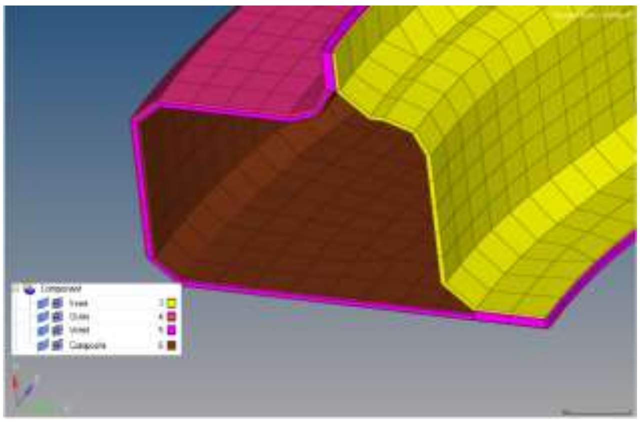

Fig. 14: Material allocation

Dent at point 1 result for Stress

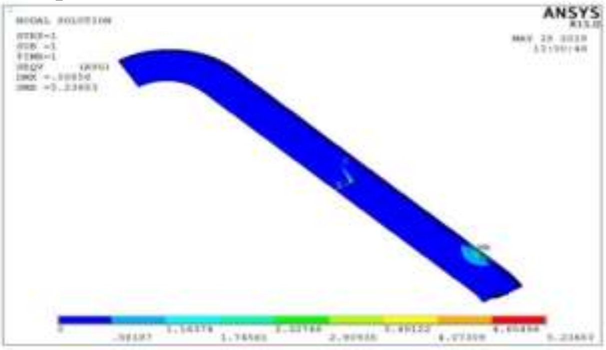

Fig. 15: von-mises stress for composite reinforced steel A-Pillar Dent at point 1

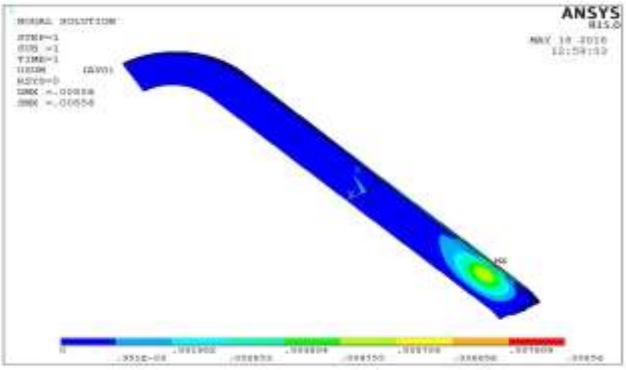

Fig. 16: Displacement result for composite reinforced steel A-Pillar dent at point 1

From figure 15 and 16, Maximum Stress at point 1 of **5.236 Mpa** is observed and it can be seen that the deformation is 0.0004 mm.

Quality Feel at point 1 Result for Stress

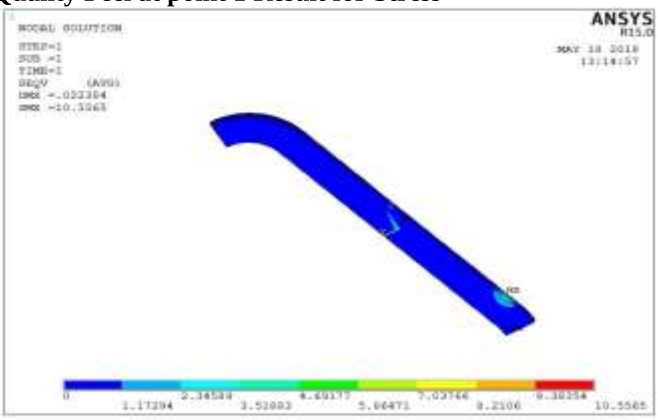

Fig. 17: von-mises stress for composite reinforced steel A-Pillar Quality Feel at point 1

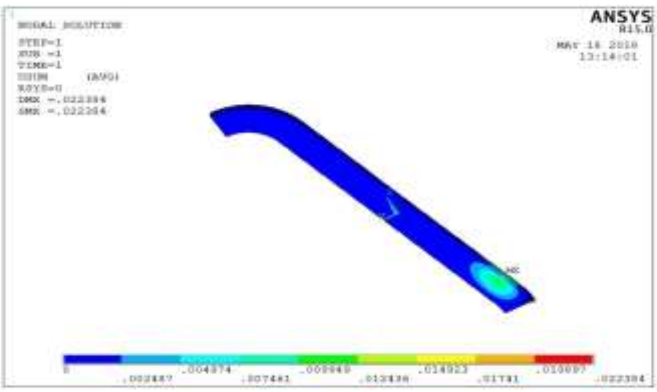

Fig. 18: Displacement result for composite reinforced steel A-Pillar Quality Feel at point 1

From figure 17 and 18, it is observed that Quality Feel Maximum Stress of **10.556 Mpa** and it can be seen that the deformation is 0.0223 mm,

Table 4: Comparison for existing and optimized A pillar for dent

Type	Dent					
	Position 1		Position 2		Position 1	
	Stress	Deformation	Stress	Deformation	Stress	Deformation
Existing	1.36 Mpa	0.0004 mm	1.30 Mpa	0.0009 mm	1.27 Mpa	0.0005 mm
Composite	5.23 Mpa	0.0004 mm	4.93 Mpa	0.0116 mm	5.27 Mpa	0.0096 mm

Table 5: Comparison for existing and optimized A-pillar for Quality feel

Type	Quality feel					
	Position 1		Position 2		Position 3	
	Stress	Deformation	Stress	Deformation	Stress	Deformation
Existing	2.78 Mpa	0.0012 mm	2.83 Mpa	0.0019 mm	2.78 Mpa	0.0012 mm
Composite	10.56 Mpa	0.0223 mm	10.79 Mpa	0.0285 mm	10.71 Mpa	0.0243 mm

Table 6: Reduction in weight

Type	Weight
Existing	1.548 Kg
Optimized	1.0385 Kg(32.91% Weight reduction)

10. CONCLUSION

By table 4, 5 and 6 observing results for considered dent and quality feel loading conditions existing steel A-pillar is so stronger and stiffer. And optimized composite A-pillar is lightweight and structurally stable. From the table it is clear that though stress and deformation in the optimized model are high, all the values are well within the limit hence design is highly safe. And there is a weight reduction of 32.91 % due to this efficiency get increased without compromising its strength.

- Maximum Stress of Dent and Quality Feel at various point increased and is within the limit
- Deformation at three different points of Dent and Quality Feel slightly increase.
- Composite Reinforced A-Pillar is light in weight and meets BIW.
- The strength of both materials high.

11. REFERENCES

[1] J.W.L.H. Maas, "Development and validation of a vibration model for a complete vehicle", DCT 2007.131.

[2] Ying Yang, Guangyao Zhao, Dongbo Ma and Xiaobin Xu, "Mode calculation and testing of a car body in white"

[3] Sameer Gupta, "USING CAE TO EVALUATE STRUCTURAL FOAM ALTERNATIVES IN B-PILLAR AND BUMPER DESIGNS".

[4] Wang Youmin, Yu Chun, Chen Mingxin, Zhou Ge Dou Zongying, Yang Xaoli, Du Xiaoyang, "CAR B-PILLAR SKID PLATE MOLDING PARAMETER OPTIMIZATION", Journal of Theoretical and Applied Information Technology 20th January 2013. Vol. 47 No.2, ISSN: 1992-8645.

[5] Luo Yun, Feng Guoying, Du Yongzhao, Zhou Shouhuan " The Dynamic Response Analysis of Auto Body Sheets to Node Loads", Int. Journal of Engineering Research and

Applications, ISSN: 2248-9622, Vol. 4, Issue 1(Version 1), January 2014, pp.201-205.

[6] Mr. G. R. Nikhade, "Modal Analysis of Body in White", International Journal of Innovative Research in Science & Engineering, ISSN (Online) 2347-3207.

[7] Sameer Gupta, "USING CAE TO EVALUATE A STRUCTURAL FOAM DESIGN FOR INCREASING ROOF STRENGTH", 8th European LS-DYNA Users Conference, Strasbourg - May 2011.