Topology Optimization of Driver Cabin Mounting Bracket of Heavy Commercial Vehicle

[1]Vijay Kalantre, [2]K. H. Munde, [3]Ashish Pawar

[1]PG scholar, [2,3]Assistant Professor
Mechanical Engineering Department,
APCOER, Savitribai Phule Pune University, Pune

***Abstract*—Today automobile sector is one of the largest growing technological fields and which is continuously striving for weight reduction of vehicles as the today's major need of fuel economy and emission reduction demands it. To reduce weight engineers have either to search for better and better materials or to do the optimization. Out of various structural optimization techniques like size, shape & topology optimization, application of Topology Optimization (TO) in automotive structure design is overviewed in this paper and using Evolutionary Structural Optimization (ESO) method driver cabin mounting bracket of a heavy commercial vehicle is optimized here. With the objective of mass reduction and compliance minimization topology optimization is performed using Ansys tool. Various topologies were studied and compared for static structural, fatigue safety factor and finding out minimum natural frequency i.e. modal analysis and an optimized topology was obtained with predefined level of compromise in constraint parameter like strength, minimum natural frequency and fatigue safety factor. Ansys results of static structural and modal analysis will be validated experimentally. 8.164 % of mass reduction is obtained with little compromise in constraint parameter. Fatigue results can be experimentally validated using component level testing or onsite testing as future scope. Also this technique can be used for other automotive structural components to reduce the overall weight of the vehicle.**

***Index Terms*—Ansys 16.0 & 18.0, BESO, Evolutionary Structural Optimization (ESO), Shape Optimization, Size Optimization, Topology Optimization (TO)**

I. INTRODUCTION

S the aim of weight reduction without compromise in strength can also be fulfilled by the optimization technology (along with Amaterial variation), structural optimization plays vital role in it. There are about three important methods of structural optimization; Sizing, Shape and Topology Optimization (TO). At various design stage these can be used separately or in combination to optimize the structural component. Sizing optimization keeps the original shape of the component while it changes the size of it as per the space constraints available. Shape Optimization has the freedom in the shape alteration but in the given size only. Whereas, Topology Optimization (TO) is a scientific method

of finding best material layout in the given set of constraints. It changes the density of the structure (not the material density) and reduces the unwanted material from the structure for specific boundary and load conditions. This is one of the software based optimization technique. Various software packages like Ansys 18.0 and above, Altair Optistruct etc are available today for the assistance.

Driver Cabin Mounting Bracket of Heavy Commercial vehicle (one of the chassis frame mount brackets) is taken here to optimize it for weight reduction. Structural topology optimization technique is used here to optimize it. Most effective load path has been found out using Ansys 16.0 and on the basis of it the structure is optimized. As Ansys has newly introduced module of Topology Optimization in workbench version 18.0 and above, it is also used in assistance with the typical ESO method. In the Evolutionary Structural Optimization (ESO) method stress analysis is carried out on software and least stressed and maximum stressed elements in the structure are found out. Then low stressed elements are gradually removed from structure to obtain required topology. This method assumes that the element with very low stress has vey less contribution in the handling of applied load. Simply that area has excess material that can be removed. ESO method was firstly introduced by Xie and Steven (1993). This bracket is optimized by the same method.

Model has only static loads on it, it is first statically analyzed and then as dynamic considerations minimum natural frequency analysis is also done to check that it should not coincide with external excitation frequency (here it is road excitations which has range of 0-20 Hz). Otherwise resonance will create leading to high amplitude vibrations and excessively high stresses.

Final Boundary condition is logically selected amongst various methods & on its basis; Optimized topology is obtained after many trials and errors. After modification in original bracket, based on software optimized profile, experimental results will be used to validate the software results and also to validate the assumed case of boundary condition.

II. LITERATURE REVIEW

Many research scholars have studied and proposed various methods of Topology Optimization for optimizing different structural components of automobile since longer time. They have found topology as very effective and powerful tool for structural optimization. The primary purpose of many experiments is found to be weight reduction.

Mayur Jagatap and Ashvin Dhoke, two CAE engineers from TechMahindra have used Altair Optistruct as tool for design and optimize cast iron Exhaust mounting bracket. Topologically Optimized design was finalized based on manufacturing feasibility and other practical constraint. They have achieved 45% mass reduction and 50% of design cycle time and without compromising in strength and fatigue life criteria. In future they are going to consider shape optimization for design. [1]

Y. S. Kong, S. Abdullah, M. Z. Omar and S. M. Harisin their paper published in LAJSS (2016), have optimized Automotive Spring Lower Seat using topological and topographical techniques. In their work 36.5% mass reduction and 27% compliance increase was achieved. [2]

Subhash Sudalaimuthu, Barry Lin, Mohd. Sithik and Rajiv Rajendramin their SAE International Paper (2016) have explained process of designing lightweight track bar bracket right from the scratch. Design of Experiments (DOE) and topology optimization is used to decide bolt locations and critical load path and followed shape optimization to finalize the shape. [3]

Suresh Kumar Kandreegula, Naveen Sukumar, Sunil Endugu and Umashanker Gupta published a SAE International paper in 2015 in which they have provided a forum to present new developments in structural Non-linear topology optimization. By this method structural optimization on irregular design domains can be carried out easily. Transmission Housing has been optimized using Non-linear Topology Optimization technique with the help of Simulation tool Altair OptiStruct& verified experimentally. They achieved cost reduction without sacrificing performance & safety. [4]

Guan Zhou, Guangyao Li, Aiguo Cheng, and Guochun Wang,Hongmin Zhang and Yi Liao (2015 SAE Paper) have done topology optimization on Auto Body for light weighting. They found weak part in BIW (Body in White) by applying Topology optimization and then performed sensitivity analysis to optimize thickness and significant weight reduction was achieved. Density method of Topology Optimization is used in this for Optimization. [5]

In another SAEresearch article (2015), **Bo Tan, Yu Yang, Jun Huang, Wenhui Liu, and Dongqing Zhanghave** have done structural optimization of Heavy Truck Propeller Shaft Bracket. Effect of bracket structure mode on the frequency response and stress on it are studied. In this they combine finite element method and the multi-body dynamics technology to present NVH vibration improvement of heavy truck drive shaft system. Topology optimization technology provides support to the structure improvement. [6]

Guangiyo Li, Xiaudong Xu and colleagues have topologically optimized an Automotive Tailor-Welded Blank(TWB) Door, tells their ASME paper in 2015. Bidirectional Evolutionary Optimization Method (BESO) is extended here to optimize TWB Door with multiple thicknesses then proposed optimization method for TWBs. This method can provide guide for light weight design for other automotive TWB components. [7]

BGN Satya Prasad and M Anil Kumar managers from Hyundai Motor India Engineering presented a paper in Altair Technology Conference 2013 India regarding Topology Optimization of Alloy Wheel. They used the technique of topology to design a lightweight Aluminum wheel using Hypermesh and Optistruct. Mass reduction of 340 gm per wheel is achieved by them. [8]

Parag Nemichand Jain and Satish Pavuluri from Ashok Leyland, Ltd. in 2013 published their work in SAE journal about Experimental and Finite Elemental Analysis of Bogie Suspension Mounting Brackets. This analysis helped to create a methodology to analyze bogie suspension brackets. [9]

Brake Actuator Mounting Bracket was optimized in 2010 by **Vasudev Rao S. and Chetan Raval** from Mahindra Engineering Services. This shows their work in HTC. Altair HyperWorksOptistruct was their optimization tool. Objective was to minimize total static deflection of bracket. They achieved it within reduced time. [10]

Some literatures have reviewed various applications of topology optimization in automotive applications [11] as well as use topology, shape & size optimization at various stages of design is also described [14].

Tool of topology optimization is mainly used for mass reduction in many structural applications like Engine Mounting Bracket, Transmission Housing Bracket, Cabin Suspension Bracket, Air filter bracket, Steering Column Bracket, tooled transmission mount, and jounce bump bracket. [13], [15], [16].

Topology Optimization is becoming more important in structural design which also can solve multiple loading condition problems. Basic formulation of TO problem can be found in SAE paper.

Main Key Highlights from the literature survey are as follows:

- Main purpose of most of the researchers was the weight reduction in individual component.
- Along with weight reduction compliance minimization (i.e. stiffness increase) and natural frequency maximization was also the important considerations.
- Shape, Size and topology optimizations are used in combinations by many researchers to get most optimized structure.
- Density method and ESO methods are more often used for the optimization.
- Altair Optistruct as most powerful Software package is used.
- Presently Ansys 18.0 and above versions are containing Topology Optimization Module separately which is used in this present work.

III. Bracket Mounting and Loading, Material details of Bracket

A. Onsite Mounting of the Bracket

Following images are showing the actual bracket and it's mounting on the vehicle. Also developed CAD model is also shown in the figure which is a weldment prepared in UG-NX 10 as the original bracket is as welded structure.

Fig. 1.Onsite Mounting of the Bracket

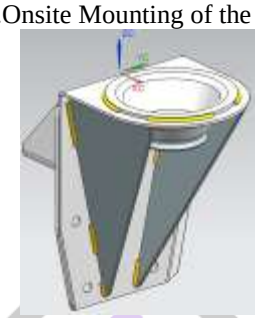


Fig. 2.CAD Model of Bracket

B. Load Calculations:

Weight of the driver cabin for selected vehicle does not exceed 400 kg as it is only a sheet metal body. And with occupant & other material we can consider 600 kg extra. Therefore, total static weight is 400+600=**1000 kg** max.

Cabin is supported on three points where two similar brackets under study are used on either side.

Then, Force Coming on 1 Bracket will be=1000/3=333.33 kg=333.33x9.81=**3270 N** approximately.

This force will be applied on upper open surface area of the conical cup vertically downwards.

C. Material Details:

Material assigned for bracket is Structural Steel which has following properties;

- Yield Strength Syt = 250 MPa
- Ultimate Tensile Strength Sut = 460 MPa
- Young's Modulus, E =210 GPa
- Poisson's ratio, ν = 0.3

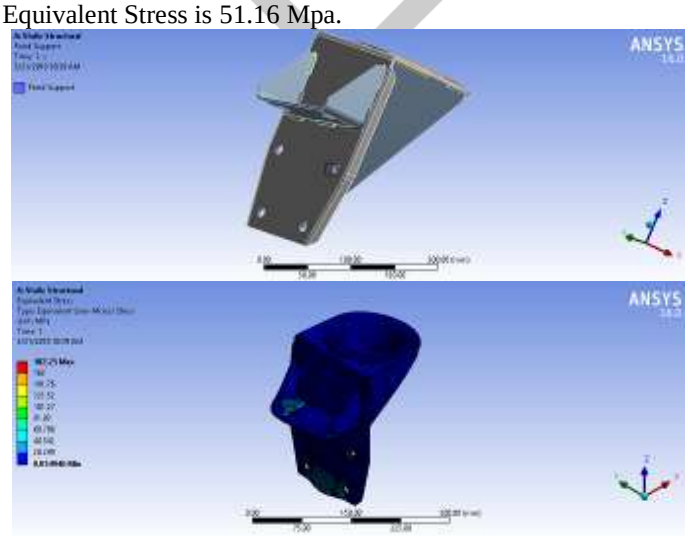
IV. Finite Element Analysis

Finite Element Analysis (FEA) is carried out using Ansys 16.0 and Ansys 18.0 Academics. Basic stress analysis and modal analysis is performed in 16.0 and 18.0 is used as assistance for the Topology Optimization.

For deciding the valid method of giving boundary constraints, following are the different boundary conditions and simulations with same mesh settings:

A. Inner surface of Bolt Clamping Hole is directly clamped; Bolts are not modeled:

Safety Factor is 1.317 & Max Equivalent Stress is 51.16 Mpa.



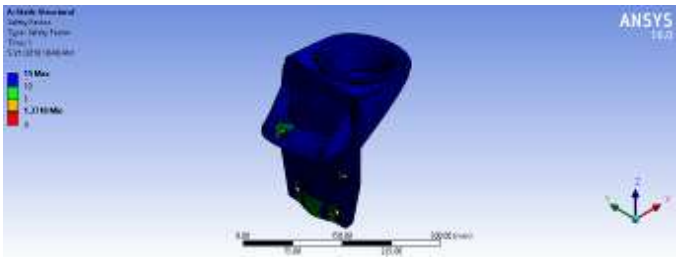


Fig. 3.Boundary Condition, Max. Equivalent Stress and Safety Factor of Case A

B. Contact surfaces of Bracket in contact with chassis frame are directly clamped; Bolts are not modeled:

Safety Factor is 4.886 & Max Equivalent Stress is 51.16 Mpa in this case.

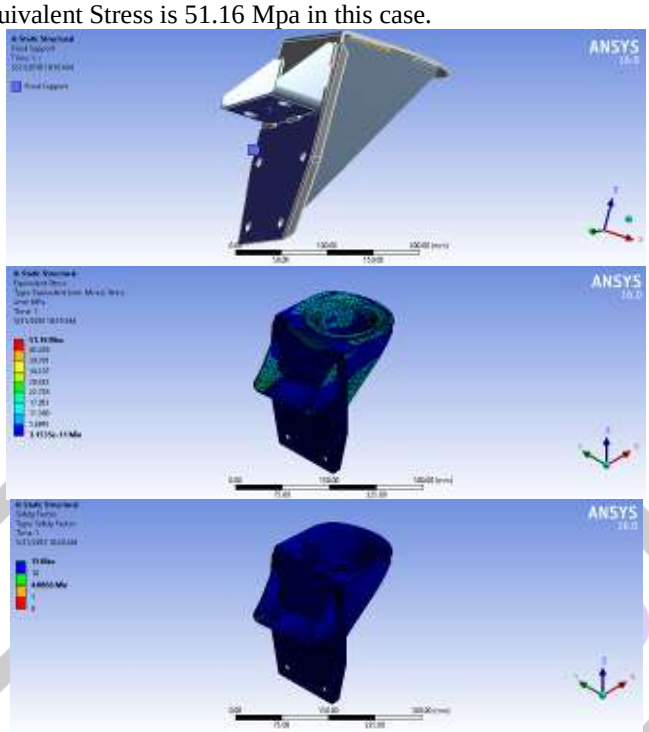


Fig. 4.Boundary Condition, Max. Equivalent Stress and Safety Factor of Case B

C. Bolts are modeled and displacement of contacting surfaces of bracket which are in contact with chassis frame member is made zero in the direction perpendicular to the surface of contact:

Safety Factor is 4.715 & Max Equivalent Stress is 53.02 Mpa in this case.

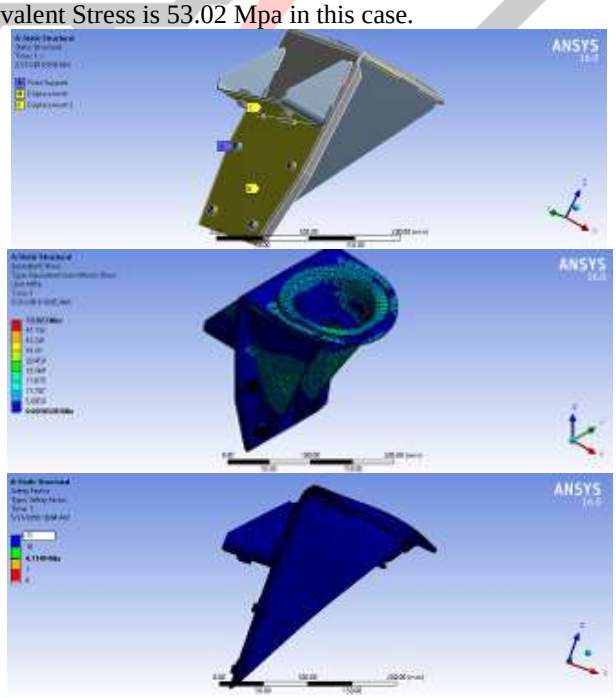


Fig. 5.Boundary Condition, Max. Equivalent Stress and Safety Factor of Case C

D. As shown in figure, on the edges C and D zero displacement is applied in the directions X & Z respectively and free in other two directions:

Safety Factor is 0.904 & Max Equivalent Stress is 276.28 Mpa in this case D. This value of safety factor is not acceptable as per design rules.

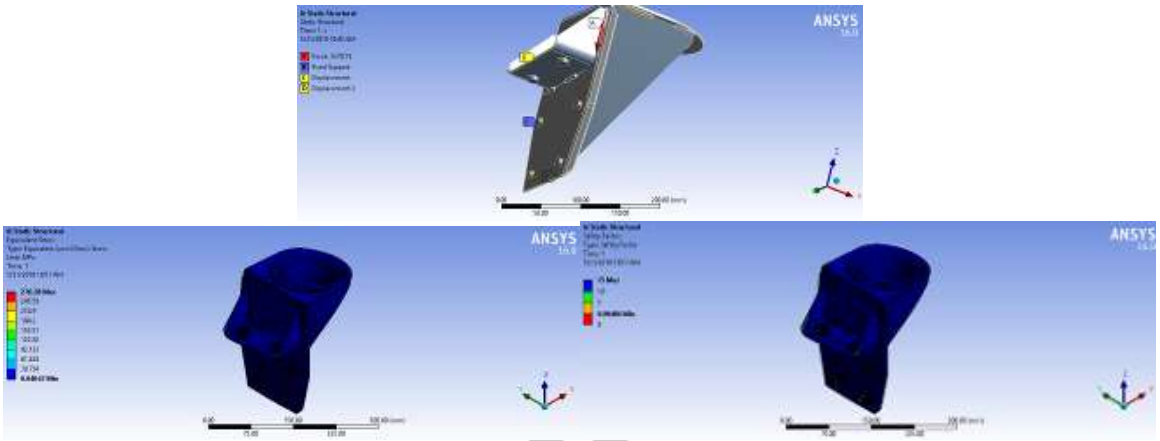


Fig. 6.Boundary Condition, Max. Equivalent Stress and Safety Factor of Case D

E. Only Bolt Ends are clamped:

Safety Factor is 1.068 & Max Equivalent Stress is 234.02 Mpa in this case E.

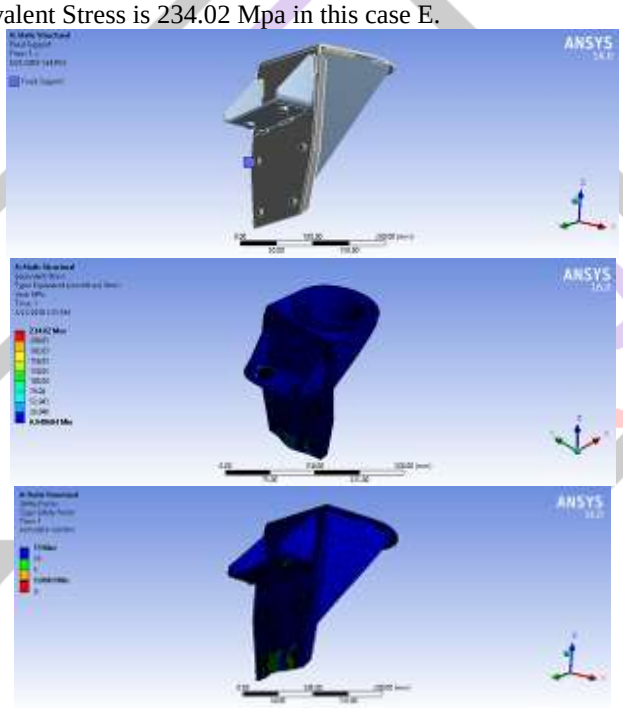


Fig. 7.Boundary Condition, Max. Equivalent Stress and Safety Factor of Case E

F. Result Comparison of above five cases:

TABLE I

COMPARISON OF RESULTS FROM VARIOUS BOUNDARY CONDITIONS

Case. No.	Equivalent Stress (MPa)	Minimum Factor of Safety
A	182.25	**1.371**
B	**51.16**	**4.886**
C	***53.02***	***4.715***
D	276.28	**0.904**
E	234.02	**1.068**

V. Comments after Comparison:

From the observation of above results simultaneously it can be said that factor of safety in Case D is below 1 which is not the acceptable case. Case A and Case E is showing the minimum factor of safety very close to 1. But present application is Automotive which is subjected to road shocks continuously and application is heavy loading such low factor of safety may cause problem and chances of failure may increase. Therefore these results and boundary conditions are not acceptable with reference to mentioned situation.

After looking at Case B & Case C, the value of factor safety is around 4.7 or approximately close to 5. As for heavy vehicle application value of safety factor is generally 3 to 6 looking logical.

But comparing and considering the boundary conditions of both, Case C looking more logical. As in case B both surfaces in contact are clamped, which means all the degrees of freedom is fixed and in Case C, translation is fixed only in perpendicular direction of surface of contact. But it is free to move in remaining two directions. Bracket after loading can move vertically downwards and in sideways due to clearances in bolts and respective holes. Accordingly results of **Case C** are chosen for further optimization by ESO method.

VI. Results for Original Brackets

Boundary conditions, safety factor and stress results are already calculated for case C and displayed in section IV. Remaining things are shown below like meshing details, Load application and total deformation.

A. Meshing (Ansys 16.0):

Default medium Ansys meshing is used. Main sheet metal body is meshed with tria-elements and welds got meshed with quad elements. Total numbers of nodes created are 88939 and Elements created are 33681.

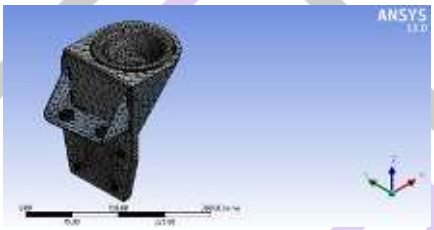


Fig. 8.Meshing of original Bracket

B. Load Application:

Load 3270 N is applied vertically downward on the top surface of conical cup as shown in figure. After observing the actual assembly elements it is clear that load is distributed as shown in figure.

Fig. 8.Applied load on bracket for analysis

C. Total Deformation of Original Bracket:

Maximum Total deformation is found to be 0.060 mm as can be seen in figure.

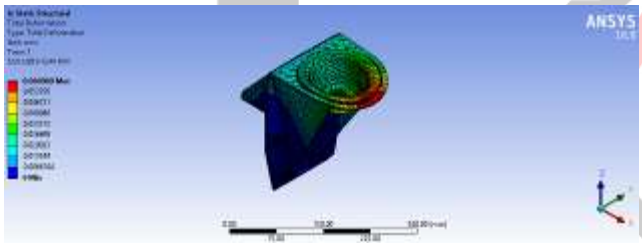


Fig. 9.Total Deformation of original bracket

D. Modal Analysis of Original Bracket:

Considering dynamic analysis for bracket as it will be subjected to continuous road excitations with frequency range 0 to 20 Hz, it is very necessary to find out the minimum natural frequency of the individual bracket. Natural frequency and road excitation frequency should not coincide with each other to avoid the resonance condition.

Following are the modal analysis results for first six mode shapes. Meshing and clamping conditions are same as above. Minimum Natural frequency of original bracket is 833.05 Hz.

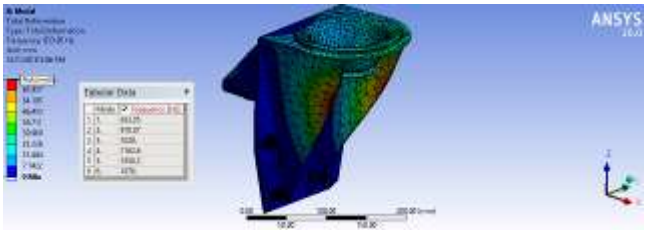


Fig. 10.Modal Analysis of original bracket (here is the first mode image)

Also following table No II is showing all the compiled results for original bracket.

TABLE II

RESULTS FOR ORIGINAL BRACKET

Sr. No.	Parameter	Value
1	Total Deformation	0.060 mm
2	Equivalent Stress	**51.02 MPa**
3	Equivalent Strain	**0.00026512 mm/mm**
4	Safety Factor Minimum	**4.715**
5	Weight of the Bracket	**3.356 kg**
6	Minimum Natural Frequency	**833.05 Hz**

VII. SUGGESTED TOPOLOGY BY ANSYS 18.0 AND OPTIMIZED TOPOLOGY USING ESO TECHNIQUE

A. Suggested Topology by Ansys

Ansys 18.0 and above versions now has separate topology optimization tool which gives the optimized topology after giving constraints to it. Present analysis here is performed in Ansys 18.0 Academic.

Following Fig is showing the areas of Design and Non-design. We can only make changes in the design area and non design area is nothing but the region where boundary conditions are applied.

Constraints of Compliance minimization and mass reduction by 20% were given to it. Following figure shows the suggested topology from the software.

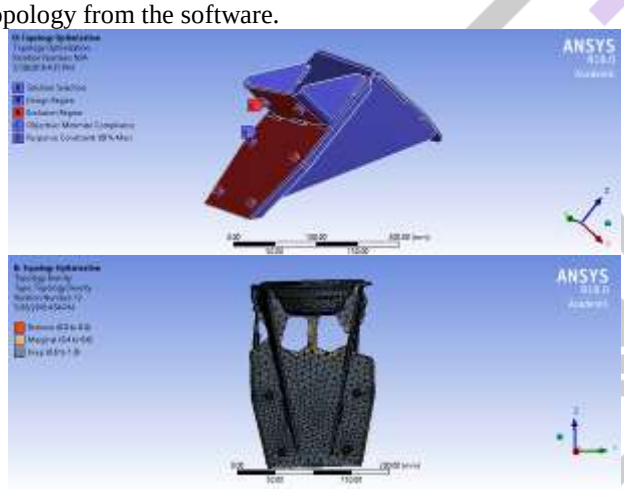


Fig. 11.Figures showing design and exclusion region and the suggested topology in Ansys 18.0

B. Application of ESO to the Bracket:

Following two figures are showing the stress distribution in the bracket. Areas with least stressed elements can be seen in Blue colour. Highlighted area is showing the scope for material removal. As per the principle of ESO, least stressed elements are inefficient and can be removed from the structure. If we see the suggested topology and highlighted area we can say that both are almost similar patterns.

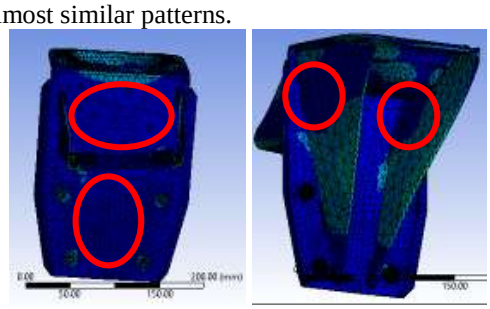


Fig. 12.Highlighted area is least stressed

C. Optimized Bracket:

Following are the nine different possibilities of optimized profiles were checked. Results are compared for % mass reduction, minimum static factor of safety and fatigue factor of safety. CAD work is done in UG-NX 10 again keeping manufacturing constraint in mind. That means along with compliance minimization and weight reduction, manufacturing is also important constraint for topology optimization.

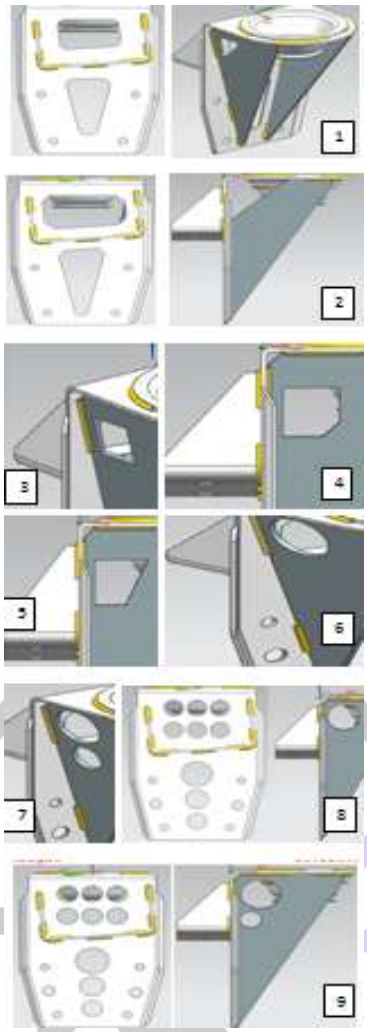
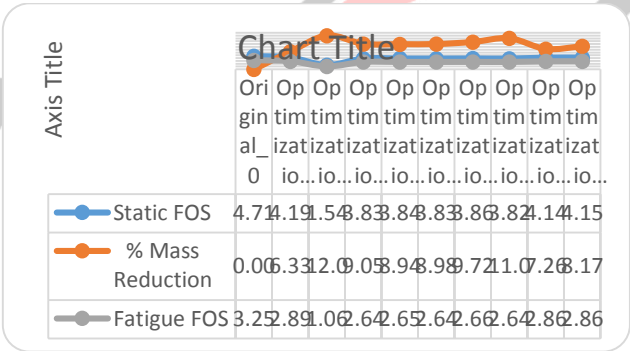


Fig. 13.Some possibilities of different topologies



	Original_0	Optimization...	Optimization...	Optimization...	Optimization...	Optimization...	Optimization...	Optimization...	Optimization...	Optimization...
Static FOS	4.71	4.19	1.54	3.83	3.84	3.83	3.86	3.82	4.14	4.15
% Mass Reduction	0.00	6.33	12.0	9.05	3.94	3.98	9.72	11.0	7.26	8.17
Fatigue FOS	3.25	2.89	1.06	2.64	2.65	2.64	2.66	2.64	2.86	2.86

Fig. 14.FEA result Comparison of different topologies

After observing the comparison graph, we can see that for case 3 to 7, mass factor of safety is almost same and below 4. Mass reduction is around 11% in case 7 but FOS is below 4. In case 8 FOS is increased but mass reduction is around 7 %. In the case 9, FOS is above 4 and mass reduction is also above 8 %. Assuming value of optimized minimum static factor of safety as 4 (lowering it by 0.7) and minimum fatigue factor of safety as 2.7 (i.e. dropping it by 0.5); Topology 9 is considered as optimized topology. Following are the FEA results for Topology 9.

Meshing & boundary conditions are kept as before & mesh statistics shows 87937 nodes and 33008 elements.

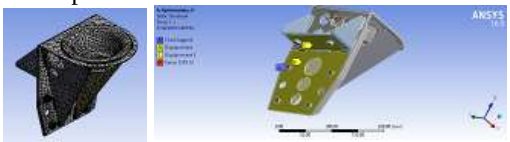


Fig. 15.Meshing & Boundary Conditions of modified bracket

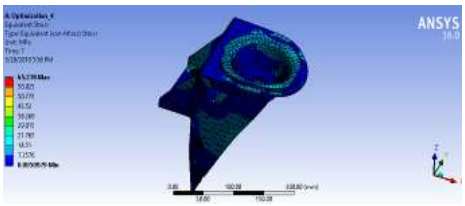


Fig. 16. Equivalent Stress for optimized bracket

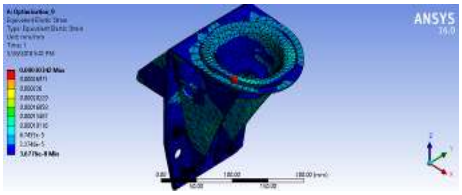


Fig. 17. Equivalent Strain for optimized bracket

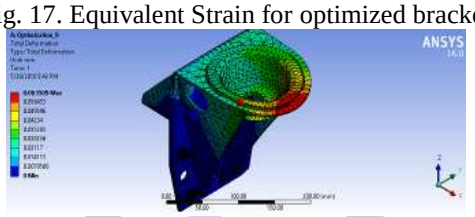


Fig. 18.Total deformation for optimized bracket

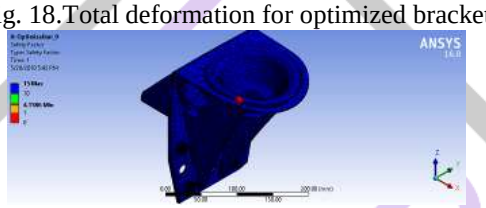


Fig. 19.Static factor of safety for optimized bracket

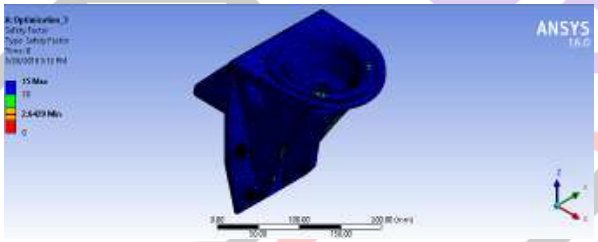


FIG. 20.Fatigue safety factor for optimized bracket

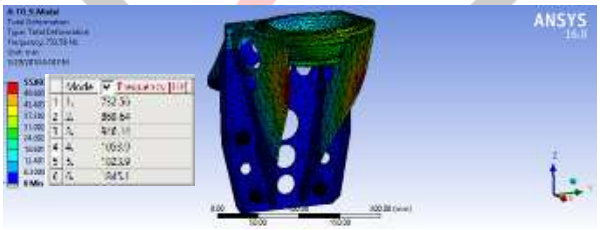


Fig. 21.Modal analysis for optimized bracket (First Mode)

TABLE III
RESULTS FOR MODIFIED BRACKET

Sr. No.	Parameter	Value for Original Bracket	Value for Modified Bracket	Difference (New-Original)
1	Total Deformation	0.0601 mm	0.0635 mm	0.0034 mm
2	Equivalent Stress	**53.024 MPa**	**60.116 MPa**	**7.092 Mpa**
3	Equivalent Strain	**0.00026512 mm/mm**	**0.00030342 mm/mm**	**0.0000383 mm/mm**
4	Static Safety Factor Minimum	**4.715**	**4.159**	**- 0.556**
5	Fatigue Safety Factor Minimum	**3.251**	**2.868**	**- 0.383**
6	Minimum Natural Frequency	**832.93 Hz**	**732.56 Hz**	**-100.37 Hz**
6	Mass of Bracket	**3.356 kg**	**3.082 kg**	**- 0.274 kg (- 8.164 %)**

VIII. EXPERIMENTAL VERIFICATION:

Selecting the most suitable boundary condition, bracket is optimized using FEA. On the basis of comparison of various topologies, Optimized topology is selected & modifications in the original brackets are done accordingly. But the test results obtained from FEA are needed to be validated experimentally. So, further step will be the Experiment to check the Static Structural results and the modal results using UTM with & strain gauge and Hammer test, respectively. Fatigue test will not performed its validation will be kept as future scope.

Below is the photograph of the fixture prepared for the tests. Tests are yet to be performed.

Fig. 22.Fixture for tests

IX. CONCLUSION

With objective of mass reduction this work was started. Ansys is used for FEA analysis. During research work various boundary conditions were studied and one selected applying general technical logics. On the basis of selected one further work was continued.

8.164 % of mass reduction is obtained using FEA results after analyzing nine different topologies as per ESO technique, with very less compromise with constraint parameters like factor of safety, natural frequency etc.

Results will be validated using experimental analysis for static and modal values. Further extension for this work will be validating fatigue values and using this technique for other automobile structural components to reduce overall weight of the vehicle and ultimately increasing the fuel economy, reducing emissions, etc.

References

[1] Mayur Jagtap, Ashvin Dhoke, "Topology Optimization of Exhaust Mounting Bracket", *Tech Mahindra, Altair Technology Conference* India, 2017.

[2] Y.S. Kong, S. Abdullah, M.Z. Omar, S.M. Haris, "Topological and Topographical Optimization of Automotive Spring Lower Seat", Latin *American Journal of Structures*, 2016.

[3] Subhash Sudulaimuthu, Barry Lin, Mohamed Sithik, Rajeev Rajendran, "Track Bar Bracket Development with the Help of Advanced Optimization Techniques", *SAE Technical Paper 2016-01-1387*, Apr 2016.

[4] Sureshkumar Kadrigulla, Navin Sukumar, "Structural Non-linear Topology Optimization of Transmission Housing and Its Experimental Verification", *SAE Technical paper, 2015-01-0098*, Mar 2015.

[5] Guan Zhou, Guangyo Li, Aiguo Cheng, Guochaun Wang, Hongmin Zhang, and Yi Liao, "The Lightweight of Auto Body Basef on Topology Optimization & Sensitivity Analysis", *SAE Technical Paper, 2015-01-1367*, Apr 2015.

[6] Bo Tan, Yu Yang, Jun Huang, Wenhui Liu, and Dongquing Zhang "Structure Optimization Method for Heavy Truck Propeller Shaft Bracket", *SAE Technical paper, 2015-01-0*638, Apr 2015.

[7] Guangyao Li, Fengxiang Xu, Xiaodong Huang, and Guangyong Sun, "Topology Optimization of an Automotive Tailor-Welded Blank Door", *Journal of Mechanical Design, ASME*, Vol. 137/ 055001-1, May 2015.

[8] BGN Satya Prasad, M Anil kumar, "Topology Optimization of Alloy Wheel", *Hyundai, presented at Altair Technology Conference India*, 2013

[9] Parag Nemichand Jain, Satish Pavulari, "Experiemental and Finite Elemnt Analysis of Bogie Suspension Mounting Brackets", *SAE Techinical Paper, 2013-01-2789*, Nov 2013.

[10] Vasudeva Rao. S, Chetan Raval, "Brake Actuator Mounting Bracket Optimization Using HyperWorks", *Mahindra Engineering, presented at HTC* 2010.

[11] Shi Jin-fa, Sun Jian-hui, "Overview on Innovation of Topology Optimization in Vehicle CAE" *International Conference on Electronic Computer Technology, IEEE*, 2009.

[12] Manoj Ukhande, Bharat Pawar, Girish Shegavi, "Control Arm Weight Optimization Using HyperWorks", *Bharat Forge, presented* at HTC 2009.

[13] Xiao-Yong Pan and Doni Zonni, Guo-Zhong Chai, Yan-Qing Zhao and Cui-Cui Jiang, "Structural Optimization for Engine Mount Bracket", *SAE Technical Paper*, 2007-01-2419, 2007.

[14] Hong Suk Chang, "A Study on the Analysis Method for Optimizing Mounting Brackets", *2006 SAE World Congress, Detroit, Michigan*, 2006-01-1480, Apr 3-6, 2006.

[15] Yugi Ogata, Satoru Suzuki, Masami Hiraoka, "Development of Topology Optimization Method for reduction of Transmission Housing Weight", *SAE Technical Paper, 2005-01-1699*, 2005.

[16] Eduardo Castelo Branco Porto, Geraldo Minoru Kato, Rogerio Ribeiro, Walter Wilhem Lorenz Junior, "Structural Optimization of Rear Cabin Suspension Mounting Bracket and of a Frame Air Filter Bracket", *SAE Technical Paper, 2004-01-3391*, 2004.