International Journal for Research in Engineering Application & Management (IJREAM)
ISSN : 2454-9150 Vol-07, Issue-06, SEP 2021

Design Optimization and Analysis of Cabin Mounting Bracket of Heavy Commercial Vehicle

Ghanshyam Gangurde[1], Dr. D M Patel [2], Kiran C. More [*3]

[2,3]Department of Mechanical Engineering, DNPCOE, Shahada, MH, India.

[1]Department of Mechanical Engineering, School of Engineering and Technology, DYPU Pune, Pune, MH, India. [1*]Corresponding Author- prishaoptimizedsolutions@gmail.com

Abstract Automobile sector is one of the largest branch of mechanical engineering industry. It consumes a lot of fuel transporting goods and people from one place to other by road. Now a day reducing automobile weight for better economy is the challenge industry faces right now. Project work is focused on design and weight optimization of HCV truck's Cab mounting. But very less focus is given by the researchers on the comparison of the optimization techniques to find out the best optimization technique for cabin mounting bracket. Study will be focused on finding optimization technique which is best suited for the weight optimization of the Cab mounting. Objectives of this study will be focused on the design of Cab mounting for HCV truck according to available space constrains, perform optimization of the Cab mounting design using different optimization techniques and analyze the output data to select the best one, manufacturing, testing the best design.

Keyword: Cabin Mounting Bracket; Optimization, Ansys, Weight optimization; HCV Truck.

I. INTRODUCTION

An Automobile is a self-propelled vehicle which is used for transportation of goods and passengers. The motor vehicle, both passenger car and truck are generally considered to be made up of two major assemblies Body and Chassis. Chassis is a main structure of a vehicle. The chassis contains all the major units necessary to propel the vehicle. Body is the super-structure of the vehicle. Body is bolted to the chassis. The chassis, body, structure make the complete vehicle. The truck consists of different assemblies, bodies, structures performing their functions smoothly. Although there are more important parts, the cabin is a place where the driver and co-driver are seated. Their weight will be mainly on the floor where it should withstand many loads acting from different ways in different direction. This makes the driver seated without any vibrations and distractions. A flat sheet of thin material. The floor panel is very flexible for out-of-plane loads. The aim of the floor is the vehicle, such as the side frames, cab side bracket. Floors are applied to loads normal to their plane. Under these circumstances they do not act as simple structural surfaces. The floor is stiffened against out of plane loads by added beam members arranged into a planar framework. The advantages of tilting cabin over rigid cabin is easy for servicing, less weight, easy for design modifications and provide less vibrations. [6]

The present bracket that is used in the vehicle Weight about 2.82 kg. There are four mounting brackets are used support cabin assembly is 1500 kg including one driver, three passengers. Two bracket are attached at the front side and remaining two are attached at rear side [6]. Rear end this cab

 DOI : 10.35291/2454-9150.2021.0545

ISSN : 2454-9150 Vol-07, Issue-06, SEP 2021

mounting component is mounting on chassis. Isolators are utilize to hold cab on centre channel. Which is fitted to the frame by cross member end bracket. The front cabin mounting bracket are assembled of two bracket attached to chassis as shown in fig. 1.

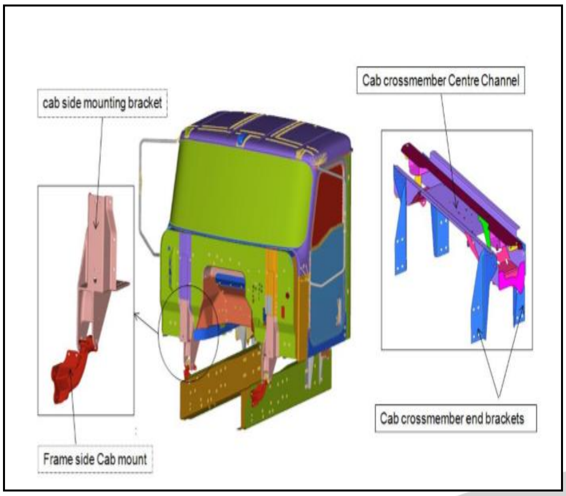


Fig. 1: Schematic diagram of cab mounting system [1]

In this investigation cabin mounting was used as component for study. Weight reduction is beings issues in truck manufacturing industry. Weight reduction will give impact on fuel efficiency, efforts to reduce emissions, save environment. Weight can be reduced through several types of technology improvement such as advance in materials, design, analysis methods and optimization technique [5].

Automobile manufacturing continuously improvement in new vehicles with more lixury, convenience, perform and safety. The design optimization should be applied to obtain minimum weight with maximum feasible performance based on design boundaries. Among the vehicle structural components the cabin mounting bracket is one of the important part in cabin system.

An cabin mounting bracket is mainly composed of five parts cab mounting bracket, bell shape, dish, black, c channel, two triangular plate. Cab mounting bracket is jointed on chassis. C channel attached to the back side of the cab mounting bracket. It used to support from back side of the entire structure. Dish is attached to the bottom surface of the cab mounting bracket. Triangular plate is used to support to the entire structure of the cabin mounting bracket. It carries the high stress. The cabin mounting bracket as shown in fig. 2.

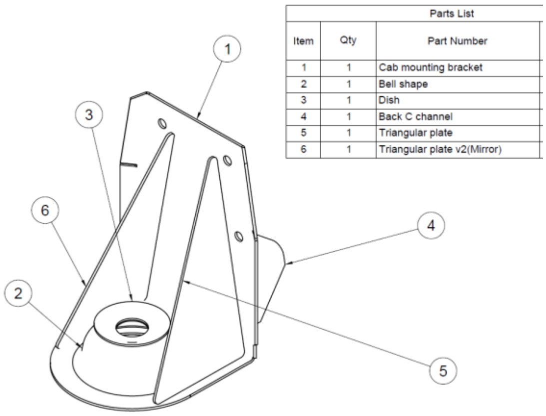


Fig. 2: Cabin Mounting Bracket

II. Literature Review

Shailesh Kadre et.al [1] focused on the The simplified approach was tried on cab mounting system of a truck assembly. All the CAE results were validated against the test data. Good test correlation was observed between test and FEA results.

Cornelia Stan et.al [2] studied an analytical study concerning the optimization of the mounting system of a truck cab using a FE model. This model is useful to studies on the vibration transmissibility among different elements of the structure, in order to improve the comfort in the truck cab. **Monali Deshmukh et.al [3]** presented that the proposed model of the engine mounting bracket 12.5%weight reduction is achieved maintaining an acceptable level of yield stress and harmonic response.

M Meghana et.al [4] Analysed that Mounting brackets are generally used support the structural component and electronic components such as batteries, seats, cabin, Chassis, rear body and also it should support the external load such as passengers. Since it is very much needed to reduce the weight and therefore it is to be redesigned or optimized for minimum weight without sacrificing the functionality. **Jeong Woo Chang et.al [5]** Studied the topology optimization is very useful engineering technique especially at the concept design stage. Structural analysis methodology of compressor bracket was verified on the static and dynamic loading condi-

DOI : 10.35291/2454-9150.2021.0545

tion with 2 bracket samples for the topology optimization base model.

P. Meghana et.al [6] Presented that The mounting brackets are meant for supporting the structural component and electronic components such as batteries, seats, cabin, chassis, rear body and also it should support the external load such as passenger's weight. **Sandeep Maski et.al [7]** Analysed the Bracket is one of the important components of an engine mount assembly, heavy performance truck has their engine supported by bracket and this engine mounting brackets assembly is used in chassis front frame which has been designed as a framework to support engine along with transmission member. **Vasudeva Rao S et.al** [8] Worked on a This simulation helps to reduce the burden on product development cost and time. A number of iterations were performed in CAE world before concluding the final design. This significantly replaced the proto part making and testing cost and time. **Vijay Kalantre et.al [9]** Studied the Topology optimization of Leaf Spring Mounting Bracket is performed here as a case study and which results in 19.39 % of weight reduction without compromising strength of the bracket. Topology is scientific tool that provides guideline about removing of inefficient material from the structure. Jadhav & Madaki [10,11] analyzed and optimized the Front Suspension Shackle Bracket.

Problem Statement

Many researchers have focused on behavior of the cabin mounting brackets and other brackets under structural loads and vibrational loads. Some of the researchers have worked on optimization of the bracket design for weight reduction and some have studied various shape optimization technique for weight reduction. But very less focus is given by the researchers on the combine the weight and shape optimization techniques to find out the cabin mounting bracket. Project work is focused on design, weight and shape optimization of Cab mounting.

Objectives

I. Design Cab mounting for HCV truck according to available space constrains.

II. Perform optimization of the Cab mounting design using different optimization techniques and analyze the output data to select the best one.

III. Manufacturing and testing of the best design option from optimization techniques.

III. METHODOLOGY

The normal mode analysis of the cabin mounting bracket is obtained using finite element analysis. Finite element modeling using ANSYS software. The weight reduction done using topology optimization Process flow chart as shown in fig 3.

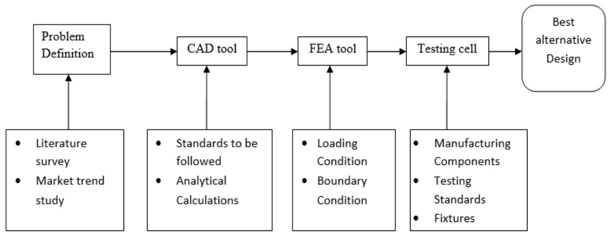


Fig. 3: Process Flow Chart

IV. CAD & FEA WORK

4.1 CAD Work

1. First Stage involved in the methodology of this project's FEA work is to design Cab mounting bracket for TATA 1613 truck.
2. Detailed modelling of the component can be performed using AUTODESK FUSION 360 software. Images below show the actual component purchased for the reverse engineering.
3. Generally mild steel of material used for cabin mounting bracket. The properties of material as shows below.

4.1.1 Material Selection

There are many materials used for manufacturing such as mild steel, structural steel, cast iron, wrought iron. Mild steel is lesser than yield point, hence mild steel better materials.

Table.1 Mechanical properties of Mild Steel [7]

Sr. No	Mechanical property	Mild Steel
1	Young's modulus (MPa)	2.1 x e+5
2	Poisson's ratio	0.30
3	Density (Kg/mm3)	7.89 x e-9
4	Yield strength (N/mm2)	370
5	Ultimate Strength (N/mm2)	440

 DOI : 10.35291/2454-9150.2021.0545

ISSN : 2454-9150 Vol-07, Issue-06, SEP 2021

By measuring the component bought using measuring instruments like measuring tape, venire caliper and scale 3-dimensional model is created using FUSION 360. Model created is shown in the image below. Model module of Fusion 360 is used to create the object shown in the image below. Series of sketching and modelling commands are used to create the object the cad model for cabin mounting bracket assembly. It is made different sheet metal components. Sheet metal module of Autodesk FUSION 360 is used to create the individual components of the cabin mounting bracket and then assembly module is used to assemble those components together. These components are made using sheet metal cutting and bending processes as well as welding is used to join them together permanently[24]. Finite Element analysis is performed on the created geometry of the component. The Mass of our Cabin Mounting bracket is 2.8222 kg. The model of Cab Mounting Bracket is design in Space Claim as shown fig 4.

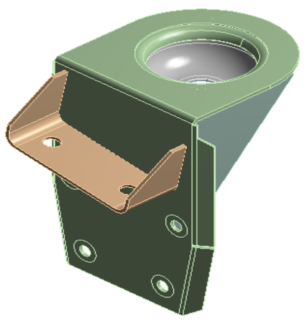


Fig. 4: CAD model of Cabin Mounting Bracket Assembly

For analysis purpose meshing of our geometry is in Ansys 18.2 with the element size of 2mm.

Table.2 FE Model Details

Element type	Hexahedran, Tetrahedran & Quadratic shell
No. of elements	312989
Average element size(mm)	2

4.1.2 Load Acting on Cabin

Truck is a vehicle with cabin mass of 1.5 ton in completely loaded conditions.

- Generally maximum mass of TATA 1613 truck cabin are 1 ton.
- Consider 1 driver, 3 passengers and mass of each 90 kg (4 x 90 =360 kg).
- Consider some luggage and other weight 140 kg.
- Total mass on cabin mounting W_1 (1000kg + 360kg + 140kg =1500kg)

Total of 1.5ton loading gets distributed in 4 different brackets on which isolators are mounted. The product we are studying is utilized to connect these isolators to the vehicle body. So we can assume that 4 brackets of the cabin can each support around 375 kg of loading at fully loaded condition.

So loading on the single support bracket of the suspension mounting can be calculated as below.

Total mass supported by cabin mounting bracket - 375 kg.

Acceleration due to gravity (g): - 9.81 m/s^2.

Total load acting on cabin mounting bracket W_2 can be given as[25].

$$W_2 = \text{Mass} \times g \qquad \text{------------------(1)}$$

$$W_2 = 375 \times 9.81$$

$$W_2 = 3678.75 \text{ N}$$

4.1.3 Loading and Boundary conditions[26]

Hence we applied total load on bracket are 3678.75 N. The load applied on bracket at the center of ball shape as shown in fig 5.

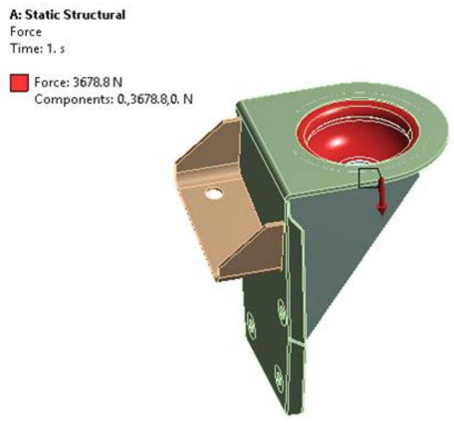


Fig. 5: Load on Cab Mounting Bracket

 DOI : 10.35291/2454-9150.2021.0545

4.1.4 Static Structural Analysis

The Structural analysis is done to find the Total Deformation and Equivalent von-mises stress in Cab mounting bracket after the application of constraint on it. The maximum deformation observed is only 0.342232mm which is very less. The results of analysis are shown 6.

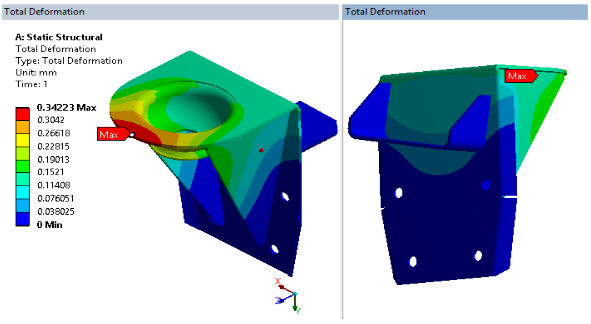


Fig. 6: Total deformation

Equivalent stress with there maximum value on each portion of cabin bracket.

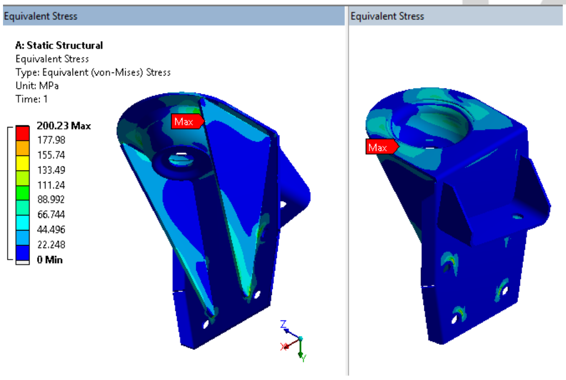


Fig. 7: Equivalent Stress Original Cabin Mounting Bracket

4.1.5 Modal Analysis

Baseline Model the Baseline model of Cab Mounting Bracket. Number of Nodes and Element In model analysis nodes are form 65560 and elements of model is 29052. The Baseline Model Cab Mounting Bracket is fixed at a working condition.

Table 3: 6 mode of Frequency for Baseline Model

Sr. No	Mode	Frequency (Hz)
1	1	457
2	2	540.87
3	3	782.07
4	4	790.33
5	5	792.16
6	6	835.39

4.2 Design Modification (Parametric Analysis)

From above analysis we have seen that stress on the C channel attached at the back of bracket is negligible. So for further weight optimization we have eliminated that C channel. The optimization of weight is done further by making a set of hole pattern on mounting bracket where stress is already negligible. These will help to reduce stress concentration at particular points and reduces the mass of final geometry.

So the set of no of holes are made parametric and the set which reduces maximum mass and is also feasible for manufacturing is chosen.

Table of Design Points

	A	B	C	D	E	F	G	H
1	Name	P1 - Group1	P2 - Group2	P3 - Group3	P4 - Group4	P5 - Group5	P6 - Equivalent Stress 2 Maximum	P7 - Geometry Mass
2	Units						MPa	kg
3	DP 0 (Current)	2.5	5	7.5	5	2.5	230.28	2.4866
4	DP 1	15	7.5	15	7.5	15	231.95	2.4371
5	DP 2	20	10	20	10	20	234.17	2.3918
6	DP 3	25	12.5	25	12.5	25	234.39	2.3335
7	DP 4	25	10	25	10	25	229.64	2.3419
*								

Fig. 8: Parametric Table

So set DP2 is chosen as most effective one in reducing mass of our geometry and also feasible in manufacturing and mass is reduced. The Geometry of DP2 set is shown in fig 9.

Meshing of DP2 Cab Mounting Bracket. Applying forces on the cabin mounting bracket see the results.

A set of hole pattern isadded diameter of 20mm, 10mm, 20mm, 10mm,20mm,20added a both sides Collar is added at both sides of Body side

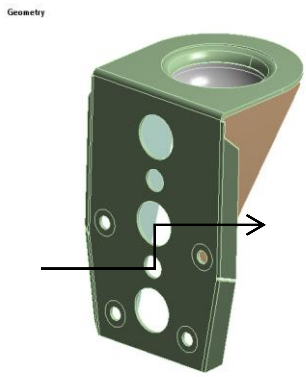


Fig. 9: Geometry with set of holes

DOI : 10.35291/2454-9150.2021.0545

V. Final Result Discussion

5.1 Static Structural Analysis

The DP2 Cab mounting Bracket is fixed at working condition which is only boundary condition. Total deformation of DP2 is 0.3126 Cab Mounting Bracket are shown in fig 10.

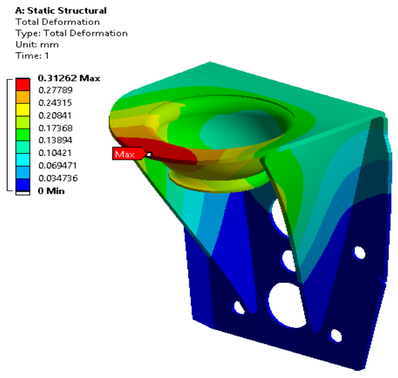


Fig. 10: Total deformation

Equivalent stress of DP2 is 234.17 MPa Cab Mounting Bracket are shown below.

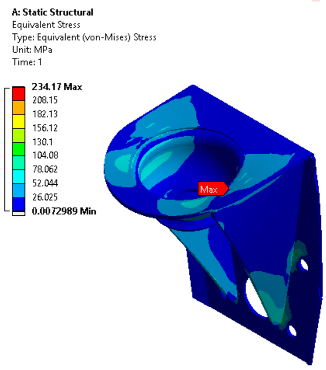


Fig. 11; Equivalent Stress

From above result it can be seen that the stress concentration at different area are reduce due to application of hole pattern on it.

5.2 Modal Analysis

Table. 4: 6 mode of Frequency for DP2

Sr. No	Mode	Frequency (Hz)
1	1	457.31
2	2	545.21
3	3	799.11
4	4	842.79
5	5	1235.3
6	6	1593.6

VI. Experimental Validation

6.1 Experimentation

The Cabin Mounting Bracket is tested on FFT Analyzer and Universal Testing Machine table test rig by mounting on fixture with loading condition as discussed above, over the operational loading range. Test conducted found alignment with values observed through Finite element analysis.

6.1.1 Testing of cabin mounting bracket on UTM

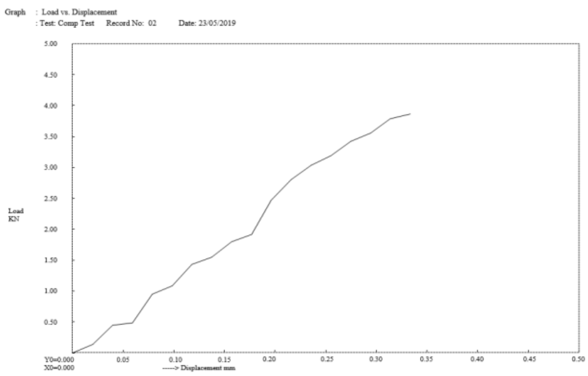
As shown in figure bracket is clamped on the Universal Testing Machine fixed jaw with the help of fixtures. Bracket is mount on fixture by using a 4 bolts. By checking all the connections the test is started by gradually opening the valve to give hydraulic pressure. Computer is used for operating the UTM and for recording the load v/s displacement

Fig. 12: Experimental Setup for UTM testing

Table. 5: Load Test

Description	Testing values	Specifications (any)
Load Applied	3.67	Client Described load
Observation	Deflection of 0.34 mm. No damage on sample.	

Graph 1: UTM Testing Plot



 DOI : 10.35291/2454-9150.2021.0545

6.1.2 Experimental Set Up For Vibration Analysis Using FFT Analyzer

1) Start the FFT analyzer.

1)Go to Run-up mode ➔ Set ➔ New set (Rename) ➔ OK.

2) Set the ranges of Force Amplitude, Frequency and Scale for Graphs.

3)A Force of a very small amplitude is applied on a bracket by a hammer which is provided with FFT analyzer and corresponding results (Natural frequencies) are obtained on FFT analyzer. These results are shown below.

Fig. 13: Experimental Setup for FFT Analysis

Fig. 14 FFT Analysis

VII. Result Summary

7.1 Result Table

Various analyses are performed on the different shapes of the bracket the results are obtained for maximum equivalent stress and maximum deformation is 234.84MPa and 0.31262mm respectively and first modal frequency values for the same shape is 457.31Hz and also reducing mass of our geometry and also feasible in manufacturing and mass is reduce by 0.4304kg.

Table.6: Result Table

Iteration No	Maximum Equivalent Stress	Maximum Deformation	Mass reduction (%)
Baseline	200.23	0.3422	-
Iteration 1	329.84	0.3211	11.5
Iteration 2	234.17	0.31262	3.5

The values observed from the experimentation are as follows.

Table. 7: FEA vs Testing

	FEA	Testing	Error %
Stress MPa	234.17	245.12	4.46%
Deformation (mm)	0.31262	0.34	8.05%

VIII. Conclusion

1. From this study we have observed that in the baseline iteration, stress observed is 200.23 MPa, and the deformation is 0.3422mm. In the first iteration, the stress observed is 329.84 MPa, and the deformation is 0.3211mm and for second iteration the stress observed is 234.17 MPa, and the deformation is 0.3126 mm.

2. Also in the practical testing, the stress observed is 245.12 MPa and the deformation observed is 0.34 mm. The values are within the conformance of 4.46% and 8.05% respectively.

3. In frist Iteration 1 removed C channel we mass reduced 11.5% and second Iteration 2 applying parametric analysis we mass reduced upto 3.5%. The total mass saving of cabin mounting bracket is 0.4304kg.

Conflict of Interest: The authors declare that they have no conflict of interest.

References

[1] S. Kadre, S. Shingavi and M. Purohit., Durability Analysis of HCV Chassis Using FPM Approach, Simulation Driven Innovation HTC 2011.

 DOI : 10.35291/2454-9150.2021.0545

[2] C. Stan, D. Iozsa, R. Oprea., Study Concerning The Optimization of The Mounting System of The Truck Cab, Advances in Automatic Control. ISBN: 978-960-474-383-4 (2016)

[3] M. Deshmukh, Prof. K R Sontakke., Analysis and Optimization of Engine Mounting Bracket, International Journal of Scientific Engineering and Research 3(5), (2014), pp. 131-136.

[4] M. Meghana, K Vinod Reddy., Field Problem of Cabin Mounting Bracket of Load King Pride, 4(13), 2017, pp. 2352 - 2361.

[5] J. W. Chang and Y. S. Lee., Topology Optimization of Compressor Bracket, Journal of Mechanical Science and Technology 22 (2008), pp. 1668 - 1676.

[6] P. Meghan, Y. Vijayakumar, Dr P. Ravinder Reddy, P. Seema Rani., Analysis of Cabin Mounting Bracket of Truck Using ANSYS, International Journal of Engineering Science Invention 5(11) 2016, PP. 82 – 89.

[7] S. Maski, Y. Basavaraj., Finite Element Analysis of Engine Mounting Bracket by Considering Pretension Effect and Service Load, International Journal of Research in Engineering and Technology, 4(8), 2015, pp. 327 – 333.

[8] Vasudeva Rao S, Soundararajan. S., Optimization of Front Axle Mounts Vibration Characteristics for Driveline Integration with Chassis, Simulation Driven Innovation HTC 2012, pp. 1 – 7.

[9] V. Kalantre, K. H. Munde, A. Pawar., Topology Optimization of Front Leaf Spring Mounting Bracket, International Journal of Scientific Development and Research, 3(7), 2018, pp. 12 -19.

[10] S, Jadhav, S. Madki, Study and Analysis of Front Suspension Shackle Bracket for Commercial Vehicle, International Journal of Engineering Development and Research (4)5, 2017, pp. 1392 - 1396.

[11] S. Jadhav, S. J. Madki., Optimization of Front Suspension Shackle Support using Finite Element Analysis, International Research Journal of Engineering and Technology, 04 2017, pp. 114 -117.

[12] H. Kim, Y. Lee., Optimization design technique for reduction of sloshing by evolutionary methods, Journal of Mechanical Science and Technology, 22 (2008), pp. 25 – 33.

[13] F. Buonamici, M. Carfagni, R. Furferi, L. Governi, A. Lapini, Y. Volpe., Reverse engineering of mechanical parts: A template-based approach, Journal of Computational Design and Engineering 5 (2018), pp 145–159.

[14] V. Neelakandan, T. Ganesan, P. Chakrapani Rao., Weight Optimization of Housing Bracket for Electrical Starter Motor using FEA, Structural Integrity Procedia 14 (2019), pp 345–353.

[15] B. V. Ramnath, C. Elanchezhian, J. Jeykrishnan, Rangavender, P.K. Rakesh, J. Sunjay Damodhar, A danasekar., Implementation of Reverse Engineering of Crankshaft Manufacturing industry, Material Toady Proceeding, pp 994-999.

[16] B. V. Ramnath, C. Elanchezhian, V. Chandrasekhar, A. Arun Kumar, S. Mohamed Asif, G. Riyaz Mohame, D. V. Raj, C .Suresh Kumar., Analysis and Optimization of Gating System for Commutator End Bracket., Procedia Materials Science 6 (2014), pp. 1312 – 1328

[17] N. Anwer, L. Mathieu., From reverse engineering to shape engineering in mechanical design, Nabil Anwer, Luc Mathieu, CIRP Annals - Manufacturing Technology 6 (2016), pp.1 – 4.

[18] M. Viqaruddin, D. R. Reddy., Structural optimization of control arm for weight reduction and improved performance, Materials Today: Proceedings 4 (2017), pp. 9230–9236.

[19] S. Porziani, C. Groth, M. E. Biancolini., Automatic shape optimization of structural components with manufacturing constraints, Procedia Structural Integrity Procedia 12 (2018) pp. 416–428.

[20] J. Fang, Y. Gao, G. Sun, C. Xu, Q Li., Multiobjective robust design optimization of fatigue life for a truc kcab, Reliability Engineering and System Safety 135(2015), pp.1–8.

[21] K. C. More, G. M. Patil, A. A. Belkhede (2020). Design and analysis of side door intrusion beam for automotive safety. Thin-Walled Structures. 153: 1-10. ISSN: 0263-8231

 DOI : 10.35291/2454-9150.2021.0545

[22] K. C. More, S. G. Dongre, G. P. Deshmukh (2019). Experimental and Numerical analysis of Vibrations in Impeller of Centrifugal Blower. SN Applied Sciences. 2, 82: 1-14.ISSN: 2523-3971.

[23] A. J. Herrema, N. M. Wiese, C. N. Darling, Baskar Ganapa-thysubramanian, A. Krishnamurthy, M. Hsu,., A framework for parametric design optimization using isogeometric analysis, Computer In Methods Applied Mechanical Engineering 26(2016) pp. 1-35.

[24] K. C. More, G. M. Patil, A. A. Belkhede (2020). Experimental and Numerical analysis of Vibrations in Impeller of Centrifugal Blower. Thin-Walled Structures. 153: 1-10. ISSN: 0263-8231.

[25] K. C. More, S. G. Dongre, G. P. Deshmukh (2019). Experimental and Numerical analysis of Vibrations in Impeller of Centrifugal Blower. SN Applied Sciences. 2, 82: 1-14.ISSN: 2523-3971.

[26] More, K.C.; Rao, R.V. Design Optimization of Plate-Fin Heat Exchanger by Using Modified Jaya Algorithm. In Advanced Engineering Optimization through Intelligent Techniques; Springer: Berlin/Heidelberg, Germany, 2020; pp. 165–172.

[27] M. Langelaar., Topology Optimization for Multi-Axis Machining, Computer In Methods Applied Mechanical Engineering 26 (2016), pp. 1-29.

[28] H. Kim, W. Hwang., Effect of deboning on natural Frequencies and frequency response functions of honeycomb sandwich beams., Composite Structures (2002), pp. 51-62

 DOI : 10.35291/2454-9150.2021.0545