International Journal for Research in Engineering Application & Management (IJREAM)
ISSN : 2454-9150 Vol-06, Issue-02, May 2020

Experimental and FEA analysis of weight optimization of existing tyre bracket mounting

[1]Mr. Swapnil D. Korde, [2]Prof. Anil V. Suryavanshi
[1,2]Mechanical Department, Vikhe Patil College of Engineering,Ahmednagar, India.
[1]kordeswapnil6001@gmail.com, [2]profanilvs@gmail.com

***ABSTRACT* - Tyre support mounting are widely used in heavy load vehicles to hold spare tyre with chassis. Heavy vehicles industry regularly improving from many years with the efforts in modification of the mechanical parts of vehicle in order to improve their performance. The most effective way of increasing vehicle mechanical efficiency while decreasing emissions is to reduce vehicle weight.. Finite element analysis of mounting will be done using ANSYS Workbench. Fixture will be designed for mounting bracket on UTM. Experimental stress of mount will be measured using strain gauge and applying corresponding loading through UTM. Validation for strain vector from FEA & Experimental results will be done. Conclusions and future scope will be suggested.**

Keywords—*FEA, UTM, Tyre mounting bracket*

I. INTRODUCTION

The power plant is that the largest targeted mass within the vehicle and if it's not properly forced and isolated it'll cause vibrations within the body and frontend sheet. The engine is subjected to numerous moving disturbances. Some are external to that et al internal. Random shocks from the road, transmitted through the suspension, shake it. Thus do periodic shaking forces from the universal joints within the mechanical device shaft. Any rotating imbalance within the engine, transmission, or engine-mounted accessories are exciters. Therefore, the mounts should isolate all of them. additionally, they have to support the static weight of the engine and restrain it from lengthwise lateral and vertical movements. The correct style of rubber mounts is also the foremost effective engineering approach to enhance the ride. The analysis of engine mounting stepony components ought to be among the vibration analysis of the engine mount system. It’s necessary not solely to understand what their properties and wherever to position the mounts, however additionally to work out the optimum style of a vicinity to realize the required properties at the side of the specified supporting capability ensuing from the system vibration analysis. Fatty tissue or spare tire stepney is a further tire carried during a car as a replacement for one that goes flat, a blowout, or different emergency. Spare tires in buses ar typically hold on during a fatty tissue well – a roof carrier space higher than the roof a vehicle, typically within the center, wherever the fatty tissue is hold on whereas not in use. In most buses, the fatty tissue isn't secured with a bolt and wing-nut vogue fastener. typically it's unorthodox methodology and cause headache at the time of tire ever-changing. it's hard to lower the one hundred metric weight unit tire by single person with none mechanism. this technique of storing the spare wheel on the roof of the vehicle has some major disadvantages within the variety of fatigue to the motive force, giant cycle time, giant physical stresses to the motive force and diminished productivity. So, this method is sort of cumbersome, time intense and occasionally unreliable. Therefore, there's want for a spare wheel bracket that makes this whole method of tire ever-changing abundant easier, safer, and quicker while avoiding any further fatigue to the motive force.

II. LITERATURE REVIEW

Gloria Alleviaet al. [1]in this paper it presents the metal sections made in titanium based-compounds are one of the most diffused contextual investigations in the field of Additive Manufacturing (AM) advancements: the present age of satellites depends on metal sections to fill in as a link between the body of the satellite and the reflectors and feeder offices mounted at its upper end. In this situation, one of the principle central focuses is the capability of the 3D printed-items, including both the characterization of the miniaturized scale/large scale structure of the segment, and the meaning of its mechanical conduct. In spite of the high nearness, in writing, of works managing the recognition of deformities in the structure, no dissertation can be found about pressure investigation on the genuine part. This theme is pivotal on the grounds that some morphological or dimensional contrasts between the ostensible (CAD) structure and the made one could prompt non-anticipated pressure focuses toward the end. We performed achievability investigation of the Thermo elastic Stress Analysis on a titanium based-combination space section, made by Electron Beam Melting (EBM). The accomplishment of our investigation, in addition to A non-Destructive Dimensional Measurement (triangulation system for figuring out) could empower the

 DOI : 10.35291/2454-9150.2020.0418

old style topology advancement procedures to be actualized posteriori, therefore giving an extra time and cost sparing.

Sagar Ingleaet al. [2]in this article it speaks to an interface structures in shuttle's and flight vehicles experience mechanical stuns from an assortment of sources, for example, stage division, discharge tasks and unexpected outside unsettling influences. The flight interface sections must be structured, broke down and tried to withstand these stun loads representing brief terms. Because of high solidarity to-weight proportion and protection from consumption carbon fiber fortified polymer (CFRP) is prominently embraced as flight interface sections and are supplanting metals. The reaction of CFRP flight interface sections under stun stacking isn't completely comprehended. The transient stun investigations performed on run of the mill flight interface sections, for example, level rectangular and L-shape CFRP sections with fiber directions in 33 distinctive stacking arrangements for fixed-fixed and cantilever limit conditions and exposed to motivation stacking. Results showed that the stacking arrangement [90/0/0/90]s demonstrated best for rectangular fixed and cantilever conditions while stacking grouping [90/90/90/90]s yielded best outcomes for L-formed cantilever and fixed-fixed limit conditions with least response load at fixed end.

S.K. Lohet al. [3]This paper presents and centers around some Finite Element (FE) examinations performed, for example, recurrence investigation to determine the auxiliary reaction because of consonant excitation over a recurrence extend. Vibration and exhaustion of engine section has been persistently a worry which may prompt auxiliary failure if the subsequent vibration and stresses are serious and extreme. It is a noteworthy report which requires sin-profundity examination to comprehend the basic qualities and its dynamic conduct. The resounding frequency can be anticipated dependent on the reactions in recurrence space. The anticipated most extreme anxieties are contrasted and inalienable material yield quality. The plastic deformation is not shrouded in the examination as just flexible property is characterized. In the push to fortify the engine section, distinctive modifications of the engine section geometry have been researched and the examination of consequences of each analysis is introduced. The rib support, edge sweep and thickness have been changed and added to examine the impact to the general static and dynamic conduct. It is discovered that the rib-bolster expansion and the increase of edge range can successfully improve the basic execution dependent on the investigations in question.

Xiao-Yong Panet al. [4], in this paper it presents ans data on structure of powertrain mounting section is always challenging in accomplishing great NVH qualities, sound durability and at the same time decreased weight. Structural optimization is a successful device to acquire an optimum design. Contingent upon the plan status, different schemes, for example size, topology and shape advancement, are applied. In this paper, a contextual investigation of utilization of structural enhancement in the plan of a mount bracket has been introduced. Right off the bat, both test and FEA results uncover issues of the initial design. Thus, it is important to overhaul the bracket. With adequate plan opportunity and time in topology optimization, structure space and streamlining parameters are characterized. Pass on bearing and different manufacturability considerations for the throwing segments are imperative. Shape enhancement is then directed to advance decrease the weight and refine nearby shortcoming. Looked at with original plan of mount section, the mass on the final design is decreased by 12%; the auxiliary unbending nature and strength are improved by about half.

Wu Qin et.al[5] in this paper it presents a strategy for computing detachment proportions of mounts at a power train mounting systems(PMS) is proposed accepting a power train as an inflexible body and utilizing the recognized power train excitation powers and the deliberate IPI (input point inertance) of mounting point sat the body side. With estimated increasing velocities of mounts at power train and body sides of one (Vehicle A), the excitation powers of a powertrain are recognized using conversational technique right off the bat. Another (Vehicle B) has the equivalent power train as that of Vehicle A, yet with various body and mount setup. The increasing speeds of mount sat power train side of a PMS on Vehicle B are determined utilizing the power train excitation forces recognized from Vehicle A. The recognized powers of the power train are approved by comparing the determined and the deliberate increasing velocities of mounts at the power train sizeof the power train on Vehicle B. A strategy for figuring quickening of mounting point at body side for Vehicle B is introduced utilizing the recognized power train excitation powers and the estimated IPI at an interfacing point between vehicle body and mount. The separation proportions are approved utilizing the experiment, which confirmed the proposed strategies for evaluating confinement proportions of mounts.

PROBLEM STATEMENT

Optimization of weight has been very critical aspects of any design. It has substantial impact on vehicle performance, and in spin minimizes the emissions. This dissertation would focally point on the design gap offered by the element practical even as crucial the scenery and scope of the weight optimization more the areas acknowledged during design optimization.

IV. OBJECTIVES

- Modeling of tyre support mounting bracket of heavy vehicle in CATIA V5 software.
- Analyzing for stresses and deformation in Tyre support mounting of heavy vehicle.

 DOI : 10.35291/2454-9150.2020.0418

- To alter the existing material to aluminum alloy to reduce weight and determine stress and deformation.
- To perform experimental testing of new, optimize Tyre support mounting of heavy vehicle on UTM.
- Experimental testing and correlating results

V. METHODOLOGY

Step 1: - We started the work of this project with literature survey. We gathered many research papers which are relevant to this topic. After going through these papers, we learnt about tyre bracket mounting.

Step2: - After that the components which are required for my project are decided.

Step 3: - After deciding the components, the 3 D Model and drafting will be done with the help of software.

Step 4: - The components will be manufactured and then assembled together.

Step 5: -The testing will be carried out and then the result and conclusion will be drawn.

CATIA MODEL

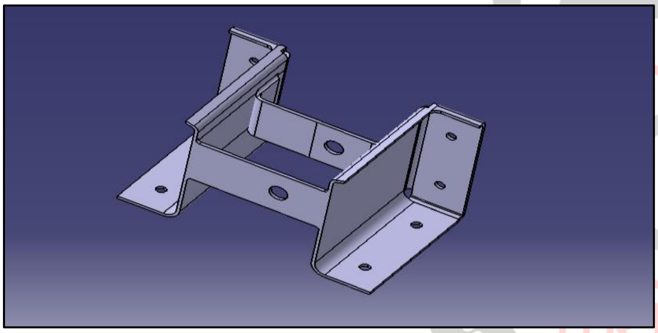


Fig 1 CATIA model of tyre bracket mounting

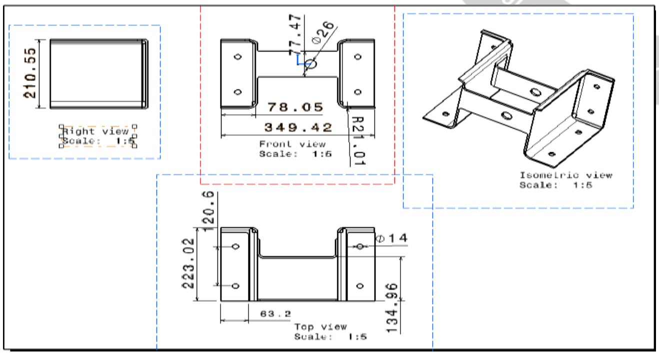


Fig. 2 Drafting of tyre bracket mounting

Material Properties

Properties of Outline Row 3: Structural Steel

	A	B	C	D
1	Property	Value	Unit	
2	Material Field Variables	Table		
3	Density	7850	kg m^-3	
4	Isotropic Secant Coefficient of Thermal Expansion			
6	Isotropic Elasticity			
7	Derive from	Young...		
8	Young's Modulus	2E+11	Pa	
9	Poisson's Ratio	0.3		

Table 1- Material properties of tyre bracket mounting

FINITE ELEMENT ANALYSIS OF TYRE BRACKET MOUNTING

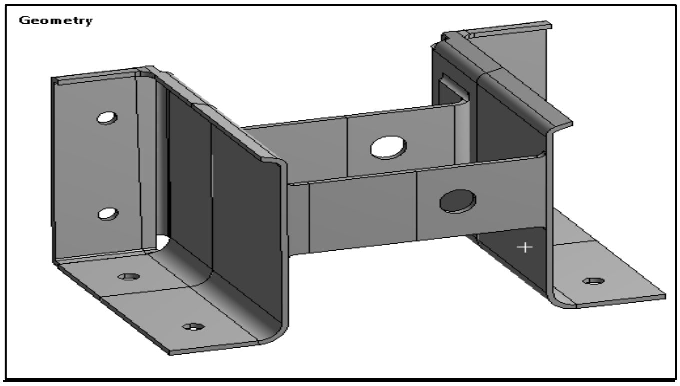
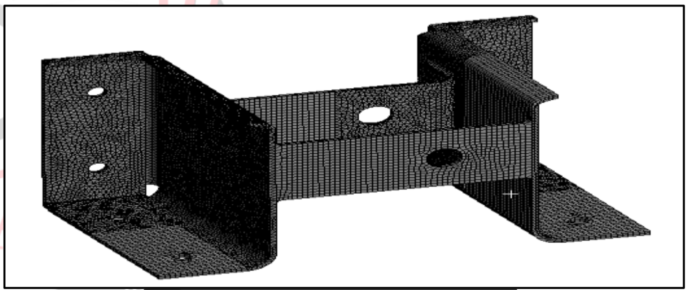


Fig. 3 Geometry of model imported in ANSYS

Mesh

ANSYS Meshing is a broadly useful, smart, robotized elite item. It delivers the most fitting lattice for exact, effective Multiphysics arrangements. A work appropriate for a particular examination can be produced with a solitary mouse click for all parts in a model. Full powers over the choices used to produce the work are accessible for the master client who needs to calibrate it. The intensity of equal handling is naturally used to decrease the time you need to hang tight for work age.



Statistics	
Nodes	222442
Elements	112147

Fig. 4 Details of meshing

Total no. of Nodes is 222442. Total no. of Elements is 1122147

Boundary condition

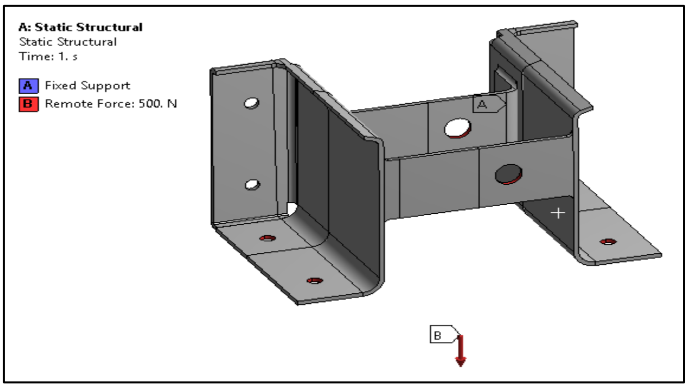


Fig. 5 Boundary condition of tyre bracket mounting

 DOI : 10.35291/2454-9150.2020.0418

Deformation results

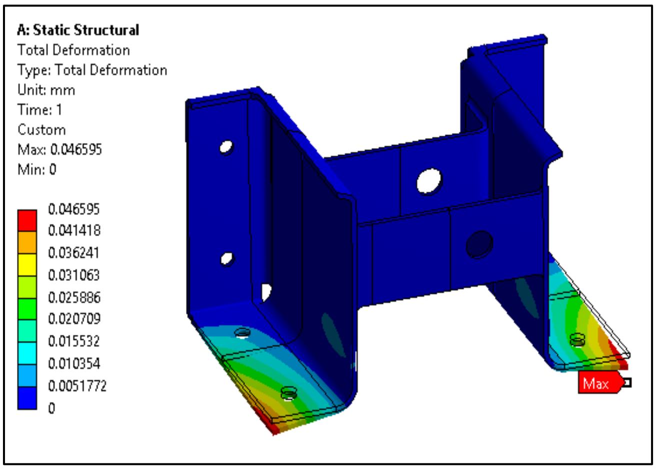


Fig. 6 Deformation results of tyre bracket mounting

Maximum deformation under static condition of tyre bracket mounting was 0.04mm

Equivalent stress

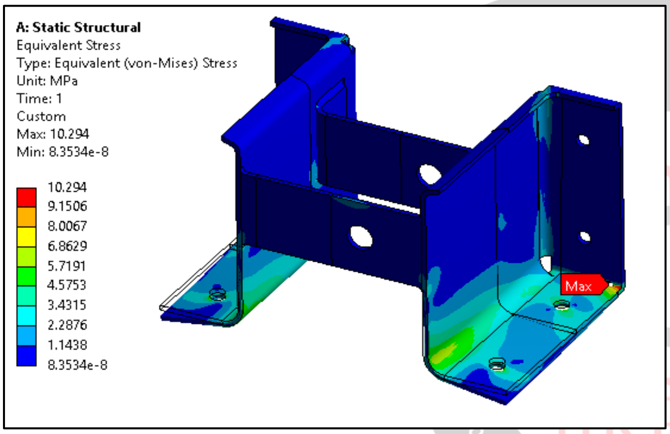


Fig.7 Equivalent stress results

Maximum Equivalent stress of original tyre bracket mounting was 10.94 MPa

Maximum Principal Stress

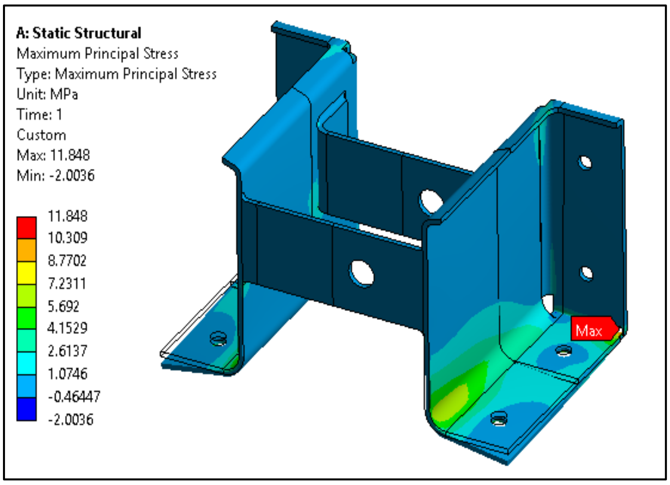


Fig.8 Maximum Principal Stress in Tyre support mounting

Maximum Principal Stress in Tyre support mounting was 11.84 MPa

Minimum Principal Stress

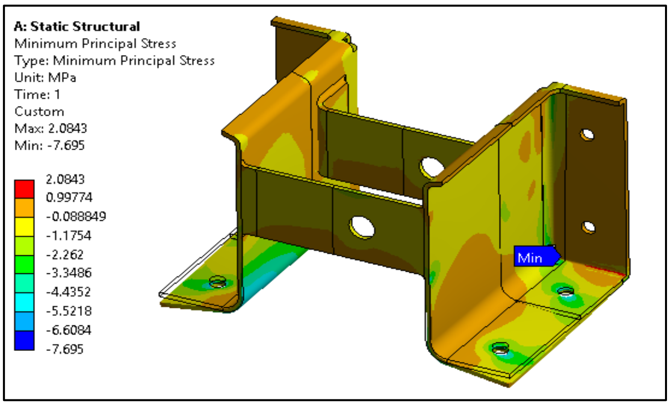


Fig.9 Minimum Principal Stress

FINITE ELEMENT ANALYSIS OF TYRE BRACKET MOUNTING AFTER REVERSE ENGINEERING

Engineering Material – Aluminium Alloy

Properties of Outline Row 4: Aluminum Alloy

	A	B
1	Property	Value
2	Density	2770
3	Isotropic Secant Coefficient of Thermal Expansion	
4	Coefficient of Thermal Expansion	2.3E-05
5	Isotropic Elasticity	
6	Derive from	Young's Modulu...
7	Young's Modulus	7.1E+10
8	Poisson's Ratio	0.33
9	Bulk Modulus	6.9608E+10
10	Shear Modulus	2.6692E+10

Table - 2 Aluminium material properties

Final result of optimized Aluminium material tyre bracket mounting

Total deformation

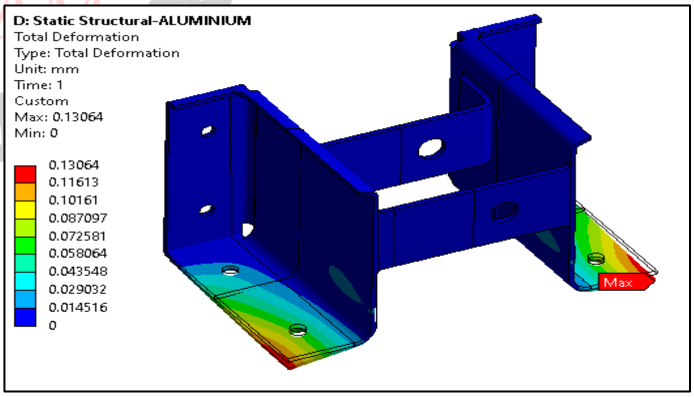


Fig. 10 Total deformation result

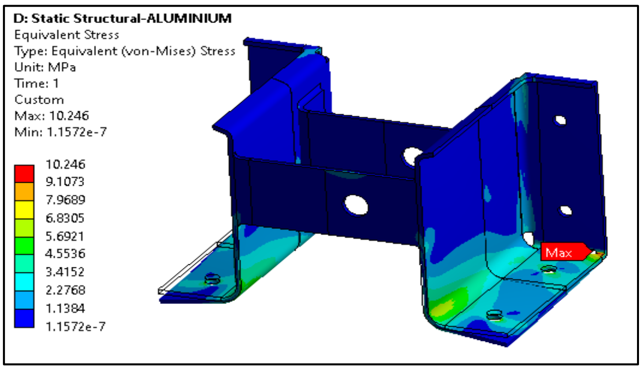


Fig.11Equivalent stress results

 DOI : 10.35291/2454-9150.2020.0418

FEA result

SR.NO	TOTAL DEFORMATION	MAXIMUM PRINCIPAL STRESS	MINIUMUM PRINCIPAL STRESS	EQUIVALENT STRESS
ORIGINAL DESIGN (steel)	0.04mm	11.84 MPa	7.27 MPa	10.29 MPa
OPTIMIZED DESIGN (aluminium)	0.13mm	11.79 MPa	2.13 MPa	10.24 MPa

EXPERIMENTAL TESTING:

A Universal Testing Machine (UTM) is employed to check both the tensile and compressive strength of materials. Universal Testing Machines are named in and of itself because they'll perform many various styles of tests on an equally diverse range of materials, components, and structures.

Universal Testing Machines can accommodate many forms of materials, starting from hard samples, like metals and concrete, to flexible samples, like rubber and textiles.

The UTM could be a versatile and valuable piece of testing equipment that may evaluate materials properties like strength, elasticity, compression, yield strength, elastic and plastic deformation, bend compression, and strain hardening. Different models of Universal Testing Machines have different load capacities, some as low as 5kN et al as high as 2,000kN.

In this experimental purpose we used 400KN load capacity UTM.

Specification of UTM

1	**Max Capacity**	**400KN**
2	Measuring range	0-400KN
3	Least Count	0.04KN
4	Clearance for Tensile Test	50-700 mm
5	Clearance for Compression Test	0- 700 mm
6	Clearance Between column	500 mm
7	Ram stroke	200 mm
8	Power supply	3 Phase , 440Volts , 50 cycle. A.C
9	Overall dimension of machine (L*W*H)	2100*800*2060
10	Weight	2300Kg

CONCLUSION

Original design and optimized design have stress value below yield limit of, Hence Design is safe.

Original components mass is 7.34 kg which is reduced to 2.59 kg i.e. total around 4.75Kg of weight is reduced which finally reduces product cost.

REFERENCES

[1] Qualification of additively manufactured aerospace brackets: A comparison between thermoelastic stress analysis and theoretical results Gloria Allevia, Marco Cibecab, Roberta Fiorettib, Roberto Marsilib, Roberto Montaninic, Gianluca Rossi, SAE International by University of Edinburgh, Thursday, August 09, 2018

[2] Optimization of Transmission Mount Bracket, BasemAlzahabi, Scott C. Simon, Logesh Kumar Natarajan, ISSSR 2007, may volume 4

[3] 3.Analysis of CFRP Flight Interface Brackets under Shock Loads, Sagar Inglea, Nikhil Tilakpurea, V. Narayanamurthy b, S.M.Hussaini, Materials Today: Proceedings 4 (2017) 2492–2500

[4] 4.Shape design of an engine mount by a method of parameter optimization, Joong Jae Kim and Heon Young Kim, compress a structures Vol. 65, No. 5, pp. 72s-731, 1997, 0 1997

[5] 5. Fatigue analysis of Package Terminal Air Conditioner motor bracket under dynamic loading, S.K. Loh a, W.M. China, Waleed F. Faris, Materials and Design 30 (2009) 3206–3216

[6] 6. Stress-based Topology Optimization with Discrete Geometric Components, ShanglongZhanga, Arun L. Gainb, Julián A. Norato

[7] 7.Structural Optimization for Engine Mount Bracket, Xiao-Yong Pan and DoniZonni, Guo-Zhong Chai, Yan-Qing Zhao and Cui-Cui Jiang, SAE International by University of Minnesota, Thursday, August 02, 2018

[8] 8. Stress-based shape and topology optimization with the level set method, R. Picelli, S. Townsend, C. Brampton, J. Norato, H.A. Kim, Comput. Methods Appl. Mech. Engrg. 329 (2018) 1–23

[9] 9.A method for estimating mounts isolations of power train mounting systems, Wu Qin, Wen-Bin Shangguan, Guohai Luo, ZhengchaoXie, Journal of Sound and Vibration 426 (2018) 278e295

[10] 10. Design method of automotive power train mounting system based on vibration and noise limitations of vehicle level, Wen-BinShangguan, Xiao-AngLiu, Zhao-PingLv, SubhashRakheja, Mechanical Systems and Signal Processing, 24 September 2014

[11] 11. Stress-based topology optimization using bi-directional evolutionary structural optimization method, Liang Xia, Li Zhang, Qi Xia, Tielin Shi.

 DOI : 10.35291/2454-9150.2020.0418