ISSN : 2350-0743

www.ijramr.com

International Journal of Recent Advances in Multidisciplinary Research
Vol. 10, Issue 09, pp.8879-8887, September, 2023

RESEARCH ARTICLE

OPTIMIZATION OF AUTOMOTIVE HOT END EXHAUST BRACKETS BY MODAL AND FREQUENCY RESPONSE ANALYSIS

[1],*Pylo, M.A., [2]Dr. Venkatesh P.R. and [3]Naseerahusen Yatageeri

[1]Mtech. Scholar, Department of Mechanical Engineering, R V College of Engineering, Bengaluru
[2]Associate Professor, Department of Mechanical Engineering, R V College of Engineering, Bengaluru
[3]Sr Engr. FEA Department, Faurecia India Private Limited, Bengaluru

ARTICLE INFO

Article History:
Received 08th June, 2023
Received in revised form
20th July, 2023
Accepted 16th August, 2023
Published online 30th September, 2023

Key Words:

Modal Analysis, Frequency Response, Modal Correlation, MAC Criterion, Tuning.

ABSTRACT

Exhaust system design has become a tricky job when analyzing the structural integrity of a marketable vehicle where its hot end components like catalytic converters are the most popular exhaust emission control devices. The T6 hot end exhaust model is modelled in FEM process where Hypermesh is used as a pre-processor, Abaqus as a solver. Shell and solid elements were utilized to provide better results in the modal frequency domain. The 3-point and 6-point engine excitation is considered towards the COG of the engine. Both Engine and gearbox control groups are coupled to the end of the roll axis nodes by RBE2 rigid elements, creating a boundary condition where the middle node of the roll axis is confined in all degrees of freedom, along with the input of Flex which is fixed in 3 degrees of freedom. The frequency response is determined for engine order 2 by the vibration amplitude of the exhaust system (displacement vs. frequency graph) for different locations on the hot end of the exhaust system to determine the peak frequencies of the base and optimized model. Further experimental tests were set up using an FFT (Fast Fourier Transform) analyzer. The accelerometers were installed to recognize the excitation and the response points. The maximum displacement occurred at two peaks 187Hz and 211Hz at an amplitude of 0.19mm. Here MAC criteria and slope of displacement plots were 16% varying with that of the simulation data. Furthermore, tuning was performed to match the respective frequency and displacement values.

INTRODUCTION

The exhaust system design takes much higher calculation times and large data requirements, along with proper knowledge of the test track profile, which is often not available in the early phases of the project. Due to the unavailability of this information, in various cases, simplistic, experience based static approaches are used that do not provide us with real dynamic behavior, thereby leading to a reliance on engineering and computational data (Jian Min Xu *et al*., 2014).

Modal analysis of exhaust system: The suspension device transmits the exhaust system's vibration energy to the underbody, causing the vehicle's body to vibrate and make noise. In Modal analysis, we observe the dynamic characteristics, which can state the inherent properties of the structure and provide a platform for performing the vibration analysis, fault diagnosis and prediction, and optimization design of the exhaust system structure. Using modal analysis, it is possible to define a structure according to its inherent qualities or characteristics, such as damping, frequencies, and mode shapes.

Corresponding author: Pylo, M.A.,
Mtech. Scholar, Department of Mechanical Engineering, R V College of Engineering, Bengaluru.

The associated eigenvectors define a set of modal degrees of freedom (Patil V *et al*., 2018). Harmonic analyses are used to investigate how the exhaust structure reacts to harmonic excitations of various frequencies. The response is shown graphically by a displacement-frequency curve. The solution for every excitation frequency is a linear combination of these modes. The highest loading of the exhaust system will occur at resonance frequencies. Frequency response analysis displays the response of the system in the frequency domain and estimates whether the design optimization of the structure can withstand the problems of resonance, fatigue, and other adverse effects caused by forced vibration (Xiong, J, 2023). The engine, braking, and, most importantly, the road cause vibrations in the automotive exhaust system. Dynamic steady state simulation and subsequent damage calculations are state of the art in terms of simulation methodology in the case of engine idling and vehicle comfort (Shenghao Xio *et al*., 2023).

Frequency response of exhaust system: Harmonic analyses are used to investigate how the exhaust structure reacts to harmonic excitations of various frequencies. The response is shown graphically by a displacement-frequency curve. The engine's structural vibrations and periodic road vibration brought on by the roughness of the road surface are the sources of the exhaust system's excitation The associated eigenvectors define a set of modal degrees of freedom. The solution for every excitation frequency is a linear combination of these modes. The highest loading of the exhaust system will occur at resonance frequencies (M.H. Shojaeifard *et al*., 2017). The expression of the real mode frequency response function of excitation at point p and response at point l for a damping system with a multi-excitation point structure and N degrees of freedom is as follows

$$H_{lp}(\varpi) = {}^{N}\sum_{r=1} 1/K_{er}\,[1-\varpi^2_r(1-\varpi^2_r)^2+g^2_r+j-g_r(1-\varpi^2_r)^2+g^2_r]$$

Where K_{er} is the rth order equivalent stiffness and $K_{er} = K_r/(\varphi_{lr}\varphi_{pr})$, g_r is the damping ratio of the r^{th} order modal structure and $\varpi_r=\omega/\omega_r$

Natural Frequency: The real frequency curve's intersection with the residual compliance line or the frequency corresponding to the virtual frequency curve's peak when $\omega = \omega_r$ can both be used to calculate the natural frequency. The peak is easier to identify since it is sharper.

Damping ratio: When ω approaches ω_r, ω_r takes the lead and is said to as the major mode. Other modes that are close to ω_r but have poor coupling or that are far from ω_r can be roughly represented by the complex constant H_c.

HOT END EXHAUST MODEL

Cad Geometry of the Hot End: The 3D CAD Geometry of the T6 Automotive exhaust Model shown in Fig 1 is taken after modelling in CATIA V5 tool suspension device transmits the exhaust system's vibration energy to the underbody.

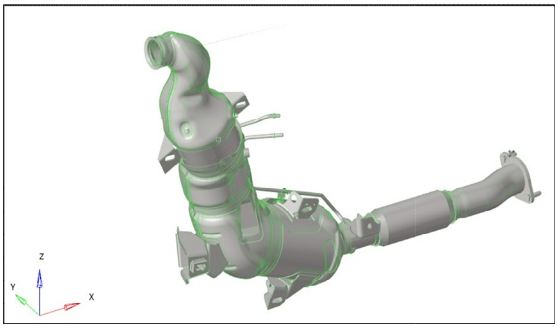


Fig.1. CAD geometry of exhaust system hot end

FE Model of the Hot End: The Finite Element Model of the Hot end exhaust system is developed using Hypermesh 2021 tool along with closely coupled Engine, Powertrain and Gearbox Components as shown in the Fig 2 The hot end exhaust system, which was pre-processed using the Hypermesh 2021 software and a solver like Abaqus and a viewer like Hyper View, is the subject of this study. It has 2182949 nodes and 3027557 elements. Most of the components of the FE Model were created using 4 node quad shell elements, while the Powertrain and Gearbox was created using tetrahedral elements. Modelling is done using 8 node hexahedral elements to represent the mat, substrate, and flanges. Modelling of the weld between the components uses shell pieces with a 2.5 mm thickness. Between solid and shell parts is modelled a shell coat with a thickness of 2.5 mm. Using rigid elements (RBE2), the connection between flanges (bolt holes) is modelled.

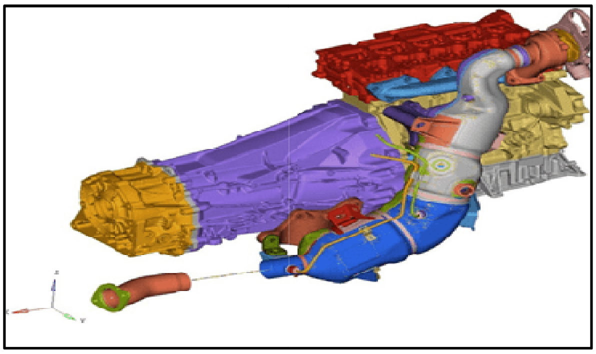


Fig.2. FEM model of the hot end

BOUNDARY CONDITIONS

The Fig 3 shows the Boundary conditions of the base Model where in the 3-point engine excitation is considered towards the COG of the engine. Engine and gearbox control groups are coupled to the end of the roll axis nodes by RBE2 rigid elements, creating a boundary condition where the middle node of the roll axis is confined in all degrees of freedom, along with the Input of Flex which is fixed in 3 degrees of freedom.

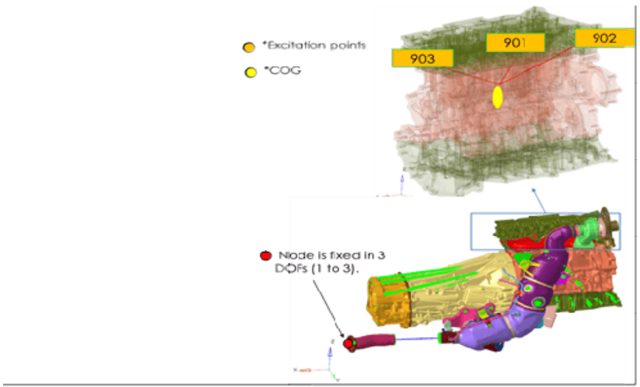


Fig. 3. Boundary conditions of base model

MODAL ANALYSIS OF HOT END EXHAUST SYSTEM

As we are dealing with the hot end, we are initially finding the nodal temperature plots with respect to different views shown in Fig 4 are given as an input to perform Modal analysis in order to define the properties of the Hot end with respect to the parent zone and weld affected zones of the model.

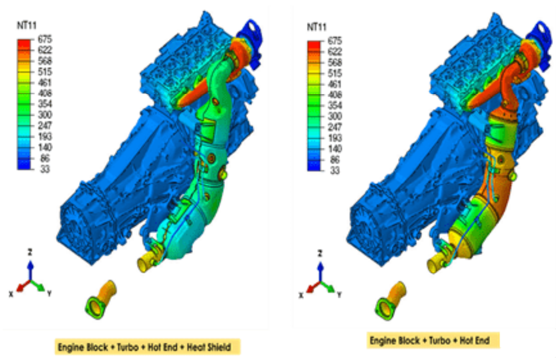


Fig. 4. Heat transfer plots

The eigen frequencies are extracted for different set of modes and their mode shapes are noted down in Table 1. [(*Max RPM=4500rpm) f ≥1.1*n/2*rpm max/60 = 165Hz.)]

Table 1. Modal Analysis details

Mode No	Eigen Frequency in Hz	Mode Shape
1-9		Rigid Body Modes
10	28.46	Flex Mode (1st Lateral Mode)
11	28.69	Flex Mode (1st Bending Mode)
12	67.87	Flex Mode (1st Twisting Mode)
13-17	78.13	Flex Modes
18	178.91	Pressure pipe (PP) Mode
19	183.72	Hot End (HE) +Heat Shield (HS) Mode
20	188.09	Flex + PP-Mode
21	190.20	PP +Turbo Mode
22	191.74	HE+ HS Mode
23-25	194.55	HS+PP Mode
26	202.51	HS+Flex Mode
27	207.54	HS +Turbo Mode
28	234.17	HS + Turbo Mode
29	238.76	HS+PP Mode
30	239.18	HS+Turbo Mode
31-32	250.07	HS+HE Mode
33-34	291.54	Flex Mode
35-37	250.07	HS+HE Mode
38-43	291.24	HS+Flex Mode

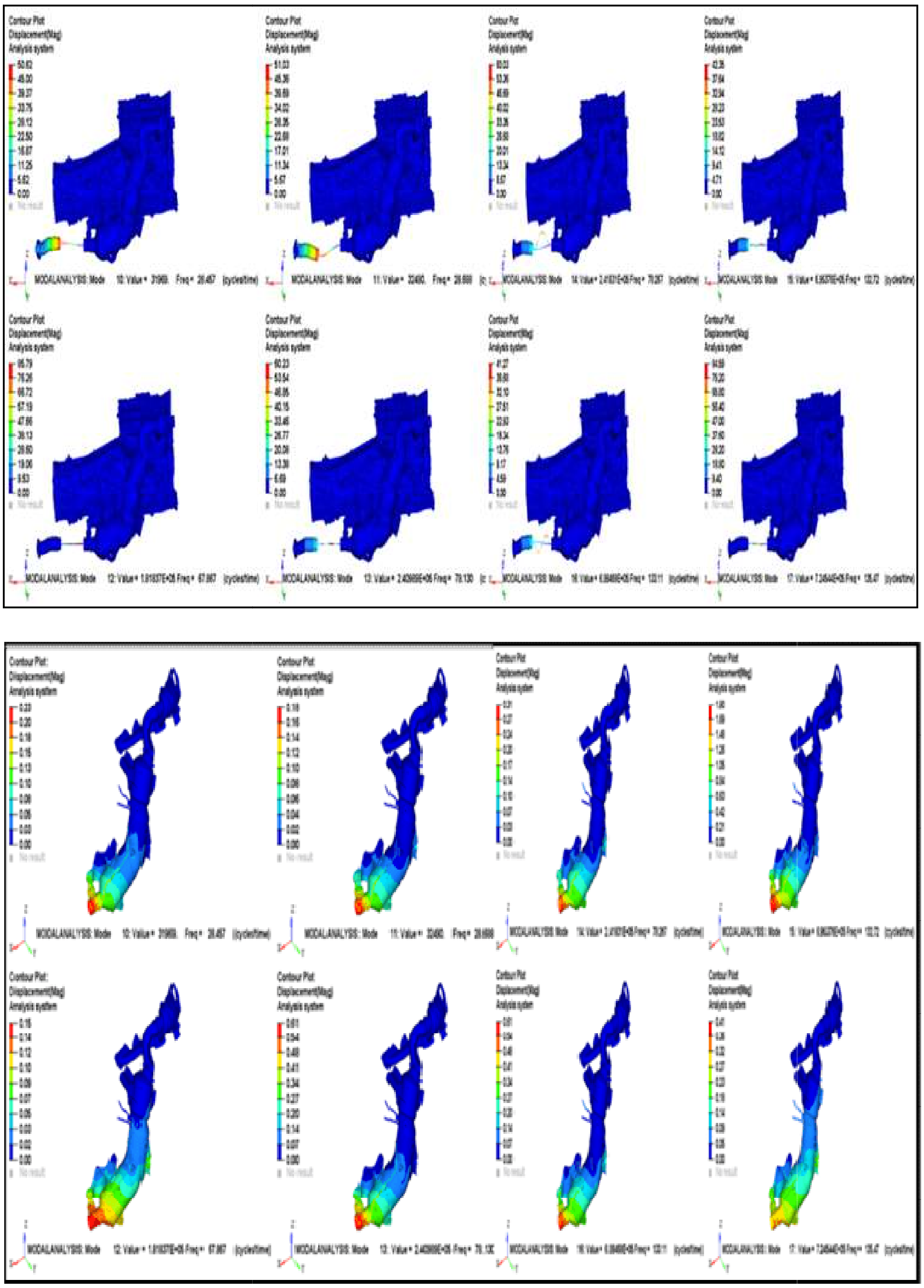


Fig.5. Mode shapes

Here we obtain 183.72Hz (First hot end mode) which is well above design frequency. The Mode shapes showing the first axial, bending and twisting mode is shown in Fig 5.The purpose of the confined mode is to mimic the exhaust system's constrained mode in its actual installation state. The powertrain's involvement will not only make it difficult to simulate the exhaust system's actual limits, but it will also have an impact on analyses of the exhaust system's dynamic performance. Therefore, the manifold linking the engine and the exhaust system is considered, and the powertrain system is taken into consideration in the exhaust system constraint modal analysis. The first Bending, axial and Twisting mode is shown and the Flex modes coupled with the hot end with their respective frequencies are also represented here.

FREQUENCY RESPONSE ANALYSIS OF HOT END EXHAUST: Initially in FRA1 the result evaluation is done for the critical regions which includes brackets, heat affected zones (HAZ), welds, etc. Then after obtaining the peak values, it is then made applicable for all the nodes and elements of various components of entire the system where results include displacements at the specific nodes and the stresses for the elements at peak frequencies. While performing the FRA Analysis the Input excitation data should be provided to the 3 co-ordinates of the engine excitation points which are obtained from the test model in the form of Displacement vs Frequency plots as shown in Fig 6 Their base motions for 3 degrees of freedom is defined.

Plotting Frequency vs Stress and Frequency vs Displacement plots: The Frequency vs Stress and Frequency vs Displacement plots for all the components of the system gives us the peak frequency at which the maximum stress is acting on the entire system as shown in Fig 7. The Frequency vs Displacements plots for all the components in x, y and z directions of the system gives us the peak frequency at which the maximum displacement occurs in all the three directions acting on the entire system.

Obtaining the stress plots for the peak frequency: The respective stresses for the peak frequency of 150Hz achieved are represented as shown in figure 4.9e with the color coding. Hence from the stress plots we can conclude that the brackets are failing due to stresses and does not meet the safety factor requirements with respective to their material properties assigned. Hence there is room for design optimization of the brackets that are failing due to excessive stresses.

OPTIMIZATION

As the starting point for the design, the best fitting approximate model's optimal solution was used. Then, the FE model-based optimization analysis may significantly lower the number of iterations and discover the global best solution quickly. The topology and design optimization of the bracket procedure takes place through the following parameters involving the FE model developed for analysis in Optistruct OSS Smooth while observing the Modal frequency and Thermo mechanical targets and checking the design objectives and constraints. This bracket developed is further checked for feasibility.

Alternative 1: Connecting rod replaced by Mixer Bracket.

Alternative 2: Outlet pipe Casting Bracket replaced with cost saving stamped bracket.

Alternative 3: Flaps added on Bracket and Bracket replaced with connecting rod.

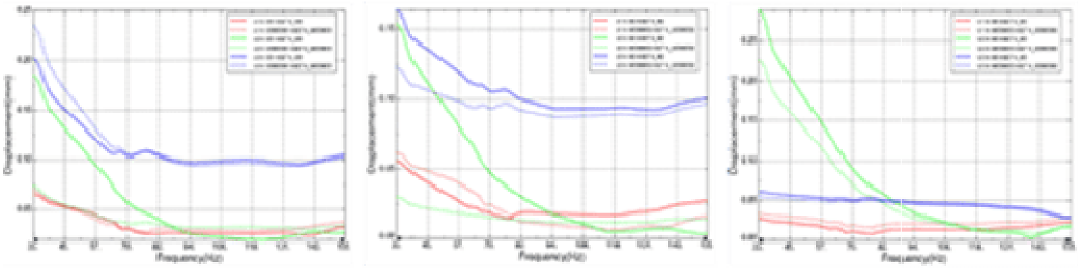
By allowing logically defined criterion relationships and weightings and by evaluating new concepts against selected reference solutions per criterion rather than against discrete products, the goal is to overcome the several known drawbacks through the Pugh Matrix shown in Table 2 Fig 9 shows the boundary conditions of the optimized model where in the 6-point engine excitation is considered towards the COG of the Engine and gearbox control groups are coupled to the end of the roll axis nodes by RBE2 rigid elements, creating a boundary condition where the middle node of the roll axis is confined in all degrees of freedom, along with the Input of Flex which is fixed in 3 degrees of freedom. These boundary conditions are followed according to the Global standards for a longitudinal engine. It can be agreed that the first hot end frequency is well above the design frequency which is obtained analytically. First 1 to 18 Modes are Rigid body modes The minimum resonant frequency should be above the following limit: [(*Max RPM=4500rpm) f ≥1.1*n/2*rpm max/60 = 165Hz. Here we obtain 181.11 Hz (First Hot End Mode)] which is well above the minimum resonant frequency. In FRA analysis the input excitation data as indicated in Fig 10 should be provided to the 6 co-ordinates of the engine excitation points which are obtained from the test model in the form of Displacement vs Frequency plots. The base motions for the 3 excitation points are defined for 3 degrees of freedom. The Frequency vs Stress plots in Fig 11 shows the peak frequencies at which the maximum stress is acting on the entire system

EXPERIMENTAL TEST SETUP

Using an FFT (Fast Fourier Transform) analyzer, the experimental validation is carried out. The input signal is sampled by the FFT spectrum analyzer, which then determines the sine and cosine component magnitudes and shows the spectrum of these determined frequency components. All the excitation locations where accelerometers are to be installed were noted as shown in Fig 12. Additionally, the response points are identified and highlighted depending on the structure 8 Excitation points and 8 response points are taken with the help of the accelerometers that are placed accordingly on the engine, hot end, gearbox, power train etc. With the excitation on these points, we obtain the response in plots of Frequency vs Amplitude. We can also relate the Engine rpm to the plots. These curves provide the peaks for maximum magnitude. For the purpose of correlation, we have taken the maximum displacement occurring in the test data occurred at two peaks at 187Hz and 211Hz.These peaks are as shown in the Fig 13.

Modal Assurance Criterion: According to Equation 1, the MAC value between two modes is effectively the complex modal vector's normalized dot product at each of their common nodes (i.e., points). In another words the square of the correlation between two modal vectors, φr and φs

$$MAC(\{\varphi r\}\{\varphi s\}) = \frac{|(\{\varphi r^{*t}\}\{\varphi s\})^2|}{(\{\varphi r^{*t}\}\{\varphi r\})(\{\varphi s^{*t}\}\{\varphi s\})}$$



*	Co -ordinates of Excitation points		
Node No	x	y	z
901	1532.936	-142.023	1045.366
902	1522.864	130.166	1110.827
903	1983.463	20.464	1039.368

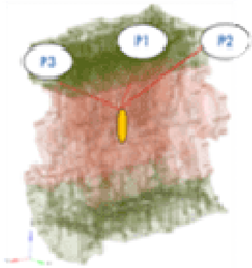
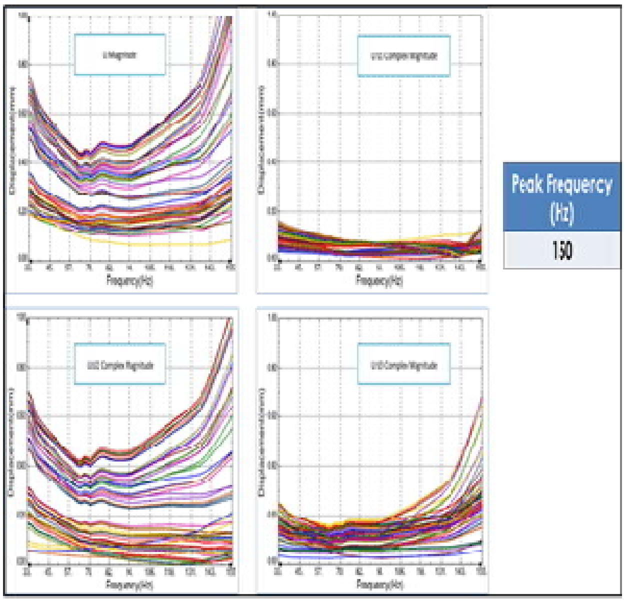


Fig. 6. 3-point Excitation details (base model)



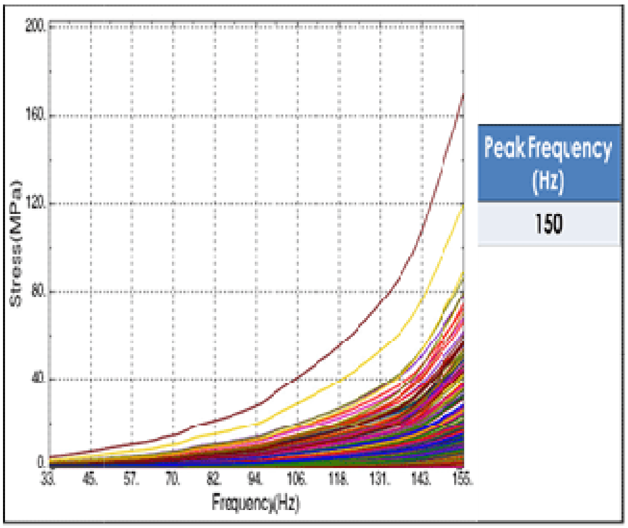


Fig. 7. Frequency vs Stress & Frequency vs Displacement plots (base model)

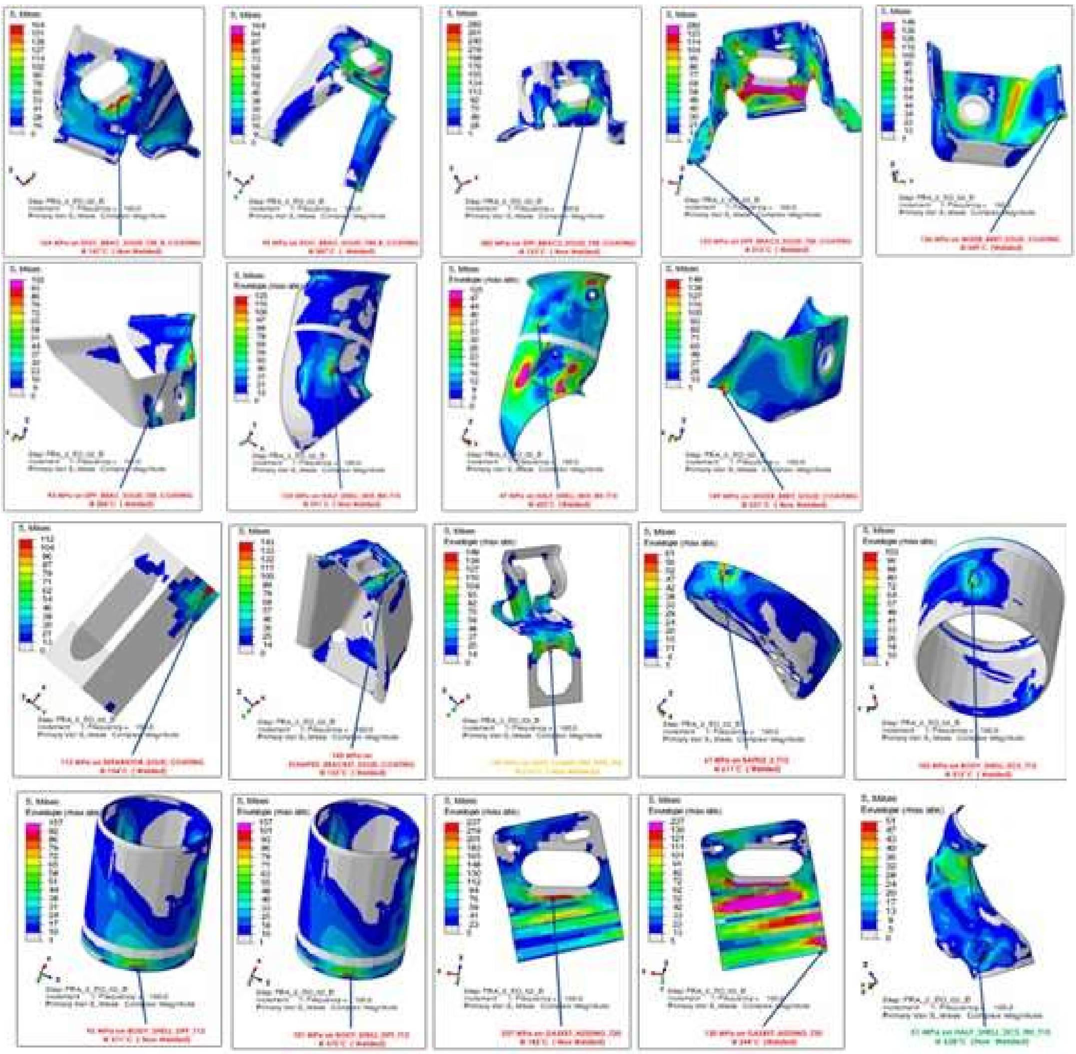


Fig 8. Stress plots

Table 2. PUGH Matrix

Sl No	Criterias	Variables			Total
		Alt1	Alt2	Alt3	
1	Machinability	1	0	0	1
2	Cost	-1	1	1	1
3	Durable	0	1	1	2
4	Stiffness	1	1	1	3
5	Weight	1	-1	1	2
6	Feasibility	1	0	1	2
7	Ease of Assembly	1	1	0	2
	Total	4	3	5	12
	Rank	2	3	1	

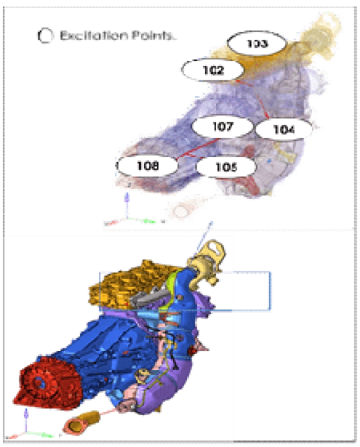
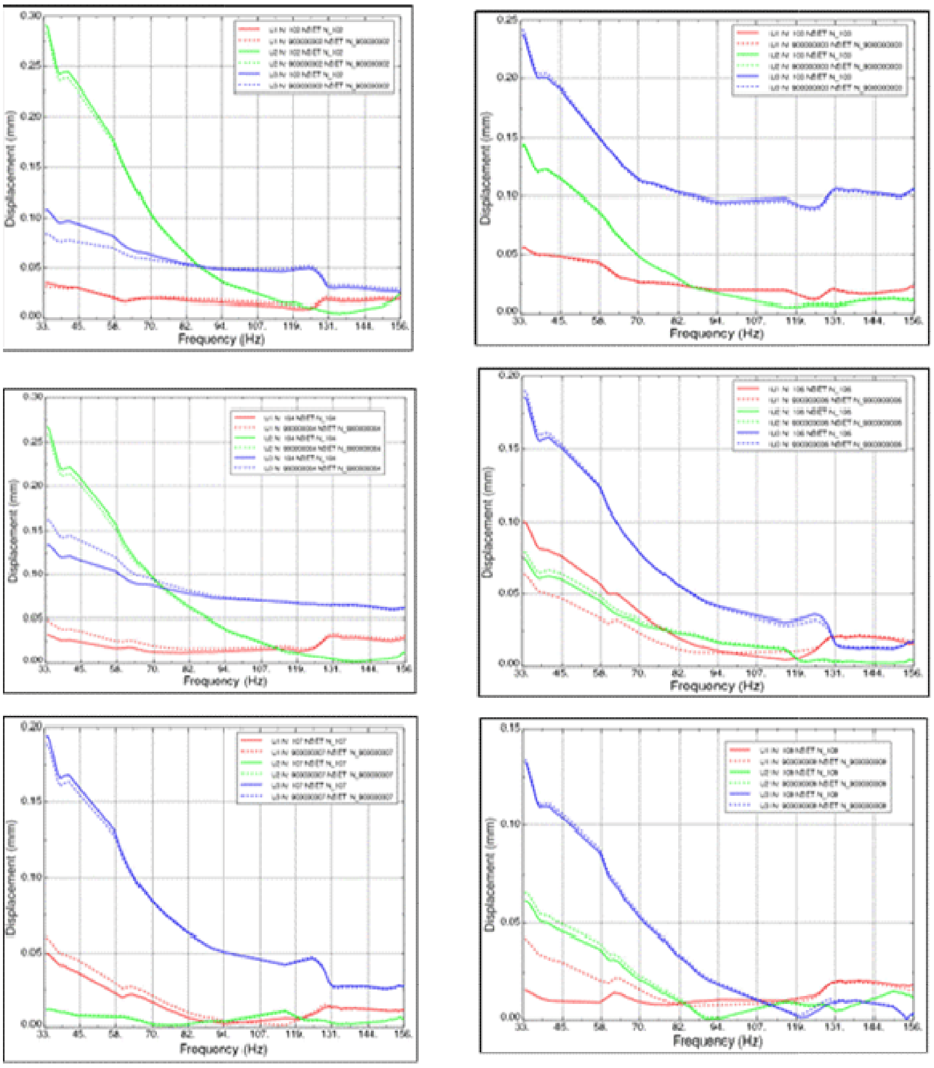


Fig. 9. Boundary conditions of optimized model



*	Co –ordinates of Excitation points		
Node No	x	y	z
102	1989.642	71.571	1044.729
103	1522.864	-128.528	1045.639
104	1767.982	133.111	697.639
105	2243.972	145.421	696.859
107	2082.312	148.221	811.919
108	2546.422	96.221	709.709

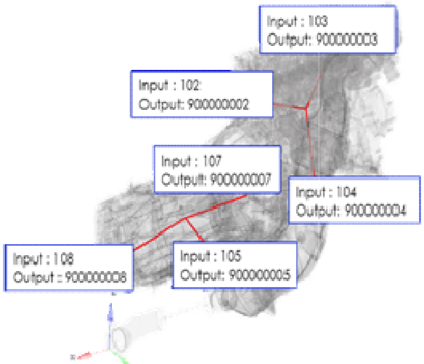
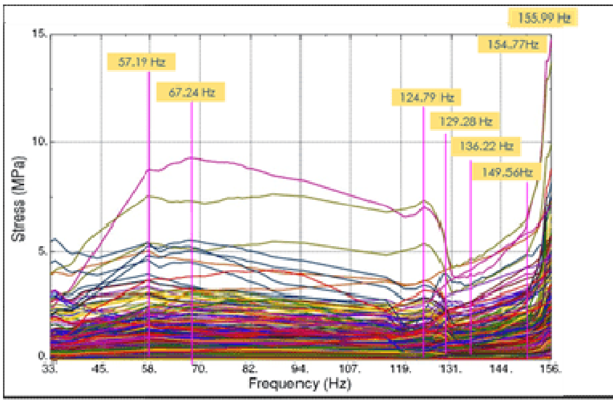


Fig. 10: 6-point excitation details (optimized model)



Peak Frequency (Hz)
57.19
67.24
124.79
129.28
136.22
149.56
154.77
155.99

Fig 11. 6-point excitation details (optimized model)

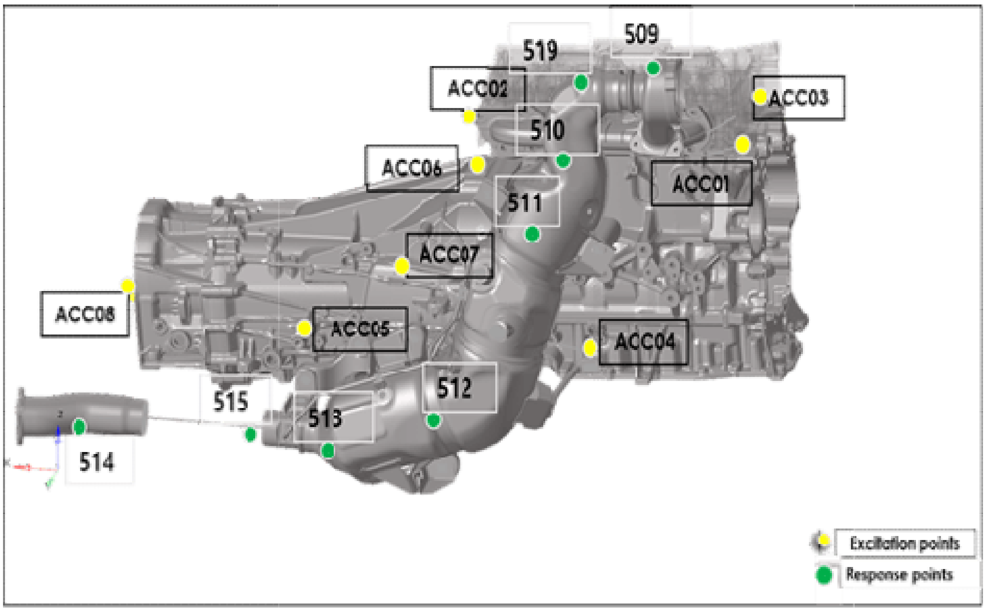


Fig. 12. Experimental test set up

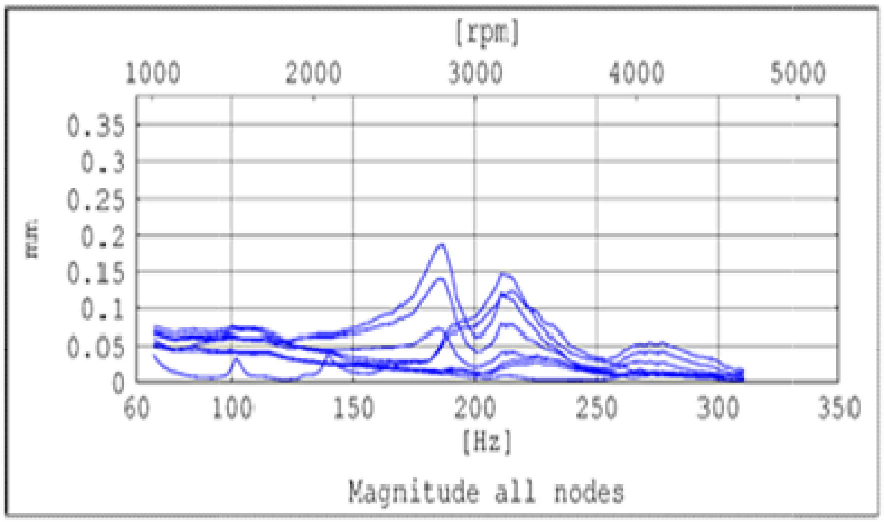


Fig.13. Experimental results

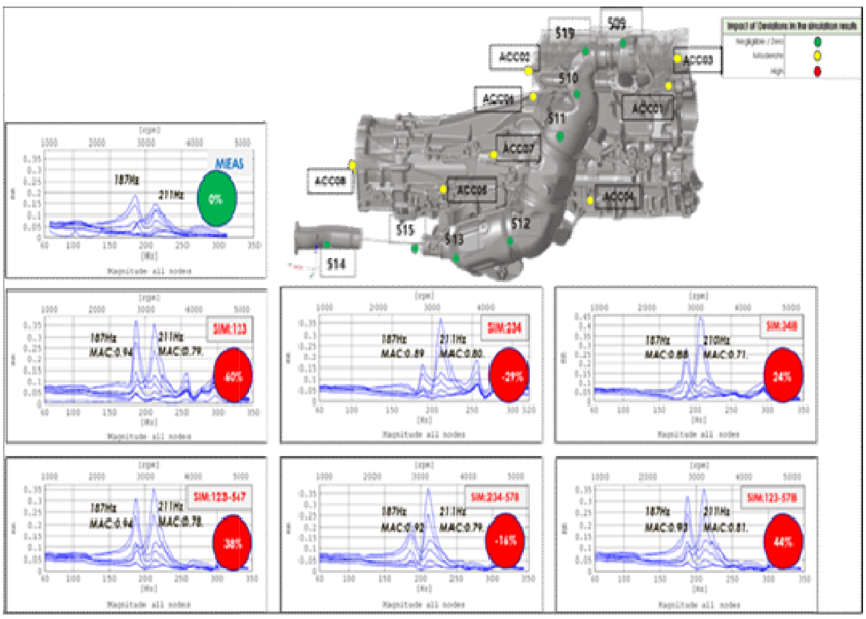


Fig. 14. Test co-relation

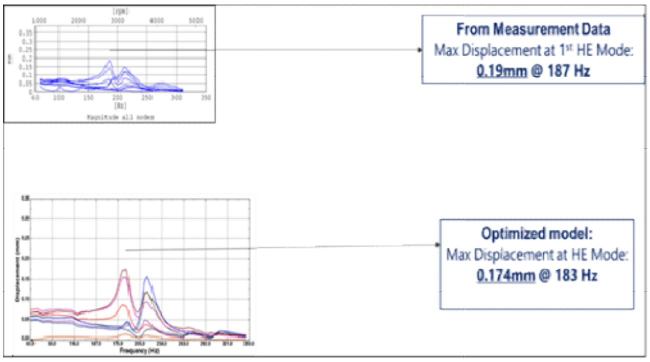


Fig. 15. Test data and simulation data comparison

CO-RELATION: In order to corelate the simulation data with the test data we need to make sure that we choose the proper combination of the excitation points that has minimal deviation from the test data as shown in Fig 14, and it can be done through comparing the differences in the displacement values and MAC (Modal Assurance criterion) values from Frequency vs Displacement. Here SIM indicates the simulation data and the number of excitation points in the combination that is selected. Both 3 point and 6-point excitation simulation data are compared. We found that the 6-point excitation combination of SIM: ACC (234-578) with a variation of about 16% with a MAC Value of 0.92 to be the most complimenting the test data. Hence, we proceed further with this combination of simulation data.

Tuning Parameters: On comparing the simulation data with the test results as shown in Fig 15, we observe that there is scope for fine tuning. After set of tuning iterations with respect to different parameters we can see that there is considerable change in the frequency as there is an increase in the mass of the flex and considerable change in the displacement value as there is an increase in the modal damping factor. After the iterations we obtain the desired frequency value at amplitude of 0. 19mm.The tuning parameters are:

- Mass (Add/remove of the mass to achieve the target frequency)
- Flex stiffness + Flex structural damping (Increase/decrease stiffness of flex to achieve the target frequency and check the model behavior)
- Modal Damping (Increase/decrease the damping percentage to achieve the target displacement amplitude)
- Scaling of the Excitation data

CONCLUSION

In order to address the issue of excessive interior and exhaust vibration that is transferred to the T6 hot end brackets when the engine is idle, the vibration characteristics of the exhaust system are analyzed using the theory of modal parameters and frequency response.

The first hot end modal frequencies were found to be 181Hz and 183Hz which are well above the design frequency obtained analytically and their mode shapes were extracted. Frequency vs stress and Frequency vs Displacement plots were obtained to determine the peak frequencies of the base and optimized model for which the stress plots are taken to provide conclusive proof for optimization. After corelating with the test data, it is found that there is a variation of 16% with MAC value of 0.92 with the experimental data. These variations are then fine-tuned with different tuning parameters to match 0.19mm amplitude at the frequency of 187Hz. Further Engine order 4 or more firing orders can be utilized to conduct the analyses and it is also possible to obtain the thermomechanical failure results to obtain the life estimation and damage occurring on the hot end of the exhaust system.

ACKNOWLEDGMENTS

Special thanks to Mr. Udayakumar Shivakanth, (Deputy Manager, CAE Department, Faurecia India Pvt Ltd, Bangalore) for his productive discussions and inputs to the research. Furthermore, the authors acknowledge all the support offered by Faurecia India Pvt Ltd, Bangalore

REFERENCES

Jian Min Xu, Shuiting Zhou, Shui xuan Chen. 2014. An Analysis of the Vibration Characteristics of Automotive Exhaust Systems and Optimization of Suspension Points. *TOMEJ.*10(2174): 574–580.

Patil V, Khan S.N. 2018. Modal Analysis of Passenger Car Exhaust System and Validation with Physical Test.*Techno-Societal Springer, Cham*:207-214.

Xiong, J.2023Study on optimization of vehicle exhaust system noise and hook position based on multi source experimental data analysis. SN Appl. . Sci. 2023; 5(57):–9

Shenghao Xio, Si Cheng. 2023 Research on noise control of automobile exhaust system under idling condition based on modal analysis. *Advances in Mechanical Engineering*15(1):1–17.

Shojaeifard, M.H., Ebrahimi-Nejad, S., Kamarkhani, S. 2017. Optimization of Exhaust System Hangers for Reduction of Vehicle Cabin Vibrations. *International Journal of Automotive Engineering*. 7(1): 2313-2324

****