



e-ISSN: 2319-8753 | p-ISSN: 2347-6710

# IJRSET

International Journal of Innovative Research in  
**SCIENCE | ENGINEERING | TECHNOLOGY**

# INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

Volume 12, Issue 4, April 2023

**ISSN** INTERNATIONAL  
STANDARD  
SERIAL  
NUMBER  
INDIA

Impact Factor: 8.423

9940 572 462

6381 907 438

[ijirset@gmail.com](mailto:ijirset@gmail.com)

[www.ijirset.com](http://www.ijirset.com)



# Design and Optimization of Dashboard Holding Fixture

Vivek Dilip Patil, Dr. Kumar. D. Sapate, Priyanka B. Sawant

Department of Mechanical Engineering, Alard College of Engineering and Management, Pune, Maharashtra, India

Professor, Department of Mechanical Engineering, Alard College of Engineering and Management, Pune,  
Maharashtra, India

Asst. Professor, Department of Mechanical Engineering, Alard College of Engineering and Management, Pune,  
Maharashtra, India

**ABSTRACT:** Dashboard holding fixture is the frame, which supports to Cockpit of vehicle while physical testing phase during car development. Fixture is used in some important physical test like vibration test, static indentation test etc. Out of them vibration test is very crucial test and in this test the design and weight of fixture is plays an important role. If the weight of fixture is more than specific range and 1st natural frequency of fixture is less, then it adds adverse effect in vibration test. The weight of fixture is tried to maintain near to 80-120Kg and first natural frequency of fixture is should more than 95Hz for better results we are trying to maintain first global mode as max as possible. Design of holding fixture is done in various phases; initially a rough frame is prepared. CAE test is carried out on simple frame to check its frequency and vibration modes. CAE outputs are analysed to get an idea to improve the stiffness of frame. The design of holding fixture is initiated by some internal standard and then its basic results are analysed. In proto level, the weight, frequency and mounting are checked. The results of proto test are analysed and team decides further changes, improvements in design and these changes are implemented in build level. Build level model is last stage of design. The results of build model is almost near to our goal and then physical frame is manufactured.

**KEYWORDS:** FEA, Dashboard Holding Fixture, Optistruct analysis, Frequency & von-mises stresses.

## I. INTRODUCTION

Now a day car are introduced with high technological advantages, light in weight , and faster speed were achieved and effect of it serious accidents are happen. Car and automotive product Manufacturers became more concerned with improving car safety and preventing injuries and deaths. Occupant safety is most important and crucial point while developing new car. There are lot of assembly level and car level testing needs to pass then and only then car is approve for launch in market. Development of new car program needs lot of testing. There are two type of testing methods are used, one is FEA testing and other is physical testing. FEA is most use tool for testing of various tests in development phases. Vibration test is one of them and holding fixture is use in vibration test. Holding Fixture is the metallic frame, which support to the Dashboard in vibration test. Dashboard assembly with car body is kept on holding fixture and this mixture is mounting on vibrating plate of vibration test machine. In vibration test the vibrating plate of vibration machine is vibrated and this vibrations are transfer to dashboard through holding fixture. Car body with dashboard assembly is mounted on fixture therefore; the design of fixture is more bulky and rigid to sustain vibration load and weight of car body with dashboard.

## II. RELATED WORK

From the study done by Ajay Varma Sagi & Darshan Bandihalli Mahalingaiah in “Design and Analysis of a Generic Fixture for Physical Squeak and Rattle Prediction”. The fixture is failing to meet a few requirements set at the connection components due to the adjustment mechanisms, it is now of interest to explore mechanisms which would avoid the connection components to extend too much away from the main support structure thereby satisfying those requirements. Even though Aluflex provides good flexibility it is known to exhibit less stiffness, it could be interesting to check for alternatives to Aluflex that provide good flexibility without compromising too



much on stiffness. A parametric or size optimization can be performed on the beam structure of the fixture to check if all the requirements on the stiffness could be satisfied. But it is unlikely for the requirements to be satisfied with this method since the overall stiffness of the fixture is already higher than that of the BIW. With the current setup of topology optimization, load paths are generated behind the connection components which is transformed into a manufacturable design. The design is then adapted to accommodate adjustment mechanisms on it, this accommodation of additional parts would cause for less stiffness to be exhibited by the final fixture at the connections. We understood that, only fixture and Dashboard assembly is not sufficient for the Vibration test setup. Fixture with BIW plays major role in the test and also it will be very close correlation on actual scenario of the vehicle condition on road.

### III. METHODOLOGY

- Step 01: Concept preparation of a fixture.
- Step 02: Cross section selection for Holding Fixture.
- Step 03: Creation of 3D CAD model.
- Step 04: Load and Boundary condition definition.
- Step 05: Natural frequency analysis and Structural analysis of Fixture.
- Step 06: Design optimization based on CAE results.
- Step 07: Manufacturing of Fixture.

### IV. EXPERIMENTAL RESULTS

Holding fixture frame experiences bending load and therefore frame should sustain vibrating load without any failure.

Some key points which to select cross section for Fixture frame -

1. The bending strength of cross section should good.
2. Heavy load is placed over fixture therefore; fixture frame should sustained heavy load.

FEA test is carried out on five different specimens to check natural frequency. It helps to decide cross section. Polar moment of inertia is one of most important factor to select cross section. The Polar Moment of Inertia is a geometric property of a cross section. Physically, it is a measure of how difficult it is to turn a cross-section about an axis perpendicular to it (the inherent rotational stiffness of the cross-section). This means that greater the Polar Moment of Inertia, the more torque is require to turn the shaft by a certain angle. Section modulus is also major factor to select cross section. Section modulus is a geometric property of the cross section used for designing beams and flexural members. It does not represent anything physically. To define section modulus, it may be defined as the ratio of total moment resisted by the section to the stress in the extreme fibre, which is equal to yield stress.

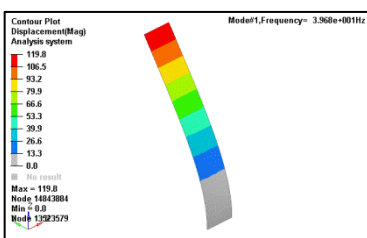


Fig. No. 1.1. Natural frequency of Plate specimen with Mild steel

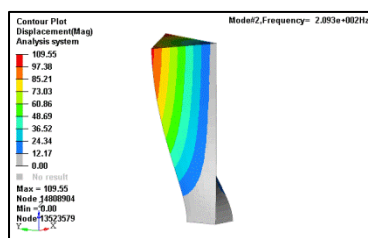


Fig. No. 1.2. Natural frequency of Angle specimen with Mild steel

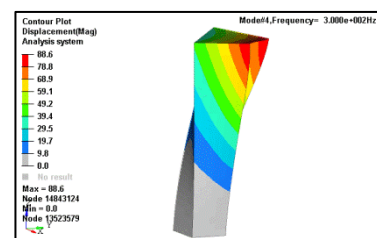


Fig. No. 1.3. Natural frequency of C-section specimen with Mild steel

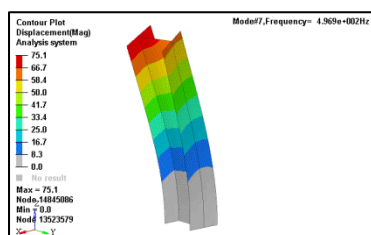


Fig. No. 1.4. Natural frequency of I section specimen with Mild steel

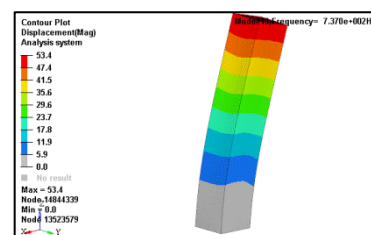


Fig. No. 1.5. Natural frequency of Square tube specimen with Mild steel.

Fig. No. 1. Cross section selection for holding fixture

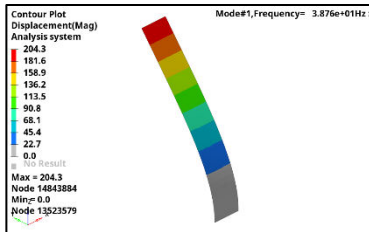


Fig. No. 2.1. Natural frequency of Plate specimen with Aluminium

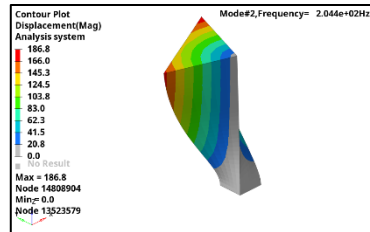


Fig. No. 2.2. Natural frequency of Angle specimen with Aluminium

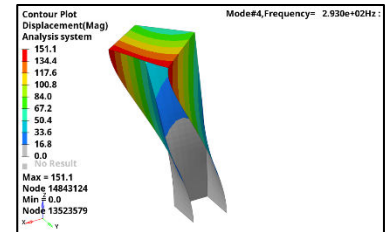


Fig. No. 2.3. Natural frequency of C-section specimen with Aluminium

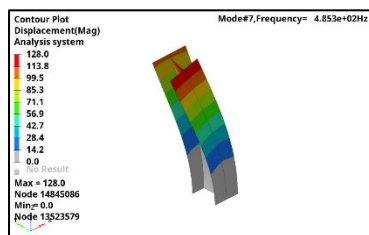


Fig. No. 2.4. Natural frequency of I section specimen with Aluminium

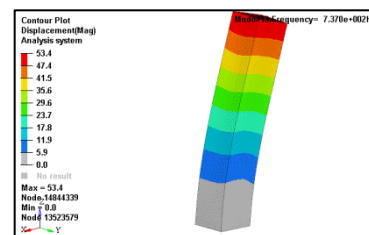


Fig. No. 2.5. Natural frequency of Square tube specimen with Aluminium

Fig. No. 2. Cross section selection for holding fixture

From above table the natural frequency of square tube section is maximum from other cross section fabrication with square pipe is much easier than other cross section for welding picture the design is proceed with square tubular cross section.

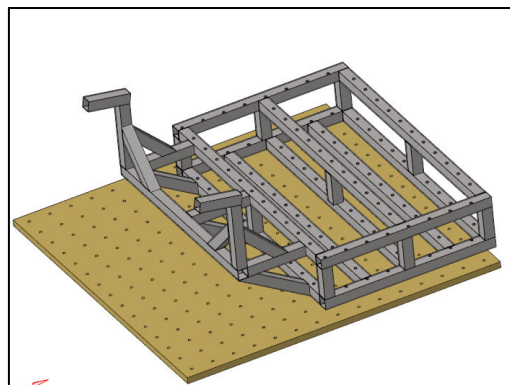


Fig. No. 3. Base design of Holding Fixture with Vibrating plate (Header)

Natural frequency analyses are used to determine the dynamic properties of a system and to identify its resonant frequencies. Some frequencies can cause elements to ‘sing.’ Imagine the sound of a guitar string being strummed. Because we understand a guitar string’s resonant frequencies, we are able to harness it, and subsequently make music. This can also be exemplified by microwave ovens. Microwaves in microwave ovens have an equal frequency to the frequency of water molecules. As these molecules are oscillated and gain kinetic energy, the temperature of food rises. At the same time, we can also use this knowledge of natural frequency to dampen an object’s resonant frequencies and vibrations. Natural frequency analysis or modal frequency analysis of structures or buildings is imperative to avoid disasters. The load carrying ability of the material almost never enters into fixture design. This is because, if the structure is designed so as to avoid resonances, it is over-designed by far with respect to static or dynamic loading. The resonant frequency then becomes the critical design factor. The prediction of natural frequencies of a structure can be very complicated. However, for the purpose of designing a fixture, only the first natural frequency is important. The first natural frequency is the most damaging and the appearance of any resonances will often disqualify a fixture for use.

A further simplification is the assumption of a single degree of freedom. Probably everyone reading this is familiar with the concept of a simple mass  $m$  supported by a pure spring of stiffness  $k$ , which when deflected, resonates at a

$$f = \frac{\omega}{2\pi} = \frac{1}{2\pi} \sqrt{\frac{k}{m}} \text{ Hz}$$

frequency.

By breaking the fixture down to a combination of simple elements and assuming a single degree of freedom system, this formula can be used as the basis for almost all frequency calculations in fixture design. The trick then comes in deciding what  $k$  should be.

As an example, take the aluminium cantilever beam shown in Fig. No. 4.

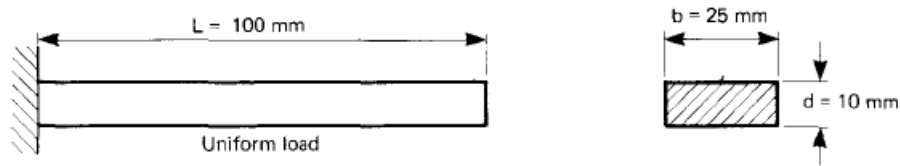


Fig. No. 4. Cantilever Beam

This is a fairly common type of configuration. For instance, resonant fatigue specimens are typically bars of material clamped at one end in a fixture.

The  $k$  value for a beam is –

$$k = \frac{BEI}{L^3} \text{ N/m}$$

Where,

$E$  = elastic modulus in  $N/m^2$

$I$  = second moment of area in  $m^4$

$L$  = length in  $m$

$B$  = a constant based on end conditions

For a rectangular section,

$$I = \frac{bd^3}{12} \text{ m}^4 \text{ and } m = \rho bdL \text{ kg}$$

Where,

$b$  = width in  $m$

$d$  = thickness of the beam in  $m$

$\rho$  = density in  $kg/m^3$

Hence,

$$\omega = \sqrt{\frac{k}{m}} = \sqrt{\frac{BEI}{mL^3}} = \sqrt{\frac{12Ebd^3}{12\rho bdLL^3}} = \sqrt{\frac{Ed^2}{\rho L^4}} = 5 \times 10^3 \frac{d}{L^2} \text{ rad/s}$$

Which shows that for a rectangular section the resonant frequency is independent of width.

For the above example,

$$f = \frac{1}{2\pi} \times 5 \times 10^3 \times \frac{d}{L^2} \approx 800 \times \frac{0,010}{0,1^2} = 800 \text{ Hz}$$

The calculations are done using Optistruct as a solver and results are post-processed using Hyperview. First frequency of holding fixture is 95.74Hz with Mild Steel material, which is near the Target of 95Hz. First frequency of holding fixture is 93.50Hz with Aluminium, which is below the Target of 95Hz. The upper side hands of holding fixture are not good support on top side therefore results are on lower side. Support is required to front hand of holding fixture.

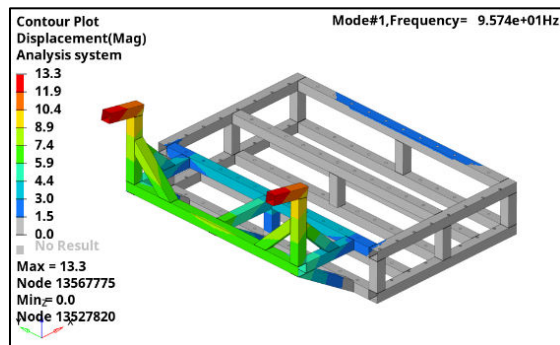


Fig. No. 5.1. Frequency plot of base design with Mild Steel Material

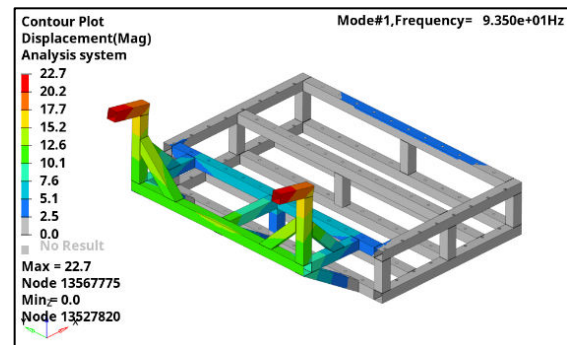


Fig. No. 5.2. Frequency plot of base design with Aluminium Material

Fig. No. 5. Frequency plot of base design

## V. CONCLUSION

Design of holding fixture is relatively complicated as small changes in it makes big impact on its performance and without FEA software; it is very difficult to do this design. It is very difficult to make holding fixture and fabricate it and test it on testing machine for its vibration and if it fails redesign is required and then again fabricate it and test on testing machine and check the results. In this all process the cost of fabrication and testing is too much high. The other factor without FEA is time for design phase. FEA provide very good platform for such complicated as we seen in report we do lot of study on holding fixture, we did lot of iteration with changing tubes and support to fixture and got the results. In this project, we reduced lot of effort with FEA tool and achieved good optimized model. Without FEA, good optimization is much difficult due to limitation of testing of physical fixture.

## REFERENCES

- [1] The Journal of Advanced Manufacturing Technology (2018) 96.4303-4311
- [2] <https://doi.org/10.100//sm1/O-018-WO/-z>
- [3] Kaustubh V. Wankhade, and N. A. Wankhade, Design and Analysis of Transfer Trolley for Material Handling, International Journal of Innovative and Emerging Research in Engineering, 2(1), Special Issue 1 MEPCON 2015
- [4] Ramkumar R, Krishnaraju A, Optimization of Material Handling Trolley using Finite Element Analysis. IOSR Journal of Mechanical and civil Engineering (IOSR-JMCE), Volume 13, Issue 6 Ver. V (Nov. - Dec. 2016), PP 137-148
- [5] International Journal for Innovative Research in Science & Technology | Volume 2 | Issue 05 | October 2015
- [6] Int J AdvManufTechnol (2002) Ownership and Copyright Springer-Verlag London Ltd
- [7] International Journal of Engineering Research and General Science Volume 2, Issue 2, Feb Mar 2014 ISSN 2091-2730
- [8] International Journal of Innovative Research in Science, Engineering and Technology (An ISO 3297: 2007 Certified Organization) Vol. 4, Issue 6, June 2015
- [9] Sai Aditya Pradeep, S. P. (n.d.). Body in White - an overview | ScienceDirect Topics. Retrieved October 7, 2020, from <https://www.sciencedirect.com/topics/engineering/body-in-white>
- [10] Park, K. H., Bae, M. S., Yoo, D. H., Shankar, S., & Kim, B. H. (2005). A study on buzz, squeak and rattle in a cockpit assembly. SAE Technical Papers. <https://doi.org/10.4271/2005-01-2544>
- [11] Martin Trapp, Fang Chen. Automotive Buzz, Squeak and Rattle: Mechanisms, Analysis, Evaluation and Prevention. Butterworth-Heinemann 2012.
- [12] Kavarana, F., & Rediers, B. (1999). Squeak and rattle - State of the art and beyond. SAE Technical Papers, 1999-2000. <https://doi.org/10.4271/1999-01-1728>
- [13] Guillaume, P. (n.d.). MODAL ANALYSIS. In CONTROL SYSTEMS, ROBOTICS AND AUTOMATION. <http://mech.vub.ac.be/avrg/publications/ModalAnalysis.pdf>
- [14] Poropatich, K. (2011). When Resonance Attacks. <https://www.vice.com/en/article/3ddj35/when-resonance-attacks>
- [15] Tabea Tietz. (2020). The Broughton Suspension Bridge and the Resonance Disaster - SciHi BlogSciHi Blog. <http://scihi.org/broughton-suspension-bridge-resonance-disaster/>
- [16] Cook, R., 1974. Concepts and Applications of Finite Element Analysis. 4th ed. New York: Wiley.



**INNO SPACE**  
SJIF Scientific Journal Impact Factor  
**Impact Factor: 8.423**



**ISSN** INTERNATIONAL  
STANDARD  
SERIAL  
NUMBER  
INDIA



# INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

9940 572 462 6381 907 438 [ijirset@gmail.com](mailto:ijirset@gmail.com)



[www.ijirset.com](http://www.ijirset.com)

Scan to save the contact details