Optimization of Three Wheeler Differential Gearbox Casing Using Modal And Stress Analysis

[1]Prof. R. S. Yadav,[2]Prof. V. R. Gambhire,[3]Prof. Dr. P. J. Patil

[1]PG Student TKIET Warananagar,[2]Professor,PG Guide TKIET Warananagar,[3]PG Coordinator TKIET Warananagar
[1]Department of Mechanical Engineering,
[1]TKIET Warananagar, Kolhapur,India

Abstract : The differential gearbox casing encloses different sets of helical gears, spur gears and three bearings to support the shafts. The vibrations in a power transmission gear system are initiated by the gear mesh.[4] The vibrations are transmitted to the gear-box casing through the shafts and bearings, due to which the gearbox casing vibrates and radiates the structure-borne noise. To reduce the vibrations, the rib stiffeners are generally used around bearing locations. Adding stiffeners increases small weight, but its increases stiffness enormously. It is observed that the overlooked variables i.e. positions and numbers of stiffeners affect the dynamic behavior of casing significantly. The casing under consideration is of Bajaj Three Wheeler Vehicle made up of die cast ALSi132 material equipped with sets of stiffeners on either side. The number of stiffeners on the casing was excessive, so it is required to reduce the number of stiffeners on the casing without affecting the mechanical functionality, strength and durability of the component so as to get simplicity in casting and some reduction in material consumption also vibrational performance should not be hampered due to renumbering of the stiffeners. The objective of the project is to analyze differential gearbox casing for modal and stress analysis. The theoretical modal analysis is validated with experimental results from Fourier frequency transformer analysis.

***IndexTerms* - Differential gearbox casing, Modal analysis, Stress analysis, Optimization, FFT analyzer, vibration.**

I. Introduction

The differential gearbox casing is very important part of any vehicles. It houses of various sets of spur and helical gears. It also consists of three bearing to support the shafts carrying gears. The gear meshing generates vibration which is transmitted to gearbox casing through bearings. Hence analysis of gearbox casing is of great importance. To reduce the vibrations generated due to gear meshing stiffeners are incorporated in casing. Optimization and structural modifications is necessary to reduce component complexity, weight and product cost. This project aims at Static stress analysis of Three Wheeler Differential Gearbox casing and optimization of stiffeners using gear mesh frequency.

The objective of this project is to do Number Optimization of 4 sets of stiffeners of Differential Gear Box Casing. So Modal analysis of differential casing with original set of stiffeners and optimized set of stiffeners which was finalized after comparison of natural frequency results with Gear Meshing Frequency is carried out. Also the linear static stress analysis of the casing was done by applying the forces which are calculated from the given data, and then the stress values are compared for casing with original number of stiffeners and casing with optimized set of stiffeners. The casing under consideration for this project is made by casting using die cast AlSi132 material with stiffeners on both sides. Still yet the work done in this area includes vibration analysis using empirical formulae and iterative methods but in this project work finite element analysis is used to carry out vibration analysis and stress distribution under applied load. The results of FEA are compared with Fast Fourier Transform results for validation and optimized models of casing are proposed..

II. ANALYSIS METHODS

Modal Analysis: It is study of dynamic properties of structures under vibration excitation. The aim of Modal analysis is to determine natural mode shapes and frequencies of casing under free-free condition using ANSYS 14.5.

Stress Analysis: The static analysis of the model is performed by applying boundary conditions and forces which are calculated according to the data provided by the company.

II. Input Calculations

Gear mesh frequency:

Gear mesh frequency is defined as the number of gear teeth that enter in the mesh per unit of time or in other words; gear mesh frequency is the number of teeth multiplied by the rotational frequency of the gear. Gear meshing frequency is one of the potential vibration sources in the transmission.

Gear mesh frequency = $(F) = k * (N/60)$ Hz (1)

Where,

k= number of teeth on gear

N= speed of the rotating shaft on which gear is mounted in RPM

The general expression for fundamental gear meshing frequency and higher harmonics is as follows:

$$Fn = n*k* (N/60) \text{ Hz.} \quad (2)$$

Table 1 shows the excited frequencies obtained by multiplying the fundamental frequencies by 2,3,4 etc.

Table 1 Excited frequencies for further calculations

DESCRIPTION	SPEED RPM	FUNDAMENTAL FREQUENCIES (f)	EXCITED FREQUENCIES				
			II (2*f)	III (3*f)	IV (4*f)	V (5*f)	VI (6*f)
First gear	403	457	914	1371	1828	2285	2742
Second gear	689	838	1676	2514	3352	4190	5028
Third gear	1108	1182	2364	3546	4728	5910	7092
Forth gear	1594	1486	2972	4458	5944	7430	8916

Net forces acting at bearing locations at gearbox casing are as shown in table no. 2

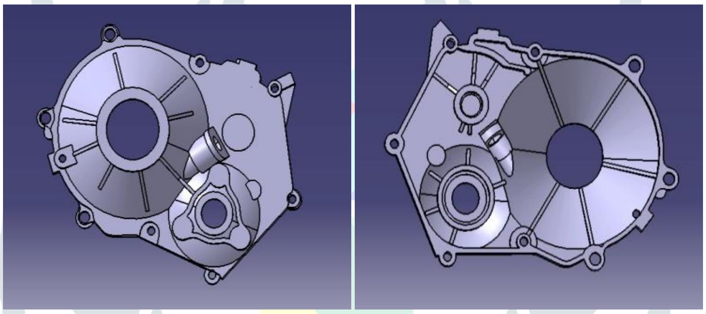
Table 2 Different forces at bearing locations

Location	Tangential force	Radial force
At Shaft No 3	371.03 N	135.06 N
At Shaft No 4	199.81N	72.73 N
At Shaft No 5	506.83 N	184.48 N

III. FINITE ELEMENT ANALYSIS :

In finite element analysis the first part is modeling. The mode of casing is prepared by using CATIA V5R20 software.

Fig. 1 Model of casing



Pre-processing

In pre processing the material properties are defined and meshing is done. The material of casing is Die Cast Aluminum (A360).

Material properties are

1) Density : 2630 Kg/m^3
2) Youngs Modulus : 7.1e^10
3) Poissons ratio: 0.33
4) Ultimate tensile strength: 46 MPa – 4.6e^7 Pa
5) Yield strength : 170 MPa – 1.7e^8 Pa

Meshing

Meshing is one of the important steps in analysis using ANSYS. Mesh tool is used to discretise the geometry into elements. To get accurate results "Tetrahedral" is used. The mesh size is kept as default under the tab Body Sizing.

Post processing- In post processing solution is obtained.

The mode shape results are as follows.

Table 3 Natural Frequencies under Free-Free condition

Mode	Natural Frequency
1	722.28
2	1020.4
3	1444.8
4	1508.8

IV. EXPERIMENTAL ANALYSIS:

When the finite element analysis is completed it is necessary to validate it experimentally. Experimental analysis is done by using FFT analyzer.

Experimental Setup

The basic test setup required for making frequency response measurements depends on a few major factors. These include the type of structure to be tested and the level of results desired. Other factors, including the support fixture and the excitation mechanism, also affect the amount of hardware needed to perform the test. Fig 3 shows a diagram of a basic test system configuration. The heart of the test system is the controller, or computer, which is the operator's communication link to the analyzer. It can be configured with various levels of memory, displays and data storage.

Fig. 2 Experimental Set Up

Experimental Results

Table 4 Experimental frequencies

Mode	Natural Frequency
1	776
2	1055
3	1487
4	1584

Comparison of ANSYS and Experimental Frequencies

Table 5 Experimental frequencies

Mode	ANSYS Frequency (Hz)	Experimental Frequency (Hz)	% Error
1	722.28	776	6.92
2	1020.4	1055	3.27
3	1444.8	1487	2.83
4	1508.8	1584	4.74

It is observed that the experimental frequencies are closer to the FEA frequencies. The maximum error recorded is 6.92 % which is less than 10 % so it in permissible limit so this validates the FEA work.

V. OPTIMIZATION OF STIFFENERS

Table 6.3 shows the different sets of stiffeners considered for optimization. In the present casing, there are three bearing locations at which stiffeners have been provided numbering as 8, 5, 5, 5. at the location A, B, C and D respectively, Where A- Bearing location at shaft no. 5 on outer side of casing – 8 Stiffeners. B- Bearing location at shaft no. 5 on inner side of casing – 5 Stiffeners . C- Bearing location at shaft no. 3 on inner side of casing – 5 Stiffeners. D- Bearing location at shaft no. 4 on inner side of casing – 5 Stiffeners, as shown in following figure.

For analysis, we created one table in which the number of stiffeners at three locations are kept constant and keep on changing the number of stiffeners at fourth hole. According to this we have considered the 12 such cases of stiffeners which are represented in table no 6

Table 6 Different sets of stiffeners

Mode	A	B	C	D	Designation
Case 1	8	5	5	5	8555
Case 2	6	5	5	5	6555
Case 3	4	5	5	5	4555
Case 4	8	4	5	5	8455
Case 5	6	4	5	5	6455
Case 6	4	4	5	5	4455
Case 7	8	5	4	5	8545
Case 8	6	5	4	5	6545
Case 9	4	5	4	5	4545
Case 10	8	4	4	5	8445
Case 11	6	4	4	5	6445
Case 12	4	4	4	5	4445

So these 12 models are prepared in CATIA and for all 12 cases natural frequencies are determined in ANSYS and compared it with operating frequencies to suggest optimized model For above stated set of stiffeners fundamental frequencies are calculated for first 10 modes by modal analysis in ANSYS. The modal analysis for above mentioned 12 cases is also carried out in a similar way to that of modal analysis of existing casing described in earlier chapter, but one major change is that for this modal analysis of 12 cases analysis is not carried out under free free condition but the boundary conditions are applied. As under operating condition casing is connected to the gearbox through nut and bolts so at the eight bolt locations of the casing, fixed support is applied in an analysis setting tab, also maximum modes to be find are taken as 10.

Table no. 7 describes the fundamental frequencies for all 12 sets of stiffeners.

Table 7 Fundamental frequencies for all 12 sets of stiffeners

Cases	Mode	1	2	3	4	5	6	7	8	9	10
Case 1	8555	2415.4	4421.1	4488	4864.5	5144.4	5399.2	5459.9	6182.8	6320.5	6574.3
Case 2	6555	2559.7	4601.8	4731.4	5047.5	5361.7	5567.3	5829.7	6621	6661.6	6841.3
Case 3	4555	2562.7	4620.1	4744	5055.8	5378.2	5581.9	5832.2	6613.2	6674.4	6849.6
Case 4	8455	2555.4	4577.9	4710.2	5005	5322	5531.8	5822.1	6556.1	6616.6	6721.1
Case 5	6455	2565.8	4610.2	4732.9	5060.7	5355.7	5568.1	5839.6	6608	6638.7	6776.9
Case 6	4455	2557.7	4605.7	4728.6	5040.5	5362.8	5560.1	5830.3	6580.4	6619.3	6754
Case 7	8545	2558.4	4588.4	4724.5	5019.6	5333.4	5552.4	5822.6	6583.8	6637.8	6819.2
Case 8	6545	2566.9	4617.4	4746.4	5072	5367.9	5587.8	5834.2	6593.8	6666.7	6874.4
Case 9	4545	2571.8	4628.8	4762.3	5080.8	5389.9	5608.2	5843	6594.8	6682.2	6868.3
Case 10	8445	2554.3	4575.8	4709.7	5004.7	5320.4	5530.1	5820.8	6555	6610	6716.3
Case 11	6445	2562.4	4607.9	4731.8	5058.1	5352.7	5566.7	5833.9	6592.9	6614.6	6771.8
Case 12	4445	2563	4610.7	4747.2	5066.7	5361.8	5566.7	5839	6590.5	6610.4	6773.6

The operating frequencies are compared with these natural frequencies. If some of these frequencies are nearly closer to each other then there will be resonance and vibrations. The cases in which both the frequencies which are closer/matching will not be recommended case of stiffeners for the gearbox casing. From above comparison it is found that fundamental frequencies for case nos. 2, 6,7 and 11 coincides with natural frequencies i.e. condition of resonance occurs. Hence these sets are excluded from consideration. But one more attempt had made by performing modal analysis by removing all the stiffeners from the existing model. After comparing these frequencies with operating frequencies it is observed that these frequencies are does not matching with exciting operating frequencies so no resonance condition will occur. So this model i. e. casing without all stiffeners is also proposed as optimized model.

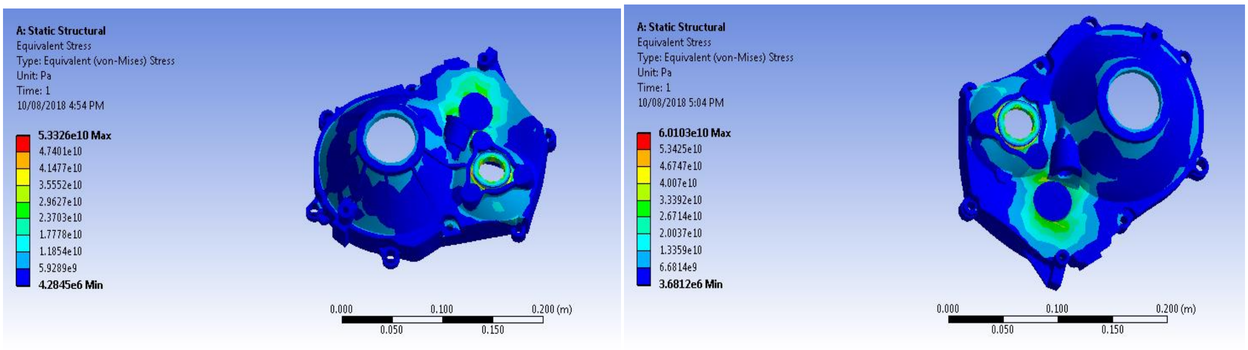
VI.STRESS ANALYSIS

The forces which are acting at the bearing locations are calculated in previous chapter by using the numerical equations. These forces are applied in at their corresponding acting positions in finite element analysis. The model was constrained at the 8 locations by applying fixed support and different forces applied at three bearing locations are calculated earlier as shown in table no. 2.

Stress analysis is performed for two cases one by presenting all the stiffeners and another by removing all the stiffeners. The following results are obtained by running the static stress analysis.

Static stress analysis of existing casing with all stiffeners. Static stress analysis of model without all stiffeners.

Fig. 3 Stress analysis



The maximum stress observed in both the case is as shown in table no 8

Table 8 Stress comparison

Model	Stress (Mpa)
Existing (Casing with all stifners)	5.33
Proposed (Casing without all stiffners)	6.01

The variation in stress values as shown in table no. 7.2 is very small so proposed model of casing is safe from stress point of view. Also adding stiffeners increases the mass and stiffness around bearing locations of casing so for other proposed cases there is no need to carry out stress analysis; those models are also safe from stress point of view.

VII. RESULTS AND DISCUSSIONS

Models with different sets of stiffeners are prepared in CATIA and their natural frequencies are determined in ANSYS. The obtained frequencies are compared with operating frequencies. The cases in which resonance would occur are not recommended and remaining cases are proposed optimized models. Those proposed models are case no. 1, 3, 5, 8, 9, 10 and 12. Also model without all ribs is also proposed.

For existing model i. e. model with all ribs and model without all rib, stress analysis is carried out the maximum stress in casing with all stiffeners is recorded as 5.33 Mpa where as it is 6.01 Mpa in model of casing without all stiffeners so it is observed that there is no any significant increase in stress in model without all stiffeners. So it is safe from stress point of view.

VIII. CONCLUSION

The following conclusions are drawn 1. For the sets of the stiffeners as mentioned in cases 1, 3, 4, 5, 8 to 10 and 12 there is no any resonance condition is occur. So models with these cases are recommended.

2. For the remaining sets of stiffeners i. e. cases 2, 6, 7 and 11 the resonance condition is observed so these sets are not recommended.

3. As casing with no stiffeners is also belongs to non resonance condition and also it satisfies the load carrying capacity i. e. stress criterion so it recommended as one of the optimized design for the casing.

4. For the analysis of components like gearbox casing and gear mesh frequencies the modal analysis is one of the best method rather than complex numerical equations.

5. There is close agreement between results obtained in finite element analysis and experimental analysis is observed.

REFERENCE

[1] Bagul A.D.Barijibhe R. B. and Patil A. V,“ Vibrational Analysis of Gearbox Casing Component using FEA Tool ANSYS and FFT Analyser” International Journal of Engineering Research & Technology (IJERT), ISSN: 2278-0181, Vol. 3 Issue 2, February 2014.

[2] R. V. Nigade, Prof. T. A .Jadhav, A.M. Bhide, “Vibration Analysis of Gearbox Top Cover”, International Journal of Innovations in Engineering and Technology (IJIET), Vol. 1 Issue 4, Dec 2012.

[3] G. Rajesh Babu, N. Amar Nageswara Rao,“ Static And Modal Analysis Of Rear Axle Housing Of A Truc,” International Journal of Mathematical Sciences, Technology and Humanities, July 2011.

[4] Balasaheb Sahebrao Vikhe,“ Design Analysis Of Industrial Gear Box Casing.,” International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056 Volume: 03 Issue: 11 Oct -2016.

[5] Sagar Pradip Walhekar,” Vibration Analysis for Two Wheeler Gearbox Casing using FEA,” International Journal on Recent Technologies in Mechanical and Electrical Engineering IJRMEE, Vol. 2 Issue 8, August 2015.

[6] William R. Kelley, L.Dean Isley, and Thomas J. Foster of Borg,” Dynamic Correlation Study Transfer Case Housings,” Warner Automotive Power train systems Sterling Heights, MI, USA..Mar 1987.

[7] M. Sofian, D. Hazry, “A study of Vibration Analysis for Gearbox Casing Using Finite Element Analysis.,” International Conference on Applications and Design in Mechanical Engineering (ICADME) October 2009.