A Review on Analysis And Optimization of Gearbox Casing

Mr. Khandekar Ranjeet S.[1], Dr. R.P. Kulkarni[2]
[1] Dept of Mechanical Engineering
[2] Principal,
[1, 2] YSPM, YTC, Satara.

Abstract- *The gearbox casing is the large weighted part of the gearbox assembly. The function of gearbox casing is to support and covers transmission components like shafts, gears, bearing. The objective of this study is to review the design optimization of gearbox casing to find out the effective and efficient design of gearbox casing leading to material and cost savings. The modeling of the gearbox casing is created by using 3-D modeling software like CREO, CATIA & Pro-E etc. Finite element analysis (FEA) is performed to carry out stress analysis and modal analysis for the gearbox casing using ANSYS software. An experimental modal analysis (EMA) validates the FEA result. It has been observed that FEA analysis and EMA gives the best output to carry design optimization.*

Keywords- EMA; gearbox casing; FEA; weight reduction;

I. INTRODUCTION

Gearbox casing is an important part of the gearbox assembly, which surrounds the mechanical part of the transmission. It supports the shaft, bearings and hence the gear loading. Thus the gearbox casing is an important component to be taken into account while designing. As its weight typically represents about 50% of the gearboxes, optimizations of shape and weight are of particular importance, leading to material and cost savings.

In this presents study detail overview of design optimization of gearbox casing including theoretical and experimental methods and their result. The efforts have been made to present various cost-effective, reliable, analytical, numerical and experimental techniques developed by various researchers to reduce weight, reduce complexity and subsequently the manufacturing costs. In this paper different modeling software, structural analyses software, Experimental method, analyzing technics have been reviewed.

II. LITERATURE REVIEW

Syed Rizwan Ul Haque et. al., Studied in his work about the Static Analysis of Gearbox Casing Using Sub modeling Approach in ANSYS. The casing consist of Upper cover, bottom and hooks are modeled in high end design software (Pro/E) which is simpatico with simulation software. The CAD model is imported into the respective file format to the FEM design software Altair Hyper mesh 8.0. The static load of transmission gear and drive shaft act on bearing hole. To obtain precise stress in a local/specific region, sub modeling separates local analysis from the global model. This allows mesh refinement in a region that might not be possible on the full model without exceeding size limits. The analysis of gearbox casing's hooks, the sub modeling utilizes two separate models; the whole model is analyzed with a relatively coarse mesh, and then sub model cut from the whole model is analyzed in ANSYS 11.0.

Aditya Nigam et. al., Study has been carried out to evaluate the static analysis of the gearbox Housing foot. The housing foot of planetary gearbox made using part and assembly design module in CATIA V5R20 software. While static analysis is done in ANSYS V14 software. Results are obtained by varying the size of housing foot by keeping loads and boundary condition constant. The obtained results are compared to help in finding the optimized design for the housing foot of planetary gear trains in which it has the best performance without any failure and with Optimum Loads acting on the housing foot.

Balasaheb Sahebrao Vikhe, In his paper, conducted Optimization of a three stage Commercial gearbox casing using Finite Element Analysis (FEA) Method. The 3D model is prepared by using Pro-E creo2.0 pre-processing is prepared by using Hyper mesh 11.0 while FEM is solved by using ANSYS 14.5 solver. It was statically and dynamically analyzed using simulation software Altair Hyper mesh and ANSYS 14.5. Static analysis is to find out the total amount of stresses and displacement of the gearbox casing and End cover. Dynamic analysis is to find out the Natural frequency of casing. Optimization is based on ANSYS Linear static and dynamic modal analysis results, which can be used to enhance the efficiency of the design process. The results obtained from optimization, the geometric model was modified and iterated until satisfactory results were achieved. The process is repeated until all specified criteria are met.

Rahul Yashwant Banekar et. al., Conducted structural analysis of a single stage industrial helical gearbox casing using Finite Element Analysis (FEA) Method. Strength, weight, manufacturability and cost of gearbox casing are an important factor is to be taken while designing. Gearbox casing Prepared by using Pro-E creo2.0, 3D solid model in STEP file format is imported in Hyper mesh. Mesh model is prepared by using Hypermesh11.0. 2D. trial meshing is carried out on all the inner and outer surfaces of the geometry, quads splits to tries and then converted to tetras. 4-node Linear Tetra 3D solid elements are used to the model of Gearbox and End covers. The element size selected for Casing and End covers mesh is 5 mm. static analyses carried out by applying load and boundary condition by using ANSYS 14.5 software. After analyses results, they can Say that acceptable limits of Stress & displacements are very higher than actually induced on the casing.

But they have manufacturing constraint that cannot optimize beyond this limit of 5 mm wall thickness. The reason is that, during a stress relieving cycle, plate sizes below 5 mm get Distortions, which results in bending of gearbox casing plates. Also, they conclude that optimum design is better than the traditional one in weight factor by comparing results.

Mitesh Patel et. al., Focused on the stress analysis of triple reduction of gearbox casing. The load calculation for helical gear was performed using the CAD Software package. The model was built by using NX-8 and analysis carried out on ANSYS software. In that research take three different cases and compare the result to get optimum design. Casing with ring and stiffeners gives better result amongst three cases, casing without ring and stiffener, casing with stiffeners and casing with ring and stiffeners.

Vasim B. Maner et. al., In his paper conducted design Optimization foot casing of planetary gearbox casing at Top Gear Transmission Industry situated at Satara, India. The author redesigns the existing foot casing to save Material as well as cost. 3D model generated in PRO-E software; static analysis is done on ANSYS software. The design optimization is based on ANSYS results. The main focus is to optimize the foot casing and to find out the effective and efficient design of gear-box concerning both cost and durability.

Smita Pawar et. al., Identifies and solves the problem. Through static analysis on the electrostatic Precipitator gearbox housing. Analysis carried out by using ANSYS software. Results are obtained by varying rib Dimension and selecting different material. Finally best optimum design for gearbox housing selected by comparative analysis by considering equivalent stress, deformation, weight, casting price, availability of material.

Zoltan- KORKA et. al., Has studied Modal - Based Design Optimization of a Gearbox Housing. They presented the natural frequencies are influenced by the housing material and how these frequencies vary by changing the dimensions of some geometric elements, which define the rigidity of the housing. In the first stage of the research, Experimental modal analysis (EMA) is a physical Test performed on an object to establish its natural frequencies. In the next stage, FEM is used to find out natural frequencies and mode shape of a structure during free vibration. Experimental modal analysis was performed to validate the FEM model developed for the housing of single stage gear. The results obtained by numerical simulation are close to the results obtained experimentally, with a maximum deviation of 4.18%. The deviations may be generated by the non-homogeneity of the material (cast iron) of the analyzed housing.

Shrenik M. Patil et. al., Compared and analyzed relative results of modal analysis and stress analysis by using ANSYS 12.0 and validate through EMA. The model created in Pro-E Wildfire 3.0 and meshed using hyper mesh 11.0. The Stress analysis and natural frequencies of the model in free-free conditions are calculated using ANSYS 12.0.Experimental modal analysis (EMA) of the actual component in free-free conditions done by using FFT analyzer. The EMA determines the natural frequency, mode shapes and damping ratio. From the experiment, natural frequencies of the component calculated. The natural frequencies are compared with ANSYS 12.0 results which validate the model created in Pro-E Wildfire 3.0. They considered five different sets of stiffeners provided on both sides of casing to reduce vibration, noise and stress. Each set of stiffener is removed and its natural frequency is evaluated for optimization of the gearbox casing.

Raghavendra Setty et. al., Have investigated the problem of very high vibration observed in the Gearbox in Bar Rod Mill (BRM) of JSW Steel Limited, Bellary at 1000-1100 rpm the velocity is 10.3 mm/sec. The model was built by using Solid Edge (V19) and Nastran (NX8) software's and analysis carried out on ANSYS 15.0 software. Optimized the resonance problem of a casing by finding the mode shapes and natural frequencies of the existing and modified casing. First 10 natural frequencies and mode shapes of gearbox casing were considered. They said, the modified casing has more stiffness also the natural frequency is shifted away from the resonance band.

Vijay N.A. et. al., Presented, FEA of Gear Box Casing used for Permanent Magnet D.C. Motors. The gearbox casing is modeled using CATIA V5 software and analyzed using ANSYS workbench software.

Static stress analysis carried out to find maximum stresses as well as deformation and Model analysis carried out to find mode shape and natural frequencies. After Analyses Results, they can Say that the outcomes through the FE analysis is found to be in a good agreement and are within the safe limits.

Table II shows the researchers view to carry their work in case of analysis method, software used, modification etc.

TABLE I Researchers Study overview

Sr. No.	Author Name	Analysis Method	Software used	Material Specification	Modification	Result	Remark
1	V. B. Maner et. al.	Static Analysis	PRO-E, ANSYS	FG260	Size variation	Max. Stress-36.544 Mpa Deformation-0.077242 mm.	Max. stress 14.05% of Material Strength . Deformation within limit
2	Balasaheb Sahebrao Vikhe	Static Analysis,	PRO-E, Hyapermesh, ANSYS	FG260,	Size variation, variation in rib thikness	Max. Stress-86.66 MPa Deformation-0.205mm	Max. stress 33.33% of Material Strength. Deformation within limit
3	Smita Pawar et. al.	Static Analysis	CATIA, ANSYS	CI FG 150	varying rib dim. and selecting different material	Max. Stress-63.66 MPa Deformation-0.078822mm	Max. stress 42.44% of Material Strength. Deformation within limit
4	R.Y. Banekar et. al	Static Analysis	PRO-E, Hyapermesh, ANSYS	Fabsteel IS2062 E-250	Size variation	Max. Stress-3.82 MPa Deformation-0.0091mm	Max. stress 1.52% of Material Strength. Deformation within limit
5	Namankumar Patel et. al.	Static Analysis, Modal Analysis	Solid work , ANSYS	FG250	Split To Monoblock Gearbox Design	Existing Model- max. stress-66MPa. Deformation-0.07mm Modified model- max. stress-60 MPa Deformation- 0.04mm	Max. stress 24 % of Material Strength. Deformation within limit
6	Veysel Yalin Ozturk et. al.	Static Analysis	MSC Nastran	Not mentioned	Irrelevant elements are removed	Existing Model- max. stress-45MPa. Deformation-12 μm Modified model- max. stress-34 MPa Deformation- 5 μm	Max. stress 17.30 % of Material Strength. Deformation within limit
7	Aditya Nigam et.al	Static Analysis	CATIA, ANSYS	Not mentioned	Size variation	Max. stress-3.3924e8 pa Deformation- 0.0004144 mm	Max. stress 45 % of Material Strength .Deformation within limit

.

The graphical representation of maximum stress percentage and deformation value of modified gearbox analysis from different researchers study shown in fig 1. From the graph, it is clear that researcher R.Y. Banekar et. al. obtained very less maximum stress value. They said that due to manufacturing constraint cannot optimize beyond the limit.

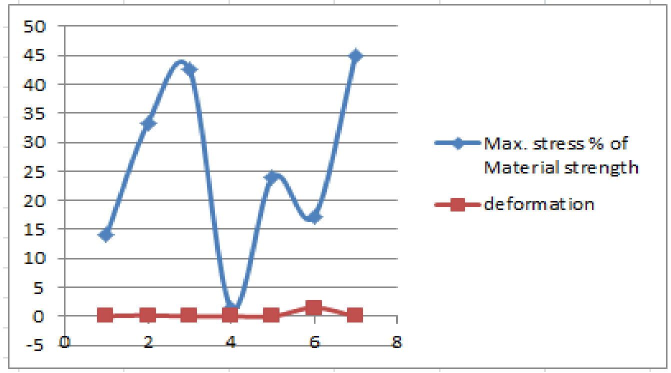

Fig 5 Max. stress % of Material Strength And Deformation from Different researchers Study.

III. CONCLUSION

From the study, the following conclusions can be drawn:

- FEA is a good tool to carry out stress analysis and modal analysis to achieve the effective and efficient design of gearbox casing relevant to cost and durability.
- FEA gives the idea about future scope in modification and optimization of gearbox casing design relevant to shape, material, time and cost.
- EMA is essential to validate the software result.

REFERENCES

[1] Aditya Nigam et. al., "Design & Analysis of Housing Foot of a Planetary Gearbox using Finite Element Analysis," in International Research Journal of Engineering and Technology (IRJET), Vol. 03 Issue: 11 | Nov -2014

[2] Syed Rizwan Ul Haque, Prof. Dongyan Shi, Azouz Radhia, Tauseef Ahmed, " Static Analysis of Gearbox Casing Using Submodeling Approach in ANSYS." in Harbin Engineering University publication, China. January 2010

[3] Balasaheb Sahebrao Vikhe, "Optimization of Industrial Gearbox Casing," in International Research Journal of Engineering and Technology (IRJET), Vol. 03 Issue: 12 | Dec -2016

[4] Rahul Yashwant Banekar, "Design & analysis of single stage industrial helical gearbox casing," in International Journal of Recent Trends in Engineering and Research(IJRTER), ISSN(online): 2455-1457 |2016

[5] Mitesh Patel et. al.," Stress and Design Analysis of Triple Reduction Gearbox Casing," in International Journal for Innovative Research in Science & Technology (IJIRST), Vol. 02 Issue: 02 | July -2015

[6] V. B. Maner et. al., "Design Analysis and Optimization for foot casing of gearbox," Proceedings of 3rd IRF International Conference, Goa, India, ISBN: 978-93-84209-15-5 | May -2014

[7] Smita Pawar et. al., "Improvement in design of gearbox housing (Code No: MFO225DR) through static analysis," International Journal of Advance Research, Ideas and Innovations in Technology, Vol. 03 Issue: 12 | 2019

[8] Zoltan Korka et. al., "Shape Improvement of Gearbox Housing Using Modal Analysis," in RJAV, Vol. 15 Issue: 01 | 2018

[9] Shrenik M. Patil et. al., " Modal and Stress Analysis of Differential Gearbox Casing with Optimization," in Int. Journal of Engineering Research and Applications, Vol. 03 Issue: 06 | Dec -2013

[10] G Raghavendra Setty, "Modeling and Dynamic Analysis of Gear Box Casing Using Finite Element Analysis," in International Journal of Innovative Research in Science, Engineering and Technology, Vol. 05 Issue: 06 | June - 2016

[11] Veysel Yalin Ozturk, "Topology Optimization of a Gearbox Casing," in European Rotorcraft Forum

[12] Sudhir Mane et. al., "Structural Stress Analysis of Reduction Helical Gearbox Casing," in International Journal Of Modern Engineering Research (IJMER), Vol. 06 Issue: 06 | June -2016

[13] Namankumar Patel et. al. , "Optimization of gearbox Casing for Multi-stage Gearbox by using FEA Approach," in Journal of Applied Science and Computations, Vol. 06 Issue: 05 | May -2019

[14] Aditya K. Nimbalkar et. al., "Optimization of Stiffeners of Differential Gearbox Casing by Vibration and Stress Analysis," in International Engineering Research Journal, Page No 1425-1430, MD 7-30