Optimization of Wind Power Generator Frame Act By Using CAE Software

Dr. B Nageswara Rao[1], R Santhi[2], Hatukesula Ramulamma[3]
Department of Mechanical Engineering
[1]Assistant Professor, [2]Assistant Professor, [3]PG Student,
[1, 2, 3]AVN Institute of Engineering & Technology, Hyderabad, India

Abstract*-The generator frame is the main load-bearing component in generator set. The generator frame consists of an assembly of the parts of generator like a generator, electrical equipment, accessories, etc. The generator frame must be strong enough to with stand the torque and vibrations loads generated by the generator. The generator frame is mostly made of welded steel plates and bolted connection, so it is complicated in structure and cannot be tabulated by applying the theoretical formula in the strength of materials. For this reason, we need to utilize the finite element method to calculate using the engineered software. In this project, we have considered a generator weights are 180Kgs with a production capacity of 10KW.This generator weight of 180kgs is applied as the static load on the generator frame. As the generator considered in this project has a lot of movable parts, it is subjected to huge vibrations. So, the generator frame is analyzed to check whether it will withstand for these vibrations caused by a generator. Computer-Aided Engineering (CAE) package is used to perform the analysis. In the present days, CAE package is vastly used for different analysis in various fields. In this project, a 3D model of the Generator frame is done in NX-CAD and is converted into Para solid. This Para solid file is imported into ANSYS to perform finite element analysis. The static structural analysis is performed with a generator weight as static load, and stresses and deflections are documented. In these project dynamic characteristics of the generator, the frame is also evaluated by performing modal analysis to calculate natural frequencies of generator frame. Spectrum analysis is performed to check structure behavior for random vibrations. Efforts are made to optimize the design for the above-said conditions NX-CAD software is used for generating the 3D model, and ANSYS software is used for doing finite element analysis.*

I. INTRODUCTION

A windmill is one type of engine. It uses the wind to make energy. Usually, a windmill is a large building. Common types of windmills are tower mills, smock mills and post mills. The energy made by windmills can be used in many ways. These include grinding grain or spices, pumping water and sawing wood. Modern wind power machines are used to create electricity. These are called wind turbines.

The sails or blades of the windmill are turned by the wind. Gears and cogs make the drive shaft inside the windmill turn. In a windmill used for making flour, this turns the grinding stones. As the stones turn, they crush the wheat between them. In a windmill used for pumping water, turning the drive shaft and moves a piston. This piston will hold up tightly and push out water as it reciprocates. In a windmill used for generating power, the drive shaft is connected to many years. This increases the speed and is used to turn a generator to make electricity.

Wind power is extracted from air flow using wind turbines or sails to produce mechanical or electrical power. Windmills are used for their mechanical power, wind pumps for water pumping, and sails to propel ships. Wind power as an alternative to fossil fuels, is plentiful, renewable, widely distributed, clean, produces no greenhouse gas emissions during operation, and uses the little land. The net effects on the environment are far less problematic than those of non-renewable power sources.

II. LITERATURE REVIEW

Static And Dynamic Analysis Of 2.0mw Generator Frame: by Chaoyi Ding Luping Dai and Hongche Guo, ANSYS 12.1 software is applied to the generator frame modelling and the static and dynamic performance analysis of the structure. The structure is calculated to find the max stress and the first six frequencies and Based on the Results; Some Suggestions Are Put Forward For Optimization Of The Design.

Analysis Of Structural Dynamic Turbo Generator Load On Foundation Structure For Estimation Stresses: during Vertical Excavation Using Finite Element Method by Sanjay Gupta, The load over the turbo power generator should be concluded. Based on the finite element value and experimental value of soil parameter as such major principal stresses and minor principal stresses, normal stresses, shear stresses, bearing pressure is analysed for the analysis of foundation structure.The analysis of factor bending moment, the intensity of pressure, maximum pressure, the main area of steel, distribution of steel were carried out for a comparative statement of the two-dimensional and three-dimensional approach. Three-dimensional approach became realistic and exact approach over two-dimensional approach.

Design And Performance Analysis Of A 6mw Medium Speed Brushless Dfig: by E. Abdi, M.R Tatlow, R.A. McMahon, PJ. Tavner The paper presents the design and performance analysis of a 6 MW medium-speed Brushless Doubly-Fed Induction Generation (Brushless DFIG) for a wind turbine drive train. Two machines with different frame sizes have been designed to show the flexibility of the design procedure. The medium speed Brushless DFIG in combination with a two stage gearbox offers a low-cost, low-maintenance and reliable drive train for wind turbine applications.

Modelling Of Turbine –Generator And Foundation Assingle Degree Of Freedom Using Ruaumoko Programme: By Shafiq Abdullah, Nor Hayati Abdul Hamid, A rigid-moment frame supporting the turbine-generator was designed according to BS 8110. This structure is subjected to vibrations of turbine-generators and seismic loading. Turbine-generator with its foundation is the model as a single degree of freedom (SDOF) using RUAUMOKO program. RUAUMOKO program is retained in this study to analysis nonlinear dynamic behaviour of turbine foundation using time-history analysis and Modified Takeda Model. Mode shape, natural period, natural frequency, nodal displacement, member forces and moment of reinforced concrete turbine foundation were obtained by running this program. The result shows that turbine foundation under Imperial Valley earthquakes does not exceed yield drift limit for the solid connection and remain within the elastic condition. Thus, RC turbine foundation is safe and able to carry gravity load as designed according to BS 8110. Contradictory, turbine Foundation experience exceeding yield drift limit but it is not safe and likely to collapse under San Fernando earthquake loading.

Design And Analysis Of Generator And Converters For Outer Rotor Direct Drive Gearless Small-Scale Wind Turbines By Yusuf Yasa, Erkan Mese, This paper focus on design and analysis issues of direct drive a gearless generator which are widely used in small-scale wind turbines. Permanent magnet synchronous generator(PMSG) is designed that is considered for 2.5 kW continuous and 6 kW peak power. The power converter topologies are briefly discussed then one of them is selected to develop. The finite element model of PMSG is integrated with power converters and simulated which is named coupled simulation provides more realistic results. The results are thought to be guidance for wind turbine developers and manufacturers.

Completion Of A 1,120-Mva Turbine Generator For Huaddian International Zouxian Power Plant In China: by Seijiro Muramatsu Kado Miyakawa Mitsuru Onoda, Dr. Eng., Hitachi, Kengo Iwashige, Kazuhiko Takahashi, Ltd. has completed construction of a 1,120-MVA turbine generator (2 poles, 50 Hz) for the Huadian International Zouxian Power Plant of China Huadian Corporation in China, and its performance has been fully verified through factory testing. With a maximum output of 1,230 MVA — some 1.6 times greater capacity than the 778-MVA output of Hitachi's next larger similar generator — the generator is one of the world's largest-class single unit turbine generators ever developed for a thermal power plant. This significant increase in output could only be used to achieve through a combination of sophisticated technologies, including an advanced distilled water cooling system and hydrogen gas and 27-kV high-voltage insulation. Maximum effort was made to improve the reliability and performance of the generator by fully exploiting advanced analytical tests and procedures including rotor vibration analysis, the stator core and electromagnetic analysis, network ventilation analysis, and stress analysis of all key parts of the generator. Full-scale rotational tests including efficiency, temperature increases, shaft vibration, and other performance measures confirm that the 1,120-MVA turbine generator more than satisfies all design specifications.

Turbo-Generator Foundation: By Sukanta Adhikari, This paper deals with the design of a turbo generator foundation for a well built thermal power plant. This article covers the typical aspect in design of turbo generator Foundation concerning IS 2974 (Part 3)-1992 and other international standards

III. PROBLEM DEFINITION AND METHODOLOGY

The objective of this paper is to design a 3d model of generator frame. The generator frame is subjected to static analysis due to generator weight. The generator frame is subjected to vibrations due to some moving parts in the generator. So, to check whether it withstands for vibrations caused by the generator, spectrum analysis is carried out. 3D modelling software (UNIGRAPHICS NX) is used for designing and analysis software (ANSYS) was used for finite element analysis.

THE METHODOLOGY FOLLOWED IN THE PROJECT ISAS FOLLOWS

3D model of the generator frame is done in NX-CAD. This 3d model is converted into Parasolid and imported into ANSYS to perform finite element analysis. The static structural analysis is performed with a generator weight as static load, and stresses and deflections are documented. Modal analysis of the generator frame is carried out to calculate natural frequencies and their mode shapes and evaluate the dynamic characteristics. Spectrum analysis is performed to check the frame behaviour due to random vibrations as per the input PSD curve. One sigma deflections and stresses are plotted. Design changes are made to strengthen the generator frame structure for operating conditions. The static structural analysis is performed with a generator weight as static load, and stresses and deflections are documented on the modified generator frame. Modal analysis is carried out to calculate natural frequencies and their mode shapes and evaluate the dynamic characteristics of the modified generator frame. Spectrum analysis is performed to check the modified generator frame behavior due to random vibrations as per the input PSD curve. One sigma deflections and stresses are plotted.

IV. 3D MODELLING OF GENERATOR FRAME

The 3D model of the generator frame is created using the NX-CAD software. NX-CAD is the world's leading 3D product development solution. This software enables engineers and designers to bring better products to the market faster. It takes care of the total product definition to serviceability. NX delivers measurable value to manufacturing companies of all sizes and in all industries.

NX is used in a wide range of industries from manufacturing of rockets to computer peripherals. With more than 1 lakh seats installed in worldwide, many Cad users are exposed to NX and enjoy using NX for its power and capability.

2d drafting of generator frame:

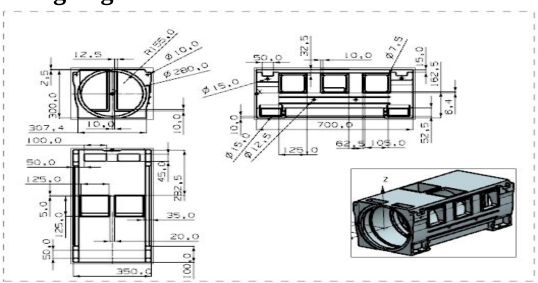


Fig: shows 2d drafting of generator frame

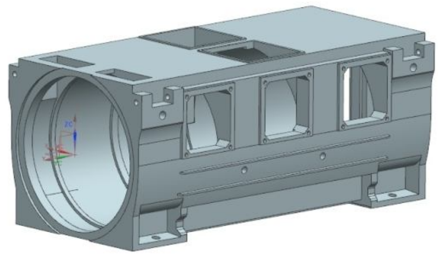


Fig. Shows 3d view of generator frame

V. FINITE ELEMENT ANALYSIS OF GENERATOR FRAME

STRUCTURAL ANALYSIS OF GENERATOR FRAME

Finite Element Modeling (FEM) and Finite Element Analysis (FEA) are two most popular mechanical engineering applications offered by existing CAE systems. This is attributed to the fact that the FEM is the most popular numerical technique for solving engineering problems. The method is general enough to handle any complex shape of geometry (problem domain), any material properties, any boundary conditions and any loading conditions. The generality of the FEM fits the analysis essential for today's complex engineering systems, and designs, where closed form solutions are governing equilibrium equations, are not available. Besides it is an efficient design tool by which designers can perform parametric design studying various cases (different shapes, material loads etc.) analysing them and choosing the optimum design.

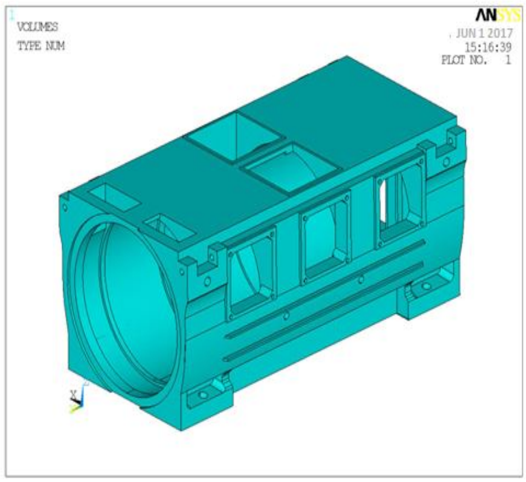


Fig: Shows the Finite element model of the generator frame

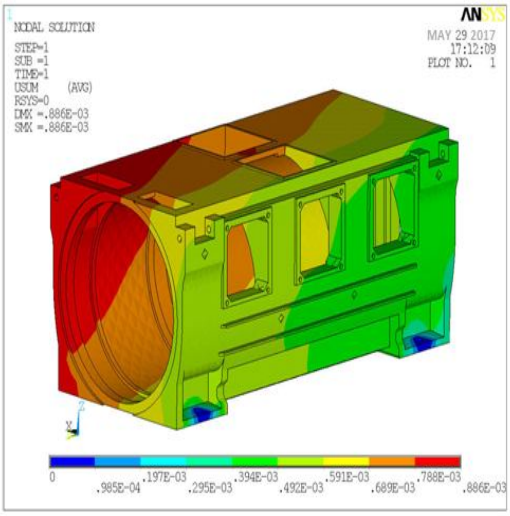


Fig. Total Deflection for static analysis of generator frame

The table. Shows the first ten natural Frequencies of generator frame.

MODE	FREQUENCY	PARTIC.FACTOR			EFFECTIVE MASS		
		X Direction	Y Direction	Z Direction	X-Direction	Y Direction	Z Direction
1	358.453	0.47956	5.01E-02	-8.12E-2	0.229977	2.51E-03	6.60E-03
2	609.42	7.54E-02	-0.45494	0.22675	5.68E-03	0.206971	5.14E-02
3	658.37	3.95E-02	0.21605	0.51504	1.56E-03	4.67E-02	0.265268
4	778.162	-4.07E-02	8.98E-03	-4.97E-2	1.66E-03	8.06E-05	2.47E-03
5	1006.71	-1.61E-02	1.44E-02	-2.78E-2	2.58E-04	2.06E-04	7.73E-04
6	1674.84	1.43E-02	0.25772	-1.26E-2	2.03E-04	6.64E-02	1.58E-04
7	1735.34	-2.51E-02	1.87E-02	1.02E-3	6.29E-04	3.50E-04	1.05E-06
8	1877.61	-5.90E-02	5.71E-04	-3.95E-3	3.49E-03	3.26E-07	1.56E-05
9	1911.39	0.28679	-4.39E-3	-3.04E-3	8.22E-02	1.92E-05	9.24E-06
10	2103.71	6.64E-04	-6.32E-3	1.54E-3	4.40E-07	3.99E-05	2.38E-06

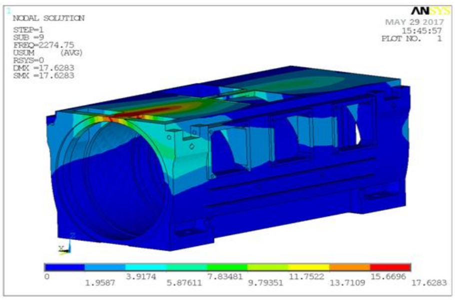


Fig: Shows mode shape of modified generator frame @2274.75Hz

10) Mode shape@2420.11Hz:

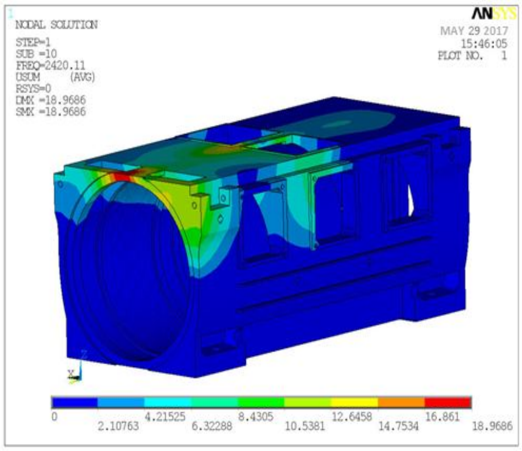


Fig: Shows mode shape of modified generator frame @2420.11Hz

From the above modal analysis results of the modified generator frame, following observations are made:

The total weight of the modified generator frame + generator weight is observed for the analysis as 148.86 Kgs + 180 Kgs = 328.86 Kgs.

The maximum mass participation of 180Kgs is observed in X-direction for the frequency of 617.71Hz.

The maximum mass participation of 239Kgs is observed in Y-direction for the frequency of 1044.64Hz.

The maximum mass participation of 294Kgs is observed in Z-direction for the frequency of 1222.33Hz.

To check the magnitude values of deflections and stresses at the frequencies mentioned above are due to the operating loads or PSD curve, spectrum analysis is carried out on the modified generator frame.

SPECTRUM ANALYSIS OF MODIFIED GENERATOR FRAME:

A Random Vibration Analysis is a form of Spectrum Analysis. The spectrum is a graph of spectral value versus frequency that captures the intensity and frequency content of time-history loads. Random vibration analysis is probabilistic because both input and output quantities represent only the probability that they take on individual values. Random Vibration Analysis uses Power spectral density to quantify the loading. PSD is a statistical measure defined as the limiting mean-square value of a random variable. It is used in random vibration analyses in which the instantaneous magnitudes of the response can be specified only by probability distribution functions that show the probability of the magnitude taking a particular value.

A PSD spectrum is a statistical measure of the response of a structure to random dynamic loading conditions. It is a graph of the PSD value versus frequency, where the PSD may be a displacement PSD, velocity PSD, acceleration PSD, or force PSD. Mathematically, the area under a PSD-versus-frequency curve is equal to the variance (square of the standard deviation of the response). PSD spectrum curves are supplied as a spec. Or are measured and calculated using vibration analysis equipment.

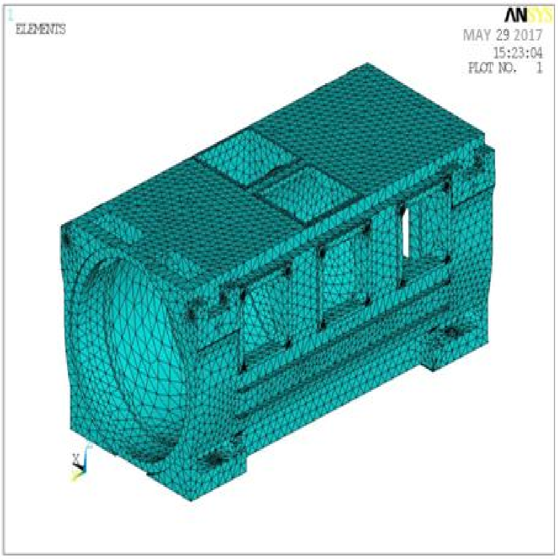


Fig: Shows the Finite element model of the modified generator frame

1σ Deflections and stress in Z-direction:

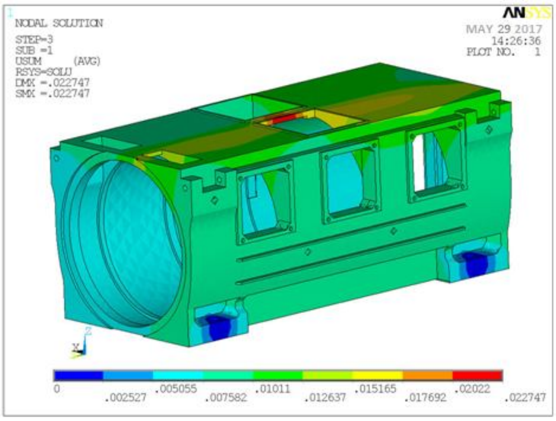
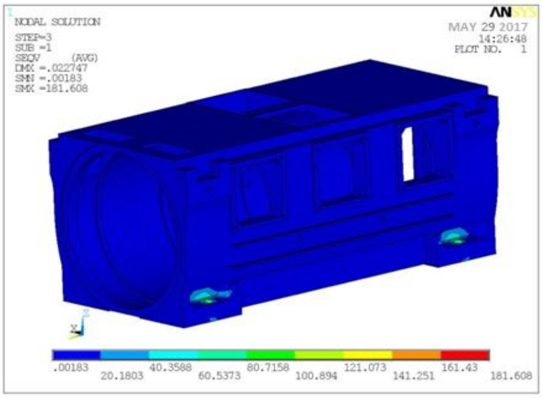


Fig: Shows total deflection of modified generator frame in Z-dir



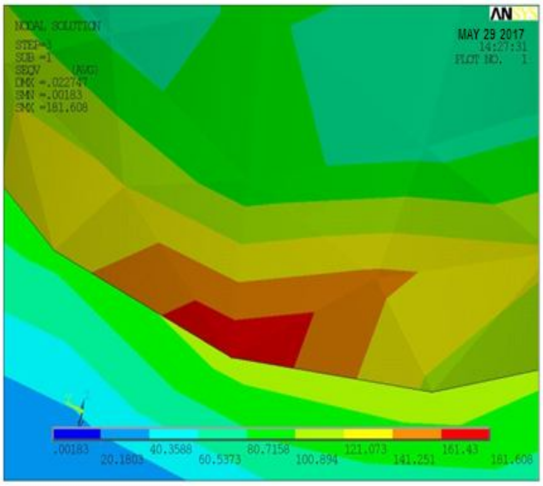


Fig: Shows von misses stress of modified generator frame in Z-dir

The von Mises stress in the structure on mounting frame for 1σ is 71.42 MPa as shown along the path (membrane + bending). The stresses are plotted along the path to identify whether the stresses are real or spurious stresses. From the above plots, the maximum stress is 181.608 N/mm2, which is a spurious stress and can be ignored.

VI. RESULTS AND CONCLUSION

Generator frame and modified generator frame were modelled in NX-CAD software. Both Generator frame and modified generator were analysed for structural analysis.
In structural analysis, generator frame and modified generator frame were analysed for three analyses. They are:

1. Static analysis
2. Modal analysis
3. Spectrum analysis

RESULTS OF GENERATOR FRAME

(A) STATIC ANALYSIS:

The maximum Von Mises stress of 14.03MPa is observed on the generator frame. The Iof the material used for generator frame is 250 Mpa. The, maximum Von Misses stress observed from the analysis is 14.03MPa, which is less than the yield strength of the material.

(B) MODEL ANALYSIS:

The results of the generator frame, following observations, are made:
The total weight of the generator frame + generator weight is observed for the analysis as 147.25 Kgs + 180 Kgs = 327.25 Kgs.
The maximum mass participation 229 Kgs is observed in X-direction for the frequency of 358.45Hz.
The maximum mass participation of 206 Kgs is found in Y-direction for the frequency of 609.42Hz.
The maximum mass participation of 265 Kgs is observed in Z-direction for the frequency of 658.37Hz.

(C) SPECTRUM ANALYSIS:

Case-1: In x-axis
The combination of bending and membrane stresses gives actual stresses developed in generator frame. The maximum Von Mises stress observed in generator frame for 1σ is
From the above results the following conclusions are made:
1σ deflection observed is: 0.079 mm
3σ deflection observed is :3 X 0.079 = 0.237 mm
This implies that only 0.3% of the time generator frame deflection reaches 0.237mm.
1σ stress observed is: 297 Mpa
3σ stress observed is : 3 X 297 = 891 Mpa
This implies that only 0.3% of the time generator frame stress reaches 891 Mpa.

Case-2: In y-axis
The combination of bending and membrane stresses gives actual stresses developed in generator frame. The maximum Von Mises stress observed in generator frame for 1σ is 312.935MPa.

From the above results the following conclusions are made:

1σ deflection observed is: 0.0380 mm
3σ deflection observed is :3 X 0.0380 = 0.114 mm
This implies that only 0.3% of the time generator frame deflection reaches 0.114 mm.
1σ stress observed is: 312 Mpa
3σ stress observed is : 3 X 312 = 936 Mpa
This implies that only 0.3% of the time generator frame stress reaches 936 Mpa.

Case-3: In z-axis
The combination of bending and membrane stresses gives actual stresses developed in generator frame. The maximum Von Mises stress observed in generator frame for 1σ is 337.542MPa.

From the above results the following conclusions are made:

1σ deflection observed is: 0.0307 mm
3σ deflection observed is :3 X 0.0307 = 0.0921 mm
This implies that only 0.3% of the time generator frame deflection reaches 0.237mm.
1σ stress observed is: 337.54 Mpa
3σ stress observed is : 3 X 337 = 1011 Mpa
This implies that only 0.3% of the time generator frame stress reaches 1011Mpa.

RESULTS OF MODIFIED GENERATOR FRAME:

(A) STATIC ANALYSIS:

The maximum Von Mises stress of 3.26MPa is observed on the modified generator frame. The maximum stress is observed on the bolting locations of the modified generator frame. The yield strength of the material used for modified generator frame is 250 Mpa. The maximum Von Mises stress of 3.26MPa is less than the yield strength of the material is 250 MPa.

(B) MODEL ANALYSIS:

The results of the modified generator frame, following observations, are made:
The total weight of the modified generator frame + generator weight is observed for the analysis as 148.86 Kgs + 180 Kgs = 328.86 Kgs.
The maximum mass participation of 180Kgs is observed in X-dir for the frequency of 617.71Hz.
The maximum mass participation of 239Kgs is observed in Y-dir for the frequency of 1044.64Hz.
The maximum mass participation of 294Kgs is observed in Z-dir for the frequency of 1222.33Hz

(C) SPECTRUM ANALYSIS:

Case-1: In x-axis
The combination of bending and membrane stresses gives actual stresses developed in modified generator frame. The

maximum Von Mises stress observed in modified generator frame for 1σ is 51.06MPa.
From the above results the following conclusions are made:
1σ deflection observed is: 0.054 mm
3σ deflection observed is :3 X 0.054 = 0.162mm
This implies that only 0.3% of the time generator frame deflection reaches 0.162mm.
1σ stress observed is: 51.06 Mpa
3σ stress observed is : 3 X 51 = 153 Mpa
This implies that only 0.3% of the time generator frame stress reaches 153 Mpa.

Case-2: In y-axis
The combination of bending and membrane stresses gives actual stresses developed in modified generator frame. The maximum Von Mises stress observed in modified generator frame for 1σ is 42.034MPa.
From the above results the following conclusions are made:
1σ deflection observed is: 0.0201 mm
3σ deflection observed is :3 X 0.0201 = 0.0603 mm
This implies that only 0.3% of the time generator frame deflection reaches 0.060 mm.
1σ stress observed is: 42.034 Mpa
3σ stress observed is : 3 X 42 = 126 Mpa
This implies that only 0.3% of the time generator frame stress reaches 126 Mpa.

Case-3: In z-axis
The combination of bending and membrane stresses gives actual stresses developed in modified generator frame. The maximum Von Mises stress observed in modified generator frame for 1σ is 71.42MPa.
From the above results the following conclusions are made:
1σ deflection observed is: 0.0227 mm
3σ deflection observed is :3 X 0.0227 = 0.0681 mm
This implies that only 0.3% of the time generator frame deflection reaches 0.0681mm.
1σ stress observed is: 71.42 Mpa
3σ stress observed is : 3 X 71 = 213 Mpa
This implies that only 0.3% of the time generator frame stress reaches 213 Mpa.

VII. CONCLUSION

Firstly, Generator frame was modeled in NX-CAD software. The generator frame was imported to ANSYS software to perform structural analysis. In structural analysis, the generator is analyzed for static, modal and spectrum analysis. From the results, the maximum Von Miss the stress of generator frame was less than the yield strength of the material, but in the case of spectrum analysis, the Von Mises stress of generator frame was more than the yield strength of the material. So, modifications are done on the generator frame. The modified generator is analyzed for three analyses. In all three cases, the Von Mises stress of the modified generator frame was less than the yield strength of the material. Hence, it was concluded that the modified generator frame was safe for structural loads.

REFERENCES

[1] Static and dynamic analysis of 2.0mw generator frame by Chaoyi Ding, Luping Dai and Hongche Guo, Applied Mechanics and Materials Vol. 229-231, pp. 933-936, 2012

[2] 'Analysis of Structural Dynamics Turbo Generator Load on Foundation Structure for Estimation Stresses during Vertical Excavation Using Finite Element Method' by Sanjay Gupta, http://shodhganga.inflibnet.ac.in/bitstream/10603/37394/4/13_chapter%207.pdf

[3] Design and performance analysis of a 6 MW medium-speed brushless dig by E. Abdi, M.R Tatlow, R.A. McMahon, PJ. Tavner, 2nd IET Renewable Power Generation Conference (RPG 2013), 2013 page 3.44

[4] Modelling of Turbine-generator and Foundation as Single Degree of Freedom Using Ruaumoko Programme by Shafii Abdullah, Nor Hayati Abdul Hamid

[5] Design and Analysis of Generator and Converters For Outer Rotor Direct Drive Gearless Small-scale Wind Turbines by Yusuf Yasa, Erkan Mese, ICRERA 2014 (3rd International Conference on Renewable Energy Research and Applications), At Milwakuee, USA DOI: 10.1109/ICRERA.2014.7016474

[6] Completion of a 1,120-MVA Turbine Generator for Huadian International Zouxian Power Plant in [7] China by Seijiro Muramatsu, Kado Miyakawa, Mitsuru Onoda, Kazuhiko Takahashi, Kengo Iwashige,

[7] Turbo–generator Foundation by Sukanta Adhikari, IS 2974(Part 3)-1992, http://www.academia.edu/5782802/By_Sukanta_Adhikari