

A Review on Determination of natural frequency of RO plant frame and its non linear stress analysis using ANSYS

Mr. Ankit Shrikrushna Ghatol¹, Prof. S. R. Shekokar²

¹ PG Student ME, Department of Mechanical Engg CAD CAM, Padm. Dr.V. B. Kolte College of Engineering, Malkapur, Maharashtra, India

² Assistant Professor, Department of Mechanical Engg CAD CAM, Padm. Dr.V. B. Kolte College of Engineering, Malkapur, Maharashtra, India

DOI: 10.5281/zenodo.19333117

ABSTRACT

Industrial RO plant frames are made with MS and has significant utilization to mount all necessary peripherals to achieve balanced structural integrity, smooth fluid flow and significant utilization of space. As per the weight capacity and the number of components mounted on the frame structure, it varies in size and shape. As it's a structural part and subjected to loading under various running and vibrating components, is frequently undergoes under vibrations, stresses and the varying loads. Hence the knowing behavior of frame under such conditions is prime area of interest while its development.

Most of such frames are designed by considering metal MS with L, H and rectangular sections. Combinations of such sections are often observed to provide stiffness to withstand at loading and vibrating conditions. A proper method is be followed while designing such frames for RO plant. Different parameters like natural frequency, factor of safety, yield strength, loading positions etc are considered. Despite of all these parameters natural frequency is one of the most important parameter, which directly intended to survive in higher vibrating conditions. Hence knowing natural frequency of frame structure will help to make frame structure stable during vibrations.

In this paper the reviews of most of the recent and important studies are done. All literatures are studied well and the conclusive statement ware drawn on the basis of the observations gained from the all papers.

Keywords: -RO frame, natural frequency, Factor of safety

1. INTRODUCTION

The dynamic behavior of a frame structure describes how it responds to sudden or changing forces like earthquakes or wind by vibrating. This behavior is characterized by the structure's stiffness, mass, and damping and the analysis considers inertia forces, which are a product of mass and acceleration. Key concepts include the structure's natural frequency and period, which influence its response, particularly the phenomenon of resonance where vibrations can amplify dangerously.

Important parameters which affect the frame behavior are as follows.

Forces : Dynamic loads are forces that change over time. Examples include: Earthquakes: Sudden ground motion causes the structure to shake.

Wind : Wind gusts can cause continuous vibration, especially in tall structures. Vehicles : Moving vehicles on a bridge create changing loads.

Machinery : Vibrating machines on a floor can introduce dynamic forces.

Structural Properties:

Mass : The structure's resistance to acceleration. **Stiffness** : The structure's resistance to deformation.

Damping : The structure's ability to dissipate energy through friction and other mechanisms.

Dynamic Response:

Natural Frequency and Period: Every structure has a natural frequency and period at which it will vibrate freely.

Forced Vibration : When a structure is subjected to a time-varying force, it undergoes forced vibration

Resonance : Occurs when the frequency of the applied force matches the structure's natural frequency, leading to a large increase in vibration amplitude that can cause damage.

Dynamic analysis is often performed using computer models that simulate the structure's response to specific loads.

2. LITERATURE SURVEY

Till date research shows us that the design of frame structure is improved and brought to withstand heavy loading. But considering various conditions in which frames needs to be withstand a proper design method needs to be adopted which is varying now a days with different manufacturers.

Aditya Sharma, Sashank Thapa, Bhaskar Goel, Raj Kumar, Tej Singh explained in their study that structural Integrity and its response to the subjected cutting forces during machining, plays a vital role in the performance and the life of the machine tool. The machine tool must meet the demands of the stiffness and easy maintenance; in absence of this the desired machining accuracy during cutting operation is not achieved. While at the same time the machine structure cannot be made heavy which increases the manufacturing cost. The use of stiffener plates is a solution to this problem. However, the selection of the appropriate locations of stiffener plates based on only designer's design knowledge and experience may not lead to desired results. Hence, Structural analysis in conjunction with the structural optimization methods are employed for getting optimized structures. In their study, experimental results obtained from the cutting force measuring machine which was manufactured for the experiment were used to analyse and optimize the structure of machine tool. The analysis and optimization were done with the help of Finite element methods technique (Optistruct solver) and modelling being done in Hypermesh. The optimized structure with stiffeners, led to the decrease in the maximum displacement thus increasing the stiffness of the machine structure while avoiding resonance. [1]

Eser Yazar, Murat Makarac, Şaban Yılmaz, Ömer Apsar, Esra Kan explained in their study that the design and analysis of a base frame for a 3 MW turbo compressor, aiming to develop a robust and reliable structural framework. The design process included comprehensive simulations, incorporating static stress-strain and dynamic vibration analyses to assess the base frame's performance under operational conditions. The static analysis evaluated the total deformation and strain behavior of the base frame, ensuring it can withstand the maximum load without yielding or excessive deformation. Dynamic vibration analysis was performed to identify the natural frequencies and potential resonance issues, minimizing the risk of vibrations that could compromise operational stability. Results from the static analysis revealed that the initial design exhibited a maximum total deformation of 0.24 mm, which was reduced to 0.09 mm in the final design—a 62.5% improvement. Elastic strain values were within safe operational limits for both designs. The effective mass ratio analysis showed significant results at frequencies close to 0 Hz, with negligible impact across the operating frequency range. Based on these analyses, an optimized base frame model was developed, meeting the design criteria for mechanical strength and vibrational stability. The results highlight the importance of integrating deformation, strain, and vibration assessments in the structural design of heavy-duty turbo compressor base frame, contributing to enhanced durability and performance. [2]

G murthi Prasad Rao explained in their study that the two-wheeler chassis consists of the frame, suspension, wheels and brakes. The chassis truly sets the overall style of the two-wheeler. Automotive chassis is the main carriage systems of a vehicle. The frame serves as a skeleton upon which parts like gearbox and engine are mounted. It can be made of steel, aluminum or an alloy. It is essential that the frame should not buckle on uneven road surfaces and that any distortions which may occur should not be transmitted to the body. The frame must therefore be torsion resistant.

The aim of their study is to model a frame using 3D modeling software Pro/Engineer. To validate the strength of a frame, Structural analysis is done by applying the wheel forces. In this analysis ultimate stress limit for the model is determined. Analysis is done for frame using four materials alloy steel, aluminum alloy A360, magnesium and carbon fiber reinforced polymer to verify the best material for frame. Model analysis is also done to determine different mode shapes for number of modes. Analysis is done in ANSYS software. [3]

Pushpadatta, Anand N, Lohith MN their study has thus been taken up to model the Bracket mountings of Hydraulic tank of BH 100 dump truck and analyze it to find the factor of safety (FOS) to check if the mountings are properly designed. Initially, 2D drawings of front and rear brackets, hoist structure, and frame rails are translated into precise 3D models using CATIA software. The assembly process involves meticulous creation of individual bracket components followed by the integration of hoist and rail structures. Subsequently, the entire assembly is imported into ANSYS for Finite Element Analysis (FEA). The ANSYS environment facilitates meshing, application of realistic loads, and imposition of constraints to accurately simulate real-world conditions. The focus shifts to evaluating the structural integrity of the mounting brackets by determining various stress components, including Max. Principal stresses, equivalent stress, shear stress, and deflection. The Factor of Safety (FOS) is then calculated, serving as a crucial metric for assessing the durability of the brackets and ensuring a robust hoist mounting structure. [4]

Mahadev Narasimhe Gowda Pavan D and Venkatesha B K explained that, The frame is the main component in a bicycle to support the external loads acting on it. As all the important accessories are mounted on the frame, different kinds of masses along with weight of rider, braking force, and the response from floor are immediately transferred to it. The frame needs to be strong, stiff, and lighter in weight, that is acquired through combining extraordinary substances and optimizing its structures. In this paper, the static, dynamic and fatigue evaluation of a bicycle frame is achieved to decide the life of frame, deformation, stresses, and von-misses stress appearing

at the frame under loading circumstances for different materials. The frame modelling carried using CATIA V5R20 and simulation performed by ANSYS 19.0 Workbench software. From the analysis it is found that for all the cases the maximum stress is less than yield strength of selected materials, so the design is safe. [5]

Raghavendrasing S. Rajput, S. K. Dubey explained in their study that Block machine foundations are used for machines which produce periodic and impulsive forces and which is operating at low speed i.e. low frequency. For Machine of rotating type and operating at high speed i.e. high frequency primarily framed machine foundation type is used. It may not be possible always to use above option for machine foundation due to local condition, in such cases any other system, which can be combination of above systems; can be used as support for machine foundation. Framed machine foundation are becoming of choice because it is space saving. Analysis and design of block and framed machine foundation is carried out manually for high speed rotary machine and their results are compared. For manual calculations MS excel sheets are prepared for both analysis of framed as well as block machine foundations. It is found that moments and forces in framed machine foundation are less than block machine foundation for high speed rotary machines. For manual calculation MS excel sheet prepared which saves time and arithmetic mistakes. [6]

P. Sriram, B.Balamugundan, S.Thamizh Selvan, R.Saravanan explained in their study that The automotive chassis forms the structural backbone of a commercial vehicle. The main function of the chassis is to support the components and payload placed upon it. When the vehicle travels along the road, the chassis is subjected to various stress distribution and displacement under various loading condition. The method used in numerical analysis is finite element technique to find the critical stress. In this dissertation work we have analysed the monologue and ladder frame for static load condition with the stress, deflection bending moment of chassis of compressed air vehicle (CAV). It is clear that the critical stress acts at the joints and it is being reduced by increasing the side member thickness, connection plate thickness and connection plate length varied. Numerical results showed that stresses on the side member can be reduced by increasing the side member thickness locally. An air powered vehicle uses the expansion of compressed air to drive the piston of a pneumatic cylinder that creates useful work by expanding compressed air. There is no mixing of fuel with air as there is no combustion. [7]

Mahesh Dattatray sankpal¹, D. H. Nimbalkar² explained their study that An Transport structure of frame will go by using high vibration amplitude in operation due to road instability, which will lead to significant changes in structural properties. The vibration of the frame will remove the formation of pressure on certain parts of the larger size. It also causes structural weakness and noise build-up due to loosening of mechanical parts and resulting in the car being uncomfortable. To rectify the above difficulties, This paper evaluates the structural integrity for the Transport Configuration Ancillaries, which contains Transport Frame, Tilting Frame, Locking Pin, Longitudinal Restraint Assembly, Front Spacer Assembly of Longitudinal Restraint, Turnbuckle Assembly of Transport frame, Tiedown Strap Assembly of Transport frame Lifting Pin of Transport frame, Trailer Adapter Plate assembly and Primary longitudinal support plate of Transport Frame. The transport Frame is used to transport the Heavy Vessel, when the frame is attached to the deck of a vehicle trailer. [8]

Heng Jiang , Pengda Zhao, Xinsheng Bi , Tingwen Pei, Jianning Yang , Jiahao Su , Jianhao Dong and Yuxin Bao found in their study that The frame of the six-row self-propelled packaging cotton picker serves as the primary load-bearing structure. During operation, the frame is subjected to multiple vibration signals, which are further intensified by coupling effects. These vibrations negatively impact the machine's operational stability and overall performance. In this study, vibration source tests were designed to collect dynamic response data, enabling systematic analysis of excitation mechanisms and vibration characteristics. Furthermore, a comprehensive analytical approach integrating finite element simulation with experimental analysis was employed to optimize the layout of the vibration sources on the frame. Finally, the frame was validated through modal testing, with multiple measurement points arranged at the interfaces between the frame and the vibration source for vibration tests and time–frequency domain analysis. The results indicate that the final optimized dimensional parameters of the frame were determined as follows: $X_1 = 1575$ mm, $X_2 = 805$ mm, and $X_3 = 275$ mm. Furthermore, time–frequency domain analysis reveals that the natural frequency of the rack designed in this study is effectively separated from the dominant excitation frequency band. This design feature successfully mitigates the risk of resonance, thereby fulfilling the intended performance objectives. [9]

Taimur Rahman, Md. Farhad Momin and Afra Anam Provasha described in their study that The accurate prediction of the fundamental period of steel-braced reinforced concrete (RC) buildings is crucial for optimizing seismic design and ensuring structural safety. Traditionally, empirical formulas provided by building codes such as Eurocode 8 and ASCE 7–22 primarily rely on building height to estimate the fundamental period. However, these height-based models often overlook the significant influence of other structural parameters, such as bracing configurations, bracing lengths, and material properties. This study addresses these limitations by offering a comprehensive evaluation of the factors affecting the fundamental period of steel-braced RC buildings, using advanced computational techniques for more precise and interpretable predictions. A dataset comprising 17,280 building models with varied structural configurations was generated using computational simulations. Key parameters, including total building height, bracing type, bracing length, and building dimensions, were

systematically varied. The study utilized machine learning techniques and employed SHapley Additive exPlanations (SHAP) and Individual Conditional Expectation (ICE) plots as post-hoc interpretability tools to analyze the contributions of structural parameters. Results show that total building height remains the dominant factor, contributing approximately 45% to the predicted fundamental period, while bracing length and bracing type significantly influence the period, reducing it by up to 20%. The inclusion of these parameters improves prediction accuracy and reveals limitations in existing height-based formulas. The study concludes that height alone is insufficient for accurate prediction of the fundamental period in steel-braced RC buildings. Incorporating bracing systems and other structural factors is essential for more reliable seismic design. These findings contribute to the development of more resilient building codes and enhanced seismic performance. [10]

3. RESEARCH GAP

Literature survey done on various studies exposes the limited work on the frame structural behavior as most of the studies are based on the steady state structural analysis. Most of the metals have non linear characteristics. If we consider MS (Mild Steel) as a metal for the frame which is widely used in structures then we must consider its non linear nature to obtain the exact stress and deformation.

Recent studies are exploring the natural frequency of frames, but there is a large scope to find the stress and deformation considering non linear characteristics. It is found that the recent studies are considering only steady state condition of frame while performing FEA analysis, which creates the research gap to understand the actual stress and deformation produced under loading condition on the frame.

To overcome on this research gap, it is important to perform virtual FEA analysis considering non linear characteristics of frame metal. Hence this study focuses on the fulfillment of this research gap by producing exact stress and deformation values considering non linear nature of metal. Also vibration analysis and drop test analysis gives extra dimension to this study and fulfill research gap between recent studies and its requirements.

4. EXPECTED OUTCOME

It is expected to obtain stable deformation and stress concentration of a structural frame so that it can withstand on loading condition. Stresses should be below yield stress value and deformation should be in controlled limit.

5. REFERENCES

[1] Aditya Sharma a , Sashank Thapa a , Bhaskar Goel a , Raj Kumar a , Tej Singh b , *, “Structural analysis and optimization of machine structure for the measurement of cutting force for wood”.

www.sciencedirect.com,

Received 2 July 2021; revised 30 August 2022; accepted 13 September 2022 Available online 24 September 2022, 1110-0168 2022 THE AUTHORS. Published by Elsevier BV on behalf of Faculty of Engineering, Alexandria University.

[2] Eser Yazar¹ , Murat Makaracı^{1,*}, Şaban Yılmaz^{1,2}, Ömer Apsar² , Esra Kan^{2,3}, “Structural design optimization and vibration assessment of a base frame for a 3 MW turbo compressor”. Sound & Vibration. 2025; 59(1): 2156. <https://doi.org/10.59400/sv2156> Received: 28 October 2024 Accepted: 18 December 2024 Available online: 24

January 2025

[3] G murthi Prasad Rao, “Structural Analysis of Two Wheeler Suspension Frame”. International Journal of Research Available at <https://edupediapublications.org/journals>. Special Issue on Conference Papers, Volume 05 Issue 06 March 2018

[4] Pushpadatta B^{*1}, Anand N^{*2}, Lohith MN^{*3}, “Integrated design and structural analysis of bracket mounting of hydraulic tank of BH 100 dump truck: a CATIA-ANSYS approach”. International Research Journal of Modernization in Engineering Technology and Science (Peer-Reviewed, Open Access, Fully Refereed International Journal) Volume:05/Issue:12/December-2023 Impact Factor- 7.868 www.irjmets.com

[5] MAHADEV NARASIMHE GOWDA PAVAN D and VENKATESHA B K, “Modelling and structural analysis of bicycle frame using FEA for different materials”. JOURNAL OF MINES, METALS & FUELS, ICAMMME 2021

[6] IRaghavendrasing S. Rajput, 2S. K. Dubey, “Analysis and Design of Framed and Block Machine Foundation”. IJEDR2001063 International Journal of Engineering Development and Research (www.ijedr.org), ISSN: 2321-9939

| ©IJEDR 2020 Year 2020, Volume 8, Issue 1

[7] P. Sriram^{1*}, B.Balamugundan² , S.Thamizh Selvan³ , R.Saravanan⁴, “Structural Analysis Of Chassis And Assembly on Compressed Air Vehicle”. IJRAR1BQP004 International Journal of Research and Analytical Reviews (IJRAR) www.ijrar.org, 0 IJRAR June 2020, Volume 7, Issue 2 www.ijrar.org (E-ISSN 2348-1269, P-ISSN 2349-5138)

- [8] Mahesh Dattatray sankpal¹, D. H. Nimbalkar², “STRUCTURAL ANALYSIS OF TRANSPORT FRAME”. International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 09 Issue: 07 | July 2022 www.irjet.net p-ISSN: 2395-0072
- [9] Jiang, H.; Zhao, P.; Bi, X.; Pei, T.; Yang, J.; Su, J.; Dong, J.; Bao, Y. “Design and Vibration Analysis of the Frame Structure in a Six-Row Self-Propelled Packaging Cotton Picker.” *Machines* 2025, 13, 1086. <https://doi.org/10.3390/machines13121086>
- [10] Taimur Rahman^{1*}, Md. Farhad Momin¹ and Afra Anam Provash¹, “Comprehensive analysis of structural parameters influencing the fundamental period of steel-braced RC buildings using machine learning interpretability”. Springer, <https://link.springer.com/article/10.1007/s43503-025-00051-z>