Tuijin Jishu/Journal of Propulsion Technology
ISSN: 1001-4055
Vol. 47 No. 02 (2026)

Determination of Natural Frequency of Ro Plant Frame and its Non-Linear Stress Analysis Using Ansys

Ankit Shrikrushna Ghatol[1], S.R. Shekokar[2], Nitin A Kharche[3], Dipak P. Kharat[4]

[1]P.G. Student, Department of Mechanical Engineering, Padm. Dr. V. B. Kolte College of Engineering, Malkapur, Maharashtra, India

[2,3,4]Department of Mechanical Engineering, Padm. Dr. V. B. Kolte College of Engineering, Malkapur, Maharashtra, India

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. Modern FEA packages like ANSYS can be utilized to find the natural frequency of any engineering objects and can also predict the possible stress and deformation in a structure with their location. In this project determination of natural frequency and stress deformation values along is to be determined for RO plant frame with 2000 ltr capacity tank. It can be done by creating CAD model of frame structure in CAD tool like Solidworks and then imported to FEA package ANSYS Workbench. By applying boundary conditions and metal properties to this virtual frame after meshing (CAD Model) natural frequency and stress, deformation is generated. These values are later validated with advanced engineering formulas utilized to find stress and deformation in structure. Factor of safety also calculated with these calculations.

Keywords: Load test, Vibration Analysis, ANSYS, CAE Tool, Natural Frequency, Shape Optimization

Introduction

An industrial RO plant frame is crucial as the sturdy backbone that holds together the entire water purification system, ensuring proper alignment, durability, and safety for high-pressure operations, protecting sensitive components like pumps, membranes, and pipes from vibration and damage, thereby guaranteeing consistent water quality for critical processes, enhancing system lifespan, and supporting cost savings and environmental compliance.

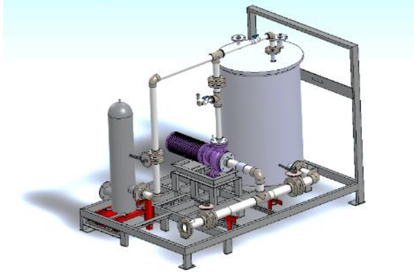


Fig. 1 Typical Industrial RO Plant

Literature Review

Previous research on The aim of their study is to model a frame using 3D modelling 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.

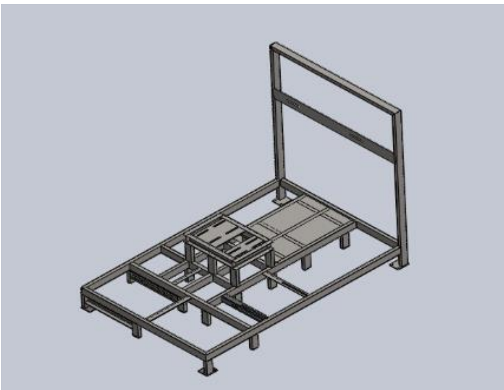
Transport structure of frame will go by using high vibration amplitude in operation due to road instability, which will lead to significant changes in structural properties. The vibration of the frame will remove the formation of pressure on certain parts of the larger size. It also causes structural 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 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.

cad modelling of ro pplant frame

SolidWorks is a powerful, industry-standard 3D CAD (Computer-Aided Design) software from Dassault Systems used by engineers and designers to create precise parts, complex assemblies, and detailed 2D drawings, featuring parametric modeling that links changes across all related files (parts, assemblies, drawings) for efficiency. Its user-friendly interface, familiar Windows tools, and focus on feature-based design make it popular for visualizing products and automating design workflows, from simple components to complex mechanical systems.

Fig. 2 RO Frame CAD model developed on Solidworks



Reverse Engineering Process

Reverse engineering is the process of analyzing an existing physical product, component, or system to understand how it is designed, how it works, and how it was manufactured, and then creating its CAD model, drawings, or technical data from it.

Finite Element Method

FEA is a good choice for analyzing problems over complicated domains (like cars and oil pipelines), when the domain changes (as during a solid state reaction with a moving boundary), when the desired precision varies over the entire domain, or when the solution lacks smoothness. FEA simulations provide a valuable resource as they remove multiple instances of creation and testing of hard prototypes for various high-fidelity situations. For instance, in a frontal crash simulation it is possible to increase prediction accuracy in "important" areas like the front of the car and reduce it in its rear (thus reducing cost of the simulation). Another example would be in numerical weather prediction, where it is more important to have accurate predictions over developing highly nonlinear phenomena (such as tropical cyclones in the atmosphere, or eddies in the ocean) rather than relatively calm areas.

Fea Analysis Of Ro Frame

ANSYS Tool

ANSYS is a finite element analysis (FEA) software used to simulate and analyze engineering problems such as temperature distribution, stress, and fluid flow. It helps predict product performance under different conditions without physical testing. Using ANSYS Workbench, complex models are analyzed efficiently by dividing them into smaller elements.

Descretisation/Meshing of Piston.

Discretization refers to the process of translating the material domain of an object-based model into an analytical model suitable for analysis. In structural analysis, discretization may involve either of two basic analytical-model types, including:

Node-element model, in which structural elements are represented by individual lines connected by nodes. In a 3D system, each node has six degrees of freedom, each either constrained or free. The geometric and material properties of structural elements are then characterized by line elements which simulate their physical behavior by following mathematical relationships. Through application of the direct stiffness method, loading at node locations translates into displacement and stress fields which indicate structural performance.

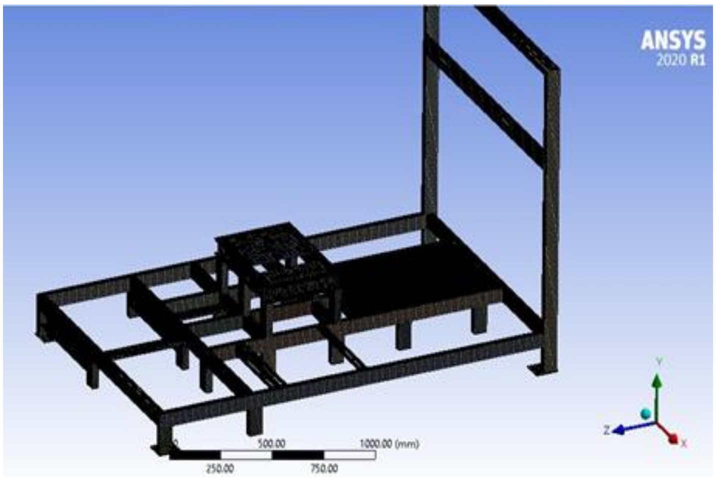


Fig. 3 Meshed view of RO frame.

Boundary Conditions

Same as structural analysis, fixed boundary conditions and loads are applied for vibration analysis. In this case the maximum loading on the RO frame is considered as 2000 Kg (2Tons). But this load is not uniformly distributed on the entire frame. At some location like Tank mounting the load is maximum and the section where filter is mounted, the load is considered as minimum. Whereas at the pump section it is moderate.

Force Calculations

Weight of each component mounted on RO frame including water weight is as follows,

1) Tank Total Weight = 1.2 Ton =1200 kg = 1200*9.81 = 11772 N

2) Inlet Piping Weight = 70 kg = 70*9.81 = 686 N

3) Pump Weight = 30 kg = 30*9.81 = 294 N

4) Outlet Piping Weight = 70 kg = 70*9.81 = 686 N

All the support base plates are fixed on the floor. Therefore fixed boundary conditions are applied on the base plate.

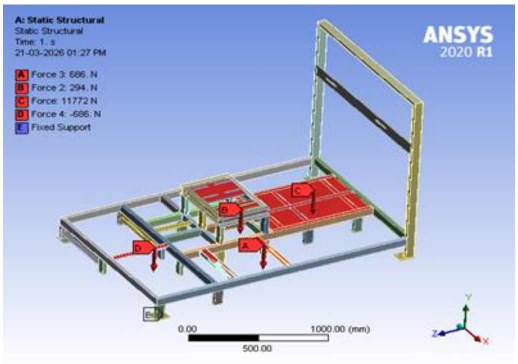


Fig. 4 Load and fixed displacement on RO frame.

Non-Linear Structural Analysis

Nonlinear structural analysis is a computational method used to evaluate structures where loads and deformations are not directly proportional, meaning stiffness changes during loading. Unlike linear analysis, it accounts for large deformations (geometry changes), material yielding (plasticity), or changing contact conditions, providing accurate safety assessments for complex failures.

Generated Results

Total Deformation due to continuous loading on RO Frame. The value is observed as a 0.44 mm. This is the maximum deformation which can observe in to the RO frame. The maximum 0.44 mm deformation is observed at the place where tank is to be mounted. So, the failures can accurse from this position as we observed maximum deformation. Continuous loading on RO frame may cause the failure, hence effect of vibration also needs to check.

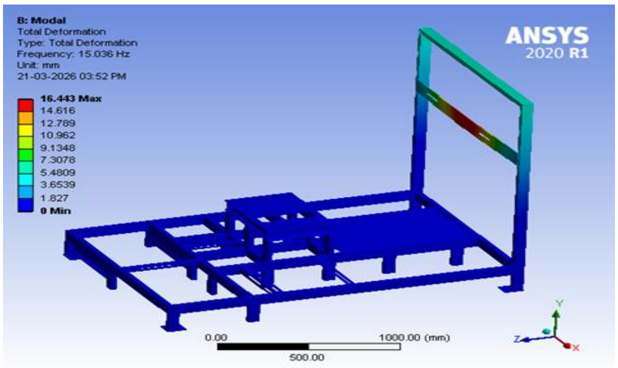


Fig.5 Total Deformation due to continuous loading on RO Frame

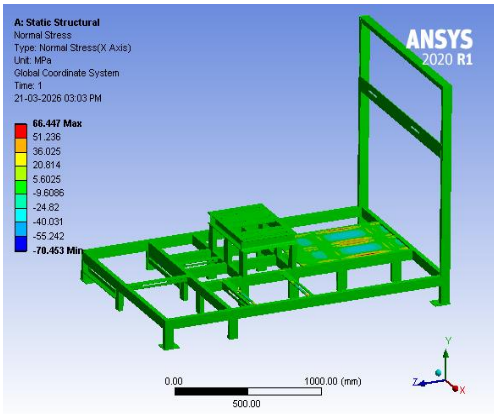


Fig.6 Normal Stresses induced in RO Frame

The value of normal stresses shown in figure are much less (66 MPa) than yield stress of the M. S. Material (350 MPa). Hence the normal stress values will not create failure issues for long life period.

Equivalent stresses are found 63 MPa for the applied loads and are under control. These stresses will not affect the workability of the RO frame and can withstand on such loading for long period of time.

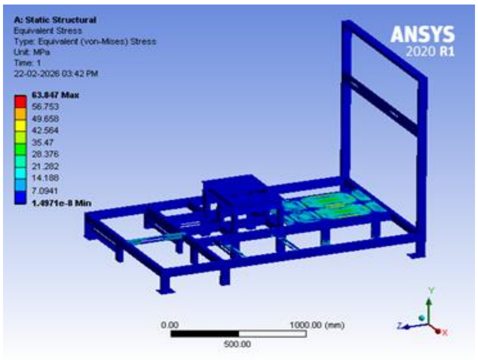


Fig. 7: Equivalent Stresses generated in RO Frame

Vibration Analysis Results

Vibration analysis is carried out for up to 6 mode shape results. Natural frequency is found and the six different mode shapes are studied along with their maximum deviation on their respective frequency values. The detailed results of vibration analysis are as follows. Figure 8 shows the Total Deformation for Mode Shape 1 at 15 Hz frequency. The value is observed as 16.44 mm. As we know that, “More the frequency, safer the object”. Hence frequency is observed as 15 Hz only. But the portion where actually deformation takes place not carrying any load. This is a support structure for the tank. Entire remaining Frame is not having any deformation and does not contain less frequency. Hence the frame is safe at first mode shape.

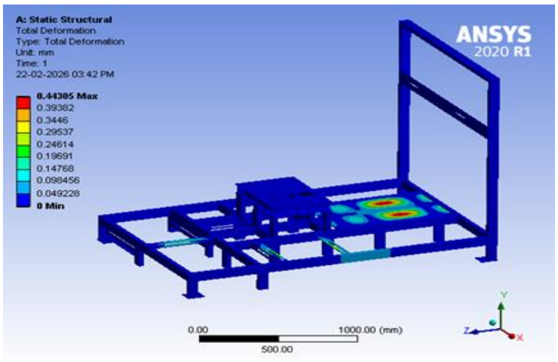


Fig. 8: Total Deformation for Mode Shape 1 at 15 Hz frequency

At second mode shape (Figure 9) the frequency is increased, and the deformation is lowered (12.69 mm) as compared to first mode shape. Also, the area of maximum deformation is not participating in load distribution. Hence the Frame is safe in this mode shape.

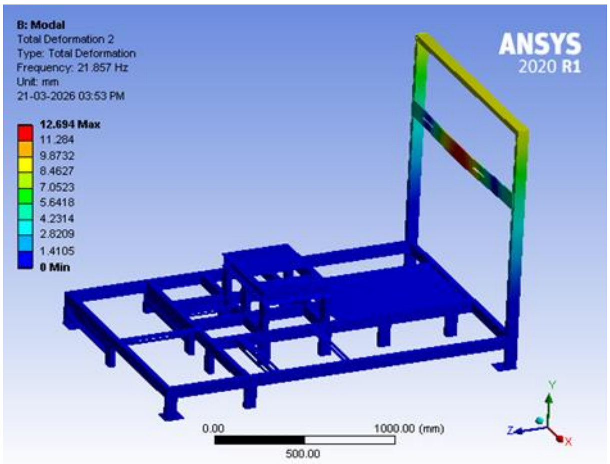


Fig. 9: Total Deformation for Mode Shape 2 at 22 Hz frequency

In third mode shape maximum deformation is found 8.24 mm at 24 Hz frequency. Here the pattern of vibration will change and deviation will accurse on the top edge of the tank supporting frame. Overall other portion of the frame which actually distributing load is kept un-vibrant of has minimum deformation. Hence at this mode shape also the frame is observed as safe.

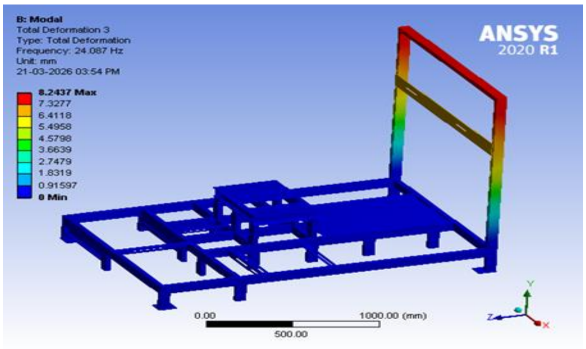


Fig. 10: Total Deformation for Mode Shape 3 at 24 Hz frequency

At mode shape 4 deformation is increased to 12mm at frequency 31.5 Hz. The vibrating frequency is low but the area of vibration is again on tank supporting member. Hence the other portion of the frame is safe and has very less chances of failure due to vibration. Hence the frame is safe on 4th mode shape.

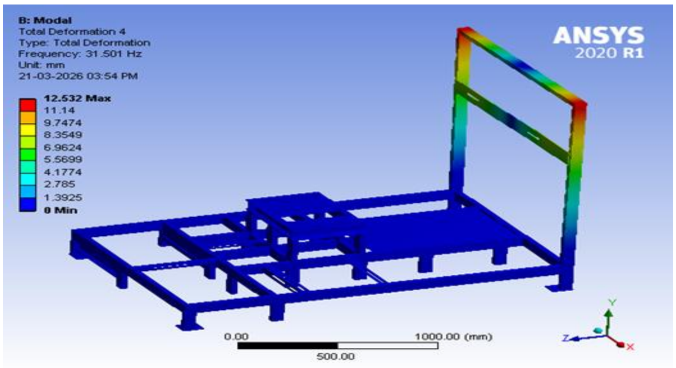


Fig. 11: Total Deformation for Mode Shape 4 at 31.5 Hz frequency

At mode shape five, maximum deformation than previous on is observed (19mm). The frequency is observed as 50 Hz. This deformation will shake the tank supporting member with deviation of approx 20mm. It may cause unbalancing of tank support member. But as tank hold by metal strip with supportive member, all vibrations will be controlled and failure can be prevented. Hence the frame will be on safe side.

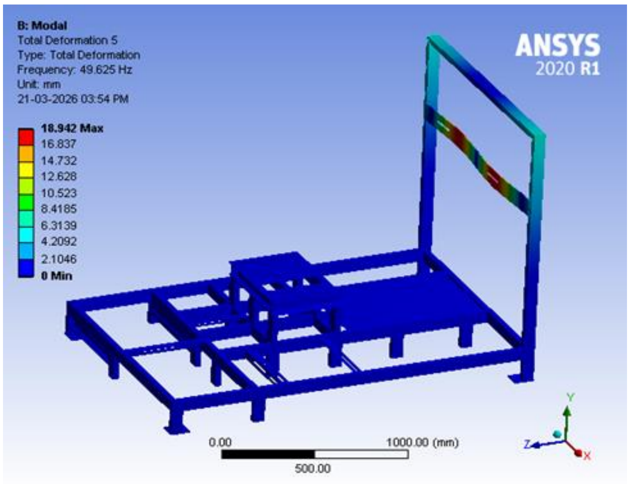


Fig. 12: Total Deformation for Mode Shape 5 at 50 Hz frequency

At the final mode shape 6 the maximum frequency is observed as 91 Hz and the deformation is about 14 mm. Again, the area of deviation concentration is tank supportive member. Hence the frame is safe.

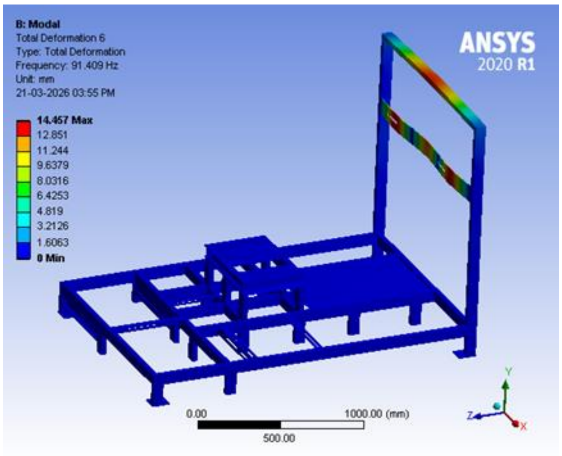


Fig. 13: Total Deformation for Mode Shape 6 at 91 Hz frequency

Result Summary: By observing all the results generated through Vibration analysis, it is observed that the maximum deviation comes at the Tank supporting member and the remaining frame structure is safe. Even the maximum frequency is only 91 Hz, still frame will be safe due to the load carrying members are having negligible deformation.

Table 6.3: Vibration analysis mode shape results

Sr. No.	Frequency (Hz)	Deformation (mm)
1.	15.036	16.44
2.	21.857	12.69
3.	24.087	8.24
4.	31.501	12.53
5.	49.625	18.94
6.	91.409	14.45

Result Synthesis

The detailed synthesis of the results obtained from both non linear structural analysis and vibration analysis can be tabulated in following table.

Table 7.1: Tabulated Results of Frame Analysis.

Sr. No.	Analysis Type	Result	Value
1	Non Linear Static Structural Analysis	Total Deformation	0.44 mm
		Shear Stress	12.195 MPa
		Normal Stress	66.44 MPa
		Equivalent Stress	63.84 MPa
2	Vibration Analysis	Deformation in Mode Shape 1	16.44 mm
		Frequency in Mode Shape 1	15.087 Hz
		Deformation in Mode Shape 2	12.69 mm
		Frequency in Mode Shape 2	21.857 Hz
		Deformation in Mode Shape 3	8.24 mm
		Frequency in Mode Shape 3	24.087 Hz
		Deformation in Mode Shape 4	12.53 mm

		Frequency in Mode Shape 4	31.501 Hz
		Deformation in Mode Shape 5	18.94 mm
		Frequency in Mode Shape 5	49.625 Hz
		Deformation in Mode Shape 6	14.45 mm
		Frequency in Mode Shape 6	91.409 Hz

The detailed observations from the above results are as follows.

1) Deformation in all cases: Deformations observed in all cases are under controlled and does not affect the stability of the frame. Even it does not affect the life of the frame.
2) Shear Stress: The value of shear stress found in acceptable range and hence it would not affect the stability.
3) Normal Stress: Normal stress also in acceptable range
4) Equivalent Stress: Normal stress also in acceptable range

By observing all the results generated through Vibration analysis, it is observed that the maximum deviation comes at the Tank supporting member and the remaining frame structure is safe. Even the maximum frequency is only 91 Hz, still frame will be safe due to the load carrying members are having negligible deformation. The results generated through Nonlinear structural analysis we came to know that the RO frame is stable for continuous loading and the stresses induced are in controlled proportion. Hence the design is safe. But to analyse the RO frame completely, vibration analysis must be carried out.

Conclusion

By observing all the results generated through non linear structural analysis and the vibration analysis. It is found that the RO frame design is safe in both nonlinear conditions and under vibrations. The entire RO frame structure can work safely with maximum loading of 2 tons for longer period of time. Hence the redesigning of the frame is not required. Also, all the mountings (Tank, Motor, filters, piping etc) can be mounted on the frame for longer period of time as the frame can sustain their weigh with stability. Adding of one more supportive angle for the tank can be suggested to increase the support frame stability during vibrations. All other frame members are in safe zone.

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 Yarar1 , Murat Makaracı1,*, Şaban Yılmaz1,2, Ömer Apsar2 , Esra Kan2,3, “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. Raghavendrasing 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. Sriram1*, 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 Rahman1* , Md. Farhad Momin1 and Afra Anam Provasha1, “Comprehensive analysis of structural parameters infuencing the fundamental period of steel-braced RC buildings using machine learning interpretability”. Springer, https://link.springer.com/article/10.1007/s43503-025-00051-z