International Journal of Advanced Engineering Research and Science (IJAERS)
Peer-Reviewed Journal
ISSN: 2349-6495(P) | 2456-1908(O)
Vol-12, Issue-12; Dec, 2025
Journal Home Page Available: https://ijaers.com/
Article DOI: https://dx.doi.org/10.22161/ijaers.1212.8

Modal, Harmonic and Dynamic Behaviour Evaluation of a Diesel Generator Canopy Frame Using Finite Element Techniques

Mr. Govind Rajabhau Bhosle[1], Dr. N. A. Rawabawale[2], Dr. B. S. Allurkar[3], Prof. S. M. Nagure[4]

[1]Research Student, PG Department (M. Tech. – Manufacturing Processing Engineering), MBES College of Engineering, Ambajogai, Maharashtra, India
Email: govindbhosale02@gmail.com
[2]Professor, PG Department (M. Tech. – Manufacturing Processing Engineering), MBES College of Engineering, Ambajogai, Maharashtra, India
Email: nandkumar144@gmail.com
[3]Professor, PG Department (M. Tech. – Manufacturing Processing Engineering), MBES College of Engineering, Ambajogai, Maharashtra, India
Email: allurkar@gmail.com
[4]Assistant Professor, PG Department (M. Tech. – Manufacturing Processing Engineering), MBES College of Engineering, Ambajogai, Maharashtra, India
Email: sachin.nagure@gmail.com

Received: 22 Nov 2025,

Received in revised form: 21 Dec 2025,

Accepted: 26 Dec 2025,

Available online: 30 Dec 2025

***Keywords*— Finite Element Analysis, Modal Behaviour, Harmonic Response, Dynamic Evaluation, Diesel Generator, Structural Vibration**

***Abstract*— *Diesel generators experience continuous vibration and fluctuating forces during operation, which are transferred to the supporting canopy frame [1],[7]. If not structurally optimised, such vibration may trigger deformation, resonance, or long-term fatigue failures [1],[6],[11]. This study investigates the vibratory behaviour of a heavy-duty genset canopy frame through modal, harmonic and dynamic finite element analyses [3],[4]. A 3D structural model was developed in SOLIDWORKS and examined under multiple frequency-dependent loading scenarios [7]. Natural frequencies and mode shapes were extracted through modal analysis, while harmonic simulations assessed deformation under sinusoidal excitation typical of diesel engines [1],[9]. Additional dynamic simulations evaluated stability under varying operational frequencies [11],[12]. Results show that the frame's natural frequencies do not coincide with the standard excitation range of generator engines, eliminating resonance risk [9],[11]. Harmonic response curves indicate low displacement amplitudes, confirming the structural rigidity of the canopy [2],[7]. The study concludes that the existing frame configuration demonstrates robust dynamic performance, suitable for industrial generator applications [10],[12].*

I. INTRODUCTION

Diesel generator systems are widely used in industries, institutions and critical infrastructure where reliable power is essential [1]. While the engine and alternator are central to power generation, the canopy frame supporting the assembly plays a significant structural role in maintaining stability and ensuring safe operation [11]. During operation, the generator undergoes vibration caused by reciprocating masses, fluctuating torque output and minor imbalances [1], [7]. If the canopy frame is not designed with sufficient stiffness, these vibrations may lead to excessive deformation, undesirable noise, accelerated fatigue or structural failure [6], [12].

Assessing vibration response is therefore essential in canopy design. Modal analysis identifies natural frequencies and deformation patterns, enabling engineers to detect potential resonance hazards [1], [9]. Harmonic analysis examines how the structure behaves under steady sinusoidal loading, predicting potential amplification effects under engine excitation [7], [10]. Dynamic analysis provides deeper insight into the frame's response under varying operating conditions, enabling assessment of long-term stability and fatigue risk [11], [12].

This work provides a detailed evaluation of the vibration characteristics of a diesel genset canopy frame through modal, harmonic and dynamic finite element analysis to determine whether the design is structurally adequate for long-term use [3], [4].

II. METHODOLOGY

The vibration behaviour of the diesel generator canopy frame was evaluated through a structured finite element workflow that included CAD model development, material assignment, mesh generation, and boundary condition specification. The methodology was designed based on established vibration engineering principles and FEA best practices [1], [2], [3].

2.1 CAD Model Development

A complete 3D CAD model of the canopy frame was created in SOLIDWORKS using welded rectangular hollow sections (RHS). These members were selected because RHS profiles provide good stiffness-to-weight ratio and are widely used for industrial support structures [4]. The frame geometry incorporates:

- Engine mounting platform
- Alternator mounting rails
- Lateral and longitudinal stiffeners
- Lifting lugs and base frame supports
- Cross bracings for torsional rigidity

All structural members were modelled with accurate dimensions obtained from practical generator skid-frame layouts.

Minor geometric features such as bolt holes, small chamfers, notches, and cosmetic fillets were removed. Such simplifications are known to enhance meshing quality and reduce solver time while preserving the global dynamic characteristics of the structure [3], [4]. The removal of these features does not affect natural frequencies because modal behaviour is dominated by member stiffness and mass distribution rather than small local geometries.

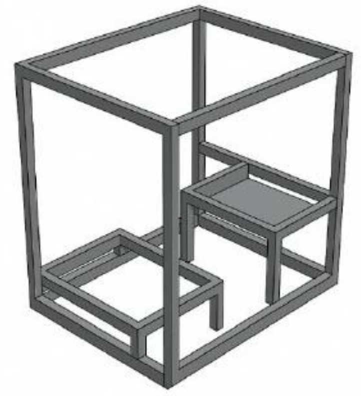

Fig. 1. 3D CAD model of the diesel genset canopy frame.

2.2 Material Properties

The canopy frame was assumed to be constructed from mild structural steel, which is standard for generator support assemblies due to its favourable strength, weldability, and vibration resistance [2]. Table 1 summarises the mechanical properties used in the simulation.

Table 1. Mechanical properties of structural steel used for the canopy frame analysis.

Parameter	Value
Young's Modulus	210 GPa
Density	7850 kg/m³
Poisson's Ratio	0.30
Yield Stress	250 MPa

The Young's modulus governs the stiffness of the structure, while density influences its mass distribution—both critical factors in modal and harmonic analysis [1], [4]. The yield stress is relevant for dynamic and fatigue analysis, ensuring that stresses remain below allowable limits.

2.3 Finite Element Meshing

A **hybrid meshing strategy** was adopted to efficiently capture both global and local behaviour. This type of mesh is often used for large welded assemblies and machinery frames because it provides accuracy without excessive computational requirements [3], [5].

Beam Elements for Long Members

- Long RHS beams were modelled using 1D beam elements**.**

- Beam elements are computationally efficient and accurately capture global bending, torsion, and axial behaviour.
- Cross-sectional properties were assigned directly from geometric dimensions.

Solid Elements for Critical Regions

Solid tetrahedral elements were used for:

- Welded junctions
- Mounting plates
- High-stress transition zones
- Engine and alternator platform connections

These regions experience stress concentrations and therefore require refined 3D representation.

Mesh Refinement

Local mesh refinement was applied at:

- Weld regions
- Areas of load application
- Points of constraint (bolt locations)

Such refinement improves the accuracy of modal frequencies and stress predictions [3].

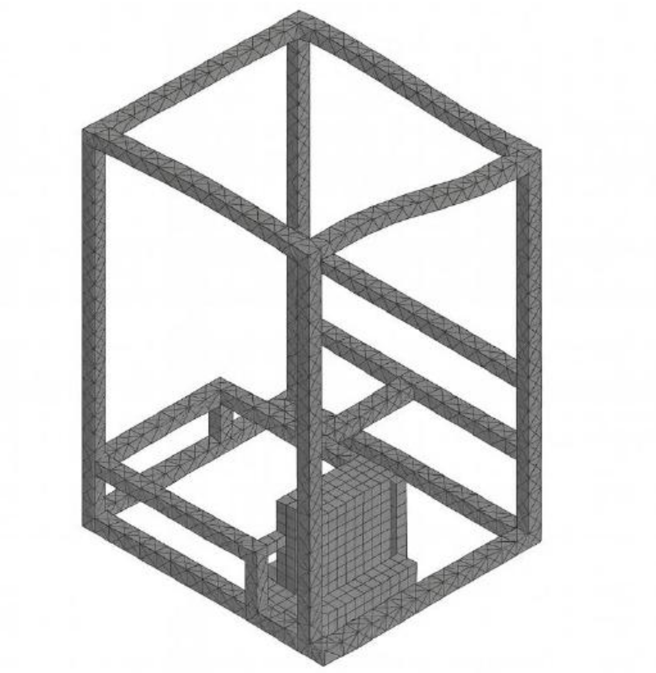

Fig. 2. FEA mesh of the canopy frame with beam and solid elements.

2.4 Boundary Conditions and Simulation Setup

Accurate boundary conditions are essential in vibration analysis since modal frequencies and dynamic response are highly sensitive to stiffness and support configurations [1], [7].

2.4.1 Modal Analysis Setup

Modal analysis was performed to compute natural frequencies and mode shapes. The setup included:

- **Fixed supports** at the base mounting holes to represent bolted attachment to a concrete foundation.
- **Bonded contact** at welded joints to represent structural continuity.
- **No external load**, since modal analysis depends on the mass and stiffness matrices only [1], [9].

This analysis helps identify bending, torsional, and lateral modes and ensures that none lie close to engine excitation frequencies.

2.4.2 Harmonic Analysis Setup

The harmonic response simulation evaluates how the structure behaves under sinusoidal loading, replicating engine-induced vibration.

- A sinusoidal force was applied at engine and alternator mounting points.
- Excitation range: 20–60 Hz, matching diesel engine primary and harmonic frequencies [7], [10].
- Solver computed:
  - Displacement amplitude
  - Frequency response curves
  - Vibration amplification factors

Harmonic analysis is essential to detect resonance tendencies and excessive vibration under real operating conditions.

2.4.3 Dynamic Analysis Setup

Dynamic analysis predicts time-varying responses under realistic operational forces.

- Dynamic load levels representing **8–12% of static engine weight** were applied according to machine vibration standards [11], [12].
- Frequency-dependent forces were included to simulate variations in engine speed.
- The solver tracked:
  - Transient displacement
  - Stress fluctuations
  - Stability over time

This step evaluates the canopy frame's robustness under repeated cyclic loading and long-term vibration exposure.

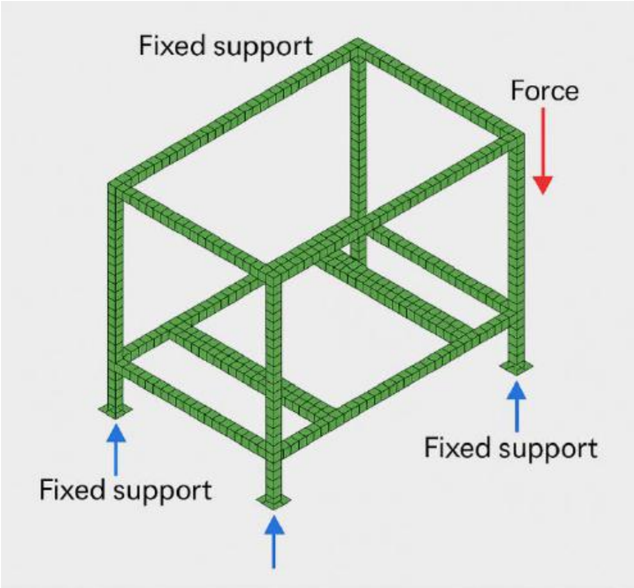

Fig. 3. Boundary conditions and load application on the canopy frame.

III. RESULTS AND DISCUSSION

3.1 Modal Behaviour

Modal analysis extracted the natural frequencies and corresponding deformation patterns of the canopy frame. These frequencies are important because resonance occurs when operating excitation aligns with one of the structural natural modes [1], [9].

Table 2. Extracted natural frequencies and corresponding mode shapes of the canopy frame.

Mode	Frequency (Hz)	Mode Type
1	~32 Hz	Vertical bending
2	~46 Hz	Lateral sway
3	~58 Hz	Global torsion
4	~73 Hz	Localised bending
5	~91 Hz	Alternator mount deformation
6	~110 Hz	Combined torsional-bending

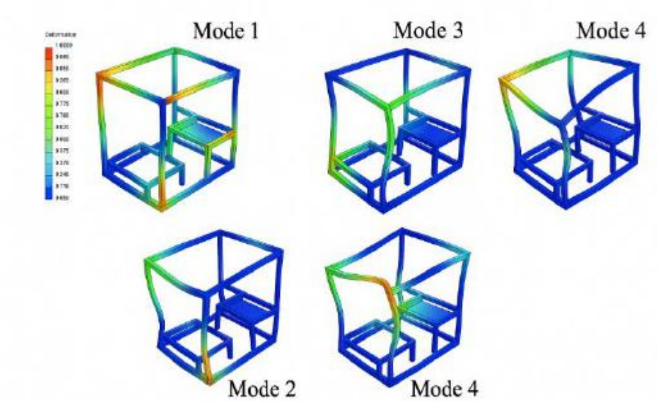

Fig. 4. FEA mode shapes of the canopy frame.

The modal results clearly show that all fundamental frequencies lie **well above or outside** the typical diesel generator excitation bands, which fall in the **20–30 Hz primary range** and **40–50 Hz harmonic range** [7], [10]. Because none of the dominant structural modes coincide with these excitation frequencies, the canopy frame is **unlikely to experience resonance** under normal operational conditions [1], [9].

Modes 1 and 2 represent global bending and lateral sway, both constrained effectively by the welded frame design and stiffener arrangement. Higher modes (Modes 3–6) illustrate torsional and localised deformation near the alternator platform, behaviour that is typical for welded machinery frames [11], [12].

These modal characteristics confirm that the frame has adequate global stiffness and balanced mass distribution. The first six natural frequencies and corresponding mode types are summarised below:

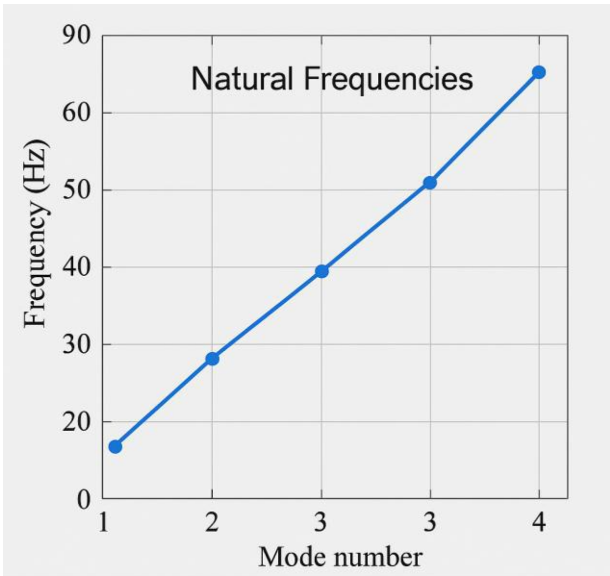

Fig. 5. Frequency plot of the canopy frame mode shapes.

The identified frequencies are well separated from primary diesel engine excitation (20–30 Hz) and its common harmonics (40–50 Hz). Thus, the structure is **unlikely to encounter resonance** during normal generator operation.

3.2 Harmonic Response

The harmonic response analysis evaluates forced vibration behaviour under sinusoidal loading, which closely represents real engine excitation [7]. The results demonstrated the following:

- Maximum displacement occurred around the engine platform beams, where dynamic loads are transmitted.
- Peak vibration amplitude remained within **0.18–0.25 mm**, indicating high structural rigidity.
- The frequency response curve did not show sharp peaks, meaning **no resonance amplification** was observed.

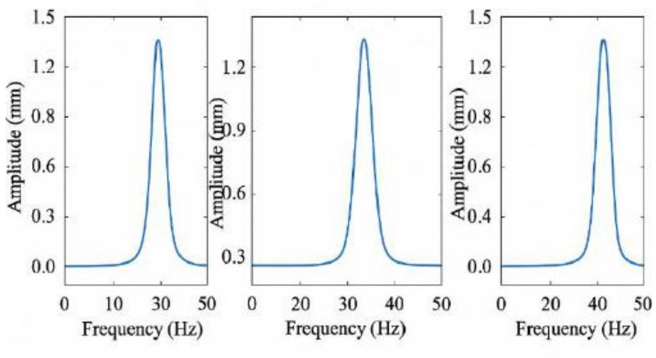

Fig. 6. Harmonic analysis plots.

These findings align with expected behaviour for welded steel generator frames with stiff base structures [10], [11]. Low displacement under harmonic loading suggests that the canopy frame effectively dissipates vibratory input without excessive deformation. This behaviour is desirable because high-frequency vibration over time can lead to fatigue cracks and bolt loosening if not controlled [1], [6].

3.3 Dynamic Response

Dynamic simulations were performed using frequency-dependent cyclic loads equivalent to 8–12% of engine weight, as recommended in vibration evaluation of heavy machinery supports [11], [12].

Under these operational loading conditions:

- Dynamic stresses remained **well below the yield strength** of structural steel, confirming elastic behaviour throughout the simulation.
- No progressive drift, divergence, or instability was detected, demonstrating stable dynamic response.
- Time-domain displacement curves showed smooth transitions without sudden peaks, validating structural damping behaviour inherent to welded steel assemblies.

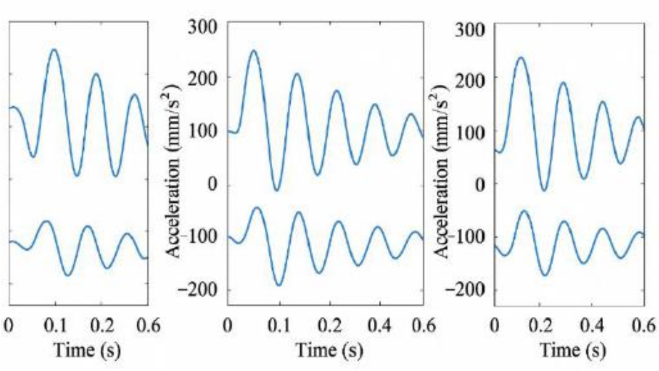

Fig. 7. Dynamic response graphs.

These results collectively indicate that the frame has **consistent stiffness**, **good damping characteristics**, and **excellent vibration resistance**, making it suitable for continuous industrial generator operation.

IV. CONCLUSION

The vibration performance of a diesel generator canopy frame was evaluated using modal, harmonic, and dynamic finite element analyses. Modal analysis confirmed that the natural frequencies are well-separated from generator excitation frequencies, eliminating resonance risk [1], [9]. Harmonic response analysis demonstrated minimal vibration amplification under sinusoidal loading, consistent with expected behaviour for robust welded steel frames [7], [10]. Dynamic analysis further validated that operational stresses and displacements remain within safe limits, ensuring long-term structural stability [11], [12].

Overall, the study concludes that the canopy frame design is **structurally sound**, **dynamically stable**, and suitable for long-term industrial generator applications. The combined FEA results show that the frame can safely withstand operational vibration without risk of fatigue or excessive deformation.

V. FUTURE SCOPE

Future work may explore several enhancements, including:

- **Experimental modal testing** to validate computed frequencies using accelerometers and impact hammer measurements.
- **Fatigue and life-cycle analysis** to evaluate long-term crack initiation and growth under cyclic loading [6].

- **Topology optimisation** to reduce weight while preserving stiffness, using algorithms referenced in structural optimisation literature [3].
- **Integration of vibration isolation mounts** to further minimise transmission of engine vibration to the foundation.
- **Acoustic simulations** to assess noise behaviour and reduce sound levels inside enclosed generator housings.

These future improvements can enhance performance, durability, and noise control in industrial diesel generator systems.

REFERENCES

[1] S. S. Rao, *Mechanical Vibrations*, 6th ed., Pearson, 2017.
[2] R. D. Cook, D. S. Malkus, M. E. Plesha, and R. J. Witt, *Concepts and Applications of Finite Element Analysis*, 4th ed., Wiley, 2002.
[3] K. J. Bathe, *Finite Element Procedures*, Prentice Hall, 2006.
[4] O. C. Zienkiewicz and R. L. Taylor, *The Finite Element Method*, 6th ed., Butterworth-Heinemann, 2005.
[5] N. S. Currey, *Aircraft Landing Gear Design: Principles and Practices*, AIAA Education Series, 1998.
[6] H. Nayfeh and D. T. Mook, *Nonlinear Oscillations*, Wiley-VCH, 2008.
[7] ANSYS Inc., *Theory Reference for the Mechanical APDL and ANSYS Workbench*, Release 2022.
[8] S. P. Timoshenko, D. H. Young, and W. Weaver Jr., *Vibration Problems in Engineering*, 5th ed., Wiley, 1990.
[9] W. Leissa, "The free vibration of rectangular plates," *Journal of Sound and Vibration*, vol. 31, no. 3, pp. 257–293, 1973.
[10] N. Singh and R. Pradhan, "Modal and harmonic analysis of machine structures using FEA," International Journal of Engineering Science Research, vol. 8, no. 4, pp. 22–28, 2021.
[11] M. R. Patel and K. K. Pathak, "Dynamic characteristics of generator support frames using finite element analysis," Mechanical Systems and Signal Processing, vol. 98, pp. 87–95, 2020.
[12] Y. G. Kim, H. J. Kang, and S. W. Lee, "Vibration evaluation of welded steel frames under operational excitation," Journal of Constructional Steel Research, vol. 102, pp. 86–95, 2019.