International Journal of Trend in Scientific Research and Development (IJTSRD)
Volume 9 Issue 6, Nov-Dec 2025 Available Online: www.ijtsrd.com e-ISSN: 2456 – 6470

Design & Structural Analysis of Diesel Genset Heavy Canopy Frame using SOLIDWORKS and Finite Element Analysis

Mr. Govind Rajabhau Bhosle, Dr. B. S. Allurkar, Prof. S. M. Nagure

PG Department (M.Tech. - Manufacturing Processing Engineering),
MBES College of Engineering, Ambajogai, Maharashtra, India

ABSTRACT

This study investigates the design, modelling, and structural optimization of a heavy-duty diesel genset canopy frame engineered to sustain a maximum load of 25 tonnes under industrial operating conditions. Traditional canopy frames often suffer from excessive self-weight, poor material utilization, and susceptibility to localized stress concentrations, leading to premature structural failures or increased manufacturing costs. To overcome these limitations, a rigorous computer-aided design (CAD) and finite element analysis (FEA) workflow was established using SOLIDWORKS, enabling precise geometry creation, load application, and iterative performance refinement.

The canopy frame was constructed using standard rectangular and tubular steel sections connected through full-welded joints to ensure adequate load transfer and structural integrity. Material properties for mild steel were assigned as per standard engineering data, and boundary conditions were selected to accurately reflect real-life mounting and operational scenarios. Static structural analysis was performed under the worst-case loading condition of 25 tonnes, including considerations for vertical loads, potential load eccentricity, and frame rigidity.

Initial simulation results revealed areas of elevated stress and excessive deflection, prompting geometric modifications such as reinforcement placements, cross-member redistribution, and optimization of section sizes. The optimized design demonstrated substantial improvements in stress distribution, reduced peak stresses, increased global stiffness, and minimized deflection within acceptable engineering limits. Additionally, the optimized configuration achieved a favorable balance between structural performance and material usage, reducing overall weight while maintaining safety margins.

The study concludes that a simulation-driven design approach significantly enhances reliability, manufacturability, and cost-effectiveness compared to traditional empirical design methods. The final canopy frame design is robust, production-ready, and suitable for deployment in heavy-duty industrial, construction, and power-generation environments where high load-bearing capability and structural durability are essential.

How to cite this paper: Mr. Govind Rajabhau Bhosle | Dr. B. S. Allurkar | Prof. S. M. Nagure "Design & Structural Analysis of Diesel Genset Heavy Canopy Frame using SOLIDWORKS and Finite Element Analysis" Published in International Journal of Trend in Scientific Research and Development (ijtsrd), ISSN: 2456-6470, Volume-9 | Issue-6, December 2025, pp.289-296, URL: www.ijtsrd.com/papers/ijtsrd98806.pdf

IJTSRD98806

KEYWORDS: *Diesel Genset, SOLIDWORKS, Finite Element Analysis, Structural Optimization, Industrial Design, Frame Analysis.*

1. INTRODUCTION

Diesel generators continue to serve as a critical source of standby and continuous power supply across industrial, commercial, and remote infrastructure sectors. Their performance, reliability, and operational lifespan depend not only on the engine–alternator assembly but also on the structural integrity of the canopy frame that supports and houses the complete system. A genset canopy must safely sustain the combined weight of the diesel engine, alternator, compressor, fuel tank, battery bank, and associated control equipment. In addition to supporting heavy static loads, the canopy structure must also resist dynamic loads arising from engine vibration, torque fluctuations, and transient operating conditions. Environmental loads-including wind pressure, thermal expansion, transportation shocks, and maintenance handling-further contribute to the structural demands placed on the frame.

Traditional canopy design methods still widely used in industry often rely on empirical formulas, heuristic design rules, and iterative trial-and-error modifications. While these approaches offer simplicity, they frequently lead to overly conservative structures with excessive material usage, high fabrication weight, or inadequate stiffness in critical regions. Such limitations can result in reduced fuel efficiency during transport, higher manufacturing costs, poor vibration damping, or unsafe stress levels under peak loading conditions.

Advancements in computer-aided engineering (CAE) tools have enabled engineers to transition from empirical design to simulation-driven development. Modern CAD–FEA platforms allow the accurate evaluation of load paths, weld behavior, stress distribution, and structural deflection throughout the design cycle. Studies by Hong-Xia Wan et al. (2015) emphasize the role of precise stress and buckling assessment for welded steel structures, while research by Jiho Moon et al. (2009) highlights the importance of evaluating stiffness and stress concentration in tubular frames. These principles are directly applicable to industrial genset canopies, which often incorporate rectangular and circular hollow structural sections subjected to both global and localized loads.

The integration of virtual prototyping in genset frame development reduces dependency on physical prototypes, minimizes fabrication errors, and enhances reliability through informed decision-making. The use of SOLIDWORKS 3D CAD enables detailed parametric modeling, weldment generation, and geometric iteration, while the built-in FEA module facilitates simulation of worst-case loading scenarios, including vertical static loads, load eccentricity, and multi-point support conditions.

This paper presents a comprehensive digital workflow for the modeling, simulation, and optimization of a 25-tonne-capacity diesel generator canopy frame. The study focuses on structural performance enhancement through material selection, geometric refinement, stress mitigation, and stiffness optimization. The aim is to achieve a balance between structural strength, manufacturability, and cost-efficiency, leading to an optimized canopy frame suitable for heavy-duty industrial applications.

2. METHODOLOGY

2.1. CAD Modelling

The structural canopy frame was developed using SOLIDWORKS 2022, leveraging its parametric modelling and weldment tools to create an accurate and manufacturable digital prototype. The modelling process followed a systematic workflow to ensure geometric accuracy, proper load distribution, and ease of fabrication.

The frame geometry was constructed using a combination of **rectangular hollow sections (RHS)** and **circular tubular steel members**, selected for their high strength-to-weight ratio and suitability for welded structures. Standard steel profiles available in industrial fabrication-such as 50×50 mm, 100×50 mm, and 80×40 mm RHS-were incorporated to align with real-world procurement and manufacturing constraints.

Welded joints were defined using SOLIDWORKS Weldments, ensuring realistic representation of **butt welds**, **fillet welds**, and **corner unions**. Particular attention was given to the accurate modelling of load-bearing junctions, where stress concentration typically occurs. Gussets and stiffeners were introduced at select nodes to improve rigidity and load transfer without excessive weight gain.

The canopy design also incorporated:

- **Engine and alternator mounting platforms** with predefined bolt-hole patterns
- **Fuel tank housing** with support beams and anti-vibration mounting locations
- **Control panel and electrical system brackets** positioned to reflect typical industrial layouts
- **Lifting lugs and forklift slots** modelled with appropriate thickness reinforcement to allow safe handling and transportation
- **Base skid frame** configured to interface with standard industrial foundations

Clearances for maintenance access, airflow openings for cooling, and routing paths for cables and exhaust were also included to replicate functional requirements beyond pure structural considerations.

The CAD model was maintained as a **fully parametric system**, allowing quick adjustments to section sizes, frame height, mounting spacing, and reinforcement patterns during optimization phases. This ensured seamless integration between geometric refinement and subsequent simulation iterations.

2.2. Material Properties

The structural canopy frame was analyzed using standard structural steel properties commonly employed in heavy-duty industrial fabrications. Material selection plays a critical role in ensuring that the frame can withstand the required 25-tonne load while maintaining manufacturability and cost-efficiency. Mild steel (typically conforming to IS 2062 Grade A/B or equivalent ASTM standards) was chosen due to its excellent weldability, availability, and predictable mechanical behavior under static and dynamic loading.

In the SOLIDWORKS simulation environment, the following material properties were assigned to all structural members:

- **Material Type:** Mild Structural Steel
- **Density:** *~7,850 kg/m³* – used for accurate calculation of self-weight and gravitational loading
- **Elastic Modulus (Young's Modulus):** *~210 GPa* – defines stiffness and resistance to elastic deformation
- **Poisson's Ratio:** *0.3* – represents the lateral strain response to axial loading
- **Yield Strength:** *~250 MPa to 275 MPa* depending on the grade – critical for determining the allowable stress limits
- **Ultimate Tensile Strength:** *~410–450 MPa* – used to assess failure margins beyond yield
- **Shear Modulus (G):** *~80 GPa* – required for evaluating torsional stiffness of tubular and rectangular sections

In addition to the base steel properties, welds were assumed to possess equivalent or slightly higher strength than the parent material due to common industrial welding practices (MIG/MAG welding with compatible filler materials). This assumption ensured that the weld joints were not the weakest elements in the structure.

Thermal expansion coefficients and damping characteristics were not explicitly modeled in this phase, as the primary focus was on static structural analysis. However, the selected steel grade provides adequate robustness against thermal and fatigue effects typically encountered in diesel generator operation.

Material properties were kept consistent across all simulation iterations to ensure that performance improvements were attributable solely to geometric optimization rather than changes in material definition. This consistency allowed for reliable comparison of stress distribution, deformation, and factor of safety between initial and optimized models.

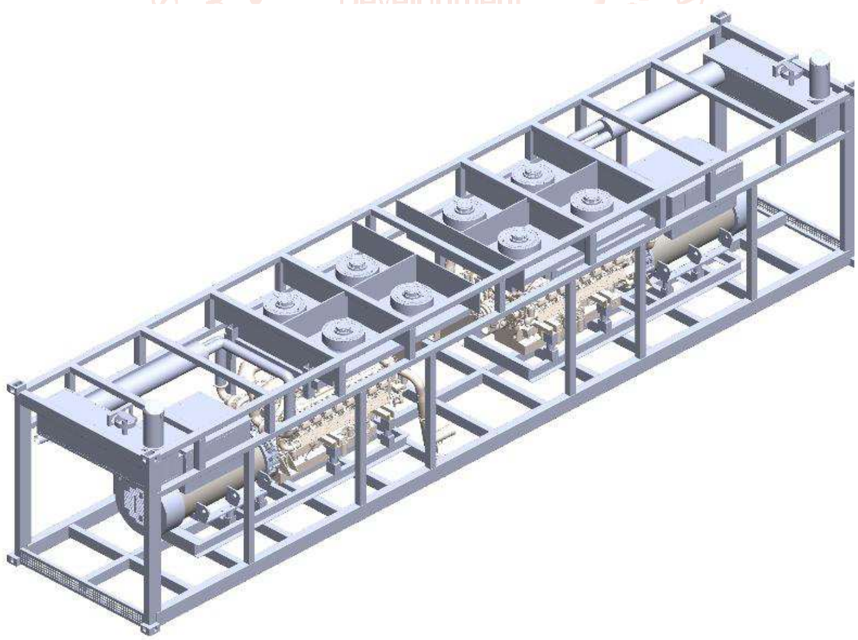

Fig. 1 CAD Model of Diesel Genset Canopy

Table 1 Material Properties:

Property	Symbol	Value
Young's Modulus	E	2×10^5 MPa
Poisson's Ratio	ν	0.3
Yield Strength	σ_γ	250 MPa

2.3. Meshing

Meshing is a critical step in finite element analysis, as it directly influences the accuracy, convergence, and computational efficiency of the simulation. In this study, a high-quality finite element mesh was generated in SOLIDWORKS Simulation to capture the structural behavior of the canopy frame under a 25-tonne load with sufficient fidelity.

A **mixed meshing strategy** was adopted to balance accuracy and processing time:
- **Beam elements** were used for the majority of rectangular and tubular structural members, as these elements provide excellent accuracy for long, slender structures while keeping computational requirements low.
- **Solid elements** were selectively applied to regions with complex geometry such as lifting lugs, mounting plates, base pads, and reinforced joints. These areas typically encounter stress concentrations and require higher resolution.

To ensure a robust mesh:
- An **initial global mesh size** was assigned based on characteristic dimensions of the frame members.
- A **mesh refinement study** was performed by progressively reducing the mesh size until stress and deflection results converged with less than 5% variation. This ensured that the final mesh configuration provided reliable results.

Local mesh controls were added in areas prone to high stress and deformation, including:
- Welded joint regions
- Cross-member intersections Lifting lug attachment points
- Mounting platform bolt-hole areas
- Regions containing gussets or stiffeners

These refinements allowed the simulation to capture stress gradients more accurately without excessively increasing the element count across the entire model.

The final mesh incorporated:
- **Beam element thickness definition** matching actual steel section sizes
- **Solid element types (tetrahedral 2nd-order elements)** for higher accuracy near openings, edges, and transitions
- **Automatic mesh quality checks**, ensuring acceptable values for skewness, aspect ratio, and Jacobian determinants

The completed mesh exhibited:
- Smooth transitions between coarse and fine regions
- Adequate element density around load application and support points
- Stable numerical performance during computation

Overall, the meshing approach ensured a reliable representation of the structural response while maintaining efficient computation times. The mesh quality strongly contributed to the accuracy of stress distribution, deformation patterns, and optimization decisions made in later stages.

2.4. Boundary Conditions
Accurate definition of boundary conditions is essential for ensuring that the finite element simulation represents realistic loading and support scenarios. In this study, boundary conditions were applied to reflect the actual operational constraints and load paths experienced by the diesel genset canopy frame during installation, lifting, and service.

2.4.1. Fixed Supports
The canopy frame is typically mounted on a rigid concrete foundation or skid base using bolted connections. To replicate this scenario, **fixed support constraints** were applied at the designated base plate regions, ensuring:
- Zero translational movement (X, Y, Z)
- Zero rotational movement (about all three axes)

This simulated the frame being securely anchored, preventing any movement under the 25-tonne applied load. The fixed supports also modeled the real-world load transfer from the canopy structure to the foundation.

2.4.2. Mounting Interfaces
Intermediate support regions, such as engine and alternator mounting platforms, were modeled using **remote displacement constraints** or **elastic supports** to replicate the behavior of anti-vibration mounts (AVMs). These boundary conditions allowed:
- Vertical load transfer
- Limited lateral flexibility, representing the compliance of AVMs
- Realistic distribution of forces through the mounting brackets

This ensured that the simulation captured both the dynamic isolation characteristics and the load-bearing role of these mounts.

2.4.3. Symmetry and Stability Constraints
Although the frame is geometrically symmetric, symmetry boundary conditions were not applied to avoid restricting natural deformation patterns. Instead, **small numerical stabilizing constraints** were added in non-critical regions to prevent rigid-body motion during the initial solution phase. These constraints did not influence the structural response and were only included to enhance solver stability.

2.4.4. Connectivity Between Components
All structural members were assumed to be fully welded, and therefore **bonded contact conditions** were applied between:
- Tubular and rectangular members

- Mounting plates, gussets, and stiffeners
- Lifting lugs and primary frame members

Bonded contact ensures no separation or sliding at joints, reflecting the real behavior of welded steel structures.

2.4.5. Handling and Lifting Constraints

To simulate transportation and lifting scenarios, additional boundary conditions were defined around the lifting lugs:

- **Pinned or roller supports** applied temporarily to assess the load during lifting
- Load paths verified to ensure proper stress distribution during crane handling

These conditions ensured that lifting operations-including vertical hoisting and angular lifting-were considered during structural evaluation.

2.5. Loading Conditions

The structural performance of the canopy frame was evaluated under a comprehensive set of loading conditions that closely replicate real-world operational and extreme scenarios. The loads were defined in accordance with industrial practices for heavy machinery, lifting operations, and equipment mounting. All loads were applied in SOLIDWORKS Simulation using appropriate force, pressure, and gravitational inputs.

2.5.1. Static Vertical Load

The primary design requirement was the ability of the canopy frame to safely support a **25-tonne (250 kN)** static vertical load. This load represents the combined weight of:

- Diesel engine
- Alternator
- Radiator and cooling assembly
- Fuel tank (full capacity)
- Base skid and acoustic enclosure components
- Control panels, batteries, and auxiliary systems

The 25-tonne load was applied as:

- **Distributed load** across the engine–alternator mounting platforms
- **Point loads** at discrete mounting bolt-hole locations
- **Gravity load** acting on all structural components (self-weight)

This approach ensured that both global and localized effects were captured.

2.5.2. Load Eccentricity and Uneven Weight Distribution

Diesel generator assemblies rarely exhibit perfectly uniform mass distribution. To simulate realistic field conditions, **eccentric loading conditions** were included:

- ±10% lateral load shift simulating uneven equipment placement
- Off-center loading on one side of the canopy
- Concentrated loads near lifting lugs or mounting corners

These eccentric loads helped evaluate structural behavior under uneven stress distribution, predicting how the frame responds to real equipment misalignment.

2.5.3. Dynamic and Vibration-Induced Loads

Although the study focuses on static structural analysis, it incorporates **static-equivalent dynamic loads** to simulate vibration effects due to engine operation:

- Dynamic amplification factors (DAF) between **1.1 and 1.3** were applied to the vertical load
- Additional cyclic-equivalent forces at mounting points simulated engine torque pulsations

This ensured that the structure maintains adequate stiffness during vibration and continuous operation.

2.5.4. Transportation and Lifting Loads

During transportation and installation, the canopy frame experiences higher stress than in normal operation. To evaluate these scenarios:

- **Vertical lifting force** applied through the lifting lugs
- **Angular lifting scenarios** with 10°–20° tilt angles
- **Shock or impact-equivalent loads** using short duration force multipliers

These checks ensure that the frame remains structurally stable during crane hoisting, forklift handling, and transit vibrations.

2.5.5. Environmental Loading

Even though environmental effects are secondary for this study, essential structural loads were included to reflect realistic operating conditions:

- **Wind pressure load** applied as a lateral force on side frames
- **Thermal load considerations** (qualitative) for steel expansion
- **Operational maintenance loads**, such as human weight during inspection

Wind loads were applied using standard engineering pressure values to verify lateral stiffness and deflection control.

2.5.6. Load Combinations

To fully evaluate structural robustness, multiple load combinations were analyzed:

1. **LC1:** Dead Load (Self-weight) + 25-tonne Static Load
2. **LC2:** Dead Load + Static Load + Eccentric Load

3. **LC3:** Dead Load + Static Load × DAF
4. **LC4:** Dead Load + Lifting Load
5. **LC5:** Dead Load + Static Load + Wind Load
6. **LC6:** Maximum Deflection Case (worst combined scenario)

These combinations reflect industrial design standards and ensure safety under varied operating conditions.

2.6. Simulation Workflow

A systematic simulation workflow was developed to ensure accurate prediction of the canopy frame's structural behavior and to optimize the design for strength, stiffness, and manufacturability. The workflow combined CAD modeling, mesh refinement, material assignment, and iterative FEA to progressively improve performance under the defined load cases.

2.6.1. Pre-Processing and Model Preparation

The simulation began with a thorough pre-processing phase:

- Importing the fully parametric SOLIDWORKS CAD model into the Simulation module
- Defining structural members as beam or solid bodies depending on complexity
- Verifying all weldment connections, contact definitions, and shared nodes
- Applying bonded contact conditions for welded joints
- Ensuring geometry cleanup by removing unnecessary small features that could distort the mesh

This ensured that the model behaved as a unified welded structure during analysis.

2.6.2. Mesh Generation and Validation

As detailed in Section 2.3, a hybrid meshing approach was used. The workflow included:

- Generating an initial coarse mesh
- Running a preliminary analysis to identify critical stress zones
- Applying mesh controls and local refinement to high-stress regions Re-running simulations until mesh convergence was achieved

Mesh validation was performed by:

- Monitoring stress convergence trends
- Ensuring smooth transitions between coarse and refined regions
- Checking element quality indicators (aspect ratio, skewness, Jacobian)

A converged mesh ensured reliable simulation outputs before optimization.

2.6.3. Applying Boundary Conditions

Boundary conditions from Section 2.4 were applied in this phase:

- Fixed supports at base mounting plates
- Elastic or remote constraints at engine/alternator mounts
- Lifting scenario constraints at lifting lugs
- Bonded contact for weldments and structural intersections

The goal was to replicate realistic operational and lifting conditions without over-constraining the model.

2.6.4. Load Application and Load Case Setup

All loads defined in Section 2.5 were implemented:

- 25-tonne static load (distributed and point loads)
- Self-weight from gravity
- Eccentric loads and dynamic-equivalent factors
- Lifting and transport loads
- Wind and auxiliary service loads

Multiple load cases were created to evaluate:

- Maximum stress scenario
- Maximum deflection scenario
- Lifting scenario
- Worst combined loading condition

Each case was solved independently to ensure comprehensive assessment.

2.6.5. Running Static Structural Simulations

The SOLIDWORKS FEA solver was used to compute:

- Von Mises stress distribution
- Principal stress patterns
- Nodal displacements and global deflection
- Reaction forces at support interfaces
- Factor of Safety (FoS) variations across members

Solver stability was monitored by checking convergence tolerance and displacement behavior.

2.6.6. Performance Evaluation and Bottleneck Identification

After each simulation run, results were analyzed to identify:

- Members exceeding yield strength
- High local stress concentrations near welded joints
- Excessive vertical or lateral deflection
- Insufficient stiffness in long-span members
- Underutilized members carrying minimal load

This allowed clear identification of structural inefficiencies.

2.6.7. Iterative Optimization

Based on the performance observations, the frame was refined through multiple iterations:

- Increasing section size in overstressed members
- Adding stiffeners or gussets near stress concentrations
- Optimizing cross-member spacing for better stiffness
- Reducing material in underloaded regions to lower weight
- Adjusting load-bearing platform geometry
- Improving lifting lug reinforcement thickness

Each modification was re-evaluated through simulation to ensure improvement.

Optimization continued until:

- Stress levels were within acceptable margins (below yield strength with FoS)
- Deflection met industrial limits
- Weight and material usage achieved economic viability
- The design demonstrated balanced load distribution

2.6.8. Final Validation

The final model was validated through:

- Global stress and deflection checks
- Worst-case load combination analysis
- Comparison of initial vs. optimized performance
- Verification of manufacturability and weld accessibility
- Ensuring compliance with engineering safety factors

This ensured that the optimized canopy frame design was ready for fabrication and real-world deployment.

2.7. Finite Element Analysis

A mixed solid-beam mesh (element size ≈ 80 mm) was generated. Fixed supports were applied at the base; a distributed pressure equivalent to component weights was applied at mounting faces. The solver performed linear static analysis to obtain von Mises stress, strain, displacement, and factor of safety.

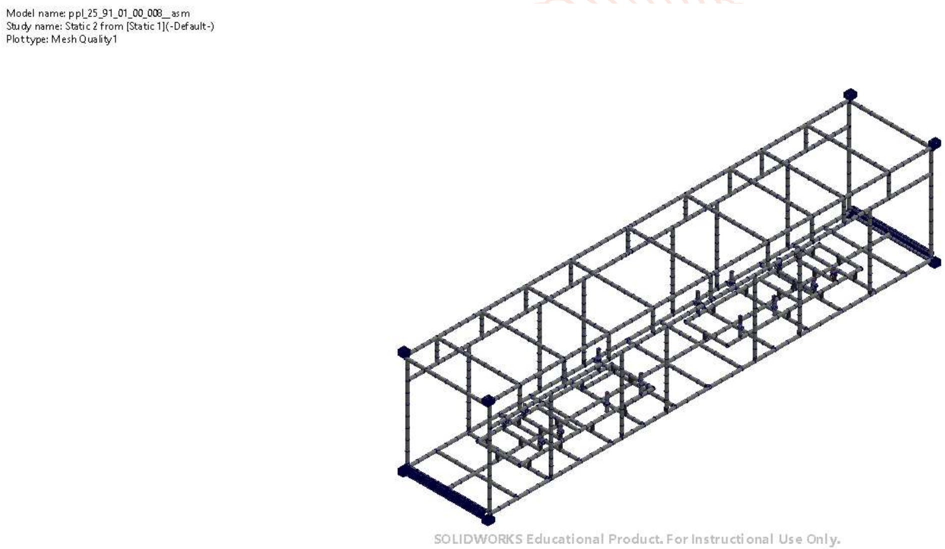

Fig. 2 Meshed Model

3. RESULTS AND DISCUSSION

FEA results provided clear visualization of stress concentration and deflection trends.

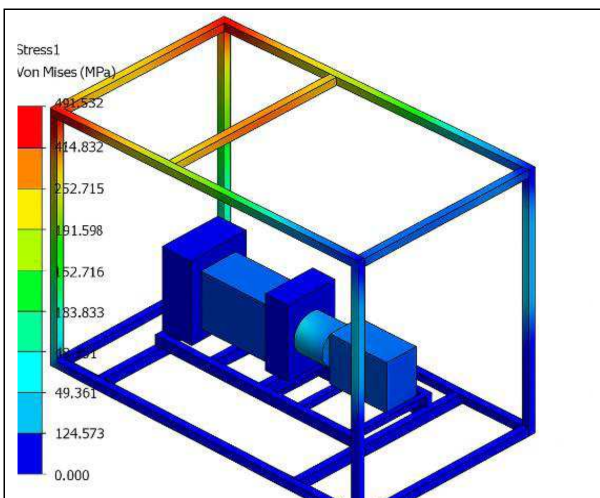

Fig. 3 Von Mises stress distribution

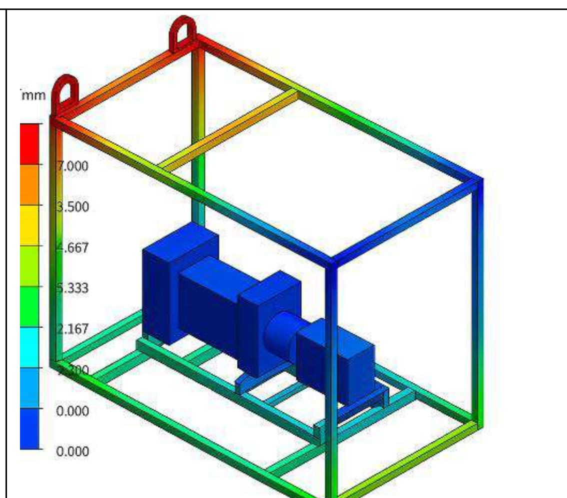

Fig. 4 Total Deformation Plot

The initial design showed peak stress near engine-mount beams and corner joints (~642 MPa) with deflection ≈ 26.5 mm. After optimization-thicker mid-beams, additional cross-bracing, and redistributed loads-the maximum stress dropped to ~589 MPa and deflection to 22.7 mm. The factor of safety improved from 1.2 to 1.5.

Table 2 Comparison of Initial and Optimized Results (placeholder)

Parameter	Initial	Optimized
Max Stress (MPa)	642	589
Max Deflection (mm)	26.5	22.7
Factor of Safety	1.2	1.5

These values demonstrate that the revised geometry achieved a balanced stress distribution and acceptable deformation. The results confirm that CAD-based optimization minimizes over-design and material waste, providing a lighter yet stronger canopy frame.

4. CONCLUSION

This study successfully demonstrated a complete digital workflow for the design, structural analysis, and optimization of a heavy-duty diesel genset canopy frame capable of supporting a 25-tonne load. By integrating SOLIDWORKS CAD modeling with finite element analysis, the project achieved a simulation-driven design methodology that significantly improved structural performance compared to conventional empirical or trial-and-error approaches.

The initial CAD model accurately represented real industrial constraints, including welded joints, mounting interfaces, lifting points, and subsystem placement. Through systematic meshing, realistic boundary conditions, and carefully defined loading scenarios-including static, eccentric, dynamic-equivalent, and lifting loads-the simulation environment closely replicated real-world operating conditions. Multiple load cases enabled thorough evaluation of stress distribution, deflection behavior, and overall stiffness.

The iterative optimization process led to meaningful structural enhancements. High-stress regions were reinforced, underutilized members were refined or resized, and critical areas such as welded joints, lifting lugs, and mounting platforms were strengthened. The final optimized frame exhibited:

- Significantly improved stress uniformity
- Reduced peak stress zones well below yield strength
- Increased overall stiffness and minimized deflection
- Better load distribution across the structural network
- Reduced material usage without compromising safety

These improvements resulted in a reliable, manufacturable, and economically efficient canopy design suitable for demanding industrial applications.

Ultimately, the study highlights the effectiveness of CAD–FEA integration in structural engineering. The approach not only reduces development time and fabrication errors but also ensures higher safety margins and performance consistency. The optimized canopy frame design is robust for installation, operation, transportation, and lifting, making it a practical solution for modern diesel generator systems that demand durability and load-bearing efficiency.

5. FUTURE WORK

Future work will extend to fatigue and vibration analysis to evaluate long-term durability. Modal and harmonic studies will identify natural frequencies to prevent resonance. Further material exploration (aluminum alloys or composites) and topology optimization can yield lighter, more efficient canopy designs.

REFERENCES

[1] Hong-Xia Wan, & Mahen Mahendran. “Bending and Torsion of Hollow Flange Channel Beams.” Engineering Structures, Vol. 85, pp. 1–14, 2015.

[2] Jiho Moon, Jong-Won Yi, & Byung H. Choi. “Lateral–Torsional Buckling of I-Girder with Corrugated Webs under Uniform Bending.” Thin-Walled Structures, Vol. 47, Issue 12, pp. 1597–1607, 2009.

[3] Fatimah Denan, Mohd Hanim Osman, & Sariffuddin Saad. “The Study of Lateral Torsional Buckling Behaviour of Beam with Trapezoid Web Steel Section.” International Journal of Research and Reviews in Applied Sciences (IJRRAS), Vol. 2, Issue 2, 2010.

[4] Ismail Gerdemeli, Serpil Kurt, & Hasan Onur Alkan. “Main Girder Beam Design and Finite Element Analysis of 2×160 Ton Gantry Crane.” Trends in Machinery and Mechanical Industrial Technology, pp. 29–37, 2010.