# Multi-Hazard Finite Element Assessment of Telecom Equipment Mounting Brackets under Wind, Vibration, and Seismic Loads

[1]Amrita Singh, [2]Dr. Sujeet Singh Baghel
[1]M. Tech Scholar, Department of Mechanical Engineering, Oriental Institute of Science & Technology
[2]Assistant Professor, Department of Mechanical Engineering, Oriental Institute of Science & Technology
[1]amritasingh51192@gmail.com, [2]sujeetsinghbaghel@gmail.com

**Abstract- Telecommunication equipment mounting brackets are critical structural components that ensure the safety, stability, and uninterrupted operation of modern communication infrastructure. These brackets are continuously exposed to extreme environmental and operational loads, including high-speed winds, steady-state and random vibrations, and seismic excitations, which can significantly affect their structural integrity. In this study, a comprehensive finite element analysis (FEA) framework is developed to evaluate and optimize the performance of telecom mounting brackets under multi-hazard loading conditions. Two bracket designs were investigated across three mounting configurations bookshelf, portrait, and combined bookshelf–portrait using wind force analysis in accordance with TIA-222-G, harmonic vibration analysis, random vibration analysis based on power spectral density (PSD), and earthquake response spectrum analysis following GR-63 core guidelines. The numerical results demonstrate that Design-2 consistently outperforms Design-1 across most loading scenarios. Under wind loading, Design-2 exhibited reduced deformation and stress, with the portrait configuration achieving the lowest deformation of 0.65 mm and a von Mises stress of 405 MPa. Harmonic and random vibration analyses identified the Z-direction as the most critical axis, with Design-2 showing improved vibration control, particularly in the combined configuration where Z-direction stress was reduced from 32 MPa to 16 MPa. Earthquake analysis revealed that the Y-axis governs seismic response, and Design-2 achieved a significant reduction in Z-axis stress (65.7 MPa) compared to Design-1 (106.85 MPa). Overall, the results confirm that Design-2 provides superior structural resilience, balanced vibration response, and enhanced safety under multi-hazard conditions, making it a robust solution for telecom infrastructure applications.**

**Keywords- Telecom mounting bracket; Finite element analysis; Wind load analysis; Harmonic and random vibration; Earthquake response spectrum; Structural optimization**

## I. INTRODUCTION

The telecommunications sector forms a critical backbone of modern infrastructure, enabling continuous connectivity across residential, commercial, and industrial domains. As reliance on communication networks increases, the structural reliability of telecom equipment becomes increasingly significant [1]. Telecom components such as antennas, base stations, and transceivers are commonly mounted on brackets fixed to towers, rooftops, or poles. These mounting brackets are exposed to dynamic environmental loads, including wind forces, mechanical vibrations, and seismic events [2]. Any structural failure of these brackets can result in equipment damage, service interruptions, and safety risks, highlighting the necessity for robust and resilient bracket designs [3].

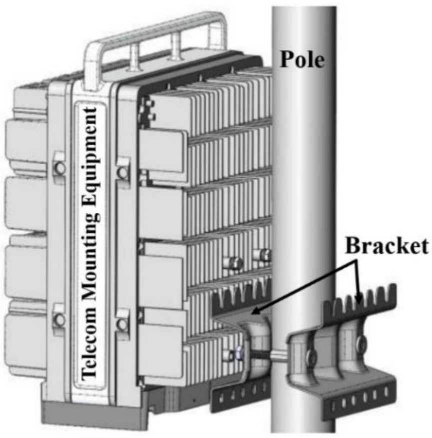


Fig. 1: Telecom Equipment with Mounting Bracket

Structural optimization of telecom mounting brackets is essential to ensure long-term performance under dynamic loading conditions [4]. Vibrations caused by wind-induced oscillations, mechanical operations, or nearby traffic can lead to fatigue and resonance-related failures, while earthquakes impose sudden, high-intensity forces that may cause excessive deformation or collapse if not adequately addressed in the design stage [5]. Conventional bracket designs often employ high safety factors, resulting in excessive material usage, increased weight, and higher costs. Hence, there is a growing demand for lightweight yet high-strength designs that balance mechanical performance, durability, and economic efficiency [6].

Advancements in computational tools, particularly Finite Element Analysis (FEA), have enabled detailed evaluation of structural behaviour under complex loading scenarios [7]. FEA allows engineers to predict stress distribution, deformation patterns, and dynamic responses such as natural frequencies and resonance characteristics without relying solely on physical prototypes [8]. Through vibration analyses including modal, harmonic, and random vibration studies and seismic analyses such as response spectrum methods, the structural integrity and stability of telecom mounting brackets can be thoroughly assessed [9]. These analytical approaches play a crucial role in identifying stress concentrations, minimizing resonance effects, and improving overall reliability, thereby supporting the development of optimized and resilient telecom infrastructure designs [10].

## II. LITERATURE REVIEW

Extensive numerical and experimental investigations on bracket structures demonstrate that vibration fatigue, geometry, and material characteristics critically govern service life and mechanical reliability. Detailed fracture and fatigue analyses revealed that combined extreme loading and welding defects significantly accelerate damage, while optimized configurations achieved a safety margin of at least 10%, with relative damage levels of 0.07, 0.46, and 0.45 under standard, frequency-domain, and uniaxial cumulative loading compared to on-track multiaxial conditions, and test–simulation variance remaining below 20% [1]. Finite element–based comparative studies on bracket geometries under controlled boundary conditions, including inclination variations from 0° to 20°, confirmed the capability of numerical modelling to accurately predict deformation and torque behaviour using detailed 3D meshing and material assignment strategies [2]. Surface and topology optimization approaches further demonstrated effective weight reduction while maintaining acceptable stress levels through evolutionary and shape-based algorithms [3,4].

Subsequent studies broadened the scope to rail, automotive, civil, and telecom applications, identifying high-frequency excitation in the 400–800 Hz range as a dominant cause of fatigue crack initiation near welded regions under operational loading [5]. Topology-optimized designs achieved up to 50% mass reduction, with static stresses as low as 142.19 MPa, safety factors reaching 3.09, and first natural frequencies exceeding operating limits at 69.043–100.49 Hz, ensuring dynamic stability [6]. Large-scale finite element modelling involving 7,560 simulations enabled the development of unified design equations for bracketed joints under varying moment conditions [7]. Telecom mounting brackets analyzed under wind speeds of 67 m/s demonstrated improved structural efficiency through reduced stress concentration and material savings [8], while dimensional optimization of fluid tank holding brackets emphasized weight reduction benefits for vehicle performance [9]. Harmonic response and fatigue assessments conducted over vibration ranges of 5–17.3 Hz with excitation levels up to 1.5 G confirmed effective resonance avoidance [10], and comparative material studies using cast iron, wrought iron, and mild steel highlighted the role of material selection in controlling von Mises stress and natural frequency response [11].

Table 1: Parametric Comparison of Bracket Design, Analysis, and Optimization Studies

| **Ref. No.** | **Bracket Type** | **Industry** | **Analysis Method** | **Optimi-zation Technique** | **Loading Conditions** | **Material** | **Software / Tools Used** | **Key Performance Parameters** | **Key Findings** |
|---|---|---|---|---|---|---|---|---|---|
| 1 | Wire mounting bracket | Railway / High-speed train | Vibration fatigue analysis, fracture analysis | Structural configuration optimization | Multiaxial vibration, extreme loads | - | Bench testing + Simulation | Stress, fatigue damage, service life | Stress reduced, service life extended, safety margin ≥10%, |

| | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|
| | | | | | | | | | simulation–test error <20% |
| 2 | Orthodontic brackets | Biomedical / Dental | Finite Element Analysis | Comparative geometric evaluation | Torque loading at varied inclinations | SS, TMA alloys | ANSYS SpaceClaim | Torque expression, displacement | Slot geometry significantly affects torque efficiency |
| 3 | Engine bracket arm | Automotive | Static structural FEA | Topology optimization (BESO) | Realistic operational loads | Not specified | MATLAB, FEA solver | Weight, von Mises stress | Significant weight reduction with structural integrity |
| 4 | Vehicle mounting bracket | Automotive | Structural analysis | Shape & topology optimization | External mechanical loads | Not specified | CATIA + CAE | Stress, deflection, weight | Optimized design meets strength with reduced mass |
| 5 | Wire bracket (cantilever) | Railway | Fracture mechanics + vibration study | Failure mechanism identification | Wheel–rail excitation, high-frequency vibration | Welded steel | Field testing + Simulation | Crack initiation, vibration frequency | Failure due to weld defects + high-frequency vibration |
| 6 | Electric motor bracket | Industrial machinery | Static & dynamic FEM | Topology optimization | Static load, vibration | Not specified | FEM software | Stress, safety factor, natural frequency | Optimized design-2 superior in static & dynamic response |
| 7 | Eaves bracket joint | Structural / Civil | Large-scale FEM study | Parametric design equations | Opening & closing bending moments | Cold-formed steel | FEM solver | Moment capacity, stiffness | Unified design equations proposed |
| 8 | Pole-mount telecom bracket | Telecommunications | Static + wind load FEA | Design modification | Wind load (67 m/s) | High tensile steel | SOLIDWORKS, ANSYS | Stress, deformation, weight | Improved design reduced stress and weight |
| 9 | DEF tank holding bracket | Automotive | Static FEA | Dimensional optimization | Static operational loads | Not specified | FEA software | Weight, stress | Weight reduction improves fuel efficiency |
| 10 | Horn mounting bracket | Automotive | Harmonic response + fatigue | Design comparison | Vibration (5–17.3 Hz), resonance | Not specified | ANSYS Workbench | Natural frequency, fatigue life | Resonance avoided, fatigue life estimated |

| | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|
| 11 | Engine mounting bracket | Automotive | Static & modal analysis | Material comparison | Static loads, vibration | Cast iron, wrought iron, mild steel | HYPERMESH, ABAQUS | Stress, natural frequency | Mild steel shows balanced performance |

Table 1 presents a structured parametric comparison of existing studies on bracket design and optimization across various engineering domains. It summarizes the analysis methods, loading conditions, optimization techniques, key performance parameters, and major findings, enabling identification of research trends and existing gaps for further investigation. Existing studies on bracket design primarily emphasize static, wind, or weight-optimization aspects, with limited focus on vibration-induced fatigue behaviour in telecom equipment mounting brackets under real-world dynamic conditions. Integrated evaluation frameworks combining wind, vibration, and seismic loads with long-term fatigue life prediction remain largely unexplored. Furthermore, current optimization approaches rarely incorporate vibration fatigue resistance as a design objective, creating a gap in developing durable and reliability-oriented telecom bracket designs.

## III. OBJECTIVES

The primary objective of the present work, to evaluate and compare the structural performance of different designs of a telecom equipment mounting bracket under various loading conditions to identify the most stable, durable, and efficient design for real-world applications, specifically in environments where the mounting bracket would be subjected to:

- Wind Force Loads – To determine how well each design withstands deformation and stress due to wind forces, ensuring the bracket can handle high-speed winds without failure.
- Harmonic Vibration Response – To assess the bracket's behaviour under harmonic (steady-state) vibrations, which occur due to operations.
- Random Vibration Response – To evaluate the bracket's durability under unpredictable, real-world vibrations. (Such as road vibrations, railway-induced vibrations, or general structural vibrations).
- Earthquake Resistance – To measure how well each design performs under seismic conditions, ensuring the bracket remains stable and does not experience excessive deformation or failure during an earthquake.

## IV. METHODOLOGY

Static structural analysis is carried out to evaluate displacement, stress, strain, and reaction forces in the structure under time-independent loads by assuming static equilibrium. The finite element formulation is governed by the equilibrium equation

$$\boldsymbol{K}\boldsymbol{u} = \boldsymbol{F} \quad (1)$$

where

$\boldsymbol{K}$= global stiffness matrix,

$\boldsymbol{u}$= nodal displacement vector,

$\boldsymbol{F}$= external force vector.

The structure is discretized into finite elements, and the global stiffness matrix is assembled from individual element stiffness matrices. Once the displacement field is obtained, strains are derived from displacement–strain relations and stresses are computed using Hooke's law for linear elastic materials. Structural safety is assessed using the von Mises stress criterion, given by

**Von Mises Stress Criterion**

$$\sigma_v = \sqrt{\frac{1}{2}\left[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6\left(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2\right)\right]} \quad (2)$$

which is compared against the material yield strength. Total deformation and stress contours obtained from the solution provide insight into stiffness adequacy and potential failure zones under static loading. Modal analysis is performed to determine the inherent dynamic characteristics of the structure, namely natural frequencies and mode shapes, which are essential for vibration-related assessments. The free-vibration equation of motion for a multi-degree-of-freedom system is expressed as

$\boldsymbol{M}\ddot{\boldsymbol{u}}(t) + \boldsymbol{K}\boldsymbol{u}(t) = 0$ (3)

where

$\boldsymbol{M}$= mass matrix,

$\boldsymbol{K}$= stiffness matrix.

Assuming harmonic motion and modal decomposition leads to the generalized eigenvalue problem

**Eigenvalue Problem**

$(\boldsymbol{K} - \omega_i^2 \boldsymbol{M})\boldsymbol{\phi}_i = 0$ (4)

or equivalently,

$\boldsymbol{K}\boldsymbol{\phi} = \omega^2 \boldsymbol{M}\boldsymbol{\phi}$ (5)

where:

$\omega^2$ = natural frequencies

$\phi$ = corresponding mode shapes.

Solving this eigenvalue problem yields the dynamic properties of the structure, which form the basis for identifying resonance-prone modes and for subsequent harmonic, random vibration, and seismic response analyses.

**Harmonic Analysis:**

The structural behaviour of the system is evaluated using finite element–based dynamic analysis, with particular emphasis on harmonic response to represent steady-state operational vibrations. Initially, the dynamic characteristics of the structure are identified through its natural frequency, which governs resonance behaviour and vibration amplification. The natural circular frequency is defined as

$\omega_n = \sqrt{\frac{K}{m}}$ (6)

$f_n = \frac{\omega_n}{2\pi}$ (7)

where $K$ is the structural stiffness and mmm is the system mass. Damping effects are incorporated using the critical damping coefficient and damping ratio, expressed as

$C_c = 2m\omega_n$ (8)

$\zeta = \frac{c}{C_c}$ (9)

to realistically model energy dissipation and avoid unrealistically large responses near resonance.

Harmonic response analysis is governed by the equation of motion for a system subjected to sinusoidal loading,

$M\ddot{u}(t) + C\dot{u}(t) + Ku(t) = F_0 \cos(\omega t)$ (10)

where $M$, $C$, and $K$ are the mass, damping, and stiffness matrices, respectively. Assuming steady-state conditions, the displacement response is expressed in the frequency domain as

$(-\omega^2 M + i\omega C + K)\hat{u} = F_0$ (11)

from which the frequency response function $H(\omega) = \frac{\hat{u}}{F_0}$ is obtained. The vibration amplitude and phase are evaluated as functions of excitation frequency, with resonance occurring when $\omega$ approaches $\omega_n$. This formulation enables identification of critical frequency ranges, assessment of deformation and stress amplification, and evaluation of the effectiveness of damping in controlling harmonic vibrations under operational loading conditions.

**Random Vibration Analysis:**

Random vibration analysis is employed to evaluate the structural response of the system under real-world, non-deterministic dynamic excitations such as wind turbulence, traffic-induced motion, and operational disturbances. Unlike harmonic analysis, the excitation in random vibration is stochastic in nature and is therefore represented statistically rather than deterministically. The dynamic behaviour of the structure is governed by the linear equation of motion under random loading,

$M\ddot{u}(t) + C\dot{u}(t) + Ku(t) = f(t)$ (12)

where $M$, $C$, and $K$ denote the mass, damping, and stiffness matrices, respectively, and $f(t)$ represents the random excitation force, commonly idealized as a zero-mean stochastic process. This formulation enables modelling of unpredictable vibration inputs that cannot be characterized by a single frequency or amplitude.

To efficiently solve the problem, the governing equation is transformed into the frequency domain, where the system response is described in terms of the frequency response function (FRF). The relationship between the input excitation and structural response is expressed using power spectral density (PSD) functions as

$$S_u(\omega) = |H(\omega)|^2 S_f(\omega) \tag{13}$$

$$H(\omega) = (-\omega^2 M + i\omega C + K)^{-1} \tag{14}$$

where $S_f(\omega)$ is the PSD of the input random force and $S_u(\omega)$ is the PSD of the displacement response. This approach allows estimation of RMS displacement and stress responses over a frequency range, providing a reliable measure of structural durability and fatigue susceptibility under random vibration environments.

**Response Spectrum (Earthquake) Analysis:**

Earthquake resistance of the structure is evaluated using the response spectrum method, which provides an efficient means of estimating the maximum dynamic response under seismic excitation without performing computationally expensive time-history analysis. The structural behaviour under earthquake loading is governed by the linear dynamic equilibrium equation

$$M\ddot{u}(t) + C\dot{u}(t) + Ku(t) = f(t) \tag{15}$$

where $M$, $C$, and $K$ are the mass, damping, and stiffness matrices, respectively, and $f(t)$ represents the seismic ground motion input. For interpretation and response estimation, the system is idealized as a set of equivalent single-degree-of-freedom (SDOF) oscillators, each characterized by its natural frequency $\omega_n = \sqrt{\frac{K}{m}}$ and damping ratio $\zeta$.

The seismic response is obtained directly from the response spectrum, which relates peak structural response to excitation frequency and damping. The maximum displacement response of an SDOF system is expressed as

$$\hat{u} = \frac{\hat{f}}{m\omega^2} \tag{16}$$

and the corresponding spectral acceleration is given by

$$S_a(\omega_n) = \frac{\hat{f}}{m} = \hat{u}\omega_n^2 \tag{17}$$

These spectral quantities allow estimation of peak displacement, velocity, and acceleration demands on the structure under earthquake loading. By extracting spectral responses at the natural frequencies of dominant modes, the maximum deformation and stress levels are determined, enabling assessment of structural stability, seismic safety, and resistance to excessive deformation during earthquake events.

**Wind force analysis:**

Wind force analysis is conducted to evaluate the structural safety of telecom mounting brackets exposed to high wind environments on poles and towers. The analysis follows the TIA-222-G standard, which provides a rational framework for estimating wind-induced forces based on site conditions, structural height, and importance factors. The wind loading is first quantified through the velocity pressure, which represents the kinetic energy of wind acting on the structure, and is subsequently converted into pressure and force acting on the projected area of the bracket–equipment assembly. This approach ensures that the bracket design can safely resist wind-induced stresses and deformations without risking structural failure or equipment detachment. The velocity pressure is computed as

$$q = 0.613\, K_z K_{zt} K_d v^2 I \tag{18}$$

where $k_z$ is the velocity pressure coefficient (height dependent), $k_{zt}$ is the topographic factor, $k_d$ is the wind direction probability factor, $v$ is the wind speed, and $I$ is the importance factor. The height-dependent coefficient is evaluated using

$$k_z = 2.01 \left(\frac{z}{z_g}\right)^{2/\alpha} \tag{19}$$

yielding $k_z$= 1.34 for a mounting height of 40 m. The ultimate wind pressure applied in the finite element model is then obtained as

$$p = nqG_h \tag{20}$$

and the corresponding total wind force acting on the bracket is calculated by

$$F = pAC_d \tag{21}$$

where $A$ is the projected area and $C_d$ is the drag coefficient. These calculated loads are applied in static structural analysis to determine total deformation and von Mises stress, enabling assessment of the bracket's structural adequacy under extreme wind conditions.

For wind force analysis, the bracket is restrained using fixed supports at the pole top and bottom surfaces with all translational and rotational degrees of freedom constrained, while gravity is applied in the negative Y-direction (−9.81 m/s²). Bonded contacts are defined for jointed regions and frictional contacts for bolted connections, with bolt pretension applied as required. Wind pressure calculated as per the TIA-222-G standard is applied on the exposed surfaces, and the structural response is evaluated in terms of total deformation and von Mises stress.

For harmonic analysis, fixed supports are applied to restrict all degrees of freedom, and gravity is again included in the negative Y-direction. Base excitation is applied in the X, Y, and Z directions to simulate operational vibrations, with a frequency sweep range of 0–200 Hz to capture resonant behaviour. A damping ratio of $\zeta = 2\%$ is specified to represent energy dissipation and prevent unrealistic resonance amplitudes.

For random vibration analysis, all degrees of freedom at the mounting locations are constrained using fixed supports, and gravity is applied in the negative Y-direction. The excitation is defined through a Power Spectral Density (PSD) function, allowing evaluation of stochastic vibration effects on deformation, stress, and fatigue-related response.

For earthquake analysis, fixed supports are imposed at all mounting points with complete restriction of translational and rotational degrees of freedom, and gravity is included in the negative Y-direction. Seismic excitation is introduced using a time-history or response spectrum approach, with acceleration components applied in the X, Y, and Z directions based on GR63 core parameters. The resulting total deformation and von Mises stress responses are assessed to evaluate structural stability and seismic resistance.

## V. RESULT AND DISCUSSION

Finite Element Analysis (FEA) was conducted on two telecom mounting bracket designs across three configurations (bookshelf, portrait, and combined) to evaluate and optimize structural performance. Wind force, harmonic and random vibration, and earthquake analyses were performed in all three axes, with results quantified in terms of total deformation, von Mises stress, frequency response characteristics, and power spectral density under dynamic and seismic loading conditions.

**Comparative result analysis:**

Table 2: Comparative wind force analysis for two designs

| Wind force Analysis | Design-1 | | Design-2 | |
|---|---|---|---|---|
| | **Total Deformation [mm]** | **Von-mises Stress [Mpa]** | **Total Deformation [mm]** | **Von-mises Stress [Mpa]** |
| Bookshelf configuration | 1.2 | 444.4 | 1.1 | 414 |
| Portrait Configuration | 1.35 | 443.5 | 0.65 | 405 |
| Combination of Bookshelf & Portrait configuration | 1.86 | 463.1 | 1.65 | 418 |

Table 2 presents a comparative wind force analysis for two designs telecom equipment mounting brackets under different configurations.

Design-2 shows lower deformation and von Mises stress in all cases compared to Design-1, indicating better performance under wind force. Portrait Configuration has the least deformation in Design-2 (0.65 mm), suggesting it might be the most stable setup. Combination configuration has the highest deformation and stress, meaning it may experience the most structural strain.

Table 3: Comparative Harmonic Analysis results for two designs

| Harmonic Analysis | Peak Response | Design-1 | | | Design-2 | | |
|---|---|---|---|---|---|---|---|
| | | **Stress** | **Deformation** | **Acceleration** | **Stress** | **Deformation** | **Acceleration** |
| Bookshelf configuration | X | 20 | 20 | 16 | 15 | 16 | 16 |
| | Y | 16 | 20 | 16 | 16 | 16 | 16 |

| | | | | | | | |
|---|---|---|---|---|---|---|---|
| | Z | 30 | 32 | 31 | 36 | 36 | 36 |
| Portrait Configuration | X | 15 | 15 | 15 | 15 | 16 | 16 |
| | Y | 15 | 15 | 15 | 15 | 16 | 16 |
| | Z | 32 | 32 | 33 | 40 | 39 | 39 |
| Combination of Bookshelf & Portrait configuration | X | 16 | 15 | 15 | 16 | 16 | 16 |
| | Y | 16 | 16 | 16 | 16 | 16 | 16 |
| | Z | 32 | 17 | 32 | 38 | 38 | 38 |

Table 3 presents the Harmonic Analysis results for two designs telecom equipment mounting brackets under different configurations, showing peak response in X, Y, and Z directions for stress, deformation, and acceleration. In the bookshelf configuration, the Z direction governs the response for both designs, with Design-2 showing reduced stress in X and Y but higher stress and acceleration in Z, while overall acceleration remains comparable. For the portrait configuration, the Z direction again dominates, and Design-2 exhibits higher Z-direction stress and deformation (40 MPa) compared to Design-1 (32 MPa). In the combined bookshelf–portrait configuration, the response is more balanced, though the Z direction remains critical, with Design-2 showing slightly higher Z-direction acceleration (38 vs. 32).

From the above comparison it can be summarized that Design-2 generally has lower X and Y stress but higher Z stress and deformation. Z direction is the most critical for deformation and stress. Acceleration differences are minimal except in Z.

Table 4: Comparative Random Vibration Analysis results for two designs

| **Random Vibration Analysis** | **Peak Response PSD** | **Design-1** | | **Design-2** | |
|---|---|---|---|---|---|
| | | **Stress** | **Acceleration** | **Stress** | **Acceleration** |
| Bookshelf configuration | X | 16 | 16 | 16 | 16 |
| | Y | 15 | 15 | 16 | 16 |
| | Z | 31 | 31 | 36 | 36 |
| Portrait Configuration | X | 15 | 15 | 16 | 16 |
| | Y | 15 | 15 | 15 | 15 |
| | Z | 32 | 32 | 39 | 39 |
| Combination of Bookshelf & Portrait configuration | X | 15 | 16 | 15 | 16 |
| | Y | 15 | 16 | 16 | 16 |
| | Z | 32 | 32 | 16 | 16 |

Table 4 presents the Random Vibration Analysis results for two designs telecom equipment mounting brackets under different configurations, showing peak response in X, Y, and Z directions for stress and acceleration.

In the bookshelf configuration, the Z direction governs the response in both designs, with Design-2 exhibiting higher Z-direction stress (36 MPa) than Design-1 (31 MPa) and increased acceleration. For the portrait configuration, the Z direction again dominates, and Design-2 shows higher Z-direction stress (39 MPa vs. 32 MPa) along with increased acceleration. In the combined bookshelf–portrait configuration, Design-2 demonstrates a significant reduction in Z-direction stress (16 MPa vs. 32 MPa), while acceleration levels remain largely similar for both designs.

From the above comparison it can be summarized that Z direction is the most critical for stress and acceleration in all configurations. Design-2 shows higher stress in Z for bookshelf and portrait configurations but lower for combination configuration. Acceleration is mostly similar except for higher values in the Z direction for Design-2.

Table 5: Comparative Earthquake Analysis results for two designs

| **Earthquake Analysis** | **Axis** | **Design-1** | | **Design-2** | |
|---|---|---|---|---|---|
| | | **Total Deformation [mm]** | **Von-mises Stress [Mpa]** | **Total Deformation [mm]** | **Von-mises Stress [Mpa]** |
| Bookshelf configuration | X | 1.38 | 106.1 | 1.5 | 99.791 |
| | Y | 2.07 | 148.2 | 2.1 | 137.21 |
| | Z | 0.636 | 94.92 | 0.49 | 77.538 |
| Portrait Configuration | X | 2.07 | 117.18 | 2.02 | 90.23 |
| | Y | 1.5 | 85.189 | 1.37 | 63.38 |
| | Z | 0.496 | 88.454 | 0.352 | 55.24 |
| Combination of Bookshelf & Portrait configuration | X | 1.7 | 116.07 | 1.8 | 91.6 |
| | Y | 1.6 | 129.06 | 1.7 | 111 |
| | Z | 0.71 | 106.85 | 0.49 | 65.7 |

Table 5 presents Earthquake Analysis results for two designs telecom equipment mounting brackets under different configurations, showing total deformation (mm) and von Mises stress (MPa) along X, Y, and Z axes for different configurations.

In the bookshelf configuration, the Y-axis governs the response with the highest deformation and stress in both designs, where Design-2 shows slightly higher deformation but reduced stress compared to Design-1. For the portrait configuration, the X-axis experiences maximum deformation, while the Z-axis exhibits the lowest deformation and stress, with Design-2 consistently demonstrating lower stress across all directions. In the combined configuration, the Y-axis remains critical, and Design-2 achieves improved performance with lower stress and deformation, including a substantial reduction in Z-axis stress (65.7 MPa versus 106.85 MPa).

From the above comparison it can be summarized that Design-2 performs better overall, with lower von Mises stress in all axes. Y-axis is the most critical for deformation and stress in all configurations. Z-axis experiences the least deformation and stress. If earthquake resistance is a priority, Design-2 is the preferred choice.

## VI. CONCLUSION

This study presented a comprehensive finite element–based evaluation of telecom equipment mounting brackets subjected to wind force, harmonic vibration, random vibration, and earthquake loading conditions to identify a structurally efficient and reliable design for real-world deployment. Two bracket designs were examined across three mounting configurations bookshelf, portrait, and combined to assess deformation, stress distribution, and dynamic response. The numerical results consistently demonstrate the superior performance of Design-2 over Design-1 under most loading scenarios. Under wind loading, Design-2 exhibited reduced deformation and stress, with the portrait configuration achieving the lowest deformation of 0.65 mm and a von Mises stress of 405 MPa, indicating enhanced stiffness and load-bearing capability. Vibration analyses identified the Z-direction as the most critical axis for both harmonic and random excitations; however, Design-2 showed improved vibration performance, particularly in the combined configuration, where Z-direction stress was reduced from 32 MPa to 16 MPa. Earthquake response spectrum analysis further confirmed the advantages of Design-2, which demonstrated consistently lower stresses and deformations, including a significant reduction in Z-axis stress from 106.85 MPa in Design-1 to 65.7 MPa, with the portrait configuration again showing the best seismic performance. Overall, Design-2 provided a more balanced and resilient response to multi-hazard loading, with vibration effects governed by the Z-direction and seismic behaviour dominated by the Y-axis. These findings validate the suitability of Design-2 for practical telecom installations. Future studies should focus on enhancing Z-direction stiffness using reinforcement ribs or damping elements, optimizing load distribution along the Y-axis, and incorporating experimental validation and fatigue life assessment to further improve structural reliability.

**References**

[1] Ziyu Dong et al. (2024) "Study on structure optimization and vibration fatigue damage of wire bracket for rail vehicles" Engineering Failure Analysis 156 (2024) 107732.

[2] Rajat Kanta Satapathy et al. (2024) "Torque moments and stress analysis in two passive self-ligating brackets across different incisor inclinations: A 3-dimensional finite element study" Journal of Oral Biology and Craniofacial Research 14 (2024) 487–493.

[3] Saurabh Srivastava et al. (2023) "Topology optimization of engine bracket arm using BESO" Advances in Modeling and Optimization of Manufacturing Processes, Int. J. Simul. Multidisci. Des. Optim.14, 2 (2023).

[4] Bhesar Yogesh Vishnu et at. (2023) "DESIGN & ANALYSIS OF CABIN MOUNTING BRACKET USING THREE DIFFERENT MATERIALS" International Research Journal of Modernization in Engineering Technology and Science, Peer-Reviewed, Open Access, Fully Refereed International Journal.

[5] Ziyu Dong et.al. (2023) "Research on vibration fatigue and failure mechanism of the wire bracket for high-speed train" Engineering Failure Analysis 151 (2023) 107409.

[6] Ma'ruf Yanuar Effendi et al. (2023) "Performance analysis on the structure of the bracket mounting for hybrid converter kit: Finite-element approach" Curved and Layered Structures 2023; 10: 20220206.

[7] Xin Chen et al. (2023) "Moment capacity of eaves brackets of cold-formed steel portal frames" Thin-Walled Structures 189 (2023) 110947.

[8] U. K. Singh & David (2021) "Experimental and finite element analysis of pole mounting bracket for performance and mass optimization" International Journal of Engineering Development and Research, IJEDR2101040 Volume 9, Issue 1 2021, 289-293.

[9] Aditi Subhedar & Bohra (2021) "Finite element analysis of diesel exhaust fluid tank holding Bracket" International Conference on Research Frontiers in Sciences (ICRFS 2021). 1913 (2021) 012087 IOP Publishing.

[10] R. Ramkumar et al. (2021) "Fatigue failure behaviour of horn mounting bracket using finite element analysis" Materials Today: Proceedings, April 2021.

[11] M.H. Reddy & J Basha (2021) "STATIC AND MODAL ANALYSIS OF AN BRACKET BY CONSIDERING THE BOLT PRETENSION AND SERVICE LOAD EFFECTS BY USING THE FINITE ELEMENT ANALYSIS" International Research Journal of Modernization in Engineering Technology and Science Volume:03/Issue:03/March-2021.