Evaluation of the Linear-Elastic Response of a Three-Dimensional Portal in a Jacket Structure Leg System under Vertical Loads Using SAP2000

Rakhmat Alghifari Arif, Arsa Aldani Saptarina, Sitti Suhaeni Sarhani Rahman, and Muhammad Zubair Muis Alie

Department of Ocean Engineering, Faculty of Engineering, Hasanuddin University
Email: arifyagi19@gmail.com

Abstract

Jacket structures are three-dimensional steel framing systems that transfer top side loads to the foundation through legs, braces, and tubular joints. This study evaluates the linear-elastic response of a three-dimensional portal model representing a jacket leg system under vertical loading using SAP2000. The model consists of four legs connected by diagonal and horizontal braces, rigid joints, fixed base supports, and hollow circular sections. The analysis includes self-weight and four concentrated loads of 1 kN applied at the top. The revised study extends the original work through section-property calculations, load-path assessment, a unit-consistency audit, and a critical interpretation of the output contours. The SAP2000 displacement legends show an E-6 scale factor; assuming the model length unit is metre, the maximum Ux and Uy are 245×10^{-6} m or 0.245 mm, while Uz is 560×10^{-6} m or 0.560 mm. These values correct the initial metre-based interpretation. The reported axial response is compression-dominated, ranging from -1059.44 kN to +7.41 kN. The stress contour identifies the middle-brace region as a critical response zone; however, yielding cannot be concluded because the active stress unit was not documented. Reported torsion ranges from -4.60 to +3.42 kN·m, indicating internal coupling associated with the three-dimensional configuration. Linear static analysis is therefore useful for preliminary stiffness and load-path assessment, but unit verification, reaction equilibrium, member discretization, joint flexibility, soil-pile interaction, environmental loading, and nonlinear analysis are required before the model can support an offshore structural safety assessment.

Keywords: Finite Element Method, Jacket Structure, SAP2000, Three-Dimensional Frame, Unit Audit, Vertical Load.

INTRODUCTION

Jacket structures are a type of fixed offshore structure widely used to support production, drilling, utility, and topside facilities in shallow to intermediate waters. The system consists of main legs, diagonal and horizontal braces, tubular joints, and piles or foundations that work as an integrated unit. ISO 19902 classifies jackets as fixed offshore steel structures, while API RP 2A-WSD provides a framework for the design and construction of fixed platforms. Both references emphasize that structural evaluation covers not only member strength but also global stability, load paths, joints, foundations, and environmental conditions [1], [2].

During operation, jacket structures are subjected to combinations of self-weight, topside loads, live loads, waves, currents, wind, earthquakes, installation effects, and accidental conditions. DNV-RP-C205 provides a basis for defining environmental conditions and environmental loads, while ISO 19902 requires modelling to represent the structural behaviour relevant to the limit state being assessed. Therefore, a pure vertical-load analysis is not intended as a final design, but as a basic step for checking stiffness, load-transfer paths, internal-force signs, and model consistency before applying more complex load combinations [1], [3].

The finite element method allows a three-dimensional frame structure to be idealized as a set of frame elements connected at nodes. Each element can carry axial force, shear force, bending moment, and torsion. SAP2000 provides frame-element formulations, local-to-global coordinate transformations, stiffness-matrix assembly, boundary-condition application, and post-processing of analysis results. Although the software interface simplifies modelling, output quality still depends on consistent geometry, material properties, active units, boundary conditions, and user interpretation [9]–[13].

Previous studies show that jacket behaviour can progress from an initial elastic response to geometric nonlinearity, material plasticity, local joint flexibility, buckling, and force redistribution as loads increase. Abdel Raheem demonstrated the importance of considering nonlinear responses under combined structural and wave loads, while Othman and Mohd used ultimate-capacity analysis to assess platform responses under operating and extreme conditions. Wang et al. showed that efficient finite element models still need realistic boundary conditions and loading when used for jacket optimization [17]–[19].

Lisensi Creative Commons Atribusi 4.0 Internasional.

Tubular joints are particularly important because chord–brace geometry can produce stress concentrations and local joint flexibility that may not be adequately captured by a simple centreline frame model. Zavvar et al.'s review summarizes the roles of stress concentration factor, hot-spot stress, degree of bending, and local joint flexibility in tubular-joint assessment. Mia et al. also showed that the behaviour of XT joints under static loading requires attention to local stress distribution, rather than only nominal member forces [20], [21].

The initial manuscript reported Ux and Uy displacements of 0.245 m and Uz of 0.560 m. However, the SAP2000 contour legends in the output figures show an E-6 factor. If the model length unit is metres, the legend values of 245 and 560 correspond to 245×10^{-6} m and 560×10^{-6} m, respectively, or 0.245 mm and 0.560 mm. This scale correction is important because misreading an exponent by 10^6 can change the interpretation from a very small deformation to an apparently unrealistic one. In addition, the stress-contour unit is not visible in the figure, so the claim that stress exceeds yield stress cannot be confirmed without checking the active units or SAP2000 output tables.

Based on these issues, this study aims to: (1) restate the theoretical basis and methodology for a three-dimensional portal model of a jacket leg system; (2) calculate section properties to support model verification; (3) reinterpret displacement using the scale factor shown in the contours; (4) carefully evaluate the distributions of axial force, stress, and torsion; and (5) compile a model-quality audit and recommendations for further development. The main contribution is the integration of available modelling results with a reproducible consistency-checking procedure for educational activities and preliminary offshore structural studies.

THEORETICAL BACKGROUND

Jacket Structural System and Load Path

In a jacket structure, vertical loads from the topside are transferred through deck legs or support points to the main legs. Braces connect the legs to provide lateral and torsional stiffness while helping distribute forces between planes. Under symmetric conditions, with vertical loads applied exactly along the stiffness axis, the ideal response is dominated by compression in the legs. In a real three-dimensional model, differences in brace geometry, local-axis orientation, node connectivity, and deformation compatibility can generate local tension, bending, and torsion even when the external loads have no lateral component [14]–[16].

Brace configuration affects effective length, load paths, mass, and the locations of response concentrations. Modern jacket optimization therefore seeks not only to reduce mass but also to maintain displacement limits, strength, fatigue performance, and manufacturability. Motlagh et al. incorporated fatigue damage into jacket-platform optimization, while Zheng et al. developed an efficient optimization method for offshore wind-turbine jacket structures. These studies reinforce that brace dimensions and configurations should be treated as structural variables affecting multiple limit states [23], [27].

Three-Dimensional Frame-Element Formulation

For linear static analysis, global structural equilibrium is expressed using the stiffness-matrix equation. The element stiffness matrix in local coordinates is transformed into global coordinates and assembled according to node connectivity. After applying the boundary conditions, nodal displacements are obtained and used to calculate element-end forces and nominal stresses [9]–[13].

$$[K]\{u\} = \{F\} \tag{1}$$

where [K] is the global stiffness matrix, {u} is the vector of nodal displacements and rotations, and {F} is the nodal force vector. This formulation assumes a linear force–displacement relationship, small deformations, linearly elastic material, and boundary conditions that remain unchanged during loading. If plasticity, buckling, large geometric changes, or contact occurs, nonlinear analysis is required. DNV-RP-C208 provides specific guidance for determining structural capacity using nonlinear finite element methods [4].

Hollow-Pipe Section Properties

The leg and brace elements are modelled as circular hollow sections. Given the outside diameter D and thickness t, the inside diameter is d = D − 2t. The cross-sectional area, second moment of area for bending, and polar torsional constant of a hollow circular section are calculated as follows [14].

Lisensi Creative Commons Atribusi 4.0 Internasional.

$$A = \frac{\pi}{4}[D^2 - (D - 2t)^2] \quad (2)$$

$$I = \frac{\pi}{64}[D^4 - (D - 2t)^4] \quad (3)$$

$$J = \frac{\pi}{32}[D^4 - (D - 2t)^4] = 2I \quad (4)$$

The nominal line weight can be calculated as w = ρgA. The values of A, I, J, and w are useful for checking whether the section properties entered into the software are consistent with the geometric dimensions. For offshore tubular members, product and fabrication specifications should match the intended application. ASTM A992/A992M is primarily a specification for rolled structural shapes; because this study uses tubular members, A992 properties should be treated as an idealization of material properties rather than as a specification for offshore pipe. API Specification 2B specifically addresses the fabrication of structural steel pipe for welded offshore structures [7], [8].

Nominal Stress, Torsion, and Utilization Indicators

In a frame element, nominal normal stress at the extreme fibre can be calculated from the combination of axial force and bending moment. For one bending plane, the simplified expression is:

$$\sigma = \frac{N}{A} \pm \frac{MC}{I} \quad (5)$$

where N is the axial force, M is the bending moment, c is the distance from the neutral axis to the extreme fibre, A is the cross-sectional area, and I is the second moment of area. Shear stress due to torsion can be estimated using:

$$\tau = \frac{Tr}{J} \quad (6)$$

where T is the torsional moment and r is the radius at the evaluation point. If the stress units have been confirmed and the analysis remains within the elastic domain, a simple utilization ratio relative to yield stress can be expressed as URσ = |σ|/Fy. However, this ratio does not replace code checks, because jacket design also requires assessment of axial–bending interaction, local and global buckling, hydrostatic effects, tubular joints, condition factors, and other limit states [1], [2], [6].

Verification and Validation of the Numerical Model

Verification asks whether the equations and numerical model have been solved correctly, whereas validation assesses whether the model adequately represents the physical system. For a static frame model, basic verification includes checking active units, geometry, connectivity, local-axis orientation, section properties, boundary conditions, the self-weight factor, reaction-force equilibrium, deformation scale, and force signs. Vertical equilibrium can be checked using:

$$\Sigma Rz \approx Wself + \Sigma Pi \quad (7)$$

where ΣRz is the sum of vertical reactions, Wself is the model self-weight, and ΣPi is the sum of external vertical loads. A large discrepancy may indicate an incomplete load case, an incorrect self-weight factor, incorrect restraints, or output taken from a different load combination. Further validation can use a benchmark model, simple hand calculations, laboratory testing, or comparison with another software package. Jacket reliability studies also show that material, geometric, loading, and model uncertainties should be considered when the analysis is extended to safety or fatigue assessment [22].

Lisensi Creative Commons Atribusi 4.0 Internasional.

RESEARCH METHODOLOGY

Research Approach

This study uses a numerical approach based on linear static analysis of a 3D frame model. The available primary data comprise member geometry, material properties, boundary conditions, loading scenarios, and SAP2000 contour images. The revision retains the original geometry and principal output values while adding section calculations, displacement-scale conversion, a unit audit, and limits on interpretation. The workflow is shown in Figure 1.

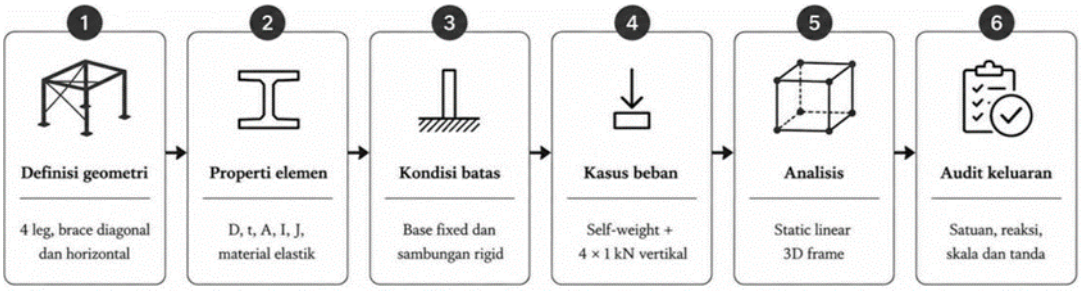


Figure 1. Workflow for modelling, analysis, and output auditing

Model Geometry

The model consists of four legs arranged in a square configuration, with plan dimensions increasing from the top toward the base. The legs are connected by diagonal and horizontal braces at several elevations. The centreline model uses rigid connections, so translations and rotations are compatible at the nodes. This geometry is a global idealization; joint eccentricity, cans, chord ovalization, weld profiles, and joint flexibility are not explicitly modelled.

Table 1. Cross-sectional dimensions of model elements

Member	Outside diameter D (m)	Thickness t (m)	Inside diameter d (m)
Leg	1,80	0,04	1,72
Top brace	0,90	0,06	0,78
Middle brace	1,20	0,06	1,08
Horizontal brace	0,90	0,03	0,84

Table 2. Lengths of diagonal braces

Diagonal-brace group	Length (m)
Brace 1 (topmost)	7,1410
Brace 2	8,6603
Brace 3	10,3410
Brace 4 (bottommost)	12,1244

Table 3. Plan dimensions of horizontal-brace levels

Horizontal-plane elevation	Plan dimensions (m × m)
Level 1	4 × 4
Level 2	6 × 6
Level 3	8 × 8
Level 4	10 × 10
Base	12 × 12

Material and Section Properties

The model uses an elastic modulus E = 200 GPa, Poisson's ratio ν = 0.30, density ρ = 7850 kg/m³, and reference yield stress Fy = 345 MPa. These values are consistent with an idealization of structural steel. However, ASTM A992/A992M-22 covers structural steel shapes and is not a specific specification for offshore pipes. Therefore, the material name in the model should be distinguished from the actual product selection: the

Lisensi Creative Commons Atribusi 4.0 Internasional.

numerical analysis may use these E and Fy values, but fabrication design must specify an appropriate tubular grade, toughness, weldability, corrosion allowance, and inspection requirements [7], [8].

Table 4. Section properties obtained from analytical calculations

Member	A (m^2)	I (m^4)	J (m^4)	Line weight (kN/m)
Leg	0,221168	0,085681	0,171361	17,032
Top brace	0,158336	0,014037	0,028073	12,193
Middle brace	0,214885	0,035005	0,070010	16,548
Horizontal brace	0,081996	0,007767	0,015534	6,314

Table 4 shows that member self-weight cannot be neglected. The nominal line weight is approximately 17.032 kN/m for a leg, 16.548 kN/m for a middle brace, 12.193 kN/m for a top brace, and 6.314 kN/m for a horizontal brace. Given the number of members and brace lengths exceeding 12 m, the total frame self-weight is logically much greater than the four concentrated loads, which total only 4 kN. Consequently, the observed response is likely dominated by self-weight, while the 1 kN load at each upper point acts as a small perturbation. This dominance should be explicitly stated when interpreting the results.

Boundary Conditions, Connectivity, and Loading

The four nodes at the leg bases are assigned fixed supports, i.e., Ux = Uy = Uz = Rx = Ry = Rz = 0. All member intersections are assumed to be rigid joints without end releases. The fixed-base assumption is suitable for an initial study but tends to produce greater global stiffness than a flexible pile–soil system.

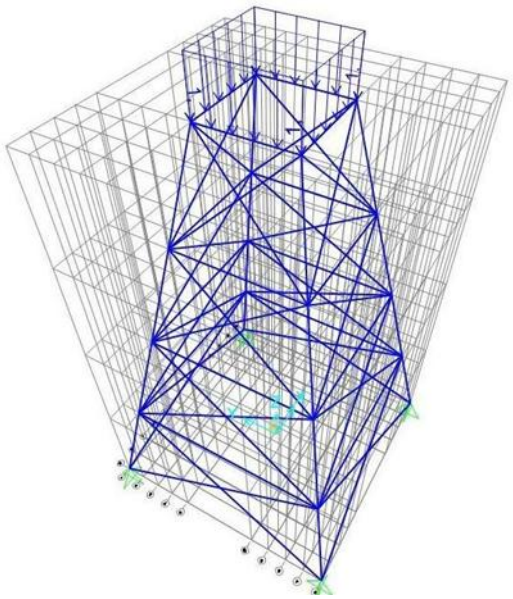


Figure 2. Three-dimensional model and vertical loading in SAP2000

In offshore design models, foundation stiffness and soil–pile interaction can be represented using pile elements, distributed soil springs, p–y, t–z, and q–z curves, or an equivalent foundation model, depending on the required level of accuracy [1], [2], [16]. A single linear-static load case is used. The self-weight factor is enabled, and four concentrated vertical loads of 1 kN each are applied at the top-brace nodes, giving a total concentrated load of 4 kN. The global axes must be defined consistently; in this discussion, the global Z-axis is assumed to be vertical. Figure 2 shows the SAP2000 model and the general load locations.

Output Parameters and Audit Procedure

The main outputs examined are global displacements Ux, Uy, Uz and their resultant, member axial forces, stress contours, and torsion. Verification is performed to ensure that SAP2000 results are read using the correct units, exponent factors, and sign conventions. The locations of maximum responses are also checked against the load path and changes in structural stiffness. The audit further includes checks of reaction equilibrium, local-axis orientation, model discretization, and physical plausibility. Displacement, force, stress, and torsion

Lisensi Creative Commons Atribusi 4.0 Internasional.

values are compared with the geometric scale and simple estimates to identify possible modelling errors or misinterpretation of outputs.

Table 5. Model-output verification checklist

Check	Purpose	Required evidence
Active units	Prevent unit-conversion errors	SAP2000 unit status or output header
Exponent factor	Ensure correct reading of contour values	Notation such as E−3, E−6, etc.
Keseimbangan reaksi	Confirm loads and restraints are applied correctly	Joint reactions and total applied load
Orientasi sumbu lokal	Memastikan interpretasi P, V2, V3, T, M2, M3	Local axes and sign convention
Discretization	Ensure response distribution is captured	Number of elements and output stations
Physical plausibility	Detect unrealistic results	Hand check dan perbandingan skala geometri

RESULTS AND DISCUSSION

Global Displacement Response

The contours in Figure 3 show maximum values of 245 for Ux, 245 for Uy, and 560 for Uz, with an E-6 multiplier in the legend. If the active length unit is metres, the conversions are Ux,max = Uy,max = 245×10^{-6} m = 0.245 mm and Uz,max = 560×10^{-6} m = 0.560 mm. Thus, the initial interpretation of 0.245 m and 0.560 m should be corrected to millimetres. This correction is directly supported by the scale notation in the figure and gives a much more plausible deformation order for a large tubular frame under self-weight and an additional 4 kN load.

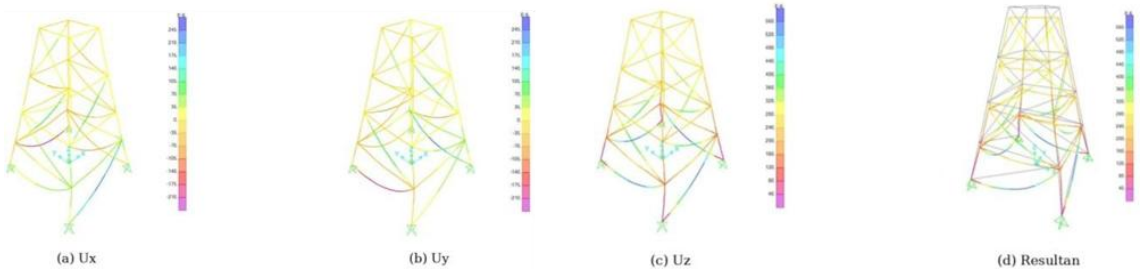


Figure 3. Global displacement contours: (a) Ux, (b) Uy, (c) Uz, and (d) resultant

Table 6. Conversion of displacement values using the contour multiplier.

Komponen	Angka legenda	Multiplier	Nilai dalam meter	Nilai praktis
Ux maksimum	245	E-6	245×10^{-6} m	0,245 mm
Uy maksimum	245	E-6	245×10^{-6} m	0,245 mm
Uz maksimum	560	E-6	560×10^{-6} m	0,560 mm

The vertical displacement is approximately 2.29 times the lateral displacement, while the ratios $\frac{Ux}{uz}$ and $\frac{Uy}{uz}$ are both 0,4375. The similarity of Ux and Uy indicates a relatively balanced global response in the two horizontal directions or the use of generally symmetric geometry. Nevertheless, nonzero lateral displacement under vertical loading can arise from changes in brace planes, coupling between translation and rotation, node eccentricity, or element orientation in the three-dimensional model. These lateral values do not by themselves prove geometric asymmetry; node coordinates and joint loads must be checked to determine the cause.

From a physical-plausibility perspective, interpreting 0.560 m as 560 mm would imply a very large displacement relative to the 4 kN concentrated load. In contrast, 0.560 mm is consistent with a possible elastic deformation order for a system with a leg area of approximately 0.221 m² and an elastic modulus of 200 GPa. A single-member axial-deformation hand check, $\delta = PL/AE$, can be used as an order-of-magnitude comparison, although the actual force distribution in the frame is more complex. Such checks are an effective way to detect unit errors before drawing technical conclusions.

Lisensi Creative Commons Atribusi 4.0 Internasional.

Axial-Force Distribution and Load Path

The reported extreme axial forces are −1059.44 kN in compression and +7.41 kN in tension. Under the sign convention used in the initial manuscript, the response is compression-dominated. The ratio of compressive to tensile magnitudes is approximately 143, indicating that the leg or certain members carry much greater compression than the maximum tension. This pattern is consistent with the role of the legs as the main gravity-load path, while braces stabilize the geometry and distribute forces between legs [15], [16].

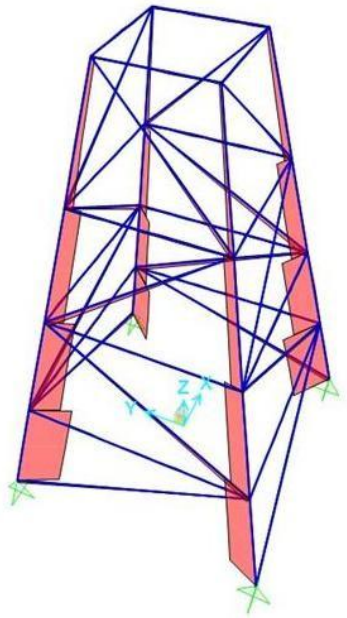


Figure 4. Axial-force distribution in frame elements

However, the magnitude of the compressive force should not be attributed solely to the four 1 kN loads. The line-weight calculation indicates that a single leg weighs approximately 17.032 kN per metre of length. The total self-weight of all legs and braces can readily reach hundreds or more than a thousand kilonewtons, depending on the number and length of members. Therefore, a compressive force of approximately 1059 kN is qualitatively more plausible as the result of accumulated self-weight and frame compatibility than as a response to the 4 kN concentrated load alone. Future work should report the total model weight and base reactions so that the contribution of each load can be separated.

For tubular compression members, evaluation must go beyond the N/A stress. Effective length, slenderness, local buckling, global buckling, axial–bending interaction, and end conditions should be checked against the applicable code. A centreline model with rigid joints may produce different moment distributions and effective lengths from actual connections. More advanced capacity analysis can use pushover or material–geometric nonlinear analysis to identify the sequence of yielding, buckling, force redistribution, and reserve strength ratio. Such approaches have been used in studies of ageing platforms and ultimate jacket strength [18], [24].

Evaluation of Stress Contours and Unit Consistency

The stress contour reports a maximum of +620.75 and a minimum of −1018.05, with high-response locations at braces and joints at the middle level. Mechanically, these locations may be affected by changes in brace dimensions, stiffness changes between levels, axial force, and secondary moments. However, the figure does not show the stress unit. SAP2000 follows the active units; if the model uses kN–m, stress may be displayed in kN/m², whereas with N–mm units, output may be expressed in N/mm² or MPa. Because the active units are undocumented, these values must not be assigned MPa directly.

Lisensi Creative Commons Atribusi 4.0 Internasional.

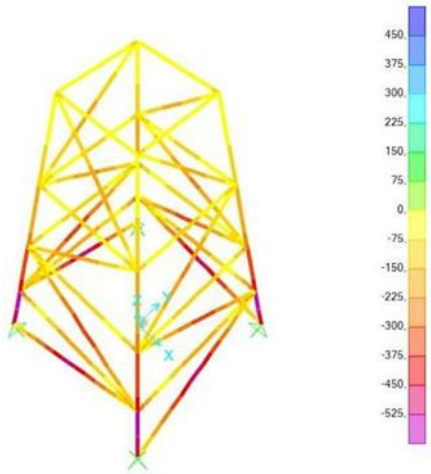


Figure 5. Stress contours from the SAP2000 frame model

Table 7. Effect of possible units on stress interpretation

Unit scenario	Interpretasi +620,75	Interpretation / −1018.05	Implication for Fy = 345 MPa
Values are truly in MPa	620,75 MPa	-1018,05 MPa	UR ≈ 1.80 and 2.95; elastic analysis exceeds the yield domain
Values in kN/m²	0,62075 MPa	-1,01805 MPa	UR ≈ 0.0018 and 0.0030; far below yield

Table 7 shows that unit ambiguity creates a 1000-fold difference in stress and completely changes the safety interpretation. If the unit is MPa, the extreme values exceed Fy = 345 MPa, and linear static analysis is no longer consistent for evaluating post-yield response; nonlinear material analysis, model checks, and verification of whether the plotted stress is axial stress, combined stress, or another output are required. Conversely, if the unit is kN/m², the actual stress is only about 0.62 to 1.02 MPa and is far below yield. Therefore, the defensible conclusion from this figure is only that the middle-brace region is a relatively high-response zone, not that a member has definitively failed.

Furthermore, the frame model provides nominal stresses based on resultant member forces. It does not capture local stress distributions on the chord wall, weld toe, punching shear, or tubular-joint ovalization. Fatigue and hot-spot stress assessment require a local shell model or an appropriate stress-concentration-factor formulation. DNV-RP-C203 identifies hot-spot stress and joint details as important aspects of fatigue assessment, while tubular-joint studies emphasize the effects of local joint flexibility and degree of bending [5], [20], [28].

Torsional Response

The reported torsional response has a maximum of +3.42 kN·m and a minimum of −4.60 kN·m, giving an extreme range of 8.02 kN·m. Although the external loads are vertical, local torsion can develop due to member local-axis orientation, rotational compatibility at rigid joints, brace directions that do not coincide with the system shear centre, and moment coupling in a three-dimensional frame. Torsion does not necessarily indicate global instability, but it can indicate that joints and braces are subjected to combined actions.

Lisensi Creative Commons Atribusi 4.0 Internasional.

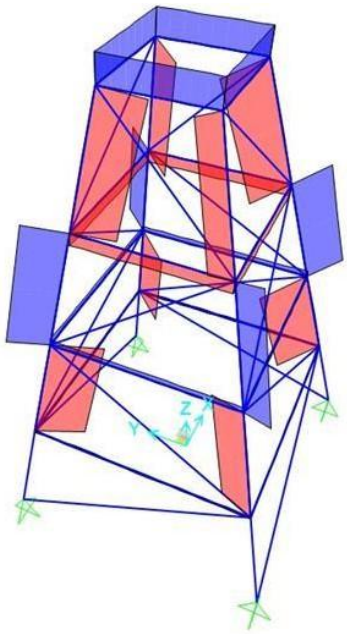


Figure 6. Distribution of torsional moment in structural members

Torsion interpretation should be accompanied by a display of local axes and the T diagram at member stations. Positive and negative signs follow the orientation of local axis 1, so two members with opposite orientations may show different signs for similar physical actions. For a circular pipe, elastic torsional stiffness is proportional to GJ/L; because J is larger for a leg than for a brace, a smaller brace may show relatively greater sensitivity in rotation or shear stress. Nevertheless, torsion magnitude must be compared with bending moments and axial forces to assess its significance for utilization.

Critical Zones and Implications for Joint Modelling

Based on the available contours, relatively high-response zones occur at middle-level brace intersections, changes in configuration between levels, and areas near the base. These locations are logical because forces from the upper planes converge and are redistributed toward the legs and supports. However, "critical zone" in this study means a location prioritized for further investigation, not a proven failure location. Establishing failure requires correct units, a demand-to-capacity ratio, local-joint checks, and clearly defined limit states.

Rigid-joint modelling tends to assume connections are stiffer than actual tubular joints. In real joints, chord-face deformation and local flexibility can change moment distribution and natural frequency. The rigid assumption can be accepted for preliminary global analysis with appropriate caveats. For joint-stress assessment, a local shell model, joint-flexibility factor, or stress-concentration factor should be used. Recent reviews of tubular joints emphasize that integrating global and local models better represents the interaction between platform response and joint hotspots [20].

Comparison with Previous Studies

The results of this study cannot be compared numerically directly with other platforms because geometry, water depth, load cases, and foundation conditions differ. However, the qualitative pattern agrees with the literature: legs primarily transfer gravity loads in compression; braces limit relative deformation and distribute forces; tubular joints are areas of response concentration; and linear models describe only the initial elastic condition. Nonlinear studies show that as loads increase, plasticity and buckling change load paths in ways not captured by linear analysis [17], [18].

For jacket design and optimization of wind turbines, geometric and sectional parameters are evaluated together with realistic loading, soil–pile interaction, and dynamic criteria. Häfele et al. developed an approach to jacket pre-design that integrates the design process, while Wang et al. and Zheng et al. demonstrated the benefits of parametric modelling for optimization efficiency. Compared with these approaches, the present model is simpler and suitable as an educational or preliminary verification model, rather than a complete representation of a platform at a specific site [19], [26], [27].

Lisensi Creative Commons Atribusi 4.0 Internasional.

Research in Indonesia also indicates relevant directions for further development. Setiawan et al. Used pushover analysis for an ageing jacket platform in shallow water and formulated mitigation strategies for life extension, while Sumardi and Rosyid studied design optimization to reduce construction costs. These topics show that a verified global model can be developed further for capacity assessment, life extension, and economic optimization. The first required step remains the same: consistency of units, geometry, loads, boundary conditions, and output interpretation [24], [25].

Model Quality Audit

The audit indicates that the model contains the basic components of a frame analysis, but the output documentation is insufficient for a safety assessment. The most important missing information is the active unit system, total reactions, identities of critical members, and definition of the stress-contour type. These four items can in fact be exported directly from SAP2000 and should be included as standard appendices in numerical studies. Good documentation enables readers to trace and reproduce results and distinguish model findings from interpretive assumptions.

Table 8. Summary of model-quality audit and required improvements

Aspect	Status based on available data	Recommended action
Global geometry	Main leg and brace dimensions are available.	Provide complete node coordinates and elevations.
Section properties	D and t are available; A, I, and J can be verified.	Export section properties from SAP2000 and compare them with Table 4.
Material	E, ν, ρ, and Fy are available.	Use appropriate offshore tubular specifications for the actual design.
Boundary conditions	Fixed base and rigid joints are specified.	Perform sensitivity analyses of foundation stiffness and joint flexibility.
Loading	Self-weight and four 1 kN loads are available.	Report the total weight, load patterns, and load combinations.
Displacement	An E-6 multiplier is visible.	Report U in mm and specify the active units.
Axial force / torsion	Extreme values are available.	Export the tables including element names, stations, and units.
Stress	The contour is available; the unit is not visible.	Do not conclude that yielding occurs until the unit and type of stress have been verified.
Reaction equilibrium	Not available yet.	Provide ΣR and compare it with the total load.
Validation	Not available yet.	Compare the results with hand calculations or a benchmark model.

SENSITIVITY ANALYSIS AND MODEL DEVELOPMENT

Sensitivity to Boundary Conditions

The fixed-support assumption restrains all six degrees of freedom at each base node. In an actual structure, piles and soil provide finite translational and rotational stiffness that depends on depth, soil type, pile diameter, and loading level. Sensitivity analysis can use three models: fixed base, equivalent spring supports, and a distributed pile–soil model. Comparing Ux, Uy, Uz, base shear, overturning moment, and leg axial forces would show how strongly the support assumption affects the global results.

Sensitivity to Joint Stiffness

Rigid joints can be compared with semi-rigid models using rotational springs or a joint-flexibility factor. Variations in joint stiffness, for example 100%, 75%, 50%, and 25% of the ideal rigid condition, can be used to map changes in displacement and end moments. If the response changes significantly, the local joint model should be developed further. For fatigue-prone areas, a shell submodel can be used to obtain hot-spot stress at the weld toe using an appropriate method [5], [20].

Lisensi Creative Commons Atribusi 4.0 Internasional.

Sensitivity to Loading and Self-Weight

To separate the effects of self-weight and point loads, at least three load cases are recommended: SW-ONLY, POINT-ONLY, and SW+POINT. Because linear analysis satisfies superposition, the combined result should equal the sum of the first two load cases within numerical tolerance. This comparison can show the contribution of the 4 kN load to displacement and internal forces. If the contribution is very small, the point-load magnitude should be adjusted to the research objective or explained as a load perturbation.

Nonlinear Analysis and Capacity

If the objective expands from understanding the initial response to assessing capacity, the analysis should include P–Δ/P–δ effects, material plasticity, initial imperfections, member buckling, and joint behaviour. DNV-RP-C208 can be used as a reference for determining resistance through nonlinear FE analysis, while pushover analysis can provide a base-shear–displacement curve, the sequence of plastic-hinge formation, the collapse mechanism, and reserve strength. Studies of jacket ultimate strength show that extreme-condition response cannot be predicted from linear-elastic results alone [4], [18], [24].

Environmental Loads and Fatigue

Further development should add waves, currents, wind, marine growth, buoyancy, hydrodynamic coefficients, and appropriate load combinations. DNV-RP-C205 provides principles for assessing environmental conditions and loads. For tubular structures, fatigue is important because cyclic stresses concentrate at joints. DNV-RP-C203 and jacket reliability studies can support the development of S–N assessment, hot-spot stress, cumulative damage, and probabilistic fatigue assessment [3], [5], [22].

Study Limitations

This study has several limitations that must be stated explicitly. First, the SAP2000 model file and raw output tables were unavailable, so active units, total reactions, critical-member names, output stations, and load combinations could not be directly verified. Second, the complete geometry, including node elevations, total height, and connectivity details, was not fully documented. Third, the model uses a fixed base and rigid joints without soil–pile interaction or local joint flexibility. Fourth, the analysis is linear static and does not include buckling, plasticity, geometric nonlinearity, dynamic response, or fatigue. Fifth, representative environmental and topside loads have not been applied. Sixth, using ASTM A992 as the material designation for tubular members is an idealization of properties and does not constitute an offshore product specification.

These limitations do not remove the value of the model as a basic study. Instead, it can be used to teach the relationships between geometry, stiffness, self-weight, and internal-force distribution, while emphasizing the importance of unit auditing. However, the results must not be used to declare the structure safe, unsafe, or suitable for construction without more complete modelling and code checks.

CONCLUSION

A three-dimensional frame structure consisting of four legs with diagonal and horizontal braces can represent the gravity-load path at the initial analysis stage. The axial response is dominated by compression, with an extreme value of −1059.44 kN in compression and +7.41 kN in tension. In the displacement output, the contour legend uses an E−6 multiplier; therefore, assuming the model length unit is metres, the maximum displacements are Ux = Uy = 0.245 mm and Uz = 0.560 mm, rather than 0.245 m and 0.560 m, respectively. Correctly reading the exponent factor is one of the key findings of the result verification.

Based on the loading characteristics, self-weight is estimated to be the dominant load component. The nominal member line weights range from 6.314 to 17.032 kN/m, so the concentrated load of 4 kN is relatively small compared with the total structural weight. The stress contours indicate that the middle-level brace is one of the zones with relatively high response. However, because the stress unit is not visible in the available output, values of 620.75 and −1018.05 cannot be directly expressed in MPa or compared with yield stress (Fy). The active units and stress-output type must therefore be verified before evaluating material capacity.

Lisensi Creative Commons Atribusi 4.0 Internasional.

Member torsion ranges from −4.60 to +3.42 kN·m. Although the applied external loads are primarily vertical, local torsion may still arise from rotational compatibility, local-axis orientation, and interaction among members in the three-dimensional frame. Before the model is used as a basis for structural safety assessment, further verification is required, including reaction equilibrium, sensitivity to support and joint conditions, unit documentation, identification of critical members and stations, and the addition of environmental loads and soil–pile interaction. Nonlinear and fatigue analyses should also be considered according to the limit states being assessed.

ACKNOWLEDGEMENTS

The authors thank the Department of Ocean Engineering, Faculty of Engineering, Hasanuddin University, for academic support and facilities for structural-analysis learning. This revision retains the geometry and principal outputs of the initial study while adding a technical audit to improve traceability of the results.

DATA AVAILABILITY STATEMENT

The data used in this article comprise the member dimensions, material properties, boundary conditions, loading scenarios, and SAP2000 result images included in the initial manuscript. The SAP2000 model file and raw output tables are not provided; therefore, verification of the active units and reaction equilibrium is recommended before final publication.

REFERENCES

[1] International Organization for Standardization, Petroleum and Natural Gas Industries—Fixed Steel Offshore Structures, ISO 19902:2020, Geneva, Switzerland, 2020.
[2] American Petroleum Institute, Planning, Designing, and Constructing Fixed Offshore Platforms—Working Stress Design, API RP 2A-WSD, 22nd ed., Washington, DC, USA, 2014.
[3] DNV, Environmental Conditions and Environmental Loads, DNV-RP-C205, Høvik, Norway, Apr. 2025, amended Mar. 2026.
[4] DNV, Determination of Structural Capacity by Non-linear Finite Element Analysis Methods, DNV-RP-C208, Høvik, Norway, Sep. 2019, amended Oct. 2022.
[5] DNV, Fatigue Design of Offshore Steel Structures, DNV-RP-C203, Høvik, Norway, Oct. 2024, amended Oct. 2025.
[6] American Institute of Steel Construction, Specification for Structural Steel Buildings, ANSI/AISC 360-22, Chicago, IL, USA, 2022.
[7] ASTM International, Standard Specification for Structural Steel Shapes, ASTM A992/A992M-22, West Conshohocken, PA, USA, 2022, doi: 10.1520/A0992_A0992M-22.
[8] American Petroleum Institute, Specification for the Fabrication of Structural Steel Pipe, API Spec 2B, 6th ed., Washington, DC, USA, 2021.
[9] Computers and Structures, Inc., CSI Analysis Reference Manual, Rev. 15, Berkeley, CA, USA, 2017.
[10] K. J. Bathe, Finite Element Procedures, 2nd ed. Watertown, MA, USA: Klaus-Jürgen Bathe, 2014.
[11] O. C. Zienkiewicz, R. L. Taylor, and J. Z. Zhu, The Finite Element Method: Its Basis and Fundamentals, 7th ed. Oxford, U.K.: Butterworth-Heinemann, 2013.
[12] R. D. Cook, D. S. Malkus, M. E. Plesha, and R. J. Witt, Concepts and Applications of Finite Element Analysis, 4th ed. New York, NY, USA: Wiley, 2002.
[13] W. McGuire, R. H. Gallagher, and R. D. Ziemian, Matrix Structural Analysis, 2nd ed. New York, NY, USA: Wiley, 2000.
[14] T. H. G. Megson, Structural and Stress Analysis, 4th ed. Oxford, U.K.: Butterworth-Heinemann, 2019.
[15] S. K. Chakrabarti, Ed., Handbook of Offshore Engineering. Amsterdam, The Netherlands: Elsevier, 2005.
[16] B. C. Gerwick, Jr., Construction of Marine and Offshore Structures, 3rd ed. Boca Raton, FL, USA: CRC Press, 2007.
[17] S. E. Abdel Raheem, "Nonlinear response of fixed jacket offshore platform under structural and wave loads," Coupled Syst. Mech., vol. 2, no. 1, pp. 111–126, 2013, doi: 10.12989/csm.2013.2.1.111.

Lisensi Creative Commons Atribusi 4.0 Internasional.

[18] N. A. Othman and M. H. Mohd, "Response of the offshore jacket platform at its ultimate strength under operating and extreme loads: A Malaysian waters case study," Ships Offshore Struct., vol. 19, no. 11, pp. 1805–1826, 2024, doi: 10.1080/17445302.2024.2312726.
[19] Z. Wang, S. K. Mantey, and X. Zhang, "A numerical tool for efficient analysis and optimization of offshore wind turbine jacket substructure considering realistic boundary and loading conditions," Mar. Struct., vol. 95, Art. no. 103605, 2024, doi: 10.1016/j.marstruc.2024.103605.
[20] E. Zavvar et al., "Analysis of tubular joints in marine structures: A comprehensive review," Mar. Struct., vol.
[21] 99, Art. no. 103702, 2025, doi: 10.1016/j.marstruc.2024.103702.
[22] M. J. Mia, M. S. Islam, and M. M. Rahman, "Numerical analysis of tubular XT joint of jacket type offshore structures under static loading," Bangladesh Marit. J., vol. 6, no. 1, pp. 299–318, 2022, doi: 10.70279/bmj-v6-i1-1057.
A. Shittu, A. Kolios, and A. Mehmanparast, "A systematic review of structural reliability methods for deformation and fatigue analysis of offshore jacket structures," Metals, vol. 11, no. 1, Art. no. 50, 2021, doi: 10.3390/met11010050.
[23] Motlagh, N. Shabakhty, and A. Kaveh, "Design optimization of jacket offshore platform considering fatigue damage using genetic algorithm," Ocean Eng., vol. 227, Art. no. 108869, 2021, doi: 10.1016/j.oceaneng.2021.108869.
[24] Setiawan, D. M. Rosyid, and R. W. Prastianto, "Pushover analysis of ageing offshore jacket platform in shallow water under extreme storm and mitigation strategy for platform's life extension," Int. J. Mar. Eng. Innov. Res., vol. 10, no. 1, pp. 12–34, 2025.
[25] S. F. Sumardi and D. M. Rosyid, "Jacket structure design optimization to reach minimum construction cost," Int. J. Mar. Eng. Innov. Res., vol. 10, no. 1, pp. 47–54, 2025, doi: 10.12962/j25481479.v10i1.22293.
[26] J. Häfele, M. Damiani, R. King, and P. Gebhardt, "A systematic approach to offshore wind turbine jacket predesign and optimization," Wind Energy Sci., vol. 3, no. 2, pp. 553–572, 2018, doi: 10.5194/wes-3-553-2018.
[27] S. Zheng, Y. Wang, Z. Shi, and X. Zhang, "Efficient optimization design method of jacket structures for offshore wind turbines," Mar. Struct., vol. 89, Art. no. 103372, 2023, doi: 10.1016/j.marstruc.2023.103372.
[28] M. A. Lotfollahi-Yaghin and H. Ahmadi, "Effect of geometrical parameters on SCF distribution along the weld toe of tubular KT-joints under balanced axial loads," Int. J. Fatigue, vol. 32, no. 4, pp. 703–719, 2010, doi: 10.1016/j.ijfatigue.2009.10.008.

Lisensi Creative Commons Atribusi 4.0 Internasional.