



IJRTSM

INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT

“STRUCTURAL ASSESSMENT OF AN OFFSHORE CONTAINER CARGO CARRYING UNIT LIFTING FRAME UNDER INCLINED DROP-IMPACT LOADING USING FINITE ELEMENT ANALYSIS”

Rohit Shakya ¹, Dr. Nitin Tenguria ², Dr. Syed Faisal Ahmad ³

^{1,2,3} Department of Mechanical Engineering, Sagar Institute of Research & Technology Excellence, Bhopal, Madhya Pradesh, India

ABSTRACT

Offshore cargo-carrying units (CCUs) are repeatedly lifted, transported, positioned and landed under demanding operational conditions. Their lifting frames must therefore provide adequate strength and stiffness not only under gravity and lifting loads but also under short-duration accidental impact. The present study evaluates an offshore CCU lifting frame subjected to an inclined drop-impact condition using three-dimensional finite element analysis. The baseline frame has overall dimensions of approximately 155.40 in × 95.20 in × 107.00 in and uses ASTM A500 Grade B structural tubing. The represented payload is 26,000 lb (11,793 kg), while the reported frame tare mass is 5,300 lb (2,404 kg), giving a gross mass of approximately 31,330 lb (14,211 kg). The investigated impact configuration uses an inclined frame, a rigid impact surface, a free-drop height of approximately 2 in (5 cm), and explicit transient solution procedures. Parabolic tetrahedral solid elements are used for the three-dimensional model, with approximately 123,667 elements and 115,418 nodes reported for the baseline mesh.

The reported results show that the maximum von Mises stress is 28,108 psi at the lowest corner that contacts the rigid surface first, with a reported factor of safety of approximately 1.63 against the cited 46,000 psi yield-strength criterion. The maximum displacement is 0.138 in, which is below the reported allowable values of 0.574 in, 0.333 in and 0.428 in for the long, short and corner members, respectively. The maximum reported plastic strain is 2.8%, below the 5% interpretive threshold used in the supplied analysis. The transient stress history rises toward the impact response over the reported 128–3200 μ s interval, while the displacement response is described as smooth and progressive.

The study contributes an integrated engineering workflow linking geometry, material selection, load representation, inclined drop configuration, contact, mesh quality, transient analysis and acceptance-oriented interpretation. The literature review further indicates that impact orientation, contact conditions, mesh sensitivity, numerical validation and structural optimization are important research themes. A staged optimization framework is therefore proposed in which critical members identified from the baseline response are modified and re-analysed. Because the supplied dissertation does not contain a quantified post-optimization model, no unreported mass reduction or performance improvement is claimed. Future validation using strain, displacement or Digital Image Correlation (DIC) measurements is recommended.

Key Words: Offshore container; cargo carrying unit; lifting frame; inclined drop test; impact loading; finite element analysis; ASTM A500 Grade B; DNV 2.7-1; transient response; structural optimization.

I. INTRODUCTION

Offshore engineering systems operate in environments in which equipment must be transported, lifted, positioned, connected, operated and periodically returned to shore under demanding logistical conditions. Unlike stationary structures that primarily experience predictable service loads, offshore transport and handling equipment can be exposed to acceleration, impact, lifting, vibration, local contact and accidental loading. A structural frame used to support offshore equipment must consequently perform as both a load-carrying structure and a handling system. Its design has to maintain adequate strength and stiffness while providing a clear load path from the supported equipment to lifting and support points.

The present research concerns an offshore container or cargo-carrying unit (CCU) lifting frame. The frame supports a distributed payload and is intended to withstand handling associated with movement between shore facilities and offshore installations. The supplied dissertation establishes a finite element representation of the frame and evaluates an inclined drop-impact condition.

The problem is important because an offshore lifting frame can experience a severe structural response during an event lasting only a few milliseconds. A hard landing can transfer kinetic energy rapidly into the frame. The response is governed by mass distribution, member stiffness, contact area, support conditions, geometry and material behaviour. A frame that appears satisfactory under static gravitational loading may therefore require a separate accidental impact assessment.

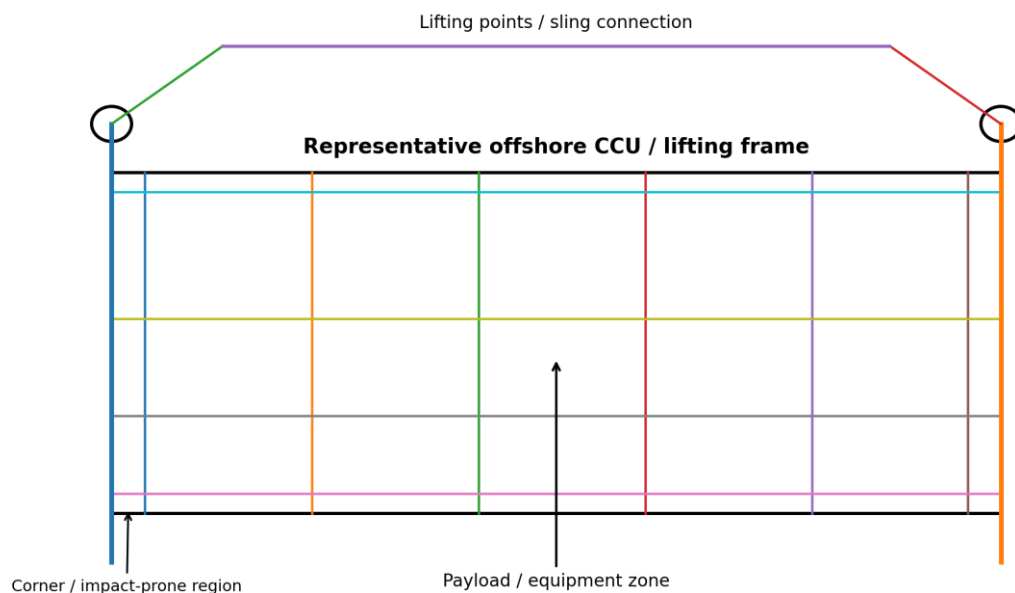


Figure 1. Representative offshore CCU/lifting-frame configuration reproduced from the supplied dissertation.

II. LITERATURE REVIEW

Finite element analysis has become an important engineering tool for short-duration impact problems because it can represent transient inertia, contact, nonlinear material response and local deformation without requiring a physical prototype for every design iteration. The supplied literature review identifies explicit dynamic formulations as particularly suitable for drop events and emphasizes that numerical credibility depends on appropriate modelling, mesh quality, contact definitions, material data and validation.

Protz, Zencker and Liebich (2015) investigated explicit finite element drop analyses of thin-walled steel containers and demonstrated the value of numerical simulation as an alternative or complement to physical testing. Jung et al. (2016) combined experimental drop tests with nonlinear LS-DYNA simulations for grated-steel deck structures, showing that nonlinear FE can reproduce impact response when modelling assumptions are appropriate. Kwon (2017) demonstrated that stress response can be sensitive to structural configuration, while Othman, Sabrah and Marzouk (2017) examined flat and corner drop scenarios and highlighted the relevance of nonlinear material modelling.

Research on dropped objects in offshore structures further shows that impact response may include local yielding, denting, buckling, permanent deformation and, in severe cases, fracture. Woo, Lee and Jeong (2017) combined nonlinear FE with risk/probability analysis for subsea manifold protection. Lu et al. (2018) used Monte Carlo simulation with an ANN-GA approach for FPSO dropped-object analysis, illustrating the computational cost of repeated high-fidelity simulations. Ozguc (2019, 2021) studied dropped-object response of offshore structures using nonlinear LS-DYNA, while Lee and Ryu (2021) demonstrated that impact orientation can strongly influence impact energy and structural consequences.

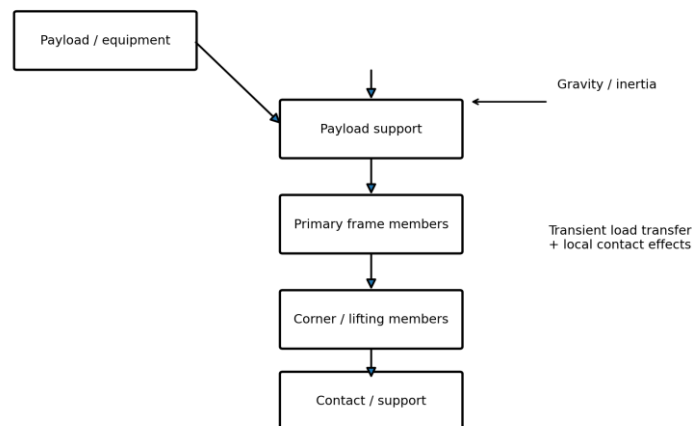


Figure 2. Simplified load-transfer path from payload to contact/support region.

The literature also increasingly connects numerical modelling with physical validation. Jiang et al. (2019) reported experimental and numerical investigation of impact response in submarine pipelines. Oka et al. (2024) used physical drop testing and Digital Image Correlation to compare measured strains with finite element predictions. Such work supports the use of measurable strain and displacement quantities as validation targets rather than relying only on a single stress contour.

A second research direction is structural optimization. Deng et al. (2021), Tian et al. (2022), Zhang et al. (2022) and related studies applied topology, size and shape optimization to offshore structural systems. Tuswan et al. (2024) provided a particularly relevant container-frame example by combining topology optimization, finite element analysis and mesh convergence for a 40-ft LNG ISO tank frame. The 2026 studies listed in the supplied review extend this trend toward sequential impact analysis and automated, code-aware multi-objective optimization.

Overall, the literature supports a staged approach: establish a credible baseline, verify mesh and numerical controls, identify governing members, modify structural dimensions or reinforcement, and re-analyse the improved design. The supplied Chapter 2 specifically identifies the need to connect impact assessment with mesh sensitivity, contact conditions, time-step controls

2.1 Comparative literature synthesis

Year	Representative study	Method	Application	Relevance to present work
2015	Protz et al.	Explicit FE drop simulation	Steel container	Supports explicit drop modelling
2016	Jung et al.	Experiment + LS-DYNA	Offshore deck	Supports nonlinear FE validation
2017	Kwon; Woo et al.; Othman et al.	Nonlinear FE / risk / experiment	Containers & subsea structures	Shows sensitivity to configuration and

				impact mode
2018–2020	Lu et al.; Ozguc; Jiang et al.; Lin & Li	FEM, MCS, experiments, DEM	FPSO, pipeline, container systems	Highlights uncertainty and internal-mass effects
2021–2022	Deng; Ozguc; Lee & Ryu; Tian; Zhang	Topology/size/shape optimization + FE	Offshore structures	Supports systematic redesign
2023	Shrivastava et al.	Dynamic FEA	ASTM A500 lifting frame	Closest comparable context in supplied review
2024	Oka et al.; Tuswan et al.	FE + physical/DIC; topology optimization	Containers/ISO frame	Supports validation and optimization
2025–2026	Residual-strength, material, sequential- impact and automated optimization studies	FE, experiments, optimization	Containers/offshore structures	Represents current research direction

III. RESEARCH GAP, PROBLEM STATEMENT AND OBJECTIVE

3.1 Problem statement

The primary engineering problem is to determine whether the offshore CCU lifting frame can withstand an inclined drop-impact event without unacceptable stress, displacement or permanent structural damage. The event is challenging because it occurs over a short time interval and includes inertia, contact, rapid load transfer and potentially nonlinear material response. The supplied problem-identification chapter frames the work as structural evaluation and improvement of the CCU lifting frame under inclined drop-impact loading.

The research gap is not simply the absence of a finite element model. Rather, the gap lies in the need for an integrated and traceable procedure that connects the physical drop condition, structural load path, material definition, mesh adequacy, contact formulation, time integration, response quantities and acceptance criteria. The supplied chapter identifies eight specific gaps, including limited work on complete CCU behaviour under inclined impact, inconsistent treatment of mesh/contact/time-step sensitivity, insufficient connection between impact analysis and optimization, limited experimental verification, and the need to compare baseline and improved designs quantitatively.

3.2 Objectives

1. Develop and characterize a three-dimensional finite element model of the offshore CCU lifting frame, including structural configuration, member dimensions, load-transfer paths, ASTM A500 Grade B material properties and payload representation.
2. Establish the inclined drop-impact simulation methodology using the DNV-oriented impact condition, boundary conditions, contact interactions, rigid impact surface, finite element mesh, mesh convergence and time-integration parameters.
3. Evaluate transient structural response in terms of von Mises stress, displacement, strain, factor of safety, energy histories and sensitivity to mesh density, contact assumptions and time-step controls.
4. Identify critical members, assess the baseline configuration against the selected acceptance criteria, and establish a rational route for member sizing, reinforcement or geometry optimization.
5. Evaluate the improved configuration in future work and recommend physical validation using strain, displacement or Digital Image Correlation measurements.

The objectives are intentionally organized around the engineering workflow rather than around individual software commands. Geometry and material define the model; drop condition and contact define the dynamic problem; mesh

convergence and numerical controls establish reliability; response assessment identifies governing regions; and optimization uses those findings to guide structural modification. This consolidated methodology is consistent with the supplied.

IV. METHODOLOGY

4.1 Three-dimensional model and dimensions

The lifting frame is represented as a three-dimensional structural assembly. The documented baseline dimensions are approximately 155.40 in in length, 95.20 in in width and 107.00 in in height. The principal members used for displacement assessment include a long beam of 143.40 in, a short beam of 83.20 in and a corner beam of 107.00 in. The load-transfer path is interpreted from the represented payload through longitudinal and transverse members, corner members and support regions toward the lifting and contact points.

The modelling philosophy is to preserve the actual member dimensions and connection arrangement unless a deliberate modification is introduced during optimization. Geometric simplification should be documented because local geometry can influence stress concentration and impact response. The lower corner expected to contact the rigid surface first is of particular importance because it represents the primary impact-load-transfer location.

4.2 Material and payload

ASTM A500 Grade B carbon-steel structural tubing is used for the lifting-frame members. The material-selection chapter describes the material as a practical choice based on structural strength, favorable weight characteristics, cost, availability, forming, machining and weldability. The final FE model should document elastic properties, density, yield strength, tensile strength and the plasticity representation actually used by the solver. If strain-rate dependence is not included, that limitation should be explicitly acknowledged.

The design payload is represented by a thick solid plate located to reproduce the inertial effect of the carried mass. The documented payload is 26,000 lb (11,793 kg), while the frame tare weight is 5,300 lb (2,404 kg), giving an approximate gross weight of 31,330 lb (14,211 kg). The supplied modelling chapter describes the payload plate as 133.40 in $\times$ 77.20 in $\times$ 9 in.

4.3 Inclined drop-impact configuration

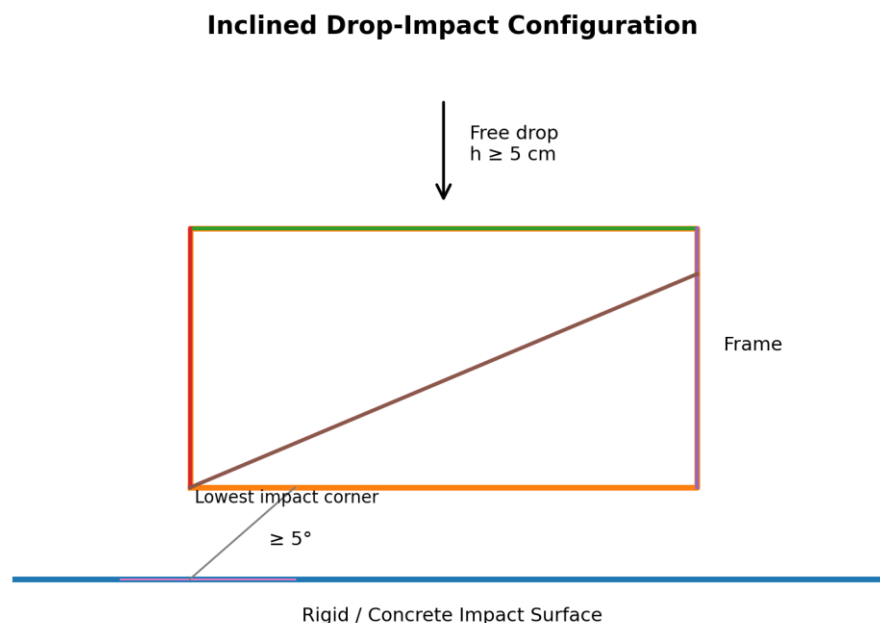


Figure 4. Inclined drop-impact condition: lowest corner contacts the rigid surface first.

The frame is inclined so that the bottom side and end rails connected to the lowest corner form an angle of not less than 5° with the floor. The documented maximum difference between upper and lower corners is approximately 293 mm; with the 50 mm free-drop height, the reported total is 343 mm, below the 400 mm limit stated in the source methodology. The documented SolidWorks setup uses a 2 in (approximately 5 cm) drop height from the lowest point/corner and gravity of 386.22 in/s^2 .

The ideal free-fall impact velocity can be checked using $v = \sqrt{2gh}$. For $h = 0.0508 \text{ m}$ and $g = 9.81 \text{ m/s}^2$, the idealized velocity is approximately 1.00 m/s. This agrees with the minimum impact-speed statement in the supplied methodology and provides a useful consistency check on the numerical setup.

4.4 Finite element discretization and numerical controls

The structural model is discretized using parabolic tetrahedral solid elements. The baseline mesh contains approximately 123,667 elements and 115,418 nodes. The modelling chapter reports high-quality meshing with Jacobian and aspect-ratio checks. Approximately 99% of the geometry is reported to have aspect ratios between 1 and 25, with one localized maximum aspect ratio of 143; the Jacobian four-point check reports that all elements meet the selected quality condition. These metrics are useful indicators, but element quality alone does not establish mesh independence.

For a credible transient result, mesh refinement should be assessed using response quantities such as peak stress and maximum displacement at the critical corner and long beam. A practical convergence criterion is to refine the mesh until the change in the selected response quantity between successive meshes is acceptably small. The same geometry, contact, material and loading conditions should be retained during the comparison.

4.5 Contact, boundary conditions and explicit time integration

The rigid floor is represented as the impact surface and the frame-floor interaction is treated through contact. Welded structural interfaces are represented by bonded contact in the supplied SolidWorks model. The physical floor interaction must permit impact and separation, and friction assumptions should be documented because tangential contact forces influence energy dissipation and load redistribution.

Explicit time integration is appropriate for the short-duration event because the response includes rapidly changing contact forces and inertia. The solution advances using small stable time increments. The simulation duration, time-step control and any mass-scaling or related solver options should be recorded. A time-step sensitivity check is recommended to establish that further reduction in numerical time increment does not materially change the governing response.

V. STRUCTURAL MODELLING AND SIMULATION SETUP

5.1 Member selection

The frame uses several tubular sections selected to provide bending resistance in multiple directions while accommodating the heavy payload and drop-test requirements. The supplied modelling chapter lists rectangular and square tubular sections together with a thick payload plate.

Component	Section / size	Role in model
Rectangular tube 1	$10 \times 6 \times 0.375 \text{ in}$	Primary base/top frame members
Square tube 1	$6 \times 6 \times 0.375 \text{ in}$	Corner members
Rectangular tube 2	$14 \times 6 \times 0.25 \text{ in}$	Forklift-pocket / handling members
Square tube 2	$4 \times 4 \times 0.25 \text{ in}$	Top support members
Plate	$133.40 \times 77.20 \times 9 \text{ in}$	Payload representation

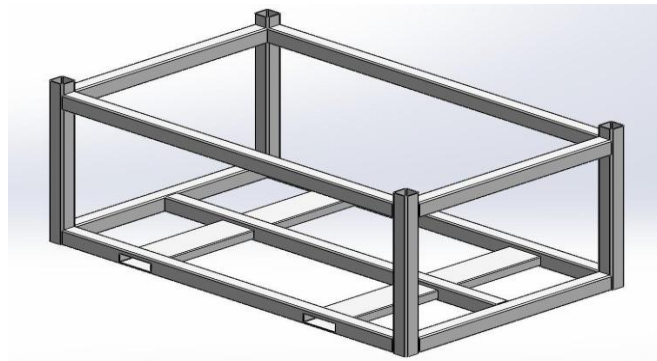


Figure 5. Three-dimensional lifting-frame assembly before payload representation.

The source modelling sequence constructs the frame progressively: the base frame is created first, forklift pockets are added for handling, vertical corner tubes are introduced, the top frame is completed, top support members are added, and finally the thick payload plate is positioned to distribute the design mass across the frame. This staged construction provides a clear geometric record and supports traceability of member functions.

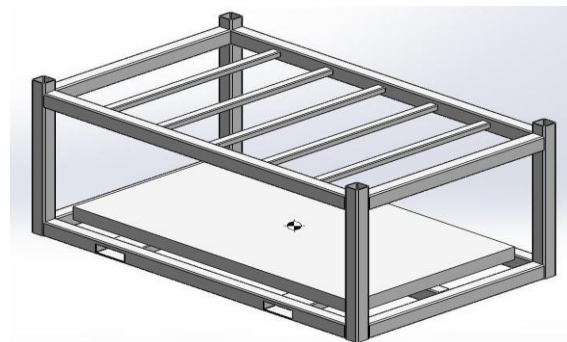


Figure 6. Lifting frame with top support members and represented payload.

5.2 Drop-test setup

The drop orientation is set so that the lowest corner reaches the rigid floor first. The supplied model explicitly examines a 5° inclination about the relevant axes and checks the maximum distance between upper and lower corners. The drop height is defined from the lowest point. Gravity is applied to the three-dimensional model, and the rigid plane represents the concrete or rigid impact surface.

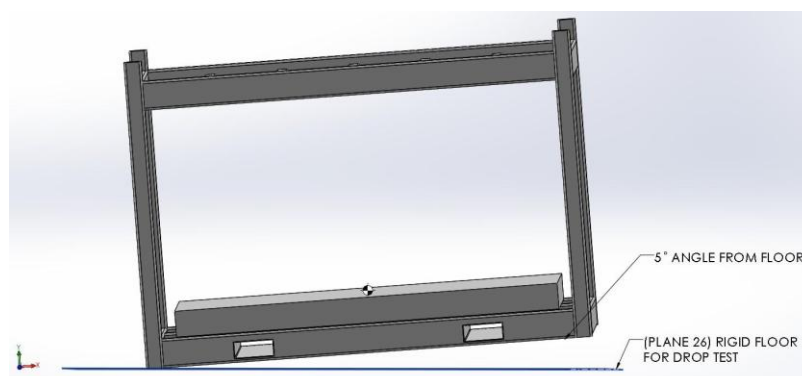


Figure 7. Inclined model orientation used to establish the lowest impact corner.

The body is allowed to move under gravity until contact with the rigid plane. This is preferable to artificially restraining the frame during free fall because the restraint could alter inertia and the load-transfer path. The first-contact corner is therefore the governing region for the initial impact impulse.

5.3 Contact and mesh quality

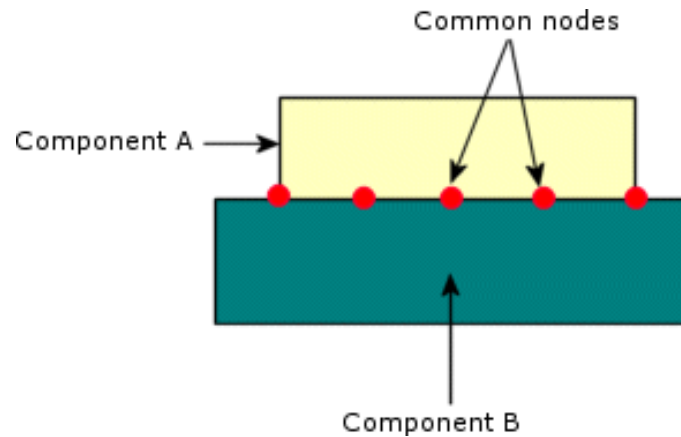


Figure 8. Conceptual bonded contact representation at welded interfaces.

The supplied SolidWorks model uses bonded contact for welded structural interfaces. The contact definition merges compatible coincident nodes and allows the welded members to behave as a continuous structural assembly. For the rigid floor, the contact formulation must represent impact, separation and, where relevant, friction. The use of high-quality parabolic tetrahedral elements improves representation of curved and geometrically complex regions.

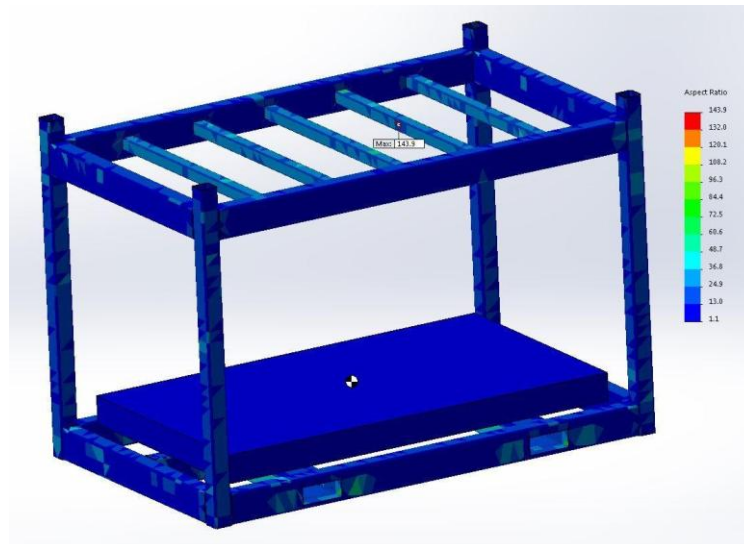


Figure 9. Example FE model / mesh-result view from the supplied simulation file.

VI. MATERIAL SELECTION AND ENGINEERING BASIS

Material selection is a structural design decision because it affects strength, stiffness, mass, fabrication, weldability and impact response. The supplied material-selection chapter selects ASTM A500 Grade B carbon-steel structural tubing for the CCU lifting frame. The choice is consistent with the need for tubular members that can be fabricated into a welded structural assembly.

6.1 Chemical and fabrication characteristics

Parameter	Source-derived value / characteristic
Carbon, C	Maximum 0.26%
Manganese, Mn	Maximum 1.35%
Phosphorus, P	Maximum 0.035%

Sulphur, S	Maximum 0.035%
Copper, Cu	Minimum 0.20% when specified
Fabrication	Suitable for conventional forming, machining and structural welding

The supplied chapter describes ASTM A500 Grade B as cold-formed welded and seamless carbon-steel structural tubing available in round, square, rectangular and special shapes. It identifies high strength-to-weight characteristics, market availability, ease of forming and welding, and uniformity of size and tolerances as important advantages.

6.2 Material response under impact

Under the inclined drop event, the frame experiences rapid load transfer when the lowest corner contacts the rigid surface. The material must therefore accommodate transient stress without unacceptable yielding, fracture or permanent deformation. The baseline result reports a maximum von Mises stress of 28,108 psi and a stress factor of safety of approximately 1.6. The results chapter reports a more specific value of 1.63. These values are treated as reported baseline outputs rather than independently recalculated material properties.

The material model is one of the principal limitations that must be considered when interpreting an impact simulation. If rate-dependent plasticity is not experimentally calibrated, the model should not be presented as a fully rate-validated constitutive description. Instead, the results should be interpreted within the assumptions documented in the supplied FE model, with physical drop testing recommended for validation.

6.3 Design implications

Material strength must be considered together with member geometry and local stress concentration.

High stiffness can reduce global displacement but may increase local contact force; therefore, stiffness and impact resistance should be evaluated together.

Welded interfaces are part of the load-transfer path and should be included in detailed validation.

Corrosion, fatigue, fabrication tolerances and lifting-point integrity remain outside the present transient FE result and require separate engineering assessment.

Any material substitution during optimization must be followed by a complete re-analysis rather than a simple strength comparison.

VII. RESULTS

7.1 Von Mises stress response

The baseline simulation reports a maximum von Mises stress of 28,108 psi. The peak occurs at the lower-most corner of the beam that contacts the rigid floor first. This location is physically consistent with the expected impact load path: the first-contact region experiences rapid deceleration and transfers the resulting load into adjacent members.



Figure 10. Resultant von Mises stress plot of the lifting frame.

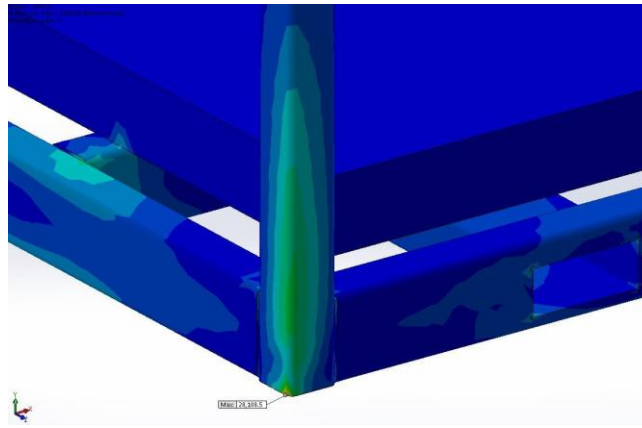


Figure 11. Enlarged view of the maximum-stress region at the first-contact corner.

The supplied results state that no element or geometry exceeds the cited minimum yield strength, and therefore the simulation does not indicate widespread plastic deformation. The reported factor of safety to yield is 1.63. The summary table in Chapter 7 uses 46,000 psi as the allowable/yield reference, giving a factor of safety of approximately 1.6. The supplied result is therefore interpreted as passing the stated baseline stress criterion.

7.2 Displacement response

The maximum displacement is reported as 0.138 in and is located on the inside of the longer base beam. This displacement pattern is consistent with global flexural response following the localized impact at the corner.

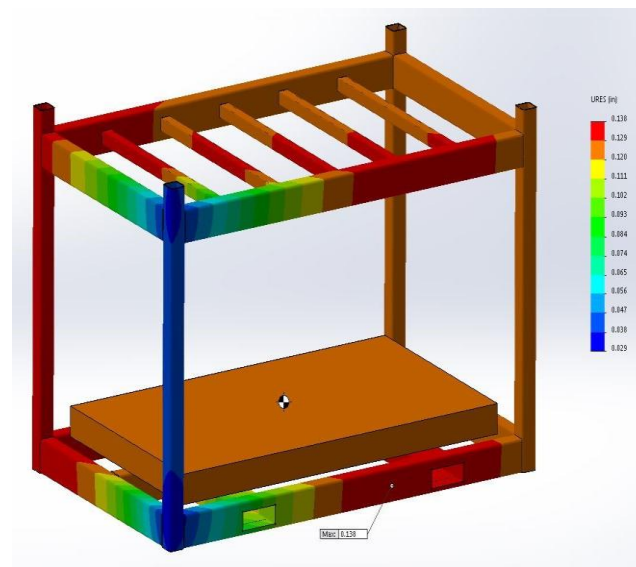


Figure 12. Resultant displacement distribution.

Member	Maximum displacement (in)	Allowable (in)	Reported FOS
Long beam	0.138	0.574	4.2
Short beam	0.138	0.333	2.4
Corner beam	0.138	0.428	3.1

The allowable displacement relation reported in the supplied results is $y = L_n/250$, where L_n is the total length of the beam, rail or post. For the long beam (143.40 in), short beam (83.20 in) and corner beam (107.00 in), the reported allowable values are 0.574, 0.333 and 0.428 in, respectively. Since the maximum computed displacement is 0.138 in, it is below each listed limit.

VIII. STRAIN AND TRANSIENT RESPONSE

8.1 Plastic strain

The supplied results report a maximum plastic strain of 2.8%. The source uses an interpretive threshold of 5% plastic strain spread over a sufficiently large region to represent significant permanent deformation. For the frame dimensions of 155.40 in $\times$ 95.20 in $\times$ 107.00 in, the corresponding 10% dimensions are approximately 15.5 in, 9.5 in and 10.7 in along the three principal dimensions. The reported 2.8% peak is below the stated 5% threshold and is described as occurring over a very small region.

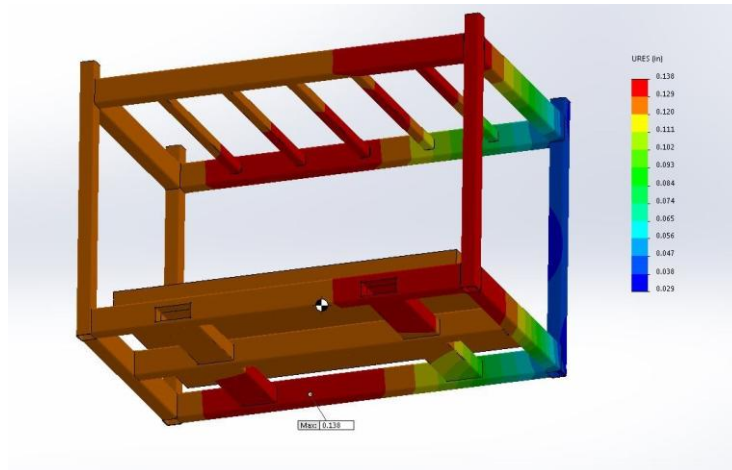


Figure 13. Resultant strain distribution from the baseline impact simulation.

This result supports the stress-based conclusion that the baseline model does not exhibit a large region of permanent deformation. However, the 5% criterion is an interpretive criterion created in the supplied analysis rather than a quantity explicitly specified as a universal DNV plastic-strain limit. It should therefore be presented as a study-specific screening criterion, and final certification should rely on the applicable standard and qualified engineering assessment.

8.2 Stress history

The supplied time-history graph covers approximately 128 to 3200 microseconds. The reported equivalent von Mises stress is initially low and then rises as the frame approaches and undergoes impact. The minimum reported stress is 0 psi at approximately 700 microseconds, interpreted in the source as a stage before significant impact. Thereafter, stress increases toward the maximum response.

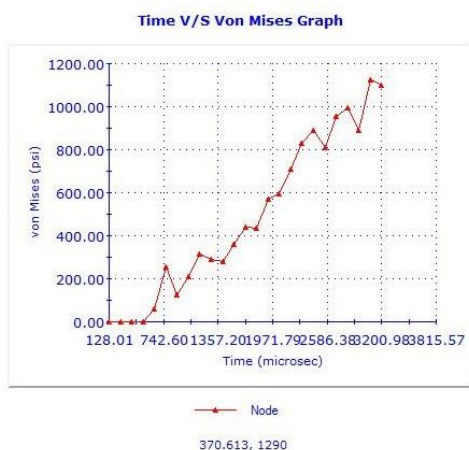


Figure 14. Equivalent von Mises stress versus time from the supplied results.

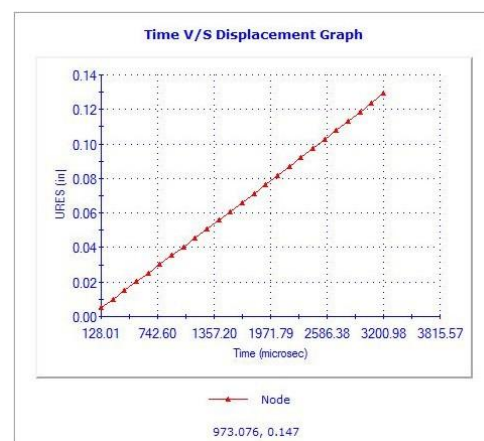


Figure 15. Displacement versus time from the supplied results.

8.3 Displacement history

The supplied displacement history is described as smooth and approximately linear over the reported interval. Taken together with the stress history, the response indicates that the frame undergoes a transient impact followed by structural redistribution. For a more rigorous publication study, force–time, kinetic-energy, internal-energy and contact-energy histories should also be reported so that numerical energy balance can be demonstrated.

IX. EXPERIMENTAL VALIDATION AND FUTURE VERIFICATION

Numerical validation is particularly important for impact analysis because peak response can be sensitive to material data, contact formulation, mesh density and time integration. The supplied literature includes physical drop testing and Digital Image Correlation as a strong validation route. A future experimental campaign should therefore measure quantities that can be extracted from the FE model at corresponding locations and times.

9.1 Recommended measurements

Strain gauges at the first-contact corner and adjacent primary members.

Displacement measurements at the long base beam where the maximum global displacement is predicted.

High-speed video or Digital Image Correlation for full-field deformation.

Impact/contact force measurement where practical.

Post-test inspection for permanent deformation, local buckling, weld damage and cracks.

9.2 Numerical-experimental correlation

Correlation should be performed using comparable physical quantities rather than only visual similarity of contour plots. Peak and time-history strain, displacement at selected points, contact duration and permanent deformation should be compared. If discrepancies are observed, material properties, contact stiffness, friction, damping assumptions and boundary conditions should be reviewed systematically rather than tuned without physical justification.

The final engineering model should distinguish between parameters that are measured, parameters obtained from standards or material certificates, and parameters assumed for the simulation. This distinction improves traceability and prevents overstatement of model certainty.

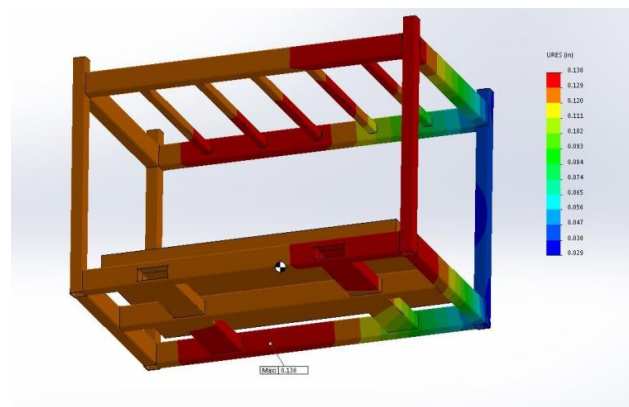


Figure 16. Predicted strain field that could be correlated with future strain/DIC measurements.

X. CONCLUSION

This study consolidates a finite element assessment of an offshore CCU lifting frame under an inclined drop-impact condition. The frame is a welded tubular structural system designed to carry a substantial payload while supporting repeated offshore handling. The inclined drop creates a sequential contact event in which one lower corner reaches the rigid surface first, producing a localized impact load and subsequent redistribution through the frame.

The baseline model uses ASTM A500 Grade B structural tubing, a 26,000 lb payload representation and a reported 5,300 lb tare mass. The frame dimensions are approximately 155.40 in × 95.20 in × 107.00 in. The documented drop

height is approximately 2 in (5 cm), with a minimum inclined condition of 5°. The FE model uses parabolic tetrahedral elements and a reported baseline mesh of approximately 123,667 elements and 115,418 nodes.

The principal result is a maximum von Mises stress of 28,108 psi at the first-contact lower corner, with a reported factor of safety of approximately 1.63 against the cited 46,000 psi yield-strength criterion. The maximum displacement is 0.138 in, below the reported allowable values for the long, short and corner members. The maximum reported plastic strain is 2.8%, below the study-specific 5% screening threshold. These results indicate that the baseline configuration passes the stress and displacement checks reported in the supplied study and does not exhibit a large region of significant permanent deformation.

The research contribution is strengthened by interpreting these results within an integrated methodology. The critical stress location is consistent with the expected first-contact load path, while the maximum displacement occurs on the longer base member, demonstrating the need to evaluate both local and global response. The literature review further supports mesh sensitivity, numerical validation, experimental correlation and optimization as important directions.

The next stage should establish mesh and time-step convergence, document energy histories, validate the model experimentally, and then optimize member dimensions or local reinforcement. Because the supplied dissertation does not contain quantitative optimized results, this paper deliberately does not claim an unverified mass reduction or stress improvement. The resulting framework provides a sound basis for a subsequent validated optimization study.

REFERENCES

- [1] Protz, C., Zencker, U., & Liebich, R. (2015). Explicit finite element analyses of drop tests with thin-walled steel sheet containers for the Konrad repository. [Source-listed literature entry].
- [2] Jung, B.-G., Jo, S.-W., Noh, S.-H., Paik, J. K., Seo, J., & Kim, Y. (2016). Experimental and numerical studies on nonlinear impact response of deck structures subjected to dropped objects. [Source-listed literature entry].
- [3] Kwon, Y.-J. (2017). Comparative study of finite element analysis for stresses occurring in various models of the spent nuclear fuel disposal canister due to accidental drop and impact onto the ground. [Source-listed literature entry].
- [4] Woo, S.-H., Lee, K.-S., & Jeong, J.-M. (2017). Design of deep-sea manifold protective structure considering dropped-object impact. [Source-listed literature entry].
- [5] Othman, H., Sabrah, T., & Marzouk, H. (2017). Feasibility of using ultra-high performance fiber reinforced concrete for radioactive waste containers: Drop test simulation. [Source-listed literature entry].
- [6] Lu, Y., Sun, L., Zhang, X., Kang, J., Zhang, Q., & Yu, B. (2018). Analysis of FPSO dropped objects combining Monte Carlo simulation and neural network-genetic approach. [Source-listed literature entry].
- [7] Ozguc, O. (2019). Structural assessment of the hull response of an FPSO unit to dropped objects: A case study. [Source-listed literature entry].
- [8] Jiang, F., Dong, S., Zhao, Y., Xie, Z., & Guedes Soares, C. (2019). Investigation on the deformation response of submarine pipelines subjected to impact loads by dropped objects. [Source-listed literature entry].
- [9] Lin, M., & Li, Y. (2020). Analysis of the interactions between spent fuel pebble bed and storage canister under impact loading. [Source-listed literature entry].
- [10] Deng, W., Tian, X., Han, X., Liu, G., Xie, Y., & Li, Z. (2021). Topology optimization of jack-up offshore platform leg structure. [Source-listed literature entry].
- [11] Ozguc, O. (2021). The assessment of impact damage caused by dropped objects on floating offshore structures. [Source-listed literature entry].
- [12] Lee, K., & Ryu, H. (2021). Dropped object impact analysis considering frequency and consequence for LNG-FPSO topside module. [Source-listed literature entry].

- [13] Tian, X., Sun, X., Liu, G., Deng, W., Wang, H., Li, Z., & Li, D. (2022). Optimization design of the jacket support structure for offshore wind turbine using topology optimization method. [Source-listed literature entry].
- [14] Tian, X., Wang, Z., Liu, D., Liu, G., Deng, W., Xie, Y., & Wang, H. (2022). Jack-up platform leg optimization by topology optimization algorithm-BESO. [Source-listed literature entry].
- [15] Zhang, C., Long, K., Zhang, J., Lu, F., Bai, X., & Jia, J. (2022). A topology optimization methodology for the offshore wind turbine jacket structure in the concept phase. [Source-listed literature entry].
- [16] Shrivastava, A. K., Sinha, A. K., Choubey, A., Choubey, N., Billore, M., & Qureshi, J. H. (2023). Drop test finite element analysis of different grades of ASTM A500 for structural lifting frame. [Source-listed literature entry].
- [17] Oka, J. M., Vaidya, R. U., Davis, J. T., Webber, K. W., & Walzel, R. K. (2024). Nuclear material container drop testing using finite element analysis with verification using digital image correlation. [Source-listed literature entry].
- [18] Tuswan, T., Andrian, M., Amiruddin, W., Muttaqie, T., Sari, D. P., Bisri, A., Yuniati, Y., Soetarjo, M., Utina, M. R., & Harmadi, R. (2024). Design improvement using topology optimization for the structural frame design of a 40 ft LNG ISO container tank. [Source-listed literature entry].
- [19] Chae et al. (2024). Machine learning-aided damage identification of mock-up spent nuclear fuel assemblies in a sealed dry storage canister. [Source-listed literature entry].
- [20] Lin et al. (2024). Recent numerical/experimental container-impact research relevant to high-strain drop response. [Source-listed literature entry].
- [21] Residual-strength study on container ships subjected to dropped-container impact. (2025). Residual ultimate strength of a container ship under dropped-object impact: Numerical simulations and empirical formulations. [Source-listed literature entry].
- [22] Study on duplex stainless steel in drop scenarios for dry-storage containers. (2025). Dynamic behavior of duplex stainless steel with improved chloride-induced stress corrosion cracking resistance in drop scenarios for dry storage containers. [Source-listed literature entry].
- [23] Roh, H., Ra, C., & Park, N. C. (2026). Finite-element analysis methodology for sequential impact events of dry storage containers. [Source-listed literature entry].
- [24] Drozdowski, J., & Niklas, K. (2026). Structural multi-objective optimization of the offshore jacket structure through rule-based buckling integrated with finite element analysis. [Source-listed literature entry].
- [25] 2026 offshore dropped-object assessment study. (2026). Simplified analytical method for dropped object assessment in offshore and new-fueled ships. [Source-listed literature entry].

