ISSN : 2455-9679
Impact Factor : 4.256

IJRTSM

INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT

"DESIGN AND ANALYSIS OF STRUCTURAL OFFSHORE CONTAINER LIFTING FRAME"

***Bhartendu Vishwakarma* [1], *Prof. Ranjeet Kumar* [2], *Dr. R.S. Sikarwar* [3]**

[1] M. Tech., Scholar, Department of Mechanical Engineering, Vaishnavi Institute of Technology and Science, Bhopal, M.P., India

[2] HOD., Department of Mechanical Engineering, Vaishnavi Institute of Technology and Science , Bhopal, M.P., India

[3] Director, Vaishnavi Institute of Technology and Science Bhopal, M.P, India

---

ABSTRACT

Offshore container is a portable unit, specially designed for repeated use in the transport of goods or equipment to, from or between fixed and/or floating offshore installations and ships. Container is normally designed and manufacture by using industry accepted standard Structural Members and the Lifting frame for analysis is designed in 3D software Solidworks. Lifting frame design consists of selection of beams, material and decides the size according to equipment being carried. The impact performance of new product is a major concern of a new design. The test procedure has to comply with international standards, which establishes minimum mechanical requirements, evaluates yield stress and deformation characteristics of products. The rapid developments in numerical analysis and simulation techniques, faster computing ability, and greater memory capacity, are allowing engineers to create and test new products in virtual environments. Through finite element analysis (FEA), these sophisticated simulations provide valuable information for designing and developing new products, as well as

perfecting existing ones. Manufacturers have found this method eminently useful, as it helps them to achieve better productivity at a lower cost per unit, and develop engineering components that are easy to manufacture, and which make the most economic use of their materials.

Drop test is generally carried out to check the strength of the component against free fall. Offshore equipment Ex. Manifolds, Flowhead, etc. is mounted inside lifting frame, which is a cagestructure created for the transport and safety of equipment. When lifting frame is transported from one place to another in event of loading or unloading, crane release it just above the ground or ship deck. This can results into impact on the lifting frame and subsequently on equipment inside. This

consequently results into high stresses and strain and eventually failure of the structure. Physical drop test involves huge cost of testing and enormous time, which can be replaced by Finite Element simulation of Drop test. In this thesis work, a new design for offshore container lifting frame is proposed and is tested using FEM as per DNV standards.

Key Words: *Offshore Container, DNV 2.7-1, Lifting Frame, Drop Test, Solidworks, CATIA.*

ISSN : 2455-9679
Impact Factor : 4.256

I. INTRODUCTION

1.1 History of Offshore Container certification

The origins of offshore container regulations and standards lay in the Safety of Life at Sea Convention, or SOLAS treaty. SOLAS was created in 1914 as a reaction to the Titanic disaster. Thirteen countries initial associated, but World War I prevented it from going into force. International agreement and adoption of SOLAS became the first major project of the IMO) when it first convened in 1958 as the Inter-Governmental Maritime Consultative Organization. It was then, as it is now, a specialized agency of the United Nations devoted to the safety and security of ships and the prevention of sea pollution. SOLAS went into force in 1965. It was revised in 1974 to simplify the process for amending the treaty. The treaty also included a "tacit acceptance" procedure where amendments will be automatically entered into force unless member nations file objections. SOLAS calls for all ships flagged by its member states to comply with minimum safety standards in construction, equipment and operation of merchant ships.

1.2 Offshore Container

1.2.1 Offshore Container Definition: An offshore container is a portable unit with a maximum gross mass not exceeding 25,000 kg, for repeated use in the transport of goods or equipment, handled in open seas, to, from or between fixed and/or floating installations and ships.

1.3 **Offshore freight containers.**

An offshore container built for the transport of goods, which can include general cargo containers, cargo baskets, bulk containers, special containers, boxes and gas cylinder racks. Offshore portable tanks are also included in this category. These are used to transport dangerous goods used offshore, and must also meet the International Maritime Dangerous Goods code.

Figure 1.1 offshore freight containers

A. Offshore service containers

Custom-built containers for a specific task those are generally temporary. Examples include labs, workshops, power plants and control stations.

ISSN : 2455-9679
Impact Factor : 4.256

Figure 1.2 Offshore Waste Skip

B. Offshore waste skip

A container which holds waste can be open or closed.

C. Skids/Lifting Frame

Offshore containers are also commonly called "skids" in the offshore oil and gas community, as they are often used to transport large components to drilling and production rigs. These can be as simple as frames that hold the contents. The skid, along with its contents, is sometimes referred to as a "skid package."

Figure 1.3 Offshore Skid/Lifting Frame

II. MESHING

Finite Element Analysis (FEA) provides a reliable numerical technique for analyzing engineering designs. The process starts with the creation of a geometric model. Then, the program subdivides the model into small pieces of simple shapes (elements) connected at common points (nodes). Finite element analysis programs look at the model as a network of discrete interconnected elements

2.1 Solid Mesh

In meshing a part or an assembly with solid elements, the software generates one of the following types of elements based on the active mesh options for the study

Linear elements are also called first-order, or lower-order elements. Parabolic elements are also called second-order, or higher-order elements. A linear tetrahedral element is defined by four corner nodes connected by six straight edges. A parabolic tetrahedral element is defined by four corner nodes, six mid-side nodes, and six edges..

ISSN : 2455-9679
Impact Factor : 4.256

III. Problem Definition

In order to achieve better performance and quality, the product design and manufacturing use a number of prototype tests (overload test, fatigue test, and impact test) to insure that the product meets the safety requirements. The test is very time consuming and expensive. Computer simulation of these tests can significantly reduce the time and cost required to perform a new product design. Create a FEA methodology to perform drop test simulation as per parameters considered in standard DNV 2.7-1 .These parameters are listed below [1]

1. The frame shall be lowered or dropped on to a workshop floor of concrete or other rigid structure.
2. The frame shall be inclined so that each of the bottom side and end rails connected to the lowest corner forms an angle of not less than 5° with the floor.
3. However, the greatest height difference between the highest and lowest point of the underside of the frame corners need not be more than 400 mm. As per DNV Common Structural steels for primary structure shall be choose from below. Extra high strength steels, with specified yield stress above 72,000 psi, shall not be used.

- Carbon steel
- Wrought alloy aluminium
- Carbon-manganese steel
- Low alloy steel.

Chemical Elements	ASTM A500 Grade B
Carbon, max. %	0.26
Manganese, max. %	1.35
Phosphorus, max. %	0.035
Sulphur, max. %	0.035
Cooper, when cooper steel is specified min. %	0.20

Table No. 1.1 Chemical Properties of ASTM A500 Grade B

3.2 Modeling and Simulation

3.3 Beam Selection for Container

DNV drop test requirements and bending in any directions, below is beam Sizes that have been selected for Design and to create model with basic structure.

1.Rectangular Tube 10” X 6” X 0.375” Qty. 9 for Base and Top Frame Portion.

2.Square Tube 6” X 6” X 0.375” Qty. 4 for all Corner Beams.

3.Rectangular Tube 14” X 6” X 0.25” Qty. 2 for Forklift Pockets.

4.Square Tube 4” X 4” X 0.25” Qty. 5 for all top support beams.

5.Plate 133.40” X 77.20” X 9” Qty. 1 as Payload

ISSN : 2455-9679
Impact Factor : 4.256

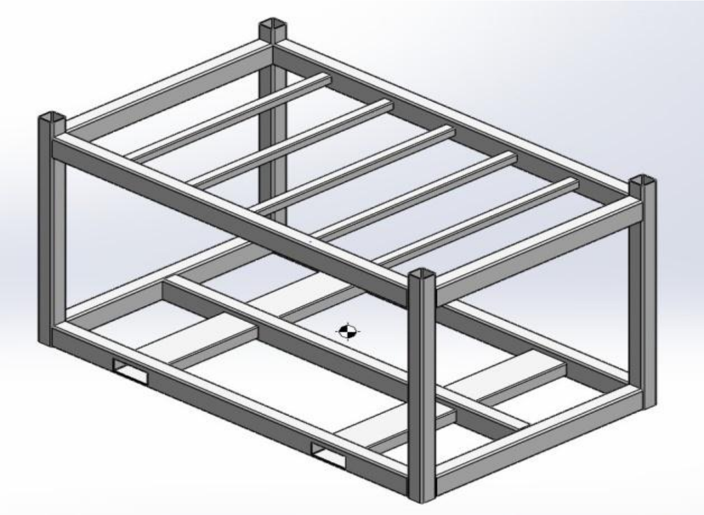


Figure No.1.4 Complete 3D Model without Payload

3.4 Meshing of 3D Model

A key step of the finite element method for numerical computation is mesh generation. The automatic mesh generation problem is to divide a physical domain with a complicated geometry say, an automobile engine, a human's blood vessels, or the air around an airplane into small, simple pieces called elements, such as triangles or rectangles (for two-dimensional geometries) or tetrahedral or rectangular prisms (for three-dimensional geometries). Meshes are also categorized as structured or unstructured. A structured mesh is a regular cubical grid, or the triangular mesh. Unstructured meshes are much more versatile because of their ability to combine good element shapes with odd domain shapes and element sizes that grade from very small to very large. Solid type unstructured Meshing is used to create mesh for lifting Frame/container Using Solidworks simulation.

Below steps are followed for meshing the frame.

1. Import the geometry.
2. Defeaturing the geometry by removing small holes, small fillets and trimming small extended surfaces etc.
3. Nodes are merged at welding location.
4. High quality mesh is selected in order to get accurate simulation result.
5. Jacobian 4point option is selected to check element quality
6. High quality mesh generates parabolic tetrahedral solid elements.

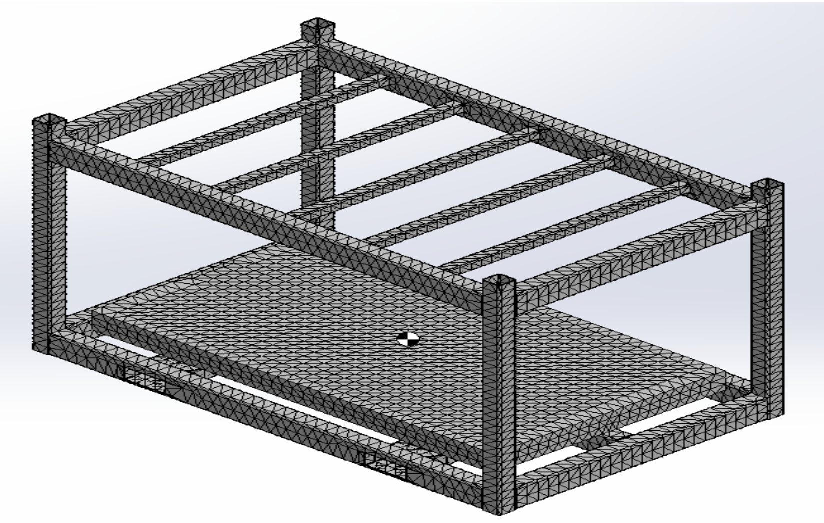


Figure No 1.5 3D Model Meshing

ISSN : 2455-9679
Impact Factor : 4.256

Since this is a bulky model automatic mesher in Solidworks selected solid type mesh. Standard Meshing in Solidworks Activates the Voronoi-Delaunay meshing scheme for subsequent meshing operations. By selecting high quality mesh option, the software Assigns 10 nodes to each solid element: four corner nodes and one node at the middle of each edge (a total of six mid-side nodes). Below Table shows total no of nodes and elements in created in FE model.

Figure No 1.6 Aspect Ratio Plot

Mesh Details

Study name	Drop Test 3 (-Default<As Machined>-)
Mesh type	Solid Mesh
Mesher Used	Standard mesh
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian points	4 points
Element size	3.5 in
Tolerance	0.175 in
Mesh quality	High
Total nodes	115418
Total elements	123667
Maximum Aspect Ratio	143.91
Percentage of elements with Aspect Ratio < 3	23.2
Percentage of elements with Aspect Ratio > 10	54.9
% of distorted elements (Jacobian)	0
Time to complete mesh(hh:mm:ss)	00:00:28
Computer name	PRAFULG

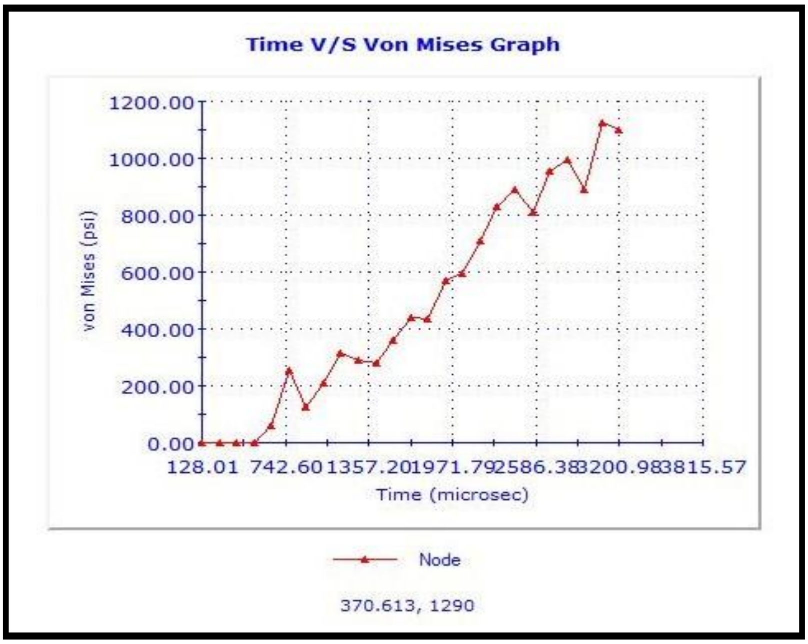
ISSN : 2455-9679
Impact Factor : 4.256

IV. RESULTS AND DISCUSSION

From the simulation results it's confirmed that no Element/geometry goes beyond the Min Yield Strength of Material which shows that there is no plastic deformation in the frame. Maximum Stress is showing on Lower most corner of beam which touches the rigid floor first is 28,108 Psi. Max stress is observed at the expected location, because it hits the rigid floor and experience the max stress. FOS to yield is 1.63 which indicates the lifting frame has passed Drop Test Stress Criteria

4.1 Time V/S Von-Mises Stress Graph

From the result, the graph for equivalent von-Mises stress versus time was constructed. Starting from 128 microseconds until 3200 microseconds, the stress was increase based on the Load impact. The stress increase because of the Impact is increase due to the lifting frame weight and payload. This is the relationship between the load applied and stress occurs. Based on the graph shown in figure 5.8, the result exposed that the minimum equivalent von-Mises stress is 0 Psi when time taken reaches approximately 700 microseconds. The minimum values of this stress means that Impact is not started yet. When time is increase, the stress also increase directly proportional to the time until reaches the maximum stress.



Resultant Von-Mises Stress Plot

V. CONCLUSION

finite element analysis of drop test is conducted in order to determine the Max Stress and Max Displacement. Inclined drop tests as per DNV Requirement and considering the worst case with corner impact, is successfully

ISSN : 2455-9679
Impact Factor : 4.256

simulated in Solidworks Simulation. The simulation results are compared with DNV Allowable Values and the Lowermost Corner which touches Rigid Floor first while drop test found to be experiencing Max stress and longer beam of base frame getting most deflection with FOS more than 1.DNV standard does not specify any value of plastic strain which can be considered as bench mark for comparison, DNV examination is based on visual inspection and NDE performed on the welds after Drop test. No significant plastic deformation is observed in FE simulation which also can be co-related with testing. The simulation results adds confidence to the modeling approach and endorses the mesh size, material, Beam selection and contact definitions specific for problem statement describe in this thesis. This drop test simulation helps to improve the design of Lifting frame while it is in design life cycle, which reduces the cost of iterative physical testing and reduce the product life cycle time to great extent. Simulation is done considering free fall from 5 cm, which consumes approx. 15 hrs. of computational time. From the simulation results it can be concluded that lifting frame is meeting all criteria for drop test allowable limits per DNV 2.7-1 and so it's passing the Drop Test and Design will survive in event of accidental drop.

.

REFERENCES

[1] Yucong Ma , Yihan Xing , Muk Chen Ong , Tor Henning Hemmingsen *et al* (2021) , "Baseline design of a subsea shuttle tanker system for liquid carbon dioxide transportation" Ocean Engineering 240 (2021) 109891.

[2] M. Saeed Khalid , Salman Nisar , Sohaib Zia Khan , Muhammad Ali Khan , Armin W. Troeschet *et al* (2020) , "Simulation of blended nonlinear hydrodynamics forces using radial basis function in uniform moving frame" , Ocean Engineering 198 (2020) 106994.

[3] Amrit Shankar Verma , Zhiyu Jiang , Zhen Gao , Nils Petter Vedvik et al (2020) , "Effects of a passive tuned mass damper on blade root impacts during the offshore mating process" Marine Structures 72 (2020) 102778.

[4] Jae-Hyun Kim, Yeong-Hoon Jeong, Dong-Soo Kim et al (2020) , "Improved load capacity of a hybrid bucket foundation for offshore structures in sand" Applied Ocean Research 101 (2020) 102197.

[5] Kyriazis Pitilakis , Sotiris Argyroudis , Stavroula Fotopoulou , Stella Karafagka , Kalliopi Kakderi , Jacopo Selva *et al* (2019) "Application of stress test concepts for port infrastructures against natural hazards.The case of Thessaloniki port in Greece." Reliability Engineering and System Safety (2018), doi: 10.1016/j.ress.2018.07.005.

[6] Xuefei Wang, Xiangwu Zeng, Xu Yang, Jiale Li *et al* (2019), "Seismic response of offshore wind turbine with hybrid monopile foundation based on centrifuge modelling" Applied Energy 235 (2019) 1335–1350.

[7] P.P. Li, Q.L. Yu (2019), Responses and post-impact properties of ultra-high performance fibre reinforced concrete under pendulum impact , Composite Structures 208 (2019) 806–815.

[8] Abhijit Ekhande, Prof S..B. Naik *et al* (2017) , "Study & FEA Analysis of Transportation Skid", Volume 5, Issue 2 , International Journal of Engineering Development and Research , 2017.

[9] J.C. Suárez, P. Townsend, E. Sanz, I. Diez de Ulzurrum, P. Pinilla et al (2016), "The Effect of Slamming Impact on out-of autoclave Cured Prepregs of Gfrp Composite" Panels For Hulls Procedia Engineering 167 (2016) 252 – 264 Science Direct.

[10] J.C. Suárez, P. Townsend, E. Sanz, I. Diez de Ulzurrum, P. Pinilla *et al* (2016), " The Effect of Slamming Impact on out-of autoclave cured prepregs of GFRP Composite Panels For Hulls", Procedia Engineering 167 (2016) 252 – 264 Science Direct.

[11] R.M.T. Raja Ismail , Nguyen D. That , Q.P. Ha *et al* (2015), "Modelling and robust trajectory following for offshore container crane systems", Automation in Construction , AUTCON-01913; 2015.

[12] Mohammed Imran *et al* (2015), Drop Test Simulation on Pen Drive by using Ansys, International Journal of Research in Engineering and Technology, Volume: 04 Issue: 05 May-2015.

[13] Ajinkya Kulkarni, RatnakarGhorpade et al (2015) "Design and Analysis of Structural Frame Based on Design Codes for Subsea Applications", International Engineering Research Journal (IERJ) Special Issue 2 Page 4599-4610, 2015.

[14] S.E Haridas, *et al* (2014) ,"Load for use of design of ships and offshore structure" Ocean Engineering , 78, (2014) 131–174.

[15] Raouf A. Ibrahim , (2014), "Recent advances in vibro-impact dynamics and collision of ocean vessels", Journal of Sound and Vibration , http://dx.doi.org/10.1016/j.jsv.2014.02.006 0022-460X & 2014.

[16] Pankaj S. Anjikar *et al* (2013) , "FEA Simulation and Validation on Prototype of Compressor Crash Frame for Drop Test as per Industry Standard DNV 2.7-1", International Journal of Engineering Research & Technology Vol. 2 Issue 12, December – 2013 2. 2013-1.

[17] M. Muni Prabaharan *et al* (2013) , "Static Analysis for Container Drop Test with a Case Study", International Journal of Modeling and Optimization, Vol. 2, No. 1, February 2012.

[18] Y.Y. Wang, C. Lu, J. Li, X.M. Tan,Y.C.*et al* (2005) , "Simulation of drop/impact reliability for electronic devices, Finite Elements in Analysis and Design", Vol. 41, Singapore, January 2005, pp. 667–680.

[19] C.Y. Zhou, T.X. Yu, Ricky S.W. Lee at al (2008), "Drop/impact tests and analysis of typical portable electronic device", International Journal of Mechanical Sciences, Vol. 50, Kowloon, Hong Kong, 2008, pp. 905- 917.

[20] T. Noguchi (1999), "Strength evaluation of cast iron grinding balls by repeated drop tests", Wear, Volume 231, Issue 2, July 1999.

[21] K. E. Jackson & E. L. Fasanella *et al* (2005), "Crash simulation of a vertical drop test of a commuter-class aircraft, International Journal of Crashworthiness", Vol.10, NASA Langley Research Centre, Hampton, VA, 2005, pp. 173-182. 2005-7.