ISSN : 2455-9679
Impact Factor : 4.256

IJRTSM

INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT

"DROP TEST ON LIFTING FRAME BY USING FEA METHOD"

***Vinay Shankar Tiwari* [1], *Krishna Waghe* [2], *Sunil Meshram* [3]**

[1] *M.Tech Scholar, Department of Mechanical Engineering, MIT Indore, MP, India*

[2,3] *Assistant Professor, Department of Mechanical Engineering, MIT Indore, MP, India*

ABSTRACT

Drop test is generally carried out to check the strength of the component against free fall. The compressor is mounted inside the crash frame, which is a cage structure created for the safety of component. When compressor with crash frame is shipped or transported from one place to another in event of loading or unloading this package, crane release it just above the ground or ship deck. Which results into impact on the crash frame and subsequently on compressor. This consequently results into high stresses/strains 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 paper a Finite Element methodology is created for drop test simulation considering the parameters of standard Det Norske Veritas (DNV) 2.7-1. A simulation is carried out on weak frame to identify the critical region which are prone to more damage then simulation is carried out on actual frame where critical regions are examined. Experimental set up is created as per the requirement of DNV standard. Finite Element Analysis result and Test results are correlated.

Key Words: *Drop simulation, FEA, DNV-2.7-1, Crash frame, Solidwork.*

I. Introduction

Drop test is free fall of component. It is generally performed to check the ability of product to withstand suddenly applied loads. Drop test is one of the functional test requirement of crash frame for DNV certification, hence crash frame has to qualify through the drop testing as per DNV guide lines before it goes to market for actual use. Crash frame is cage structure, generally manufactured by combinations of standard tube sections, bended C sections and plates. Its purpose to provide safety to the components which are mounted inside it. Since, compressor is mounted inside this frame hence it is called as "Compressor Crash Frame". These frames are generally used for packing industrial components or industrial machine to protect it from damage when its being transported from one place to another or lifted by crane to place it on ship or ground. Crane lift the crash frame containing the equipment inside it to move the frame from one place to other, while placing it onto the ground most of the time crane releases the crash frame just above the ground which results into impact on the crash frame. This impact may results into high stresses/strains on a frame and eventually components inside it. Hence impact carrying capacity of the crash frame should be known, before it is actually being tested so as to design it in more efficient way to avoid the damages while handling. DNV is an autonomous and independent foundation with the objectives of safeguarding life, property and the environment, at sea and onshore. DNV undertakes classification, certification, and other verification and consultancy services relating to quality of ships, offshore units and installations, and onshore industries worldwide, and carries out research in relation to these functions.

Figure 1: Physical drop test

II. LITERATURE REVIEW

Some of the papers are studied, which gives an idea about how to perform FE simulation of drop test. Y.Y. Wang et.al [7] had presented paper on Simulation of drop/impact reliability for electronic devices. In this paper, the finite element method (FEM) is used to simulate drop test numerically, while the attention is paid to the methodology for analyzing the reliability of electronic devices under drop impact. Modelling and simulation method for such kind of complex structure is discussed. Some important issues, such as control of the simulation and material model, are addressed. Numerical examples are presented to illustrate the application of FEM on virtual product development. Effective modelling and simulation method are concluded from the numerical example and authors' experience accumulated from serial industry projects on drop impact simulations. C.Y. Zhou et.al [4], had presented paper on Drop/impact tests and analysis of typical portable electronic devices. This paper presents investigation on the dynamic behavior of typical portable electronic devices under drop impact loading. First, an idealized system which contained an outer case and a Printed Circuit Board (PCB) with an attached packaged chip was adopted as specimen. The actual impact force pulses were measured by employing a Hopkinson bar in a dynamic test rig. Dynamic strains at several locations of the PCB were simultaneously recorded to explore the correlation between the dynamic strains and the impact force pulse. K. E. Jackson & E. L. Fasanella [8], had presented paper on Crash Simulation of a Vertical Drop Test of a Commuter-Class Aircraft. In this paper a finite element model of an ATR42-300 commuter-class aircraft was developed and a crash simulation was executed. Analytical predictions were correlated with data obtained from a 30-ft/s (9.14-m/s) vertical drop test of the aircraft. The purpose of the test was to evaluate the structural response of the aircraft when subjected to a severe, but survivable, impact. The aircraft was configured with seats, dummies, luggage, and other ballast. The wings were filled with 8,700 lb. (3,946 kg) of water to represent the fuel. The finite element model, which consisted of 57,643 nodes and 62,979 elements, was developed from direct measurements of the airframe geometry. The seats, dummies, luggage, simulated engines and fuel, and other ballast were represented using concentrated masses. The model was executed in LS-DYNA, a commercial finite element code for performing explicit transient dynamic simulations. Analytical predictions of structural deformation and selected time-history responses were correlated with experimental data from the drop test to validate the simulation. T. Noguchi et.al [12], had presented paper on Strength evaluation of cast iron grinding balls by repeated drop tests. In this paper, Repeated drop tests were performed on Ni-hard and high-Cr cast iron grinding balls with material toughness varied by heat treatment. Instrumented impact tests and bending fatigue tests were also performed on bar specimens with the same heat treatment, and correlation between drop strength and other strength characteristics were discussed. In the drop tests from various heights, balls fractured by breakage or spalling, with longer life (Nf) at lower drop heights (H) giving H–Nf curves similar to the S–N curves in fatigue tests. Experiments show that drop strength correlated better with fatigue strength and hardness than with impact toughness (KId) in both irons. The stress causing spalling by

ISSN : 2455-9679
Impact Factor : 4.256

repeated drops was inferred to be repeated contact stress, and internal tensile stress caused by surface plastic deformation assists the fracture. Breakage from the ball center is caused by cyclic tensile radial stress by impact body force, and is assisted by residual casting stress.

III. PROBLEM STATEMENT

Create a FEA methodology to perform drop test simulation as per parameters considered in standard DNV 2.7-1 .These parameters are listed below [5] 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. 4. When released, the frame shall drop freely for at least 50 mm to give it a speed at initial impact of at least 1 m/s. 5. Acceptance criteria- No significant permanent damage shall occur. Cracks in welds and minor deformations may be repaired.

IV. MATERIALS AND METHOD SELECTION

Due to significant structural welding, ASTM A500 Grade B was chosen. Carbon steel, cold formed welded, and seamless structural tube in both round and shaped shapes are covered by the ASTM A500 standard specification. ASTM A500 is a standard specification for cold-formed welded and seamless carbon steel structural tubing in round, square, and rectangular shapes, published by the American Society for Testing and Materials (ASTM). In the United States, it is frequently specified for hollow structural components. ASTM A501, which is a hot-formed counterpart of A500, is another similar standard.

Table 1 Mechanical properties of Different grades of ASTM A500 [4].

Properties	**Grade A**	**Grade B**	**Grade C**
Minimum Yield Strength, Psi	39,000 (269MPa)	46,000 (317MPa)	50,000 (345MPa)
Minimum Tensile Strength, Psi	45,000 (310MPa)	58,000 (400MPa)	62,000 (427MPa)
Elongation at Break, %	25	23	21

Method: Produce a FEA methodology to perform drop test simulation as per parameters considered in standard DNV2.7-1. These parameters are listed below.

The frame must be lowered or dumped on to a concrete or other solid structural plant floor. The frame must be angled so that the bottom of each of the lowermost side and end rails attached to the smallest corner makes a 5°angle with the bottom.

1. The highest height difference between the highest and lowest point of the underside of the frame corners must not exceed 400 mm.

2.The frame must drop freely for at least 5 cm after being released, giving it a speed of at least 1 m/s upon impact.

3.Acceptance criteria: No severe and unending damage will be tolerated. Weld cracks and mild deformities can be rectified.

Boundary condition detail

1. Payload + Lifting Frame Weight = Max Gross Weight (MGW).
2. To replicate internal load for the lifting frame, a payload of thick plate is placed to the lifting frame.
3. A plate with a length of 133.40 inches, a width of 77.20 inches, and a height of 9 inches was designed to reflect a uniform payload across the whole lifting frame.

ISSN : 2455-9679
Impact Factor : 4.256

VI. Result and Discussion

Define Contacts for 3D model

Assign the Bonded for welding contact set to all structural Beams in the lifting frame. In solidworks 2016 simulation, bonded entities act as if they were welded, and the computer merges coincident nodes at the interface for a compliant mesh with created nodes and elements.

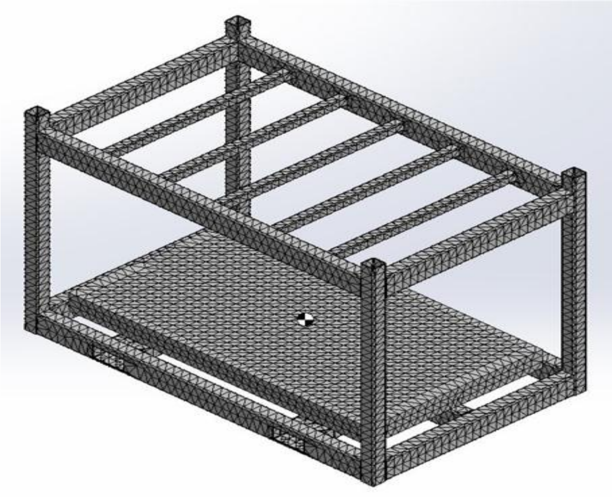

Figure2**:** Mesh Model of crash frame

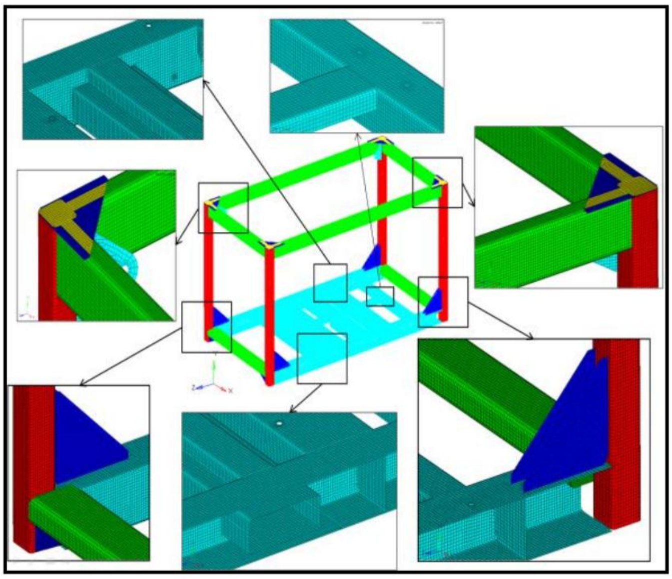

Figure 3. Meshed Model of crash frame

Due to huge size of model, solidworks' automated mesher chooses a solid type mesh. In solidworks, standard meshing turns on the Voronoi-Delaunay meshing technique for further meshing processes. The software assigns 10 nodes to each solid element when the high quality mesh option is selected: four corner nodes and one node in the middle of each edge (a total of six mid-side nodes). The Jacobian 4 point approach is chosen for analysis because of its high meshing quality. The automatic mesher generates parabolic tetrahedral solid components parabolic elements in high grade mesh [6].

ISSN : 2455-9679
Impact Factor : 4.256

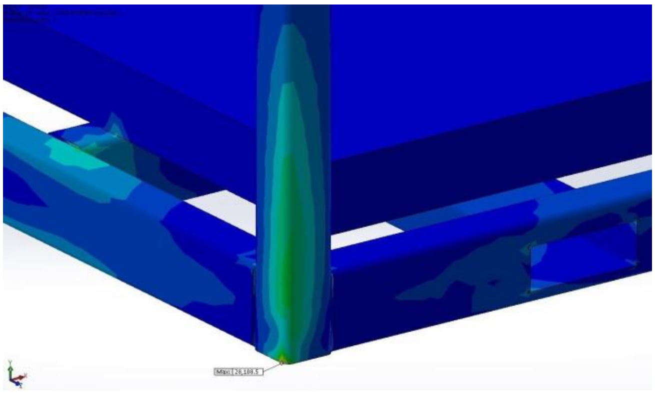

Figure.4 FEA Model of crash frame zoom in stress results

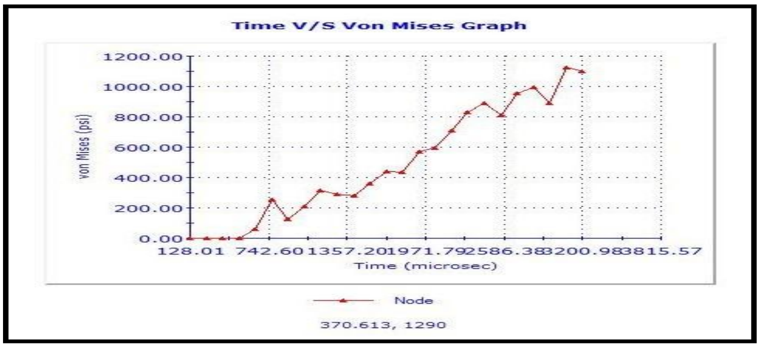

Figure.5 FEA Model of crash frame time ver stress graph

Resultant Displacement Plot

Maximum deflection or displacement is noticed on the longer beam of the base frame from the maximum tension inside the frame. If the factor of safety (FOS) is greater than 1.5, the lifting frame has passed the Drop test stress criteria. DNV has established an acceptable deflection limit for each individual beam, as shown in the equation below.

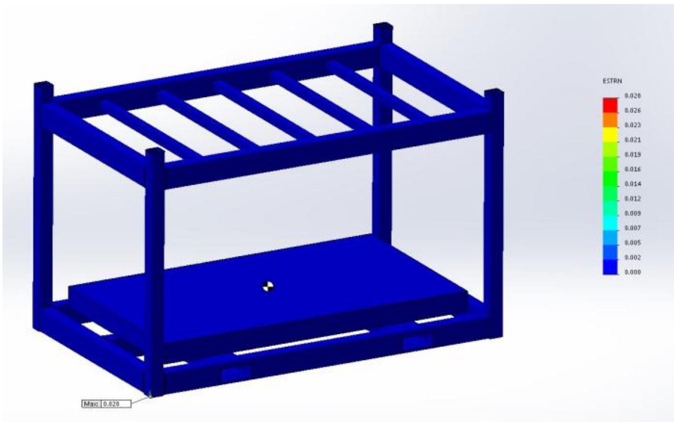

Figure.6 FEA Model of crash frame displacement results

y = Ln/250

where, y = Allowable deflection.

Ln = Total length of the Beam, rail or post.

Corner and base structural beams, which experience the largest bending in drop tests, are key parts of design. The maximum deflection in the longest and most stressed beam is less than the permissible limit. The displacement criterion are applied to the lifting frame [7].

ISSN : 2455-9679
Impact Factor : 4.256

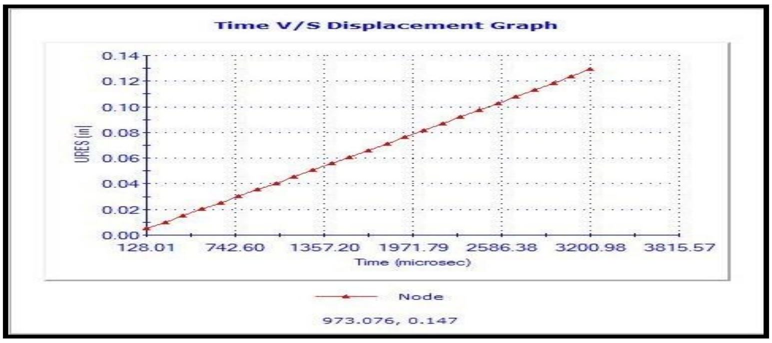

Figure.7 FEA Model of crash frame time ver displacement results graph

Resultant Strain Plot

The DNV2.7-1 standard does not indicate how much of a strain is considered substantial. To be recognized as a major unending distortion by bare eyes, a zone of plastic distortion must be quite large. As a consequence, a credible criterion is developed to better interpret the results, as shown below. When the spread of a plastic strain exceeds 10 in length, it is deemed substantial. Figure 8 shows the maximum plastic strain plot. The maximum plastic strain recorded is less than the permissible limit of 5; hence the model does not shatter. This plastic strain is distributed across a very small area, as illustrated in the larger view in Figure 7.

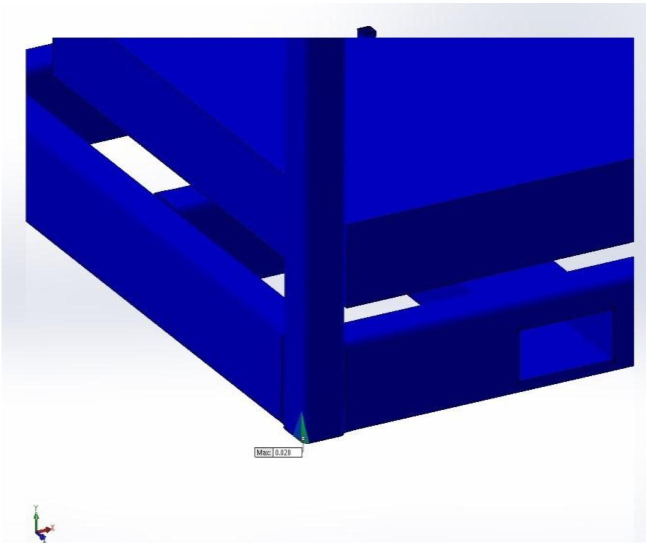

Figure.8 Resultant Strain plot zoom in results

Table.2 Von Misses Stress Summary

Sr.No	Types	Analysis Value (Psi)	Allowable Value (Psi)	FOS
1	Von-Mises Stress	28,108	46,000	1.6

Table.3 Displacement Summary

Sr.	Beam	Analysis value (in)	Allowable Value (in)	FOS
1	Long Beam	0.138	0.574	4.2
2	Short Beam	0.138	0.333	2.4
3	Corner Beam	0.138	0.428	3.1

The above tables show the results of drop test analysis, table.2 shows the stress results and table.3 describe the displacements values of analysis.

VII. CONCLUSION

Based on literature study, it can be said that there is always a variation in results obtained through fe analysis compared with test results. acceptable percentage variation in result has to be decided by individual based on complexity considered in simulation and severity of failure. based on fea results of this thesis it can be said that reasonable agreement is achieved between test and simulation results. 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, small plastic deformation are difficult to identify by the naked eyes, which make it difficult to correlate results point to point. however some broad correlations are achieved like, no cracks or rupture strain are observed in fe simulation which co relates with testing results. no significant plastic deformation are observed in fe simulation which co-relates with testing results. energy balance plot and mass scaling plot gives a confidence of correctness of fe simulation. velocity plot and reaction forces are acceptable.

REFERENCES

[1] H. R. Zarei and M. Kröger(2007), “Crashworthiness optimization of empty and filled aluminum crash boxes,” *Int. J. Crashworthiness*, vol. 12, no. 3, pp. 255–264, doi: 10.1080/13588260701441159.

[2] Anon, "LS-DYNA Keyword User's Manual", Volume I, Version R7.0, Livermore Software Technology Corporation, February 2013. [2] N. Dharwadkar, K.P. Adivi," Modelling of Engine Suspension Components for Crash Simulations" Masters thesis, Department of Applied Mechanics, Chalmers University of Technology, Sweden 2011.

[3] Y. Liu, B. Liaw, " Drop-weight impact tests and finite element modeling of a cast acrylic/aluminum plates", Polymer Testing, Vol. 28, New York, USA , July 2009, pp. 808-823.

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

[5] . DET NORSKE VERITAS (DNV), standard for certification no. 2.7-1 offshore containers April 2006.

ISSN : 2455-9679
Impact Factor : 4.256

[6] J. Lonn and J. Navred, "Drop test of a soft beverage package experimental tests and a parameter study in abaqus", Masters thesis, Division of Solid Mechanics, Lund University, Sweden, 2006.

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

[8] K. E. Jackson & E. L. Fasanella, " Crash simulation of a vertical drop test of a commuter-class aircraft", International Journal of Crashworthiness, Vol.10, NASA Langley Research Center, Hampton, VA, 2005, pp. 173-182.

[9] A. Harrysson," Drop Test Simulation of Cellular Phone", Masters thesis, Division of Solid Mechanics, Lund University, Sweden, 2003.

[10] L. T. Huang, , "Study on the drop test of thin-wall product", Ph.D. Thesis, June 2003.

[11] E. L. Fasanella and K. E. Jackson,“Best Practices for Crash Modeling and Simulation” NASA/TM□ 2002-211944 ARL-TR-2849, Hamption, Virginia, 2002.

[12] T. Noguchi , K. Shimizu, N. Takahashi , T. Nakamura, "Strength evaluation of cast iron grinding balls by repeated drop tests", Wear, Vol.231, Japan, May 1999, pp. 301-309.

[13] K.E. Jackson, and E.L. Fasanella, "Crashworthy Evaluation of a 1/5-Scale Model Composite Fuselage Concept," 55th American Helicopter Society Annual Forum and Technology Display, Montreal, Canada, 1999, pp. 25-27.

[14] J. O. Hallquist, “LS-DYNA Theoretical Manual”, Livermore Software of Technology Corporation, May 1998.

[15] E. W. Klamerus, M. P Bohn, D. A. Wesley, C. N. Krishnaswamy "Containment Performance of Prototypical Reactor Containments Subjected to Severe Accident Conditions " prepared for U. S. Nuclear Regulatory Commission, Chicago, December 1996.

[16] D. McDowell, R. Ellis editors "Advances in Multiaxial Fatigue", ASTM STP 1191, Saptember 1993.