FEA Simulation and Validation on Prototype of Compressor Crash Frame for Drop Test as per Industry Standard DNV 2.7-1

Pankaj S. Anjikar,
D. Y. Patil college of engineering, Akurdi, pune-44

Prof. Dr. Dhananjay R. Panchagade
(HOD Mechanical Department)
D. Y. Patil college of engineering, Akurdi, pune-44

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. Conclusion is made based on result correlation. LS-DYNA version 971, commercial used software is used for drop test simulation, Hypermesh v11.0 is used for meshing and set up, Hyper graph v11.0 and Hyper view v11.0 is used for post processing.

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

1. 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.[5].

2. 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 (N_f) at lower drop heights (*H)* giving H–N_f 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 (K_{Id}) in both irons. The stress causing spalling by 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.

3. 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.

4. Scope of work

This paper can be used to understand methodology used to crate FE model for drop test simulation using LS- DYNA as solver, Hypermesh v11.0 is used to build FE model, HyperView v11.0 and HyperGraph v11.0 is used for post-processing. FE methodology covers Geometry import, geometry clean up, procedure to create meshed model, time step calculation, material application, mass application, boundary conditions application, post processing results, interpretation of results and correlating with test results and concluding the results.

5. Modelling Approach

5.1. CAD Model

Figure 1 shows CAD Model of crash frame, created in Autodesk Inventor 2011.

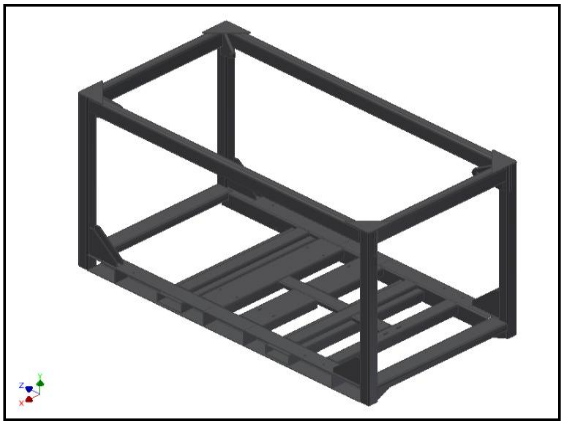

Figure 1. Geometry of crash frame

5.2. Meshing

Crash frame is discretized using 2d shell elements (QUAD4 and TRIA3 elements) using meshing software Hypermesh v11.0. Following steps are followed for meshing the frame.

1. Import the geometry in Hypermesh software.
2. Difeaturing the geometry by removing small holes, small fillets and trimming small extended surfaces etc.
3. Mid surfaces extraction. It is a process by which surface is generated exactly at mid of the thick plate.
4. Surface modification operations like extend, trim, fill etc are performed to get correct midsurface geometry.
5. All mid surfaces are manually meshed using quad4 and tria3 elements, meshing is done considering quad dominant mesh, which accounts for maximum quad elements. Tria elements are used only where quad elements are not possible total number of tria elements are restricted to 5%.
6. Nodes are merged at welding location.
7. Different Material is created, based on material of different component in crash frame.
8. Different Properties are created, which contains the information about material and thickness of the member.
9. Properties are assigned to shell elements as per geometry.

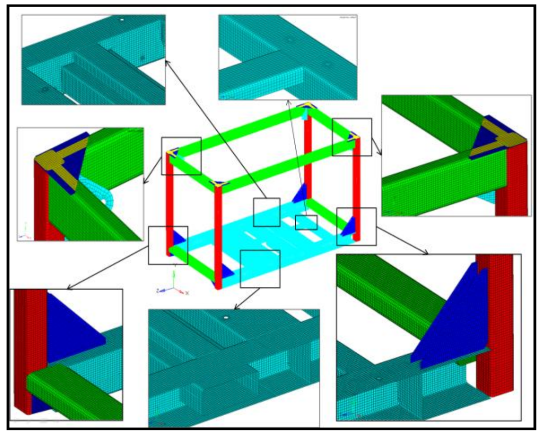

Figure 2. Meshed Model of crash frame

Table 1 shows total no of nodes and elements in FE model.

Table 1. Element and node count

Mesh Size	No of Elements	No of Nodes
10	3,13,754	2,16,782

While meshing the geometry minimum element size restricted to 5mm and average mesh size is 10 mm. Limiting value for different quality criterion parameters, which are considered while creating mesh as shows in Figure 3.

Figure 3. Quality criteria

In an explicit method to achieve accurate result requires smaller time increment. The critical time step depends on the material properties and the size of the element in explicit time integration. LS-DYNA calculates the time step for each element and the minimum element time step is used in the simulation. For shell elements the time step calculation is given by [14],

$$\Delta \mathbf{t_e} = \frac{\mathbf{L_s}}{\mathbf{c}} \qquad (5.1)$$

Where, Δt_e = Time step, L_s = Minimum characteristic length of an element, c = Speed of sound in element material. Sound speed for 2d continuum is given by,

$$c = \sqrt{\frac{E}{\rho(1-v^2)}} \qquad (5.2)$$

Where, E = Young's Modulus, ρ = Mass density, c = Positions ratio. By putting value of above parameters in equation 5.1 and 5.2 we get value of critical time step equal to 10^{-6} s (1 micro second). An automatic contact (CONTACT_AUTOMATIC_SINGLE_SURFACE) was specified for the model, which is a generic contact definition in LS-DYNA that prescribes that no node can penetrate through any surface in the model[8].

6. Material Properties

The material model used for all three material is MAT_PIECEWISE_LINEAR_PLASTICITY, it is used to define nonlinear material properties using a card called MAT_24. Material properties are listed in Table 2 used in FE model for different components material [15] & [16].

Table 2. Material properties

Component	All Plates	Big Tubes	Small Tubes
Material	**Steel A1**	**Steel A2**	**Steel A3**
E-modulus [MPa]	210,000	210,000	210,000
Poisson ratio [-]	0.3	0.3	0.3
Density [Tons/mm³]	7.85 e-9	7.85 e-9	7.85 e-9
Yield strength [MPa]	260	240	220
Ultimate strength [MPa]	487	417	360
Strain at rupture [%]	21	21	25

Figure 4 shows material plot of components of crash frame.

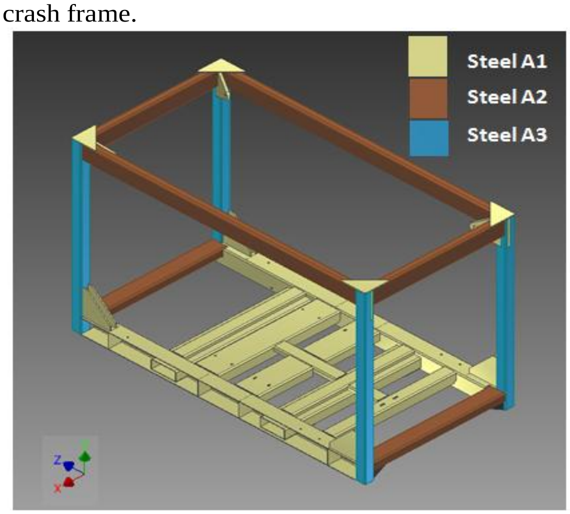

Figure 4. Material plot

7. Loads and Boundary conditions

Mass details of the crash frame and compressor is listed in Table 3.

Table 3. Mass details

Components	Mass (kg)
Crash Frame	1370
Compressor (Mass of Steel plates in testing)	2162
Total	3532

Mass of the compressor is applied on nodes of lower part of the frame on which steel plates are resting as shown in Figure 5. Acceleration due to gravity 9810 mm/s^2 applied in negative Y direction.

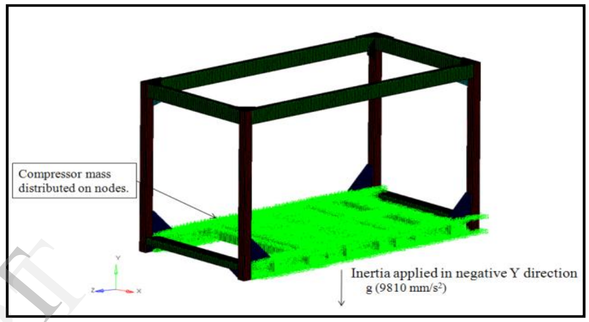

Figure 5. Load application

Crash frame is tilted about x- axis and z- axis such that lowest corner of frame forms an angle of 5° with ground floor and then frame is translated 50 mm above ground floor as per the requirement of the DNV standard. Figure 6 explains the position of Crash frame from which it dropped on concrete floor. Concrete floor is defined as rigid wall and all nodes of the frame are defined as salve nodes. Since concrete floor is rigid wall, it will not have any deformations on event of drop. Coefficient of friction value consider is 0.45. Automatic single surface contact type is used to define the contacts of all components with each other and ground floor is defined as rigid wall in this analysis.

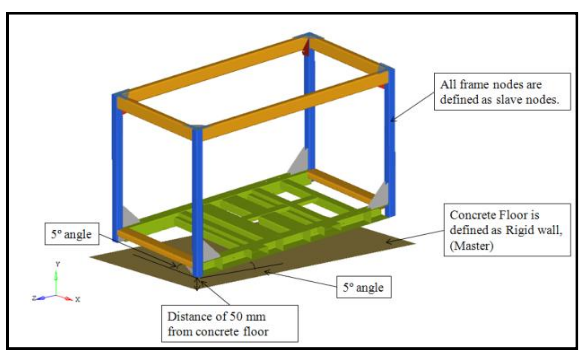

Figure 6. Dropping position in FEA

ISSN: 2278-0181
Vol. 2 Issue 12, December - 2013

8.Experimental Set up

Experimental set up is shown Figure **7** .Crane is used to lift the crash frame with the help of lifting slings from floor. Crash frame is lifted such that one of the corner is 50 mm above the floor and other diagonal corner is 370 mm above the floor, which is less than 400 mm as per requirement of DNV standard. Steel plates of equivalent mass of compressor are used to represent compressor mass, which are placed on wooden battens to get more even load distribution. Frame is dropped freely from this height to have impact with floor. Crash frame is visually inspected for cracks and any permanent deformation/damages, no cracks and permanent deformations are observed. Images are captured on four corner, four fork slots and different location after drooping the frame. Experimental arrangement and visual observations are witnessed by DNV inspector.

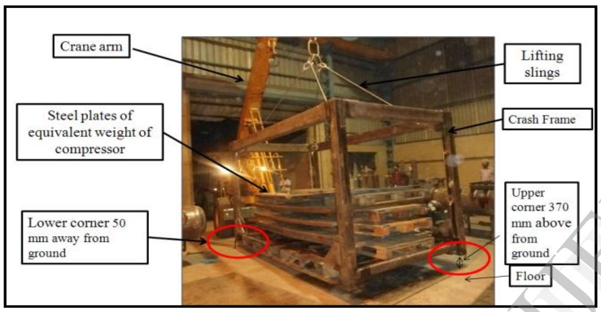

Figure 7. Experimental set up

9. Simulation to identify critical regions

Before performing simulation of actual frame it was necessary to identify critical regions in geometry which are prone to more damage, so that more attention will be given to those regions along with other region while witnessing the physical testing. To achieve this, a simulation has to be carried out by weakening the frame. So, the weakening of the frame is done by reducing the thickness of all members by 50%. This will make frame weak compared with actual frame and weak regions of frame results into higher stress/deformations (damage) to the frame.

Based on results received four corners and four fork lift slots are identified as critical region which are prone to more damage. Hence in actual model these regions has to be witnessed carefully while performing physical testing.

Figure 8 and Figure 9 shows plots for Effective plastic strain, which is set to 0.05 that is 5% plastic strain. All red colour region in frame indicates plastic strain above 5%. Maximum plastic strain observed is 23%, which is more than strain value at rupture that is 21% near fork slot 3, for location refer Figure 11, so it can be considered that crack or rupture is initiated in weak frame.

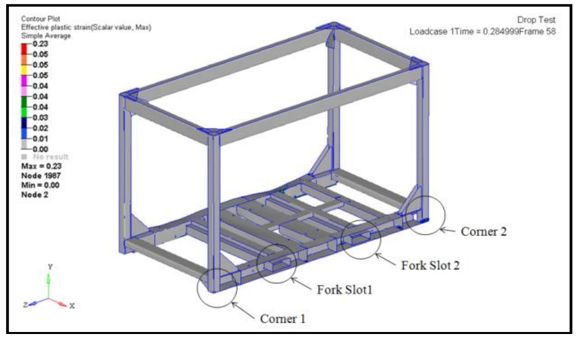

Figure 8. Plastic Strain, Orientation1 (weak frame)

Figure 8, shown a Plastic strain plot, encircle region Corner 1 and Corner 2, fork slot 1 and fork slot 2 are identified as weak regions as plastic strains are spread across wide area in this region with some places exceeding plastic strain more than 5%. Refer Figure 10 and Figure 11 for zoomed plots.

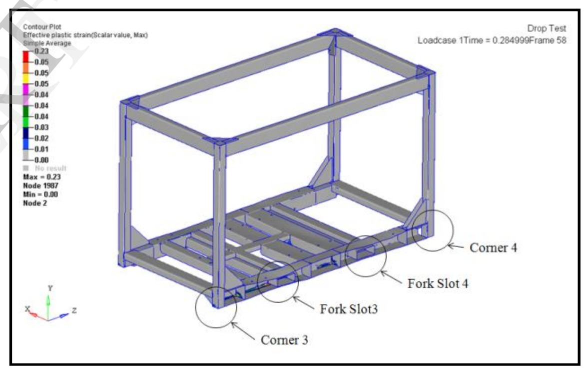

Figure 9. Plastic Strain, Orientation2 (weak frame)

Figure 9, shown a Plastic strain plot, encircle region Corner 3 and Corner 4, fork slot 3 and fork slot 4 are identified as weak regions as plastic strains are spread across wide area in this region with some places exceeding plastic strain more than 5%. Refer Figure 10 and Figure 11 for zoomed plots.

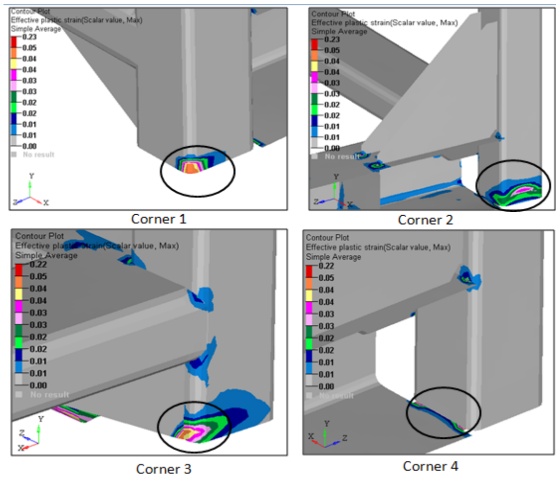

Figure 10. Plastic Strain on four Corners (weak frame)

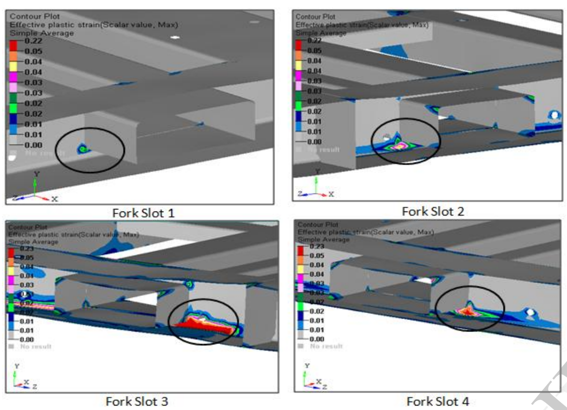

Figure 11. Plastic Strain near four fork slots (weak frame)

Encircled region in Figure 10 and Figure 11 indicates weak region as plastic strain is spread across wide region and exceeding 5% plastic strain at some location, Red colour indicates plastic strain above 5%.

10.Results and Discussion

To verify if the solution is accurate the energy values can be examined. A good way to verify the solution is to investigate if the total energy is constant during the cause of the simulation. The total energy term is constant since the energy cannot disappear only be transformed. In numerical methods the solution is not completely constant but should not vary by more than 10%, which is standard practice followed in industry.

In the explicit solution method the energy levels are controlled to establish the correctness in the discretization of the FE-simulation model. If the energy levels are acceptable a mesh density with fewer elements can be tested to achieve a shorter solution time. A mesh containing a larger amount of elements can also be tested to investigate if the deformation obtains a smother appearance.[6]

Figure **12** shows a energy balance plot, total energy curve is very smooth and no abrupt changes observed and variation of energy remain within 10% range, this give confidence of correctness of FE simulation.

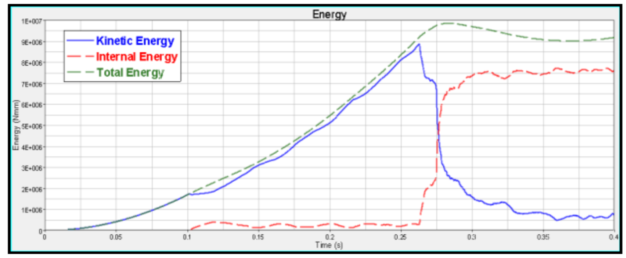

Figure 12. Energy Balance plot

Figure 13 shows mass scaling plot, Since very small time steps are needed for an explicit analysis the element size is of greater importance for numerical stability. LS-DYNA automatically detects elements which have time steps less than the critical time step and adds nonphysical mass to its nodes in order to achieve numerical stability at higher time steps [11],which is called mass scaling. Percentage increase of non physical mass is 0.07% which is less than 2%, which is acceptable. This also give us confidence of correctness of FE modeling.

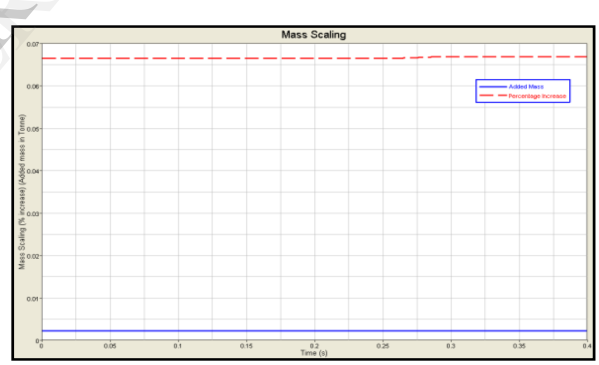

Figure 13. Mass Scaling

Figure 14 shows velocity plot in Y direction, plot shows that first impact occurs at 1m/s velocity (highlighted by red colour circle), which was the requirement of DNV standard.

Figure 14. Velocity Plot in Y direction (mm/s)

Figure 15 shows plot of reaction force on ground and maximum reaction coming on ground.

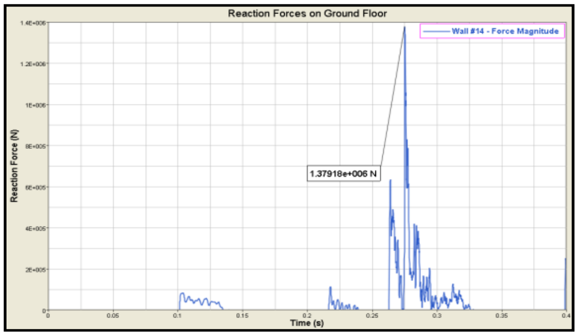

Figure 15. Reaction Force (N) on Floor

As per DNV2.7-1 standard after experimental test visual inspection is carried out to identify region of significant permanent deformation. DNV2.7-1 standard does not specify the quantity of plastic strain that can be considered as significant value. A region of plastic deformation has to be considerable large, in order to identify it by naked eyes, as a significant permanent deformation. Hence acceptable criterion is created to interpret results in better way, described as below.

A plastic strain above 5% is considered as significant value, provided its spread over 10% in length. The dimensions of frame is 3600 mm x 1840 mm x 2220 mm.

A plastic strain above 5% with the spread region 360 mm across the length or 184 mm across the width or 220 mm across height can be considered as significant plastic deformation. Maximum plastic strain plot is shown in

Figure 16, maximum plastic strain observed is 18% which is less than allowed limit of 21%, hence no rupture or crack is initiated in the model. This plastic strain is spread over very minute region of one element as shown by enlarge view in

Figure **16**

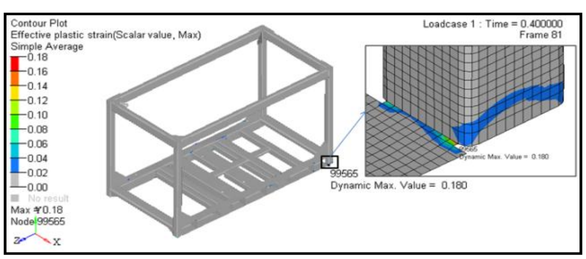

Figure 16. Maximum Plastic Strain (Actual frame)

Figure 17 shows a plot of maximum vonMises stress, Maximum vonMises stress 356 MPa which is observed near corner4, enlarge view in figure shows the same.

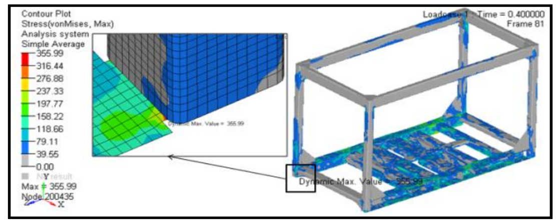

Figure 17. Maximum vonMises stress (MPa)

Figure 18 shows plastic strain in which fringe is set to 5% to identify the region in which plastic strain is above 5%. Red colour region indicates plastic strain above 5%.

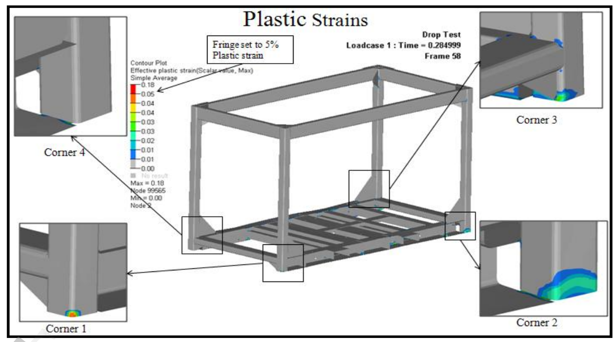

Figure 18. Plastic Strain(Actual Frame)

Figure 19 shows a enlarged view of plastic strain on 4 corners , its observed that a region of plastic strain above 5% spread over very local area in all four corners, which can not be considered as significant.

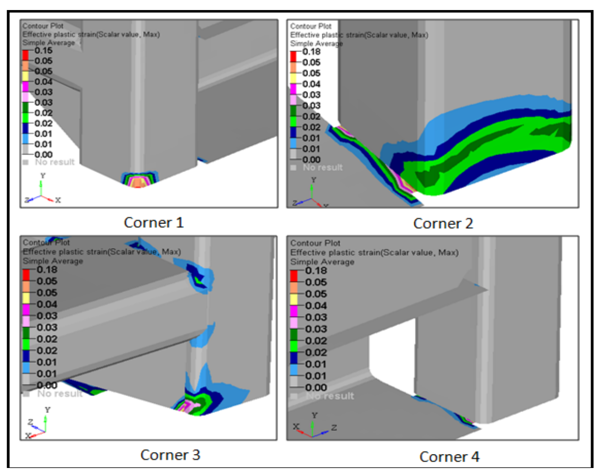

Figure 19. Plastic Strain on four Corners(Actual frame)

Figure 20 shows a plastic strain plot on four fork slots at time step 0.285 s, except fork slot 3 remaining fork slot does not show plastic deformation above 5%. Near fork slot 3 some region shows plastic deformation above 5% encircled region in the figure. Red colour indicates plastic strain above 5%, the distance across which it is speeded is 170 mm, which is less than 360 mm. However this region were inspected carefully in physical test to identify plastic deformations.

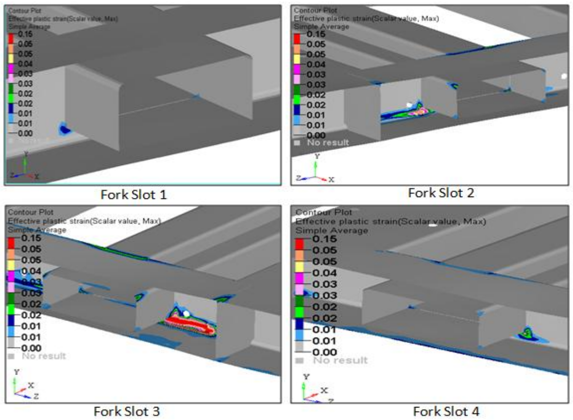

Figure 20. Plastic Strain near four fork slots (Actual frame)

11.Comparison of Physical test and FE Analysis Results

Figure 21 shows comparison plot of maximum plastic strain observed in physical test and FE analysis results for corner 2, since region of maximum plastic deformation is very small no significant plastic deformation observed in physical test.

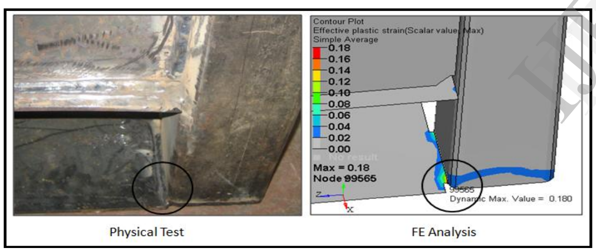

Figure 21. Comparison of Physical test and FE analysis Results (Maximum Plastic Strain on Corners 2)

Figure 22, shows a comparison plot for plastic strain observed in physical test and FE analysis results for all 4 corners. Since, region of plastic deformation is very small no significant plastic deformation observed in physical test.

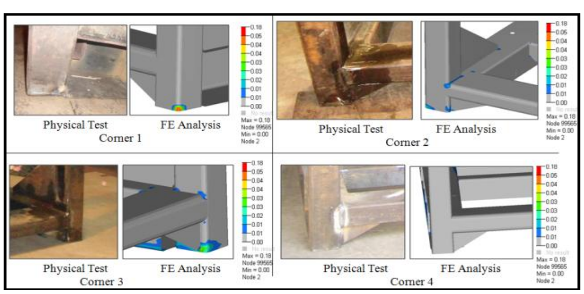

Figure 22. Comparison of Physical test and FE analysis Results (Plastic Strain on four Corners)

Figure 23, shows a comparison plot for plastic strain observed in physical test and FE analysis results for fork slot3, since we observed some region with plastic strain near fork slot 3, it was essential to give attention to this region separately in physical test. In physical test this region is carefully inspected visually however no significant deformation observed. Region with red color indicates plastic strain above 5%.

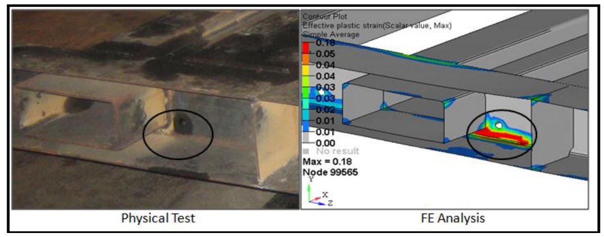

Figure 23. Comparison of Physical test and FE analysis Results (Plastic Strain on fork slots 2)

Figure 24, shows a comparison plot of plastic strain observed in physical test and FE analysis results for all 4 fork slots. Since, region of plastic deformation is very small no significant plastic deformation observed in physical test.

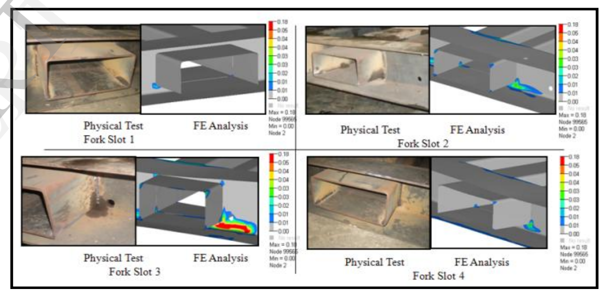

Figure 24. Comparison of Physical test and FE analysis Results (Plastic Strain on four fork slots)

Physical testing is done in presence of DNV inspector, testing certificate is evidence for test set up compliance as per DNV 2.7-1 and observation of physical testing is same no cracks and permanent deformation observed after drop test.

12. 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.

This concludes FE modeling strategy considerably acceptable for this case, however different model should be checked with different loading condition to arrive at correctness of modeling strategy of the simulation. The simulation results adds confidence to the modeling approach and endorses the mesh size, material models, element selection and contact definitions specific for problem statement describe in this paper. This drop test simulation helps to improve the design of crash 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 50 mm, which consumes approx 76 hrs of computational time. Velocity can be calculated before the frame touches to the ground, by imposing this velocity simulation time can be reduced.

13. References

[1] 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.
[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.
[17] ABAQUS Analysis User's Manual, version 6.11.
[18] Anon, "LS-DYNA online User's Guide" Livermore Software Technology Corporation.