ISSN 2395-1621

Nonlinear Elastic Analysis of Structural Frame Under Dynamic Load Condition

[#1]Mr.Vrushabha sagar Patil,[#2] Mr.D.G.Joshi,[#3] Mr.Kiran Bhagate[#]

[1]vsagar.patil@gmail.com

[2]dinesh.joshi@sinhgad.edu

[3]kbhagate@googlemail.com

[#123]Department of Mechanical Engineering, SPPU

ABSTRACT

In a Genset, a single point lifting frame is a frame that enables a crane to lift the Genset with a single hook and with minimum human resources. There by offering an ease of mobilization in a remote construction sites or while loading / unloading.Due to a high competition in the market, one of the biggest challenges in the current mechanical industry is cost optimization. Therefore, the organizations like JCB are continuously looking for advanced analysis techniques to be able to predict the structural response accurately so that design & cost optimization can be carried out. The current project highlights various levels of analysis and its impact on the structural optimization process of a lifting frame.A lifting frame with operational load 2700 kg was proposed by JCB based on their past experience. The initial solid geometry of the lifting frame was modeled using Unigraphics. Less important features such as small holes, radii and champers were deleted in order to avoid concentration of finite element mesh. This allows the solution process to work faster and at the same time maintain the accuracy of the global results. The solid geometry was then imported into ANSYS Workbench 15. The material properties of standard structural steel were defined. A frictional non-linear contact was defined between components connected using bolts. Also, due to the slender size of the frame, geometric non-linearity was considered for the analysis. The meshing was carried out using Solid 185 elements from ANSYS Workbench 15. Two different analysis as: a) Contact non-linearity, b) geometry & contact non-linearity, were carried out to compare the results.The study helped to decide the level of complication that should be considered in the current lifting frame analysis and its effect on the optimization process. Optimization is a methodology of making something(as a design, system, or decision) as fully perfect, functional & effective as possible. FEM tool is used for analyzing existing frame with different types of FEA techniques. Strain Gauging is done for correlation with FEA strain to confirm the loadings.

Keywords— **a Single Point lifting, Thickness Optimization, Finite Element Analysis, Large Deformation**

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

I. Introduction

Single point lifting frames are designed to enable mobilization of a Genset using single hook of a crane. This lifting frame is assembled to a base frame by bolting arrangement. Point of attachment in the base depends upon the center of gravity of total Genset.

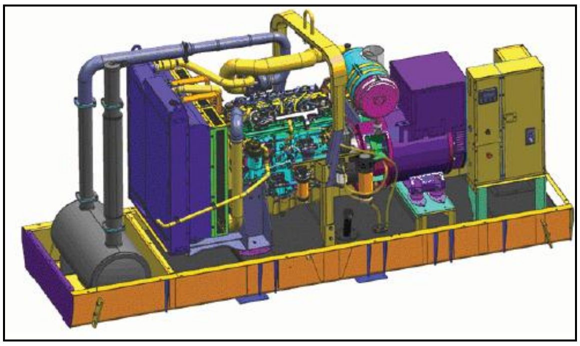

Fig. 1 Single point lifting frame assembled to a base frame.

A 'U' bolt is a forging member and it is a top most member of the lifting frame in which a hook of a crane is engaged. A lifting frame is made up of 8 mm thick plates. In the proposed lifting frame mainly bolted and welded joints are used. The material properties of standard structural steel were defined. A frictional non-linear contact was defined between components connected using bolts. Also, due to the slender size of the frame, geometric non-linearity was considered for the analysis. For testing such conditions a finite element analysis tool is used. Using this tool, the loading conditions are applied, which on solving gives the stress results. It is important to take a note that the values that we are looking for stress are on the frame and not the in the U bolt. The stress plots help us to know the highly stressed areas. Taking into account the stressed areas, a stain gauge could be placed at those locations for experimental setup, so as to compare with the results of FEA.

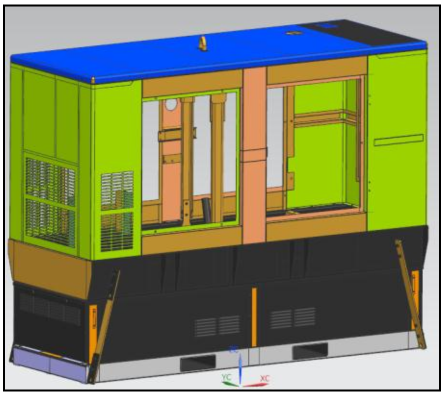

Fig. 2 Genset and base frame

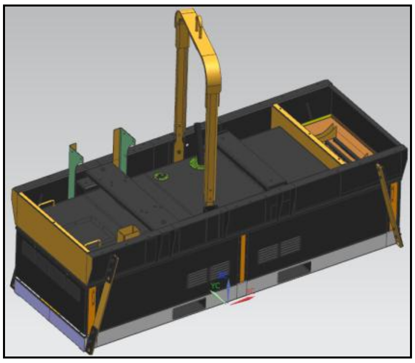

Fig. 3 Single point lifting frame assembled to a base frame.

II. MATERIAL

A. *Material Properties of Frame.*

1) *Material: AISI 1045 [1]*
2) *Yield strength: 241N/mm2*
3) *UTS: 448N/mm2*
4) *Poisson Ratio: 0.27 – 3.0*
5) *Density: 7800 kg/m3*

III. Geometry

Only a quarter model with symmetric boundary conditions was applied. This reduces the solution time of the analysis with keeping same accuracy.

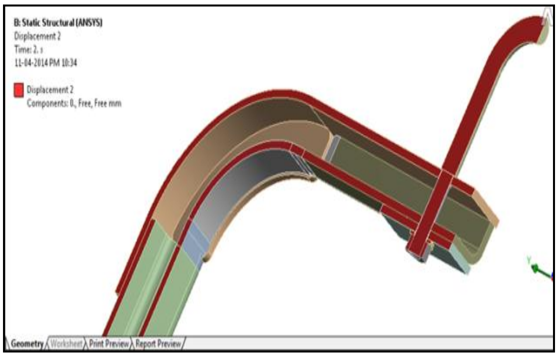

Fig. 4 Symmetric boundary condition 1

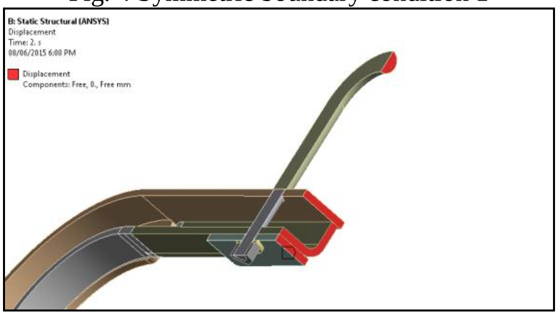

Fig. 5 symmetric boundary condition 2

IV. Finite Element Analysis

A. *Meshing.*

1) *Meshing: Solid 185 elements.*
2) *Sphere of influence mesh size: 5mm.*

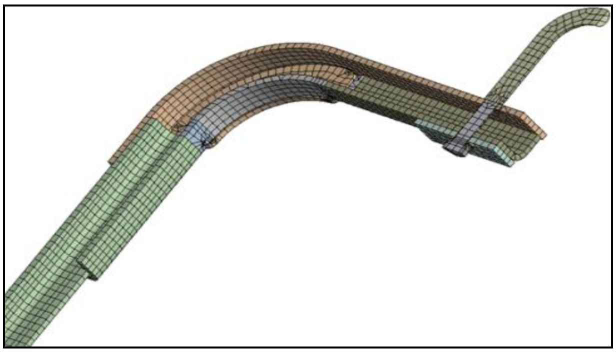

Fig 6 Meshing with 15mm mesh size

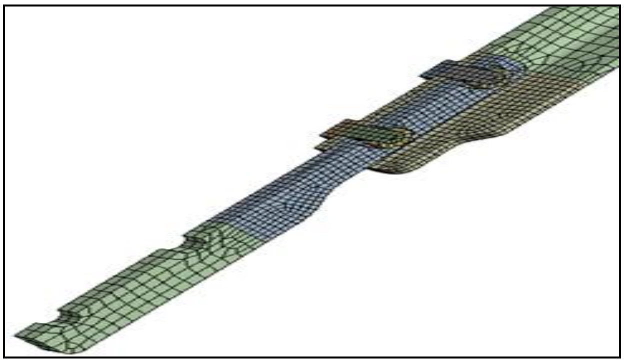

Fig 7 Meshing with 15mm mesh size

V. LOADING CONDITIONS

A. Earth gravity with given load were gradually increased till 10000 kg is applied at the overall C of G of total machine. Earth gravity of 9.81 m/s^2 was applied. The incremental load was applied to study the stress – strain curve of the structure.

B. Fixed support is given at the U bolt.

C. Remote displacement is given to the bottle shape joining plates to facilitate the actual lifting motion.

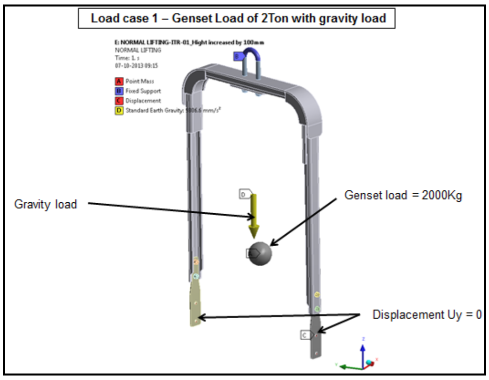

Fig.8 loading condition

VI. RESULT

Initially, linear analysis is done so as to check the linear characteristics of behavior. For linear type, contact type is considered as ‘No Separation’ type. Further, nonlinear analysis is carried out by defining the contacts as ‘Frictional’ with the coefficient of friction as 0.3 for dry & sliding type.

Two types of models (1) frictional contact, (2) frictional contact & geometric non-linearity were solved & checked for gradual increase of load upto 10,000 N on quarter model.

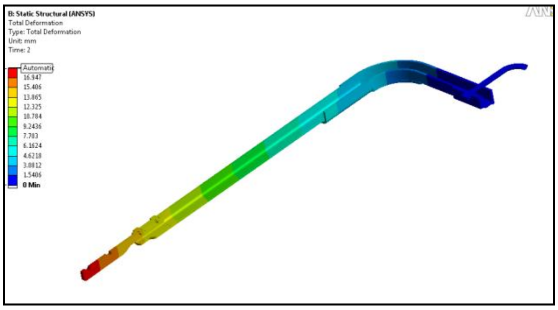

Fig 9 Maximum Deflection for 10000 N with Geometrical Non-linearity i.e. large deformation theory

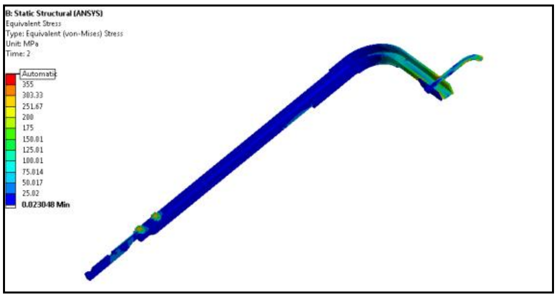

Fig 10 Equivalent Stress (Large Deformation Theory)

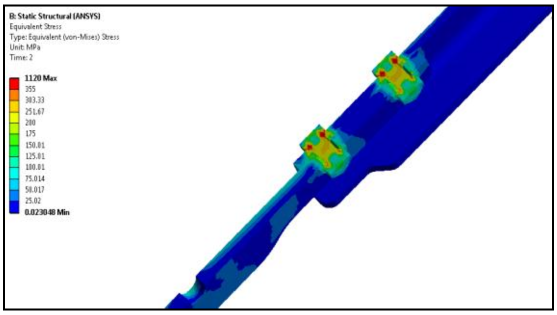

Fig.11 Equivalent Stress (Large Deformation Theory)

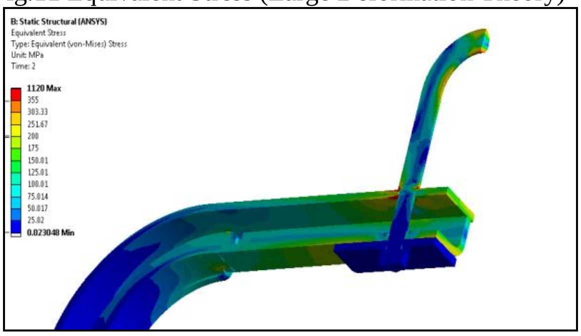

Fig.12 Equivalent Stress (Large Deformation Theory)

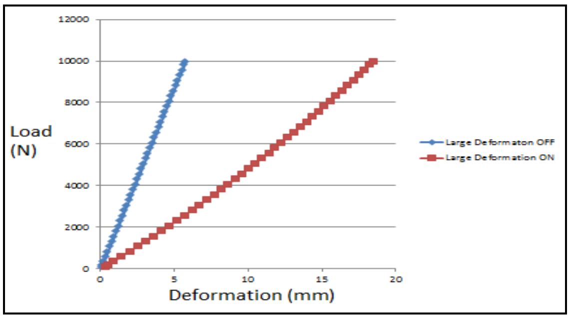

Fig 13 Load Vs Displacement

Nonlinear analysis allows for accurate modeling of structures that undergo large deformation. Hence, nonlinear analysis type is selected for further analysis. It is seen that the stress occurs on the bottle shape plate while the rest portion is safe.

Initial thickness of the plate was 8 mm. Results from Fig.9 show that the maximum deformation appears at a lower end of the bottle shaped plate. Fig 18 to Fig 12 it shows the results obtained are safe i.e. no failure (red spot) is seen on plate.

Observing the results from Fig 13 (Load Vs Displacement plot) a large difference in the deformation values is obtained. It shows that, when large deformation is 'OFF' the deflection is 5.82mm whereas when large deformation is kept 'ON' the deflection is 18.5mm. It shows that large deformation theory has significant contribution in obtaining structural response correctly, which could be comparable with experimental results.

VII. Experimental Setup

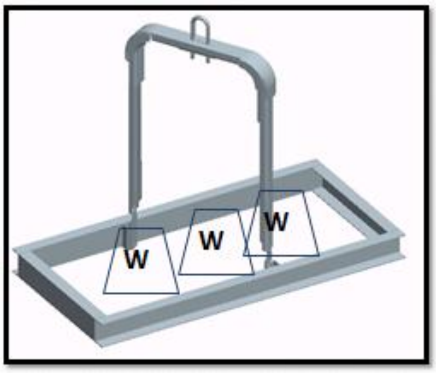

Fig.14: Mockup base frame for experimentation purpose

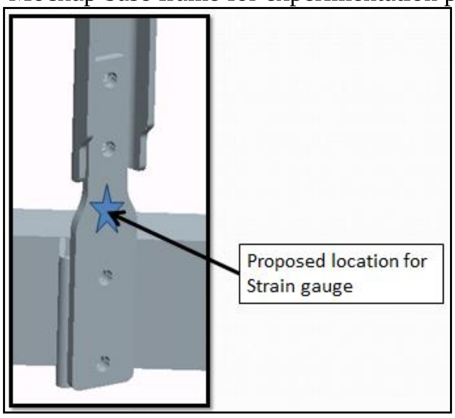

Fig.15 Proposed location for strain gauge

Once the frame will be available after manufacturing to the concluded dimensions, strain gauging on the same will be done. The mockup base frame will be manufactured locally to support the weight. This base frame will be manufactured with standard channels and will represent the actual base frame of the product. Strain gauge is applied at the area where stress occurs. The area of stress could be seen in the analytical data generated to get the idea for placing strain gauges.

Fig 14 shows the proposed mockup base frame.Results are generated using strain gauge which is connected to the data logger for recording the results which could further be decoded and compared with the analytical data.

Fig 15 shows the proposed location for strain gauge.

TABLE I
RESULTS

Sr.No.	Condition	Maximum Total Deformation (mm)
1	Contact nonlinearity	5.82
2	Geometric nonlinearity + contact	18.5

VIII. CONCLUSION

1. Two different analysis as:
   a. Contact non-linearity
   b. Geometry & contact non-linearity, were carried out to compare the results.

2. Large variations in the results were observed when geometric non linearity was used in the analysis. This was mainly due to slender nature of the frame. The experimental response is expected to be closer to the model having frictional + geometric non linearity. This will be confirmed after experiment is carried out.

Acknowledgment

The authors would like to thank the JCB India Ltd, and IDAC India Pvt Ltd. for the timely and important help in the field of Design & Finite Element Analysis. The authors also wish to thank the reviewers for their very detailed review of the manuscript and for their help in clarifying some aspects of the reported results.

References

[1] "Handbook of Comparitive World Steel Standards" by John E. Bringas.

[2]http://www.engineershandbook.com/Tables/frictioncoefficients.html

[3] Books : R.D. Cook – Introduction to FEA

[4] Chen WF, Kim SE. LRFD steel design using advanced analysis. CRCPress, 1997.

[5] Ekhande SG, Selvappalam M, Madugula MKS. Stability function for three-dimensional beam-columns.J StructEngng ASCE 1989;115(2).

[6] Fleming JF. Nonlinear static analysis of cable-stayed bridge structures. In: Computers and structures,vol. 10. New York, London: Pergamon Press Inc. 1979. p. 621–35.

[7] Harrison HB. Computer methods in structural analysis. Englewood Cliffs (NJ): Prentice-Hall, Inc,1973.

[8] Horne MR, Merchant W. The stability of frames. London, England: Pergamon Press Ltd, 1965.

[9] Kanaan AE, Powell GH. General purpose computer program for dynamic analysis of inelastic planestructures.Department of Civil Engineering, University of California, Berkeley, CA, 1973.

[10] Liew JYR. Advanced analysis for frame design. PhD thesis, School of Civil Engineering, PurdueUniversity, West Lafayette, IN, 1992