MANTECH Publications

Journal of Product Design, Quality Engineering & Technology

Volume 1, Issue 2, May-August 2016

Linear Static Analysis of Pressure Vessel Skid Frame and Its Optimization

Ms. Kalyani G. Katole[1], Prof. Sandip B. Patil[2]

Lecturer[1,2]

Department of Mechanical Engineering

[1]Jayawantrao Sawant Polytechnic, Hadapsar, pune-28

[2]Jayawantrao Sawant College of Engineering, Hadapsar, Pune-28

E-mail: *kalyanikatole9@gmail.com[1]*

Abstract

This project will outline the issues and approaches involved in skid design for static loads such as in pressure vessels. The project presents the study of performance characteristics of the support frame structures also called as skid frame, according to changes in their connections, considering the loads caused by machine has been done and compared with the limits of displacements of the structures. Skid Base Frame is a structural assembly consisting of beams of various cross sections and dimensions. The base frame is subjected to gravitational loading of all the components mounted like as Compressor, Air Receiver vessel etc. The frame discussed in this report was designed with conventional CAD design practices and then analysed statically with FEA software.

The analysis was carried out to determine the induced stresses and the deflections at various locations on proposed frame. The structure was optimized to reduce weight. In this project the experimental and Finite Element Analysis of skid Frame for Rigidity is been carried out. The base frame is analysed using Hyper mesh and ANSYS software. The static loads are applied on the frame and analysis is carried out. The test is carried out to check the conformance of the existing design. The skid base frame is modified for making a stiffer design within the allowable stress limits. Based on these results, best feasible design solution is proposed and validated experimentally.

Page 78-90 © MANTECH PUBLICATIONS 2016. All Rights Reserved

Keywords: *Skid Base Frame, CAD model, FEA loading, Static Analysis, Weight Optimization*

INTRODUCTION

Pressure vessel

A pressure vessel is a closed container designed to hold gases or liquids at a pressure substantially different from the ambient pressure. The pressure differential is dangerous, and fatal accidents have occurred in the history of pressure vessel development and operation. Consequently, pressure vessel design, manufacture, and operation are regulated by engineering authorities backed by legislation. For these reasons, the definition of a pressure vessel varies from country to country, but involves parameters such as maximum safe operating pressure and temperature, and are engineered with a safety factor, corrosion allowance, minimum design temperature (for brittle fracture), and involve non-destructive testing, such as ultrasonic testing, radiography, and pressure tests, usually involving water, also known as a hydro test, but could be pneumatically tested involving air or another gas.

Skid Frame

Skid Frame is a structural assembly consisting of beams of various cross sections and dimensions. The sections used may be of equal dimensions and cross sections, or a combination can be used for optimum strength and weight. The sections can be of IS standard dimensions or custom made. Placement of beams is done in such a way that all the footprints of the components are covered. Different arrangements for assembly lifting (for e.g. hook lifting lugs, fork lifting insertion passages) are provided in the skid frame. The frame was designed and analysed for 3g loading under both hook lifting and fork lifting process. For sufficiently lower values of deflection (< 2.5 mm), and higher value of FOS(> 1.15), the structure was redesigned using thinner and smaller IS standard sections.

The skid frame consists of the space frame with I channel Section, and eight support strut with hollow square section, placed at the equal span. Stress analysis using Finite Element Method (FEM) can be used to locate the critical point which has the highest stress. This critical point is one of the factors that may cause the fatigue failure. The magnitude of the stress can be used to predict the life span of the skid. So, it should be reduced.

Page 78-90 © MANTECH PUBLICATIONS 2016. All Rights Reserved



Fig1. Skid frame of a pressure vessel

Finite Element Mthod

Finite element method has become a very powerful tool for a wide range of engineering problems. Applications range from deformation and stress analysis of automotive, aircraft, building and bridge structures.In this method of analysis, a complex region defining a continuum is discredited into simple geometric shapes called finite elements. The element material property and the governing relationships are considered over these elements and expressed in terms of unknown values at nodes. An assembly process, duly considering the loading and constraints, results in the set of equations. Solution of this equation gives an approximate behavior of the continuum.

The following are the steps involved in F.E.M:

i. a) Modeling

The first step in FEM is to model the given body or region so as to enable one to discretize the region of interest. As an example, a uniformly varying bar can be modeled to a stepped bar. Majority of the practical problems is modeled before discritization.

b) Discritization of the structure

The second step of FEM is to consider discritization of structure into sub-divisions called finite elements. These elements are considered to be interconnected at nodes. The size, type, number and arrangement of such finite

Page 78-90 © MANTECH PUBLICATIONS 2016. All Rights Reserved

elements are decided based on factors such as discontinuity or abrupt change in structure, type of loading, degree of accuracy required, etc. these parameters are carefully selected so the original domain is stimulated as closely as possible without increasing the computational effort needed for solution.

c) Derivation of element displacement models

Since the actual variation of field variables such as temperature, pressure, stress, displacements, etc. are unknown inside the continuum; simple functions are assumed to represent the field variables within the element. The assumed functions will usually be interpolating functions, which must satisfy the convergence conditions. To satisfy the convergence requirements, these functions

- Must be continuous within the element.
- Must contain rigid body displacements.
- Must contain constant strain states.

d) Derivation of element stiffness matrix

The characteristics stiffness matrix [K] and the driving force/load vectors [f] are to be derived by using either equilibrium conditions or suitable variation principle. In case of continuum problems such as fluid dynamics the terms used are characteristics matrix and driving force vector whereas, in case of elasticity problem, the respective quantities are referred as stiffness matrix and load vectors.

ii. e) Assemblage of elemental equations to obtain overall equilibrium equations

The equilibrium equations for the entire region are obtained by combining the equations for the individual elements in such a way that continuity of displacement / field variables is prescribed at the interconnecting nodes. The derived characteristics of stiffness matrices and load vectors are assembled in a suitable manner and overall equilibrium equations are formulated as,

$[K]\,[q] = [F]$

Where,

[K] = global stiffness matrix

[q] = global displacement matrix

[F] = global force matrix

iii. Solution of unknown nodal displacement / Field variables

In this step, boundary conditions are incorporate and global displacements are obtains by solution of equilibrium equations using techniques such as "GAUSS ELIMATION" method etc.

iv. Computation of results

From the solved nodal displacements, the

Page 78-90 © MANTECH PUBLICATIONS 2016. All Rights Reserved

result / element strains and stresses are calculated using classical relations.

DESIGN OF A SKID FRAME

When designing a structural steel skid (baseplate) for a compressor or pump package, the design must balance stiffness, mass, and cost. High stiffness will help avoid alignment problems due to skid deflection during transportation and installation. Heavier skids tend to have lower overall vibrations, but can have high deflections when lifted. The challenge when optimizing the design is to know where steel can be added or removed to maximize the stiffness and minimize the costs.

TABLE I: CLASSIFICATION OF CARBON STEEL

Low-Carbon Steel	0.05% to 0.30% carbon
Medium-Carbon Steel	0.30% to 0.45% carbon
High-Carbon Steel	0.45% to 0.75% carbon
Very High-Carbon	0.75% to 1.70% carbon

By observing the material properties for application of skid mild steel is used.

TABLE II.:MATERIAL PROPERTIES USED FOR SKID DESIGN

Material used	M.S IS:2062 Gr.A
Young's Modulus	210,000 MPa
Shear Modulus	79,000 MPa
Density	7,850 kg/m3
Yield limit	230 MPa

Page 78-90 © MANTECH PUBLICATIONS 2016. All Rights Reserved

MODELLING

The CAD model of skid frame is generated in CATIA. The loads on skid frame are determined using the weight on the skid and constraints to be applied for fixing the skid frame.

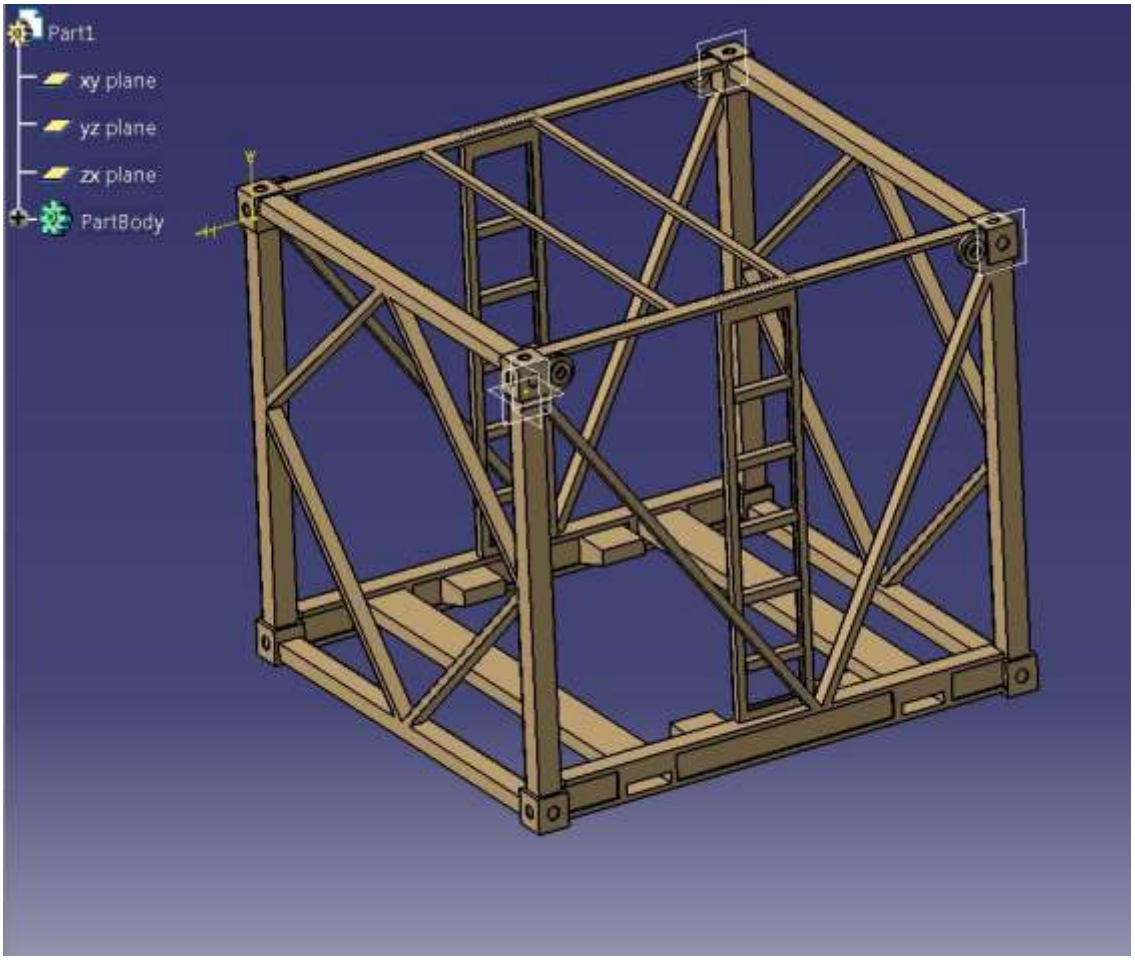

Fig. 2. Generated cad model of skid frame.

MESHING

Basic theme of FEA is to make calculations at only limited number of points and then interpolate the results for entire domain i.e. surface or volume. Any continuous object has infinite degrees of freedom and it's just not possible to solve the problem in this format. Finite element method reduces degrees of freedom from infinite to finite with the help of discretization i.e. meshing.

CAD model is imported in hyper mesh and meshed with appropriate element size and checked for quality criterion. The boundary conditions determined are applied.

Page 78-90 © MANTECH PUBLICATIONS 2016. All Rights Reserved

Finite Element Analysis using HYPERMESH

This section discusses geometry generation used for finite element analysis, describes the accuracy of the model and explains the simplifications that are made to obtain an efficient FE model. Mesh generation and its convergence are also discussed. Using proper boundary conditions and type of loading are important since they strongly affect the results of the finite element analysis. Identifying appropriate boundary conditions and loading situation are also discussed. A finite element model of one component is analyzed. This facilitates proper comparison of this component made from two different manufacturing processes. The results of finite element analysis for this crankshaft are discussed in this chapter. Above mentioned FE models are used for static analysis considering the boundary conditions according to the mounting of the crankshafts in the engine.

General Outline of FEA

a) Pre-processing

The pre-processing consists of initial data input before processing for solution. It consist of defining scalar parameters, model generation, element types and various material properties, mesh sensitivity analysis, loading condition etc. The preprocessing consists of all information about working condition about part and its environment.

b) Defining Element Type and Material Property

For the wishbone suspension system before doing the meshing, first select the element type. Meshing is done using the HYPERMESH software. Element types selection depends on three possibilities. - Geometry size & shape of element, types of analysis and time allotted to project. There are three types of element 1D, 2D&3D. The element type used for crankshaft is solid 3D.In 3D meshing four types of element shapes are available. These are Tetra, Penta, Hex, and Pyramid. For the analysis of crankshaft the tetra element shape is used. Tetra element has four nodes. Solid 3D tetra element has only three translations degree of freedom in the x, y, and z directions. Tetra element has no rotational degree of freedom. A tetrahedral-shaped, prism shaped and a pyramid-shaped element can be formed. In the meshing of wishbone suspension system tetrahedral element is used.

The material property inputs for analysis of wishbone system are given below:

Page 78-90 © MANTECH PUBLICATIONS 2016. All Rights Reserved

- *Young's modulus,*
- *Poisson's ratio and*
- *Density*
- *Yield strength*

a) Defining Mesh Size and Quality Checking

Any continuous object has infinite degrees of freedom and it's just not possible to solve the problem in this format. Finite element method reduces degrees of freedom from infinite to finite with the help of discretization that is Meshing (Nodes and Element).Mesh generation refers to representation of solution domain as an assemblage of small domain whose geometrical properties are known. Mesh size is the element edge length which is required to define earlier to meshing. Large element edge length give coarse mesh, while small element edge length gives fine mesh. Fine element edge length gives accurate results but takes more computation time.

To recognize the effect of element size on the results of solution, mesh size sensitivity analyses are carried out. These analyses are performed iteratively at different element lengths until the solution obtained appropriate accurate. This is also called grid independent stage. Grid independent stage is the mesh element size at which the solution of load steps is independent of mesh size, below this size if mesh size is reduced then the solution does not change with changing mesh size. This stage is achieved by solving the problem at various element sizes for the same load steps and the resulted von misses stresses are observed for various mesh sizes. This is repeated cyclically by refining the element edge size and solving the solution and getting von mises stresses, again refine the mesh and so on. Finally the stage at which the von mises stresses remains unchanged with varying mesh size is achieved. This element edge length size is called grid independent size and this stage is called grid independent stage.

b) Quality Checks Used for Both 2-D and 3-D Elements

These checks apply to both types of elements, but when applied to 3-D elements they are generally applied to each face of the element. The value of the worst face is reported as the 3-D element's overall quality value.

Warp angle:

Ii is define as angle between normal to two planes formed by splitting the quad element along diagonals. Maximum angle out of the two possibilities is reported as warp angle as shown in Fig. 3

Page 78-90 © MANTECH PUBLICATIONS 2016. All Rights Reserved

Ideal value = 0° (acceptable < 10°).

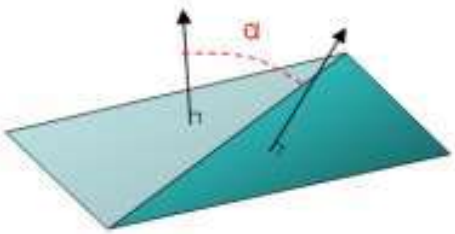

Fig. 3 Warp angle of element

Aspect ratio:

Maximum element edge length to the minimum element edge length.

Ideal value = 1 (Acceptable < 5).

Skew:

Skew for quadrilateral element = 90° - Minimum angle between two lines joining opposite mid-side of the element. Skew for triangular element = 90° - Minimum angle between the lines from each node to the opposing mid-side and between the two adjacent mid-sides at each node of the element as shown in Fig. 4.

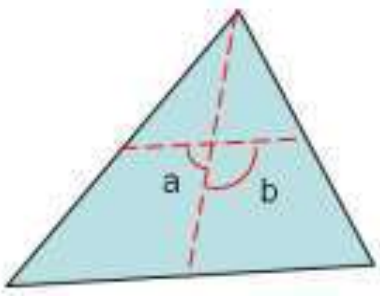

Fig 4. Skew angle on the element.

Jacobian: Jacobian is a scale factor arising because of transformation of co-ordinate system. Elements are transformed from global coordinates to local coordinates (defined at centroid of every element), from faster analysis point of view.

Ideal value = 1.0 (Acceptable > 0.6)

Included angle:

Skew is based on overall shape of element and it does not take in to account individual angle of quadrilateral or triangular element. Included angle check is applied for individual angle.

Quad: Ideal value = 90° (Acceptable = 45° < θ < 135°)

Tria:

Ideal value = 60° (Acceptable = 20° < θ < 120°)

Minimum Length / Size:

The shortest edge (in which the length of the shortest edge of each element is used) and the height to closest node are shown in Fig.5. In the height to closest node method, HYPERMESH calculates the closest (perpendicular) distances to the planes formed by the opposite faces for each corner node [Gokhale et al, 2008].

Page 78-90 © MANTECH PUBLICATIONS 2016. All Rights Reserved

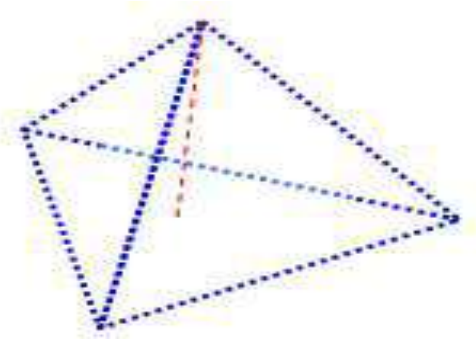

Fig 5. Shortest edge of the element

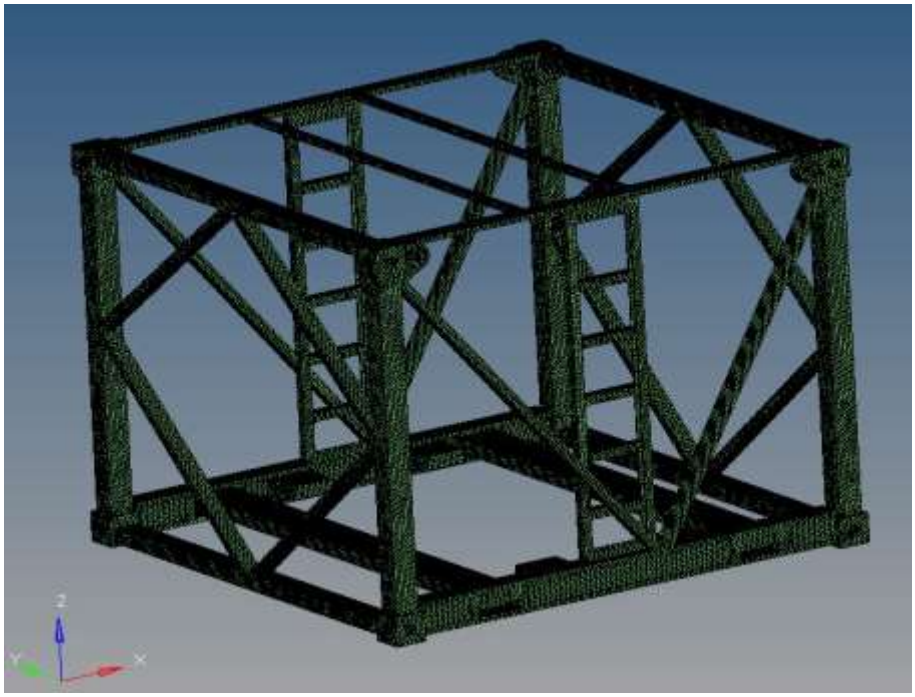

Fig 6. Meshed model of skid frame

The prepared CAD skid frame is imported in Hypermesh and then geometry cleanup is carried out. The smallest fillet or the geometry is considered for defining the meshing size of the element. The skid frame is meshed with 2D tria elements at first to create an enclosure with the elements. Then using the 2D elements 3D tetra elements was created. The tetrahedral elements were checked for mesh quality.

The boundary conditions on the skid frame will be applied on the meshed model. The frame is constrained at the base arresting all degrees of freedom as shown in the below fig.7.

Page 78-90 © MANTECH PUBLICATIONS 2016. All Rights Reserved

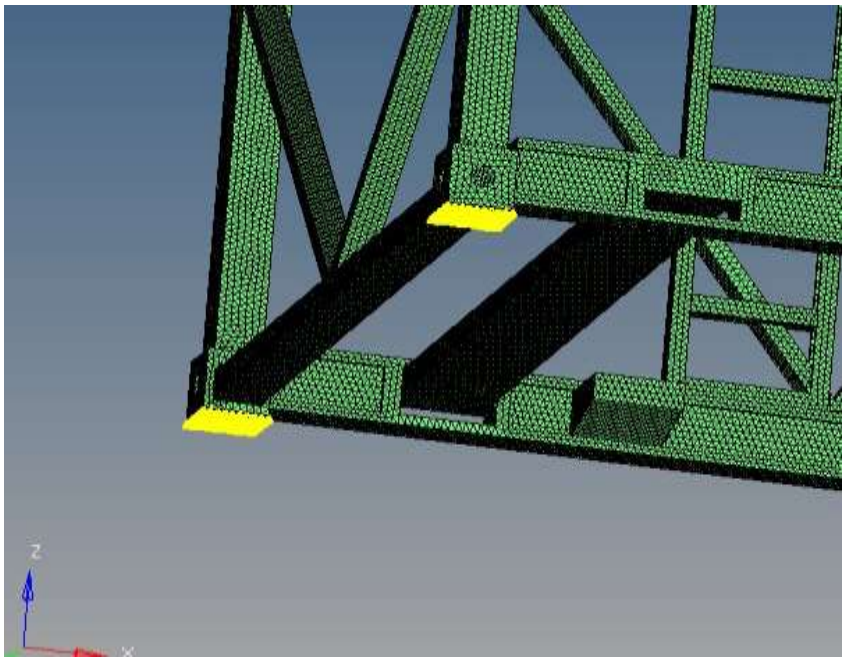

Fig. 7 Frame is constrained at base.

The self weight of the pressure vessel along with the operating pressure is added on the frame on the supporting lugs. These are modeled in Hypermesh with the help of Rigids which transfers the load applied to the elements. The Force applied is as shown in the figure 8.below

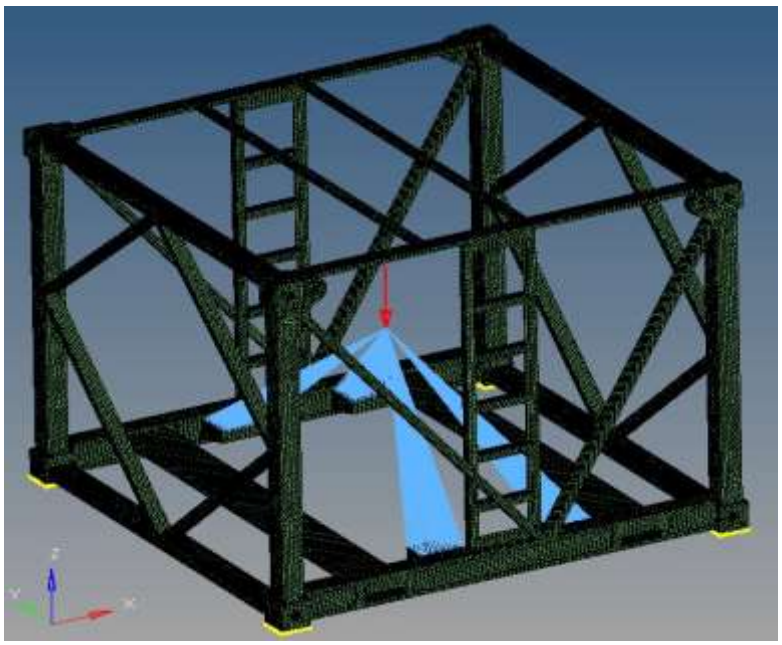

Fig. 8 Force applied at C.G.

Page 78-90 © MANTECH PUBLICATIONS 2016. All Rights Reserved

ANALYSIS

The designed frame was analysed with Finite Element Method. The analysis was required to check the values of induced stresses and deflections caused due to the weights of components.

A Static Analysis

The static analysis is applied when the value of any load acting on frame does not change with time. Generally linear behaving materials are used for manufacturing of frames. Thus the Skid Frame structure was subjected to linear static analysis.

The prepared model is imported in ANSYS for solving and Analysis is carried out. The results for displacement and von mises stresses were plotted.

FABRICATION AND EXPERIMENTATION

The skid frame is fabricated according to the CAD model dimensions. The skid frame is then mounted on the experimental set up. The testing is carried out and the results were recorded.

Here we will get a plot between load and displacement.

VALIDATION

The result from simulation and experimentation will be compared and validate. The difference in values of analysis and testing should not be greater than 10 %, then the design is valid.

OPTIMIZATION

Finally by adding or removing the steel to maximize the stiffness and minimize the cost design will be optimized. Where we will get plot between maximum deflection and iteration number, FOS Vs iteration number, weight Vs iteration number. Weight of the material required for every Iteration no. goes on reducing due to use of thinner/ smaller sections. This shows the gradual reduction in weight of the Skid Frame. With reference to the deflection and FOS results, weight reduction up to Iteration N will be taken as safe.

CONCLUSION

The project will develop an illustration of Skid Frame modelling, analysis for given operating conditions and will optimize its weight, within safety limits.

REFERENCES

1. Prof. Pandhare A. P. *, 2Chaskar S. T., 3Patil J. N., 4Jagtap A. S., 5Bangal P. M ,“Design, Analysis and Optimization of Skid Base Frame” International journal of technology and emerging

Page 78-90 © MANTECH PUBLICATIONS 2016. All Rights Reserved

engineering research, vol 2, issue 7, ISSN 2347-4289

2. Kishor d. jadhav & maneet.r.dhanvijay [2], presented a paper on "Design and Standardization of Base Frame & ant Vibration Mounts for Balanced Opposed Piston Air Compressor", International Journal of Applied Research in Mechanical Engineering (IJARME) ISSN: 2231 –5950, Vol-2, Iss-2, 2012

3. Joshi. A. M. *, 2Jadhav T. A. presented a paper on "Analysis and Testing of Skid of Frame of Compressor". International Journal of Applied Research and Studies (iJARS) ISSN: 2278-9480 Volume 2, Issue 9 (Sep - 2013)

4. Chris Harper P.Eng. & Hongfa Wu, Ph.D., P.Eng. presented a paper on " Optimized Skid Design for Compressor Packages', Gas Machinery Conference 2013 October 7 - 10, 2013 Albuquerque, NM

5. Zhanghai Wang and Daryl Bast, presented a paper on ,"Design By Analysis as an Alternative to CSA B620 and 49CFR Structural Requirements for Pressurized Highway Tanks", ASME Pressure Vessel & Piping Conference 2010

6. HEMIL R. SHAH#1, MR.A.K.SAHU #2 , PROF.M.Y.PATIL*3,"Structural analysis and Design modification of The Compressor Skid of Helium Liquifier Plant" International Journal of Engineering Research & Technology (IJERT) Vol. 2 Issue 11, November – 2013 ISSN: 2278-0181

Page 78-90 © MANTECH PUBLICATIONS 2016. All Rights Reserved