
International Journal of Analytical, Experimental and Finite Element Analysis (IJAEFEA), Issue. 2, Vol. 3, October 2016.
e-ISSN: 2394-5141, p-ISSN: 2394-5133, pp 25-30

# Static Structural Analysis of the Crane Hook using Ansys

**Vishal A. Rane**[1]
ranevishala@gmail.com

**Ganesh V. Patil**[2]
ganeshvpatil29@gmail.com

[1,2]Assistant Professor
Department of Mechanical Engineering
D .Y. Patil School of Engineering and Technology,
Lohgaon, Pune, India

*Abstract*— **Material handling equipments are an eminent part of the human life. Cranes are amongst one of the material handling equipment which finds wide applications in different fields of engineering. The present work is an effort to cover complete design and analysis of industrial crane (EOT) of 63T/20T capacity. In this work the stress analysis of four major loads bearing components; lifting hook, trunnion, pulley supporting plate and main girder is carried out. The finite element analysis has been carried out for the rated load condition with some impact factor. Maximum stress and deformation locations were obtained for each of the components to check the validity of design values. The design and analysis results were validated by comparing with literature & calculation results.**

*Index terms:* Industrial crane (EOT), design, analysis. element.

## I. INTRODUCTION

Material handling equipment is employed for moving loads in premises or areas, department, factories and plants, at construction sites, point of storage and reloading, etc.

As distinct from the so-called long distance transport (railway, automobile, water, air) moves load over a considerable distance, material handling equipment moves loads over comparatively short distances. In practice these distance are usually confined to ten or hundreds of meters and reach thousands of meters only occasionally to ensure a constant load transfer between two or several points connected by common production activities.

Handling and loading operations in each enterprise depend on the available external and interplant facilities. External transporting facilities supply the enterprise with raw materials, semi finished items, fuel, auxiliary materials, etc. And the interplant transporting facilities distributes the load, which come in, throughout the enterprise, move material between processing unit directly engaged in production and bring finished products and waste to loading points to be loaded and shipped from the enterprise by external transporting facilities.

The transporting facilities insides the plant is in turn subdivided into inter department and intradepartmental facilities. Interdepartmental transporting facilities move loads between department, for example, at a mechanical engineering plant, between blank manufacturing, machining and auxiliary departments, as well as between department and warehouses, loading and unloading points, etc. Intradepartmental transporting facilities move loads between sections, stores, machines, etc. within the limits of one department.

### A. *Type of Material Handling Equipments*

Mainly there are three type of material handling equipments, categorized as-

1. Hoisting Equipment
2. Conveying Equipment
3. Surface and Overhead Equipment

a) Hoisting Equipment

It is a group of machine with lifting gear intended for moving loads mainly in batches. It is intended mainly for unit loads - various parts of machines and whole machines, elements of metal structure, hopper and ladles, girders, building blocks and materials, etc.

Type of Hoisting Equipments:

1. Hoisting Machinery
2. Cranes
3. Elevators

b) Conveying Equipments

It is a group of machine which may have no lifting gear and which moves loads in a continuous flow. It includes all types of Conveyor.

c) Surface Overhead Equipments

It is a group of machines which also may not be provided with lifting gear and usually handle loads in batches. Conveying equipment can be used to handle either only bulk or only unit loads while surface or overhead facilities can be used to handle both bulk and unit loads. Materials handled in bulk are composed of a large number of homogeneous particles or lumps, for example: coal, ore, cement, sand, clay.

## II. EOT CRANES

### *A. Type of EOT Cranes*

- Single Girder EOT Crane
- Double Girder EOT Crane
- Under Slung Crane

a) Single Girder Overhead Traveling Crane: (Normally up to 10 tones capacity)

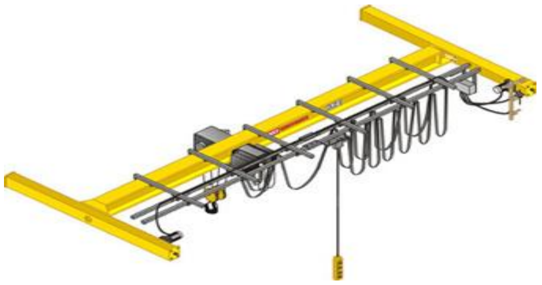

Fig. 1. Single girder overhead traveling crane

b) Double Girder Overhead Traveling Crane: (Normally above 10 tones capacity)

When heavy load and wide span are required double girder overhead traveling Crane are generally used. They consist of two torsion free box girder. This makes them especially suitable for lifting and transporting load over 10 tones and for span of more than 25 meter.

Fig. 2. Double girder overhead traveling crane

c) Under Slung Crane: (Normally up to 10 tones capacity)

It is a special type of crane and provides an optimal solution where the building structure makes the normal traveling cranes less suitable. The main feature is that the crane track is not fastened to pillars but to the beams of the building.

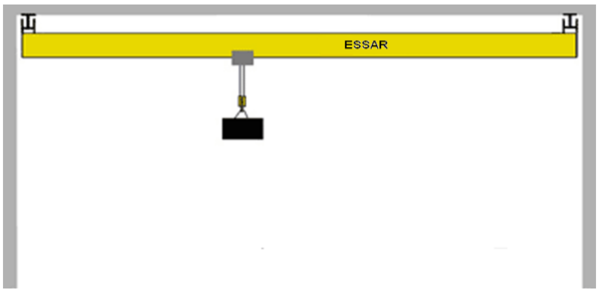


Fig. 3. Under slung crane

## III. Literature Review

Zuberi Rehan H, Long Kai, Zuo Zhengxing, [1]: Authors have presented paper on “Design Optimization of EOT Crane Bridge”. This paper demonstrates design optimization of EOT crane thin walled welded box girder subjected to rolling loads. A simple and innovative procedure has been introduced to use Generalized Reduced Gradient (GRG2) nonlinear optimization code for optimization of various parameters of the welded box section bridge and then comparing the results with the FE simulation. Local buckling of web and compression flange has also been taken in account while performing GRG2 optimization. Later vigorous random and 1st order Design Optimization is performed to establish design space and convergent solutions utilizing commercially available FEA software.

Alkin C., Imrak C. E, Kocabas H., [2]: Authors have presented paper “The design of an overhead crane bridge with a double box girder” in which case study of a crane with 35 tonne capacity and 13 m span length has been conducted. In the initial phase of the case study, conventional design calculations proposed by F. E.M Rules and DIN standards were performed to verify the stress and deflection levels. The crane design was modeled using both solids and surfaces. Finite element meshes with 4-node tetrahedral and 4-node quadrilateral shell elements were generated from the solid and shell models, respectively. After a comparison of the finite element analyses, the conventional calculations and performance of the existing crane, the analysis with quadratic shell elements was found to give the most realistic results. As a result of this study, a design optimization method for an overhead crane is proposed.

Gerdemeli Ismail, Kurt Serpil, Yildirım Metin, [3]: The paper “Calculations, Modeling and Analysis with Finite Element Method of rubber tyred container stacking Crane” presents the modeling, calculations and analysis using Finite Element Method (FEM) of the rubber tyred container stacking crane in ports. All elements of the rubber tyred container stacking crane was modeled and made its calculations. Although stress and deformation analysis of crane bridge girder and buckling analysis of the crane legs are performed. ANSYS Workbench program has been used to perform the finite element method. In addition, rubber tyred container stacking crane has been modeled by using Autodesk INVENTOR 2010 program. Stress, deformation and buckling analysis have been compared with calculations. The aim of this work is to consider the new possibilities and the gains of finite element method over conventional calculation methods on rubber tyred container stacking crane design. When we examine the results, deformation formed on crane system is not significant when considering geometric dimensions of model and it was observed that the stress values remain under the yield strength of the steel which is used for crane bridge and legs.

Maharana Pradyumna keshari, [4]: The project “Computer aided analysis and design of hoisting mechanism of an EOT crane” has been carried out to overall design of the hoisting mechanism of an EOT crane. The dimensions of the main components have been determined for a load capacity of 50 ton crane having 8 rope falls. Various dimensions for cross sections of various shapes for crane hook have been found. After the system was designed, the stress and deflection are calculated at critical points using ANSYS and optimized. Which cross section would be better keeping some parameters constant for all the case. Various dimensions and load per wire for wire ropes has been found. Using various formulae found the dimensions for pulley, Rope-drum. Also calculated the Power and ratings for the motor, brakes used in the hoist mechanism.

Camelia Bretotean Pinca, Gelu Ovidiu Tirian, Ana Josan, [5]: Authors have presented a paper on "Finite element analysis of an overhead crane bridge". This paperwork analyzes the tension and deformation estate of the resistance structure of an overhead crane bridge that is used for all the processes performed in the hall of a continuous casting department of an iron and steel plant in order to find out the best sizes. This analysis is made up with the help of the COSMOS software who enables us to make evolved finite items – shell-type with 3 or 4 nodes per element. The shell-type finite items belong to the C1–class items, and they have a field variable and the I-type derivates working continuously alongside the frontier, meanwhile the II-type continuous derivates per item are not continuous alongside the frontier. These finite items allow us to design some complex structures more accurately, such as the resistance structure of the crane bridge. This example is not intended to be the only solution when designing similar structures, but the authors of this paperwork believe that it is providing enough information and useful solutions for the analysis of the tension and deformation state in case of finite items.

Jat Hareshwar .R., [6]: The project "design &analysis of 150T EOT crane" has been done in ESSAR STEEL, Hazira, Surat. The project includes complete design of EOT crane for 150T capacity. It includes designing of hook, crossbar, sheave, rope drum, main girder etc. & further analysis of designed components using ANSYS 13.0.

The project came with following analysis results with confirming safety components for critical loading conditions i.e. Maximum Stress & deformation for crossbar is 230.31N/mm2 & 0.0885mm resp. and for pulley supporting plate is 478N/mm2 & 0.34696mm resp.

Karmakar R., Mukharjee A., [7]: Authors have presented a paper on normal operating condition "Electric Overhead Cranes are subjected to sever dynamic loading". Economical design of such crane calls for more realistic study of Dynamic behavior crane operation. This paper presents a bond graph simulation of Electric Overhead Crane dynamic for three crane critical operations namely load hoisting, breaking load lowering and carriage motion.

## IV. Finite Element Analysis

Finite element analysis (FEA) is a method for numerical solution of field problems. Engineering structures that have complex geometry and loads, are either very difficult to analyze or have no theoretical solution. However in FEA, a structure of his type can be easily analyzed. Commercial FEA programs, written so that a user can solve complex engineer problems without knowing the governing equation or the mathematics; the user is required only to know geometry of the structure and its boundary conditions. FEA software provides complete solution including deflections, stresses, strains, reactions, etc. In order to become a skillful FEA user, a thorough understanding of techniques for modeling a structure, the boundary conditions and the limitations of the procedure, are very crucial. Engineering structures, e.g., bridge, aircraft wing, high-rise buildings, etc., are examples of complex structure that are extremely difficult to analyze by classical theory. But FEA technique facilitates an easier and a more accurate analysis. In this technique the structure is divided into very small but finite size elements. Individual behavior of these elements is known and based on this knowledge; behavior of the entire structure is determined.

*1.1.1. Applications of Finite Element Analysis:*

(a) **Structural Analysis** consists of linear and non-linear models. Linear models use simple parameters and assume that the material is not plastically deformed. Non-linear models consist of stressing the material past its elastic capabilities. The stresses in the material then vary with the amount of deformation. Examples are stress analysis including truss, frame, beam analysis, stress concentration problems typically associated with holes, fillets or other changes in geometry in a body, buckling analysis of connecting rod subjected to axial compression etc.

(b) **Vibration Analysis** is used to test a material against random vibrations, shock and impact. Each of these incidences may act on the natural vibration frequency of the material which in turn may causes resonance and subsequent failure. Examples are a beam subjected to different types of loading, analysis of machine components, modal analysis of pressure vessels etc.

(c) **Fatigue Analysis** helps designers to predict the life of a material or structure by showing the effects of cyclic loading on the specimen. Such analysis can show the areas where crack propagation is most likely to occur. Failure due to fatigue may also show the damage tolerance of the material. Example is fatigue analysis of crankshaft and connecting rod of I.C engines.

(d) **Thermal Analysis** models the conductivity or thermal fluid dynamics of the material or structure. This may consist of a steady-state or transient transfer. Steady-state transfer refers to constant thermo properties in the material that yield linear heat diffusion. Examples are steady state thermal analysis on composite cylinder, transient analysis of cantilever beam etc.

*A) Loading and Boundary Conditions in FEA*

**a) Loading:** Loads and supports are thought of in terms of the degrees of freedom (DOF) available for the elements used. In solids the DOF are x, y and z translations (for shells we add rotational DOF rotx, roty and rotz). Supports, regardless of actual names, are always defined in terms of DOF.

In this present project only force loading is involve

**Force loading:**

– Forces can be applied on vertices, edges, or surfaces.
– The force will be evenly distributed on all entities.
– Force can be defined via vector or component methods.

**b) Boundary conditions:**

Boundary conditions (BC) are the way that a specific node in a FEA model is attached to the ground or some other node in the model. A variety of BC's are available including: rigid or fixed, elastic spring, thermal, etc. BC's can be specified to be fixed in any or all of the six DOF. BC's define how a part or assembly of parts is attached to the real world and the loads and constraints that represent the effect of the surrounding environment on the model

A boundary condition for the model is the setting of a known value for a displacement or an associated load. For a particular node you can set either the load or the displacement but not both. The simplest essential boundary conditions are support and symmetry conditions. These appear in many practical problems. More exotic types, such as multi freedom constraints, require more advanced mathematical tools

Following are the main Boundary conditions that has been used in present project for the static structural analysis of the components

**Fixed Support:**

In this type support all 6 Degree of freedom (DOF) is restricted, means it does not allow moving or translating any member i.e. it restricts 3 translational motions or displacements and 3 rotational motions.

**Cylindrical Support:**

This is support where 4 Degree of freedom has restricted, means it does not allow displacing body in 2-direction and 2 rotations i.e. it allows body do displace along axis of the body member and allows rotation about the cylindrical axis.

*B)Meshing*

In the present static structural analysis for meshing of object solid 10 Node tetrahedral (Quadrilateral) element is used because,

When geometries are complex or the range of length scales of the flow is large, a triangular/tetrahedral mesh can be created with far fewer cells than the equivalent mesh consisting of quadrilateral/hexahedral elements. This is because a triangular/tetrahedral mesh allows clustering of cells in selected regions of the flow domain. Structured quadrilateral/hexahedral meshes will generally force cells to be placed in regions where they are not needed. Unstructured quadrilateral/hexahedral meshes offer many of the advantages of triangular/tetrahedral meshes for moderately-complex geometries.

A characteristic of quadrilateral/hexahedral elements that might make them more economical in some situations is that they permit a much larger aspect ratio than triangular/tetrahedral cells. A large aspect ratio in a triangular/tetrahedral cell will invariably affect the skewness of the cell, which is undesirable as it may impede accuracy and convergence. Therefore, if you have a relatively simple geometry in which the flow conforms well to the shape of the geometry, such as a long thin duct, use a mesh of high aspect ratio quadrilateral /hexahedral cells. The mesh is likely to have far fewer cells than if you use triangular/tetrahedral cells.

Converting the entire domain of your (tetrahedral) mesh to a polyhedral mesh will result in a lower cell count than your original mesh. Although the result is a coarser mesh, convergence will generally be faster, possibly saving you some expense.

*C) Theories of Failure*

When a component is subject to increasing loads it eventually fails. It is comparatively easy to determine the point of failure of a component subject to a single tensile force. The strength data on the material identifies this strength. However when the material is subject to a number of loads in different directions some of which are tensile and some of which are shear, then the determination of the point of failure is more complicated. Metals can be broadly separated into DUCTILE metals and BRITTLE metals. Examples of ductile metals include mild steel, copper etc. Cast iron is a typical brittle metal.

Ductile metals under high stress levels initially deform plastically at a definite yield point or progressively yield. At failure a ductile metal will have experienced a significant degree of elongation. Brittle metals experience little ultimate elongation prior to failure and failure is generally sudden. A ductile metal is considered to have failed when it has suffered elastic failure that is when a marked plastic deformation has begun. A number of theories of elastic failure are recognized including the following:

- Maximum principal stress theory (for brittle metals )
- Maximum shear stress theory (for ductile metals)
- Maximum shear strain energy theory(for ductile metals)
- Maximum principal strain theory(for brittle metals )
- Maximum strain energy theory (for brittle metals ).

## V. Analysis Of Crane Hook

Material used for Hook: **Forged steel**

TABLE I
MECHANICAL PROPERTIES OF FORGED STEEL

| Young's modulus | 210GPa | Tensile ultimate strength | 560 MPa |
|---|---|---|---|
| Tensile yield strength | 460MPa | Density | 7850 $Kg/m^3$ |
| Poisson's ratio | 0.285 | | |

For meshing of Hook model **10 Node Tetrahedral** is used, following is the graphical representation of same element

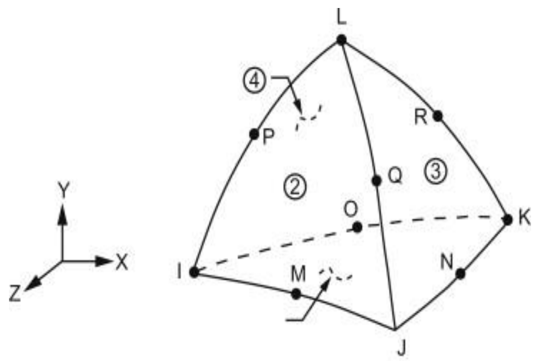


Fig 4. Solid 10 Node Tetrahedral element

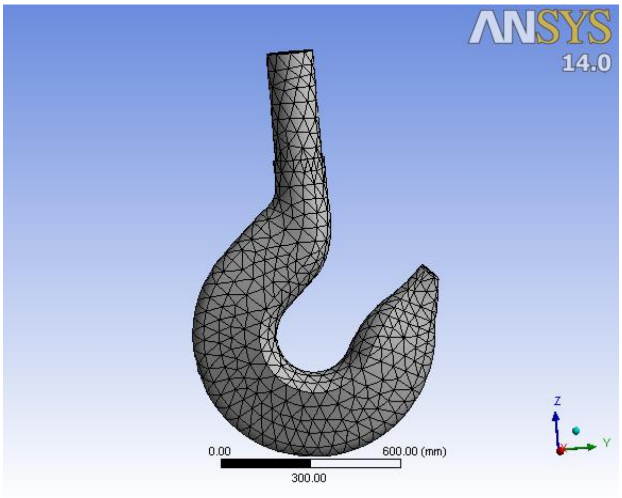


Fig. 5 Meshing of hook model

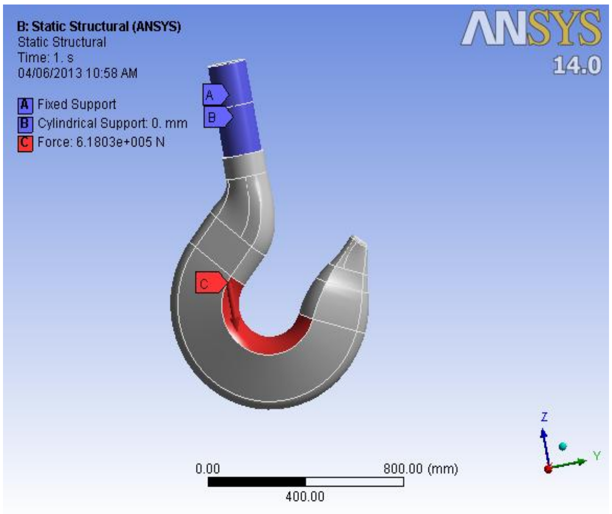


Fig. 6. Loading & boundary conditions of hook

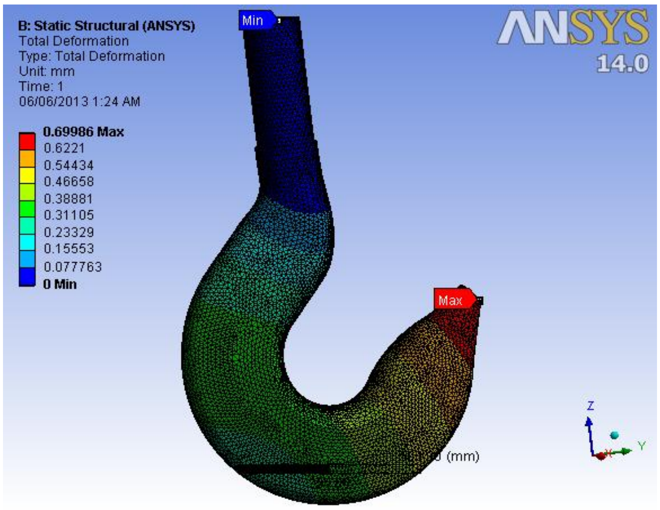


Fig. 7. Deformation analysis of hook

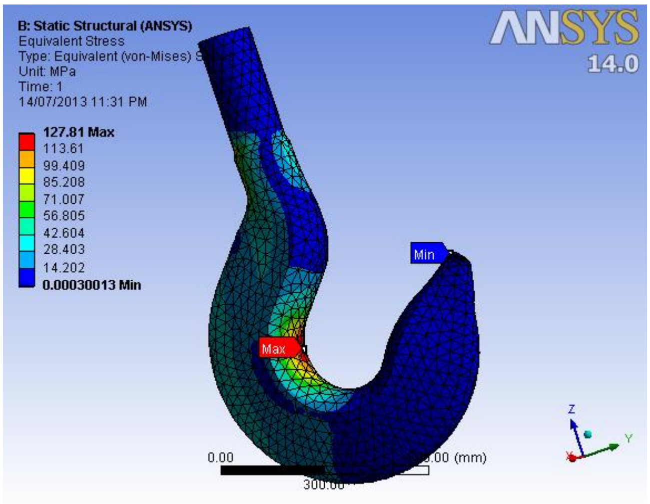


Fig. 8. Stresses of Hook

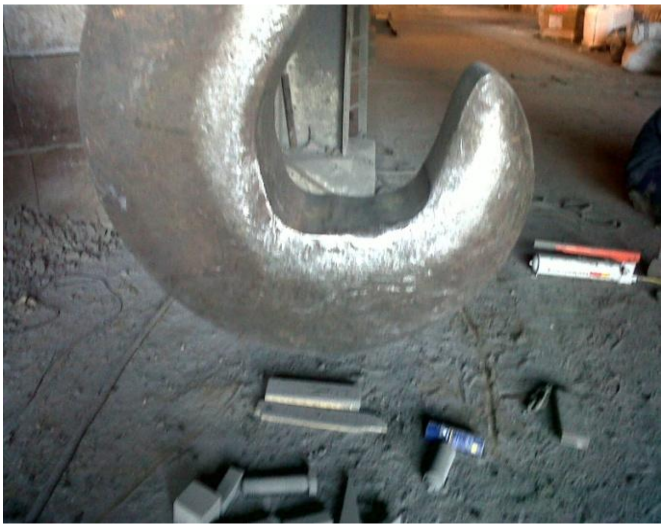

Fig. 9. wearing of Hook

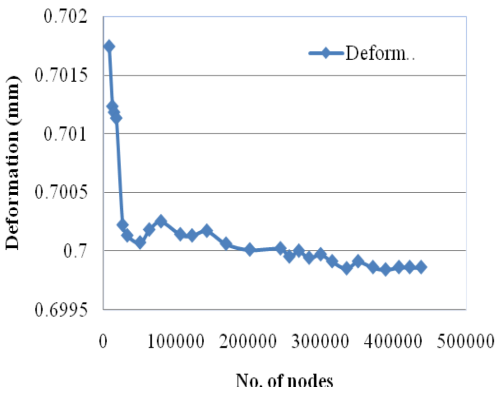


Fig. 10. Deformation convergence of hook

- Finite element mesh was generated using solid tetrahedral elements with various element sizes; the deformation was checked for convergence, Von Mises stress cannot be checked for convergence because it varies sinusoidal way. Maximum stress is selected as stress value of that component.
- Fig 8 above shown is deformation versus no. of nodes uses for convergence checking. Thus element size was found to be 12.30 for working in convergence zone
- The ANSYS analysis results of Hook obtained as follows,
  Deformation = 0.69986mm
  The Maximum Von Mises Stress = 125.33Mpa

## VI. Analysis Result And Discussion

First model of Hook is meshed with 10 node tetrahedral solid elements in ANSYS workbench, then for the required boundary conditions & applied load 63T to hook model. Maximum & minimum equivalent Von Mises stress & deformation values noted shown.

These ANSYS results compare with literature results for confirming the design is safe & correct.

TABLE II
COMPARISON OF STRUCTURAL ANALYSIS RESULTS OF HOOK WITH LITERATURE SURVEY RESULTS

| | ANSYS Analysis Results | Literature Survey Results | Analytical Results |
|---|---|---|---|
| Maximum Von Mises Stress | 125.33 $N/mm^2$ | 150.72 $N/mm^2$ | 122.07 $N/mm^2$ |
| Maximum Deformation | 0.69986 mm | 2.05 mm | - |

Error in the stress results = 3%

The comparison of the literature results and the analysis for the structural analysis (critical load) is as shown in Table 1 For the structural analysis, results can be taken into consideration, because there is no big difference between literature and analysis.

## VI. Conclusion

In this work manual crane hook 3-D modeling has been done using CATIA V5. Finite element analysis of the part has been done using ANSYS 14.

Maximum stress and deformation location were obtained for each of the components to check the validity of design values. For comparing the stress and deformation analysis with the literature survey ,examining the results there is no significant difference between the analysis and calculation result for the stresses and deformations of hook.. i.e. the error is near to 3%. Therefore analysis result can be taken into consideration.

## References

[1] Zuberi Rehan H, Kai Long, Zuo Zhengxing, "Design Optimization of EOT Crane Bridge", *International Conference on Engineering Optimizationm,* Rio de Janeiro, Brazil, 2008, pp. 1-9
[2] Alkin C., Imarak C. E, Kocabus H., "Solid Modeling and Finite Element Analysis of an Overhead Crane Bridge". *Acta Polytechnica Vol. 45 No. 3* ,2005, pp. 61- 67.
[3] Gerdemeli Ismail, Kurt Serpil, Yildırım Metin,**"**Calculations, modeling and analysis with finite element method of rubber tyred container stacking", *Trends in the Development of Machinery and Associated Technology*. Mediterranean, Cruise, 11-18 September 2010, pp. 545-548
[4] Maharana Pradyumna keshari**, "**Computer aided analysis and design of hoisting mechanism of an EOT crane", NIT, Rourkela, 2012.
[5] Camelia Bretotean Pinca, G. O, "Finite element analysis of an overhead crane bridge". *2nd WSEAS Int. Conf. on Finite Differences, Finite Elements, Finite Volumes, Boundary Elements*, Timişoara: "Politechnica" University, Romania, 2005, pp. 51-56
[6] Jat, H. R., *"Design & analysis of 150T capacity EOT crane".* ESSAR Steel, Surat: 2012.
[7] Karmakar R., Mukharjee A., "Electric overhead cranes subjected to sever dynamic loading",1989, pp-56-63.
[8] Duggal, S. K., *"Design of Steel Structure"*, Tata McGraw Hill, NewDelhi, 1960
[9] Uddanwadiker, Rashmi, "Stress Analysis of Crane Hook and Validation by Photo-Elasticity", review from july27-aug12, *Scientific research*,2005, pp.935-941.
[10] IS 3177 and IS 4137, "Codes of practice for electric overhead travelling cranes and Gantry Crane other than steel work cranes (Second revision)", 2006.
[11] Shigley J.E., Mischke C.R., *"Mechanical Engineering Design"*, McGraw Hill, Singapore, 1989.
[12] Oguamanam, D. and Hansen, J. S., "Dynamics Response of an Overhead Crane System"; *Journal of Sound and Vibration*, Vol. 213(5); 1998, pp 889-906.