

Design and Optimization of Crane Hook Using Structural Analysis at Lifting Heavy Load

K Imran¹, A. Vidyasagar²

¹P.G. Scholar, ²Guide, Head Of The Department

^{1,2} BRANCH : MACHINE DESIGN

^{1,2} Geethanjali College Of Engineering & Technology

Email : ¹kimranshahid477@gmail.com, ²sagaramadala1410@gmail.com

Abstract:

To transfer the heavy objects is very difficult. By using crank hook it is easy to carry the heavy loads. Crank hook is a mechanical component used to carry heavy loads and machinery elements to transfer from one place to another place. Heavy crane hooks are used in ship yards. Steel, carbon and high carbon steel are the materials used for manufacturing of crane hooks.

Crane hook are used to elevate the heavy loads, machineries, material in industries. A crane is a machine with a hoist, wire ropes or chains and sheaves used to lift and move material. When crane hook is subjected to heavy loading condition, it may causes to produce ruptures along center of curvature of hook. This problem is yields to loss in business. However this problem will solve by reducing the ruptures in crane hook. This reduction is takes place when redesign of hook is established only. Previous works explained only static analysis of crane hook using different materials. Among these works, this project explained that reduction of ruptures in crane hook by application of redesign.

The main aim of this project is to reduce the ruptures in crane hook by redesigning of crane hook. Design of crane hook is done by using Unigraphics software and structural analysis results was estimated by using Ansys software. From analysis results rupture region on hook is found at loading

conditions and redesign the region of rupture in hook in Unigraphics software for avoiding ruptures and again analysis is done for redesigned hook and continuously doing this process up to removing ruptures in hook.

INTRODUCTION

1.1 Introduction to crane hook and lifting machinery:

To transfer the heavy objects is very difficult. By using crank hook it is easy to carry the heavy loads. Crank hook is a mechanical component used to carry heavy loads and machinery elements to transfer from one place to another place. Heavy crane hooks are used in ship yards.

Classification of equipment used for lifting loads with lifting machinery:

1. Steel wire rope sliding
2. Chain sliding
3. Synthetic round slings and lifting belts
4. Lifting beam
5. Lifting eye bolts
6. C-hook
7. Container spreader beams
8. Lifting forks
9. Forks attached to a lift truck

1. Steel wire rope sliding:

History: Modern wire rope was invented by the German mining engineer Wilhelm Albert in the years between 1831 and 1834 for use in mining in the Harz Mountains in Clausthal, Lower Saxony, Germany. It was quickly accepted because it proved superior

to ropes made of hemp or to metal chains, such as had been used before Wilhelm Albert's first ropes consisted of three strands consisting of four wires each.

In 1840, Scotsman Robert Stirling Newall improved the further.

In the last half of the 19th century, wire rope systems were used as a means of transmitting mechanical power including for the new cable cars. Wire rope systems cost one-tenth as much and had lower friction losses than line shafts. Because of these advantages, wire rope systems were used to transmit power for a distance of a few miles or kilometers.

In America wire rope was later manufactured by John A. Roebling, forming the basis for his success in suspension bridge building. Roebling introduced a number of innovations in the design, materials and manufacture of wire rope.

Wire rope construction:

Wire: Steel wires for wire ropes are normally made of non-alloy carbon steel with a carbon content of 0.4 to 0.95%. The very high strength of the rope wires enables wire ropes to support large tensile forces and to run over sheaves with relatively small diameters.



Standards:

In the so-called cross lay strands, the wires of the different layers cross each other. In the mostly used parallel lay strands, the lay length of all the wire layers is equal and the wires of any two superimposed layers are

parallel, resulting in linear contact. The wire of the outer layer is supported by two wires of the inner layer. These wires are neighbors along the whole length of the strand. Parallel lay strands are made in one operation. The endurance of wire ropes with this kind of strand is always much greater than of those (seldom used) with cross lay strands. Parallel lay strands with two wire layers have the construction Filler, Seale or Warrington.

Spiral ropes:

In principle, spiral ropes are round strands as they have an assembly of layers of wires laid helically over a centre with at least one layer of wires being laid in the opposite direction to that of the outer layer. Spiral ropes can be dimensioned in such a way that they are non-rotating which means that under tension the rope torque is nearly zero. The open spiral rope consists only of round wires. The half-locked coil rope and the full-locked coil rope always have a centre made of round wires. The locked coil ropes from (invented by Latch and Batchelor Ltd) and from Fatzer AG has one or more outer layers of profile wires. They have the advantage that their construction prevents the penetration of dirt and water to a greater extent and it also protects them from loss of lubricant. In addition, they have one further very important advantage as the ends of a broken outer wire cannot leave the rope if it has the proper dimensions.

Stranded ropes:

Stranded ropes are an assembly of several strands laid helically in one or more layers around a core. This core can be one of three types. The first is a fiber core, made up of synthetic material. Fiber cores are the most flexible and elastic, but have the downside of getting crushed easily. The second type, wire strand core, is made up of one additional strand of wire, and is typically used for suspension. The third type is independent wire rope core (IWRC), which

is the most durable in all types of environments. Most types of stranded ropes only have one strand layer over the core (fiber core or steel core). The lay direction of the strands in the rope can be right (symbol Z) or left (symbol S) and the lay direction of the wires can be right (symbol z) or left (symbol s). This kind of rope is called ordinary lay rope if the lay direction of the wires in the outer strands is in the opposite direction to the lay of the outer strands themselves. If both the wires in the outer strands and the outer strands themselves have the same lay direction, the rope is called a Lang lay rope (formerly Albert's lay or Lang's lay). Multi-strand ropes are all more or less resistant to rotation and have at least two layers of strands laid helically around a centre. The direction of the outer strands is opposite to that of the underlying strand layers. Ropes with three strand layers can be nearly non-rotating. Ropes with two strand layers are mostly only low-rotating.

1.2 UNIGRAPHICS INTRODUCTION

Overview of Solid Modeling

The Unigraphics NX Modeling application provides a solid modeling system to enable rapid conceptual design. Engineers can incorporate their requirements and design restrictions by defining mathematical relationships between different parts of the design.

Design engineers can quickly perform conceptual and detailed designs using the Modeling feature and constraint based solid modeler. They can create and edit complex, realistic, solid models interactively, and with far less effort than more traditional wire frame and solid based systems. Feature Based solid modeling and editing capabilities allow designers to change and update solid bodies by directly editing the dimensions of a solid feature

and/or by using other geometric editing and construction techniques.

Advantages of Solid Modeling

Solid Modeling raises the level of expression so that designs can be defined in terms of engineering features, rather than lower-level CAD geometry. Features are parametrically defined for dimension-driven editing based on size and position.

Features

- Powerful built-in engineering-oriented form features-slots, holes, pads, bosses, pockets-capture design intent and increase productivity
- Patterns of feature instances-rectangular and circular arrays-with displacement of individual features; all features in the pattern are associated with the master feature

Blending and Chamfering

- zero radius
- Ability to chamfer any edge
- Cliff-edge blends for designs that cannot accommodate complete blend radius but still require blends

Advanced Modeling Operations

- Profiles can be swept, extruded or revolved to form solids
- Extremely powerful hollow body command turns solids into thin-walled designs in seconds; inner wall topology will differ from the outer wall, if necessary
- Fixed and variable radius blends may overlap surrounding faces and extend to a Tapering for modeling manufactured near-net shape parts
- User-defined features for common design elements (Unigraphics NX/User-

Defined Features is required to define them in advance

General Operation

- **Start with a Sketch**

Use the Sketcher to freehand a sketch, and dimension an "outline" of Curves. You can then sweep the sketch using Extruded Body or Revolved Body to create a solid or sheet body. You can later refine the sketch to precisely represent the object of interest by editing the dimensions and by creating relationships between geometric objects. Editing a dimension of the sketch not only modifies the geometry of the sketch, but also the body created from the sketch.

- **Creating and Editing Features**

Feature Modeling lets you create features such as holes, slots and grooves on a model. You can then directly edit the dimensions of the feature and locate the feature by dimensions. For example, a Hole is defined by its diameter and length. You can directly edit all of these parameters by entering new values. You can create solid bodies of any desired design that can later be defined as a form feature using User Defined Features. This lets you create your own custom library of form features.

- **Associatively**

Associatively is a term that is used to indicate geometric relationships between individual portions of a model. These relationships are established as the designer uses various functions for model creation. In an associative model, constraints and relationships are captured automatically as the model is developed. For example, in an associative model, a through hole is associated with the faces that the hole

penetrates. If the model is later changed so that one or both of those faces moves, the hole updates automatically due to its association with the faces. See Introduction to Feature Modeling for additional details.

- **Positioning a Feature**

Within Modeling, you can position a feature relative to the geometry on your model using Positioning Methods, where you position dimensions. The feature is then associated with that geometry and will maintain those associations whenever you edit the model. You can also edit the position of the feature by changing the values of the positioning dimensions.

- **Reference Features**

You can create reference features, such as Datum Planes, Datum Axes and Datum CSYS, which you can use as reference geometry when needed, or as construction devices for other features. Any feature created using a reference feature is associated to that reference feature and retains that association during edits to the model. You can use a datum plane as a reference plane in constructing sketches, creating features, and positioning features. You can use a datum axis to create datum planes, to place items concentrically, or to create radial patterns.

- **Expressions**

The Expressions tool lets you incorporate your requirements and design restrictions by defining mathematical relationships between different parts of the design. For example, you can define the height of a boss as three times its diameter, so that when the diameter changes, the height changes also.

- **Boolean Operations**

Modeling provides the following Boolean Operations: Unite, Subtract, and Intersect. Unite combines bodies, for example, uniting two rectangular blocks to form a T-shaped solid body. Subtract removes one body from another, for example, removing a cylinder from a block to form a hole. Intersect creates a solid body from material shared by two solid bodies. These operations can also be used with free form features called sheets.

- **Undo**

You can return a design to a previous state any number of times using the Undo function. You do not have to take a great deal of time making sure each operation is absolutely correct, because a mistake can be easily undone. This freedom to easily change the model lets you cease worrying about getting it wrong, and frees you to explore more possibilities to get it right.

- **Additional Capabilities**

Other Unigraphics NX applications can operate directly on solid objects created within Modeling without any translation of the solid body. For example, you can perform drafting, engineering analysis, and NC machining functions by accessing the appropriate application. Using Modeling, you can design a complete, unambiguous, three dimensional model to describe an object. You can extract a wide range of physical properties from the solid bodies, including mass properties. Shading and hidden line capabilities help you visualize complex assemblies. You can identify interferences automatically, eliminating the need to attempt to do so manually. Hidden edge views can later be generated and placed on drawings. Fully associative

dimensioned drawings can be created from solid models using the appropriate options of the Drafting application. If the solid model is edited later, the drawing and dimensions are updated automatically.

- **Parent/Child Relationships**

If a feature depends on another object for its existence, it is a *child* or *dependent* of that object. The object, in turn, is a *parent* of its child feature. For example, if a HOLLOW (1) is created in a BLOCK (0), the block is the parent and the hollow is its child. A parent can have more than one child, and a child can have more than one parent. A feature that is a child can also be a parent of other features. To see all of the parent-child relationships between the features in your work part, open the Part Navigator.

Creating A Solid Model

Modeling provides the design engineer with intuitive and comfortable modeling techniques such as sketching, feature based modeling, and dimension driven editing. An excellent way to begin a design concept is with a sketch. When you use a sketch, a rough idea of the part becomes represented and constrained, based on the fit and function requirements of your design. In this way, your design intent is captured. This ensures that when the design is passed down to the next level of engineering, the basic requirements are not lost when the design is edited.

The strategy you use to create and edit your model to form the desired object depends on the form and complexity of the object. You will likely use several different methods during a work session. The next several figures illustrate one example of the design process, starting with a sketch and ending with a finished model. First, you can

create a sketch "outline" of curves. Then you can sweep or rotate these curves to create a complex portion of your design.

Introduction to Drafting

The Drafting application is designed to allow you to create and maintain a variety of drawings made from models generated from within the Modeling application. Drawings created in the Drafting application are fully associative to the model. Any changes made to the model are automatically reflected in the drawing. This associativity allows you to make as many model changes as you wish. Besides the powerful associativity functionality, Drafting contains many other useful features including the following:

- An intuitive, easy to use, graphical user interface. This allows you to create drawings quickly and easily.
- A drawing board paradigm in which you work "on a drawing." This approach is similar to the way a drafter would work on a drawing board. This method greatly increases productivity.
- Support of new assembly architecture and concurrent engineering. This allows the drafter to make drawings at the same time as the designer works on the model.
- The capability to create fully associative cross-sectional views with automatic hidden line rendering and crosshatching.
- Automatic orthographic view alignment. This allows you to quickly place views on a drawing, without having to consider their alignment.
- Automatic hidden line rendering of drawing views.
- The ability to edit most drafting objects (e.g., dimensions, symbols, etc.) from the graphics window. This allows you to create drafting objects and make changes to them immediately.

- On-screen feedback during the drafting process to reduce rework and editing.
- User controls for drawing updates, which enhance user productivity.

Finally, you can add form features, such as chamfers, holes, slots, or even user defined features to complete the object.

• Updating Models

A model can be updated either automatically or manually. Automatic updates are performed only on those features affected by an appropriate change (an edit operation or the creation of certain types of features). If you wish, you can delay the automatic update for edit operations by using the Delayed Update option. You can manually trigger an update of the entire model. You might, for example, want to use a net null update to check whether an existing model will successfully update in a new version of Unigraphics NX before you put a lot of additional work into modifying the model. (A net null update mechanism forces a complete update of a model, without changing it.)

The manual methods include:

- The Unigraphics NX Open C and C++ Runtime function, `UF_MODL_update_all_features`, which logs all the features in the current work part to the Unigraphics NX update list, and then performs an update. See the *Unigraphics NX Open C and C++ Runtime Reference Help* for more information.
- The Playback option on the Edit Feature dialog, which recreates the model, starting at its first feature. You can step through the model as it is created one feature at a time, move forward or backward to any feature,

or trigger an update that continues until a failure occurs or the model is complete.

The Edit during Update dialog, which appears when you choose Playback, also includes options for analyzing and editing features of the model as it is recreated (especially useful for fixing problems that caused update failures). Methods that users have tried in the past that has led to some problems or is tricky to use:

- One method uses the Edit Feature dialog to change the value of a parameter in each root feature of a part, and then change it back before leaving the Edit Feature dialog. This method produces a genuine net null update if used correctly, but you should ensure that you changed a parameter in every root feature (and that you returned all the parameters to their original values) before you trigger the update.
- Another method, attempting to suppress all of the features in a part and then unsuppressed them, can cause updates that are not net null and that will fail. The failures occur because not all features are suppressible; they are left in the model when you try to suppress all features. As the update advances, when it reaches the point where most features were suppressed, it will try to update the features that remain (this is like updating a modified version of the model). Some of the "modifications" may cause the remaining features to fail. For these reasons, we highly recommend that you do not attempt to update models by suppressing all or unsurprising all features. Use the other options described here, instead.

1.3 ANSYS INTRODUCTION

ANSYS has evolved into multipurpose design analysis software program, recognized around the world for its many capabilities. Today the program is

extremely powerful and easy to use. Each release hosts new and enhanced capabilities that make the program more flexible, more usable and faster. In this way ANSYS helps engineers meet the pressures and demands modern product development environment.

The ANSYS program is flexible, robust design analysis and optimization package. The software operates on major computers and operating systems, from PCs to workstations and to super computers. ANSYS features file compatibility throughout the family of products and across all platforms. ANSYS design data access enables user to import computer aided design models in to ANSYS, eliminating repeated work. This ensures enterprise wide, flexible engineering solution for all ANSYS user.

5.5 PROGRAM AVAILABILITY

The ANSYS program operates on 486 and Pentium based PCs running on Windows 95 or Window NT and workstations and super computers primarily running on UNIX operating system. ANSYS continually works with new hardware platforms and operating system.

Analysis types available:

1. Structural static analysis
2. Structural dynamic analysis
3. Structural buckling analysis
 - Linear buckling
 - Non linear buckling
4. Structural non linearity
5. Static and dynamic kinematic analysis
6. Thermal analysis

5.6 PROCEDURE FOR ANSYS

Static analysis is used to determine the displacements, stresses, strains ad forces in structures or components due to loads that do not induce significant inertia and damping effects. Steady loading in response conditions are assumed. The kinds of

loading that can be applied in a static analysis include externally applied forces and pressures, steady state internal forces such as gravity or rotational velocity imposed (non zero) displacements, temperatures (for thermal strain). A static analysis can be either linear or non-linear.

The procedure for structural analysis consists of these main steps:

1. Building the model
2. Obtaining the solution
3. Reviewing the results

Build the model:

In this step we specify the job name and analysis title, element types, real constants, material properties and geometry element types both linear and non-linear structural elements are allowed. The ANSYS element library contains over 80 different element types.

Material properties:

Young's modulus (EX) must be defined for a structural analysis. If we need to apply inertia loads (such as gravity), we define mass properties such as density (DENS). Similarly, if we plan to apply thermal loads (temperatures), we define coefficient of thermal expansion (ALPX).

Obtain the solution:

In this step we define the analysis type and options, apply loads and initiate the finite element solution. This involves three phases

- Pre-processor phase
- Solution phase
- Post-processor phase

PRE-PROCESSOR:

Pre-processor has been developed so that the same program is available on micro, mini, super-mini and mainframe computer system. This allows easy transfer of models one system to other.

Pre-processor is an interactive model builder to prepare the FE model and input data. The solution phase utilizes the input data developed by the pre-processor, and

prepares the solution according to the problem definition. It creates input files to the temperature etc., on the screen in the form of contours.

Model generation:

Two different methods are used to generate a model.

- Direct generation
- Solid modelling

With solid modelling we can describe the geometric boundaries of the model, establish controls over the size and desired shape of the elements and then instruct ANSYS program to generate all the nodes and elements automatically. By contrast, with the direct generation method, we determine the location of every node and size, shape and connectivity of every element prior to defining these entities in the ANSYS model. Although, some automatic data generation is possible (by using commands such as FILL, NGEN, EGEN etc). The direct generation method essentially a hands on numerical method that requires us to keep track of all the node numbers as we develop the finite element mesh. Solid modelling is usually more powerful and versatile than direct generation is commonly preferred method of generating a model.

Mesh generation:

In the finite element analysis the basic concept is to analyze the structure, which is an assemblage of discrete pieces called elements, which are connected together at a finite number of points called nodes. Loading boundary conditions are then applied to these elements and nodes. A network of these elements is known as mesh.

Finite element generation:

The maximum amount of time in a finite element analysis is spent on generating elements and nodal data. Pre-processor allows the user to generate nodes and elements automatically, at the same

time allowing control over size and number of elements. There are various types of elements that can be mapped or generated on various geometric entities.

The elements developed by various automatic element generation capabilities of pre-processor can be checked element characteristics that may need to be verified before the finite element analysis for connectivity, distortion-index etc.

Generally, automatic mesh generating capabilities of pre-processor are used rather than defining the nodes individually. If required, nodes can be defined easily by defining the allocations or by translating the existing nodes. Also one can plot, delete, or search nodes.

Boundary conditions and loading:

After completion of the finite element model it has to constrain and load has to be applied to the model. User can define constraints and loads in various ways.

Model display:

During the construction and verification stages of the model, it may be necessary to view it from different angles. It is useful to rotate the model with respect to the global system and view it from different angles. Pre-processor offers this capability. By windowing feature pre-processor allows the user to enlarge a specific area of the model for the clarity and details. Pre-processor also provides features like smoothness, scaling, regions, active set, etc for efficient model viewing and editing.

Material definitions:

All elements are defined by nodes, which have only their location defined. In the case of plate and shell elements there is no indication of thickness. The thickness can be given as element property. Different types of elements have different properties. For e.g.

- Beams: Cross sectional area, moment of inertia etc
- Shells: Thickness

- Springs: Stiffness
- Solids: None

The user also needs to define material properties of the elements. For linear static analysis, modulus of elasticity and Poisson's ratio need to be provided. For heat transfer, coefficient of thermal expansion, densities etc are required.

SOLUTION:

The solution phase deals with the solution of the problem according to the problem definitions. All the tedious work of formulating and assembling of matrices are done by the computer and finally displacements are stress values are given as output. Some of the capabilities of the ANSYS are linear static analysis, non-linear static analysis, transient analysis etc.

POST-PROCESSOR:

It is powerful user-friendly post-processing program using interactive colour graphics. It has extensive plotting feature for displaying the results obtained from the finite element analysis. One picture of the analysis results (i.e. the results in a visual form) can often reveal in seconds, what would take an engineer hour to assess from a numerical output. The engineer may also see the important aspects of the results that could be easily missed in a stack of numerical data.

Employing state of art image enhancement techniques, facilities viewing of:

- Contours of stresses, displacements, temperatures etc.
- Deform geometric plots
- Animated deformed shape
- Time-history plots
- Solid sectioning
- Hidden line plot
- Light source shaded plot
- Boundary line plot etc.

The entire range of post processing options of different types of analysis can be accessed through the command/menu mode

there by giving the user added flexibility and convenience.

LITERATURE REVIEW

Patel Ravin B, Patel Bhakti K., Patel Priyesh M, Patel Dixit H. (All B.E students) ,Patel hirak s.(assistant professor): written on “DESIGN AND ANALYSIS OF CRANE HOOK WITH DIFFERENT MATERIAL”. In this paper the design of the hook is done by analytical method and design is done for the different materials like forged steel and high tensile steel. After the analytical method design and modeling of hook is done in modeling software (solid edge). The modeling is done using the design calculation from the modeling the analysis of hook is done in FEA software (ANSYS). This result lead us to the determination of stress in existing model. By predicting the stress concentration area, the hook working life increase and reduce the failure stress.

Joseph Leo .A.1 , ArutPranesh .K.2 , Balasubramani .V: written on “STRUCTURAL ANALYSIS OF CRANE HOOK”. Crane hooks are highly liable components that are typically used for industrial purposes. Thus such components in an industry must be manufactured and designed in a way to deliver maximum performance without failure. Failure of a crane hook mainly depends on three major factors i.e. dimension , material , overload .The project is concerned towards increasing the safe load by varying the cross sectional dimensions of the three different sections .The selected sections are rectangular ,triangular, and trapezoidal . The area remains constant while changing the dimensions of the three different sections. The crane hook is modelled using PTC CREO software. The stress analysis is done using ANSYS 14.5 workbench. The normal stress along y direction, deformation along y direction and strain is considered. It is found that trapezoidal cross section yields

maximum load of 700 kg for constant cross section area among three cross sections.

G Bhagyaraj1, K Suryaprakash2, K Subba Rao 3: They published a journal on “Crane Hook Design and Analysis”. Hoists and cranes have a very important role in many sectors of various industries like the logistics, in ship building industries, steel industries, some transport vehicles, etc., for lifting loads and carry them. Many types of hooks having different ratings and sizes are used according to the specified objective. The types of hooks used are based on the type of hoist or the type of crane to which they are attached. This paper aims at building a 245 KN or 25-ton resistant block hook with a hook made up of using lead alloy in the market, Alloy 1.2367 (X38CrMoV5-3), which have high tensile strength, compression strength, and yield strength compared to the presently used alloy in the manufacturing of crane hooks. The design of block hook and its components is done in Solid works Part Design and the individual components are assembled in the Solid works assembly. The analysis on the hook using the Alloy 1.2367 (X38CrMoV5-3) is carried out in Solid works Simulation. The images are rendered using Solid works Visualize.

Dr. P.Ravikanth Raju1, Somagani Upendar2: They published a journal on “DESIGN AND ANALYSIS OF A CRANE HOOK”. In this paper the design of the hook is done by analytical method and design is done for the different materials like forged steel, wrought iron and high tensile steel. Crane hook is very significant component used for lifting the load with the help of chain or wire ropes. Crane hooks are highly liable components and are always subjected to bending stresses which leads to the failure of crane hook. To minimize the failure of crane hook, the stress induced in it must be studied. A crane is subjected to continuous loading and unloading. This may causes structural failure of the crane

hook. After the analytical method design and modeling of hook is done in modeling software. The modeling is done using the design calculation from the modeling the analysis of hook is done in FEA software. This result lead us to the determination of stress in existing model. By predicting the stress concentration area, the hook working life increase and reduce the failure stress.

DESIGNING OF CRANE HOOK

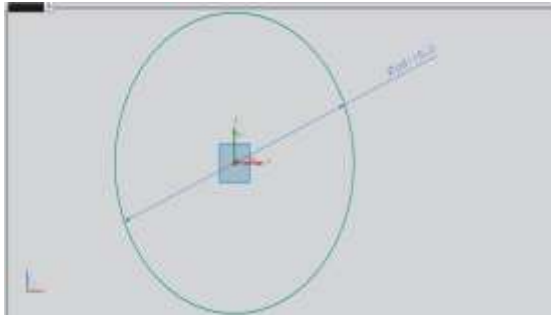


Fig.3.1 2D sketch of crane hook

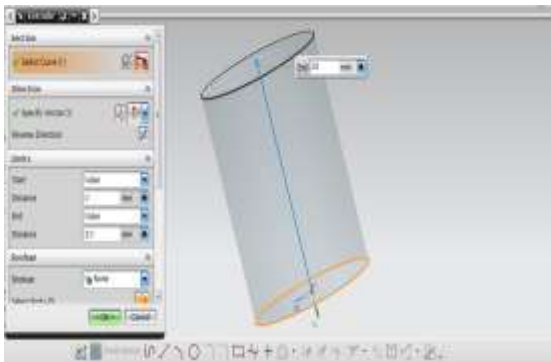


Fig.3.2 Extrude of 2D sketch of crane hook

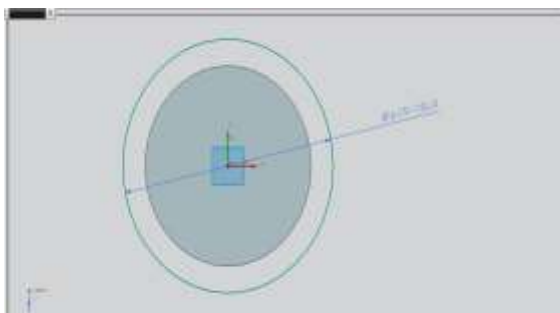


Fig.3.3 2D sketch of crane hook

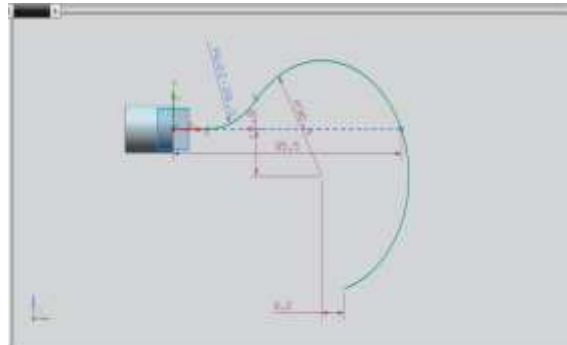


Fig.3.4 Profile 2D sketch of crane hook



Fig.3.5 Sweep along 2D sketch of crane hook

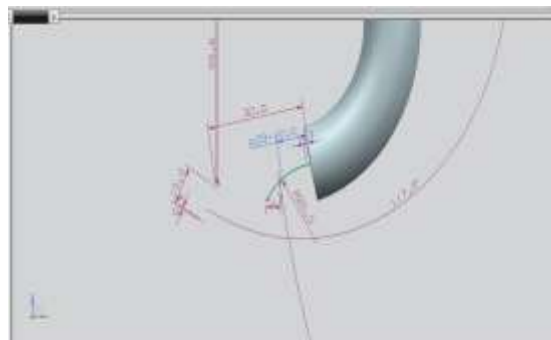


Fig.3.6 2D sketch of crane hook



Fig.3.7 Extrude of 2D sketch of crane hook



Fig.3.8 Final crane hook

STRUCTURAL ANALYSIS OF ORIGINAL CRANE HOOK

4.1 INTRODUCTION ABOUT FEM

The Basic concept in FEA is that the body or structure may be divided into smaller elements of finite dimensions called “Finite Elements”. The original body or the structure is then considered as an assemblage of these elements connected at a finite number of joints called “Nodes” or “Nodal Points”. Simple functions are chosen to approximate the displacements over each finite element. Such assumed functions are called “shape functions”. This will represent the displacement within the element in terms of the displacement at the nodes of the element.

The Finite Element Method is a mathematical tool for solving ordinary and partial differential equations. Because it is a numerical tool, it has the ability to solve the complex problems that can be represented in differential equations form. The applications of FEM are limitless as regards the solution of practical design problems.

Due to high cost of computing power of years gone by, FEA has a history of being used to solve complex and cost critical problems. Classical methods alone usually cannot provide adequate information to determine the safe working limits of a major civil engineering construction or an automobile or an aircraft.

In the recent years, FEA has been universally used to solve structural engineering problems. The departments,

which are heavily relied on this technology, are the automotive and aerospace industry. Due to the need to meet the extreme demands for faster, stronger, efficient and lightweight automobiles and aircraft, manufacturers have to rely on this technique to stay competitive.

FEA has been used routinely in high volume production and manufacturing industries for many years, as to get a product design wrong would be detrimental. For example, if a large manufacturer had to recall one model alone due to a hand brake design fault, they would end up having to replace up to few millions of hand brakes. This will cause a heavier loss to the company.

The finite element method is a very important tool for those involved in engineering design; it is now used routinely to solve problems in the following areas.

- Structural analysis
- Thermal analysis
- Vibrations and Dynamics
- Buckling analysis
- Acoustics
- Fluid flow simulations
- Crash simulations
- Mold flow simulations

Nowadays, even the most simple of products rely on the finite element method for design evaluation. This is because contemporary design problems usually cannot be solved as accurately & cheaply using any other method that is currently available. Physical testing was the norm in the years gone by, but now it is simply too expensive and time consuming also.

4.2 THE BASIC STEPS INVOLVED IN FEA

Mathematically, the structure to be analyzed is subdivided into a mesh of finite sized elements of simple shape. Within each element, the variation of displacement is

assumed to be determined by simple polynomial shape functions and nodal displacements. Equations for the strains and stresses are developed in terms of the unknown nodal displacements. From this, the equations of equilibrium are assembled in a matrix form which can be easily be programmed and solved on a computer. After applying the appropriate boundary conditions, the nodal displacements are found by solving the matrix stiffness equation. Once the nodal displacements are known, element stresses and strains can be calculated.

Basic Steps in FEA are:

1. Descritization of the domain
2. Application of Boundary conditions
3. Assembling the system equations
4. Solution for system equations
5. Post processing the results.

Descritization of the domain: The task is to divide the continuum under study into a number of subdivisions called element. Based on the continuum it can be divided into line or area or volume elements.

Application of Boundary conditions: From the physics of the problem we have to apply the field conditions i.e loads and constraints, which will help us in solving for the unknowns.

Assembling the system equations: This involves the formulation of respective characteristic (Stiffness in case of structural) equation of matrices and assembly.

Solution for system equations: Solving for the equations to know the unknowns. This is basically the system of matrices which are nothing but a set of simultaneous equations are solved.

Viewing the results: After the completion of the solution we have to review the required results.

The first two steps of the above said process is known as pre-processing stage, third and fourth is the processing stage and final step is known as post-processing stage.

4.3 STRUCTURAL ANALYSIS

Static analysis is one in which the loads/boundary conditions are not the functions of time and the assumption here is that the load is applied gradually. The most common application of FEA is the solution of stress related design problems.

Typically in a static analysis the kind of matrix solved is

$$[K] * [X] = [F]$$

Where K is called the stiffness matrix, X is the displacement vector and F is the load matrix. This is a force balance equation. Sometimes, the K matrix is the function X. Such systems are called non-linear systems.

Once the displacement vector {X} is known, the response quantities can be calculated.

4.4 STRUCTURAL ANALYSIS OF CRANE HOOK

Material:

Carbon Steel ASTM A148

Properties

Young's modulus: 190GPa

Poisson ratio: 2.9

Yield stress: 660MPa

Density: 7000Kg/m³



Fig.4.1 Imported design

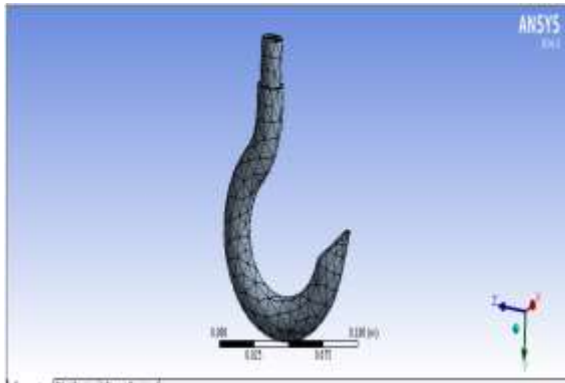


Fig.4.2 Created mesh on hook

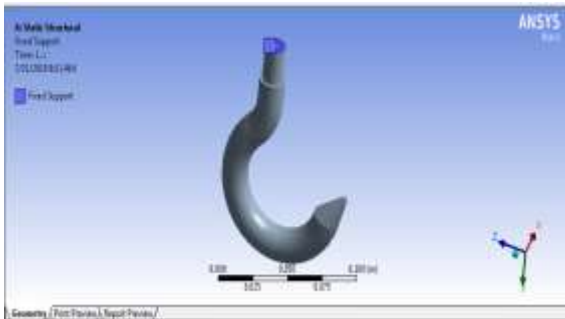


Fig.4.3 Applied fixed load on hook

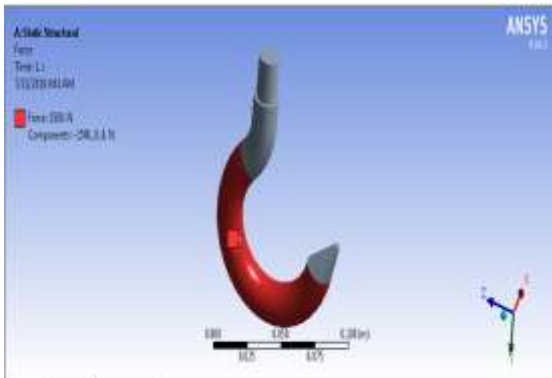


Fig.4.4 Applied load on hook

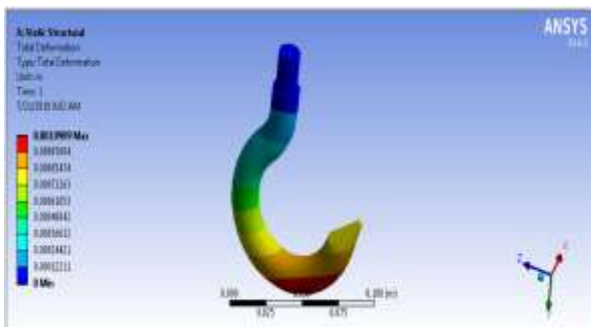


Fig.4.5 Deformation results of hook

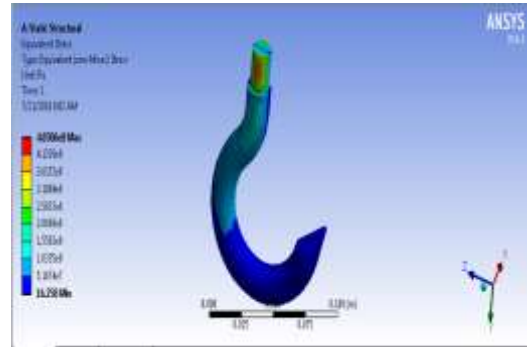


Fig.4.6 Stress results of hook

From static analysis results, observed that crane hook formed 465MPa stress. And this stress dangerous zone (rupture zone) existed at top cap of hook.

DESIGNING OF MODIFIED CRANE HOOK

5.1 DESIGN PROCEDURE

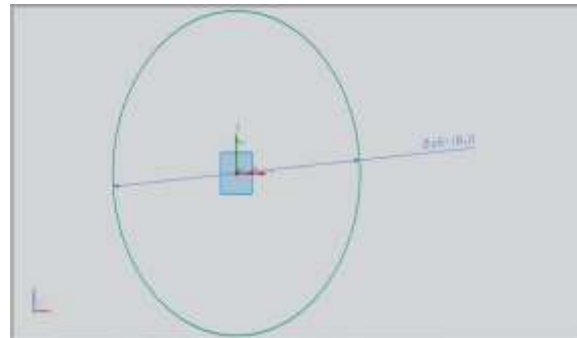


Fig.5.1 2D sketch of modified crane hook

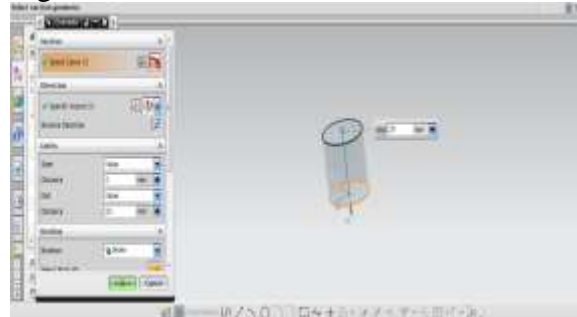


Fig.5.2 Extrude of 2D sketch of modified crane hook

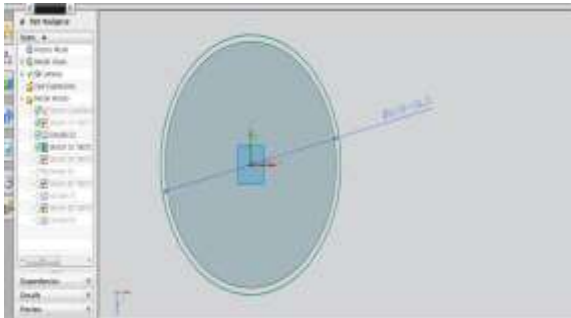


Fig.5.3 2D sketch of modified crane hook

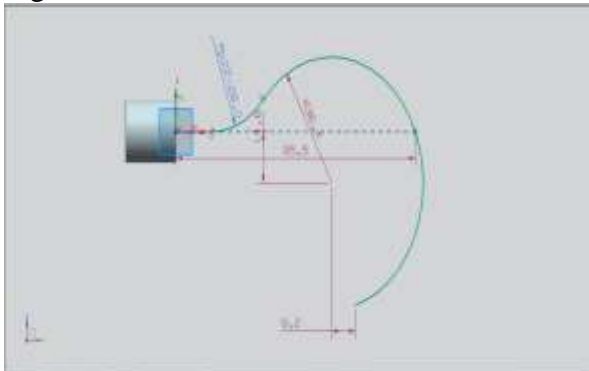


Fig.5.4 Profile 2D sketch of modified crane hook



Fig.5.5 Sweep along 2D sketch of modified crane hook

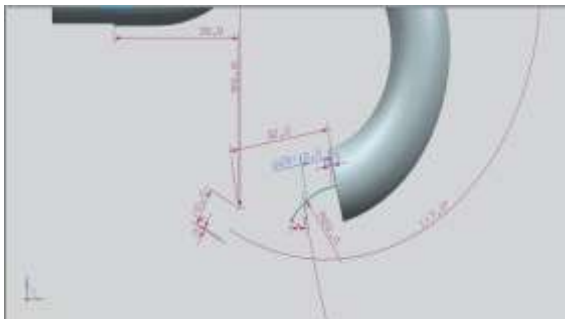


Fig.5.6 2D sketch of modified crane hook

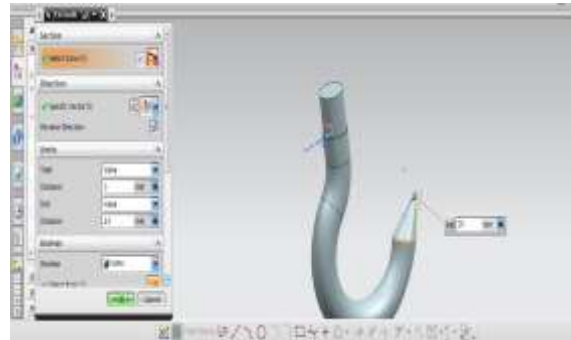


Fig.5.7 Extrude of 2D sketch of modified crane hook

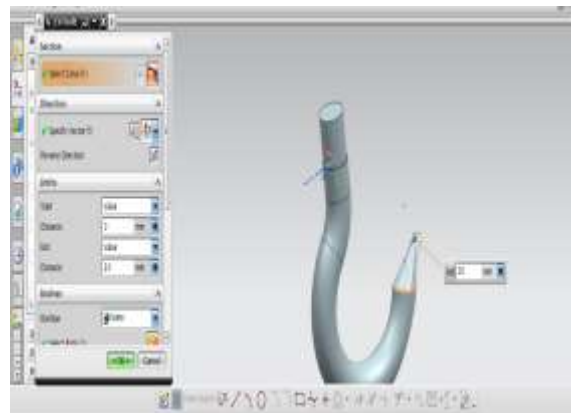


Fig.5.8 Extrude of 2D sketch of modified crane hook

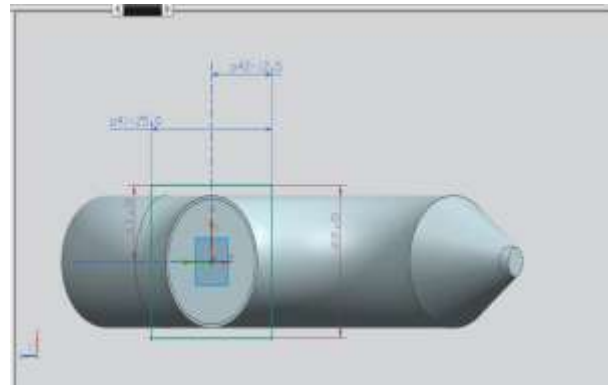


Fig.5.9 2D sketch of modified crane hook



Fig.5.10 Extrude of 2D sketch of modified crane hook



Fig.5.11 Final modified crane hook

STRUCTURAL ANALYSIS OF MODIFIED CRANE HOOK

6.1 STRUCTURAL ANALYSIS OF CRANE HOOK

Material:

Carbon Steel ASTM A148

Properties

Young's modulus: 190GPa

Poisson ratio: 2.9

Yield stress: 660MPa

Density: 7000Kg/m³

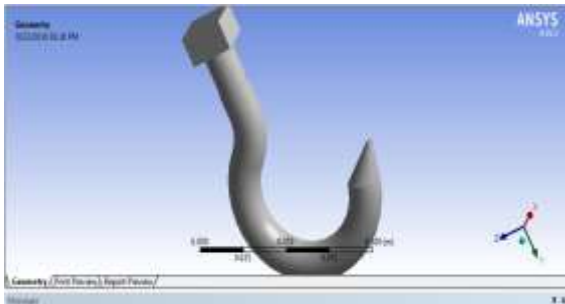


Fig.6.1 Imported design

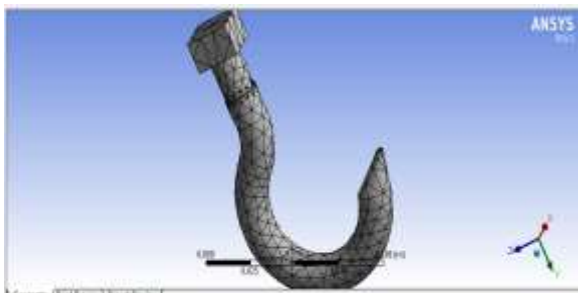


Fig.6.2 Created mesh on modified hook

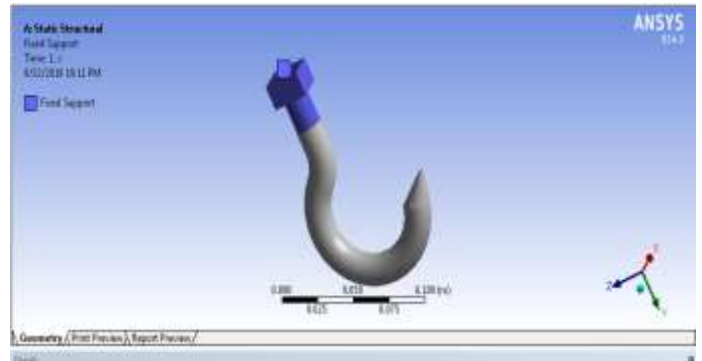


Fig.6.3 Applied fixed load on modified hook

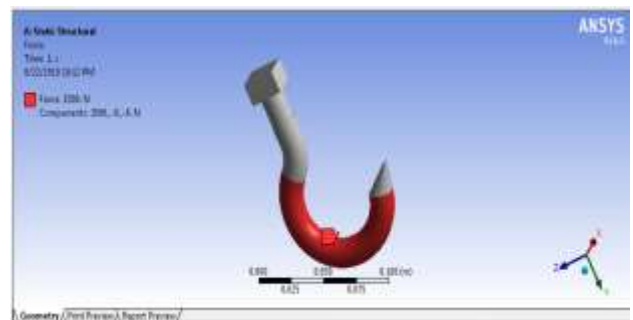


Fig.6.4 Applied load on modified hook

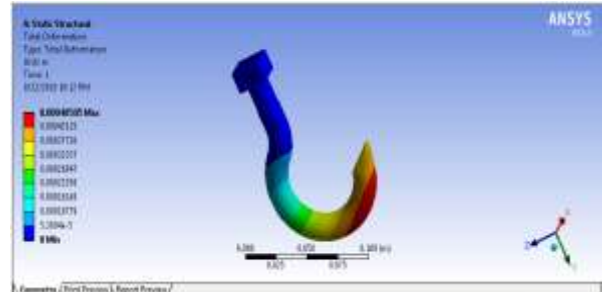


Fig.6.5 Deformation results of modified hook

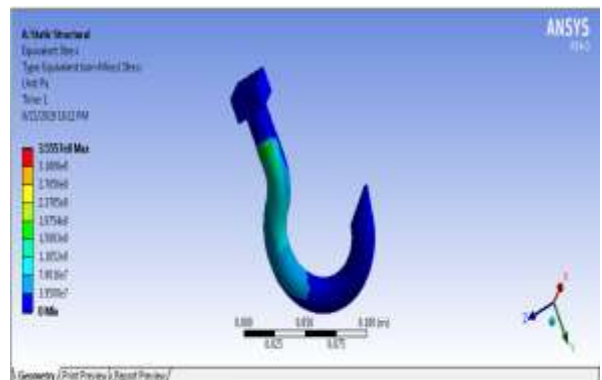


Fig.6.6 Stress results of modified hook

From static analysis results, observed that crane hook formed 355 MPa stress.

RESULTS AND CONCLUSION

In This project reduced the ruptures in crane hook by redesigning of crane hook. This process done by using structural analysis. Analysis results are given below

| Results | Crane hook | Modified crane hook |
|------------------|------------|---------------------|
| Deformation (mm) | 1.09 | 0.48 |
| Stress(MPa) | 465 | 355 |

From static analysis results of crane hook, observed that crane hook formed 465MPa stress. And this stress dangerous zone (rupture zone) existed at top cap of hook.

From static analysis results of modified crane hook , observed that crane hook formed 368MPa stress. Here no ruptures are existed and formed stress (355MPa) is less than of yield strength of Carbon Steel ASTM A148 material (660MPa). So finally concluded that modified has high mechanical efficiency.

REFERENCES

- [1] Naresh Chauhan, P.M.Bhatt, "Improving the durability of the E.O.T crane structure by finite element analysis, and optimize the hook material for improving its solidity", *Procedia Engineering*, vol. 38, pp 837 – 842, 2012.
- [2] Santosh Sahu, Ritesh Dewangan, Manas Patnaik, Narendra Yadav, "Study of Crane Hook Having Trapezoidal Section by Finite Element Method & Design of Experiments", *International Journal of Modern Engineering Research* , vol. 2, issue.4, pp 2779-2781, July-August 2012.
- [3] Ajeet Bergaley, Anshuman Purohit, "Structural Analysis of Crane Hook Using Finite Element Method", *International Journal of Science and Modern Engineering*, vol. 1, issue-10, September 2013
- [4] Mr.A.Gopichand, Ms. R.V.S.Lakshmi, Mr. B. Maheshkrishna, "Optimization of design parameters for crane hook using taguchi method", *International Journal of Innovative Research in Science, Engineering and Technology*, Vol. 2, Issue 12, December 2013.
- [5] Rashmi Uddanwadiker, "Stress Analysis of Crane Hook and Validation by Photo-Elasticity", *Engineering*, vol. 3, pp 935-941, 2011.
- [6] Patel Ravin B, Patel Bhakti. K, Patel Priyesh M, Patel Dixit. H, Patel Hirak S, "Design and analysis of crane hook with different material", *International Journal of Advanced Computer Technology*
- [7] Chetan N. Benkar, Dr. N. A. Wankhade, "Finite element stress analysis of crane hook with different cross sections", *International Journal for Technological Research in Engineering*, vol. 1, Issue 9, May-2014.
- [8] Y. Torres, J.M. Gallardo, J. Domínguez , F.J. Jiménez E, "Brittle fracture of a crane hook", *Engineering Failure Analysis*, vol. 17, pp 38–47, 2010.
- [9] Chetan N. Benkar, Prof. N. A. Wankhade, "Design and Analysis of Crane Hook – Review", *International Journal of Engineering Research & Technology*, vol. 3, Issue 1, January 2014