International Journal for Innovative Engineering and Management Research

A Peer Revieved Open Access International Journal

www.ijiemr.org

IJIEMR Transactions, online available on 12th July 2017. Link :

http://www.ijiemr.org/downloads.php?vol=Volume-6&issue=ISSUE-5

Title: Analysis and Optimization of a Material Handling Loading Crane Hook.

Volume 06, Issue 05, Page No: 1686 – 1708.

Paper Authors

*** GARJAKUNTLA DHANRAJ, K. SAINATH.**

* Dept of Mechanical, Sreyas Institute of Engineering & Technology.

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

ANALYSIS AND OPTIMIZATION OF A MATERIAL HANDLING LOADING CRANE HOOK

*** GARJAKUNTLA DHANRAJ,**K. SAINATH**

*PG Scholar, Dept of Mechanical, Sreyas Institute of Engineering & Technology, Nagole, Bandalaguda (vill), Saroornagar(Mdl), R R Dist T.S.

** Associate Professor, Dept of Mechanical, Sreyas Institute of Engineering & Technology, Nagole, Bandalaguda (vill), Saroornagar(Mdl), R R Dist T.S.

danraj.g@gmail.com sainathkasuba@gmail.com

ABSTRACT

A crane hook is a hoisting fixture designed to engage a ring or link of a lifting chain, or the pin of a shackle or cable socket. A crane hook is a curved beam and the simple theory of straight, shallow beam bending does not apply. The stress distribution across the depth of such a beam, subject to pure bending, is non linear (actually hyperbolic) and the position of the neutral surface is displaced from the censorial surface towards the centre of curvature. The aim of this project is to design, optimization and model a Crane hook which is used in material handling crane in work shop. The modeling is done in 3D modeling software Pro/Engineer. The static structural analysis and harmonic analysis is done on the crane hook by applying loads of 2.5tonnes, 2.6tonnes and 2.7tonnes. The analysis is done to validate whether the design withstands the applied loads. We are conducting the analysis on crane hook by applying the materials 1.Forged steel 2.High carbon steel 3.High come steel. The results of structural analysis are displacement and stress and results of the harmonic analysis is natural frequency of the component. In this project we are going to validate our design at different load conditions and material optimization is done by applying different load conditions.

INTRODUCTION TO CRANE HOO

A crane hook apparatus includes a hook which is rockably mounted on a rotatable stem member extending from a hanging body. A swingable locking device is pivotally mounted on the stem member, and normally assumes by gravity its operative position in which it substantially closes the opening of a recess in the hook. When located in its operative position, the locking device is movable toward its inoperative position inward into the recess, but is blocked from an outward movement. The locking device includes a portion which bears against the interior surface of the hook when it is moved to the inoperative position within the recess. The hanging body is provided with remotely controllable drive means which hauls a suspension member which extends through openings formed in alignment with the axis of

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

the stem member for connection with the locking device. The drive means is operative to cause the suspension member to move the locking device to its inoperative position and then to cause the bearing portion to move the hook in a manner such that the recess opens in a horizontal direction. In this manner, the engagement or disengagement of the hook with or from a load is remotely controlled from a driver's cab.

INTRODUCTION TO CAD

Computer-aided design (**CAD**), also known as **computer-aided design and drafting** (**CADD**), is the use of computer technology for the process of design and design-documentation. Computer Aided Drafting describes the process of drafting with a computer. CADD software, or environments, provides the user with input-tools for the purpose of streamlining design processes; drafting, documentation, and manufacturing processes. CADD output is often in the form of electronic files for print or machining operations. The development of CADD-based software is in direct correlation with the processes it seeks to economize; industry-based software (construction, manufacturing, etc.) typically uses vector-based (linear) environments whereas graphic-based software utilizes raster-based (pixelated) environments.

CADD environments often involve more than just shapes. As in the manual drafting of technical and engineering drawings, the output of CAD must convey information, such as materials, processes, dimensions, and tolerances, according to application-specific conventions.

CAD may be used to design curves and figures in two-dimensional (2D) space; or curves, surfaces, and solids in three-dimensional (3D) objects. CAD is an important industrial art extensively used in many applications, including automotive, shipbuilding, and aerospace industries, industrial and architectural design, prosthetics, and many more. CAD is also widely used to produce computer animation for special effects in movies, advertising and technical manuals. The modern ubiquity and power of computers means that even perfume bottles and shampoo dispensers are designed using techniques unheard of by engineers of the 1960s. Because of its enormous economic importance, CAD has been a major driving force for research in computational geometry, computer graphics (both hardware and software), and discrete differential geometry.

The design of geometric models for object shapes, in particular, is often called *computer-aided geometric design* (*CAGD*).

Current computer-aided design software packages range from 2D vector-based drafting systems to 3D solid and surface modelers.

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

Modern CAD packages can also frequently allow rotations in three dimensions, allowing viewing of a designed object from any desired angle, even from the inside looking out. Some CAD software is capable of dynamic

mathematic modeling, in which case it may be marketed as **CADD** — *computer-aided design and drafting*.

CAD is used in the design of tools and machinery and in the drafting and design of all types of buildings, from small residential types (houses) to the largest commercial and industrial structures (hospitals and factories).

CAD is mainly used for detailed engineering of 3D models and/or 2D drawings of physical components, but it is also used throughout the engineering process from conceptual design and layout of products, through strength and dynamic analysis of assemblies to definition of manufacturing methods of components. It can also be used to design objects.

CAD has become an especially important technology within the scope of computer-aided technologies, with benefits such as lower product development costs and a greatly shortened design cycle. CAD enables designers to lay out and develop work on screen, print it out and save it for future editing, saving time on their drawings.

Types of CAD Software

2D CAD

Two-dimensional, or 2D, CAD is used to create flat drawings of products and structures. Objects created in 2D CAD are made up of lines, circles, ovals, slots and curves. 2D CAD programs usually include a library of geometric images; the ability to create Bezier curves, splines and polylines, the ability to define hatching patterns; and the ability to provide a bill of materials generation. Among the most popular 2D CAD programs are AutoCAD, CADkey, CADDS 5, and Medusa.

3D CAD

Three-dimensional (3D) CAD programs come in a wide variety of types, intended for different applications and levels of detail. Overall, 3D CAD programs create a realistic model of what the design object will look like, allowing designers to solve potential problems earlier and with lower production costs. Some 3D CAD programs include Autodesk Inventor, CoCreate Solid Designer, Pro/Engineer SolidEdge, SolidWorks, Unigraphics NX and VX CAD, CATIA V5.

3D Wireframe and Surface Modeling

CAD programs that feature 3D wireframe and surface modeling create a skeleton-like inner structure of the object being modeled. A surface is added on later. These types of CAD models are difficult to translate into other software and are therefore rarely used anymore.

Solid Modeling

Solid modeling in general is useful because the program is often able to calculate the dimensions of the object it is creating. Many sub-types of this exist. Constructive Solid

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

Geometry (CSG) CAD uses the same basic logic as 2D CAD, that is, it uses prepared solid geometric objects to create an object. However, these types of CAD software often cannot be adjusted once they are created. Boundary Representation (Brep) solid modeling takes CSG images and links them together. Hybrid systems mix CSG and Brep to achieve desired designs.

INTRODUCTION TO PRO/ENGINEER

Pro/ENGINEER Wildfire is the standard in 3D product design, featuring industry-leading productivity tools that promote best practices in design while ensuring compliance with your industry and company standards. Integrated Pro/ENGINEER CAD/CAM/CAE solutions allow you to design faster than ever, while maximizing innovation and quality to ultimately create exceptional products. Customer requirements may change and time pressures may continue to mount, but your product design needs remain the same - regardless of your project's scope, you need the powerful, easy-to-use, affordable solution that Pro/ENGINEER provides.

Pro/ENGINEER Wildfire Benefits:

•Unsurpassed geometry creation capabilities allow superior product differentiation and manufacturability

•Fully integrated applications allow you to develop everything from concept to manufacturing within one application

•Automatic propagation of design changes to all downstream deliverables allows you to design with confidence

•Complete virtual simulation capabilities enable you to improve product performance and exceed product quality goals

•Automated generation of associative tooling design, assembly instructions, and machine code allow for maximum production efficiency

Pro ENGINEER can be packaged in different versions to suit your needs, from Pro/ENGINEER Foundation XE, to Advanced XE Package and Enterprise XE Package, Pro/ENGINEER Foundation XE Package brings together a broad base of functionality. From robust part modelling to advanced surfacing, powerful assembly modelling and simulation, your needs will be met with this scalable solution. Flex3C and Flex Advantage Build on this base offering extended functionality of your choosing.

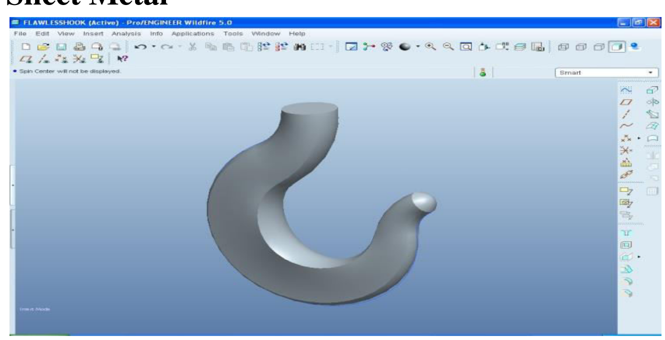
The main modules are

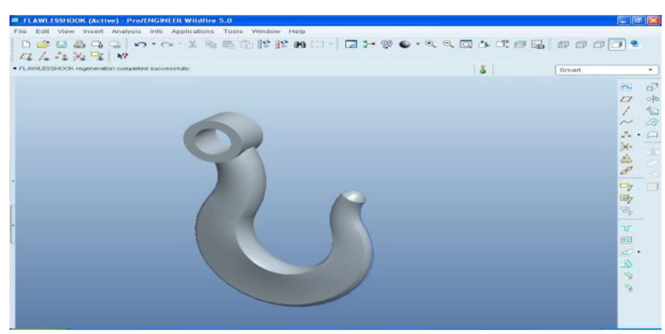
Part Design

Assembly

Drawing

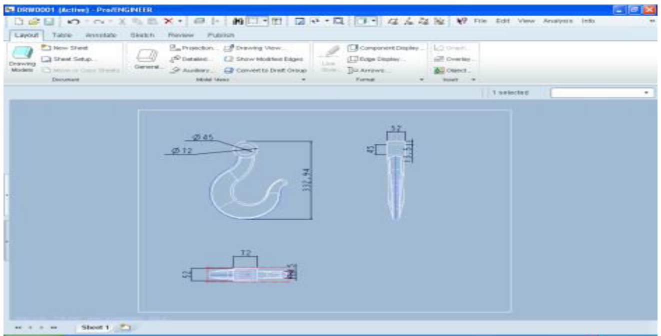
Sheet Metal





International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

2D Drawings of Crane hook



INTRODUCTION TO FEA

Finite Element Analysis (FEA) was first developed in 1943 by R. Courant, who utilized the Ritz method of numerical analysis and minimization of variational calculus to obtain approximate solutions to vibration systems. Shortly thereafter, a paper published in 1956 by M. J. Turner, R. W. Clough, H. C. Martin, and L. J. Topp established a broader definition of numerical analysis. The paper centered on the "stiffness and deflection of complex structures".

FEA consists of a computer model of a material or design that is stressed and analyzed for specific results. It is used in new product design, and existing product refinement. A company is able to verify a proposed design will be able to perform to the client's specifications prior to manufacturing or construction. Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition. There are generally two types of analysis that are used in industry: 2-D modeling, and 3-D modeling. While 2-D modeling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3-D modeling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively. Within each of these modeling schemes, the programmer can insert numerous algorithms (functions) which may make the system behave linearly or non-linearly. Linear systems are far less complex and generally do not take into account plastic deformation. Non-linear systems do account for plastic deformation, and many also are capable of testing a material all the way to fracture.

FEA uses a complex system of points called nodes which make a grid called a mesh. This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. Regions which will receive large amounts of stress usually have a higher node density than those which experience little or no stress. Points of interest may consist of: fracture point of previously tested material, fillets, corners, complex detail, and high stress areas. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes. This web of vectors is what carries the material properties to the object, creating many elements. A wide range of objective functions (variables within the system) are available for minimization or maximization:

Mass, volume, temperature
Strain energy, stress strain
Force, displacement, velocity, acceleration
Synthetic (User defined)

There are multiple loading conditions which may be applied to a system. Some examples are shown:

Point, pressure, thermal, gravity, and centrifugal static loads
Thermal loads from solution of heat transfer analysis
Enforced displacements
Heat flux and convection
Point, pressure and gravity dynamic loads

Each FEA program may come with an element library, or one is constructed over time. Some sample elements are:

Rod elements
Beam elements
Plate/Shell/Composite elements
Shear panel
Solid elements
Spring elements
Mass elements
Rigid elements
Viscous damping elements

Types of Engineering Analysis

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 as in.

Vibrational

analysis is used to test a material against random vibrations, shock, and impact. Each of these incidences may act on the natural vibrational frequency of the material which, in turn, may cause resonance and subsequent failure. 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.

Heat Transfer

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.

Results of Finite Element Analysis

FEA has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material and allowing designers to see all of the theoretical stresses within. This method of product design and testing is far superior to the manufacturing costs which would accrue if each sample was actually built and tested. In practice, a finite element analysis usually consists of three principal steps:

Preprocessing: The user constructs a model of the part to be analyzed in which the geometry is divided into a number of discrete sub regions, or elements," connected at discrete points called nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads. These models can be extremely time consuming to prepare, and commercial codes vie with one another to have the most user-friendly graphical “preprocessor" to assist in this rather tedious chore. Some of these preprocessors can overlay a mesh on a preexisting CAD file, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process.

Analysis: The dataset prepared by the preprocessor is used as input to the finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations
Kijuj = fi
where u and f are the displacements and externally applied forces at the nodal points. The formation of the K matrix is dependent on the type of problem being attacked, and this module will outline the approach for truss and linear elastic stress analyses. Commercial codes may have very large element libraries, with elements appropriate to a wide range of problem types. One of FEA's principal advantages is that many problem types can be addressed with the same code, merely by specifying the appropriate element types from the library.

Post processing: In the earlier days of finite element analysis, the user would pore through reams of numbers generated by the code, listing displacements and stresses at discrete positions within the model. It is easy to miss important trends and hot spots this way, and modern codes use graphical displays to assist in visualizing the results. A typical postprocessor display overlay colored contours representing stress levels on the model, showing a full field picture similar to that of photo elastic or moiré experimental results.

INTRODUCTION TO ANSYS

ANSYS is general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behaviour of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand. Systems that may fit into this category are too complex due to their geometry, scale, or governing equations.

ANSYS is the standard FEA teaching tool within the Mechanical Engineering Department at many colleges. ANSYS is also used in Civil and Electrical Engineering, as well as the Physics and Chemistry departments. ANSYS provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping.

With virtual prototyping techniques, users can iterate various scenarios to optimize the product long before the manufacturing is started. This enables a reduction in the level of risk, and in the cost of ineffective designs. The multifaceted nature of ANSYS also provides a means to ensure that users are able to see the effect of a design on the whole behavior of the product, be it electromagnetic, thermal, mechanical etc.

Generic Steps to Solving any Problem in ANSYS:

Like solving any problem analytically, you need to define (1) your solution domain, (2) the physical model, (3) boundary conditions and (4) the physical properties. You then solve the problem and present the results. In numerical methods, the main difference is an extra step called mesh generation. This is the step that divides the complex model into small elements that become solvable in an otherwise too complex situation. Below describes the processes in terminology slightly more attune to the software.

Build Geometry

Construct a two or three dimensional representation of the object to be modeled and tested using the work plane coordinate system within ANSYS.

Define Material Properties

Now that the part exists, define a library of the necessary materials that compose the object (or project) being modeled. This includes thermal and mechanical properties.

Generate Mesh

At this point ANSYS understands the makeup of the part. Now define how the modeled system should be broken down into finite pieces.

Apply Loads

Once the system is fully designed, the last task is to burden the system with constraints, such as physical loadings or boundary conditions.

Obtain Solution

This is actually a step, because ANSYS needs to understand within what state (steady state, transient… etc.) the problem must be solved.

Present the Results

After the solution has been obtained, there are many ways to present ANSYS' results, choose from many options such as tables, graphs, and contour plots.

Specific Capabilities of ANSYS:

Structural

Structural analysis is probably the most common application of the finite element method as it implies bridges and buildings, naval, aeronautical, and mechanical structures such as ship hulls, aircraft bodies, and machine housings, as well as mechanical components such as pistons, machine parts, and tools.

Static Analysis

- Used to determine displacements, stresses, etc. under static loading conditions. ANSYS can compute both linear and nonlinear static analyses. Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.

Transient Dynamic Analysis –

Used to determine the response of a structure to arbitrarily time-varying loads. All nonlinearities mentioned under Static Analysis above are allowed.

Buckling Analysis –

Used to calculate the buckling loads and determine the buckling mode shape. Both linear (eigen value) buckling and nonlinear buckling analyses are possible. In addition to the above analysis types, several special-purpose features are available such as

Fracture mechanics, **Composite material analysis**, **Fatigue**, and both **p-Method and Beam analyses.**

Thermal

ANSYS is capable of both steady state and transient analysis of any solid with thermal boundary conditions. Steady-state thermal analyses calculate the effects of steady thermal loads on a system or component. Users often perform a steady-state analysis before doing a transient thermal analysis, to help establish initial conditions. A steady-state analysis also can be the last step of a transient thermal analysis; performed after all transient effects have diminished. ANSYS can be used to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time. Such loads include the following:

- Convection
- Radiation
- Heat flow rates
- Heat fluxes (heat flow per unit area)
- Heat generation rates (heat flow per unit volume)
- Constant temperature boundaries

A steady-state thermal analysis may be either linear, with constant material properties; or nonlinear, with material properties that depend on temperature. The thermal properties of most material vary with temperature. This temperature dependency being appreciable, the analysis becomes nonlinear. Radiation boundary conditions also make the analysis nonlinear. Transient calculations are time dependent and ANSYS can both solve distributions as well as create video for time incremental displays of models.

Fluid Flow

The ANSYS/FLOTRAN CFD (Computational Fluid Dynamics) offers comprehensive tools for analyzing two-dimensional and three-dimensional fluid flow fields. ANSYS is capable of modeling a vast range of analysis types such as: airfoils for pressure analysis of airplane wings (lift and drag), flow in supersonic nozzles, and complex, three-dimensional flow patterns in a pipe bend. In addition, ANSYS/FLOTRAN could be used to perform tasks including:

- Calculating the gas pressure and temperature distributions in an engine exhaust manifold
- Studying the thermal stratification and breakup in piping systems
- Using flow mixing studies to evaluate potential for thermal shock
- Doing natural convection analyses to evaluate the thermal performance of chips in electronic enclosures
- Conducting heat exchanger studies involving different fluids separated by solid regions

Coupled Fields

A *coupled-field analysis* is an analysis that takes into account the interaction (coupling) between two or more disciplines (fields) of

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

engineering. A piezoelectric analysis, for example, handles the interaction between the structural and electric fields: it solves for the voltage distribution due to applied displacements, or vice versa. Other examples of coupled-field analysis are thermal-stress analysis, thermal-electric analysis, and fluid-structure analysis.

Some of the applications in which coupled-field analysis may be required are pressure vessels (thermal-stress analysis), fluid flow constrictions (fluid-structure analysis), induction heating (magnetic-thermal analysis), ultrasonic transducers (piezoelectric analysis), magnetic forming (magneto-structural analysis), and micro-electro mechanical systems (MEMS).

Modal Analysis –

A modal analysis is typically used to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. It can also serve as a starting point for another, more detailed, dynamic analysis, such as a harmonic response or full transient dynamic analysis.

Modal analyses, while being one of the most basic dynamic analysis types available in ANSYS, can also be more computationally time consuming than a typical static analysis. A reduced solver, utilizing automatically or manually selected master degrees of freedom is used to drastically reduce the problem size and solution time.

Harmonic Analysis –

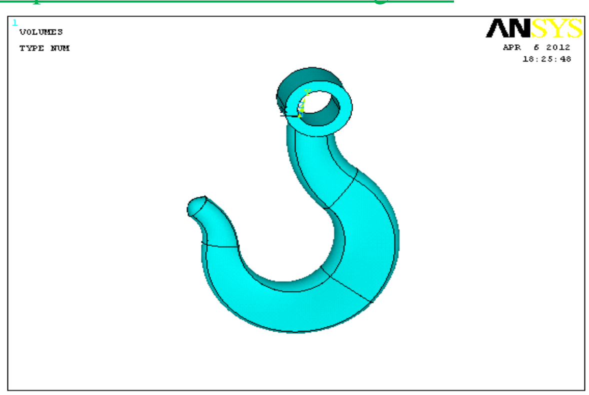
Used extensively by companies who produce rotating machinery, ANSYS Harmonic analysis is used to predict the sustained dynamic behavior of structures to consistent cyclic loading. Examples of rotating machines which produced or are subjected to harmonic loading are:

Turbines Gas Turbines for Aircraft and Power Generation
Steam Tubines
Wind Turbine
Water Turbines
Turbo pumps
Internal Combustion engines
Electric motors and generators
Gas and fluid pumps
Disc drives

A harmonic analysis can be used to verify whether or not a machine design will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations.

Structural Analysis of crane hook using Cast iron of 2.5 tons

Imported Model from Pro/Engineer



Element Type: Solid 20 node 95

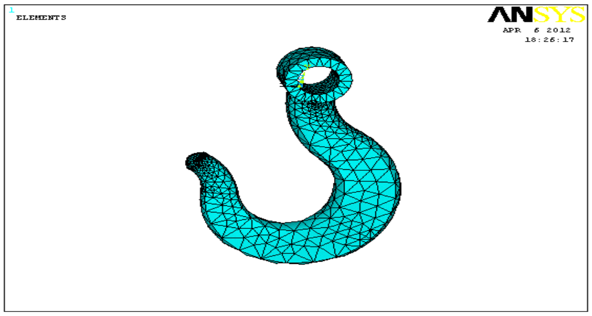
Material Properties: Youngs Modulus (EX) : 103000N/mm^2

Poissons Ratio (PRXY) : 0.211

Density :0.000007 kg/mm^3

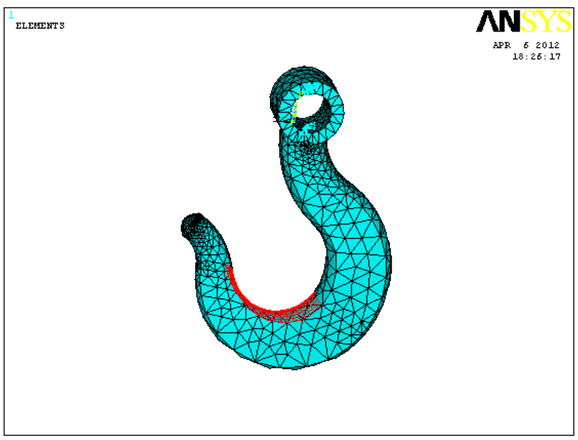
International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

Meshed Model



Loads

Pressure – 13.389N/mm^2

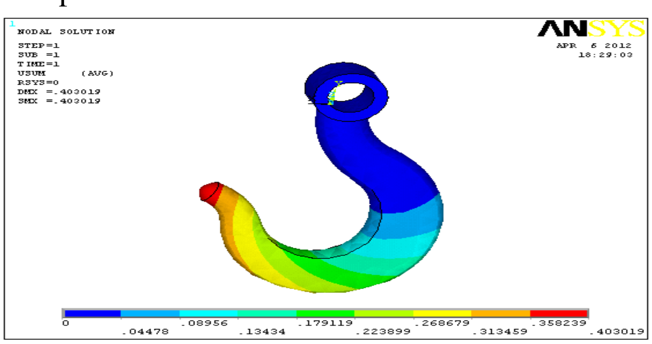


Solution

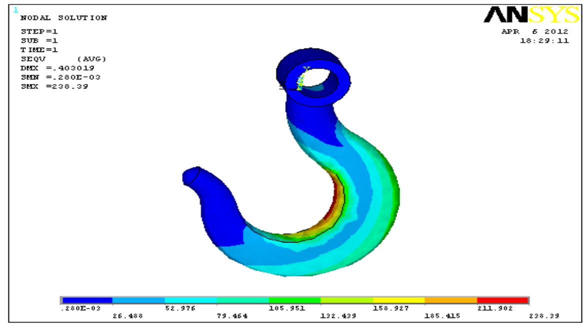
Solution – Solve – Current LS – ok

Post Processor

General Post Processor – Plot Results – Contour Plot - Nodal Solution – DOF Solution – Displacement Vector Sum

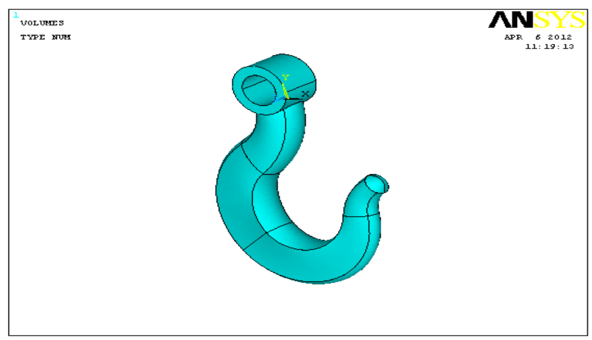


General Post Processor – Plot Results – Contour Plot – Nodal Solution – Stress – Von Mises Stress



MODAL ANALYSIS of crane hook using Cast iron of 2.5 tons

Imported Model from Pro/Engineer



c

Element Type: Solid 20 node 95

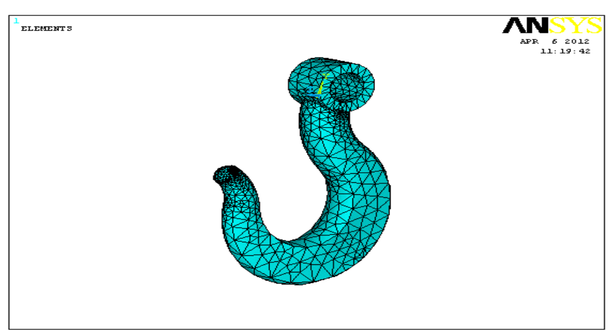
Material Properties: Youngs Modulus (EX) : 103000N/mm^2

Poissons Ratio (PRXY) : 0.211

Density :0.0000071 kg/mm^3

Meshed Model

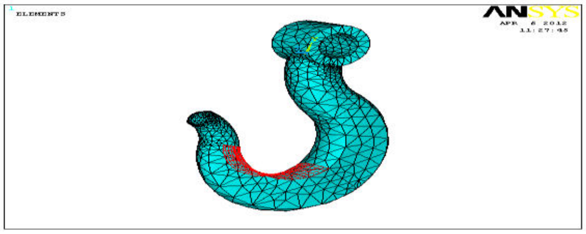
5 ir sx bhn411204



International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

Loads

Pressure –13.398N/mm^2



Main menu>Preprocessor>Loads>Analysis Type>

New Analysis> Select Modal>

Click> OK

Main menu>Preprocessor>Loads>Analysis Type>

Analysis Options>

No. Of Modes to Extract: 3

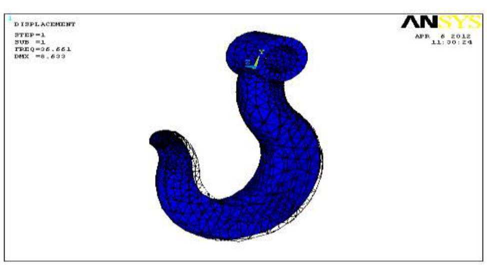
Click> OK

Main menu>Solution>Solve>Current Ls>Ok

Results

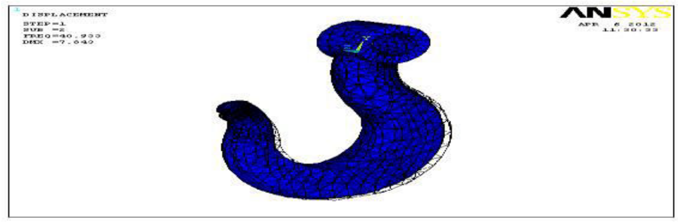
Main menu>General Postproc>Read Results> First Set

Plot result>Deformed Shape> Def+ Undeform > Click> OK



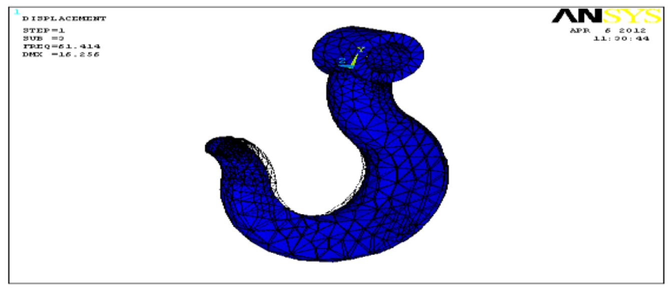
Main menu>General Postproc>Read Results> Next Set

Plot result>Deformed Shape> Def+ Undeform > Click> OK



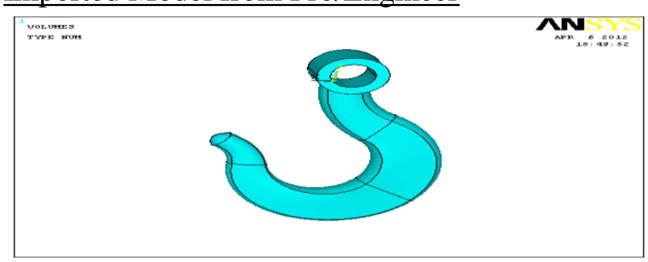
Main menu>General Postproc>Read Results> Next Set

Plot result>Deformed Shape> Def+ Undeform > Click> OK



Structural Analysis of crane hook using High Carbon Steel of 2.5 tons

Imported Model from Pro/Engineer



Element Type: Solid 20 node 95

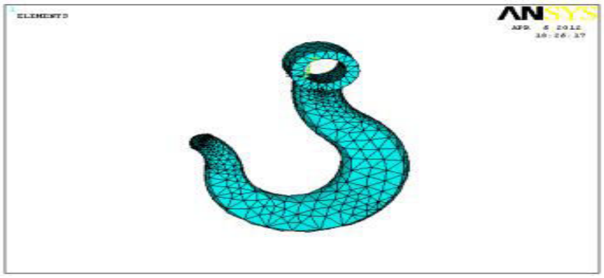
Material Properties: Youngs Modulus (EX) : 200000N/mm^2

Poissons Ratio (PRXY) : 0.295

Density :0.000007872kg/mm^3

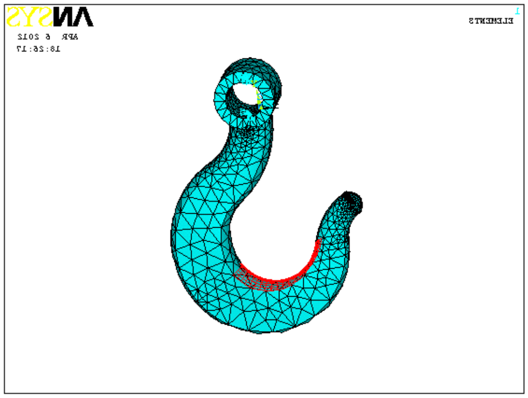
International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

Meshed Model



Loads

Pressure – 13.389N/mm^2

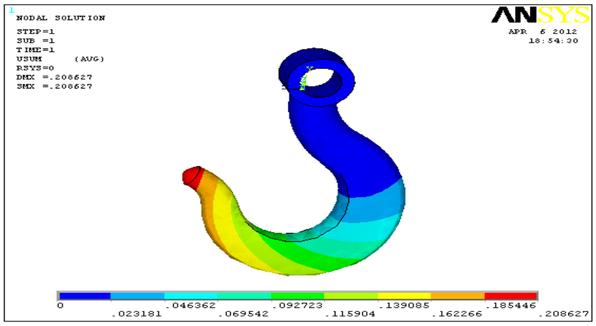


Solution

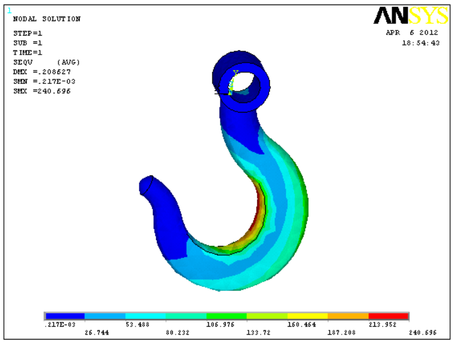
Solution – Solve – Current LS – ok

Post Processor

General Post Processor – Plot Results – Contour Plot - Nodal Solution – DOF Solution – Displacement Vector Sum

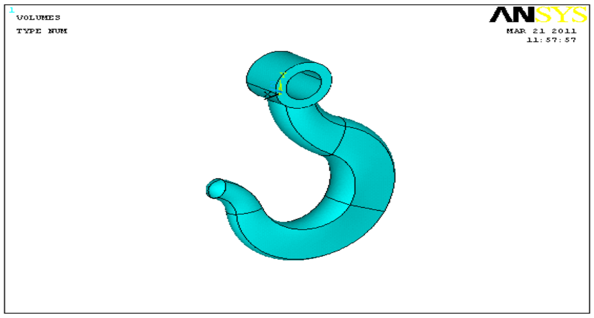


General Post Processor – Plot Results – Contour Plot – Nodal Solution – Stress – Von Mises Stress



MODAL ANALYSIS crane hook using High Carbon Steel of 2.5 tons

Imported Model from Pro/Engineer



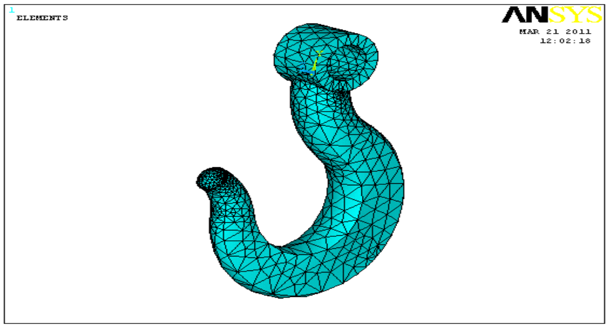
Element Type: Solid 20 node 95

Material Properties: Youngs Modulus (EX) : 200000N/mm^2

Poissons Ratio (PRXY) : 0.295

Density :0.000007872kg/mm^3

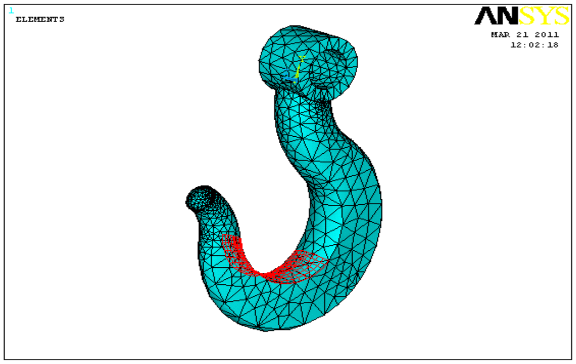
Meshed Model



International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

Loads

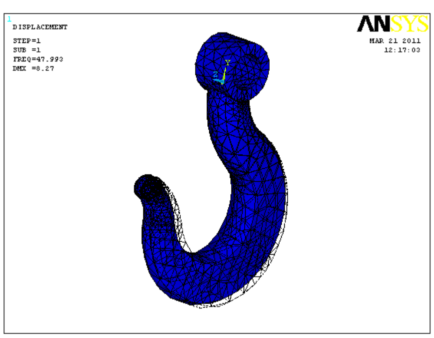
Pressure – 13.898N/mm^2



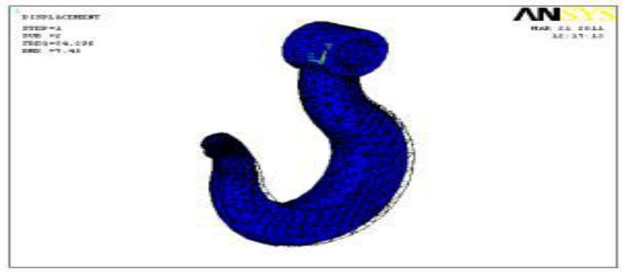
Main menu>Preprocessor>Loads>Analysis Type>
New Analysis> Select Modal>
Click> OK
Main menu>Preprocessor>Loads>Analysis Type>
Analysis Options>
No. Of Modes to Extract: 3
Click> OK
Main menu>Solution>Solve>Current Ls>Ok

Results

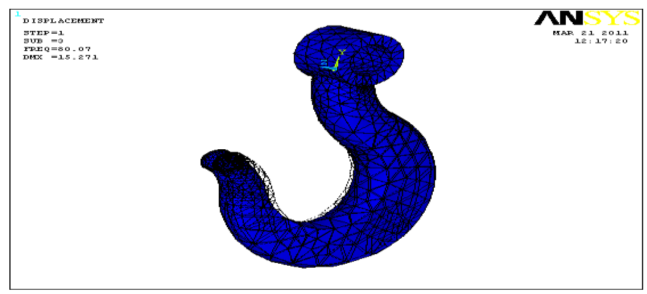
Main menu>General Postproc>Read Results> First Set
Plot result>Deformed Shape> Def+ Undeform > Click> OK



Main menu>General Postproc>Read Results> Next Set
Plot result>Deformed Shape> Def+ Undeform > Click> OK

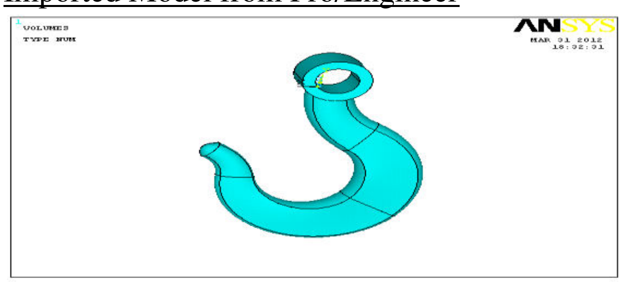


Main menu>General Postproc>Read Results> Next Set
Plot result>Deformed Shape> Def+ Undeform > Click> OK



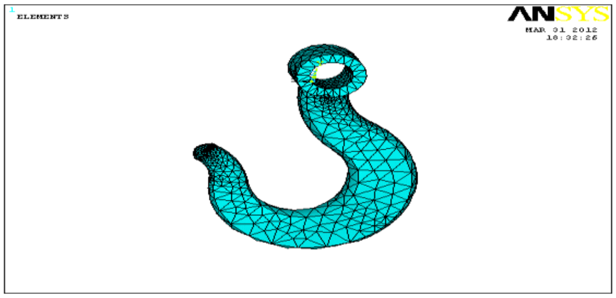
Structural Analysis of crane hook using Cast Iron of 2.6 tons

Imported Model from Pro/Engineer



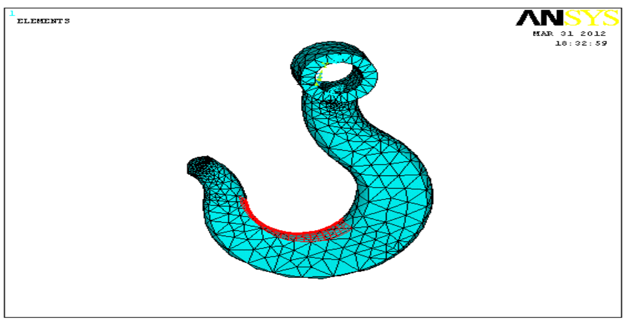
Element Type: Solid 20 node 95
Material Properties: Youngs Modulus (EX) : 103000N/mm^2
Poissons Ratio (PRXY) : 0.211
Density :0.000007 kg/mm^3

Meshed Model



Loads

Pressure – 13.9252N/mm^2

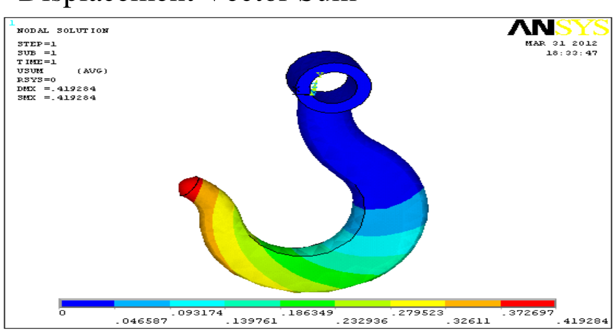


Solution

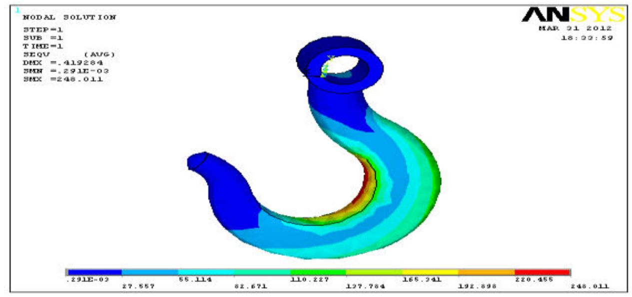
Solution – Solve – Current LS – ok

Post Processor

General Post Processor – Plot Results – Contour Plot - Nodal Solution – DOF Solution – Displacement Vector Sum

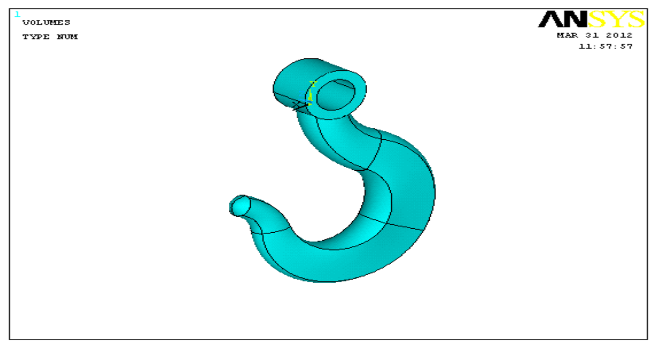


General Post Processor – Plot Results – Contour Plot – Nodal Solution – Stress – Von Mises Stress



MODAL ANALYSIS of crane hook using Cast Iron of 2.6 tons

Imported Model from Pro/Engineer



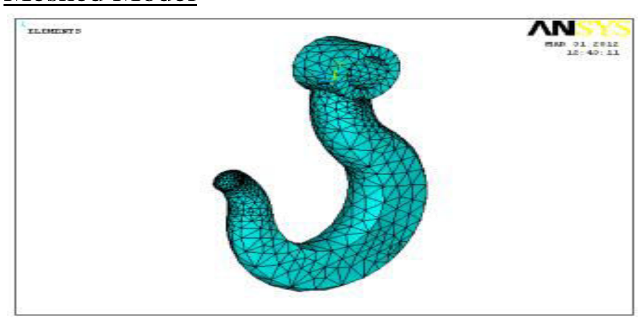
Element Type: Solid 20 node 95

Material Properties: Youngs Modulus (EX) : 103000N/mm^2

Poissons Ratio (PRXY) : 0.211

Density :0.0000071 kg/mm^3

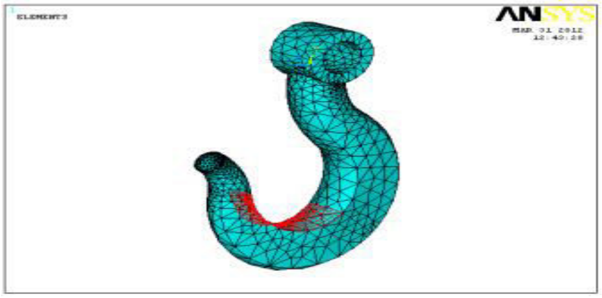
Meshed Model



Loads

Pressure – 13.9252N/mm^2

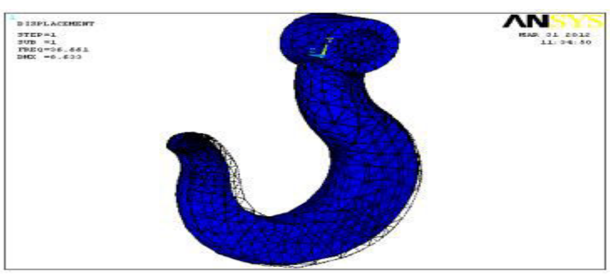
International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org



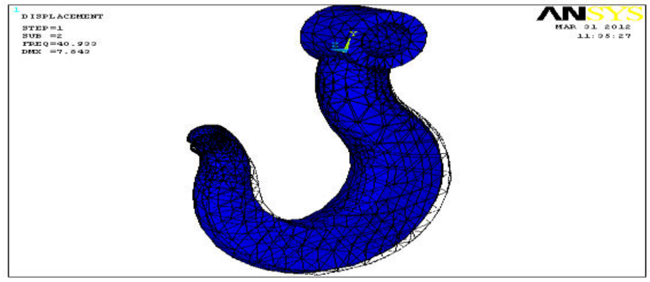
Main menu>Preprocessor>Loads>Analysis Type>
New Analysis> Select Modal>
Click> OK
Main menu>Preprocessor>Loads>Analysis Type>
Analysis Options>
No. Of Modes to Extract: 3
Click> OK
Main menu>Solution>Solve>Current Ls>Ok

Results

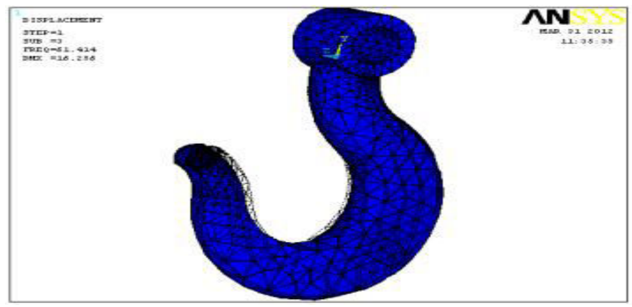
Main menu>General Postproc>Read Results> First Set
Plot result>Deformed Shape> Def+ Undeform > Click> OK



Main menu>General Postproc>Read Results> Next Set
Plot result>Deformed Shape> Def+ Undeform > Click> OK

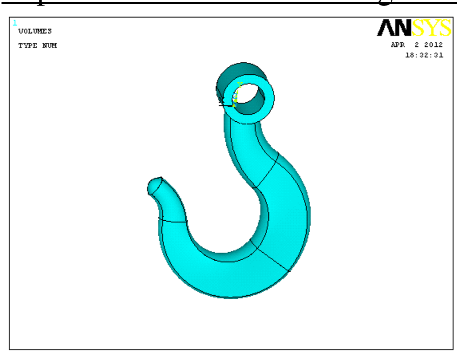


Main menu>General Postproc>Read Results> Next Set
Plot result>Deformed Shape> Def+ Undeform > Click> OK



Structural Analysis of crane hook using High Carbon Steel of 2.6 tons

Imported Model from Pro/Engineer



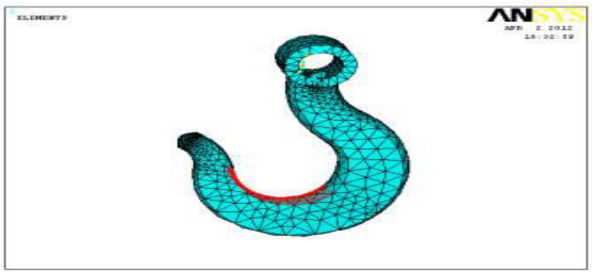
Element Type: Solid 20 node 95
Material Properties: Youngs Modulus (EX) : 200000N/mm^2
Poissons Ratio (PRXY) : 0.295
Density :0.000007872kg/mm^3

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

Meshed Model

Loads

Pressure – 13.9252N/mm^2

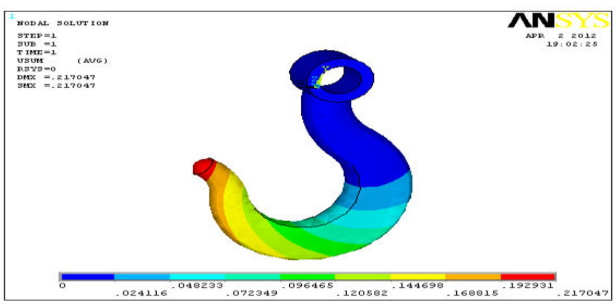


Solution

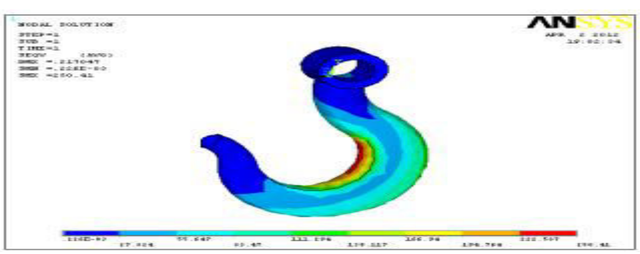
Solution – Solve – Current LS – ok

Post Processor

General Post Processor – Plot Results – Contour Plot - Nodal Solution – DOF Solution – Displacement Vector Sum

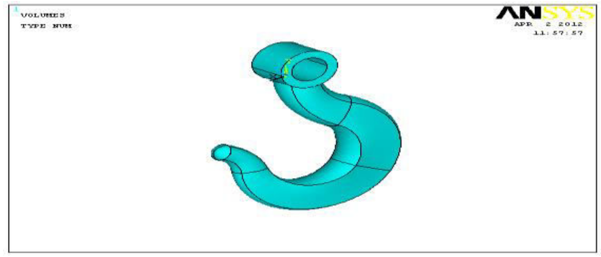


General Post Processor – Plot Results – Contour Plot – Nodal Solution – Stress – Von Mises Stress



MODAL ANALYSIS of crane hook using High Carbon Steel of 2.6 tons

Imported Model from Pro/Engineer



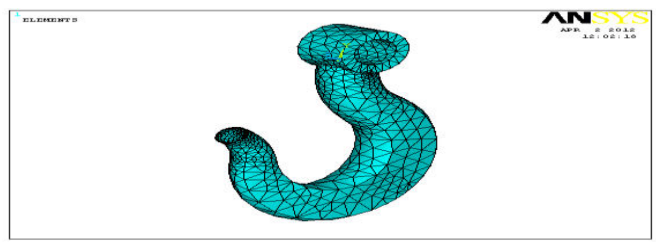
Element Type: Solid 20 node 95

Material Properties: Youngs Modulus (EX) : 200000N/mm^2

Poissons Ratio (PRXY) : 0.295

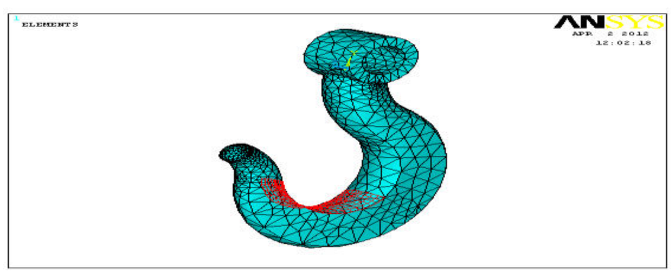
Density :0.000007872kg/mm^3

Meshed Model



Loads

Pressure – 13.9252N/mm^2



Main menu>Preprocessor>Loads>Analysis Type>

New Analysis> Select Modal>

Click> OK

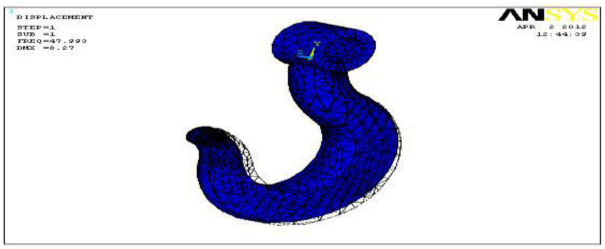
Main menu>Preprocessor>Loads>Analysis Type>

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

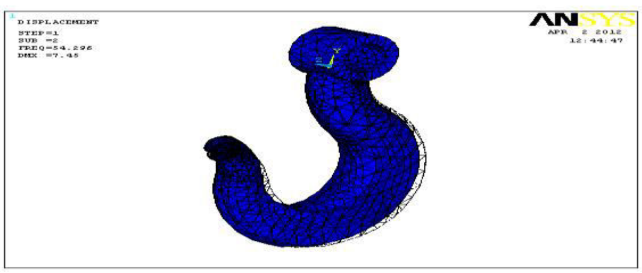
Analysis Options>
No. Of Modes to Extract: 3
Click> OK
Main menu>Solution>Solve>Current Ls>Ok

Results

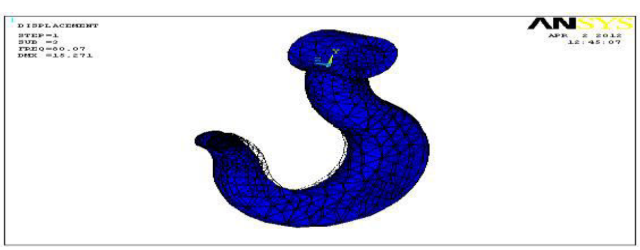
Main menu>General Postproc>Read Results> First Set
Plot result>Deformed Shape> Def+ Undeform > Click> OK



Main menu>General Postproc>Read Results> Next Set
Plot result>Deformed Shape> Def+ Undeform > Click> OK

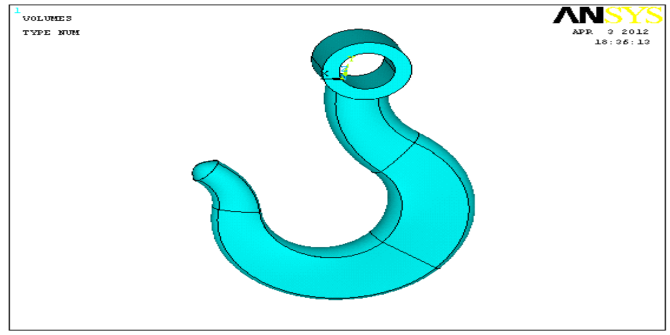


Main menu>General Postproc>Read Results> Next Set
Plot result>Deformed Shape> Def+ Undeform > Click> OK



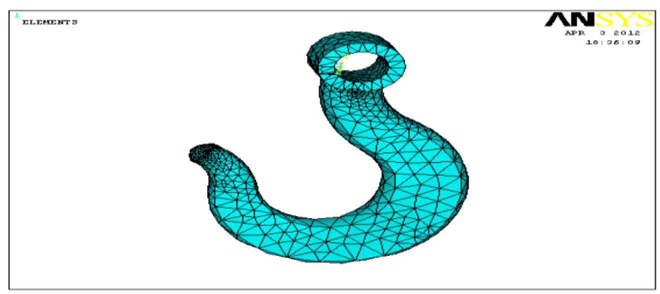
Structural Analysis of crane hook using Cast Iron of 2.8 tons

Imported Model from Pro/Engineer



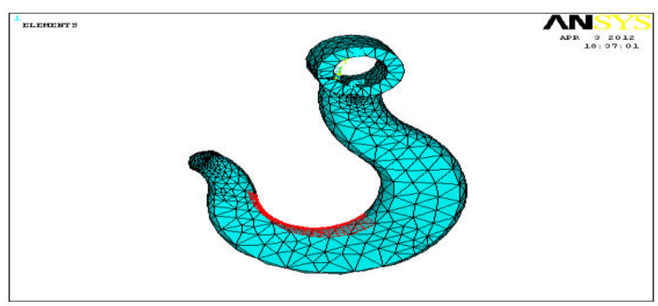
Element Type: Solid 20 node 95
Material Properties: Youngs Modulus (EX) : 103000N/mm^2
Poissons Ratio (PRXY) : 0.211
Density :0.000007 kg/mm^3

Meshed Model



Loads

Pressure – 14.9964N/mm^2



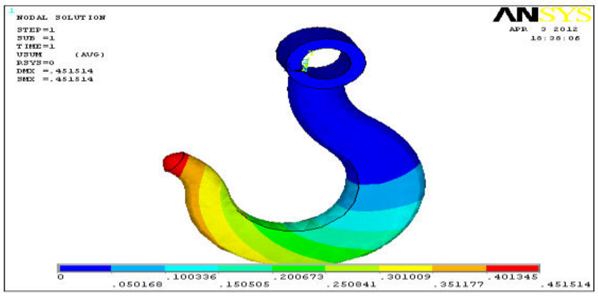
Solution

Solution – Solve – Current LS – ok

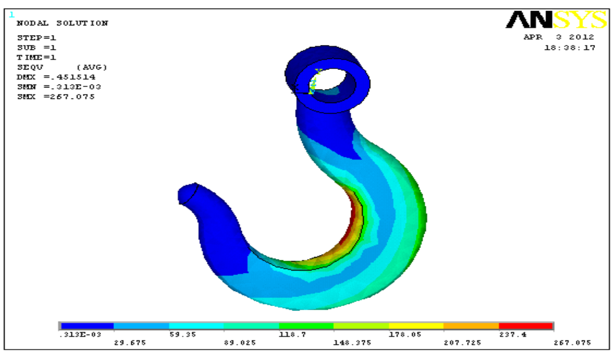
Post Processor

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

General Post Processor – Plot Results – Contour Plot - Nodal Solution – DOF Solution – Displacement Vector Sum

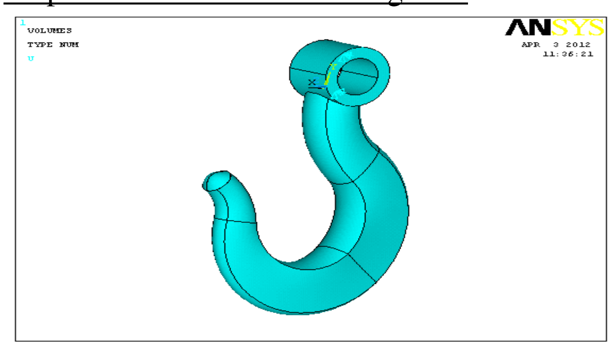


General Post Processor – Plot Results – Contour Plot – Nodal Solution – Stress – Von Mises Stress



MODAL ANALYSIS of crane hook using Cast Iron of 2.8 tons

Imported Model from Pro/Engineer



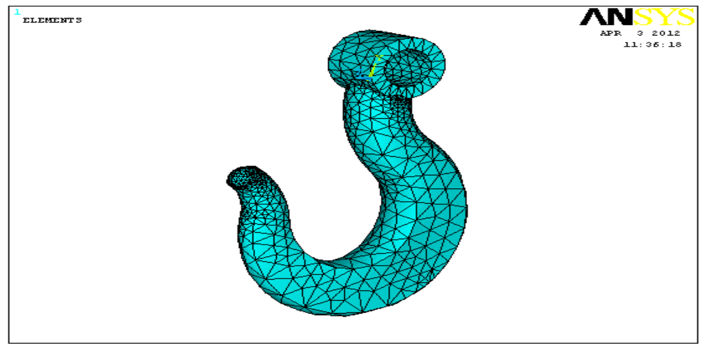
Element Type: Solid 20 node 95

Material Properties: Youngs Modulus (EX) : 103000N/mm^2

Poissons Ratio (PRXY) : 0.211

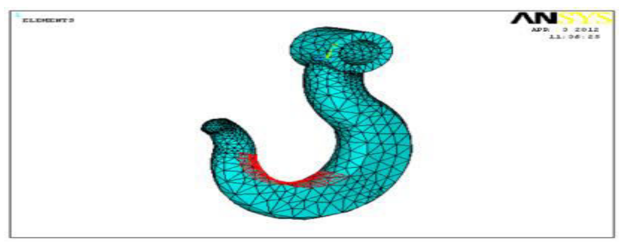
Density :0.0000071 kg/mm^3

Meshed Model



Loads

Pressure – 14.9964N/mm^2

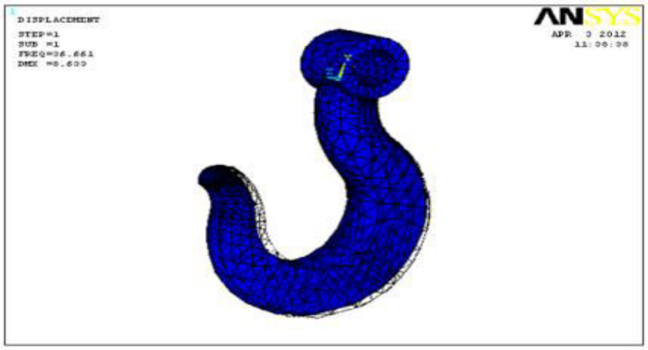


Main menu>Preprocessor>Loads>Analysis Type>
New Analysis> Select Modal>
Click> OK
Main menu>Preprocessor>Loads>Analysis Type>
Analysis Options>
No. Of Modes to Extract: 3
Click> OK
Main menu>Solution>Solve>Current Ls>Ok

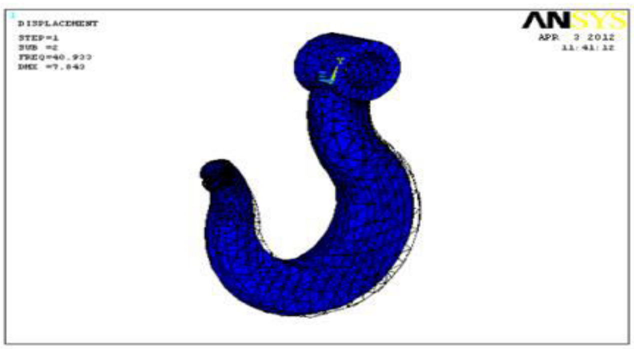
Results

Main menu>General Postproc>Read Results> First Set
Plot result>Deformed Shape> Def+ Undeform > Click> OK

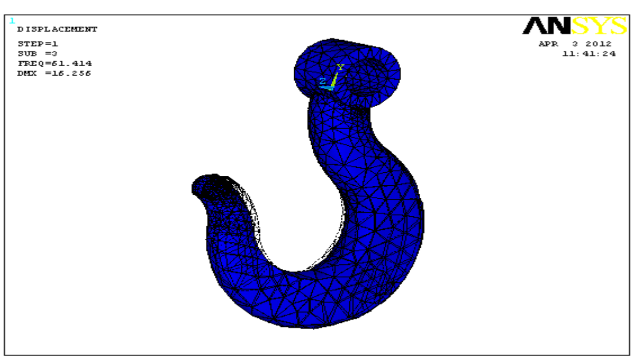
International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org



Main menu>General Postproc>Read Results> Next Set
Plot result>Deformed Shape> Def+ Undeform > Click> OK

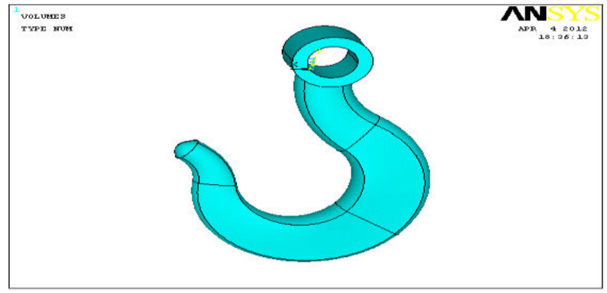


Main menu>General Postproc>Read Results> Next Set
Plot result>Deformed Shape> Def+ Undeform > Click> OK



Structural Analysis of crane hook using High Carbon Steel of 2.8 tons

Imported Model from Pro/Engineer



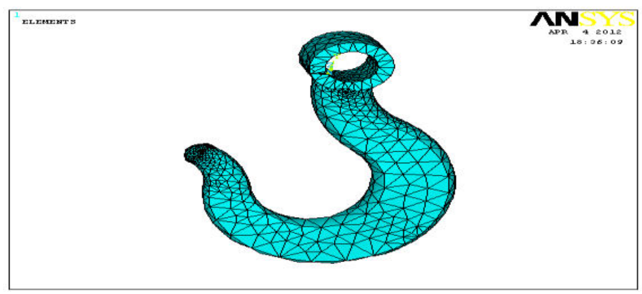
Element Type: Solid 20 node 95

Material Properties: Youngs Modulus (EX) : 200000N/mm^2

Poissons Ratio (PRXY) : 0.295

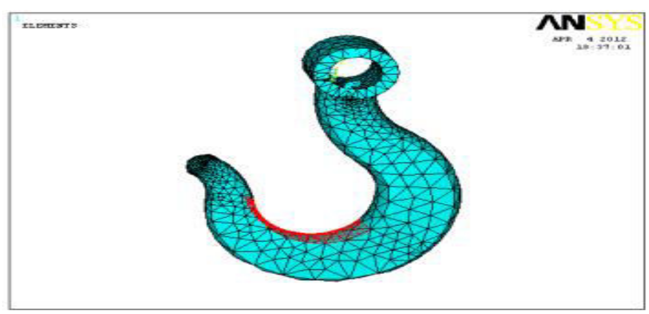
Density :0.000007872kg/mm^3

Meshed Model



Loads

Pressure – 14.9964N/mm^2



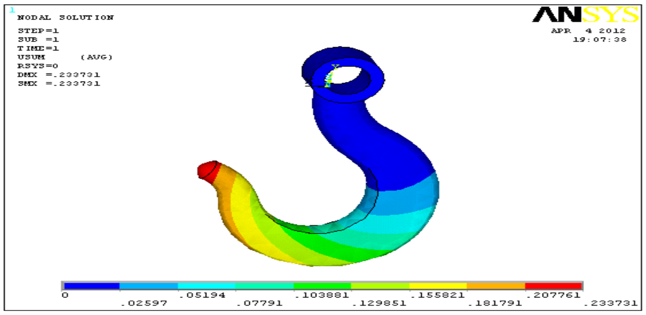
Solution

Solution – Solve – Current LS – ok

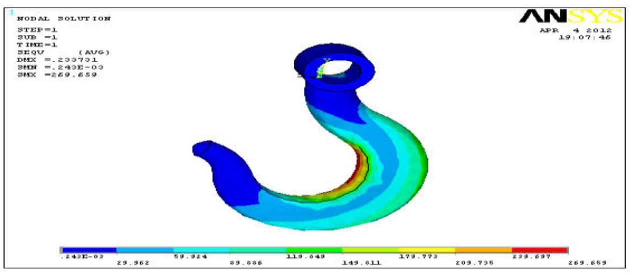
Post Processor

General Post Processor – Plot Results – Contour Plot - Nodal Solution – DOF Solution – Displacement Vector Sum

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

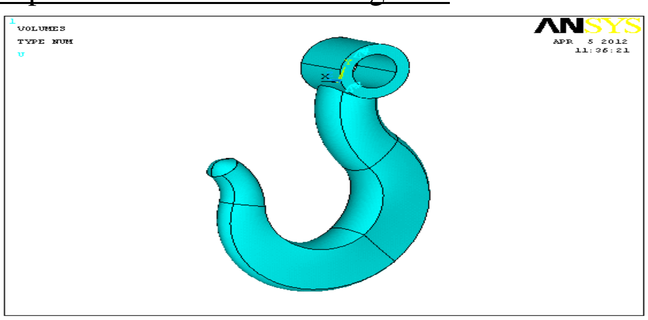


General Post Processor – Plot Results – Contour Plot – Nodal Solution – Stress – Von Mises Stress



MODAL ANALYSIS OF CRANE HOOK USING HIGH CARBON STEEL OF 2.8 TONS

Imported Model from Pro/Engineer



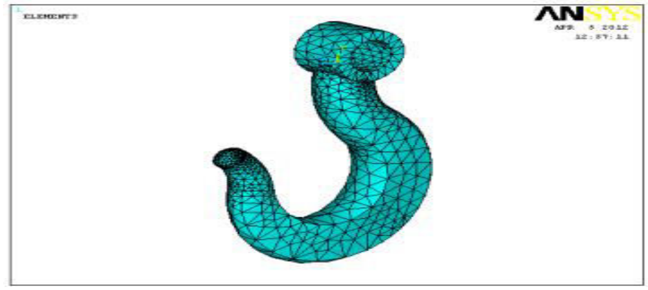
Element Type: Solid 20 node 95

Material Properties: Youngs Modulus (EX) : 200000N/mm^2

Poissons Ratio (PRXY) : 0.295

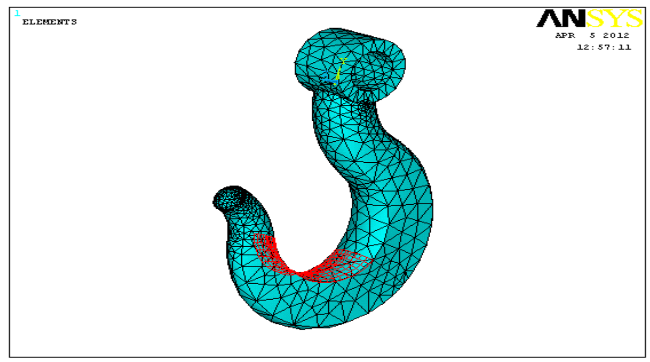
Density :0.000007872kg/mm^3

Meshed Model



Loads

Pressure – 14.9964N/mm^2



Main menu>Preprocessor>Loads>Analysis Type>

New Analysis> Select Modal>

Click> OK

Main menu>Preprocessor>Loads>Analysis Type>

Analysis Options>

No. Of Modes to Extract: 3

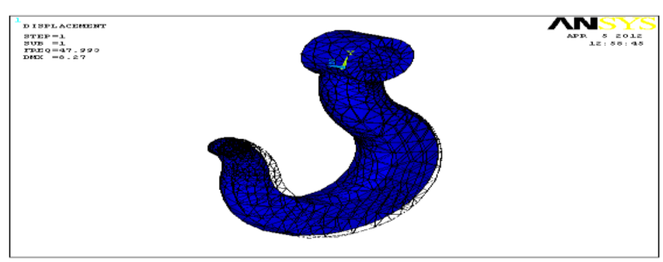
Click> OK

Main menu>Solution>Solve>Current Ls>Ok

Results

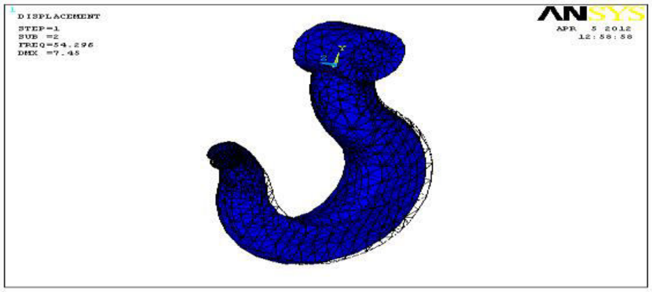
Main menu>General Postproc>Read Results> First Set

Plot result>Deformed Shape> Def+ Undeform > Click> OK

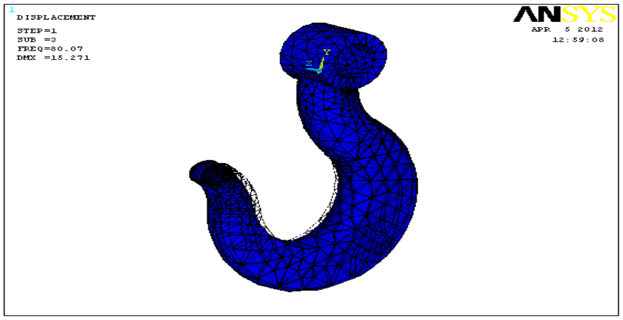


International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

Main menu>General Postproc>Read Results> Next Set
Plot result>Deformed Shape> Def+ Undeform > Click> OK



Main menu>General Postproc>Read Results> Next Set
Plot result>Deformed Shape> Def+ Undeform > Click> OK



RESULTS

CAST IRON WITH 2.5 TONS

	RESULTS	PERMISSIBLE
DISPLACEMENT (mm)	0.403019	
VONMISES STRESS (N/mm^2)	238.39	520

CAST IRON WITH 2.6 TONS

	RESULTS	PERMISSIBLE
DISPLACEMENT (mm)	0.419284	
VONMISES STRESS (N/mm^2)	248.011	520

CAST IRON WITH 2.8 TONS

	RESULTS	PERMISSIBLE
DISPLACEMENT (mm)	0.451514	
VONMISES STRESS (N/mm^2)	267.075	520

HIGH CARBON STEEL WITH 2.5 TONS

	RESULTS	PERMISSIBLE
DISPLACEMENT (mm)	0.208627	
VONMISES STRESS (N/mm^2)	240.696	285

HIGH CARBON STEEL WITH 2.6 TONS

	RESULTS	PERMISSIBLE
DISPLACEMENT (mm)	0.217047	
VONMISES STRESS (N/mm^2)	250.41	285

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

HIGH CARBON STEEL WITH 2.8 TONS

	RESULTS	PERMISSIBLE
DISPLACEMENT (mm)	0.233731	
VONMISES STRESS (N/mm^2)	269.659	285

CONCLUSION

In our project we have modeled a Crane hook which is used in material handling crane in work shop.

Structural and Model analysis are done on the crane hook by applying 2.5tons, 2.6tons and 2.8tons load on it.

The analysis is done by varying the materials Cast Iron and High Carbon Steel. By observing the results, the stress values obtained are less than their permissible stress values. So our design of crane hook is safe.

By comparing the stress values for both materials they are same for both, but the displacement values are less for High carbon Steel than Cast Iron.

So we conclude that High Carbon Steel is better for crane hook.

AUTHORS:

GARJAKUNTLA DHANRAJ
M Tech,
Department of Mechanical, CAD/CAM Branch.
Sreyas Institute of Engineering & Technology,
Nagole, Bandalaguda (vill),
Saroornagar(Mdl), R R Dist T.S.
Email: danraj.g@gmail.com

K. SAINATH
Associate Professor,
Department of Mech, CAD/CAM Branch.
Sreyas Institute of Engineering & Technology,
Nagole, Bandalaguda (Vill),
Saroornagar(Mdl), R R Dist T.S.
Email: Sainathkasuba@Gmail.Com