iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 8 Issue: IX Month of publication: September 2020

DOI: https://doi.org/10.22214/ijraset.2020.31348

Design and Analysis of Crane Hook with Catia and Ansys

G. R. Shinde[1] A. V. Patil[2]
[1]PG Student, [2]Associate Professor, Department of Mechanical Engineering, SSGBCOET, Bhusawal, Maharashtra, India

Abstract: A crane hook is a device used for lifting up the loads by means of a crane. Crane hooks with circular, triangular cross section, rectangular, trapezoidal are used commonly. The crane hook mostly subjected to failure due to accumulation of large amount of stresses. Failure of a crane hook mainly depends on three major factors i.e. dimension, material, overload. To minimize the failure of crane hook, the stress occurred in it must be studied. In this paper the design of the hook is done by analytical method for the different materials like
SAE-AISI 1040 and ASMT GRADE 60. CATIA software is used for modelling the crane hook and ANSYS software used to find out the stresses induced in it. This result helps us for determining of stress in existing model. By predicting the stress concentration area the hook working life increase and reduce the failure stress.
Keywords: ANSYS, FEA, CAD, CATIA

I. INTRODUCTION

Crane hooks are highly liable components and are always subjected to failure due to accumulation of large amount of stresses which can eventually lead to its failure. Crane hooks are the components which are generally used to elevate the heavy load in industries and constructional sites. A crane is a machine, equipped with a hoist. A crane hook is a device used for grabbing and lifting up the loads by means of a crane. It is basically a hoisting fixture designed to engage a ring or link of a lifting chain or the pin of a shackle or cable socket. Crane hooks with trapezoidal, circular, rectangular and triangular cross section are commonly used. So, it must be designed and manufactured to deliver maximum performance without failure. Crane hooks mostly employed in transport, construction and manufacturing industry. Overhead crane, mobile crane, tower crane, telescopic crane, gantry crane, deck crane, jib crane, loader crane are some of the commonly used cranes. A crane hook is a device used for grabbing and lifting up the loads by means of a crane. It is basically a hoisting fixture designed to engage a ring or link of a lifting chain or the pin of a shackle or cable socket

II. LIFTING HOOK TYPES, USES, AND DESIGN

A. Eye Hooks

On an eye hook, a chain or fittings are welded for a permanent connection to the sling. With an eye hook, you get far more flexibility in terms of movement and ergonomics to position the hook and attach it to the load. However, an eye hook is a permanent solution—if the throat of the hook becomes stretched, cracked, or bent during use, the whole sling would have to be failed out upon inspection and removed from service.

1. Eye Hook

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue IX Sep 2020- Available at www.ijraset.com

B. Clevis Hooks

A clevis fastener is a fastener system consisting of a clevis and clevis pin. The clevis is a U-shaped piece that has holes at the end of prongs to accept the clevis pin. The clevis pin is similar to a bolt, but is only partially threaded or unthreaded with a cross-hole for a split pin. A clevis hook is a hook, with or without a snap lock, with a clevis and bolt or pin at the base. The clevis is used to fasten the hook to a bracket or chain. Some rigging shops and end users who are not certified to weld alloy chain slings, utilize clevis hooks to make a mechanical connection to a chain sling.

The advantage of a mechanical connection is that if a clevis hook becomes damaged due to stretch, bending, or cracking, it can easily be removed and replaced without scrapping the entire chain sling. If this occurs on a chain sling, this is considered a repair to the sling and must be proof-tested prior to the sling being put back into service.

2. Clevis Hook

III. MATERIAL SELECTION

Generally crane hooks are made from wrought iron or steel. For heavy duty crane hooks low alloy steels are used but material is not the only factor behind its enormous load bearing capacity. Steel grade, heat treatment and forging are equally important to make a durable crane hook. Proper forging is very important. Forging provides better structural integrity than any other metal working processes. It eliminates any kind of defect such as gas pockets or voids in the hook which can affect its long term performance; thus increasing its strength, toughness, load bearing capacity and fatigue resistance.

For analysis and design consider as
SAE-AISI 1040 and ASMT GRADE
60.

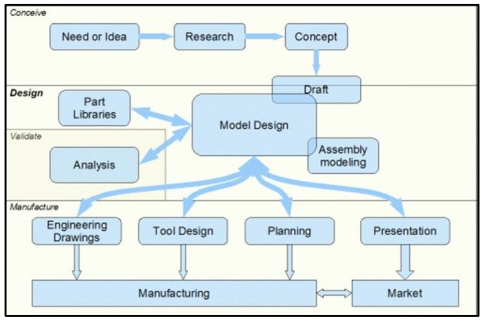
IV. PROPERTIES OF MATERIAL

Properties	SAE-AISI 1040	ASMT GRADE 60.
Tensile strength	620 MPa	415 MPa
Yield Strength	415 MPa	265 MPa
Density	7845 Kg/M^3	$7759Kg/M^3$
C %	0.37	0.18

V. CAD (COMPUTER AIDED DESIGNING)

A. Commuter Aided Manufacturing Procedure

Computer-aided design (CAD) is the use of computer systems to aid in the creation, modification, analysis, or optimization of a design. CAD software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing. CAD output is often in the form of electronic files for print, machining, or other manufacturing operations. Each stage requires specific knowledge and skills and often requires the use of specific software



3Commuter Aided Manufacturing Procedure

1) *Need or Idea:* Usually, the design process starts with a defined need. The need can be defined by market research, by the requirements of a larger body of work (for example airplane part). Sometimes, but more rarely than you may think, the design process is begun with a new idea or invention. At any rate, a needs analysis should precede any decision to undertake a project. This includes defining the need in a highly detailed way, in writing. This is similar to the requirements specification process in software engineering.
2) *Research:* Professionals tend to research available solutions before beginning their work. There is no need to "reinvent the wheel". You should study existing solutions and concepts, evaluating their weaknesses and strengths. Your research should also cover available parts that you can use as a part of your design. It is obvious, that Internet and search engines like Google are very helpful for this task. There are also many libraries of standardized parts which you can import into your project.
3) *Concept :* Based on your research, start with a high level concept. You should specify the main principles and major parts. For example, you can consider Diesel or Sterling engines for stationary electric generators.
4) *Draft:* You can choose to create a draft by pen and paper. Some prefer to use simple vector graphics programs, others even simple CAD (for example Smart Sketch), yet others prefer to start directly in their main CAD system.
5) *Model Design:* 2D and 3D modeling in CAD. The designer creates a model with details, and this is the key part of the design process, and often the most time consuming. This will be described in greater detail in further lessons. asa ceva
6) *Part Libraries:* Standard parts, or parts created by other team members, can be used in your model (you don't have to reinvent the wheel). Files representing a part can be downloaded from the Internet or local networks. They are also distributed on CD ROMs or together with CAD as an extension (library). By putting these predefined parts into your project, you ensure that they are correct and save a lot of time and effort. When working on a large project, this becomes a requirement to ensure the parts operate together, swap out equivalent parts, and coordinate distributed teams' work. This was, a standard part can be inserted into the project by one team member.
7) *About CATIA V5* CATIA (an acronym of computer aided three-dimensional interactive application) is a multi-platform software suite for computer-aided design (CAD), computer-aided manufacturing (CAM), computer-aided engineering (CAE), PLM and 3D, developed by the French company Dassault Systems. CATIA started as an in-house development in 1977 by French aircraft manufacturer Avions Marcel Dassault, at that time customer of the CADAM software to develop Dassault's Mirage fighter jet. It was later adopted by the aerospace, automotive, shipbuilding, and other industries.

VI. COMPUTERAIDED ENGINEERING (CAE)

A. *Finite Element Analysis (FEA)*

FEA is a numerical method. It is very commonly used in finding the solution of many problems in engineering. The problem includes deigning of the shaft, truss bridge, buildings heating and ventilation, fluid flow, electric and magnetic field and so on. The main advantage of using finite element analysis is that many designs can be tried out for their validity, safety and Studying or analyzing a phenomenon with FEM is often referred to as finite element analysis (FEA). The subdivision of a whole domain into simpler parts has several advantages: Accurate representation of complex geometry Inclusion of dissimilar material properties
Easy representation of the total solution Capture of loc

B. ANSYS

Ansys Inc. is an American public company based in Canonsburg, Pennsylvania. It develops and markets engineering simulation software. Ansys software is used to design products and semiconductors, as well as to create simulations that test a product's durability, temperature distribution, fluid movements, and electromagnetic properties.

Ansys was founded in 1970 by John Swanson. Swanson sold his interest in the company to venture capitalists in 1993. Ansys went public on NASDAQ in 1996. In the 2000s, Ansys made numerous acquisitions of other engineering design companies, acquiring additional technology for fluid dynamics, electronics design, and other physics analysis.al effects.

VII. GENERAL PROCEDURE OF FINITE ELEMENT METHOD

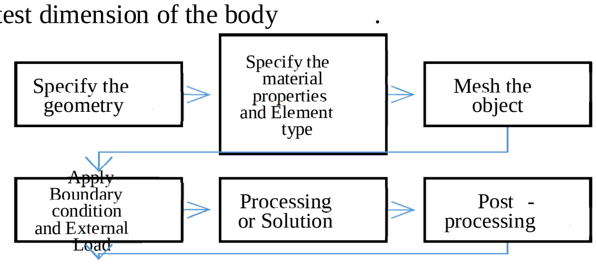
The discretization of the domain or solution region into sub-regions (finite elements) is the first step in the finite element method .The process of discretization is essentially an exercise of an engineering judgement. The shapes, sizes, number and configuration of the body have to be chosen carefully so that, the computational efforts needed for the solution are minimized. Basic Element Shape Mostly, choice of the type of element is dictated by the geometry of the body and the number of independent spatial co-ordinates necessary to describe the system .The element may be one, two and three dimensional When the geometry, material properties and other parameters (stress, displacement, pressure and temperature) can be described in terms of only one spatial co-ordinate, Hexahedron can also be used advantageously. The problems that possess axial symmetry like pistons, storage tanks, valves, rocket nozzles fall into this category. For the discretization of the problem involving curved geometries finite elements with curved size are useful. Discretization Process Type Of Elements- The type of element to be used will be evident from he physical problem itself , For example- If the problem involves analysis of a truss structure under a given set of load condition , the type of elements to be used for idealization is line or bar element Similarly for stress analysis of short Size Of Element: Generally, small sized elements gives accurate final solution but here the computational time increases. Sometimes, we may have to use elements of different sizes in the same body. In the stress analysis of a plate with hole stress concentration is expected around hole. Node Numbering Scheme:

Bandwidth (B)=(Maximum difference between the numbered degrees of freedom at the ends of any number +1) This can be generalized as,:

Bandwidth (B)=(D+1)*f

Where,

D=maximum largest difference in the nodes numbers occurring for all elements of assemblage, f=degrees of freedom at each node. For good results ,bandwidth should be minimum and this can be achieved by minimizing D,which in turn can be achieved by numbering the nodes across the shortest dimension of the body .



1) *Specify the Geometry:* In this import the geometry from CAD software to FEA software.
2) *Specify the Material Properties and Element Type:* In this step, the selection of element type is done and the material properties are given. The Young's modulus and Poisson's ratio are the input for material properties.
3) *Mesh the Object:* Here the object is broken in to small elements. This involves defining the type of element into which structure will be broken as well as specifying how the structure will be divided in to the element. This subdivision in to elements can either be input by the user or with same finite element programs can be chosen automatically.
4) *Apply Boundary condition and External Load:* This is followed by specifying the boundary condition and the external loads are specified.
5) *Processing or Solution:* The modified algebraic equations are 4.FEA Procedure solved to find the nodal values of the primary variable.

6) *Post-processing:* It involves improving the result of processing in to the model. These results are graphically displaced to enable user case of high deflection and stress.
7) *Refine the Mesh:* For the case of a judge of the accuracy of the result, there is need to increase or decrease no elements of an object.

VIII. RESULT

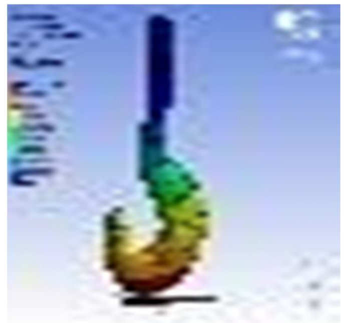
Below Table shows the comparative results between two materials As
ASMT Grade 60 & SAE-AISI 1040

Material	ASMT GRADE 60.	SAE-AISI 1040
Node	33293	33293
Element	19132	19132
Equivalent Stress	1.802 MPa	07921 MPa
Total Deformation	0.026 mm	0.0117 mm

5. Proposed Design

6. Meshing of Object

7. Loading Dia



8. Total Deformation

IX. CONCLUSION

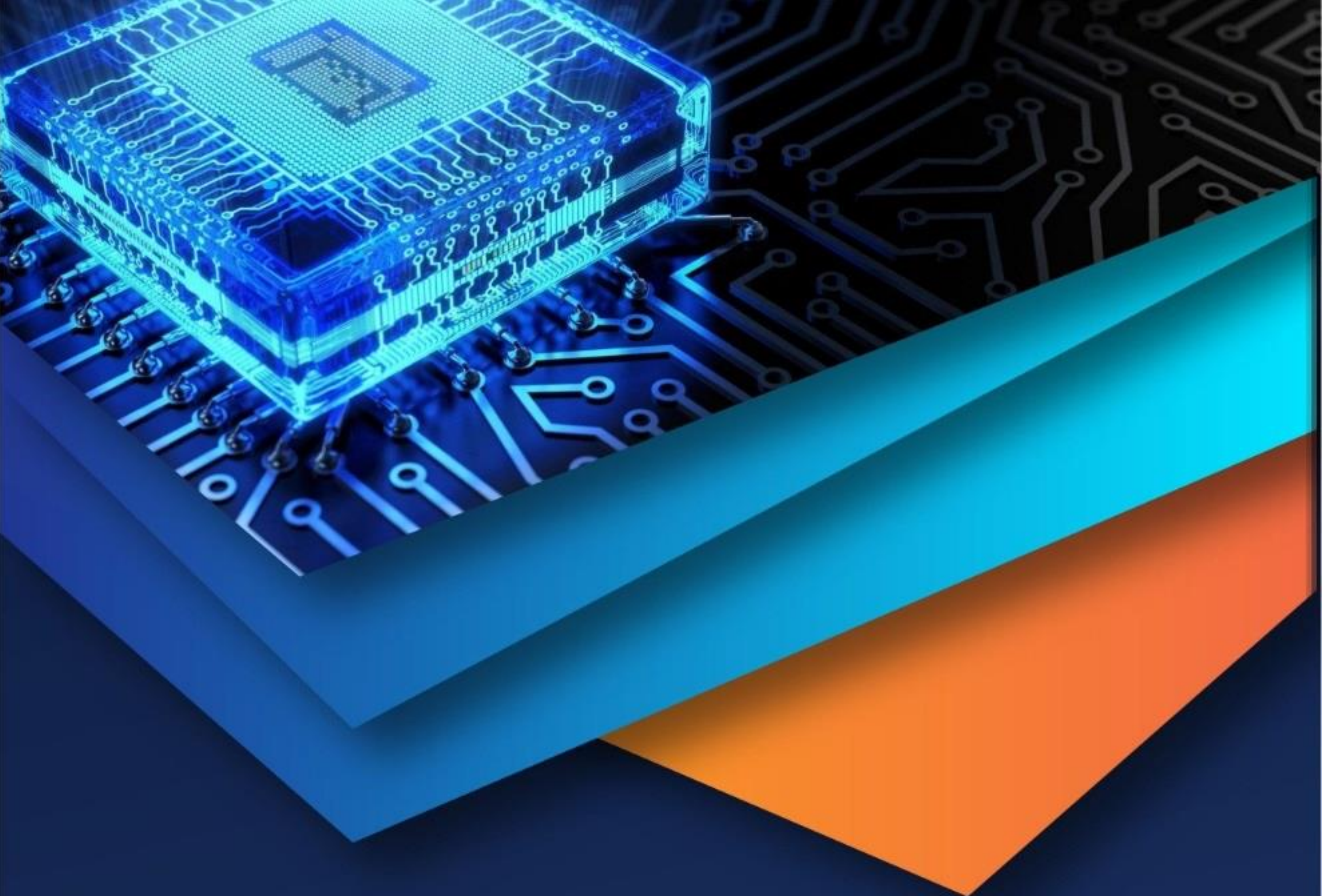
From above comparative result see that ASMT Grade 60 has higher stress generation as compare to SAE AISI 1040 ASMT Grade 60 has higher Deflection on same Loading condition and Boundary condition As from result we conclude that the SAE – AISI 1040 has Higher load Sustaining capacity as compare to ASMT Grade 60Also it is more suitable for Making Crane Hook The manufacturing process and Design may be change the actual result also

X. ACKNOWLEDGEMENT

We take opportunity to thanks Prof. A.V. Pati, H.O.D.
MechEngg. Dept. for his valuable guidance and providing all the necessary facilities which were indispensable In completion of this work. We are also thankful to Dr.R.B.Barjibhe, Dean A & A to give us Presentation facilities. We are also thankful to all staff member of the mechanical engineering. We are also thankful of the college for providing required journals, books and asses in the library.

REFERENCE

[1] L.S. Srinath, Advanced Mechanics of SolidsPage no. 209213, Tata McGraw-Hill Education 3rd Addition, 2009, ISBN-13:9780-07-013988-6
[2] Nishantsoni, "crane-hook shapes optimization and thermal analysis using finite element tool" in international journal of advanced and innovation research", ISSN: 2278-7844.
[3] Rashmi Uddanwadiker Scientific research Engineering, 2011, 3, 935-941 Published Online September 2011 Stress Analysis of Crane Hook and Validation by Photo- Elasticity. Ajeet Bergaley, Anshuman Purohit International Journal of Science and Modern engineering (IJISME) ISSN: 2319- 6386, Volume-1, Issue-10, September 2013 Structural Analysis of Crane Hook Using Finite Element Method.



10.22214/IJRASET

INDEX COPERNICUS 45.98

IMPACT FACTOR: 7.129

TOGETHER WE REACH THE GOAL

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY