JIMMA UNIVERSITY
Jimma Institute of Technology
Faculty of Mechanical Engineering

Thesis Title: Conceptual Modelling and Structural Optimization Framework of Crane Hook, A case study of the tower crane hook at Afro-Tsion construction site.

A Thesis Submitted to the School of Graduate Studies of Jimma University in Partial Fulfilment of the Requirements for the Degree of Master of Science in Mechanical Engineering (Design of Mechanical System)

By: Firankor Teshome Daba

Jimma, Ethiopia

May 2020

JIMMA UNIVERSITY

Jimma Institute of Technology

Faculty of Mechanical Engineering

Thesis Title: Conceptual Modelling and Structural Optimization Framework of Crane Hook, A case study of the tower crane hook at Afro-Tsion construction site.

By: Firankor Teshome Daba

Advisor: Prof. Hirpa G. Lemu (Ph.D.).
Co-Advisor: Mr. Yohanis Dabesa (MSc)

Jimma, Ethiopia
May 2020

Declaration

I declare and affirm that this thesis is my very own work. All ethical and technical principles have been followed while conducting the literature review of this thesis on conceptual modeling of topology and shape optimization framework. Any scholarly matter that is included in the thesis has been given recognition through proper citation.

Firankor Teshome ____________________ ______________________

Signature date

This thesis has been submitted for examination with our approval.

Hirpa Lemu (Prof.) ____________________ May 17, 2020

Advisor signature date

Mr Yohanis Dabesa (MSc) ________________________ ____________________

Co-advisor signature date

Approval

Name: **Firankor Teshome**

Degree: **Masters of science (MSc)**

Title: **Conceptual Modelling And Structural Optimization Framework Of Crane Hook, a case study of the crane hook at Afro-tsion construction site.**

Mr. Yohanis Dabesa (MSc). ______________________ ____________________

Chair signature date

Prof. Hirpa Lemu (PhD) ____________________ **May 17, 2020**

Main Advisor signature date

Mr.Yohanis Dabesa (MSc). ______________________ ____________________

Co-Advisor signature date

Examining committee

Dr Getachew Shunki (Dr;Ing) ____________________ **23/06/2020**

External Examiner Signature date

Dr Mesay Alemu (PhD) ______________________ ____________________

Internal Examiner signature date

ABSTRACT

Tower cranes are widely utilized in heavy industries and construction sites to lift and carry heavy materials. So, to save life, time, and cost, the right design of the crane has an important role within the safe operation of construction sites. A crane is subjected to continuous loading and unloading condition that exposes the crane components to fatigue failure, among others, the crane hook and causes serious accidents because most construction sites are very confined and shut to the general public, tower crane accidents not only hazard to workers in construction sites but also pedestrians within the vicinity. The main objective of this study is to develop the conceptual framework of design optimization and then apply topology optimization designxplorer to enhance the strength and endurance requirements of the crane hook. In order to fulfil strength and endurance requirements on the tower crane hook, optimization is a useful tool to predict an optimal design in the early phases of the design process. In this thesis work, a completed design of a hook constructed and topology and shape optimization has been conducted on the crane hook. Topology optimization is an optimization technique that employs mathematical tools to optimize material distribution in a crane hook to be designed. So topology optimization is conducted on the crane hook without sacrificing strength and durability. The topology optimization conducted as part of this thesis reduced the mass by 6.685% (mass reduction, of crane hook from 15.75 kg to 13.678 kg). Simulation of the hook was done using the topological approach, where the model was created, and then meshing was done and FEA analysis (ANSYS 19.2) was carried out. The core part of the work done in this thesis includes Parameterized finite element analysis, designxplorer, and Topology optimization. Designxplorer is a simulation tool or module in ANSYS workbench that is implemented using surface response sensitivity and design of the experiment to define the input and output relationship. These input parameters decide the surface of the crane hook and the output relation minimum factory of safety and maximum equivalent stress decide the life and strength of crane hook. The main contribution of this study is to investigate the possible methods of optimizing the strength and endurance required of the crane hook.

Keywords: *conceptual modeling, crane hook, topology and shape optimization, design optimization, surface respone sensitivity.*

ACKNOWLEDGMENT

Almighty God thank you for giving me such an opportunity. Next, it is my pleasure to express the deepest appreciation of his excellency to **Prof. Hirpa G. Lemu (Ph.D., Professor of Mechanical Design Engineering at the University of Stavanger, Norway)** for exposing me to such indispensable work. Without his continuous support and guidance, as well as his encouragement during difficult periods the completion of this thesis work would not possible.

Then, I wish to express heartfelt thanks to my co-advisor **Mr Yohanis Dabesa (MSc)** Program Coordinator at the School of Mechanical Engineering, **JiT** and. my Co-Advisor for his careful review, comments, and suggestions with regards to this thesis.

And I would like to acknowledge and appreciate all my friends and teachers who have been played a valuable role in my work by sharing supportive materials, data, information, and concrete ideas when I get confused and lost. The last, but not least, I thank my family, for all their help and understanding. Without your encouragement, love, and support, this work would never have been accomplished.

TABLES OF CONTENT

LIST OF FIGURES

LIST OF TABLE

ACRONYMS

2D, 3D	Two dimensional, three dimensional
3D-CAD	Three dimensional computer aided design
AISI	American Iron and Steel Institute
ASM	Aerospace specification metals
CAD	Computer aided design
CM	Conceptual modelling
DOE	Design of experiments
DOF	Degree of freedom
DX	DesignXplorer, Design explorer
EOT	Electric overhead traveling
FEA	Finite element analysis
FEM	Finite element method
GDO	Goal driven optimization
KLR	knowledge level reflection
KADS	knowledge acquisition and design of knowledge design system.
M&S	Modeling and simulation
ODE	Ordinary differential equation
Opti-struct	Optimization of structure.
RSM	response surface modelling
SADT	Structured Analysis and Design Technique
S-N	stress verse cycle to failure
TO	Topology optimization
UTM	Universal Testing Machine.
ISO	International organization for standardization

SYMBOLS

A	Area of the cross-section
E	Young Modulus
C_i	Distance from the neutral axis to the inner radius of the curved beam,
C_o	Distance from the neutral axis to the outer radius of curved beam
C_L	Centre Line of Curvature
M_b	Applied Bending Moment
r_i	Inner radius of curved beam
r_o	Outer radius of curved beam
r_c	Radius of centroid axis
r_n	Radius of neutral axis
ρ	Initial radius of curvature of the centroid surface
e	Distance of the centroid from natural axis
Y	Distance of Neutral Axis
σ	Stress,
σ_a	Stress amplitude,
σ_x	Direct stress
ε	Strain
$\tau_{r\theta}$	Shear stresses
σ_{bo}	Bending stress in outer part
t	Thickness
h	Height of the trapezoidal cross section
b	Base
P	Load,
σ_{rr}, $\sigma_{r\theta}$	Radial stress. Circumferential stress

Cs	Surface factor
C_G	Gradient factor
C_L	Load factor
Cr	Reliability factor
Se	Endurance limit
N	Number of cycle
b_1	Width (inner side of trapezoidal cross-section),
b_2	Width (outer side of trapezoidal cross section),
v	Poisons ratio

CHAPTER. 1 INTRODUCTION

In this chapter, a brief introduction of the conceptual modeling of the topology optimization framework with a case study of crane hook is presented. The statement of problem, objective, research, methodology, scope, and limitation of the study are explained afterward and at the end this chapter the structure of the research is provided.

1.1 Problem Background

Material handling equipment is the mechanical equipment that involved in the complete system that relates to the movement, storage, control, and protection of materials, goods, and products throughout the process of manufacturing, distribution, consumption, and disposal. Nowadays, in heavy industries and construction sites, tower cranes are used to lift and carry heavy materials, so to save time and cost also the Tower crane has a major role in the safe operation of construction sites (Babu & Rao, 2015).

The device uses one or more simple machines to create mechanical advantage and thus move loads beyond the normal capability of a human. Cranes are commonly employed in the transport industry for the loading and unloading of freight, in the construction industry for the movement of materials, and in the manufacturing industry for the assembling of heavy equipment (Pavlovic, 2018).

Cranes were used for the construction of tall buildings. Larger cranes were later in the High Middle Ages, harbor cranes were introduced to load and unload ships and assist with their construction, and some were built into stone towers for extra strength and stability. The earliest cranes were constructed from wood, but cast iron, iron, and steel took over with the coming of the Industrial Revolution (Bhise & Deshpande, 2018).

A crane hook is a device used for grabbing and lifting the loads by means of a crane. It is 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-sections are commonly used. So, it must be designed and manufactured to deliver maximum performance without failure (Kankotiya & Modh, 2016).

A hook block allows for a considerable amount of flexibility and safety in lifting operations as opposed to a direct connection. One of the most important functions of any hook block is facilitating a free turning or rotating hook arrangement. When loads are lifted, it is often necessary to turn the load to position it in a new location or to avoid striking obstructions. A crane hook attached directly to the hoist ropes would cause the ropes to twist if the load was turned from its original orientation. This would have several undesirable effects such as over-stressing the ropes and boom pulleys, creating an unbalanced load, and causing the load to swing back in an uncontrolled fashion when released. A hook block allows loads to be freely rotated without changing the orientation of the hoist ropes. To minimize the failure of crane hook, the stress-induced in it must be studied. A crane is subjected to continuous loading and unloading(Kumar & Prasad, 2014).

Crane hooks are highly liable components and are always subjected to premature failure due to the accumulation of unwanted stresses which can eventually lead to its failure. Crane hooks are the primary components used in industries and constructional sites to elevate the heavy load. 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. Catastrophic failure of crane hook is one of the main reasons for the industrial disaster. The hook failed from a step where there was a cross-sectional change; rough-machining marks and chatter marks were observed near the location of failure (Fig.1.1). Such locations involving cross-sectional change are the potential sites of stress concentrations leading to crack initiations(Mukhopadhyay & Souvik, 2018).

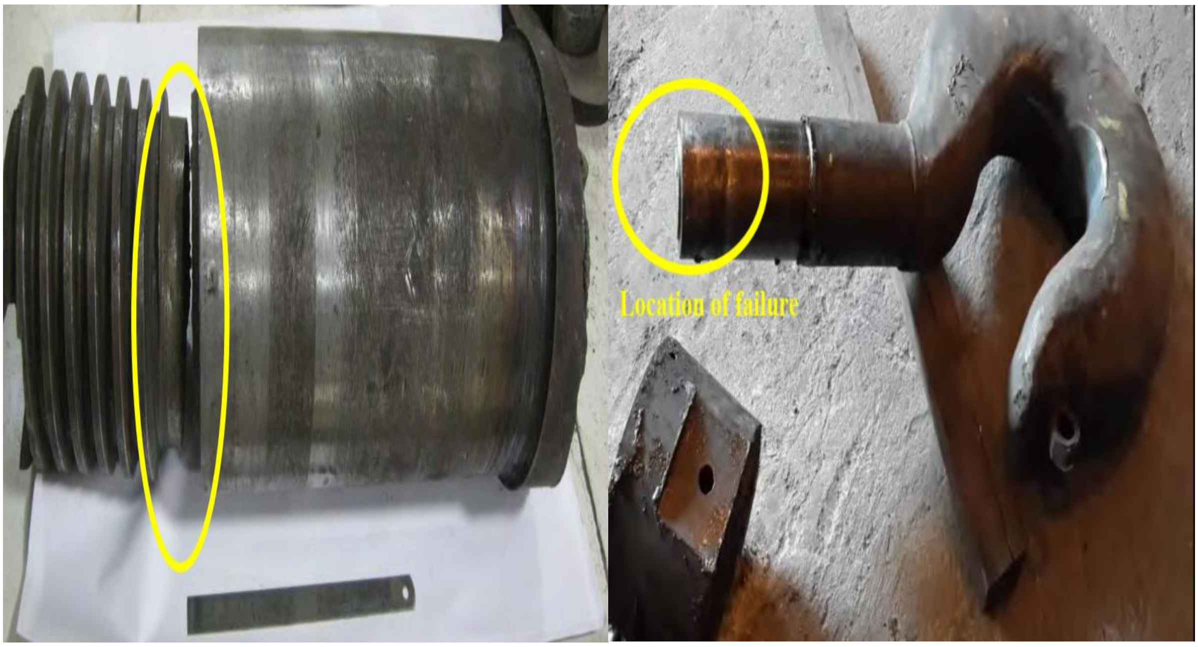


Figure 1.1 Overall view failure location of failed crane hook(Mukhopadhyay & Souvik, 2018).

Failure of a crane hook mainly depends on three major factors i.e. dimension, material, overload and working condition.

The project is concerned with 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 development of a hook is a long process that requires the number of tests to validate the design and manufacturing variables. A systematic procedure is obtained where CAE and tests are used together. In fact, their use has enabled the engineers to reduce product development cost and time while improving the safety, comfort, and durability of the crane hook they produce. The optimized geometry is analysed using FEA tool. The stress concentration factors are widely used in strength and durability evaluation of structures (Prashant, 2015).

1.2 Problem Statement

Nowadays, in heavy industries and construction sites, tower cranes are used to lift and carry heavy materials. So, to save time and cost also the tower crane has a major role in the safety of constructions. A crane is subjected to continuous loading and unloading conditions. These will cause fatigue failure of the crane hook and lead to serious accidents. Most construction sites are very confined and close to the public, as a result tower crane accidents are not only hazardous for the workers in construction sites, but also for other pedestrians. From those disasters caused by the cranes, hook failure leads to the destruction of the construction apparatus, people hit by the heavy falling-down components, and induces failure of subordinate structures.

Several researchers have proposed different design solutions and analysis methods in order to solve the challenges, however, the problem still exists and accidents happen at many construction sites. Some have proposed, traditional failure criteria, which depends on maximum stress concentration or strain energy density, and also weak design, wrong handling, and problems during the manufacturing process, material problems. But these can’t justify failure of many structures of crane hook because of the stress induced due to repetitive loading and unloading conditions.

To reduce the failure of the crane hook, the stress-induced in the hook must be analysed to reduce the stress as much as possible or improve the strength of the hook material. Therefore, the objective of this study is to address the problem of crane hook failure and overcome this through structural and topology optimization with developing the conceptual framework to improve the strength of crane hook, as well as making the primary focus on engineering tools and implementing simple optimization strategy.

1.3. The objective of the study

1.3.1 General objective

The general objective of this research is to develop the conceptual framework of the tower crane hook to improve strength and endurance requirements.

1.3.2 Specific Objectives

The specific objectives will include;

- To develop a conceptual framework with topology optimization.
- To analyze load distribution in the crane hook.
- To parameterize crane hook geometry by using ANSYS19.2 and SoildWorks2018.
- To analysis optimized parameter by Conducting Design explorer and FEM analysis for structure optimization.

1.4. Significance of the study

The importance of this study is to improve the strength of the crane hook which means to make stronger, light, and safe weight of crane hook as well as to save the waste of material for manufacturing companies. And to establish conceptual modeling for crane hooks topology and shape optimization framework in early phases of the product development process, the methodology will include optimization related subjects such as objective function, constraints and also so-called manufacturing constraints for both the topology and shape optimization of the crane hook. Furthermore, load distribution cases and boundary conditions which are relevant to include with respect to the shape optimization process are treated. Then, the other important element of this study is to implement DesignXplorer topology optimization and parameterized finite element analysis by the use of software packages (ANSYS and SolidWorks). In addition to this, the contribution of study can serve as a baseline or reference for other researchers, either individual or company, who want to investigate crane hook later. In addition, an optimized crane hook will reduce injury and damage or life loss.

1.5. Scope and limitation of the study

1.5.1 The scope of the research

The scope of this thesis is constrained to focus on developing the conceptual modeling, of crane hook structural optimization with improving the strength and endurance of hook. In general, this study focuses on Conceptual modeling of topology and shape optimization frameworks using a case study on crane hooks. Improving the strength and endurance of the crane hooks load distribution on shape effect. Optimize the crane hook with means of topology optimization by using software packages and Other optimization and analysis methods are not considered in this study.

1.5.2 Limitation of the Research

The major limitation of the work of this thesis is the results of the analytical analysis and FE analysis will be compared. Due to lack of sufficient laboratory and expensive material cost, the results are not verified by experimental results. And lack of getting the licensed software packages. The student version software provided by companies doesn't provide full analysis. Basic analysis and simulation can be carried out in the student version, not an advanced simulation. Student version software was used in this thesis for modeling and simulation.

1.6 Research Methodology

This study is concerned with improving the strength and endurance of crane hook with a conceptual design framework based on theoretical works and principles reviewed of literature design optimization tools. To address the above-mentioned objective and achieve results, the following research methodologies were employed

- Literature review to identify the cause of failure and optimization of the crane hook.
- The Mathematical expression of a three-dimensional curved beam equation.
- 3-D modeling by SolidWorks 2018, FEM and DX analysis in ANSYS 19.2.
- Parameterize via finite element analysis and load distribution of the crane hook.
- Improving the strength and topology optimization of the hook.

A schematic flow chart given in Fig.1.2 shows the methodology of the research work.

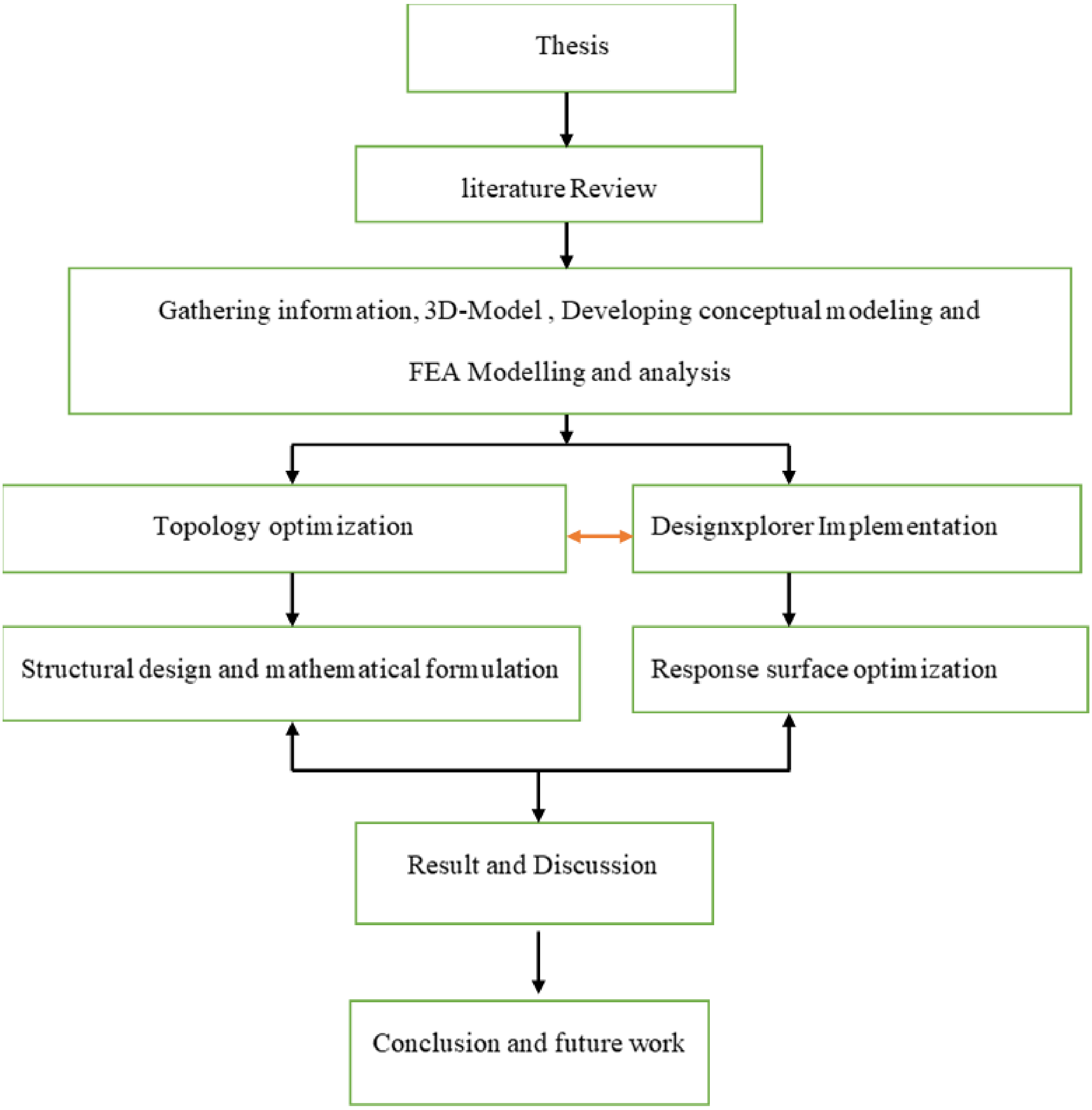


Figure 1-1 Schematic flow chart of the Research methodology

1.7. Structure of the Thesis

This thesis is organized into six chapters, following the introduction part in Chapter 1, the previous work related to this study that highlights the cause of failure in crane hook, optimization, and improving the crane hook, classification including their advantages are reviewed and presented in Chapter 2.

In Chapter 3, the material assessment and methodologies which related to improving crane hooks. The modeling the 3D, of crane hook and design, structural analysis crane hook with theory curved beam, and structural optimization with topology optimization. theory based on analysis of the load distribution and deformation, stress analysis, derivation, circumferential stress in the curved beam, and elastoplastic analysis in a curved beam.

In Chapter 4, the theoretical approach, and DesignXplorer implementation with optimization computational experimental like the design of experiments (DOE), surface response implementation and goal derive optimization with ANSYS19.2 module optimization. The theoretical approach provides parametric analysis using finite element analysis, hook parameterize model force analysis, and procedure for finite element analysis and procedure parametric optimization analysis is investigated in this chapter, also presents in this chapter.

Lastly, Chapter 5 and Chapter 6 followed by results and discussion then the conclusion and recommendation of the thesis are summarized for its future works. Reference and appendices are given at the end of the document.

CHAPTER. 2 LITERATURE REVIEW

This chapter provides a review of the literature from the various previous works of researchers, which are published in books and articles. The literature review works are intended to get indications on previous work of the implementation of design optimization and fatigue failure analysis of cranes and their hooks to enhance the strength and endurance requirements. A summary of the crane hook, failure, and optimization crane hook and conceptual modeling are discussed in the chapter.

2.1 Crane hook

Crane is a useful and frequently used equipment that has a wide, global application. The construction of large and tall structures is impossible without the use of a crane. In most building construction, tower cranes are used to lift and move payloads. Payloads always have a tendency to sway about the vertical position under excitations (Hamid, 2014). The material of the crane is modified to increase its working life and reduce failure rates.

A crane uses one or more simple machines to create mechanical advantage and thus move loads beyond the normal capacity of a human. Cranes are commonly used in the transport industry for the loading and unloading of goods, in the construction industry for the movement of materials and in the manufacturing industry for the assembling of heavy equipment. It is a specially designed structure equipped with mechanical means for moving a load by raising or lowering by electrical or manual operation. Cranes can range in capacity from a few hundred pounds to several hundred tons (Chunkawan & Subramaniyam, 2017).

Crane hooks are one of the important components used in industries to carry heavy loads basically, crane hooks are designed with pulley elongated by rope or a chain. Analytical calculation of crane hooks is based on the curved beam theory (Joseph & ArutPranesh, 2015). where the mathematical calculations are done by varying cross-sections of triangular, trapezoidal, and rectangular sections with constant cross-sections area for two different materials of steel, cast-iron analysis is evaluated. Due to the accompanied calculation inaccuracies, however, the FEM is widely recommended.

As previously described, the crane hook is used to transfer the materials having heavy loads. A hook is a tool consisting of a length of material that contains a portion that is curved or indented, so that this portion can be used to hold another object. In a number of uses, one end of the hook is pointed. The Real-time pattern of stress concentration in the 3D model of the crane hook is obtained. Crane hook is one of the main and important components of cranes basically a hoisting fixture designed to engage a ring or link of a lifting chain or the pin of a shackle or cable socket and must follow the health and safety guidelines. Crane hook is a curved beam and is used for lifting loads in cranes. Crane hook is the component that is generally used to elevate the heavy load and transfer it from one place to another in industries, factories, and constructional sites (Ankit, 2017).

2.1.1 Types of crane hooks

Crane hooks can be classified according to their shapes, method of manufacture, mode of operation or other unique characteristics [I].They are made in a variety of styles to meet specific needs and they are rated for loads of specific type and size [II].

I. Types of crane hooks based on their shape

Based on their shapes, there are two types of crane hooks which are (a) single crane and (b) double crane hooks. As the name suggests, the main difference between these two options is the number of hooks included, and there are different sub-types possible such as the C-hooks (which is essentially a single hook variant with a slightly different shape). Figures 2.1 and 2.2 show single and double crane hooks respectively.

a) Single Crane Hooks are the right choices if the machinery deals with loads of up to 75 tons. This lifting hook is very simple and easy to use no matter which variant one chooses.

Figure 2. 1 Single crane hook. (G, DONGQL, 2019)

b) Double Crane Hooks are similar in concept with single crane hooks, but their design provides superior bearing which is suitable for heavier loads of over 75 tons. A Ramshorn hook is a shank hook with two throat openings, sometimes called sister hooks or twin hooks. Commonly they are used in applications with shipyard cranes and container cranes. There are two types of Ramshorn hooks: The Ramshorn Form A hook, which has a solid lower hook design and the Ramshorn Form B hook which have a hole at the lower hook design. The hole of a hook is used to attach rigging.

a) b)

Figure 2. 2 a) Ramshorn from crane hook, b) Ramshorn form hook with a hole at the lower (Elebia.com, 2019).

II. Types of crane hooks based on their manufacturing processes

When it comes to the manufacturing methods used to create crane hooks there are two main styles available; **(a)** forging crane hooks and **(b)** laminated crane hooks, where each option offers particular benefits and drawbacks that must be kept in mind while deciding which is better suited for specific requirements.

a) Forging Crane Hooks are defined as a metalworking process in which the useful shape of the workpiece is obtained in the solid-state by compressive forces applied through the use of dies and tools, then forged from a single piece of high-quality steel with low carbon which is cooled slowly to ensure optimum stress resistance. These hooks typically feature very simple manufacturing and are also very simple to use, since they're made of a single piece, with installation being very straightforward.

b) Laminated Crane Hooks are a little more complex and are comprised of steel plates riveted together to ensure higher stress resistance and increased safety. These hooks are conceived in a way that allows the hook to keep working even if some of the individual crane hook parts are damaged. Laminated hooks are available as single hooks or double laminated hooks, and there are several models available to match different applications. Figure 2.3 indicates the single laminated crane hook. Laminated hooks are widely used in the steel industry. Due to its high load-carrying capacity, they are used for loading and unloading of material in the heavy steel industry (Lanjekar & Patil, 2016).

Figure 2. 3 a) forging crane hook, b) Laminated Crane Hook (Elebia.com, 2019)

Generally, crane hooks are made from wrought iron or carbon steel. For heavy-duty crane hooks, low alloy steels are used but the 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 metalworking process. It eliminates any kind of defects 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.

Carbon steel is a popular material for the production of the crane hook. It is available in numerous grades and can be heat-treated to improve its strength, ductility, and machinability. There are four main grades of carbon steel: low carbon steel, medium carbon steel, high carbon steel, and very high carbon steel. Depending on the amount of carbon present in the material, carbon steel forgings are hard enabled by heat treatment to increase yield and impact strength as well as wear resistance. The material cost of carbon steel forging is relatively much lower than other steel forgings, especially compared with stainless steel forging.

The load is usually handled by means of a chain or rope slings attached to the hook. There are two most popular design hooks which are standard (single) and Ramshorn (double) hooks. Standard and Ramshorn hooks may be flat-die or close die forged or else made of a series of sharped plates. One piece forged hook is used to lifting loads up to 100tons. Mostly hooks are forged from low carbon steel. In the process of production, hooks are carefully annealed after forging and machining (Sharma, 2013).

2.2 Failure of the crane hook

Due to the continuous working of crane hook nanostructure of crane hook are changes and some problems like weakening of hook due to wear, tensile stresses, and excessive thermal stresses these are some other reasons for failure. Hence continuous working of crane hook may increase the magnitude of these stresses and eventually fail the crane hook. Due to some design modifications, all the above-mentioned failures may be prevented (Sharma. 2013).

The causes of failure are stress concentration, the material of hook, manufacturing process of crane hook, wear, plastic deformation due to loading, and continuous use of crane hook may accelerate the failure of the crane hook. Crane hooks are liable components subjected to failure due to stress in the accumulation of heavy loads. The area of cross-section, material, and radius of the crane hook is the design parameters for crane hook (Guo., 2015; Patel., 2015). Failure of a crane hook mainly depends on three major factors i.e., dimension, material and overload. Besides, working conditions can influence the failure condition.

To minimize the failure of the crane hook, the stress that caused it must be studied. Structural failure of the crane hook may happen as a crane hook is subjected to continuous loading and unloading (Joseph & ArutPranesh, 2015). Under this investigation, fatigue damage is considered to be initiated by a crack due to fluctuating loading. It is caused due to stress levels which are insufficient to cause damage in a single application. In 2013, Sharma studied the stress, deformation, and fatigue life contour plots of crane hook using ANSYS Workbench (Sharma., 2013).

The contribution deals with full-field stress analysis of the crane hook model. The investigation was realized using numerical as well as experimental analysis. In numerical analysis, the software based on the finite element method was used. Experimental analysis was performed in laboratory conditions via a non-contact optical method of digital image correlation.

(Hagara & Pástor, 2017) briefly described the methodologies of both, experimental and numerical types of modeling. In this same paper, it is described that fracture surfaces of broken pieces of hook reveal the initiation of beach marks from both sides with a granular rough surface in the middle of the fracture zone. Beach marks initiated from both sides indicate the origin of reverse bending fatigue. The distinct granular rough zone in the middle is due to the final brittle fracture. The microstructure of the polished sample revealed numerous inclusions which indicate that the steel was not clean. Such a huge number of inclusions are not desirable as they can act as stress concentration sites and lead to fatigue crack initiation (Mukhopadhyay & Souvik, 2018).

The objectives of the works reported in (Torres, Gallardo, & Domínguez, 2010 & Zade, 2017) are to identify the causes that led to a failure of the crane hook in service. The study of the accident includes: (1) a summary and analysis of the peculiarities inherent to the standards that determine the manufacture and use of this type of device, (2) metallographic, chemical and proctographic analyses, (3) assessment of the steel mechanical behavior in terms of Vickers hardness profile, its tensile strength and fracture energy, and (4) simulation of the thermal history of the hook. The visual and microstructural inspections reveal some pieces of evidence that a welding bed was deposited on the hook surface. Several cracks grew from that area into the material.

In the work of (Shaban & Mohamed, 2013), to study the stress pattern of crane hook in its loaded condition, a solid model of crane hook is prepared with the help of ABAQUS software. A real-time pattern of stress concentration in the 3D model of the crane hook is obtained. The stress distribution pattern is verified for its correctness on an acrylic model of crane hook using the optical method (Caustic method) set up. By predicting the stress concentration area, the shape of the crane is modified to increase its working life and reduce the failure rates. This paper (Kumar & Prasad, 2014) analyzed the stress variation in crane hooks for different cross-sections such as circular and square and different radii of curvature as well, experimentally and theoretically.

In 2016, Thakur and his Co-workers studied the experimentally, the loads are obtained for different crane hooks for 5 mm elongation on UTM (Universal Testing Machine). And then the stresses induced in the crane hooks against the loads obtained from experimentation are also calculated theoretically using curved beam theory. Life of crane hook mainly depends on crane hook material mechanical properties, load frequency of loads (Thakur, Pawar, Nadar, ghorpade, 2016).

2.3 Crane hook optimization

This paper focuses on the optimization of the performance of the crane hook based on stress, geometry, and weight. A single load is considered and multiple cross sections-including square, circular, and trapezoidal are analyzed (Hagara & Pástor, 2017 & Zade, 2017). The stress concentration factors are widely used in strength and durability evaluation of structures and machine elements. A large number of research works have been performed in this field and recommendations for the engineers developed (Gopichand. & Lakshmi., 2013).

Structure failure of the crane hook occurs because of the stress-induced due to repetitive loading and unloading conditions. In this study, solid modeling of crane hook having trapezoidal cross-section referring to one of its existing designs will be done using SOLIDWORKS. Further, analyses are carried out in ANSYS Workbench and nCode. The lengths of two parallel sides of the cross-section of crane hook are varied and different candidates are obtained for loading capacity of 30 tons based on mass, total displacement, and von-Mises stress. This is done to reduce weight and balance the economy (Gopichand. & Lakshmi., 2013).

The design parameters for crane hook are the area of cross-section, material, and radius of the crane hook, and optimization of design parameters is carried out using the Taguchi method, a total of three parameters are considered with mixed levels and L16 orthogonal array is generated. The optimum combination of input parameters for minimum von Misses stresses are determined (Singh & Rohilla, 2015). There have been accounts of repeated failure of crane hooks at the coil yard of a Hot Strip mill which poses a serious threat to safety in the area. More than 4 hooks failed in less than 5 years. The crane hook (rated for 36000 kg) failed from the threaded shank while lifting a load of 18143 kg (Desai & Zeytinoglu, 2016).

Beam et al. (2018) presented the analysis and optimization of the geometric parameters of the T-cross section of the crane hook. The reduction of the cross-sectional area of the hook is set as the main objective of this study. The permissible stresses in the crane hook characteristic points at the most critical place of the construction are taken as the limitation functions. Also, in the second part, analysis and optimization of certain geometric constraints are taken. The maximum stresses in characteristic points are calculated according to the Winkler-Bach theory, where the construction of the hook is treated as a curved beam.

In this paper (Mehendale, 2016), the overall design of the hoisting mechanism of an EOT crane. The dimensions of the main components have been determined for a load capacity of 10 tons, crane having 8 rope falls. Various dimensions for cross-sections of various shapes for crane hook have been found. After the system was designed, the stress and deflection are calculated at critical points using ANSYS and optimized.

Dimensions and load per wire for wire ropes have been found. A solid model of crane hook is prepared as per the standard dimension of 10 tone hook with the help of SolidWorks parametric software having material mild steel and then it is exported to ANSYS software and load is applied. The location of maximum stress produced within the member is located and identified using FEM. Modal analysis concerning boundary condition for propose weigh and stress effect on the crane hook and find total deformation with respected frequency according to geometry. Laminated hook got its name from the process of manufacturing it. In an investigation reported by (Lanjekar & Patil, 2016). In this paper investigation plates which are rivets together. In an industry when continuous and unloading occur the failure of the hook takes place, but in the laminated hook when crack or faults occur the faulty plate can replace by the new one this reduces time and cost.

To calculate the bending stress that occurred in hook and the fractures affecting its life Weight optimization of the hook using a topological approach it's taken into consideration. The topological approach of weight optimization, in this weight, will be reduced in the area where more stress is present. Simulation of the hook is done using a topological approach. model is created, then meshing is done, FEA analysis (ANSYS 14.0) carried out.

The design of the crane hook contains such parameters as the cross-section of hook, material, and radius of curvature. The design is generated in Creo2.0 and analyzed in ANSYS 15.0 workbench (FEM). By applying a modified cross-section in place of Standard Trapezoidal section of hook and considering stress analysis, weight optimization is carried out. The modified design is further compared with Trapezoidal, Triangular, and Circular hook .The stress concentration factors are used in strength and durability evaluation of structure and machine element. Changing the cross-sectional area, and removing material from the low-stress concentration area in lifting hook and then comparing design stress, the hook is to be optimized (Upendar, 2018). To enhance the structure that has the least contribution to the overall stresses or stiffness can be identified. Topology optimization is a mathematical technique that optimizes material layout within a given design space to reduce its weight for a given set of loads, boundary conditions. Hyper works opti-struct solver which uses the density approaches have been used for this purpose.

To demonstrate the usefulness of the topology optimization approaches, a crane hook has been used to carry out the study. The model of the crane hook with a trapezoidal cross-section is created with CATIA V5. The finite element analysis reveals the region of low stress and the scope for the removal of material (Wang &Ji, 2019).

The hooks are prone to fatigue fracture due to the frequent impact loads on the hooks at sea. To further strengthen the structural strength of the hooks, the influence of the three parameters on the hooks is studied, such as the deflection angle of the hooks with large openings, the opening diameter of the hooks, and the position of the maximum thickness of the hook walls. The main points are as follows: parameterized modeling of crane hooks, data simulation and analysis with ANSYS, and optimization analysis of hooks' strength, which provide a basis for further enhancing the reliability of hooks (Norato, 2010).

More recently, several research groups have developed methodologies that can incorporate stress constraints, where the constraint is applied in large portions or all of the structure. These methods are important predecessors of the work presented here, in that they: a) introduce ways to enforce local constraints in the entire design space; b) address the problem of stress singularity (which is also relevant in this work since in the proposed method fatigue is a function of the stresses), and c) highlight the importance of using consistent sensitivities due to the high nonlinearity of the problem (Singh. & Rohilla., 2015).

The crane hooks design for support 2 tons of loading and material made from high carbon steel AISI 4340. The design by ISO 7597 standard that defines factors to study by the Finite Element Analysis method compatible with the Optimum Design approach and the Weighted Factors Rating Method. From all method that was found the best result thus can support more loads and reduce weight (Manee-ngam. & Fournier, 2017).

2.4 Conceptual modeling

Over the past decades, there has been a growing interest in, and concern about “conceptual modeling.” within the modeling and simulation community. Generally accepted as crucial for any modeling and simulation project addressing a large and complex problem, conceptual modeling is not well-defined, nor is there a consensus on best practices. “Important” and “not well understood” would seem to qualify conceptual modeling as a target for focused research. With this context, developing an engineering discipline of conceptual modeling will require a much better understanding of,

1. how to make conceptual models explicit and unambiguous, for both the target system (or referent) and the target analysis,
2. the processes of conceptual modeling, including communication and decision-making involving multiple stakeholders
3. architectures and services for building conceptual models(Bock., Dandashi., & Friedenthal., 2017).

The terms model, modeling, and simulation are at the heart of conceptual modeling (Karagoz., 2008). According to Merriam Webster’s Online Dictionary, a model as a noun has several possible definitions (Meriam, 2019). The most appropriate definitions in our context are “a usually miniature representation of something; also: a pattern of something to be made” and “an example for imitation or emulation”. Model or modeling as a verb is defined as “to produce a representation or simulation of” and “to construct or fashion in imitation of a particular model”. Conceptual modeling is probably the most important aspect of a simulation study. It is also the most difficult and least understood (Robinson, 2014, Roca, Pace & Tolk, 2015).

Conceptual Modeling (CM) has gained a lot of interest in recent years and it is widely agreed that CM is the most important phase of the simulation study. Despite its significance, there are very few techniques that can help to develop well-structured and concise conceptual models. The use of the Structured Analysis and Design Technique (SADT) from software engineering to develop conceptual models (Ahmed & Antuela, 2014).

(Reinders, Akkermans & Balde, 1991) developed a conceptual modeling framework for knowledge-level reflection (KLR), i.e., the modeling of tasks that require a self-representation of a knowledge system's object-level problem-solving tasks. This framework builds upon the KADS methodology for knowledge acquisition and design of knowledge systems.

2.5 Research Gap

Several researchers have proposed different methods in order to combat the fatigue failure and the optimization of crane hooks. The following points can be summarizing from the pieces of literature review. Fatigue failure, Structure failure of the crane hook occurs because of the stress-induced due to repetitive loading and unloading conditions. Failure of crane hooks mainly depends on three major factors i.e. dimension, material, and overload, and optimization of the performance of the crane hook is mostly based on stress, geometry, and weight. A single load is considered and multiple cross sections-including square, circular, and trapezoidal are analyzed. In the reviewed literature it is observed that the optimization of the crane hook has not taken consideration of surface properties and topology optimization of the crane hook. Also, there is no information about the crane hook shape, safety factory minimum, and solid mass and finally, some papers, during optimization, did not consider stresses and strains that are concentrated at holes, slots, or changes of a section in elastic bodies. Plastic flow, fracture, and fatigue cracking start at these places.

CHAPTER 3 MATERIAL SELECTION AND METHODS

This chapter deals with the applied methodologies and material assessment for the crane hook topology optimization. Material assessment is done depending on the aspect of mechanical properties and fatigue analysis. The methodologies used in this thesis involve analytical structural analysis, geometrical analysis, topology optimization, and fatigue analysis.

3.1 Assessment of material selection

Crane hooks and beams are manufactured from steel materials of different cross-sections. Other than the load-carrying capacity, crane hooks must also be able to absorb the vertical load and deflection (induced due to variable loads). The ability to store and absorb more amount of strain energy ensures the safety of crane (Babu & Rao, 2015).

Material selection for every manufactured product is very important because it is directly related to availability, cost, mechanical, chemical, the physical property of material, and durability of components. There are many properties of materials that should be fulfilled for enough performance operation of the specified product. And also one of the most important tasks that an engineer may be called upon to perform is that of materials selection about component design. Inappropriate or improper decisions can be disastrous from both economic and safety perspectives (Callister, 2007). Material selection depends on the comparison of the mechanical properties and fatigue analysis respectively.

3.1.1 Material depending of mechanical properties

The mechanical properties of materials are ascertained by performing carefully designed laboratory experiments that replicate as nearly as possible the service conditions. This is particularly true for the cases where the component or structure is subjected to fatigue loading, the fatigue resistance can be greatly influenced by the service environment, surface condition of the part, method of fabrication and design details.

In some cases, the role of the material in achieving satisfactory fatigue life is secondary to the above parameters, as long as the material is free from major flaws. Commonly used material types for design against fatigue is steel. Steel materials are widely used as structural materials for fatigue application as they offer high fatigue strength and good processability at relatively low cost. The optimum steel structure for fatigue is tempered martensite since it provides maximum homogeneity. Steel with high hardenability gives high strength with relatively mild quenching and hence, low residual stresses, which is desired in fatigue applications. Normalized structure, with its finer structure, gives better fatigue resistance than a coarse pearlite structure obtained by annealing (Wiley, 2001). Table 3.1 gives the key mechanical properties of four typical structural plates of steel, AISI 4340 Steel normalized, AISI 4340 Steel annealed, and AISI 4130 steel normalized based on their mechanical properties.

Table 3- 1 Comparison of the materials based on the mechanical properties (ASM aerospace specification metals Inc, 2019).

Properties	Structural Steel	AISI 4340 Steel, Normalized	AISI 4340 Steel, Annealed	AISI 4130 Steel, Normalized
Elastic modulus (GPa)	200	205	205	205
Poisson's ratio (N/A)	0.3	0.32	0.285	0.285
Shear modulus, (GPa)	76.923	80	80	80
Mass density (Kg/m^2)	7850	7850	7850	7850
Ultimate Tensile strength (MPa)	460	1110	745	670
Yield strength (MPa)	250	710	470	436

Table 3.1, shows a comparison of four different materials which are structural steel, AISI 4340 Steel normalized, AISI 4340 Steel annealed, and AISI 4130 steel normalized based on their mechanical properties. The structural steel is not heat-treated and also the strength of the material is very small compared with the others. As mentioned above the normalized material is better than the annealed material due to the fatigue resistance. Therefore, depending on the above comparison criteria, AISI 4340 Steel is better than the others. Therefore, this study selected AISI 4340 steel normalized by consideration of the above steel criteria's.

3.1.2 Material selection depending on the fatigue analysis

Fatigue is the failure of a component after several repetitive load cycles. In the study of fatigue failure except for one important reason: the desire to know why fatigue failures occur so that the most effective method or methods can be used to improve fatigue strength.

The three major fatigue life methods used in design and analysis are the stress-life method, the strain-life method, and the linear-elastic fracture mechanics method. These methods attempt to predict the life in the number of cycles to failure, N, for a specific level of loading. Life of $1 \leq N \leq 10^3$ cycles is generally classified as low-cycle fatigue, whereas high-cycle fatigue is considered to be $N > 10^3$ cycles. The stress-life method, based on stress levels only, is the least accurate approach, especially for low-cycle applications. However, it is the most traditional method, since it is the easiest to implement for a wide range of design applications, has ample supporting data, and represents high-cycle applications adequately (Nisbett, 2006).

There are many great factors for the fatigue strength factors to reduce the strength or life of the steel material. The temperature factors $\mathbf{C_T}$ accounts for the fact the strength of material decreases with increased temperature. The reliability factor, $\mathbf{C_R}$, acknowledges that a more reliable above 50% estimate of endurance limit requires using a lower value of endurance limit. Surface factor $\mathbf{C_S}$ is the effect of surface finish, for this research the surface factor for forged is 0.6. This means there is no effect of surface scratches and geometrics at the stress concentration. Parts that are more than 50mm in diameter and that are subjected to reverse bending should carry a gradient factor $\mathbf{C_G}$ and the load factor $\mathbf{C_L}$ for bending are 0.8 and 1 respectively (Nisbett, 2006).

$$S_e = C_L C_G C_S C_T C_R, \mathrm{S}'_E \tag{3.1}$$

Where: $\mathrm{S}'_E = 0.5 S_U$

But the value of correction factors is 0.48 depending on the assumption and the standard of the material type. Then endurance limit and the correction endurance limit are 555 MPa and 266.4 MPa respectively.

The strength bending load types is 1000 cycles

$s'_E = 0.5 S_U = 999 Mpa$

Se = (0.6) (0.8) (1) (0.48) (0.48) (999Mpa) = 230.17 MPa

Fatigue S-N Curve diagram values

The strength-life (S-N) diagram provides the fatigue strength S_f versus cycle life N of a material. ANSYS workbench, by default, displays the S-N curve of structural steel, but the material of this study is AISI 4340 Steel normalized. The S-N curve values of this material can be solved using the generalized S-N formula.

For design, an approximation of the idealized S-N diagram is desirable. To estimate the fatigue strength at 10^3 starts with Eq. (3.2).

$$\frac{\Delta\varepsilon_e}{2} = \frac{\sigma'_F}{E}(2N)^b \tag{3.2}$$

Define the specimen fatigue strength at a specific number of cycles as $(s'_f)_N = {E\Delta\varepsilon_e}/{2}$ then combine with Eq. 3.2,

$$(s'_f)_N = \sigma'_F (2N)^b \tag{3.3}$$

The two constants a and b are unknown and can be determined as follows. Let the strength at N = 10^6 be S = S_e (endurance limit), and let the strength at N = 10^3 be S = S'_e. Substituting these values in Eq. (3.2).

At 10^3 cycles,

$$(s'_f)_{10^3} = \sigma'_F (2.10^3)^b = fs_{ut}$$

Where f is the fraction of s_{ut} represented by $(s'_f)10^3$ cycles solving for f gives

$$f = \frac{\sigma'_F}{s_{ut}}(2.10^3)^b \tag{3.4}$$

If this true stress - true strain equation is not known, the SAE for steel with $HB \le 500$ may be used. $\sigma'_F = S_{ut} + 345Mpa$

To find b, substituting the endurance strength and the corresponding cycles S_e and N respectively into Eq. (3.2) and solving for b,

$$b = -\frac{\log\left(\frac{\sigma'_F}{s'_e}\right)}{\log(2N)} \tag{3.5 a}$$

Thus, the equation $(s'_f)_N = \sigma'_F (2N)^b$ is known at the 10^6 cycles. Using the same procedure.

$$a = \frac{(fs_{ut})^2}{s_e} \tag{3.5b}$$

Thus, the general S-N formula is given by;

$$sf = aN^b \tag{3.6}$$

Where N is cycles to failure and the constants a and b are defined at the cycles 10^3, $(sf)_{10^3}$ and 10^6, Se with $(sf)_{10^3} = fs_{ut}$,. Now by substituting any value number of cycles (N) in Equations (3.6), it is possible to find the alternative stress and the corresponding strength.

Then, the fatigue S-N diagram values for structural steel, AISI 4340 steel annealed, and 4130 steel normalized can be computed by using general S-N formula. Table 3-2 shows a comparison of the values of the S-N curve for structural steel, AISI 4340 steel normalized, AISI 4340 steel annealed, and 4130 steel normalized. In the table, the comparison of alternating stresses and the corresponding number of cycles for each material are given. The values of alternating stress are determined using Eq. (3.4) by substituting numbers of cycles.

Table 3- 2 Comparison of the values of S-N curved for structural steel(Tigabey, 2018).

Number of cycles (N)	Alternating stress of structural steel in MPa	Alternating stress of AISI 4340 steel normalized in MPa	Alternating stress of AISI 4340 steel annealed in MPa	Alternating stress of AISI 4130 steel normalized in MPa
10	999.28	2411.31	1618.4	1455.48
20	875.16	2111.8	1417.38	1274.69
50	734.42	1772.19	1189.44	1069.7
100	643.2	1552.06	1041.7	936.83
200	563.3	1359.28	912.31	820.47
2000	362.58	874.91	587.22	528.1
10000	266.47	643.02	431.57	388.13
20000	233.38	563.15	377.97	339.92
10^5	171.52	413.88	277.79	249.82
10^6	110.4	266.4	178.8	160.8
10^7	71.06	171.47	115.09	103.5

Crane hooks are mostly fabricated from structural steel materials. But for this study, AISI 4340 steel normalized steel is selected and compared with the other two materials as well as structural steel to check if it is better. In this case, the comparison is done based on the S-N curve or fatigue life of the materials. Figure 3-1, shows comparative values of the materials which are structural steel, AISI 4130 steel normalized, AISI 4340 steel normalized and AISI 4340 steel annealed depending on fatigue life (S-N curve). The graphs in Figure 3.1, are plotted by using the results of Table 3-2. The results of AISI 4340 steel normalized shows that at a stress level of 2411.31 MPa, the crane hook can survive only 10 cycles and at 266.4 MPa the fatigue life cycle is 10^6 cycle.

And the AISI 4340 steel annealed counter the plot shows 10 and 10^6 cycles can survive at 1618.4 MPa and 178.8 MPa respectively. The maximum number of the cycle which is 10^6 of the structural steel is at 110.4 MPa and the minimum cycles are at 999.28 MPa. The results of AISI 4130 steel normalized, shows that at a stress level of 1455.48 MPa, the hook can survive only 10 cycles and at 160.8 MPa, the fatigue life cycle is 10^6 cycles. Therefore, AISI 4340 steel normalized has a better life than the other two materials.

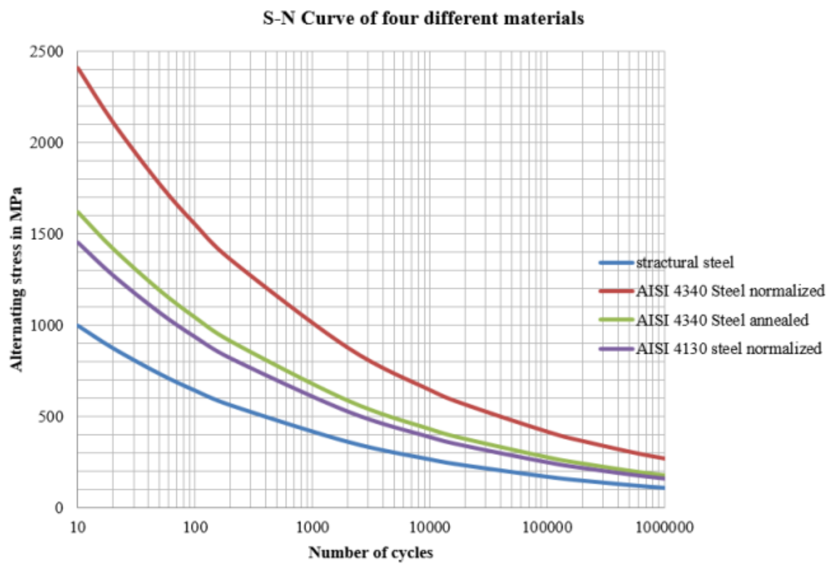


Figure 3. 1 Comparison of S-N curve for steel AISI 4340 normalized and annealed

Figure 3.1 is a comparison of S-N for structural steel AISI 4340 normalized and annealed steel. The S-N curve in this figure is a very useful way to visualize time to failure for a specific material with the S-N, curve. The S-N means stress vs. cycle to failure, which when plotted uses the stress amplitude, σ_a plotted on the vertical axis and the logarithm of the number of cycles to failure. The objective of this analysis is to evaluate the fatigue failure or to evaluate the number of cycles the hook is going to experience before it fails.

While many parts may work well initially, they often fail in service due to fatigue failure caused by repeated cyclic loading. Characterizing the capability of a material to survive many cycles a component may experience during its lifetime is the aim of fatigue analysis. In a general sense, Fatigue Analysis has two main methods: (1) Strain Life and (2) Stress Life. These analysis approaches are available within the ANSYS Fatigue Module. Fatigue analysis can be classified into high cycle fatigue (HCF), which is for several cycles greater than 10^5 and low cycle fatigue (LCF), which means less than 10^5 cycles. For this study, a stress life approach is used, in which the total life of the hook is analyzed. The total life of the hook is the summation of crack initiation and crack life of the hook. The hook experienced the number of cycles greater than 10^5 cycles which means it is within the HCF and it includes the infinite life.

The fatigue strength factor and scale factor are equal to one which means there is no surface imperfection and cracks on this model (surface of geometry). The mean stress theory is chosen Goodman theory and the stress component is equivalent to Von-Mises stress.

3.2 Topology Optimization analysis methods of the crane hook

3.2.1 Structural design and optimization.

The purpose of many structural design problems is to find the best design among many possible candidates. For this reason, the design engineer has to specify the best possible design as well as the best possible candidates (Kim and Nam-Ho, 2005). Structural design tools include topology, topography, and free-size optimization. Sizing, shape and free shape optimization are available for structural optimization.

In the formulation of design and optimization problems, the following responses can be applied as the objective or as constraints: compliance, frequency, volume, mass, the moment of inertia, the center of gravity, displacement, velocity, acceleration, buckling factor, stress, strain, composite failure, force, synthetic response and extremely (user-defined) functions. Static, inertia relief, nonlinear gap, normal modes, buckling and frequency response solutions can be included in a multidisciplinary optimization set up (Sharma, 2013). Structural optimization techniques consist of various aspects. For example, structural optimization may depend on the application fields it will be used for. Then it is divided into size, shape and topology optimization.

A numerical optimization method provides a unique and versatile tool for design optimization. It is defined as the process of finding the conditions that give the maximum or minimum value of a function. To enhance the structure that has the least contribution to the overall stresses or stiffness can be identified. Topology optimization is a mathematical technique that optimizes material layout within a given design space to reduce its weight for a given set of loads and boundary conditions (Thejomurthy & Ramakrishn., 2018).

In the variable density approach, a density function $\rho(\mathbf{x})$ $(0 \leq \rho(\mathbf{x}) \leq 1)$ is introduced into the problem formulation to represent the material distribution in the design domain. To achieve the goal of topology design, the density function $\rho(\mathbf{x})$ is related to the stiffness of the material by a power law. This choice has the effect of penalizing the intermediate densities (i.e. for $\rho(\mathbf{x})$, such that $0 < \rho(\mathbf{x}) < 1$), since in this case volume is proportional while stiffness is less proportional to $\rho(\mathbf{x})$. In this way, it is hopeful that the optimal structure may almost consist of elements that only have 0 or 1 densities function. It is worth noting that most of the numerical algorithms based on these two approaches are element-based.

In the element-based computational framework, the initial design space is always discretized by uniform rectangular finite elements and the design variables are assumed to be constant within each finite element (Guo, Zhao & Yu, 2005).

Before using size or shape optimization, an initial design proposal has to be available. In the planning phase, a fundamental structure of the object can be found using topology optimization. Starting from known loads and boundary conditions and the maximum design space available, a design concept can be found. This design concept is as light as possible while meeting all requirements, for example, stiffness and durability. Areas that are not needed are removed from the given design space. The new structure shows an indication of the optimal energy flow. The result serves as a design draft for the creation of a new FE model for the subsequent simulation calculation and shape optimization. This method provides the designer, even in the early planning stage, with a tool capable of creating a weight-optimized design proposal for a given space.

3.2.2 Mathematical formulation of the optimization problem

The optimization task for the problem is to determine the optimal geometric parametric of the cross-section of crane hook which will lead to the minimization of its optimal cross-section area (Goran Nebojsa, 2018). The optimization problem is defined in the following way;

Minimization of the objective function

$$f(x) \tag{3.7}$$

Subject to the constraint function

$$g_i(x) \le 0, i = 1...., m \tag{3.8}$$

where it is fulfilled; $x_j \ge 0$ (3.9)

and

$$1 \le x_i \le u_i, i = 1..., n \tag{3.10}$$

Where $f(x)$ = the objective function

$g_i(x) \le 0, i = 1...., m$ = the constraint function

l_i, u_i = Lower and upper limits of design variables,

i, j = Number of constraints and number of design variables,

$X = \{x_1 ..., x_n\}^T$ a projected vector of n variables; project variables are the value which should be determined during the optimization process (each the project variable is defined by its lower and upper limit)

3.2.3 Objective function and constraints

1. Objective function

The objective function is represented by the area of cross-section of crane hook at the most critical place and the cross-section area or the objective function is weight reduced, maximum (von Mises) stress, total deformation and equivalent elastic strain.

2. Constraints functions

Optimization processes are based on permissible stresses, according to the Winkler-Bach theory. The total deformation in the curved beam is proportional to the distance from the neutral surface (axis). The strains of the surface are not proportional to these distances, since the fibers are not equal in length, unlike the straight beam. In the case of bending stress that does not exceed the permitted flow stress limit; the stress of any fiber of the beam is proportional to the stress of the fibers so that the elastic stresses in the fibers of the curved beam are not proportional to the distance from the neutral axis (surface). For the same reason, the natural axis in the curved beam does not pass through the center of gravity of the cross-section. The cross-section is characteristic points with allowed stresses, according to the constraint function of the mathematical form.

3.2.4 Topology optimization

Topology optimization is an optimization method that employs mathematical tools to generates an optimized material distribution for a set of loads and constraints within a given design space. The design space can be defined using shell or solid elements or both. The classical topology optimization set up solving the minimum compliance problem, as well as the dual formulation with multiple constraints is available. Constraints on Von-Mises stress and buckling factors are available with limitations (Sharma, 2013).

Topology optimization is different from shape optimization because shape optimization methods work in a range of allowable shape which has fixed topological properties. Topology optimization generates the optimal shape of the mechanical structure. Topology optimization is a powerful approach for determining the best distribution of material within a defined design domain (Brackett, Ashcroft & Hague, 2011).

Topology optimization can be implemented through the use of finite element methods for the analysis and optimization techniques based on the Homogenization method, level set, optimality criteria methods brief discussions on these methods are given below.

1. Topology Optimization of Homogenization method or density method

The main idea of the homogenization method is to replace the difficult layout problem of material distribution by a much easier sizing problem for the density and effective properties of a perforated composite material obtained by cutting small holes in the original homogeneous material. The power-law approach must be combined with perimeter constraints, gradient constraints or filtering techniques to ensure the existence of solutions.

2. The performance-based topology optimization method

Performance-based optimality criteria were proposed and incorporated in PBO algorithms to identify the optimum from an optimization process. In this method, practical design requirements are taken into consideration to aim at a specific performance level. In PBO design, strength, serviceability, and cost performance requirements must be satisfied with the design. Limiting values specified by the design codes govern the strength and serviceability requirements. The performance objective is the weight of the structure and performance-based constraints are stresses, displacements and mean compliance.

3. Topology optimization with the Level set method

The level-set method is a numerical method for finding shapes. Numerical computations can be done on grids with curves and surfaces using the level set method. This approach is called the Eulerian approach. Also, the level-set method makes it very easy to follow shapes that change topology, for example, when a shape splits in two, develops holes, or the inverse of these operations. To solve minimum stress, stress-constrained shape and topology optimization problems (Picelli & Townsend, 2018)

3.2.5. Development of methodology

The objective of this thesis work is to create a methodology of how to use topology optimization in the design process of a crane hook. The work is based on a specific topology optimization configuration, which is tested and analyzed throughout the proposed component development processes using the structural optimization tools. Since the aim of this thesis is to establish a suitable topology optimization process starting from original design to the end of the optimal results design. The methodology will be presented as a flow chart with recommendations for how to perform the design optimization processes. Figure 3.2. Indicates the schematics of the conceptual modeling frameworks methodology within a flowchart.

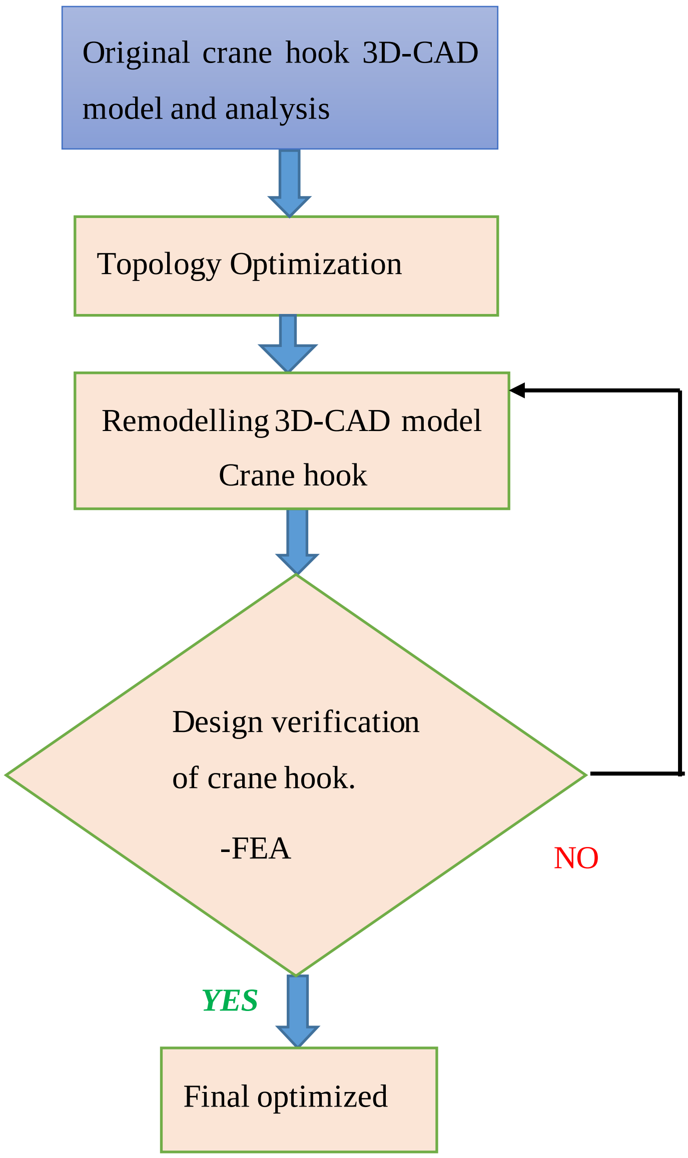


Figure 3. 2 The schematics of the conceptual modeling frameworks methodology

3.3 Analytical analysis in the structural design of crane hook methods.

3.3.1 Dimensions of the standard crane hook

The dimensions of the hook have been determined for a load capacity between 5 to 12.5 Tonnes for Trapezoidal, Rectangular and Circular cross-sections. These dimensions are calculated on the basis of design criteria i.e. keeping area same for all cross-sections (Mehendale, 2016).

Figure 3-3 shows all parameters of the shank crane hook and the standard (proportional) dimensions. The high and low-stress concentration areas of a cross-section are indicated; which are modification areas of crane hook to get better results of weight and maximum stress of the crane hook, and the proportional dimensions of a single shank hook are indicated in Table 3.1. For analytical stress analysis method used only the high concentration stress area dimensions which are the height of the cross-section (h), inner and outer width of the cross-section (b_i), and (b_o). Because the analytical stress analysis method is done using the curved beam flexure formula (Winkler-Bach formula for curved beam).

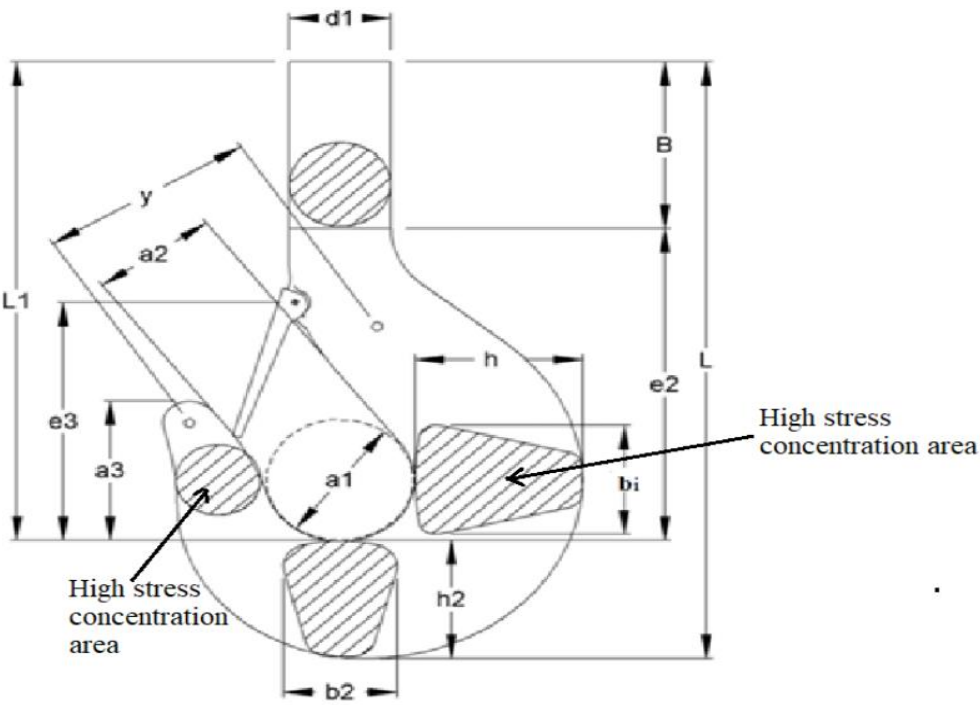


Figure 3.3 Cross-sectional view of the standard (trapezoidal) hook (Columbus. M. Corporation, 2017)

Table 3-3The overall dimensions of the trapezoidal (standard) crane hook (Columbus Mackinnon corporation, 2017).

Crane hook parameters	Values (mm)
Distance from top to the load applied portion (L_1)	318
The inner width of the cross-section at load applied portion b_2	60
Distance from lock pin to an applied load surface (e_3)	165
The height of the nose part of the crane hook (a_3)	90
A gap of the curvature (a_2)	63
The diameter of the inner curvature of the crane hook (a_1)	80
Length of the shank (B)	103
The total height of the crane hook (L)	393
Distance from the bottom of the shank to the load applied portion (e_2)	215
The Inner width of high-stress concentration area of cross-section (b_i)	71
Height of high-stress concentration area of cross-section (h)	90
Height of cross-section at load applied portion (h_2)	75

3.3.2 Design of crane hook.

The design of a crane hook contains such different parameters like cross-section, material, the radius of curvature, and loading capacity. The hook is to be designed with having load-carrying capacity. A hook is made of high tensile steel. Different types of cross-sections are used in the design of a hook i.e. trapezoidal, rectangular and circular are considered. By keeping area the same for all cross-sections as design criteria, direct stress, bending stress, and shear stress are found.

In the design of the crane hook, the following information is required (Sharma., 2013).

- Dimensions of the cross-section of the crane hook.
- The shape of the cross-section of the crane hook.

The crane hook is a curved beam and the stress in a curved beam is calculated.

The calculation for trapezoidal Cross-section

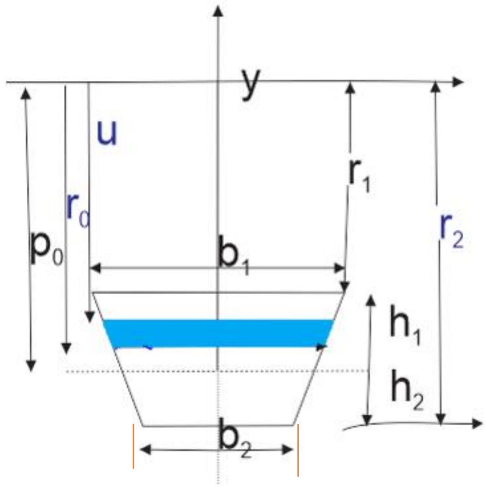


Figure 3. 4 The Standard Trapezoidal hook cross-section of the trapezoidal.

The Winker-Bach formula for the curved beam is

$$\sigma_x = -\frac{M}{Ae}\frac{y}{r_o - y}$$

Where M = Uniform bending moment applied to the beam positive when tending to increase curvature. This bending moment is given by M = P x ρ_o, (Where P = load, KN)

y = distance from the neutral axis

A = Area of the cross-section, mm, given by: $A = \frac{(b_1 - b_2)}{2h}$

e = distance of the centroid from the natural axis, mm: $e = \rho_o - r_o$

ρ_o = initial radius of curvature of the centroidal surface: $p_o = 3 + \frac{(b_1 + 2b_2)h}{3(b_1 + 2b_2)}$

r_o = radius of curvature of centroid axis, mm : $r_o = \frac{A}{\int \frac{dA}{u}}$

h = height of the trapezoidal cross-section

b_1 = width (inner side of trapezoidal cross-section), mm

b_2 = width (outer side of trapezoidal cross-section), mm

σ_x = Direct stress, N/mm^2 And $\int \frac{dA}{u} = [b_1 + \frac{r(b_1 - b_2)}{h}]\log\frac{r_2}{r_1} - (b_1 - b_2)$

3.3.3 Stress analysis in a curved beam

The crane hook is a curved beam application in the case of the straight beam, the neutral axis of the selection coincides with the centroid axis, which is not true for the section of the curved beam**.** Although exact stress analysis of a curved beam is available, it is limited in use due to cumbersome calculations involved. These calculations are eased to a greater extent by the use of the computer. So today's competitive environment, cost reduction is being the major role factor for which various industries are aspiring for many technological innovations that are used in this direction. Curved beam design is generally done with considering many parameters and better design can be done (Nagar, 2018).

A curved beam is a beam in which the neutral axis in the unloaded condition is curved instead of straight. For the study of stress pattern in the curved beam, the following assumptions are made through the plane sections perpendicular to the axis of the beam remain plane after bending, the moduli of elasticity in tension and compression are equal and the material is homogeneous and obeys Hook's law.

Consider a curved beam subjected to bending moment M_b as shown in Figure 3-5. There are two factors, which distinguish the analysis of straight and curved beams. They are as follows;

1. The neutral and centroid axes of the straight beam are coincident. However, in a curved beam the neutral axis is shifted towards the center of curvature, and

2. The bending stresses in a straight beam vary linearly with the distance from the neutral axis. However, in curved beams, the stress distribution is hyperbolic.

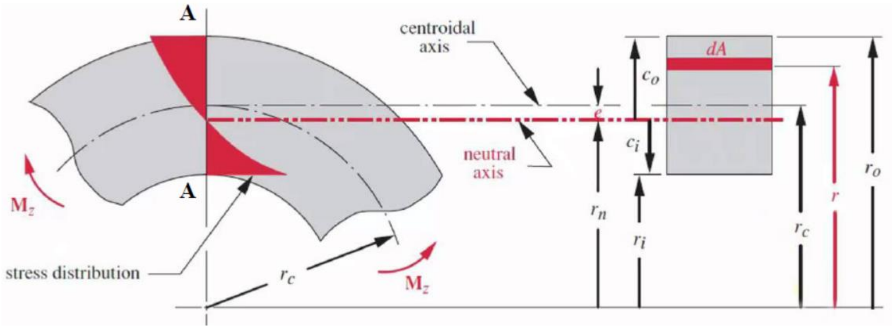


Figure 3. 5 Theoretical stress distribution on the curved beam (Prasad & Ghouse, 2018).

a) Stress analysis for crane hook (original and redesign)

To design the crane, hook cross-section(trapezoidal), the first draw should have to draw the curved beam with its cross-section to shows that the neutral and centroid axes are not coincident. And also show the parameters and the applied load on the curved beam (crane hook). Stress concentration adversely affects the structural properties and hence the knowledge of stress distribution becomes essential. In the investigation of stress distributions in the crane hook. The reduction in stress concentration improves the strength and endurance of the crane hook.

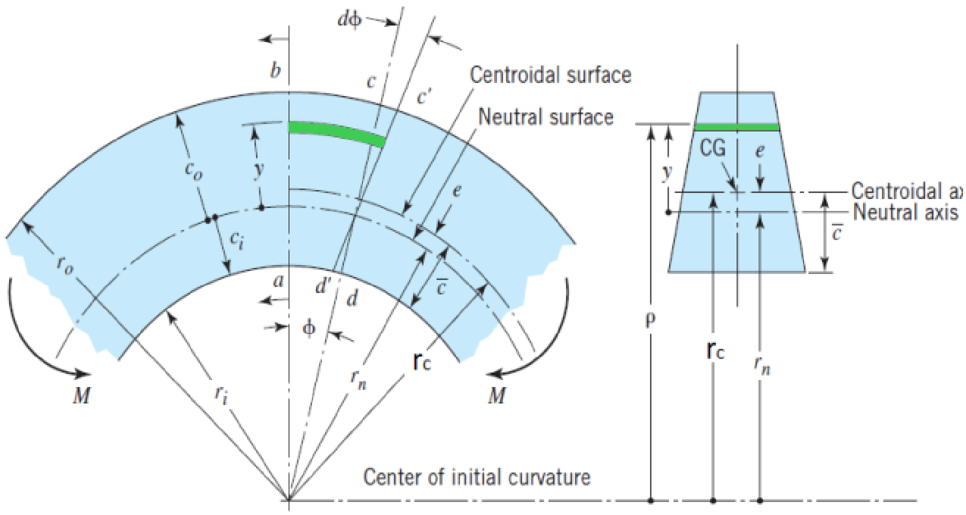


Figure 3. 6 Curved beams with its cross-section, with the neutral and centroid axis.

The purpose of this design is to get a safe condition of the hook due to maximum stress by using appropriate design dimensions of the hook to lifting the applied load.

Table 3- 4 The measured dimension tower crane hook parameters.

Crane hook parameters	Values(mm)
The total height of the crane hook (L)	338
The radius of the outer surface (ro)	140
Radius of inner surface (ri)	42
Width of the outer surface (side) of the cross-section (b_o)	38
Width of the inner surface (side) of the cross-section (bi)	76
Depth of section (h)	90

The cross-sectional area of trapezoidal is; $A = \left(b_i + b_o\right)\frac{h}{2}$ (3.11)

$$A = \left(76 + 38\right)\frac{90}{2} = 5130mm^2$$

To compute the radius of the centroid axis (rc) based on the cross-section type in case of the existing crane is trapezoidal. This is the distance from the origin of the curvature to the centroid axis of the curved beam (crane hook). For the original crane hook trapezoidal cross-section to calculate the radius of the centroid axis is used the following equation.

$$rc = ri + \frac{h}{3} + \frac{b_i + 2b_o}{b_i + b_o} \tag{3.12}$$

$$rc = 42 + \frac{90}{3} + \left(\frac{76 + 2\times 38}{76 + 38}\right) = 73.33mm$$

The radius of the neutral axis (rn) based on the cross-section. This is the distance from the origin of the curvature to the neutral axis of the curved beam (crane hook). For trapezoidal cross-section to calculate the radius of the neutral axis is used in the following equation.

$$rn = \frac{A}{bo - b_i + \left[\frac{b_i r_o - b_o r_i}{h}\right]\ln\left(\frac{r_o}{r_i}\right)} \tag{3.13}$$

$$rn = \frac{5130}{38 - 76 + \left[\frac{(76\times 150) - (38\times 42)}{90}\right]\ln\left(\frac{150}{42}\right)} = 51.01mm$$

The neutral axis and centroid axis are not coincident. Therefore, there is a gap (distance) between the axes and the distance is called eccentricity (e).

$$e = rc - rn$$

$$e = 73.33 - 51.01 = 22.315mm$$

The bending moment about the radius centroid axis (r_c). The force is through the canter of curvature. The maximum applied load (P) of this design is 6 tons.

$$M = P.rc \tag{3.14}$$

$$6000kg)(9.81\text{m/s}^2)(73.33*10^{-3}\text{ m}) = 4316.2Nm$$

To calculate the distance from the neutral axis to the inner (Ci) and distance from the neutral axis to the outer surface (Co)

$Ci = rn - ri$ and $Co = ro - rn$

Ci =9.01mm and Co =98.99mm

Calculate the critical maximum stresses at the inside and outside surfaces due to the bending moment. The stress in the inner surface of the hook is tension and in the outer surface is compression

The stress in the inner surface $\sigma_i = \dfrac{Mc_i}{Aer_i}$ (3.15)

$$\sigma_i = \frac{(4316.2Nm) \times (0.00901\text{m})}{0.00513m^2 \times 0.022315m \times 0.042m} = 81.7 \text{ MPa}$$

The stress in the outer side.

$$\sigma_o = \frac{-Mc_o}{Aer_o} \tag{3.16}$$

$$\sigma_o = \frac{4316.2Nm \times (-0.09899m)}{0.00513m^2 \times 0.022315m \times 0.15m}$$

σ_o =24.88 MPa (compression)

The hook is stretched to downward by the applied load due to the load the hook is subjected to direct tensile stress.

$$\sigma_t = \frac{P}{A}, \sigma_t = \frac{6000 \times 9.81}{0.00513} = 11.47 \text{ MPa}$$

Calculate the resultant stresses at the inner and outer surface add or subtract maximum stresses at inner and outer surfaces from direct stresses (using superposition) because the hook is subjected to bending and direct tensile stress.

The resultant stress in the inner surface.

$\sigma_{Ri} = \sigma_t + \sigma_i$ =93.17 MPa

The resultant stress in the outer surface

$\sigma_{Ro} = \sigma_t - \sigma_o$ =13.41 MPa (compression)

Calculations of load carrying capacity existing crane hook; superimposing the two stresses and equating the resultant to permissible stress, safety factor 3.5) and yield strength 436 MPa.

$$\sigma_i + \sigma_t = \sigma_{\max}$$

$$\frac{1}{fs} = \frac{\sigma_m}{\sigma_y} + \frac{\sigma_v}{\sigma_e} \tag{3.17}$$

$$\frac{1}{fs} = \frac{53.29Mpa}{436Mpa} + \frac{39.68Mpa}{261.6Mpa}, = 3.5$$

$$\sigma_{\max} = \frac{S_{yt}}{fs} = \frac{436Mpa}{3.5} = 124.6$$

From the Eq. 3.17 Determine the load-carrying capacity,

$$\sigma_t = \frac{P}{A}, = \frac{P}{5130mm^2} \text{ and } \sigma_i = \frac{Mc_i}{Aer_i} = \frac{P(73.33*10^{-3}\text{ m})}{0.00513m^2 \times 0.022315m \times 0.042m}$$

$$\sigma_i + \sigma_t = \sigma_{\max}, = \frac{P}{5130mm^2} + \frac{P(73.33*10^{-3}\text{ m})}{0.00513m^2 \times 0.022315m \times 0.042m} = 124.6,$$

P =59610N

The load-carrying capacity of the existing crane hook is 59610 N.

b) Stress analysis for remodeling crane hook

All cross-sectional dimensions except the width of the outer side of the cross-section of the model crane hook is the same with trapezoidal crane hook cross-sections. The changed dimension is the only the width of the outer side of the cross-section. Therefore, a remodel crane hook can design with the same procedure of the trapezoidal crane hook. Figure 3.7, shows the cross-sectional dimensions of the high-stress concentration area of the remodelled crane hook. In this optimization method, the geometry is modified by reducing both sides of the curved member of the hook cross-section.

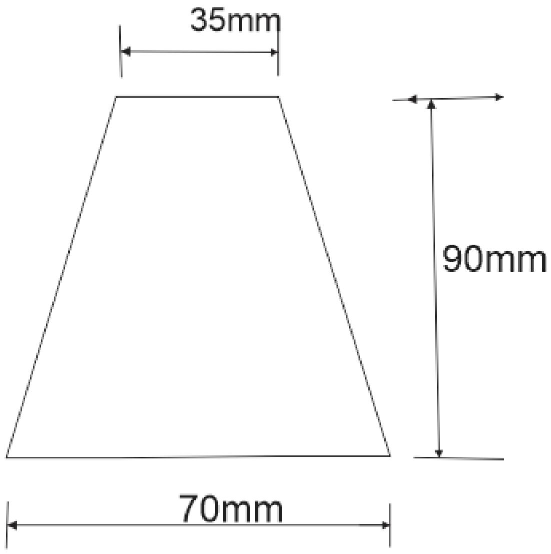


Figure 3. 7 The remodelled crane hooks cross-section of the stress concentration area.

Table 3- 5 The remodelled crane hook parameters.

Crane hook parameters	Values(mm)
The radius of the outer surface (ro)	150
The radius of the inner surface (ri)	42
Width of the outer (side) of the cross-section (bo)	35
Width of the inner (side) of the cross-section (bi)	70
Depth of section (h)	90

The cross-sectional area of remodeling cross-section is

$$A=\left(b_i+b_o\right)\frac{h}{2}=4725\text{mm}^2$$

The radius of the centroid axis (rc) for the remodeling crane hook

$$rc=ri+\frac{h}{3}+\frac{b_i+2b_o}{b_i+b_o}$$

$$rc=42+\frac{90}{3}+\frac{70+2\times 35}{70+35}=73.3mm$$

$$rn = \frac{A}{bo - b_i + \left[\frac{b_i r_o - b_o r_i}{h}\right] \ln\left(\frac{r_o}{r_i}\right)}$$

$$rn = \frac{4725mm^2}{35 - 70 + \left[\frac{70 \times 150 - 35 \times 42}{90}\right] \ln\left(\frac{150}{42}\right)} = 51.12 \text{ mm}$$

There is a gap (distance) between the neutral and centroid axes and the distance is called eccentricity (e)

$e = rc - rn = 22.12$ mm

The bending moment about the radius centroid axis

$M = \text{P}.\text{r}_c, = 6000kg \times 9.18m/s^2 \times 73.3 \times 10^{-3} = 4314.4Nm$

Calculate the distance from the neutral axis to the inner (Ci) and distance from the neutral axis to the outer surface (Co).

$$c_i = rn - ri = 9.12 \text{ mm}$$
$$c_o = ro - rn = 98.88 \text{ mm}$$

The stress in the inner surface (σi).

$$\sigma_i = \frac{Mc_i}{Aer_i} = \sigma_i = \frac{4314.43N/m \times 0.00912m}{0.004725m^2 \times 0.02212m \times 0.042m} = 89.6 \text{ MPa}$$

The stress in the outer surface (σo)

$$\sigma_o = \frac{-Mc_o}{Aer_o} = \sigma_o = \frac{4314.43 \times (-0.09888)}{0.004725m^2 \times 0.02212m \times 0.15m} = 27.2 \text{ MPa}$$

The hook is stretched to downward by the applied load due to the load the hook is subjected to direct tensile stress.

$$\sigma_t = \frac{P}{A} = \frac{6000kg \times 9.18m/s^2}{0.004725m^2} = 11.6MPa$$

Calculate the resultant stresses at the inner and outer surface add or subtract maximum stresses at inner and outer surfaces from direct stresses (using superposition) because the hook is subjected to bending and direct tensile stress.

The resultant stress in the inner surface.

$\sigma_{Ri} = \sigma_t + \sigma_i = 102.2MPa$

The resultant stress in the outer surface.

$\sigma_{Ro} = \sigma_t - \sigma_o = 15.6MPa$ (compression)

Calculations of load-carrying capacity the new model of the crane hook: superimposing the two stresses and equating the resultant to permissible stress, safety factor (3.5), and yield strength 436 MPa.

$\sigma_i + \sigma_t = \sigma_{\max}$,

$$\sigma_{\max} = \frac{S_{yt}}{fs} = \frac{436Mpa}{3.5} = 124.6$$

Then from the Eel 3.determine the load-carrying capacity of the new model of the crane hook.

$$\sigma_t = \frac{P}{A}, = \frac{P}{4725mm^2} \text{ and } \sigma_i = \frac{Mc_i}{Aer_i} = \frac{P(9.12*10^{-3}\text{ m})}{0.004725m^2 \times 0.02212m \times 0.042m}$$

$$\sigma_i + \sigma_t = \sigma_{\max}, = \frac{P}{0.004725mm^2} + \frac{P(9.12*10^{-3}\text{ m})}{0.004725m^2 \times 0.02212m \times 0.042m} = 124.6,$$

P =58395N,

The load-carrying capacity of the new model crane hook is 58395 N.

Table 3- 6 Analytical stress analysis of crane hooks.

Analytical Stress analysis	Original hook	Remodel hook
A bending moment about centroid axis (M) in (Nm)	4316.2	4314.4
Resultant stress in the inner surface (σ_{Ri}) in MPa	93.17	102.2
Resultant stress in the outer surface (σ_{Ro}) in MPa	-13.41	-15.6

Table 3 shows the stress analysis results of the bending moment and maximum stress of the crane hook.

3.3.5 Elastoplastic stress deformation response curved beam

To obtain a realistic design, it is a common practice to both, material topology optimization as well as subsequent shape optimization on linear elastic response. It might be essential to base the optimization on a more realistic physical behavior, i.e. to consider materially or geometrically nonlinear effects (Schwarz, Maute & Ramm, 2001). When the structure experiences a large deformation, the classical theory of Elastoplasticity with the assumption of infinitesimal deformation is modified to consider rigid body motion. The objective rate plays an important role in Elasto-plasticity problems to systematically express rigid body motion.

Plastic deformation can be physically explained by atomic dislocation. An elastic deformation corresponds to the variation in the interatomic distance without causing atomic dislocation, while a plastic deformation implies relative sliding of the atomic layers and a permanent shape change without changing the structural volume. (Kim & Nam-ho, 2005). The elasticity of curved beam stress analysis; Elasticity analysis of curved beams was formerly limited to structures having rectangular cross-sections. A lifting hook with a double-trapezoidal cross-section is an analysis using this extended elasticity method. Mechanics of materials approximations and finite element analysis are also applied to the same hook (Sloboda & Honarmandi, 2014).

Numerical solution of the stress distribution in a curved beam of constant radius and a single arbitrary cross-section. This is accomplished by variable substitutions that transform the compatibility equation and its boundary conditions into a system soluble regardless of how the section thickness changes with the radius. Both pure moment and force situations are considered. When both force and moment loading are present simultaneously, the load can be considered independently and the true stress state resulting in stress superimposed.

3.4 Geometrical analysis using ANSYS

ANSYS is a general-purpose finite element analysis (FEA) software package. Finite Element The analysis is a numerical method of constructing a complex system into very small pieces called elements. The software implements equations that govern the behavior of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. Firstly, the geometric model and finite element model were created. Then the material parameters and boundary conditions were applied to the finite element model, the load was studied. Creating model Firstly, the geometric model needed to be created in 3D modeling software SolidWorks, and then the geometric model was imported to the workbench to create a finite element model.

3.4.1 Geometrical modeling of 3D of the existing hooks

Today, SolidWorks software is being used around the world to design products, develop machinery, and create production systems. Mechanical engineering, industrial design, and transport technologies are just a few of the functions in which SolidWorks software is successfully used as an advanced tool by designers and engineers. Therefore, for this study, SolidWorks is selected to design or 3D modeling of the hooks because the hooks have complex shapes and cross-sections. And SolidWorks is easy to design these kinds of parts.

The below Figure 3.8 a) is a picture of a tower crane hook to be used for 3D geometrical modeling purposes using SolidWork18. This crane hook has a standard trapezoidal geometry and it will be used for the modeling to visualize the shape and topology of the crane hook. And for this crane hook, all the necessary parameters and the dimensions of this hook are taken from the Afro-Tsion construction company site of Jimma University, this tower crane hook has the capacity of carrying six tones. Creating the Geometry Model. The accuracy of the geometric model would directly determine the accuracy of the finite element model, so the geometric model needed to maximize reflect the actual situation.

a) b)

Figure 3. 8 a) Crane hook photo captured from the site (29 Nov. 2019). b) 3D modelled hook.

3.4.2 Meshing generation of the crane hook

The mesh density increases as more elements are placed within a given region. Mesh refinement is when the mesh is modified from one analysis of a model to the next analysis to yield improved results. ANSYS Meshing is a component of ANSYS Workbench. In the finite element method, the structure of interest is subdivided into discrete shapes called elements. Finite element methods have proved indispensable for physical simulation. These methods discretize the simulated domain for example, for this study the different cross-sections crane hooks dividing into many small elements. The most common element types include a one-dimensional beam, two-dimensional elements, or three-dimensional bricks, typically triangles or quadrilaterals in two dimensions and tetrahedral or hexahedra in three. The complex of elements is the mesh. The Purpose of the Mesh generating is Domain is required to be divided into discrete cells (meshed) and Equations are solved at the cell/nodal locations (Metin Ozen, 2014).

1. Importing the CAD model

The model used in this thesis is prepared in IGES (initial graphics exchange specification) format which is compatible with all CAD software. After importing the CAD file into ANSYS19.2 it is then saved. Geometry can be into ANSYS WB from many sources. So importing the CAD data, the first step is geometry clean-up. Geometry clean-up tools are used to restore proper surface connectivity to part geometry. The geometry penal contains tools like quick edit, edge edit, point edit and auto clean-up, etc. which help is preparing surface geometry for meshing. Geometry clean-up is one of the most time-consuming tasks in the project. Meshing quality depends very much on the quality of the geometry. The benefits of repairing CAD are

a) Correcting any error in the geometry from import
b) Creating the simplified part needed for the analysis
c) Ensuring proper connectivity of the mesh
d) Obtaining a desirable mesh pattern and quality

2. Meshing crane hook.

The basic theme of FEA is to make calculations only at the limited number of points and then interpolate the results for the entire domain. Any continuous object has an infinite degree of freedom and it is not possible to solve the problem in this format. Finite element method reduces the degree of freedom from infinite to finite with the help of discretization i.e. meshing.

For the meshing of Crane hook, the 3D, the mesh is generated on all surfaces. Element types used for 3D surface meshing are R-trias and Tetra mesh type as these are more accurate for 3D parts. The Average Element size used is 2.5mm. Element size is decided after doing meshing from 1mm to 3mm and then found that the value of stresses is stable at 1.55mm. Thus the value of the meshing element size is taken as 2.5 mm shows the von Mises stress and displacement on the crane hook for different elements mesh size.

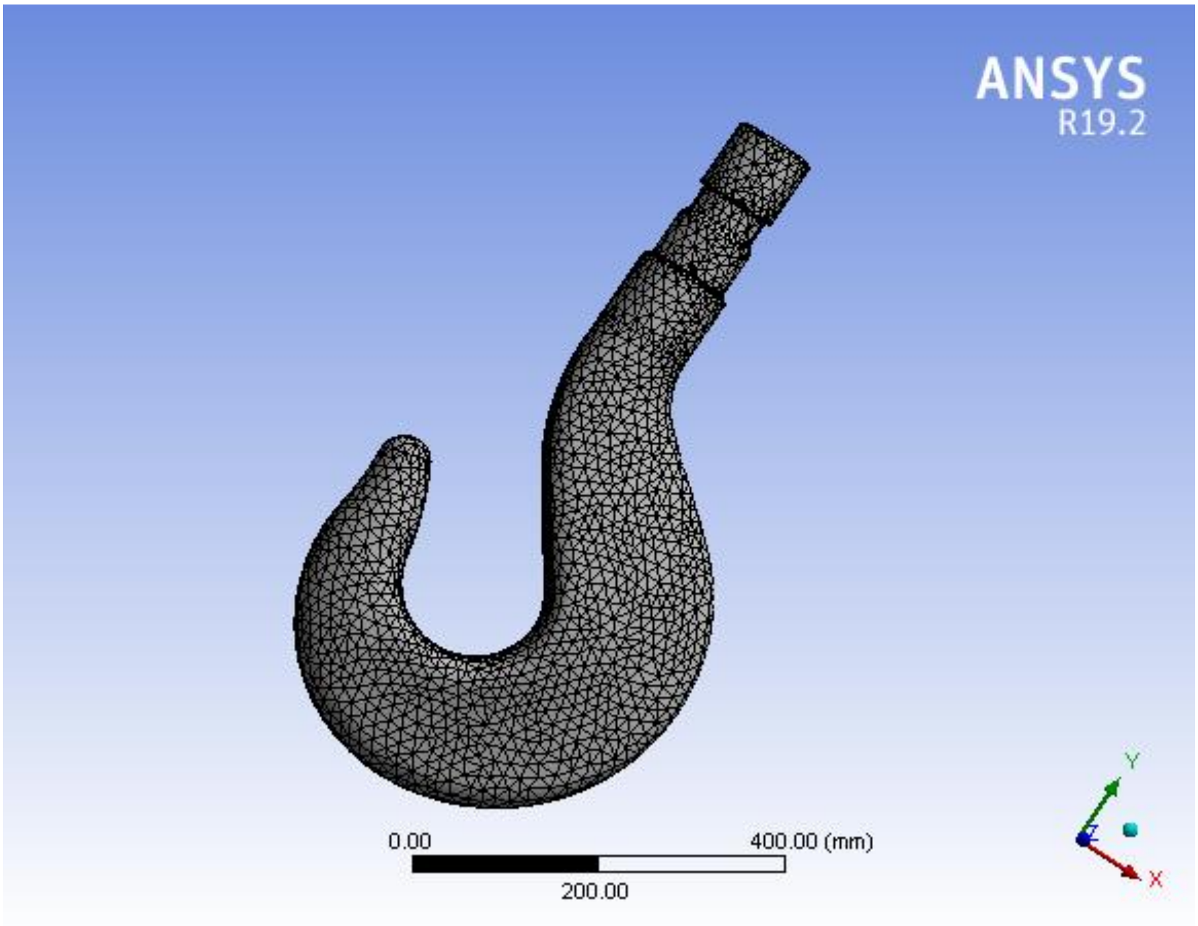


Figure 3. 9 The mesh generation of the crane hook.

To conduct the finite element analysis of the existing hook, the following tasks were performed. Solution accuracy & stability deteriorates as mesh cells, The mash size for all cross-sections is the same which is 3mm but to get the accurate answer of FEA results in the mesh size at high-stress concentration area is 1 mm. Depending on the volume of the models of crane hooks the number of elements and nodes for each model is different. The numbers of elements and nodes for each model of crane hook are listed in table 5-2.

3.5 Fatigue analysis of crane hook

3.5.1 Fatigue Life and Damage estimation of hooks by FEM

There are several different factors that can influence fatigue life including the type of material being used, structure, shape, and temperature changes. In most cases, fatigue life is calculated as the number of stress cycles that an object or material can handle before the failure. There are several different types of stress values that are considered when computing fatigue life, including the maximum stress value, which is usually less than the ultimate tensile stress limits. When a material is put into use, the design can increase the stress that is put on the object. For instance, certain sharper angles, such as the corners in a square object, can be a significantly higher stress area than a rounded area, which disperses the weight and stress of a load more evenly over a larger area. Smooth transitions or fillets will increase the fatigue strength of the structure. Fatigue damage is defined as the design life divided by the available life. For Fatigue Damage, values greater than 1 indicate failure before the design life is reached.

3.5.2 Safety factor of crane hooks.

During determining the appropriate safety factor to apply, the design constraints to consider must take into account the expected. The factor of safety (FS) is how much could a system withstand beyond the expected actual loads. Essentially, the factor of safety is how much stronger the system is than it needs to be for an intended load. For most metal (ductile) materials, it is often required that the factor of safety be checked against both yield and ultimate strengths. The yield calculation will determine the safety factor until the part starts to plastically deform. The ultimate calculation will determine the safety factor until failure. To choose the appropriate design factors are based on several considerations, such as the accuracy of predictions on the applied loads, strength, wear estimates, and environmental effects.

CHAPTER. 4 THEORETICAL BASIS AND DESIGNXPLORER ANALYSIS.

This chapter deals with the theoretical basis of improving the strength of the hook and its conceptual framework development, some definitions of components and terminologies, a framework for structural optimization. Overview of parameterizing via finite element analysis(FEA) with the geometry CAD software; formulation for geometric parameterization of finite element models, designXplorer implementation with the procedure of parametric and finite element analysis. At the end parameterization with FEA implementation response surface modeling in optimization.

4.1 A framework for structural optimization in conceptual modeling

The conceptual design phase could take advantage of a novel methodology, that would not be based on empirical or semi-empirical equations to estimate weights, performances, costs and loads, but relay on analytical models to a greater extent. The presented design framework is thought to meet also the requirements of modern complex product development. Many companies are located all over the world and are tightly involved in several global partnerships, where modules of the product are designed and manufactured at different locations. Today's product development is carried out in a distributed, collaborative, and competitive fashion and this forms a rather complex environment for the employment of modeling and simulation tools (Robinson, 2014). This framework provides a scientifically coherent methodology for refinement, analysis, modeling, comparison, and evaluation of design solutions at the early stage of the design process.

4.2 Conceptual modeling for simulation.

For modeling and simulation to mature both as an industry and academic discipline, the modeling, and simulation community must develop standards and frameworks in the areas of conceptual modeling because conceptual development stages carry implications for multiple decisions that heavily impact subsequent stages of system development and performance. M&S at the conceptual level is consequently a strategic domain poorly explored at the moment (Coatanea, Roca, Mokhtariain, and Mokammel, 2016).

Conceptual modeling is the activity of deciding what to model and what not to model – 'model abstraction'. A conceptual model is 'a non-software specific description of the computer simulation model (that will be, is or has been developed), describing the objectives, inputs, outputs, content, assumptions, and simplifications of the model'.

The figure, 4.1 provides an overview of the conceptual modeling framework that is described in more detail below. In this framework, conceptual modeling consists of five key activities that are performed roughly in this order: understanding the problem situation, determining the modeling and general project objectives, identifying the model outputs (responses), identify the model inputs (experimental factors), determining the model content (scope and level of detail) and identifying any assumptions and simplifications (Wang &Ji, 2019).

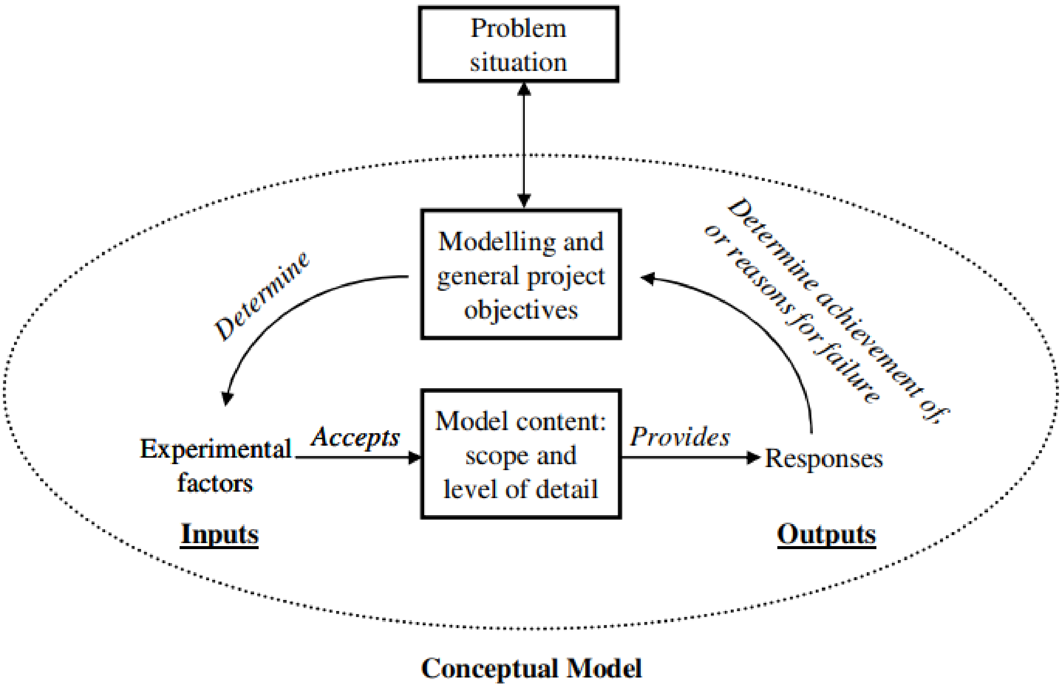


Figure 4. 5 A framework for designing the conceptual model(Roca et al., 2015).

4.3 Parameterization with Finite Element Analysis

Finite Element Analysis (FEA) is a numerical method for solving problems of engineering and mathematical physics. Typical problem areas of interest include structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential. The analytical solutions of these problems generally require the solution to boundary value problems for partial differential equations.

Then FEA methods divide the structure into small but finite, well-defined, elastic substructures called elements. By using a system of simultaneous algebraic equations polynomial functions, together with matrix operations, the continuous elastic behavior of each element is developed in terms of the element's material and geometric properties. Loads can be applied within the element, on the surface of the element, or at the nodes of the element. The element's nodes are the fundamental governing entities of the element, as it is the node where the element connects to other elements, where elastic properties of the element are eventually.

Workbench is a suite of powerful engineering simulation programs based on the finite element method, the workbench is designed as a general-purpose simulation tool, and workbench can be used to study more than just structural (stress/displacement) problems. It can simulate problems in such diverse areas as heat transfer, mass diffusion, thermal management of electrical components (coupled thermal-electrical analyses), acoustics, soil mechanics (coupled pore fluid stress analyses), and piezoelectric analysis. Workbench offers a wide range of capabilities for simulation of linear and nonlinear applications. In a nonlinear analysis workbench automatically chooses appropriate load increments and convergence tolerances and continually adjusts them during the analysis to ensure that an accurate solution is obtained efficiently.

Design variables are shape parameters of basic geometric features. The number of design variables of this formulation is small whereas various constraints can be considered. The finite element method formulation of the problem results in a system of algebraic equations. The method yields approximate values of the unknowns at a discrete number of points over the domain. To solve the problem, it subdivides a large problem into smaller, simpler parts that are called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem (Bhis & Deshpande, 2018). A formulation for geometric parameterization of finite element models is derived from efficient shape optimization. The formulation allows us to express the stiffness and the mass matrix for the geometrically parameterized hexagonal element in an explicit form allowing versatile design parameterizations (Benjamin Jan & Florian, 2018).

4.4 Hook parametric model and force analysis

The strength of the hook and the rationality of its design is crucial to the safety of the crane's work. Taking the hook bearing 6 tons produced by a factory as an example, the strength analysis of the hook is carried out, the deformation and stress distribution laws of the hook are explored, and its critical section is analyzed, which provides a theoretical basis for the study of hook structure optimization and has important engineering significance. The hook is a lifting device on the crane and is also the main component of the crane (Wang &Ji, 2019). When modeling the hook, some unimportant details and details that have little influence on the strength of the hook are ignored, thus simplifying the hook model and eliminating interference on the optimization parameters.

4.5 Finite element analysis of crane hook

There are three methods to solve any engineering problem, analytical method, numerical and experimental method. An analytical solution is a mathematical expression that gives the values of the desired unknown quantity at any location in the body; as a consequence, it is valid for an infinite number of locations in the body. An analytical method is a classic approach that gives accurate results. But this method is best suitable for simple problems like finding the deflection of the cantilever. Simply supported beam and stress and strains etc. But it consumes more time as compared to numerical methods(Chen, Shapiro, Suresh, & Tsukanov 2007& Upendar, 2018).

The use of numerical methods enables the engineer to expand his ability to solve practical design problems. It is not possible to obtain analytical mathematical solutions for many engineering problems. For problems involving complex materials properties and boundary conditions, the engineer's prefer to numerical methods that provide approximate, but acceptable solutions. A numerical method is a mathematical representation which gives approximate results. An experimental method is an actual measurement method. It physically tests the prototype under various conditions. Thus it gives 100% accurate results. But engineers can't prefer because it requires an expensive setup and a more time-consuming method as compared with the analytical method and numerical method.

4.6 Procedure for finite element analysis

The Finite element procedure is now an important and frequently indispensable part of engineering analysis and design. Extensively employed in the analysis of solids, structure and of heat transfer, fluids and indeed, finite element methods are useful in virtually every field of engineering analysis (Bathe, 2016). Certain steps in formulation a finite element analysis of a physical problem are common to all such analyses, whether structural, heat transfer fluid flow or some other problem steps are described as follows;

1) pre-processing (build the model)
2) processing or solution phase (obtain the solution)
3) post-processing (review the results)

Finite element analysis for Design Engineer, the FEA offers many important advantages;

a) Easily applied to complex, irregular-shaped objects and with complex boundary conditions.
b) Applicable to the problem like steady-state time-dependent. And also for linear and nonlinear problems.
c) The FEA can be coupled to CAD programs to facilitate solid modeling and mesh generation and model bodies composed of several different because the element equations are evaluated individually (Logan, 2007).

Topology optimization is an iterative procedure adapted to the computer-aided design (CAD). The main goal of this method is the best structural performance through the identification of the optimum material distribution inside the available volume of a structure with respect to its loads, boundary conditions, and constraints. However, the most crucial step at FEA is the definition of the problem statement and its equivalent mathematical model with all the required parameters (material properties, loads, and restraints). The optimum results occur through the discretization (meshing) of the model and with a repetitive convergence method. The topology optimization method offers a new optimized design geometry with a notable mass reduction (or increment) which can be used as a new starting point for the FEA. Finally, the new FEA results validate or evaluate the success of the TO approach (Tyflopoulos Flem & Steinert, 2018).

4.7 Parametric optimization procedure

After the analysis with a finite structural element of hook has been carried out, for the condition imposed the client, a parametric optimization project is carried out, starting from the optimization input parameters: a) the thickness of the crane hook (mm) b) the acting force (N), then adjust the optimization output parameters: c) strength of deformation) the equivalent deformation stress (von Mises), and also the inner and outer radius of the crane hook. whose values are determined according to the results of the static structural analysis as well as the limit values as objectives of topology optimization.

4.8 What is designXplorer in ANSYS workbench.

DesignXplorer i**s** a component of ANSYS Workbench that can help you make your designs more efficient and robust. And also a powerful tool for designing and understanding uses response surfaces and assemblies. Determine the sensitivity of the response of the system to variations in the input quantities. Identify which input variables play a dominant role in the response. Develop a surrogate function that enables you to quickly predict the system output for any parameter combination within the design space. Use the surrogate function to determine the optimum input settings for a defined set of goals and constraints (Nagar, 2018 & Schwarz et al, 2001).

4.8.1 DesignXplorer Implementation with Optimization

In module ANSYS the DesignXplorer provides a much more efficient approach by providing a response surface that is based on a finite element solve combined with the use of mesh morphing (Jovanovi, 2011). The ever-increasing demand to lower the production costs due to increased competition has prompted engineers to look for rigorous methods of decision making such as optimization. Optimization in its broad sense can be applied to solve any engineering problem.

And methods coupled with modern tools of computer-aided design are also being used to enhance the creative process of the conceptual and detailed design of engineering systems. There is no single method or technique for solving all optimization problems efficiently. Hence a number of optimization methods have been developed for solving different types of optimization problems. It is in the entire discretion of the engineer to choose a method which is computationally efficient, accurate, and appropriate for design problem (Fiedler, 2014; Jiaqiin et al., 2007).

Input parameters can either come from design modeler or from various CAD system these parameters can be in terms of thickness, length, and depth, etc. they can also come from mechanical in terms of force, materials properties, etc. the output parameters are calculating in mechanical and can, for example, be in terms of total mass stress or response. After setting up an analysis with several input parameters and out parameters there are the steps that can be run within designXplorer.

4.8.2 Design of experiments and response surface modeling in optimization

Optimization methods known as mathematical programming techniques are generally studied as a part of Operations Research. Mathematics scientific methods and techniques to decision making problems to establish the best or optimal solutions. The design of the experiment is one such well-defined area of operation research. This method enables one to analyze the experimental data and build empirical models to obtain an accurate representation of the physical situation. Design of experiment (DOE) and response surface modeling (RSM) is made to minimize the computational expense incurred in solving such a problem.

4.8.3 Set up generate for the response surface and response surface optimization.

During set up and generate the input and output parameters are known in which the geometry of the parameter of thickness radius and depth of the hook is the input parameter and the safety factory minimum and mass are the output parameters for response surface. In figure 4.2, shows the ANSYS19.2 workbench for designxplorer set up for analysis. The purpose of the response surface to interpolate value the multiple dimensions.

To define the design of experiments (DOE) is used to effect a design space parameters for crane hook so that a statistical model can be built to predict responses like the maximum stress, safety factory minimum, total deformation, and solid mass of a given design. DOE is useful when one can only sample a limited number of points (i.e., run a limited number of simulations). The key idea of DOE is to ``spread out'' the samples so that the resultant statistical model has low uncertainty in its model estimation and thus high accuracy in prediction. Define parameters and response; To conduct DOE for a given model first define the list of design variables and objectives that we care about (In Ansys, these are called input and output parameters). To do so, open the “Project Schematic” window, which shall look like figure 4.2.

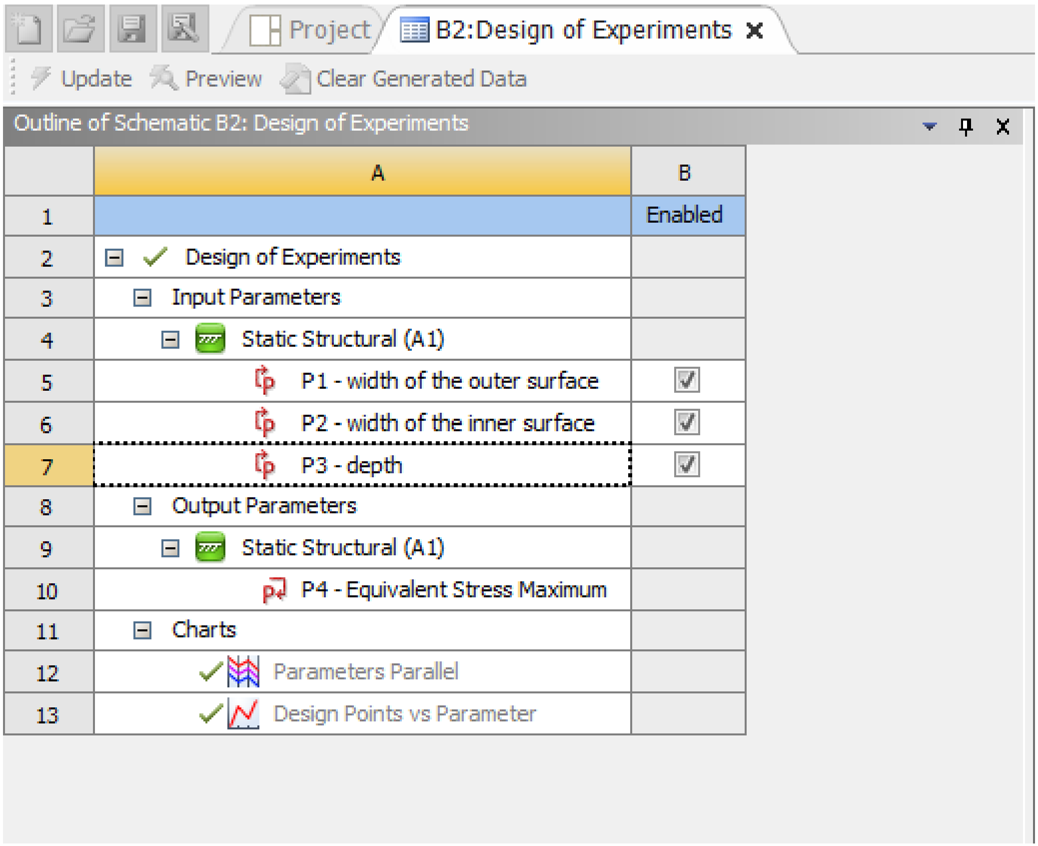


Figure 4. 6 The outline of the schematic Design of the Experiment.

Choose a Design Exploration method, in the design exploration window, find the response surface. This will allow us to perform DOE for the purpose of creating a predictive model, called a response surface. Drag the "Response Surface" tab from the Toolbox on anyone dashed box near "Parameter Set" this is shown in figure 4.4. The Design Of Experiments (DOE); is the procedure to collect a representative set of data relating to a process, technology, or an engineering project, adequate, data to calculate a response surface, and then executing an optimization (for optimization of a Response Surface too). The Response Surface accuracy will depend plenteously on the DOE scheme adopted, and in particular, the number of Design Points have been computed.

The Parametric correlation; uses the responses that can be easily obtained as the study offers an excellent graphical approach through the Parameters Correlation and the parametric correlation study allows two very important things: which input parameters have the greatest impact on design and identifies how the input-output relationship becomes linear or quadratic. The finite element analysis has performed, and the influence and impact of the input parameters to the output parameters is described.

Defining the parametric simulation model with ANSYS workbench.

- Start ANSYS workbench 19.2
- Insert a static structural (ANSYS) system in the schematics
- Right-click on the geometry then browse and select the crane hook file
- Select the model and double click the hook item.
- The select model then edit or double click
- Note that the project page now contains a parametric set bar that holds the DM parameters.

In this geometric parametric are defined from ANSYS DesignModeler and are automatically collected in the parameter set regardless of their name. Geometric parameters can also be defined directly from the CAD system using a prefix to flag the ones that are relevant for the simulation. And also mix parameter source some could be imported from the CAD model and additional ones defined in ANSYS DesignModeler.

Under mesh, insert a sizing, pick the hook body and set the size of 3mm and insert a mapped face meshing and select all faces. Under static structural insert cylindrical support and free the tangential degree of freedom(radial and axial should be fixed). Insert a force of 6 tons in the y-direction (set the force definition to the component). The force is applied on the small surface inside the hook. Under solution insert total deformation, maximum equivalent stress fatigue tool. then solve the model optional.

Output parameters are quantities maximum equivalent stress and safety factor minimum. To set these as parameters, go to the solution under the properties of the bodies check the maximum equivalent stress and minimum safety factory.

Parametric variation for the crane hook, going to perform the deterministic analysis of the hook for the following parameter ranges

- Ds- width of the outer surface 30mm to 40mm
- Ds -width of the inner surface 70mm to 80mm
- Ds- depth 85mm to 95mm. do not need to specify how many points are to be taken for each parameter the DOE method will give us necessary points.

Setup of the response surface, go back to the schematic page and insert a ‘’response surface’’ a cell from the design exploration toolbox. Select the design of experiments then set the upper and lower bounds of each input parameter. Once simulation have been performed select the ‘’response surface’’ then ‘’update’’

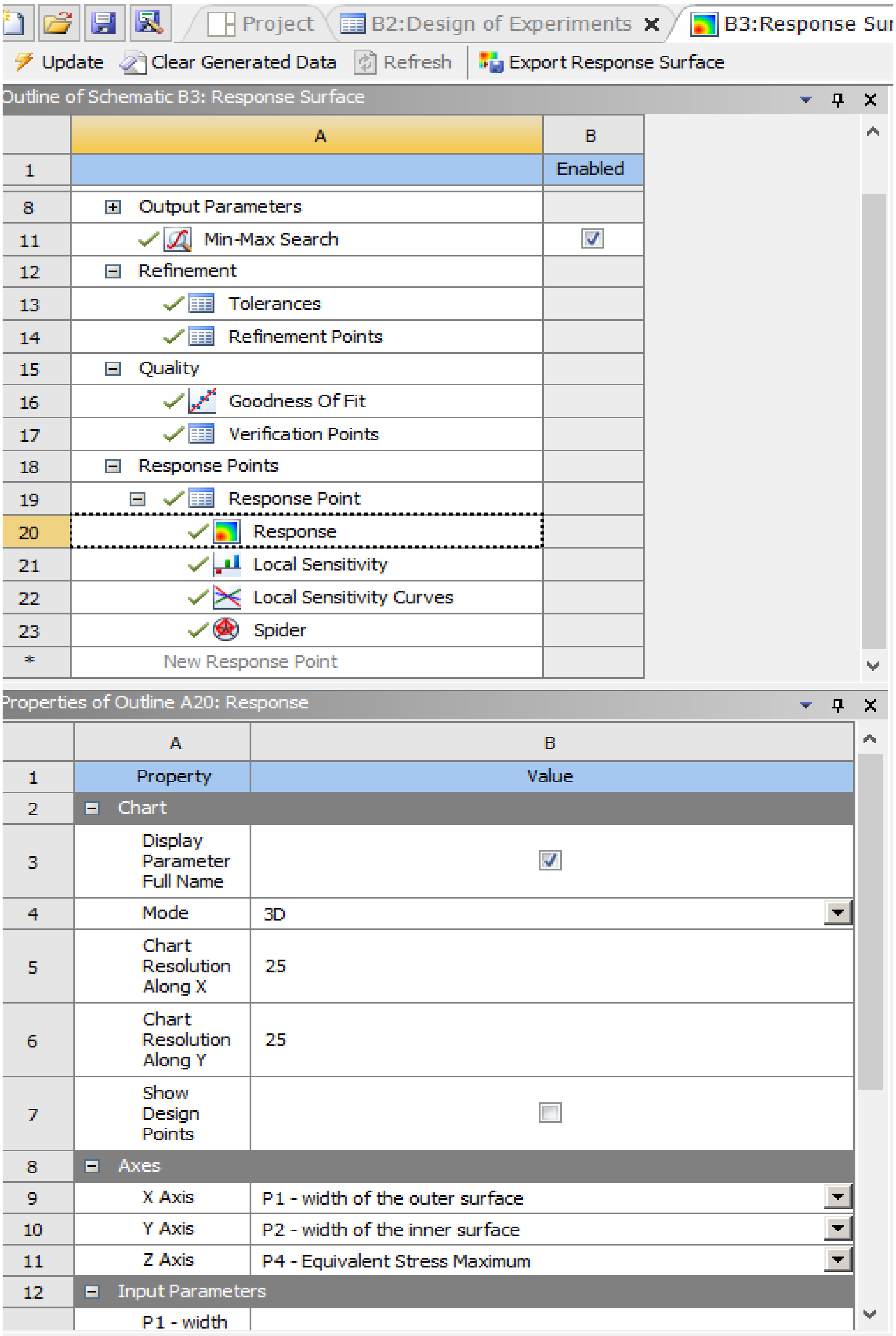


Figure 4. 7 Outline schematic of the response surface set-up.

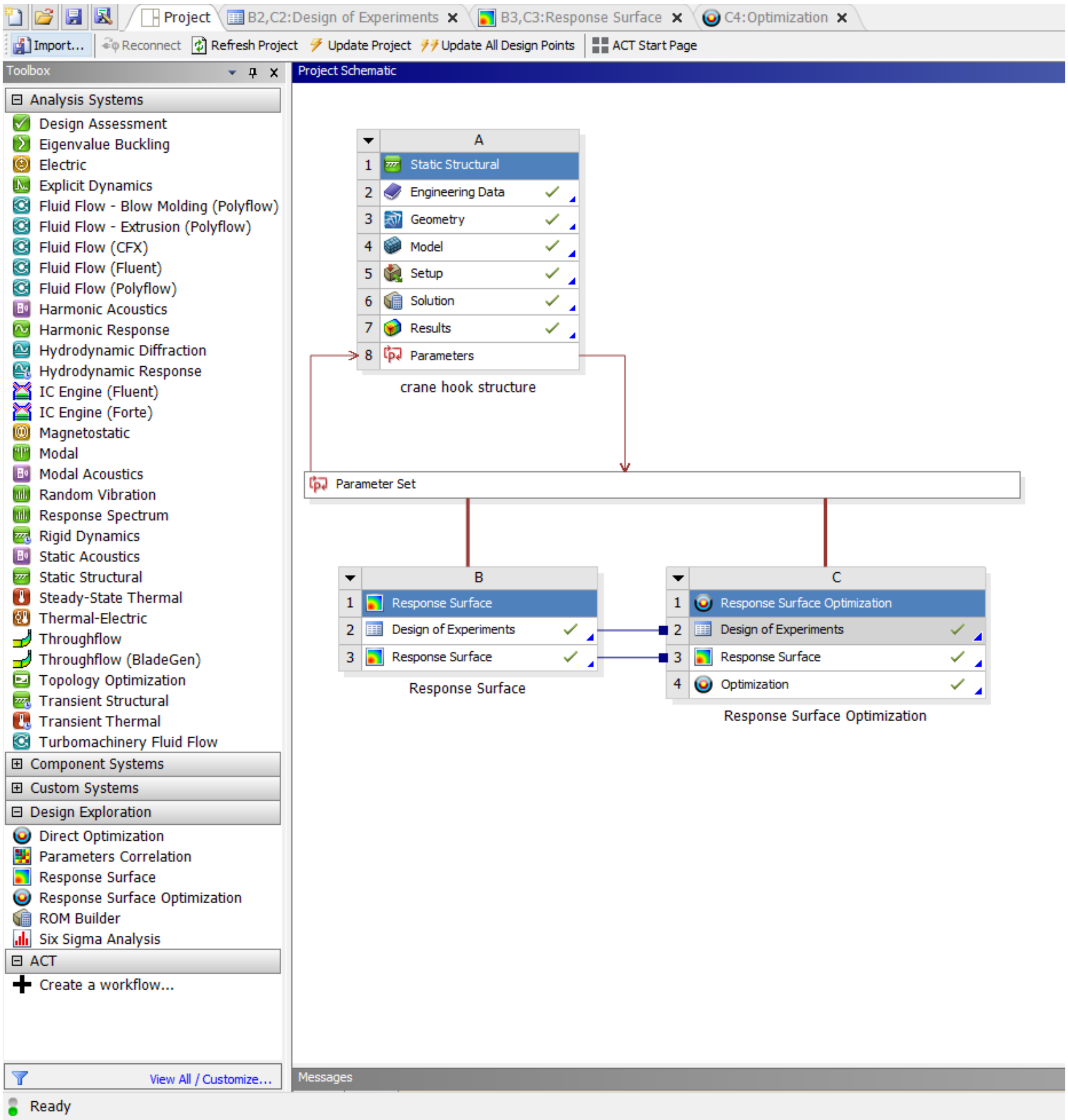


Figure 4. 8 Set-up demonstration for surface response optimization.

CHAPTER. 5. RESULTS AND DISCUSSION

This chapter deals with the discussion of the results obtained to determine the effects of its topology optimization of hook crane design depends on the conceptual modeling frameworks. The CAD geometry of the existing hook crane is simulated in ANSYS19.2 workbench topology optimization analysis and DesignXplorer implementation. The results obtained from the model and simulation analysis are explained briefly.

5.1 Structural analysis of the crane hook.

The results of static structural analysis are obtained from the FEA software ANSYS19.2. The results involve maximum von-Misses stress, optimum weight, total deformation, and equivalent elastic strain of the crane hook designs that are used to compare the crane hooks with each other's. However, the main comparison was done based on the stress and weight of the hooks because the safety of the hook depends on the stress-induced on the hook.

5.2 Structural analysis of the original crane hook.

The static structural analysis of the original hook is cross-section is trapezoidal. Figure 5.1 shows the von-Misses stress, total deformation, and equivalent elastic strain of trapezoidal (original design) crane hook by applying a load of 6 tons. The stress analysis has been done by ANSYS software, where the results have been presented by contours and numerical values. For the trapezoidal crane hook, the maximum von-Misses stress result from ANSYS is 92.216 MPa. And also, the maximum total deformation and equivalent elastic strain results are 0.52652 mm and 0.00046 respectively. The weight of the trapezoidal (standard) hook indicated in the ANSYS workbench properties of geometry is 15.75 Kg.

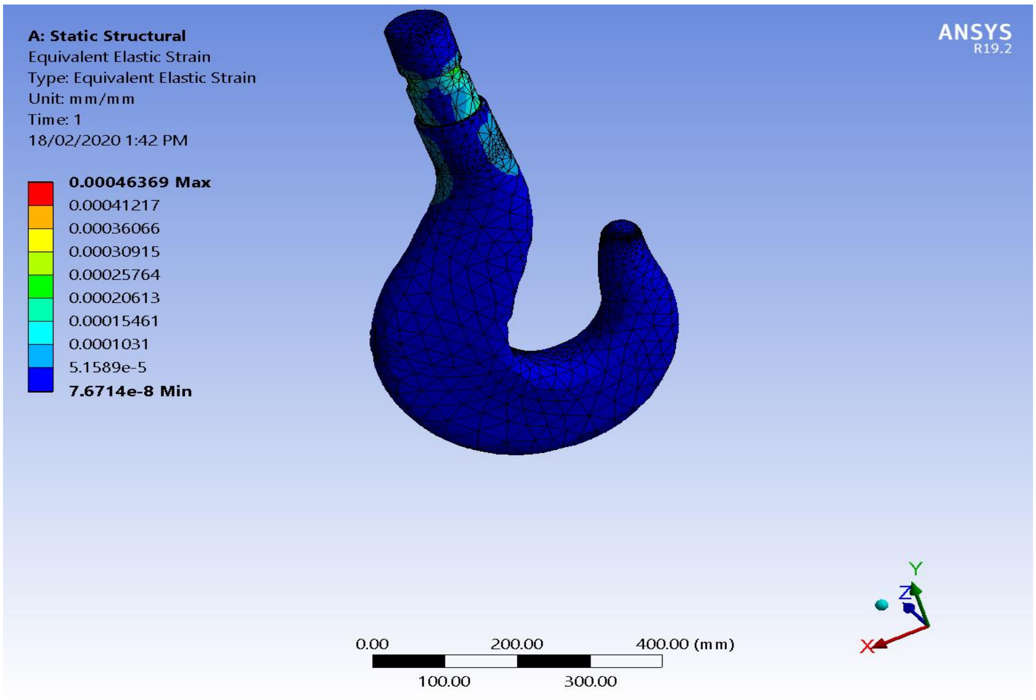


Figure 5-1. Static structural analysis of the equivalent elastic existing crane hook

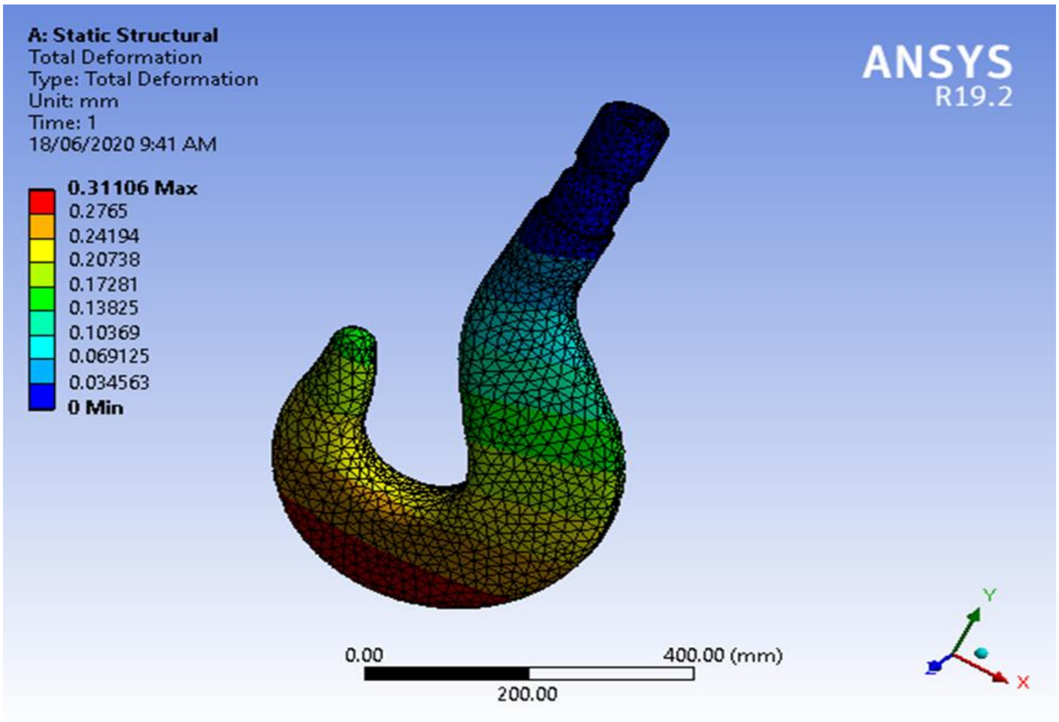


Figure 5-2. Static structural analysis of the total deformation existing crane hook

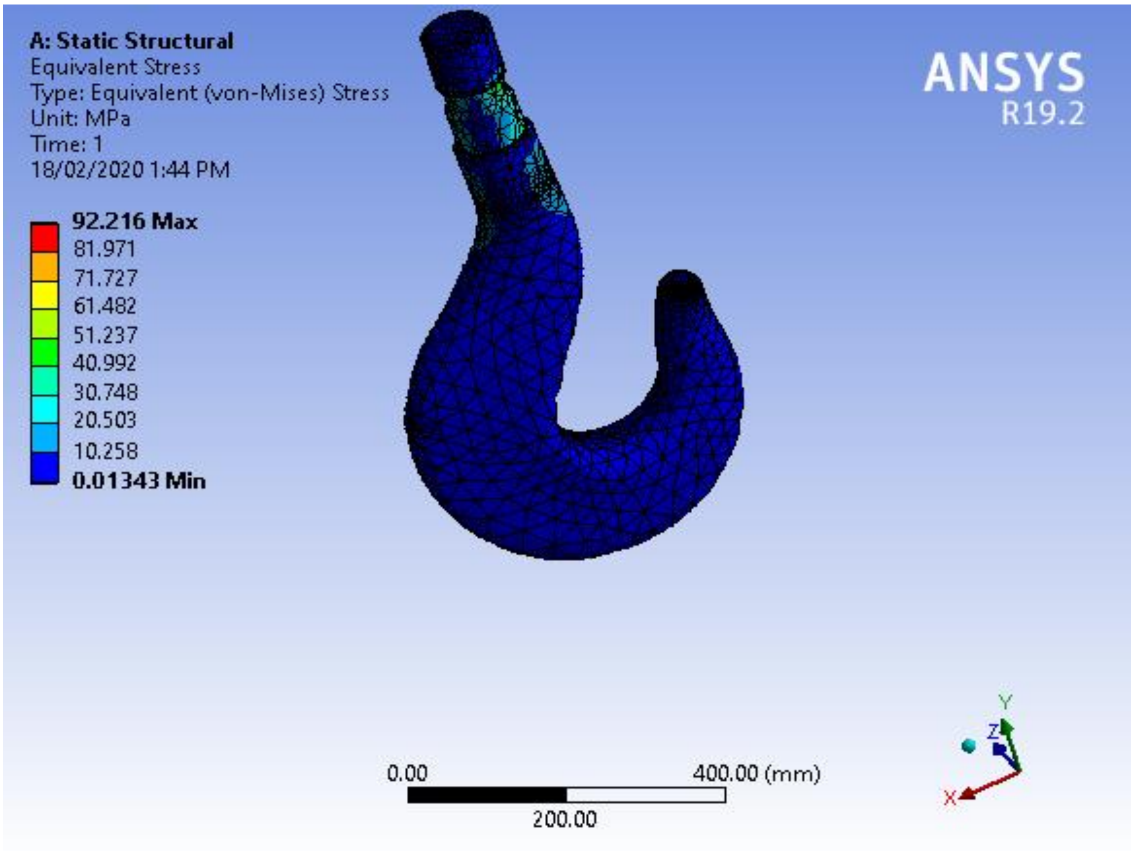


Figure 5-3 Static structural analysis of the equivalent (von-Misses) stress

5.3 Topology optimization in ANSYS19.2 workbench results

Figure 3-2 a) shows the crane hook from the site and Fig. 3.2 b) shows SolidWorks 2018 the 3D Model before the topology optimized design process executes. This 3D-CAD dimension of the tower crane hook is measured (by using vernier caliper) from the Afro-Tsion Constriction company, around Jimma University Main campus, construction site.

Due to the measured data from the original design is the structurally analyzed with the given loading conditions to see the stress and displacement distribution. Based on the stress and displacement distribution, the topology optimization removes material from areas that do not significantly contribute to carrying the applied loads. Based on the topology optimization results, the part is remodeled in CAD software. The new CAD model is then verified with FEA to carry the loads and to satisfy the design requirements.

If the model satisfies the verification, physical model verification is done using any of the physical prototyping methods. If not, the remodeling is done again until verification is done. The final design is then prepared for the final design. The process is employed in the next section, case study, to redesign a crane hook to show the potential of topology optimized design. In this, a crane hook shown in Fig. 5.6 is considered as a case study to show the potential of topology optimized design approach in reducing the weight of a product the original crane hook is based on the standard dimension design from the construction site.

Topology optimization applied by using the computational software package (ANSYS19.2 workbench) shown in Fig. 5.4 In this module of ANSYS workbench, during applying the topology optimization, several preference options govern the workbench behavior of topology optimization. Then, the material selection up to the results of static structural (A) into the topology optimization processes taken place in this module, by adjusting the material selection of steel alloy import the IGES CAD SolidWorks 2018 through geometry and display through the model then apply the necessary information for topology optimization needed in the WB. Finally, the topology density optimized for the crane hook material removal shown in Fig 5.5.

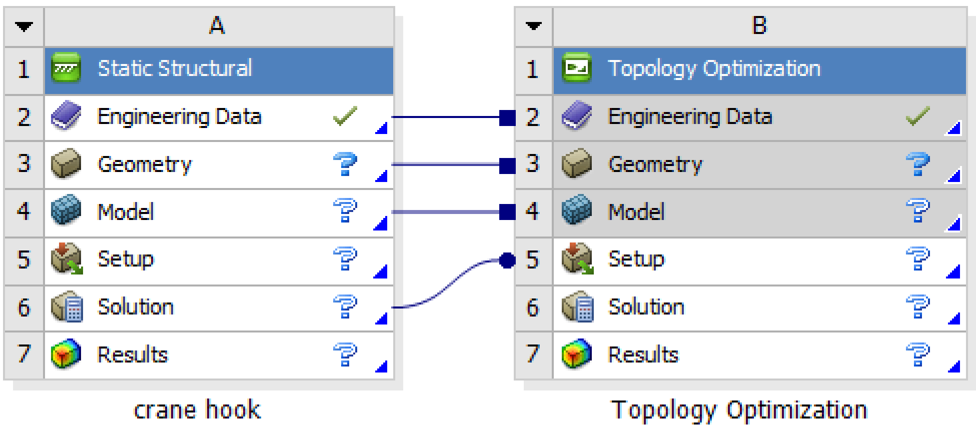


Figure 5.4 Topology optimization module on ANSYS 19.2 workbench.

5.4. Structural analysis of the remodel (optimal) crane hook.

The large areas of the hook that are shown in blue color shown in figure 5.3, in the contour plot indicate inefficient use of material. It is very likely that these areas of the part need material removal as they have a negligible effect on the performance of the hook. In this static structural analysis, the remodel is the first topology optimized, and then the static structural analysis is implemented. The ANSYS analysis results of the new model crane hook are presented by contours and numerical values (as shown in Figure 5.6). For this crane hook the maximum Von-Mises stress, total deformation, and equivalent elastic strain are 95.007 MPa, 0.31775 mm, and 0.00047506 m/m respectively and the weight of the new model indicated in the ANSYS workbench is 13.67 kg. Table 5.1 shows the comparison of the results for the original design and the optimized design. The maximum stress, strain, deformation, and weight of the hooks are given in the table 5-1.

Figure 5.6, shows topology density optimization. The model is obtained after material removal in the low-stress region up to a safe design limit and the weight of the 3D model is measured from the original weight and after material removal. The models before the topology optimized model with finite element analysis and after topology optimized model with ANSYS19.2 are 15.75 kg and 13.678 kg respectively. Redistribution of the material topologically optimized model compared with before and after an optimized model. The after optimized model 6.685 percentage of reduction mass crane hook is obtained.

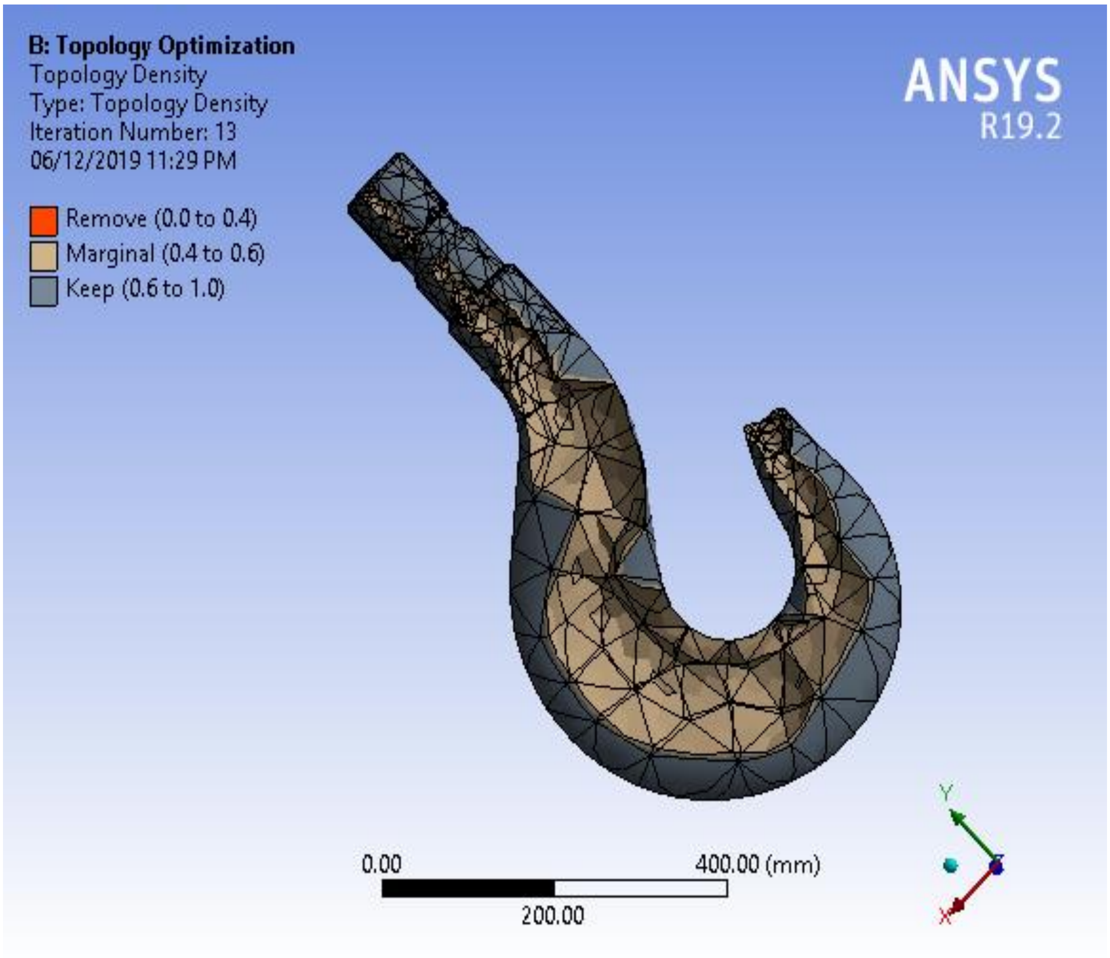


Figure 5-5 Topology density optimization with ANSYS19.2 workbench

After removal, the materials that have a negligible effect on the performance of the hook, redesign the hook crane is done based on the remaining material topography. In this study, the crane hook is redesigned using topology optimization approach considering loading conditions. The objective of the study to reduce the weight of the crane hook while satisfying all the design requirements due to the developed methodology of conceptual modeling.

Figure 5.6 Crane hook of a 3D model after topology density optimization SolidWorks 2018.

The newly designed part will have to sustain the same mechanical load while fulfilling the same design requirements. The final design has to be verified with the given design criteria that are the yield strength. The von Misses stress values for all the load cases should not exceed the yield strength. The structural verification analysis is done on ANSYS R19.2 as shown in Figure 5.7, the von Misses stress for all the cases are below the yield strength of the material. The final design satisfies the yielding condition with the safety factor not less than 1, the minimum safety factory of the crane hook to avoid failure. The final design resulted in 6.685 % weight reduction, which is from 15.75 kg original part to 13.678 kg optimized one.

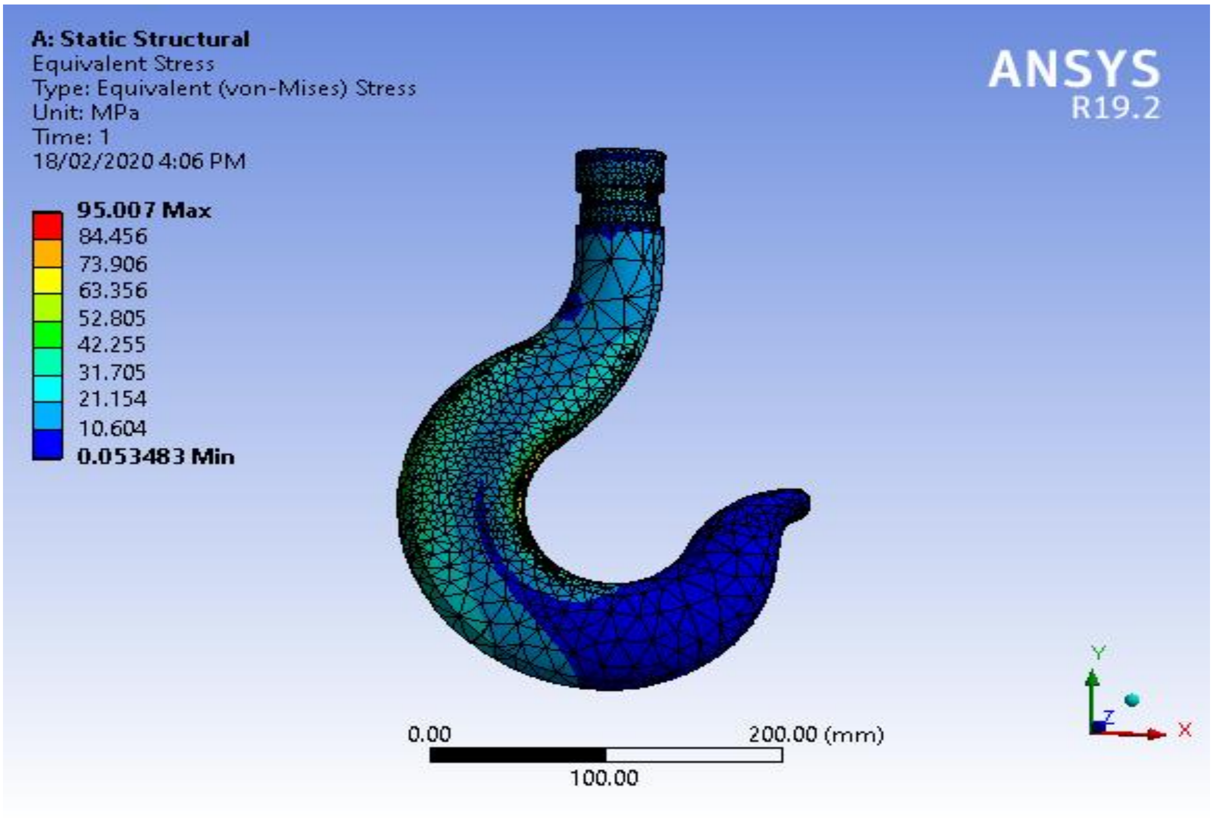


Figure 5.7 Static analysis of optimal crane hook with Equivalent (Von-Mises) stress

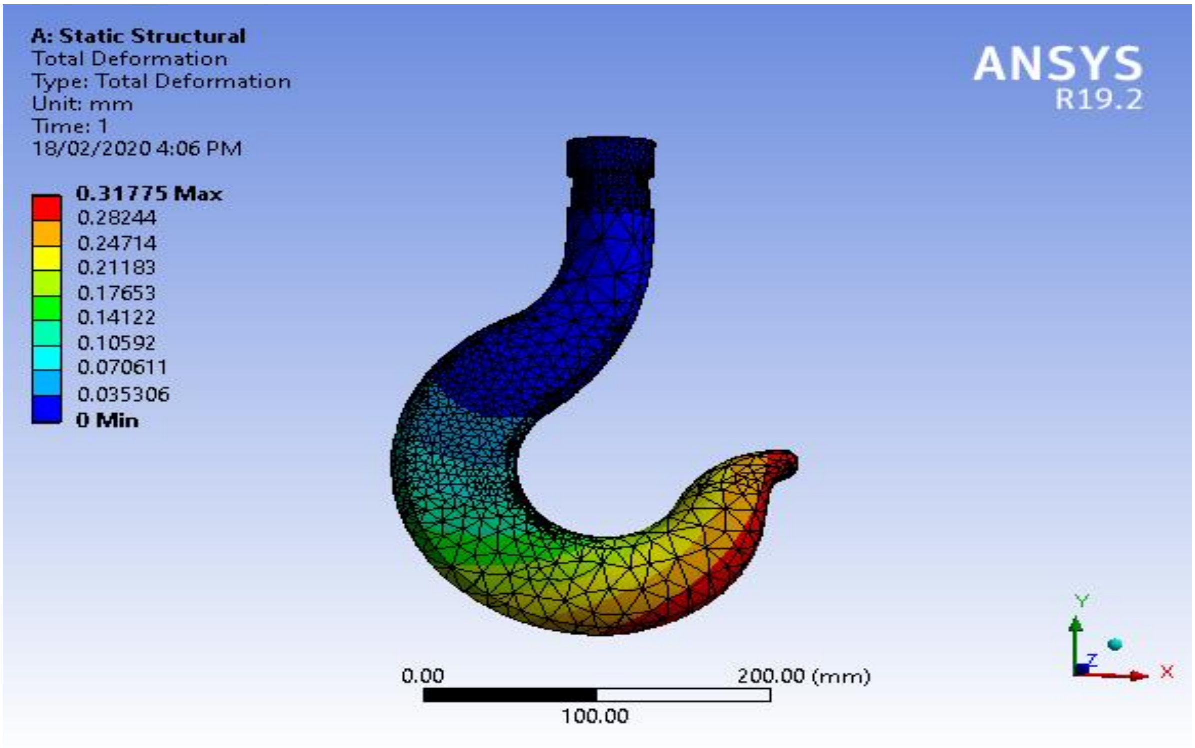


Figure 5.8 Static analysis of optimal crane hook with Total deformation analysis

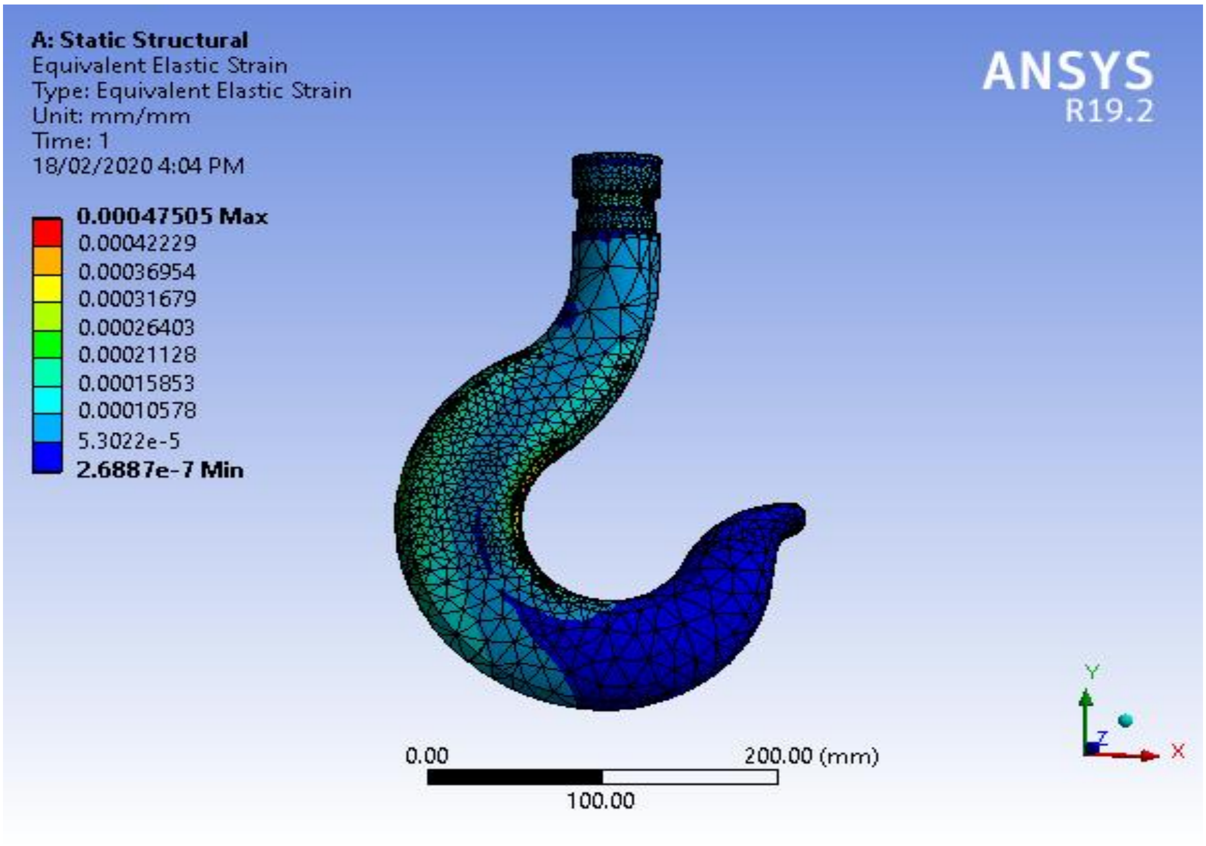


Figure 5.9 Static analysis of optimal crane hook with equivalent elastic strain analysis

Table 5-1 Comparison of existing and new model of crane hook depending on the static structural analysis with ANSYS.

Crane hook stress analysis	Remodel hook	Original hook
Load (tons)	6	6
Maximum total deformation (mm)	0.31775	0.31106
Max. equivalent elastic strain (m/m)	0.00047506	0.00046369
Max. equivalent (von-Mises) stress	95.007	92.216
Weight (kg)	13.67	15.75

Figure 5.7, shows the implementation of the topology optimization processes depending on the conceptual modeling frameworks developed in Figure 3.2, which showed the schematics of the conceptual modeling frameworks methodology within a flowchart of the topology optimization process. The figure shows all processes, starting from the 3D model in SolidWorks and then meshing to the simulation of FEM within ANSYS analysis software. In general, structural

optimization has huge potential benefits in the product development process. Topology optimization, in particular, has the following benefits in the design process.

- Creating lightweight.
- Reducing time-to-market.
- Saving a huge amount of material.
- Saving a large amount of processing energy.
- Reducing physical prototype build

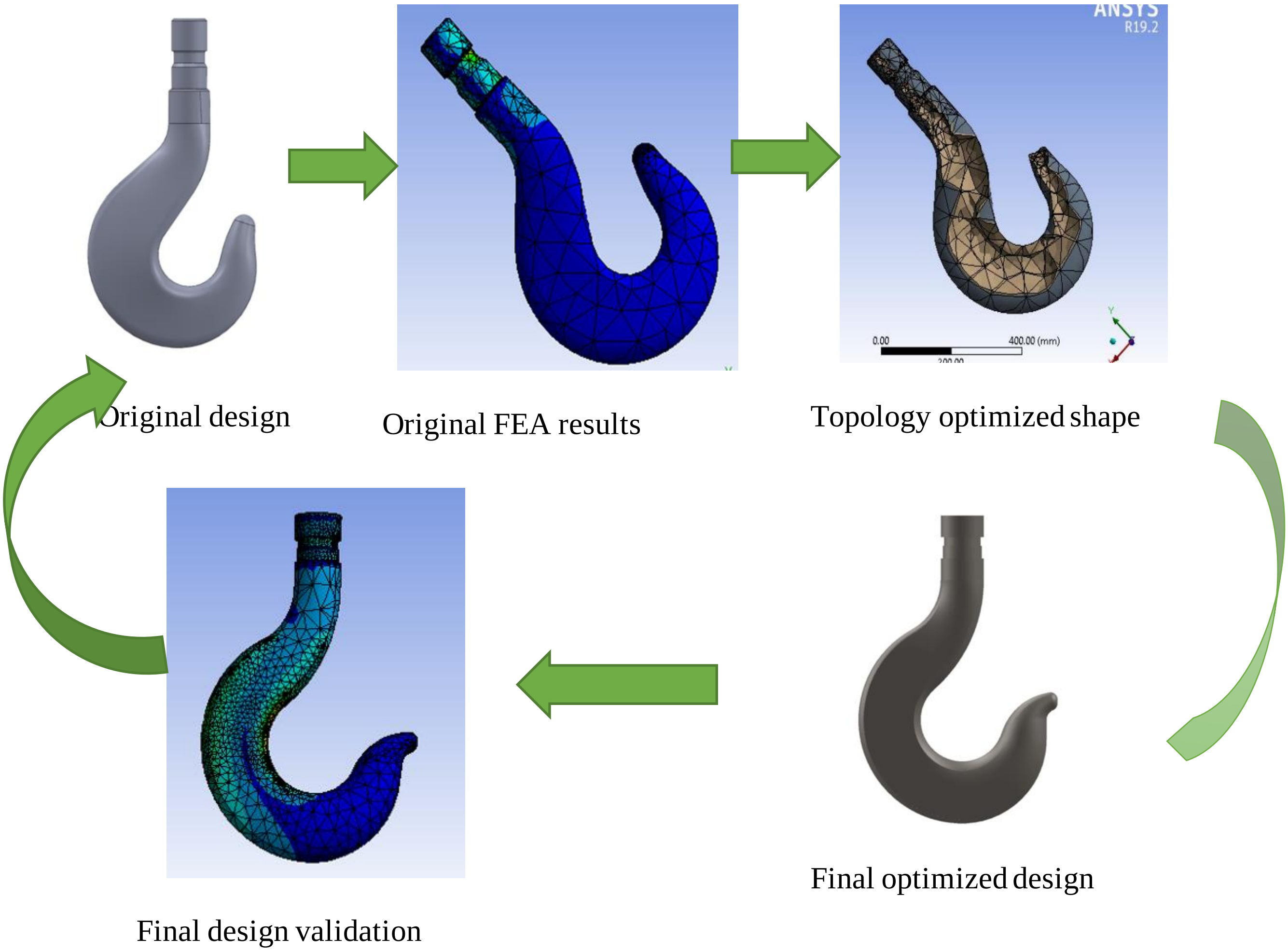


Figure 5.10 Topology optimization processes

5.5 Stress and weight comparison between the existing and optimal crane hook.

From the topology optimization, as it is known, the reduction of weight of any mechanical part has more advantages and increasing stress is reversed because it will decrease the lifetime of the material (hook). Therefore, the comparison to be based on the stress and weight of the crane hook.

Table 5- 2 Crane hooks comparison with weight and maximum stress.

Crane hook	Max. von-Mises stress in (MPa)	Weight of hook in (kg)
Original (existing) model	92.216	15.75
Optimal (new) model	95.00	13.69

Table 5-2 shows the comparison of the stress and weight of the modelled crane hooks. During topology optimization, models are comparing with the original crane hook from the site.

5.6 Parametric via finite element analysis results

The simulation in ANSYS resulted in the finite element parameters listed in Table 5-2. The first step in pre-processing is to prepare a CAD model of the crane hook. CAD modeling of any project is one of the most time-consuming processes and the base of any project. Finite element software will consider shapes, whatever is made in the CAD model. While most of the CAD modeling software has capabilities of analysis to some extent, most of the finite element software has capabilities of generating a CAD model directly for analysis. In this study, the CAD modeling of the complete crane hook is generated using SolidWorks 2018 to utilize the modeling capabilities of complex geometries.

Table 5- 3 The mesh quality with the number of the node and element number.

Property	Parametrize topology optimization
Nodes of number	179968
Elements of number	105110
Element size (mm)	3

5.7 Crane hook fatigue life analysis results

5.7.1 Damage estimation and Fatigue life with the finite element method.

Predicting fatigue damage for structural components subjected to variable loading conditions is a complex issue. The results from this approach do not take into account the effect of load sequence on the accumulation of damage due to cyclic fatigue loading. Since the introduction of the linear damage rule many different fatigue damage theories have been proposed to improve the accuracy of fatigue life prediction

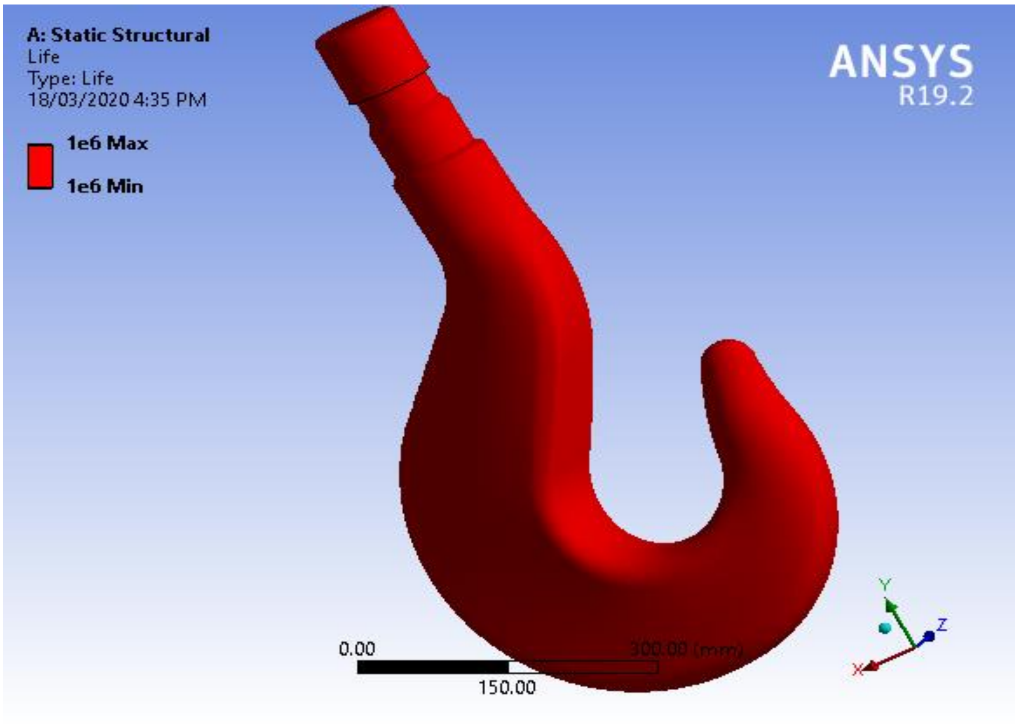


Figure 5.11 Life analysis for existing model hook

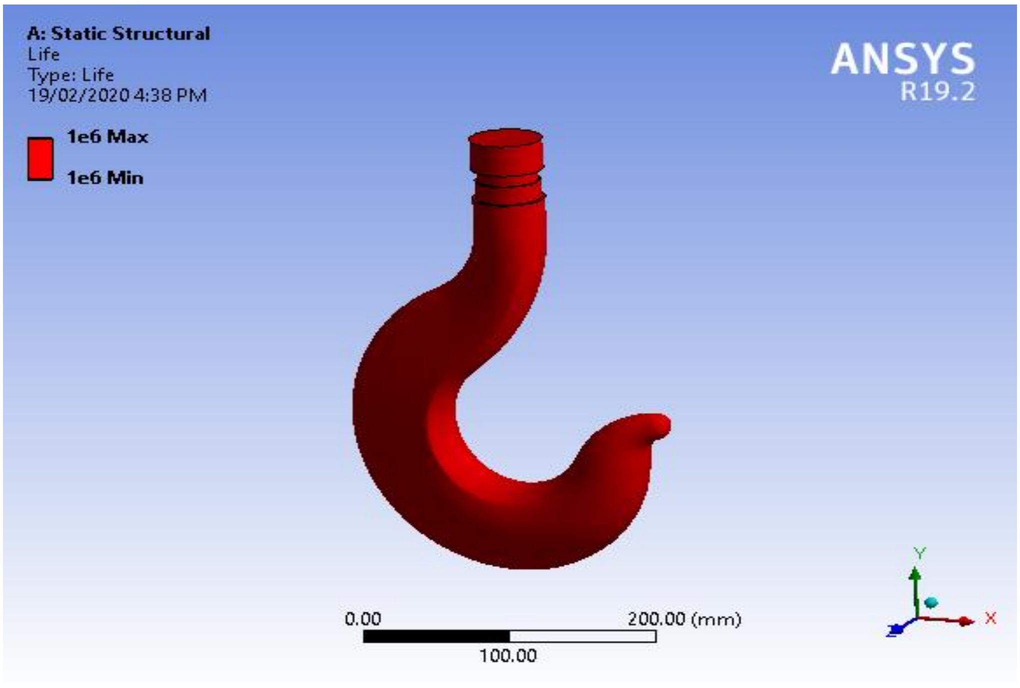


Figure 5.12 life analysis for new model hook

According to the tower crane hook working environment, the crane is operating over the head of the human beings and other fixtures (machines). So crane hook design is highly critical and components must be designed for safe operation under all conditions. A failure of any component under operation can lead to serious accidents. Therefore, critical components such as the hook must-have a design for the infinite life of cycles (more than 10^6 cycles). In general, to define the fatigue damage is the ratio of the design life to the available life. So a value of fatigue damage greater than one indicates safe fatigue damage the design life has reached. Results in the plot contour of Figure 5.13, and figure 5.14 show the fatigue damage for the fatigue analysis of both original and new model crane hook.

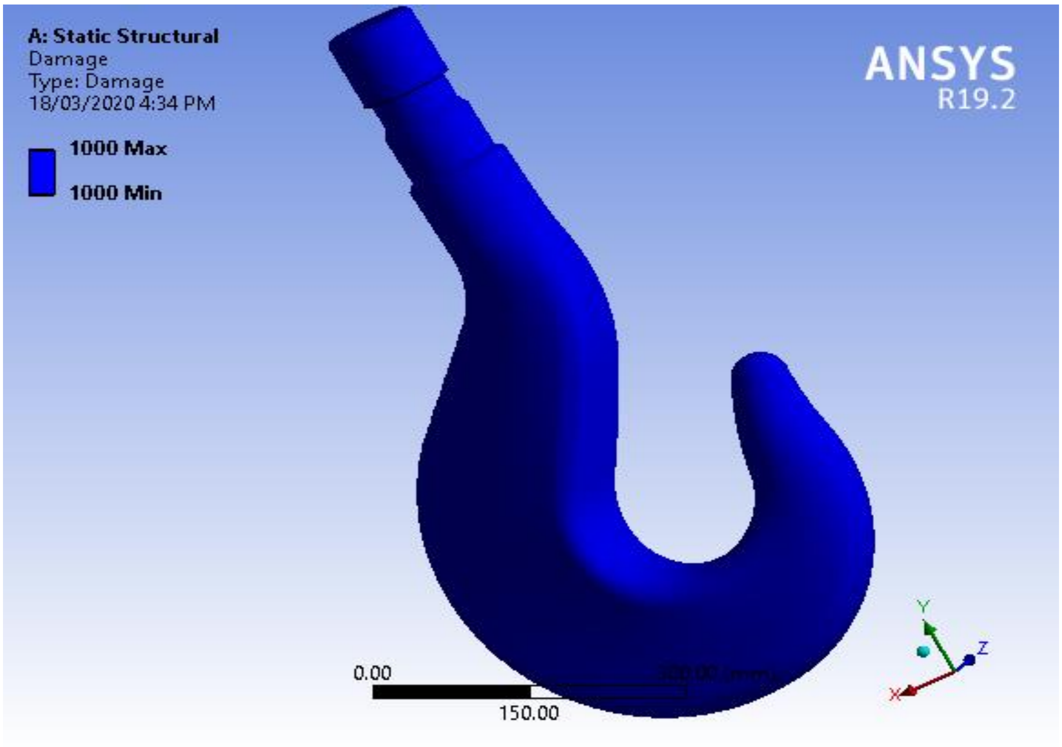


Figure 5.13 Damage analysis existing crane hook

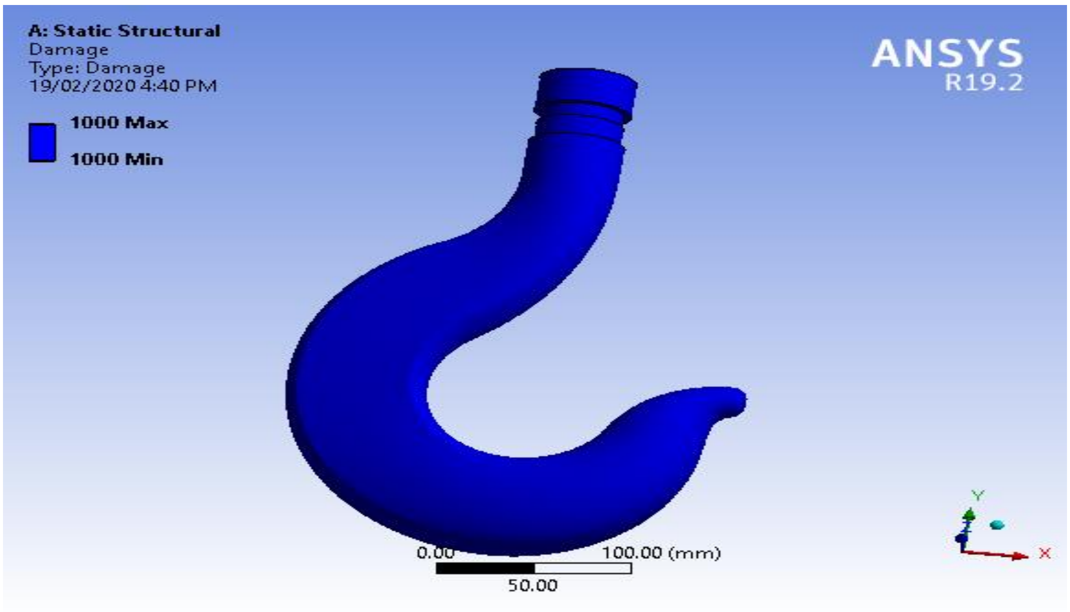


Figure 5.14 Damage analysis new model crane hook.

5.7.2 Safety factor of the hooks

The results contour plots of Figure 5.16 and Figure 5.17 show the maximum and minimum safety factor of all the modelled crane hooks. For fatigue safety factor values less than one indicates failure before the design life is reached. Crane hook is subjected to loading and unloading conditions with different loads. (less than 6ton) of the design load of the crane hook. When the applied load increases the safety factor decreases. The maximum value of safety factor for both models of crane hook is 15 and almost equal minimum safety factor(for original 1.0567 and the new model is 1.0552).

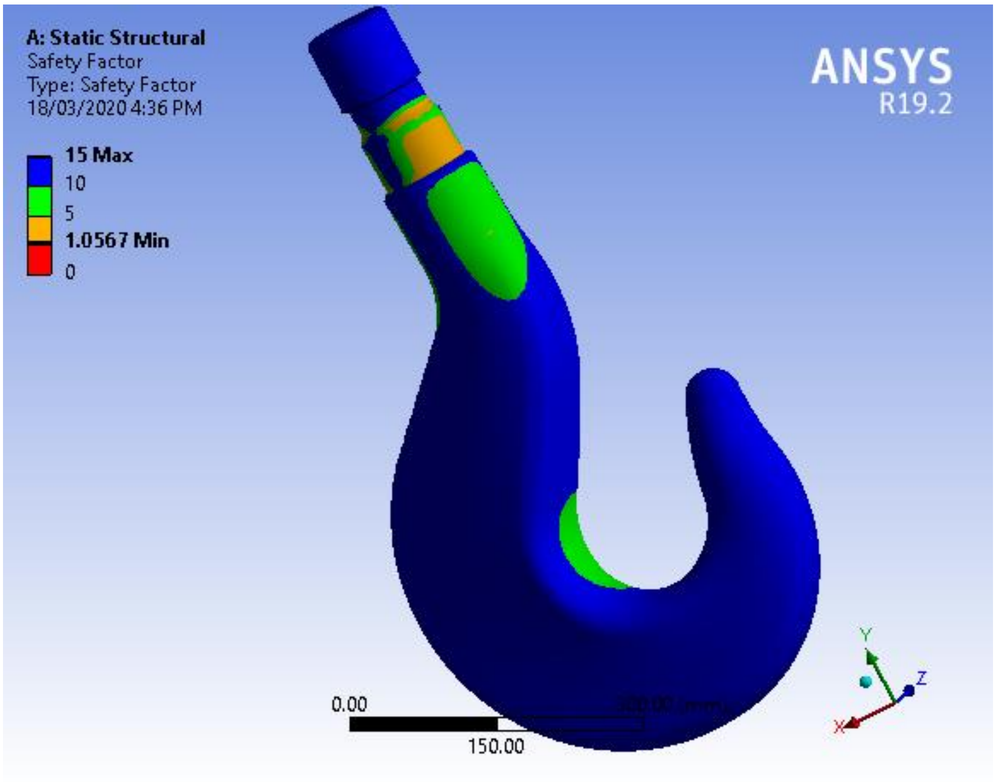


Figure 5.15. Safety factor minimum contour plot of the existing crane hook.

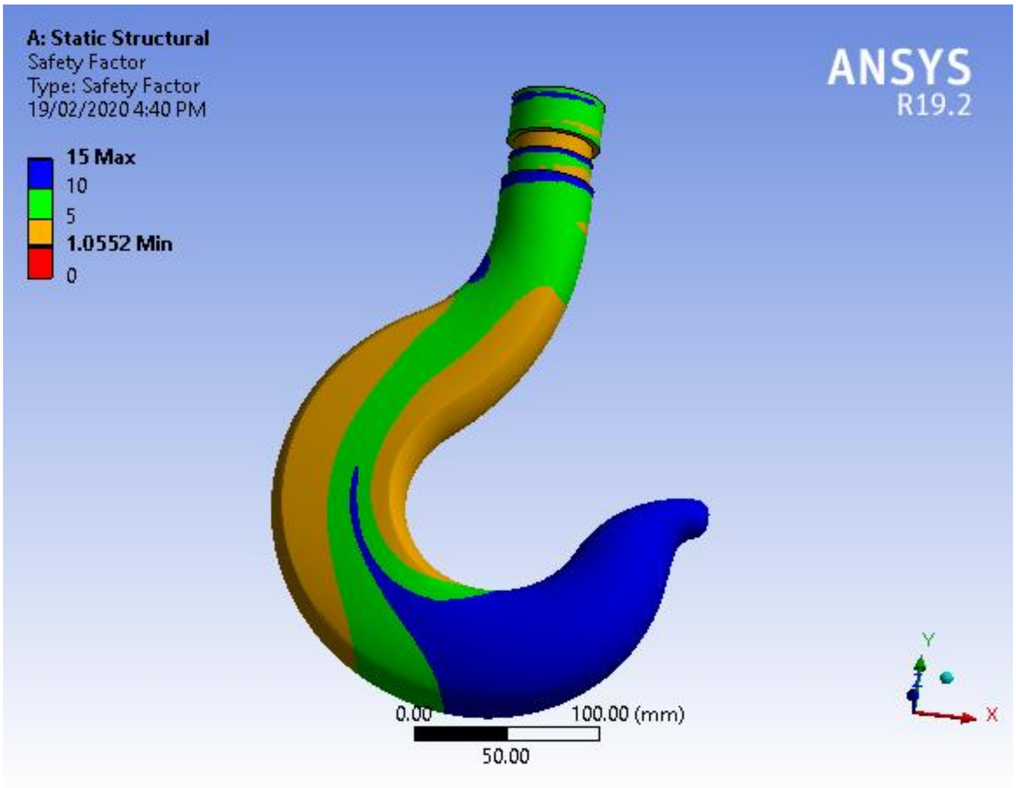


Figure 5.16. Safety factor minimum contour plot of the new model crane hook.

The minimum safety factor result of the new model crane hooks is greater than one, at 6 tons applied load. This means it cannot fail before the design life. But the minimum value safety factor of the original design is 1.0567 which implies that both have almost equal minimum safety factories.

5.8 DesignXplorer analysis results.

Designxplorer is a simulation tool or module in ANSYS workbench that is implemented using surface response sensitivity and design of the experiment to define the input and output relationship. In the design of crane hook analysis, the factor safety and maximum equivalent stress analysis as output parameters are related to the width of the outer surface, a width of the inner surface, and depth of the cross-section of the crane hook which is the input parameters. These input parameters decide the surface of the crane hook and the output relation minimum factory of safety and maximum equivalent stress decide the life and strength of crane hook. In this thesis work, the main objective is improving the strength and the endurance requirements of the crane hook. So, the maximum equivalent stress and safety factor in relation to the design optimization of the crane hook are determined.

Design of experiments: When the stress is reduced the safety factor is increased and in this case thickness, radius, and depth of the crane hook affect the stress and this leads to the minimum safety factor and still minimizes the weight, and cost of materials. The goal is to find the exact combination of the parameter values to get minimum mass and safety factor requirements.

5.8.1 Response surface modeling and optimization.

Based on the number of input parameters (ANSYS workbench-simulation), a given number of solutions (design points) are required to build a response surface. A design of experiment determines how many and which design points should be solved. Once the required solution is complete, a response surface is fitted through the results allowing the design to be queried where no hard solution exists. In the outline of the response surface, then it is possible to see a response points folder defaults response points under it (usually the center of the design space)

Each response point can be affected by different charts.

- ✓ Spider chart to examine the value and variation of all output parameters on a single graph
- ✓ Local sensitivity to examine the weight of each parameter around the response point.
- ✓ Response 2D or 3D graph represents the variations of one output with the response to one or two input parameters.

Design points in X-axis and minimum safety factor in Y-axis, for the checking value for each design point at which values of the minimum safety factor are the safe value. Design points and output parameters are shown in figure 5.17, this output is a minimum safety factory. At this point 1.11 to 1.12 the safety factory shows at the design points at 4, and 9.8 to 11.3 safety factor minimum which is safe. If the minimum safety factory is which is less than one the crane hook fails. So figure 5.17, shows the input and output relation of the design point and safety factory.

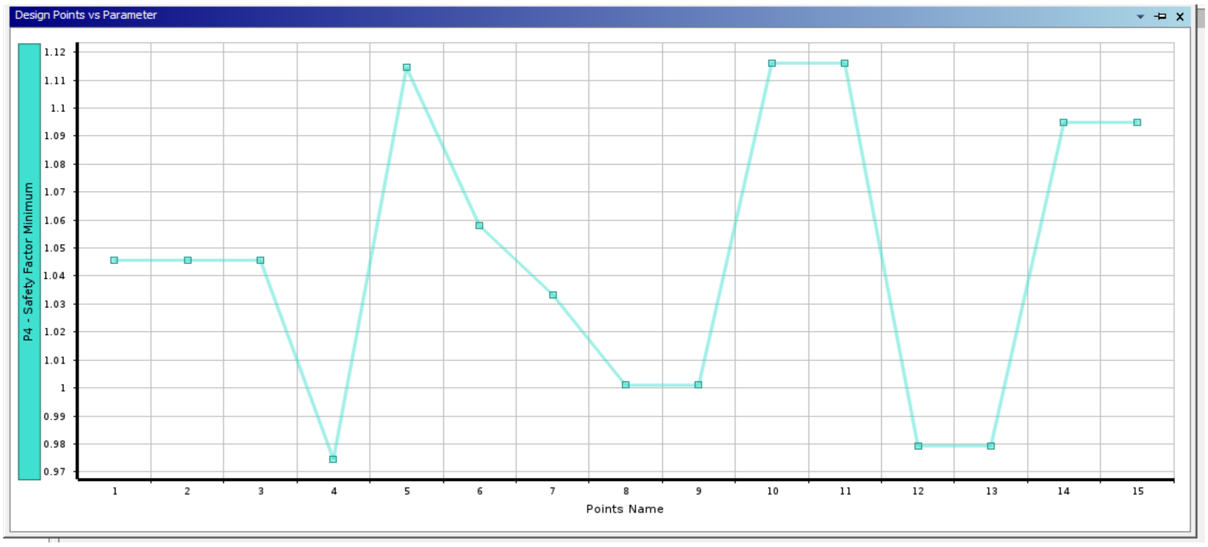


Figure 5.17 Plot of minimum safety factor vs design points.

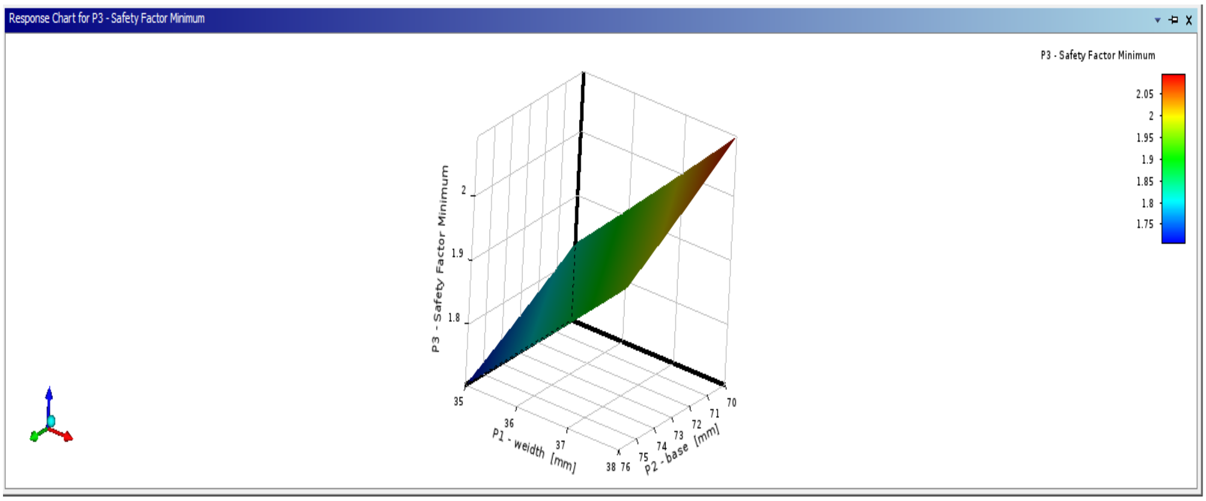


Figure 5.18.The 3D response chart minimum safety factory

5.8.2 Sensitivity of both minimum safety factory and maximum equivalent stress.

Figure 5.19 shows the Sensitivity of minimum safety factory and max. equivalent stress. This point implies both safety factory minimum and equivalent stress maximum with the impact of the input parameters of the depth, width of the inner and outer surface with a very small impact on the hook with output parameters.

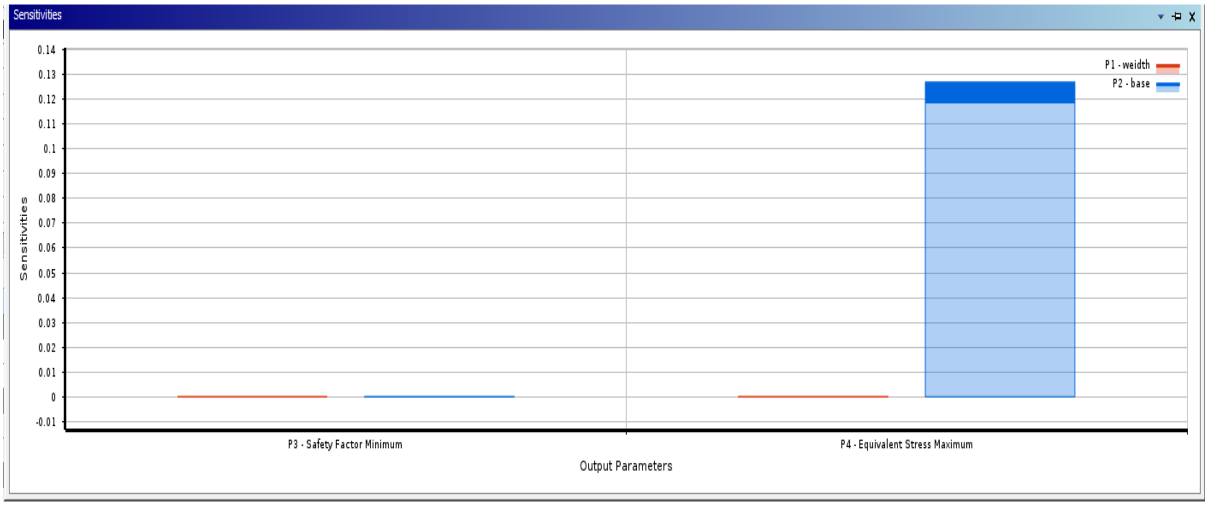


Figure 5.19. The sensitivity of minimum safety factory and max. equivalent stress

5.8.3 Response surface local sensitivity curve of safety factory minimum

There are two types of Sensitivity in response surface sensitivity (1) local sensitivity and (2) local sensitivity curve. In Figure 5.20, the ‘’X’’ axis represents the design points and the ‘’Y’’ axis stands for p4-safety factory minimum and represents the local sensitivity curve, while the black squares are the response points. As shown in figure 5.20, the P2-width of the inner surface shows the highest sensitivity, p1-width of the outer surface has neutral sensitivity and p3-depth has the lowest sensitivity.

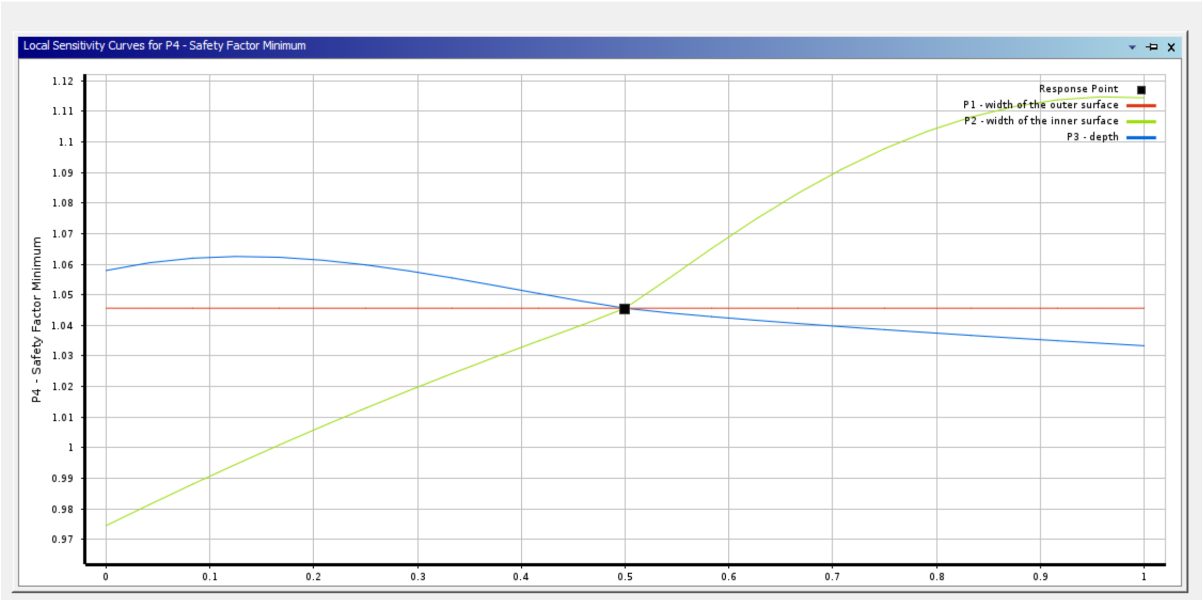


Figure 5.20. Local sensitivity curve for the minimum of the safety factory

5.8.4 Comprasion of local sensitivity of the safety factory minimum.

Figure 5.21 represents the response local sensitivity chart, the impact of both input and output. This shows the sensitivity range across the chart that the output correlated with the input that shows the bar heights with sensitivity. The negative correlated influence the inversely, neutral points have the little impact, and all the positive shows the proportional impact on the optimization of the crane hook. For this reason, a P3-depth decrease in the safety factory minimum increases, p2- width of the inner surface increase the safety factory minimum is also increases and p1-width of the outer surface has little impact on the safety factory minimum.

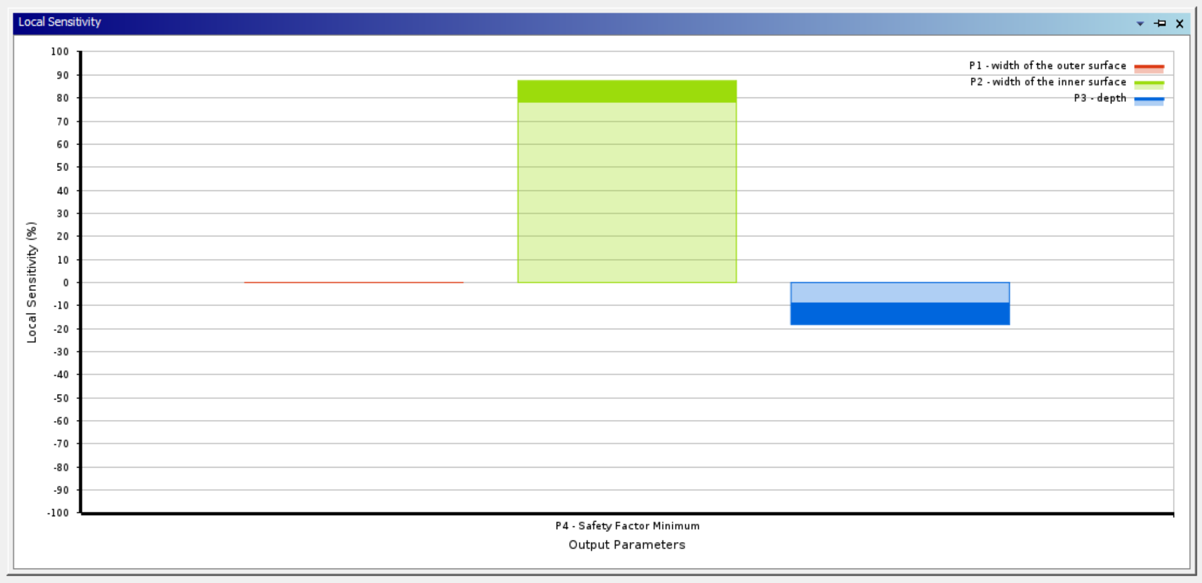


Figure 5.21. response surface local sensitivity with safety factory minimum

According to (Rathore, & Chandrakar., 2012) study the Approach to optimize ANN Meta model with Multi Objective Genetic Algorithm for multi-disciplinary shape optimization. The safety factor is mostly influenced by the thickness and depth of the hook, and then by the lower radius and angle has no effect on it. Thickness or depth increase will increase the safety factor, while the radius will lower it. The parameter sensitivity for the output functions are shown in figure below. The figure 5.21, and figure 5.22, simailrites is depends on the depth, angle and radius of crane hook for the figure 5.21 depends on the width of the outer surface, the width inner surface of the crane hook. In case of this thesis the safety factor minimum there posetive impact, negative impact and neutral is there. Then in case of study of (Rathore et al., 2012) the lower radius is the negative impact and the posetive impact is thickness and depth of crane hook The mass is influenced by all parameters increase of the parameter increases the mass

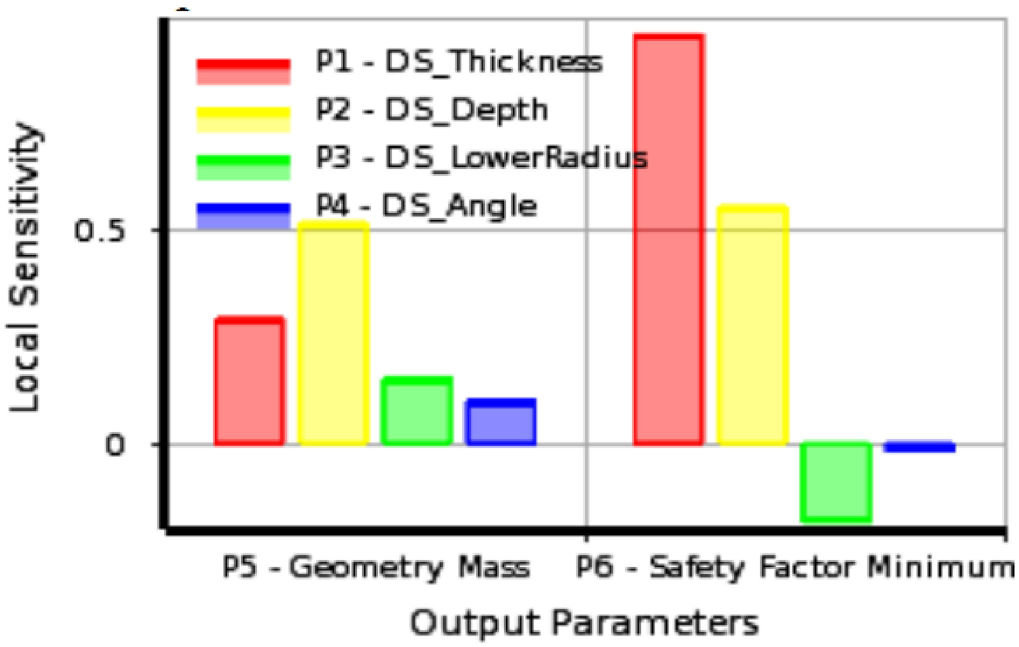


Fig.5.22 Local sensitivity of input parameter vs. output parameter (Rathore, & Chandrakar., 2012)

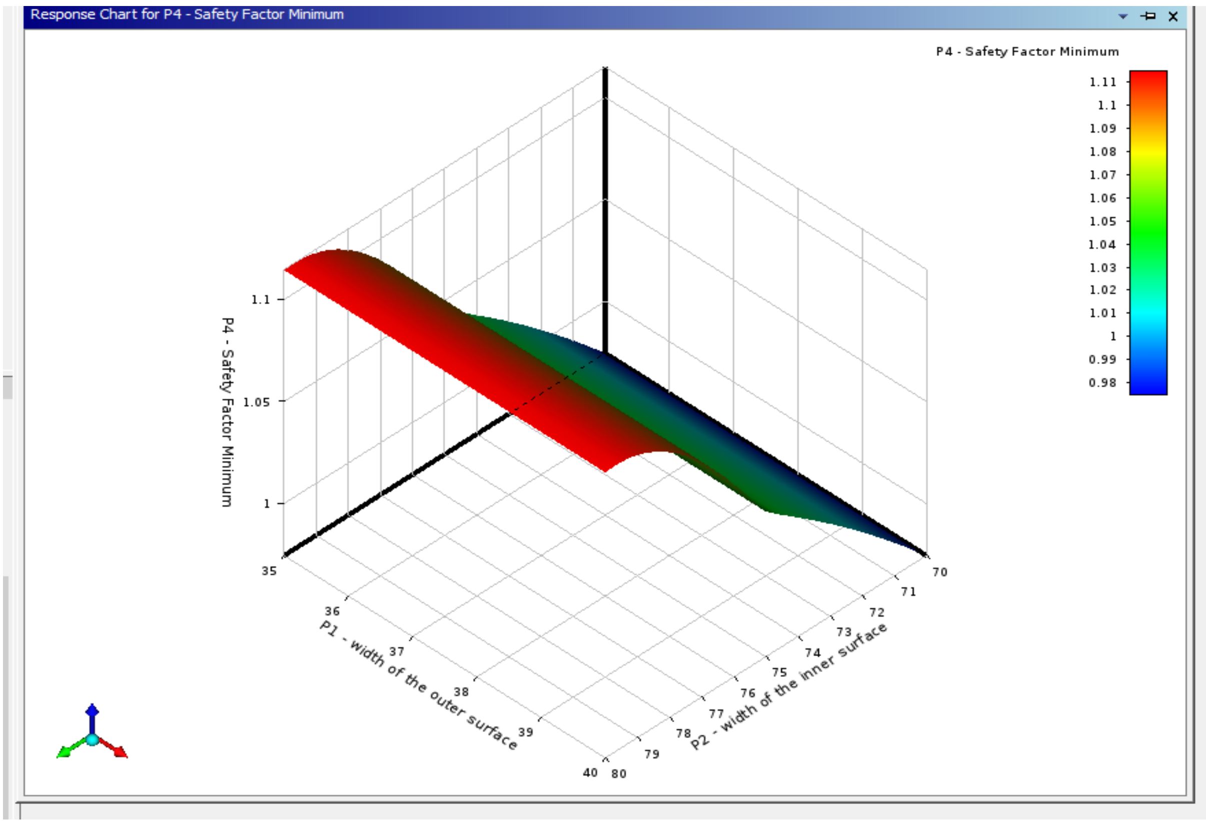


Figure5.23, The 3D response chart p1,p2 input parameters and p4 output parameter.

Spider chart

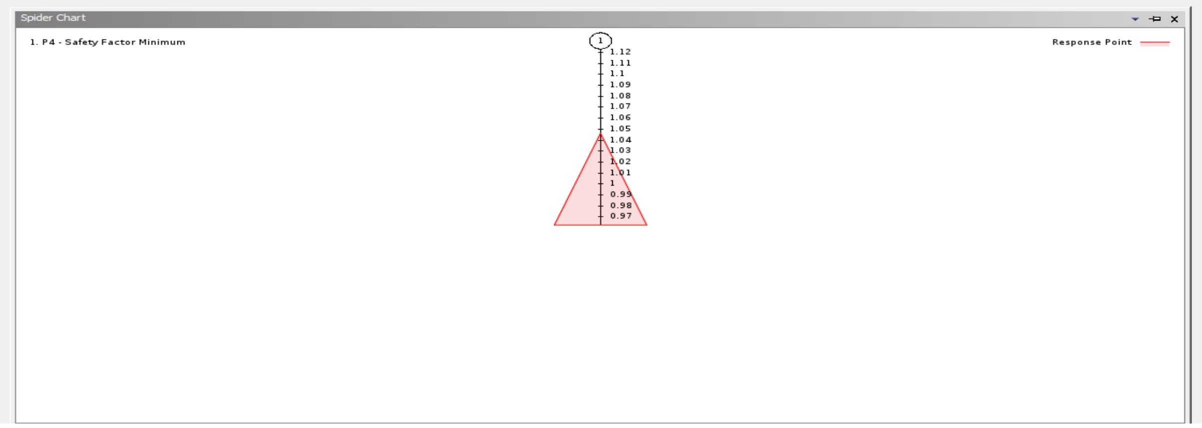


Figure 5.24 The spider chart of the minimum safety factory

The design points and parameters for crane hook equivalent stress maximum shown in figure 5.24, equivalent stress maximum at 5, design point which is the equivalent max. stress is 95 MPa. And also in figure 5.25, there is p3-depth and equivalent maximum stress. At the von-Mises stress is 95 MPa, the p3-depth is 90 mm this parameter shows that the best candidates for the crane hook.

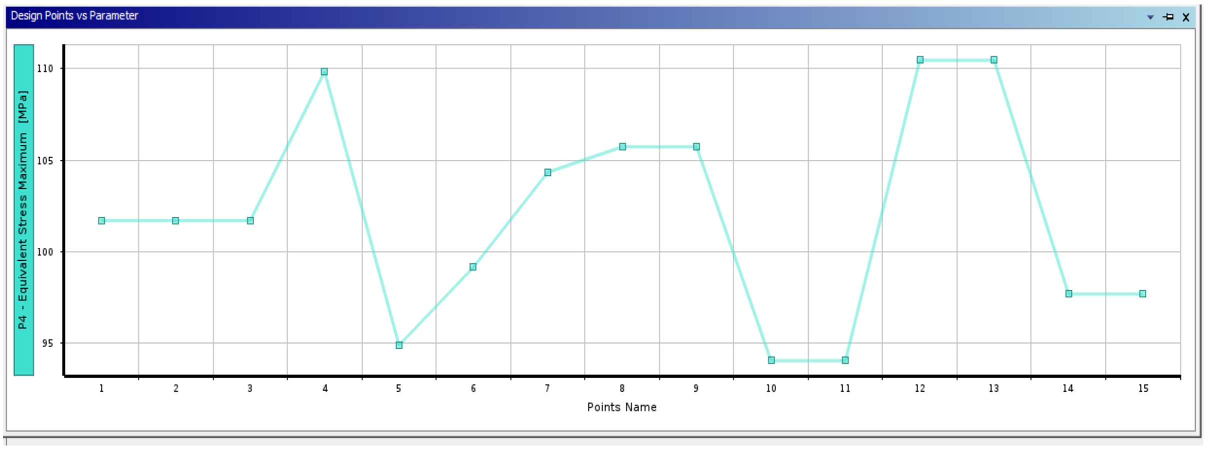


Figure 5.25, The point name vs maximum equivalent stress

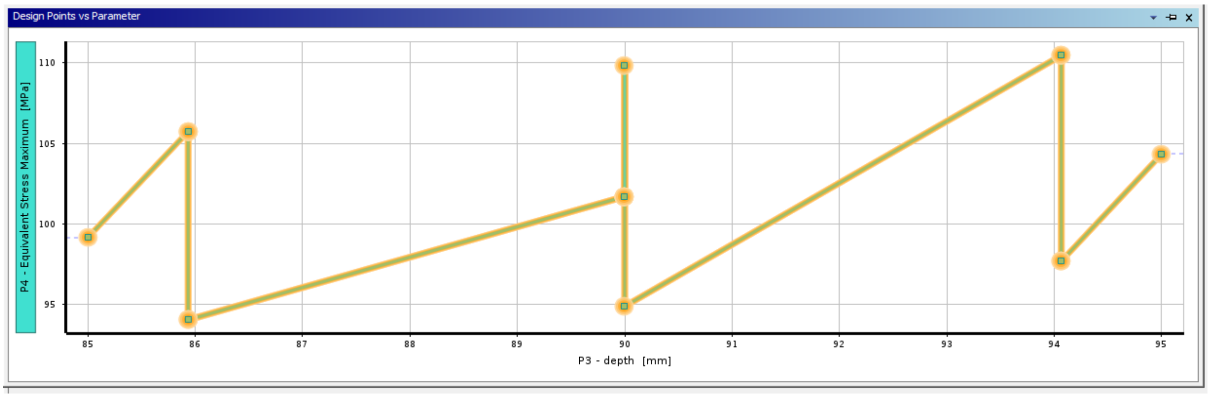


Figure 5.26, The p4-equivalent stress maximum vs p3-depth input parameters.

5.8.5 Response chart 3D Equivalent stress maximum

In parameter design, analysis and simulation are typically used as a means to assess the performance of a part of a system for one given set of CAD dimensions and loads. The curves have a width of the inner surface and stress intensity plotted with one other input variable as the width of the outer surface is the most sensitive variable. so, justifies to Sensitivity is depends on input parameter.

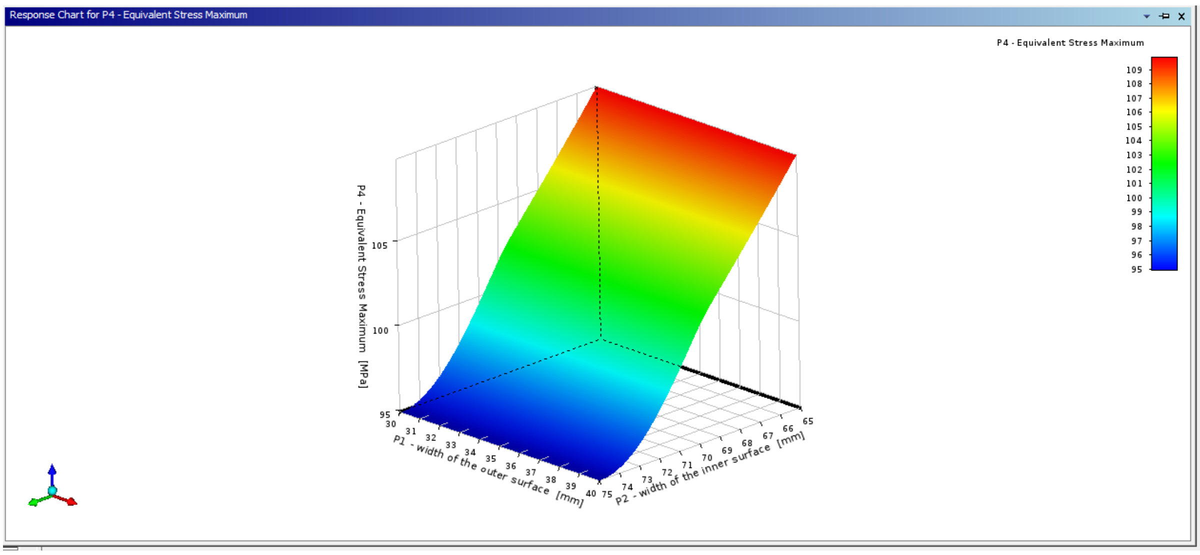


Figure5.27, The 3D response chart for p4-equivalent stress maximum

In this figure 5.27, shows the 3D response chart for equivalent stress maximum analysis the depends on the input parameters with outer and inner surface parameter the equivalent stress maximum 95 MPa to 109 MPa and the from FEM analysis of equivalent stress maximum crane hook is 95MPa this simulation has depends on the outer and inner width surface of the crane hook.

5.8.6 Local sensitivity analysis for von-Mises stress.

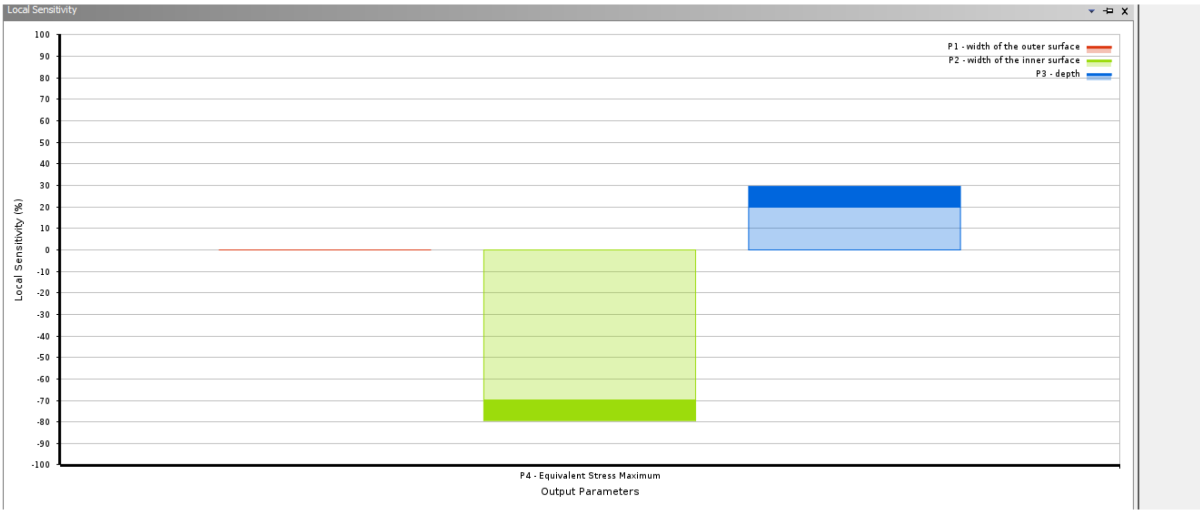


Figure5.28, The local sensitivity p4-equivalent stress maximum

In this local sensitivity p4-equivalent stress maximum or the out parameter p2-width of the inner surface more sensitivity than p3-depth, p1-width of the outer surface is neutral, the p2-width of the inner surface and p3-depth shows the local sensitivity of the equivalent stress maximum. So the equivalent stress maximum is increases the width of the inner surface is decreasing, but the depth of hook increases the equivalent stress maximum is increases and the width of the outer surface has little impact on the equivalent stress maximum.

5.8.7 Local sensitivity curve of the equivalent stress maximum.

Response surface curve for the equivalent stress maximum in figure 5.29 shows the Y-axis represents the equivalent stress and X-axis is a design point, the two black square shows the response points the chart, the high sensitivity, and low sensitivity cross the neutral points. The p3-depth of the hook section parameter, at the response points of 0.5, cross the von-Mises stress of 95 MPa. And at the 0.92 of the p2-width of the surface of the 95 MPa, Von-Mises stress maximum.

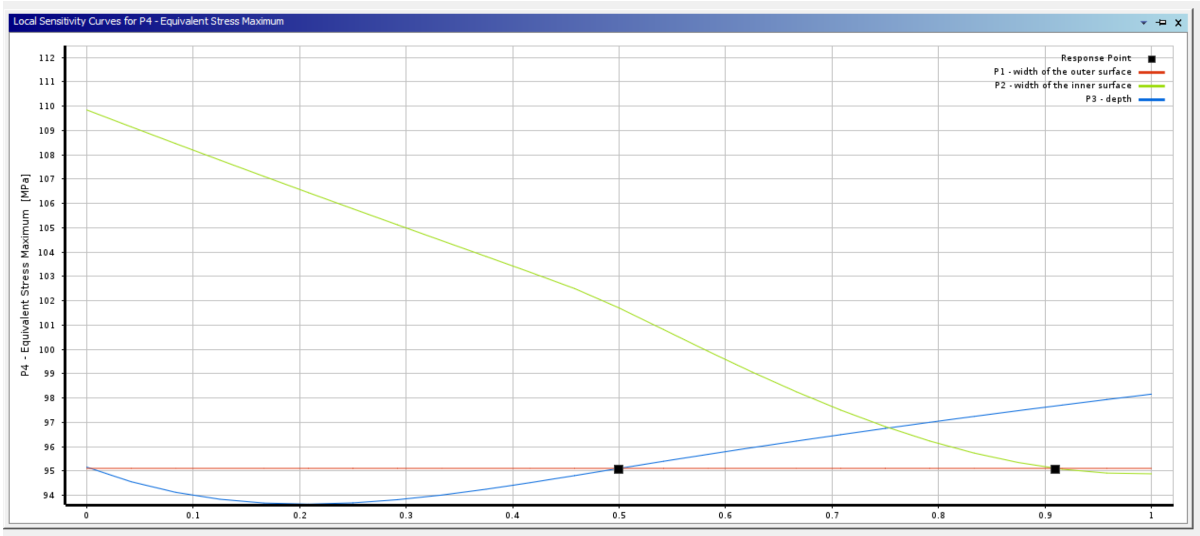


Figure 5.29, The local sensitivity curve for equivalent stress maximum.

Spider chart; the show single graph for the given input parameter of the equivalent stress maximum analysis.

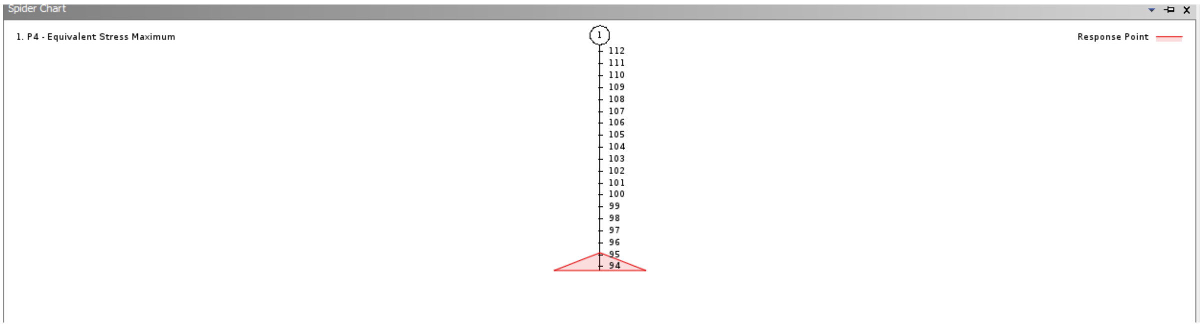


Figure 5.30, The spider chart of the equivalent stress maximum

Response surface optimization analysis Equivalent stress maximum, the same true for the this all bar is positive and the proportional impact of the correlated parameters, then p1-width of the outer surface and p2-width of the inner surface has a small impact on the equivalent stress maximum and when the p3-depth increase the equivalent stress maximum increases.

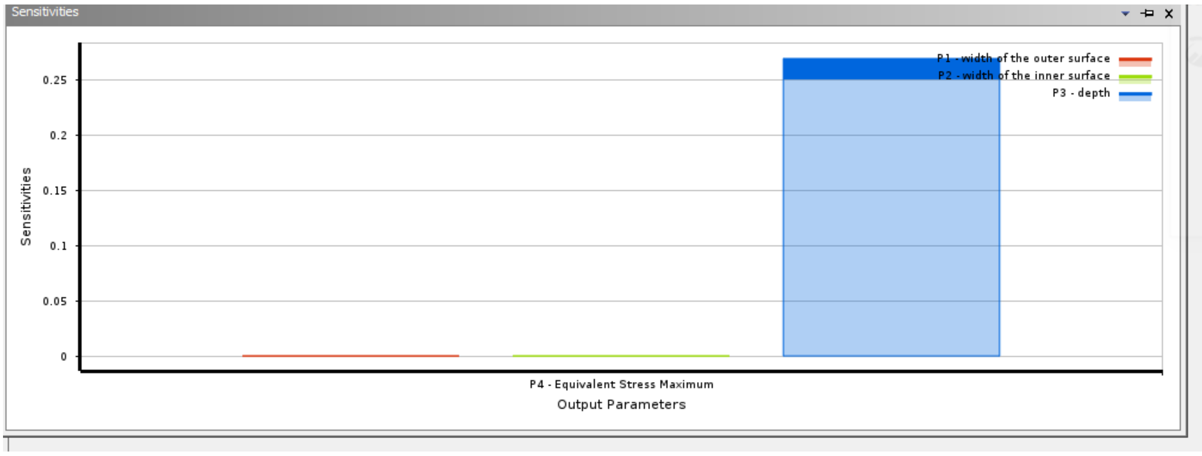


Figure 5.30, The sensitivity vs Equivalent stress maximum

In response surface optimization analysis equivalent maximum analysis in figure 5.30, shows the sensitivity of the input parameters p1-width of the outer surface, p2-width of the inner surface, and p3-depth, the output parameter of the p-4 equivalent stress maximum analysis more sensitivity. The p3- depth is more sensitive than other input parameters and p1- and p2 the neutral sensitivity with the output parameters of the equivalent stress maximum.

In this thesis the topology optimization of the crane hook with DesignXplorer implementation of the ANSYS workbench, during topology optimization the surface(side) of the crane hook removed and the weight of the crane hook is reduced. Then DX implementation shown on the chart and graph above is approve the removed part of the crane hook is under the safe condition of the crane hook without sacrifice the original status.

CHAPTER. 6, CONCLUSION AND RECOMMENDATION OF FUTURE WORK.

6.1 Conclusion

From the case study, it can be concluded that topology optimization is a powerful design concept to reduce the weight of structural products. Simulation of the hook is done using the topological approach. Model is created, then meshing is done, FEA analysis (ANSYS 19.2) carried out. The reduction of weight saves a huge amount of material without losing the design strength and processing energy amount of money. It also shows that the capability of topology optimization can be fully utilized and from the case study results, which is 6.685% weight reduction, it can also be concluded that topology optimized design can reduce a huge portion of the mass thus results in a lightweight design.

Topology optimization has been carried out on a machine member to optimize its shape and thus reduce its weight. ANSYS 19.2 WB, solver which uses the density approaches has been used for this purpose. To demonstrate the usefulness of the topology optimization approaches, a crane hook has been used to carry out the study.

In the improve the fatigue life, the hook structure, particularly the structure of the maximum stress zone were optimized to improve the fatigue life, so then the safety using time was extended.

During the demonstration of Designxplorer(DX) with ANSYS 19.2. The designxplorer in this study depending on the response surface of the crane hook with a given input parameter and output. So the input parameter used in this demonstration is the width of the outer surface(side), the width of the inner surface, and depth of the crane hook, the output is the safety factor minimum and equivalent maximum stress.

Designxplorer implementation in the topology optimization of crane hook uses response surface optimization to efficiently explore the solution space.

- ✓ Explore and understand the performance at other or operating conditions crane hook.
- ✓ Find the conditions which give the best performance of crane hook.
- ✓ Determine the key parameters influencing the design crane hook.

- ✓ The response surface techniques used in Designxplorer allow working on a set more input parameter at a time.
- ✓ Quantify the Quality of Products with ANSYS DesignXplorer

ANSYS DesignXplorer describes the relationship between the design variables and the performance of the product by using Design of Experiments (DOE) combined with Response Surfaces. DOE and Response Surfaces provide all the information required to take advantage of Simulation-Driven Product Development. When performance variations due to design variables are known, it is easy to understand and identify all changes required to meet the product requirements. Once the Response Surfaces are created, information about curves, surfaces, sensitivities, and other variables can be shared in terms that are easy to understand and can be used any time in the crane hook optimization, cycle without requiring additional simulations to test a new configuration.

6.2 Recommendation and its future work.

In this thesis work conceptual modeling and structural optimization framework of crane hook is studied with topology optimization and designxplorer simulation using the finite element method (FEM). Using FEM for stress analysis, weight optimization, fatigue life, and designxplorer simulation estimation have many advantages.

Generally, the advantages are requiring a short period for analysis and less cost as compared to the experimental method. The literature review allows related to this title are very few articles have been published so need more extensive investigation on cross-section depends on the optimization of crane hook using geometrical modification. However, this paper can be extended to conceptual modeling and structural optimization by topology optimization framework and also can study about designxplorer simulation of the crane hook.

Particularly for a specific application, it is better if the following areas are carried out for its future work:

- ➢ Analysis of Shape optimization and designxplorer simulation crane hook.
- ➢ Design for deflection degree analysis of the crane hook a slight influence on the strength and shape variables of the hook.
- ➢ Detail analysis of the opening diameter of the hook has a great influence on the strength and shape variables of the hook.

- optimizing the design of the hook through the three parameters of the deflection angle of the hook, the opening diameter of the hook, and the position of the thickest section of the hook.
- Detail analysis of response surface modeling and shape optimization crane hook.

REFERENCE

Ahmed, F, stewart R., & Antuela.A. (2014). using the structred analysis and design technique (SADT) in simulation conceptual modeling. *Proceedings of the 2014 Winter Simulation Conference*, *3*(14), 1038–1049.

Ankit. (2017). Fatigue failure of a crane hook. *Internarional Journal of Recent Technology Scienceand Management.*, *2*(2), 1–7.

Babu.A, Rao.S, S. . (2015). Design optimization and contact analysis on dual joint. *International Journal of Science, Engineering and Technology Research (IJSETR).*, *4*(7), 2485–2493.

Bathe. (2016). *Finite Element Procedures second edition*.

Benjamin., Jan., & Florian. (2018). Geometric element parameterization and parametric model order reduction in finite element based shape optimization. *Springer Computational Mechanics*, *9*(7), 1–16. https://doi.org/10.1007/s00466-018-1626-1

Bhise., & Deshpande. (2018). *Optimization of Design Parameters for Crane Hook Using Finite Element Analysis*. *2*(8), 53–59.

Bock., Dandashi., & Friedenthal. (2017). Conceptual Modeling. In *Research Challenges in Modeling and Simulation for Engineering Complex Systems, Simulation Foundations,* (pp. 23–44). https://doi.org/10.1007/978-3-319-58544-4

Brackett., Ashcroft., & Hague. (2011). Topology optimization for additive manufacturing. *Proceedings of the 6th International Conference on Additive Manufacturing, Loughborough, July 2011*, 348–362.

Callister.w. (2007). *Materials Science and EngineeringMaterials Science 7th edition*.

Chunkawan, & Subramaniyam. (2017). Static Structural Analysis Of Crane Hook. *International Research Journal of Engineering and Technology (IRJET)*, *04*(07), 2265–2274.

Coatanéa.E. (2016). A Conceptual Modeling and Simulation Framework for System Design. *IEEE CS and the AIP*, *16*(August 2016), 42–52.

Desai, N., & Zeytinoglu, N. (2016). *Design and Optimization of the Geometric Properties of a Crane Hook*. (August), 391–397.

Fiedler, K. (2014). *An integrated Approach to shape and topology optimisation of mechanical structures* (MMath TU Berlin, Germany). https://doi.org/10.2495/OP120131

Gopichand., & Lakshmi. (2013). Optimization of design parameters for crane hook using taguchi method. *International Journal of Innovative Research in Science, Engineering* *and Technology*, *2*(12), 7780–7784.

Guo. (2015). Doing Topology Optimization Explicitly and Geometrically: a New Moving Morphable Components Based Framework. *Journal of Applied Mechanics*, *81*(9), 31–32. https://doi.org/10.1142/9781783266852_0016

Guo., Zhao., & Yu. (2005). *Simultaneous Shape and Topology Optimization with Implicit Topology Description Functions*.

Hagara, M., & Pástor, M. (2017). Full-field Stress Analysis of a Crane Hook Model Performed by Finite Element Analysis and Digital Image Correlation Method. *American Journal of Mechanical Engineering*, *5*(6), 293–297. https://doi.org/10.12691/ajme-5-6-12

Hamid. (2014). *Dynamic Analysis of the Tower Crane*.

Jani, T. P. (2015). Weight optimization of Crane hook having 8tons load capacity by Modifying cross section and comparison with various basic cross sections. *International Journal of Innovative Research in Advanced Engineering (IJIRAE)*, *2*(4), 160–163.

Jiaqiin, Krishnan., & Igor. (2007). Shape optimization with topological changes and parametric control. *Internation Journal for Numerical Methods in Engineering*, *3*(December 2006), 313–346. https://doi.org/10.1002/nme

Joseph. ArutPranesh. (2015). Structural analysis of crane hook. *(IJETCSE)*, *12*(2), 108–111.

Jovanovi, J. D. (2011). Finite element evaluation and optimizationof geometry with DOE. *International Journal for Quality Research*, *4*(4), 39–45.

Kankotiya, H. P., & Modh, P. N. (2016). *Design , Analysis and Optimization of Crane Hook*. *4*(2), 773–781.

Karagoz. (2008). *A framework for developing conceptual models of the mission space for simulation systems.*

Kim. & Nam-ho. (2005). *book, Structural Sensitivity Analysis and Optimization 1 linear system, Mechanical Engineering Series*. USA.

Kumar, B. J., & Prasad, K. (2014). Design and Stress Analysis of Various Cross-Section of Hook. *International Journal of Innovations in Engineering and Technology (IJIET)*, *4*(4), 90–99.

Lanjekar., & Patil. (2016). Weight Optimization of Laminated Hook by Topological Approach. *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE*, *13*(5), 7–20. https://doi.org/10.9790/1684-1305010720

Logan. (2007). *A First Course in the Finite Element Method Fourth Edition*.

Manee-ngam., & Fournier. (2017). Hook Design loading by the optimization method with weight factors rating method. *International Conference on Alternative Energy in Developing Countries and Emerging Economies AEDCEE, 25-26 May 2017, Bangkok, Thailand , 138*, 337–342. https://doi.org/10.1016/j.egypro.2017.10.132

Mehendale, W. (2016). Design and analysis of EOT crane hook for various cross sections. *International Journal of Current Engineering and Scientific Research (IJCESR).*, *3*(12), 53–58.

Metin Ozen. (2014). Meshing workshop. *Thursday, November 13th, 2014 Metin Ozen, Ph.D., ASME Fellow*, 1–116.

Mukhopadhyay & Souvik. (2018). Failure analysis of a 40 ton crane hook at a Hot Strip Mill. *MATEC Web of Conferences 165*, *10006*, 1–10.

Nagar, G. (2018). Design and Optimization of Trapezium in Curved Beam. *International Journal of Science and Research (IJSR)*, *7*(11), 263–267. https://doi.org/10.21275/ART20192640

Nisbett. (2006). *Shigley's Mechanical Engineering Design, Eighth Edition Front*.

Norato, J. (2010). *A Method for Fatigue-Based Topology Optimization*. *41*, 61820.

Patel. (2015). Design and Analysis of 50 Tonne Crane Hook for Optimization. *International Journal for Scientific Research and Development(IJSRD)*, *3*(08), 581–584.

Patil.&Thakur. (2016). Study and Analysis of Crane Hook in Loading Area. *International Journal of Advanced Research in Education & Technology (IJARET)*, *3*(1), 88–89.

Pavlovic. (2018). Analysis and Optimization of T-cross section of Crane Hook Considered as a Curved Beam. *IMK-14 – Research and Development in Heavy Machinery*, *2*(5), 53–60.

Picelli., & Townsend. (2018). ScienceDirect Stress-based shape and topology optimization with the level set method. *Comput. Methods Appl. Mech. Engrg.*, *329*, 1–23. https://doi.org/10.1016/j.cma.2017.09.001

Prashant. (2015). Design, Structural Analysis and Optimization of Crane Hook. *International Engineering Research Journal (IERJ)*, *2*(2), 4298–4302.

Rathore, &, & Chandrakar. (2012). An Approach to optimize ANN Meta model with Multi Objective Genetic Algorithm for multi-disciplinary shape optimization. *International Journal of Soft Computing and Engineering (IJSCE)*, *2*(1), 200–207.

Reinders., Akkermans., & Balder. (1991). A Conceptual Modelling Framework for Knowledge-level Reflection. *AICOM*, *4*(2), 74–87.

Robinson, S. (2014). *Conceptual modelling for simulation Part II : a framework for conceptual modelling*. (January). https://doi.org/10.1057/palgrave.jors.2602369

Roca, R., Pace, D., & Tolk, A. (2015). Paradigms for Conceptual Modeling Paradigms for Conceptual Modeling. *Society for Modeling & Simulation International (SCS)*, 1–9.

Schwarz, S., Maute, K., & Ramm, E. (2001). Topology and shape optimization for elastoplastic structural response. *Comput. Methods Appl. Mech. Engrg*, *190*, 2135–2155.

Shaban, & Mohamed. (2013). Determination of Stress Distribution of stress distribution in Crane Hook by Caustic. *International Journal of Innovative Research in Science, Engineering and Technology*, *2*(5), 1834–1840.

Sharma. (2013). *Finite element analysis of a crane hook using cae tools* (Vol. 2). https://doi.org/13611

Singh., & Rohilla. (2015). Optimiztion and Fatigue analysis of crane hook using finite element method. *International Journal of Recent Advances in Mechanical Engineering (IJMECH)*, *4*(4), 31–43.

Sloboda., & Honarmandi. (2014). Generalized Elasticity Method for curved beam stress analysis,analytical and numerical comparison for a lifting hook. *Mechanics Based Design of Structures and Machines: An International Journal.*, *35*(3), 319–332. https://doi.org/10.1080/15397730701473820

Thejomurthy., & Ramakrishn. (2018). Topology Optimization and Analysis of Crane Hook Model. *Journal of Engineering Research and Application*, 60–64.

Tigabey, S. (2018). Optimization and Fatigue Analysis of Crane Hook Using Finite Element Method.

Torres., Gallardo., & Domínguez. (2010). Brittle fracture of a crane hook. *Engineering Failure Analysis*, *17*(1), 38–47. https://doi.org/10.1016/j.engfailanal.2008.11.011

Tyflopoulos., Flem., & Steinert. (2018). State of the art of generative design and topology optimization and potential research needs. *NordDesign 2018*, 1–15.

Upendar, S. (2018). Design and analysis of a crane hook 1,2. *International Journal of Current Engineering and Scientific Research (IJCESR)*, *5*(4), 10–15.

Wang.&Ji. (2019). Shape Optimization of Hook for Marine Crane Shape Optimization of Hook for Marine Crane. *IOP Conf. Series:Journal of Physics: Conf. Series*, *7*(11), 1–7. https://doi.org/10.1088/1742-6596/1187/3/032047

Wiley, J. (2001). *Fundamentals of Materials Science and Engineering An Interactive*.

Zade, R. (2017). Finite element analysis and fatigue analysis of crane hook with different materials. *International Research Journal of Engineering and Technology (IRJET)*, *04*(01), 1429–1436.

APPENDIX A

MATERIAL PROPERTY

Structural Steel > Strain-Life Parameters

Strength Coefficient MPa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient MPa	Cyclic Strain Hardening Exponent
920	-0.106	0.213	-0.47	1000	0.2

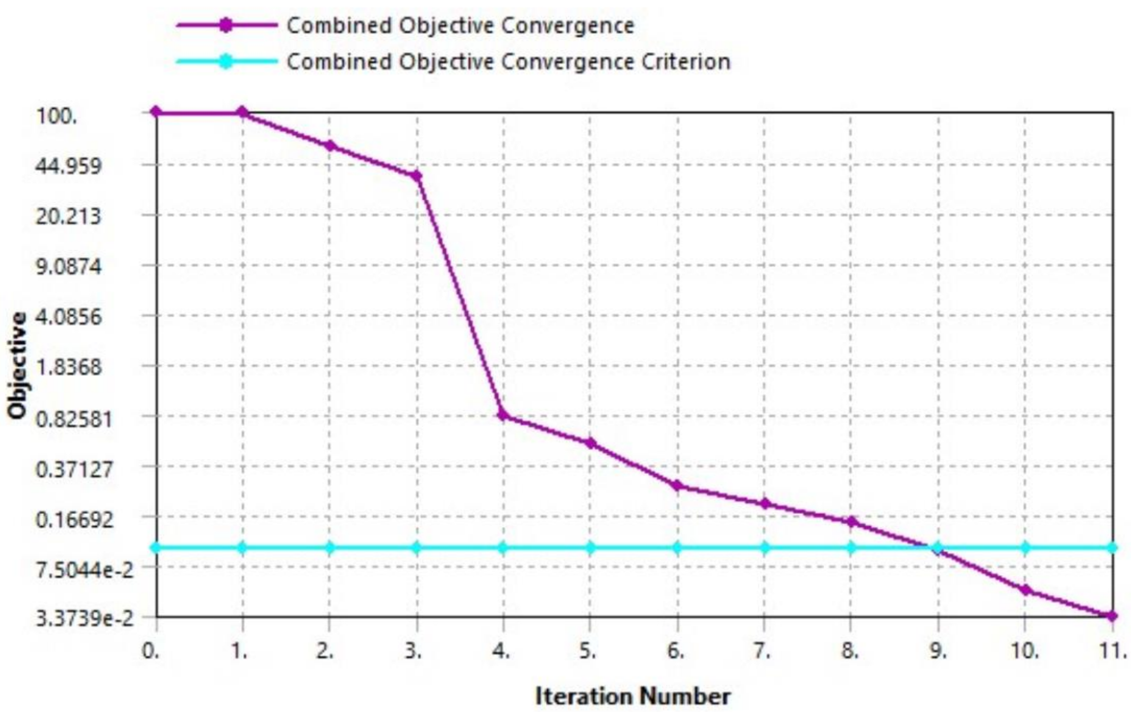
Structural Steel > Isotropic Elasticity

young's Modulus MPa	Poisson's Ratio	Bulk Modulus MPa	Shear Modulus MPa
2.e+005	0.3	1.6667e+005	76923

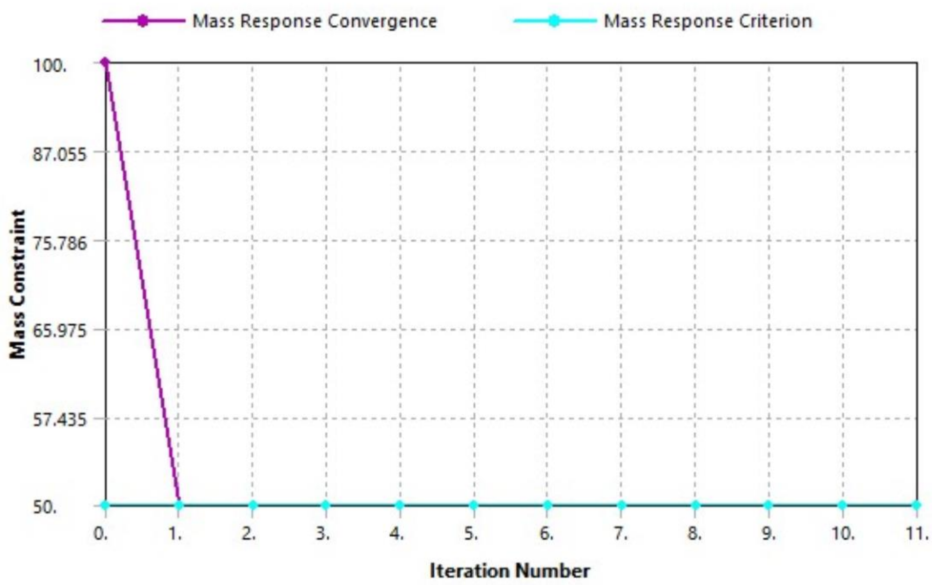
APPINDEX B,

TOPOLOGY OPTIMIZATION PROPERTIES

Model (A4, B4) > Topology Optimization (B5) > Solution (B6) > Solution Information



Model (A4, B4) > Topology Optimization (B5) > Solution (B6) > Solution Information

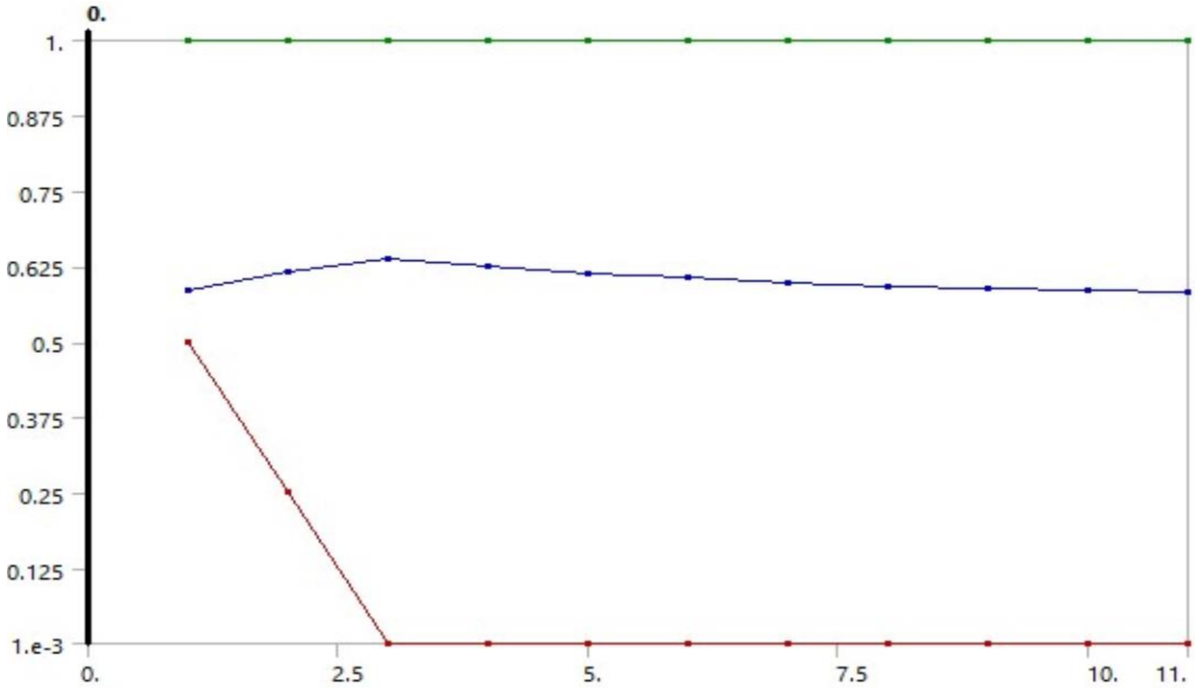


Model (A4, B4) > Topology Optimization (B5) > Solution (B6) > Results

Response Type	Goal	Formulation	Environment Name	Weight	Multiple Sets	Start Step	Step End	Step	Mode Start	Mode End	Mode
Compliance	Minimize	Controlled Program	Structural Static	1	Enabled	1	1	1	N/A	N/A	N/A

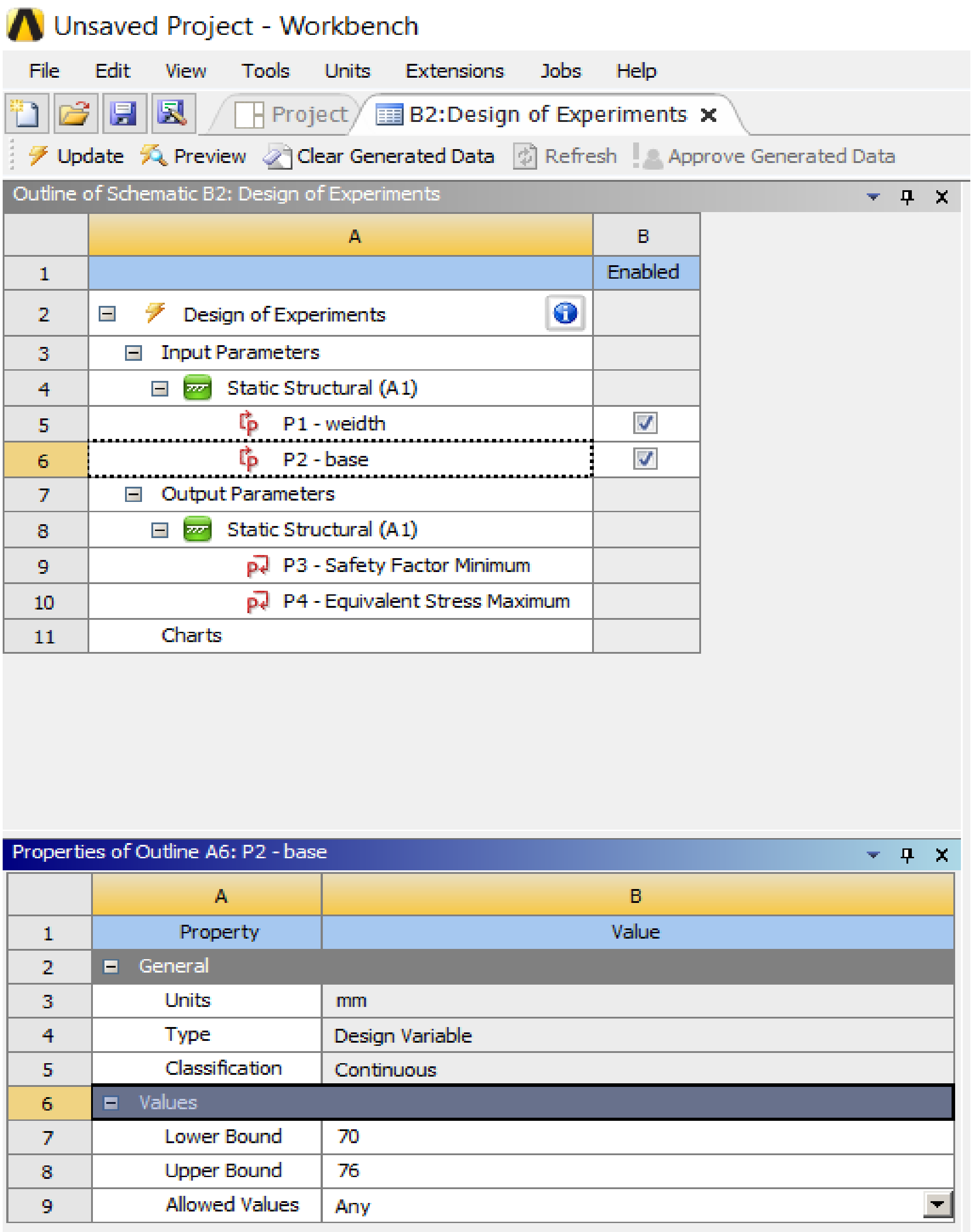
Object Name	Topology Density
State	Solved
Scoping Method	Optimization Region
Iteration	Last
Retained Threshold	0.5
Calculate Time History	Yes
Suppressed	No
Results	Yes
Minimum	1.e-003
Iteration	Last
Retained Threshold	0.5
Maximum	1.
Average	0.58418
Original Volume	1.3406e+007 mm^3
Final Volume	1.251e+007 mm^3
Percent Volume of Original	15.75kg
Final Mass	13.678kg
Percent Mass of Original	93.315
Show Optimized Region	Retained Region
Information	
Iteration Number	11

Model (A4, B4) > Topology Optimization (B5) > Solution (B6) > Topology Density

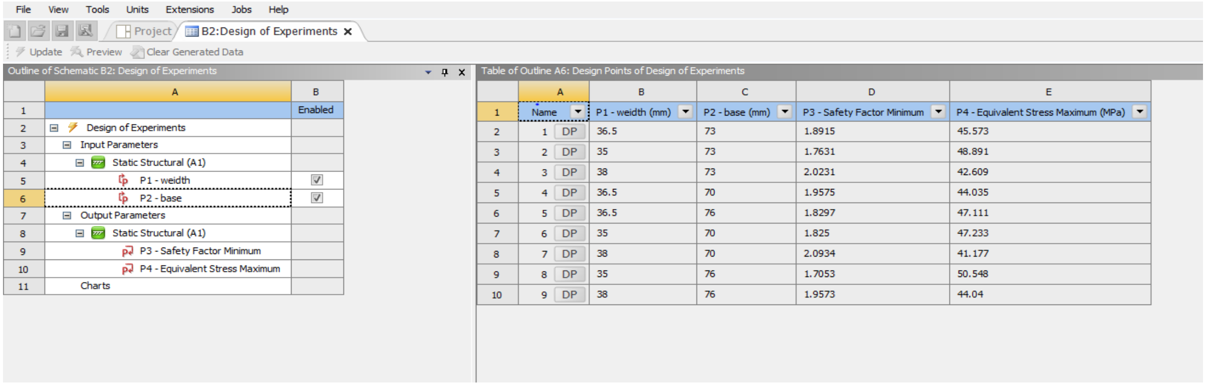


DESIGNXPLORER(DX)

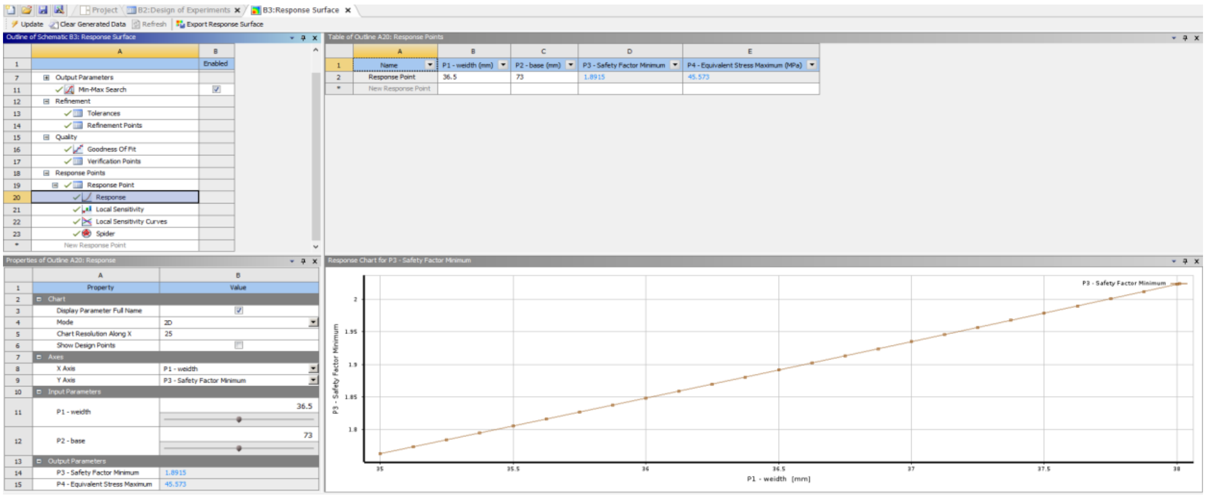
1. *Response surface analysis with designxplorer.*

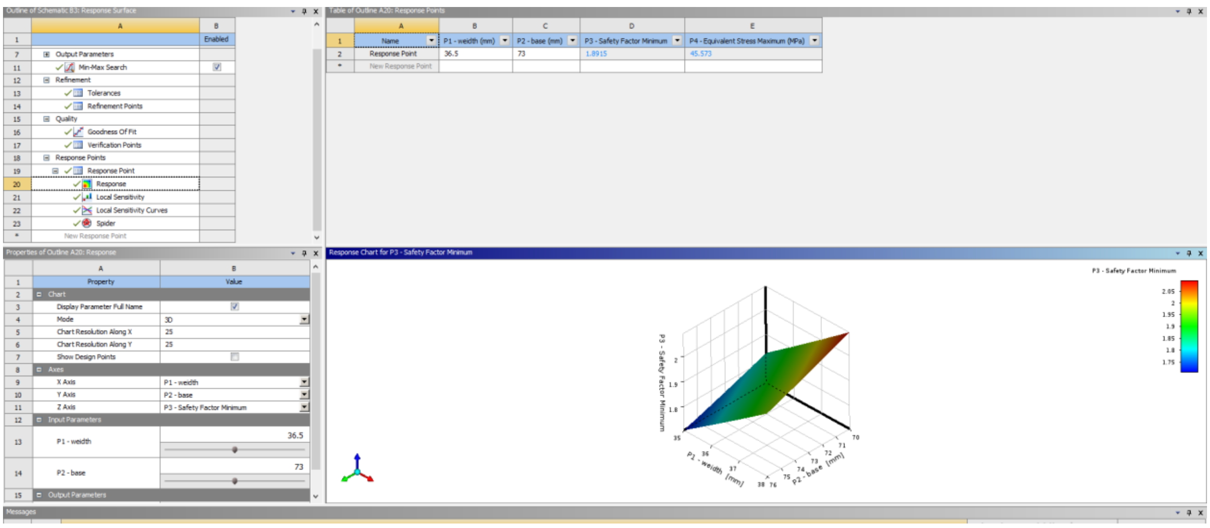


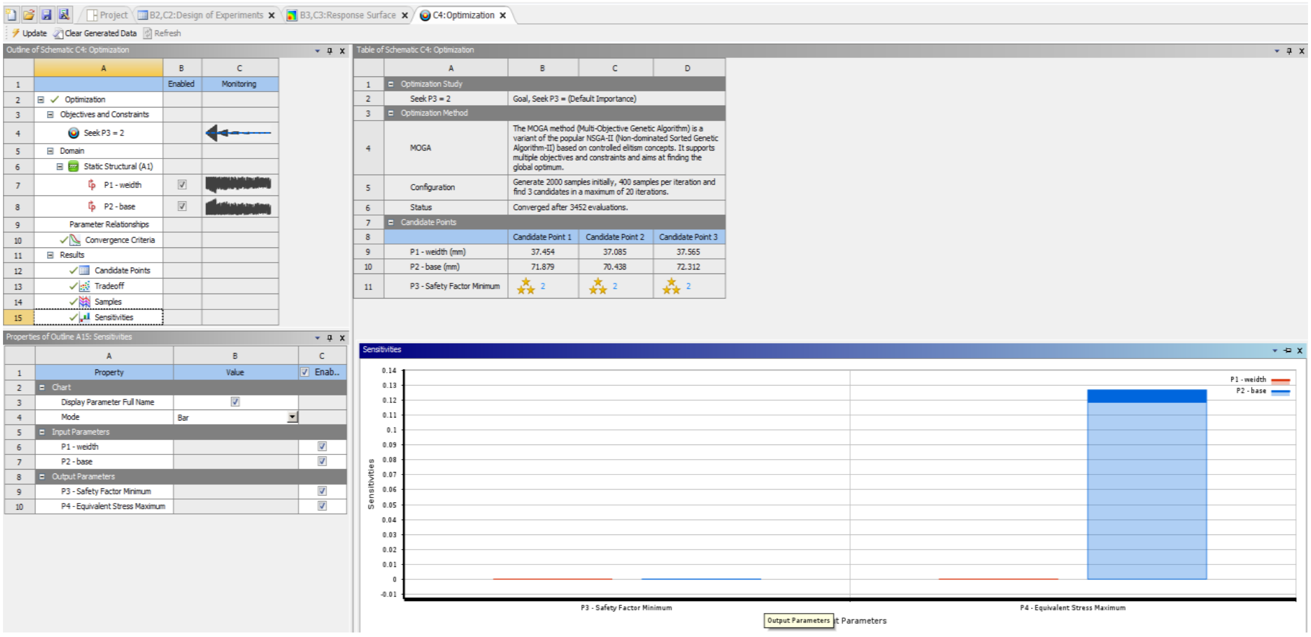
2. *Outline of schematic B2 design of experiment*

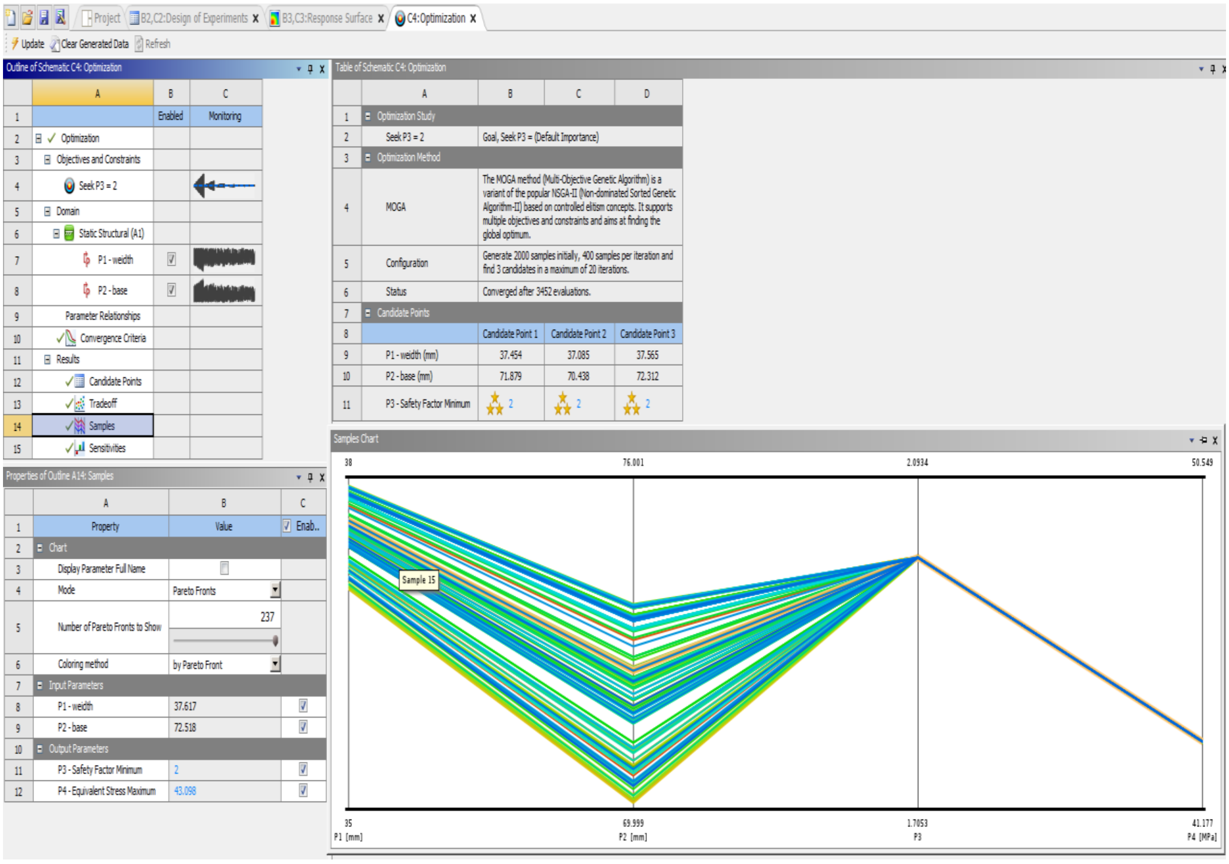


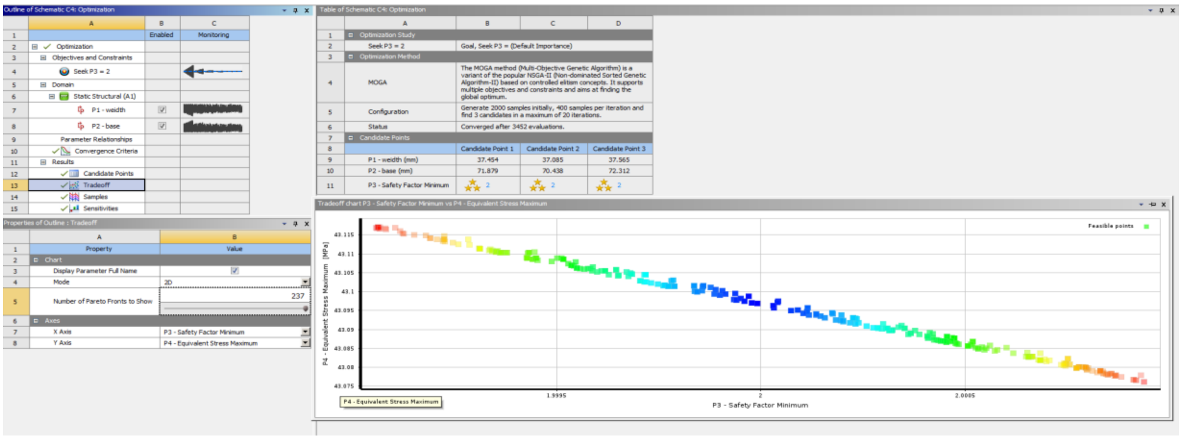
3. *outline of the schematic response surface*



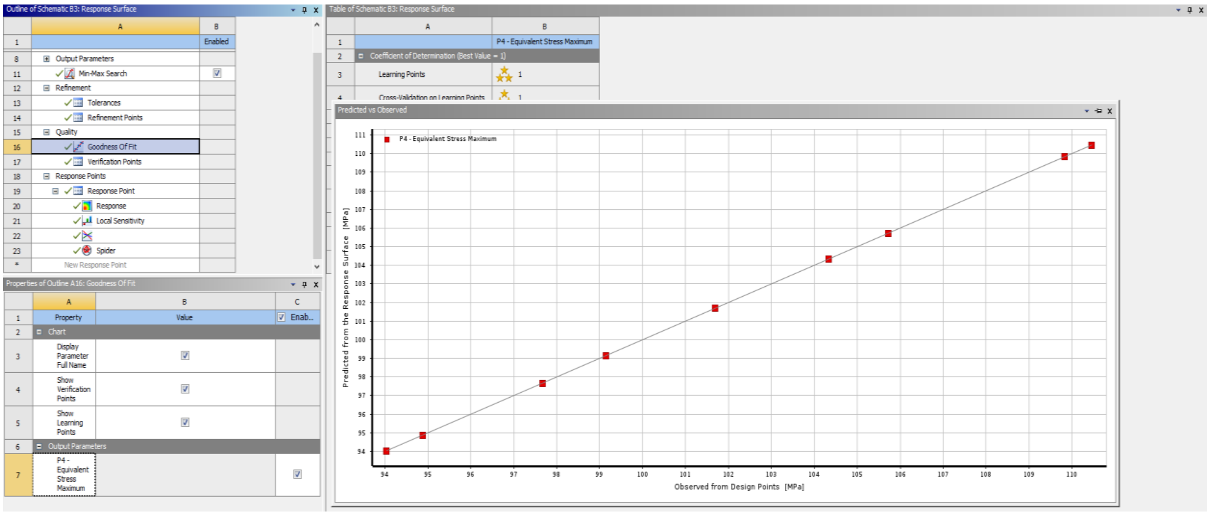



Project
B2,C2:Design of Experiments
B3,C3:Response Surface
C4:Optimization
Update
Clear Generated Data
Refresh
Outline of Schematic C4: Optimization
Enabled
Monitoring
Optimization
Objectives and Constraints
Seek P3 = 2
Domain
Static Structural (A1)
P1 - weidth
P2 - base
Parameter Relationships
Convergence Criteria
Results
Candidate Points
Tradeoff
Samples
Sensitivities
Table of Schematic C4: Optimization
Optimization Study
Seek P3 = 2
Goal, Seek P3 = 2 (Default Importance)
Optimization Method
MOGA
The MOGA method (Multi-Objective Genetic Algorithm) is a variant of the popular NSGA-II (Non-dominated Sorted Genetic Algorithm-II) based on controlled elitism concepts. It supports multiple objectives and constraints and aims at finding the global optimum.
Configuration
Generate 2000 samples initially, 400 samples per iteration and find 3 candidates in a maximum of 20 iterations.
Status
Converged after 3452 evaluations.
Candidate Points
Candidate Point 1
Candidate Point 2
Candidate Point 3
P1 - weidth (mm) 37.454 37.085 37.565
P2 - base (mm) 71.879 70.438 72.312
P3 - Safety Factor Minimum 2 2 2
Properties of Outline A15: Sensitivities
Property
Value
Chart
Display Parameter Full Name
Mode
Bar
Input Parameters
P1 - weidth
P2 - base
Output Parameters
P3 - Safety Factor Minimum
P4 - Equivalent Stress Maximum
Sensitivities
P1 - weidth
P2 - base
P3 - Safety Factor Minimum
P4 - Equivalent Stress Maximum
Output Parameters

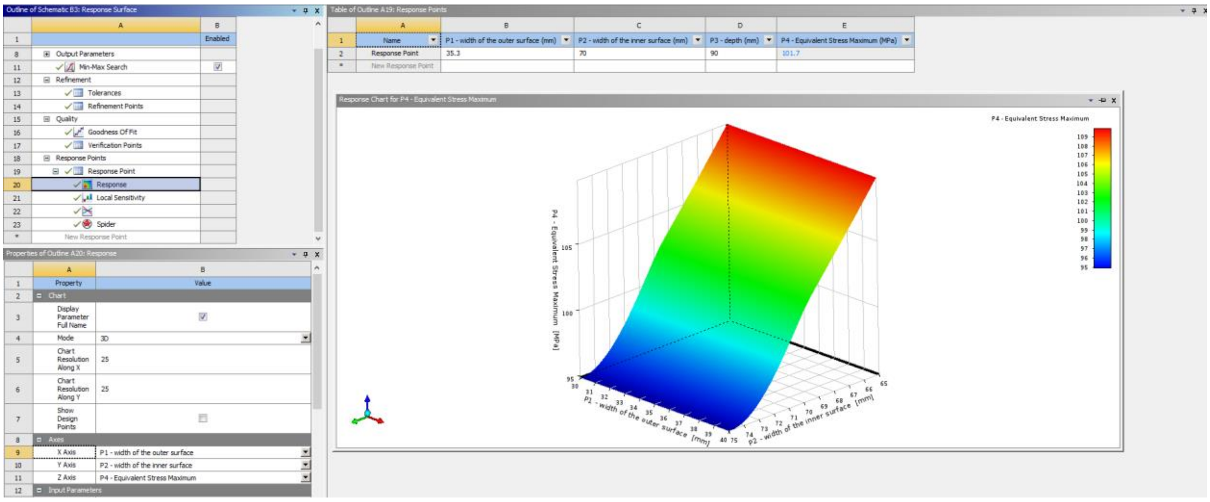

Project
B2,C2:Design of Experiments
B3,C3:Response Surface
C4:Optimization
Update
Clear Generated Data
Refresh
Outline of Schematic C4: Optimization
Table of Schematic C4: Optimization
Properties of Outline A14: Samples
Property
Value
Chart
Display Parameter Full Name
Mode
Pareto Fronts
Number of Pareto Fronts to Show
237
Coloring method
by Pareto Front
Input Parameters
P1 - weidth 37.617
P2 - base 72.518
Output Parameters
P3 - Safety Factor Minimum 2
P4 - Equivalent Stress Maximum 43.098
Samples Chart
38
76.001
2.0934
50.549
Sample 15
35
P1 [mm]
69.999
P2 [mm]
1.7053
P3
41.177
P4 [MPa]



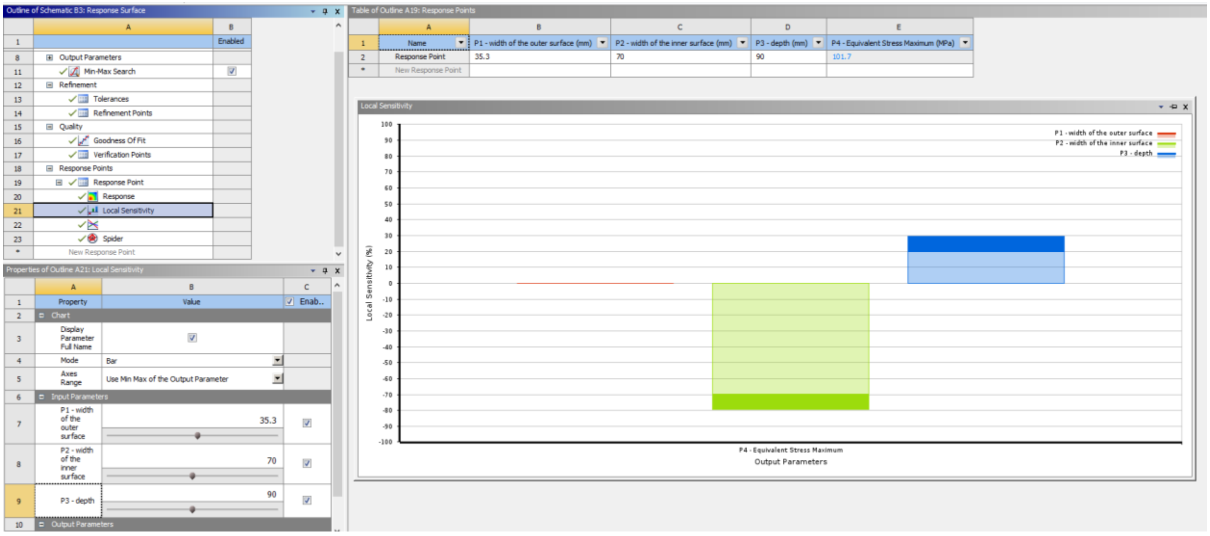
4. *Goodness of Fit*



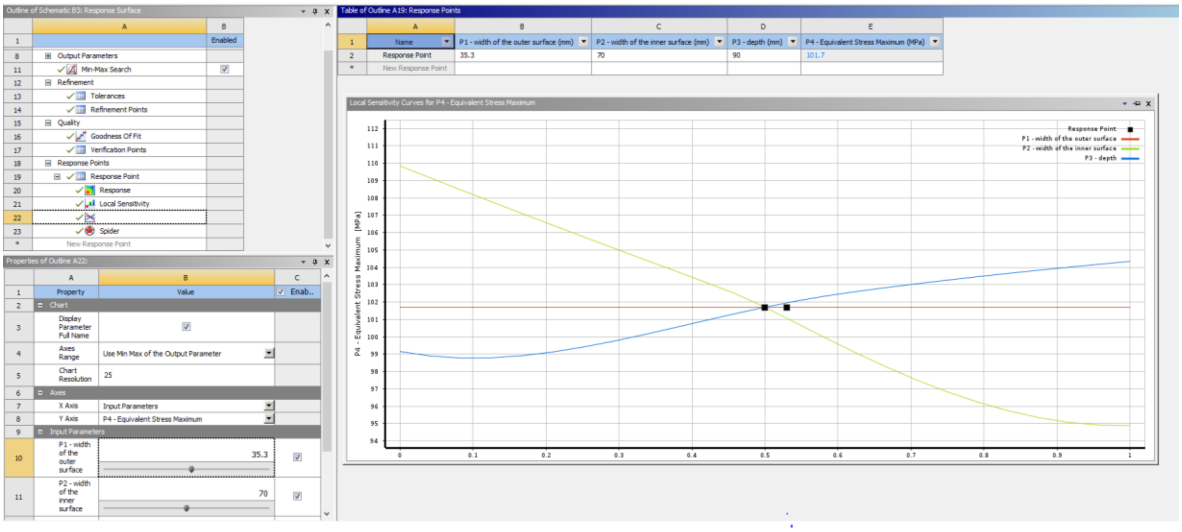
5. Response point with equivalent max. stress



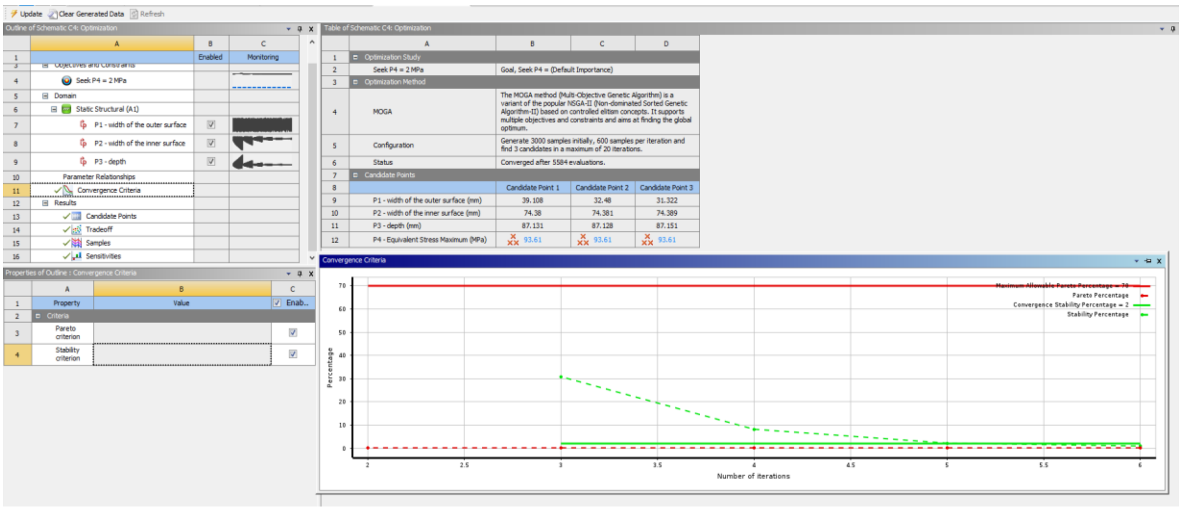
5. *Local sensitivity of von-Mises stress.*

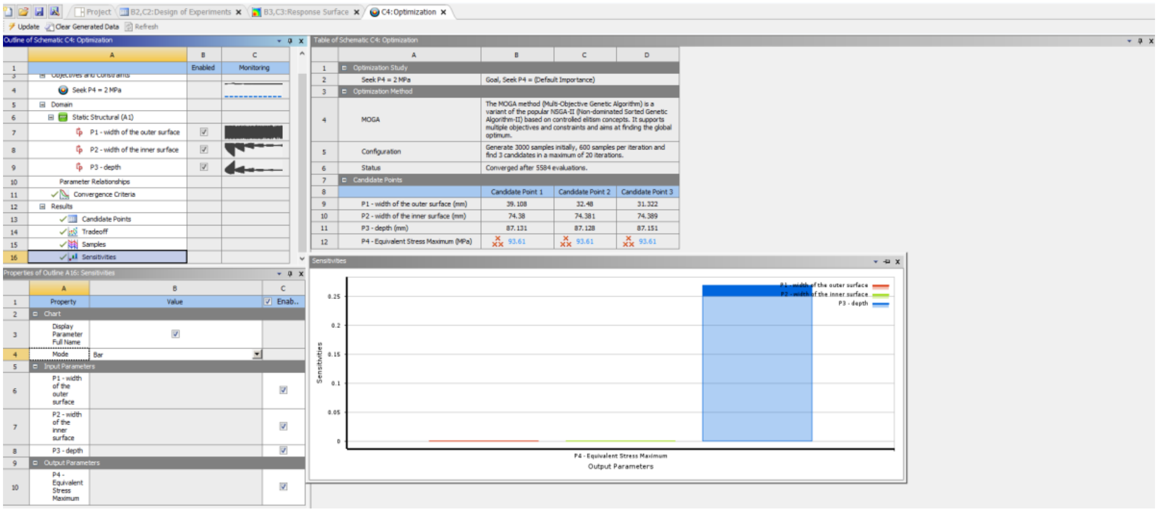


6. *Local sensitivity curve for von-Mises stress max.*

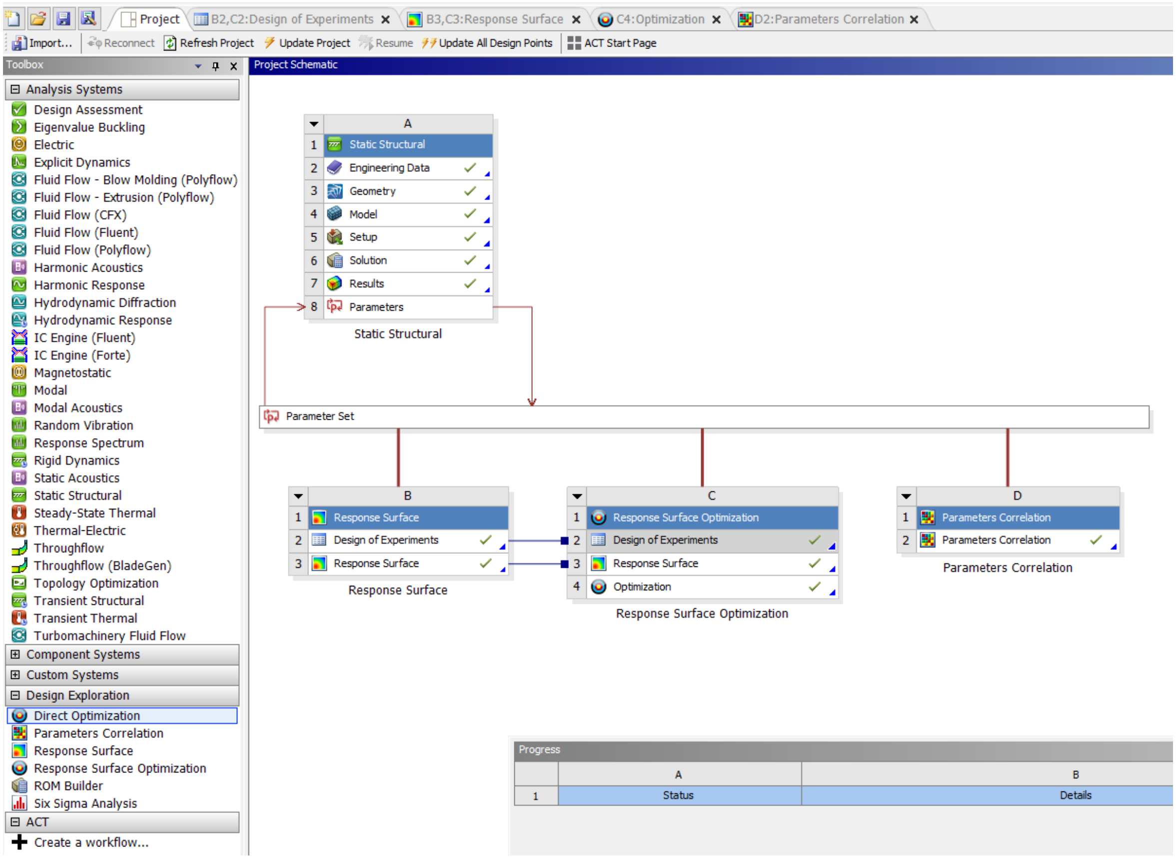


7. *Response surface optimization.*

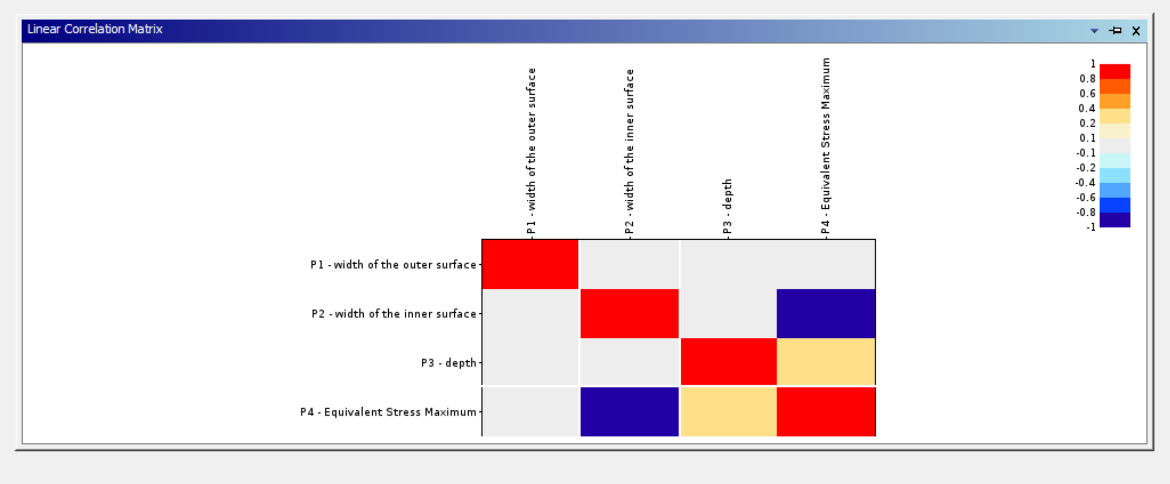


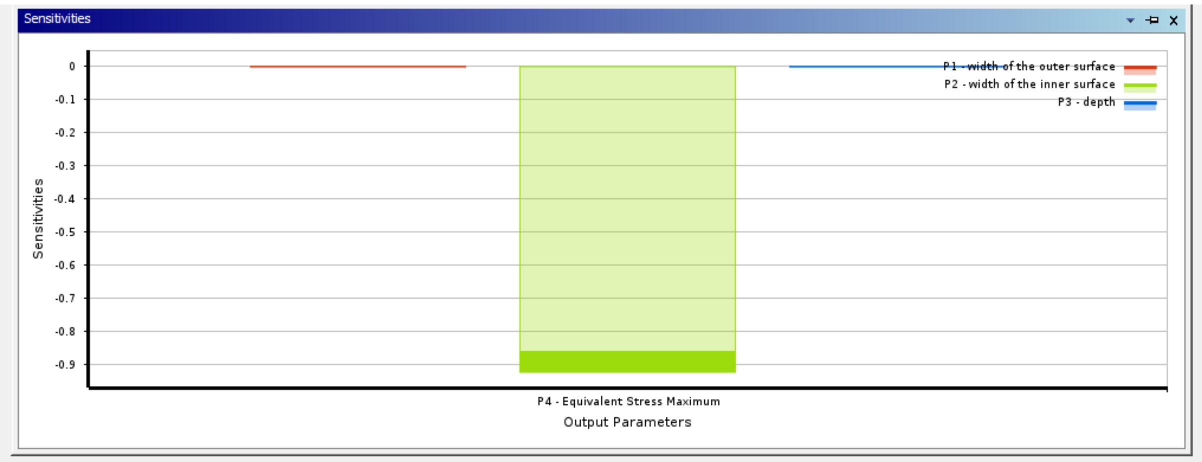


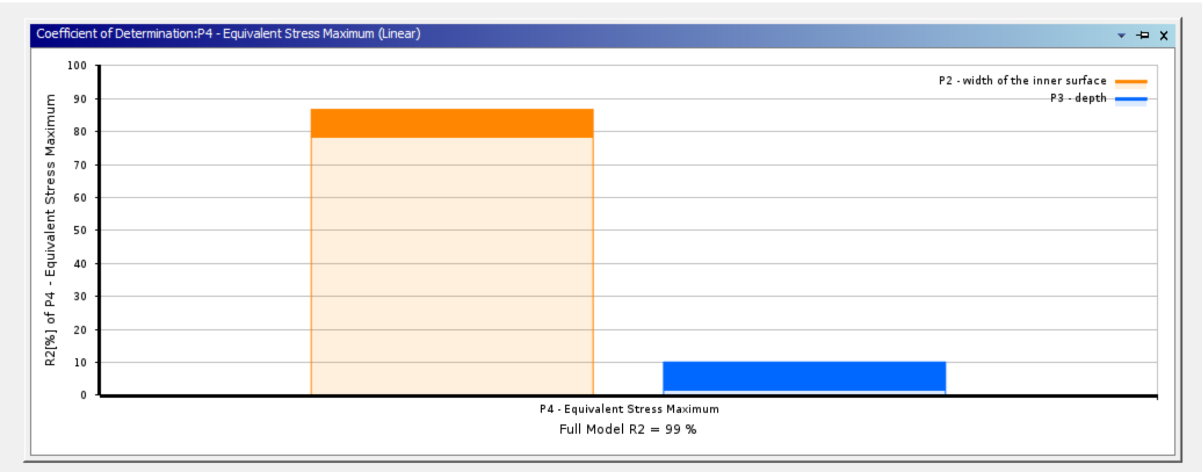
Parameters correlation

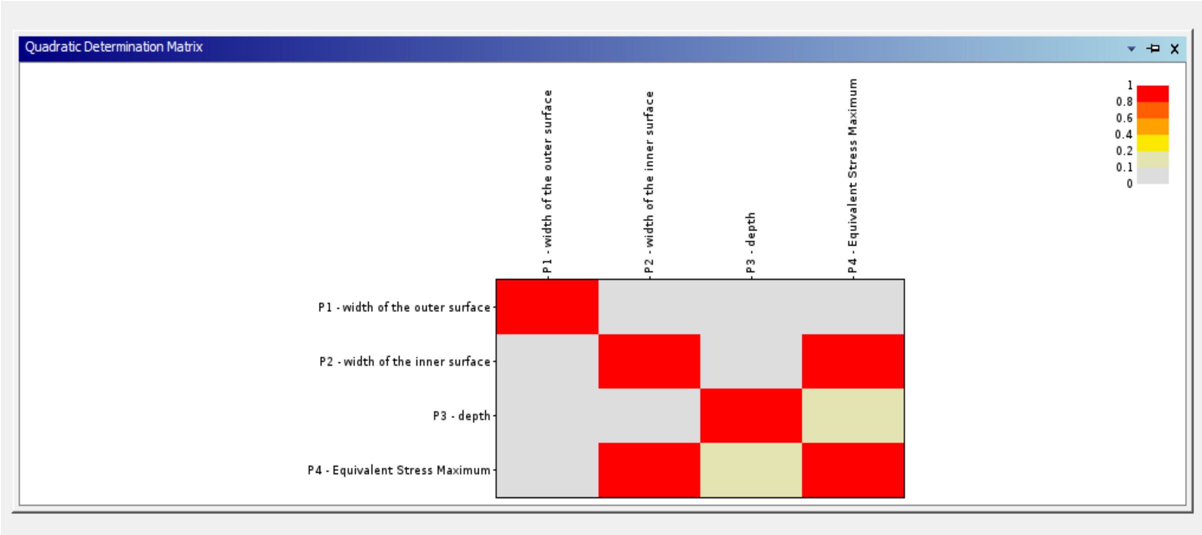


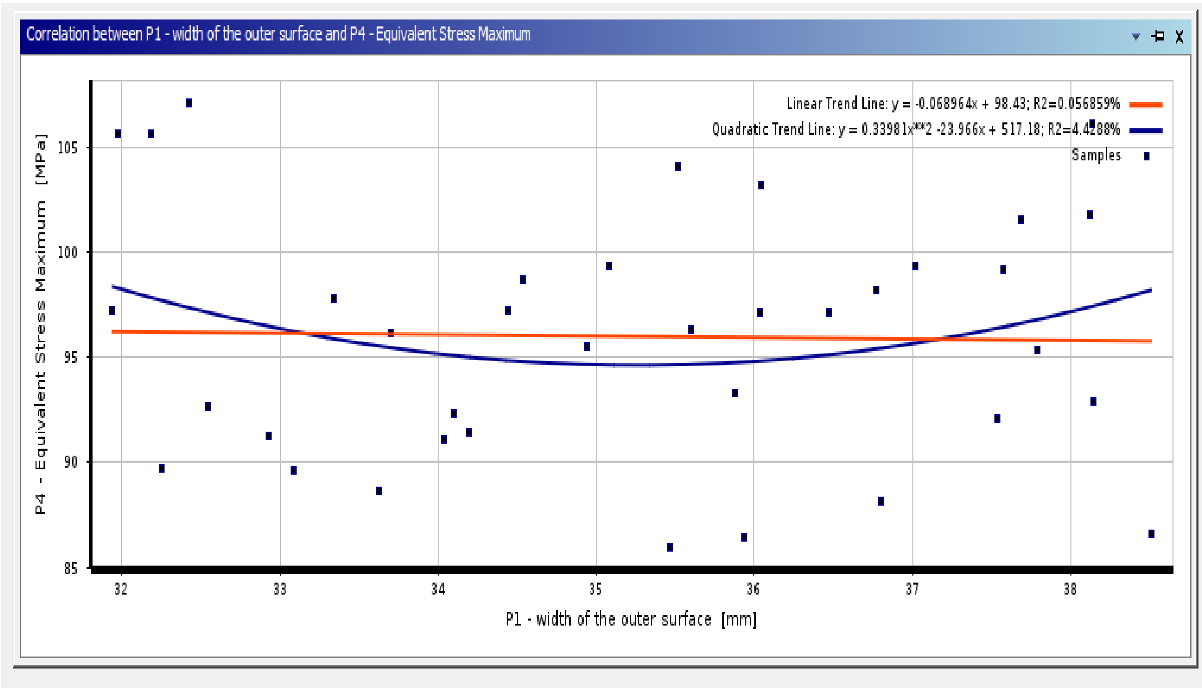
8. Parametric correlation.








Quadratic Determination Matrix
P1 - width of the outer surface
P2 - width of the inner surface
P3 - depth
P4 - Equivalent Stress Maximum
1
0.8
0.6
0.4
0.2
0.1
0


Correlation between P1 - width of the outer surface and P4 - Equivalent Stress Maximum
Linear Trend Line: y = -0.068964x + 98.43; R2=0.056859%
Quadratic Trend Line: y = 0.33981x**2 -23.966x + 517.18; R2=4.4288%
Samples
P4 - Equivalent Stress Maximum [MPa]
P1 - width of the outer surface [mm]
105
100
95
90
85
32
33
34
35
36
37
38