Finite Element Analysis for a Knuckle Crane Boom

by

Mohd Ashraf Arif b. Mohamad Refaee

Dissertation submitted in partial fulfilment of

the requirements for the

Bachelor of Engineering (Hons)

(Mechanical Engineering)

SEPTEMBER 2011

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Finite Element Analysis for a Knuckle Crane Boom

by

Mohd Ashraf Arif b. Mohamad Refaee

A project dissertation submitted to the

Mechanical Engineering Programme

Universiti Teknologi PETRONAS

in partial fulfillment of the requirement for the

BACHELOR OF ENGINEERING (Hons)

(MECHANICAL ENGINEERING)

Approved by,

Idris bin Ibrahim, P.Eng. MIEM
Senior Lecturer
Mechanical Engineering Department
Universiti Teknologi PETRONAS

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

September 2011

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

MOHD ASHRAF ARIF BIN MOHAMAD REFAEE

ABSTRACT

A knuckle crane is a very unique crane due to its ability to bend and reach the place near it based and increased in payload with smaller size body compared to other crane. In order to implement the used of knuckle crane in various condition, an analysis should be done to understand the behavior of the crane. From the result of the analysis, a solution can be made possible as to increase the ability of the crane and make it more efficient in various working environment. The project objectives are to study the stress profile on the knuckle crane boom using finite element analysis method. The effect of the different material on the maximum stress of the crane under specific load will be included in the scope of study.

In this project, a 3D model of knuckle crane boom is made using Autodesk inventor 2012 based on the drawing of the reference crane that is selected. The model of the knuckle crane boom is drawn in 3 parts which each part of the crane is assembled before the model is simulated using finite element method. Load is applied to the model based on the load chart provided with the reference model throughout the simulation process. Second objective is achieved by assigning different material on the model and simulated. Result will be compared again current material which is high-strength low-alloy steel and recommendation will be made.

The project result will identify the maximum location of stress profile on the boom structure and the best material that can be used in crane manufacturing process. Stress analysis on the model based on the load chart proves that the highest concentrated stress is at the assembly point where the area is small enough to develop high stress. Second reason for the stress distribution to behave as shown in the result is due to the existed joint, pin joint, between the main boom and the second boom structure.

Thus, it can be concluded that the strength of the joint is more as the contributed factor to the limitation of the crane lifting capability compared to the boom itself. It is proved that stainless steel 440c is the best material for boom manufacturing and it is believed to be one of the solutions for high end crane product. However, the process of implementation the material in current world might have some difficulty in term of cost and manufacturing ability.

ACKNOWLEDGEMENTS

First and foremost, I would like to praise God the Almighty for His guidance. Though difficulties occurred, His guidance and blessing gave me the chance to complete this final year project successfully. Here, I would like to use this special opportunity to express my heartfelt gratitude to everyone that has contributed in order to complete the final year project.

My deepest appreciation goes to my supervisor, Ir Idris B. Ibrahim, Lecturer of Mechanical Engineering Department, Universiti Teknologi PETRONAS who advises and guides me throughout two (2) semester of Final Year Project. I really acknowledge all the precious words from him and hope the moment working with him remains as valuable experience for my future undertakings.

I would also like to address my thankfulness to Favelle Favco company especially Mr. Derek. He was very generous and helpful in helping me during the progress of my research. Apart from them, I really appreciate all the lecturers of UTP, my beloved family members, and also my colleagues that gave feedback's and helped a lot through the useful ideas and advises. With their contribution, my research completed very well.

Thank you.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLE

CHAPTER 1

INTRODUCTION

1.1 Background of Study

1.1.1 Crane

A crane is basically a machine that is used both to lift and lower item and to move them horizontally. The main purpose is a heavy weight lifter and transporting it to certain places. Crane machine is mostly employed in transport industry where loading of item from the manufacturing company to the assembling company. The crane design consists of three major considerations that will affect entirely the crane features and lifting capability. The considerations of the crane are

1. Crane must be able to lift the designed load.

2. Crane should not topple while operating.

3. Crane must not rupture.

Most of the crane use internal combustion engine or electric motor and hydraulic system to operate. Although the crane basic system is nearly similar for most type of crane, the function and the advantages of the crane is greatly differ. Figure 1.1 shows several type of crane which is commonly used worldwide.

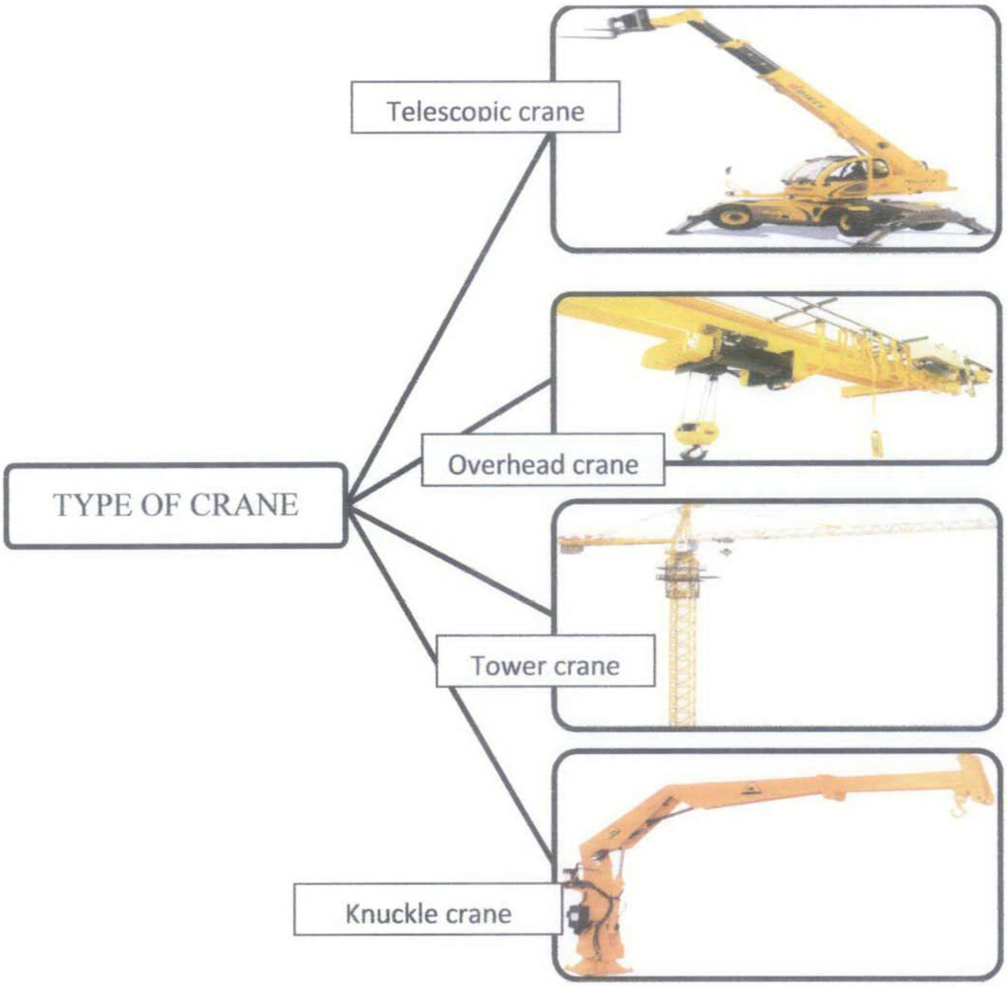


Figure 1.1: Type of crane

1.1.2 Boom

The boom structure is a structure which includes a plurality of elongated sections which can be interconnected to permit formation of a boom structure of selected length [1]. The boom structure is used for lift supporting structure and consists of an outer boon which is pivotally mounted on the lift supporting structure for movement about horizontal axis [2]. One of the most important considerations that need to be considered while designing the boom is the load carrying capacity as compared to certain size limitations. The design boom should be able to withstand the load being lifted and the weight of the boom itself.

As the boom is operating, great stress are developed within the boom sections. More to that, crane boom structure will be subjected to concentrated reactions at the contact point between boom sections and some of the contact points can vary in location along the length of the boom, as the sections are extended and retracted[3]. `

1.1.3 Knuckle Crane

Knuckle crane have lots of advantages which is why this type of crane is used in most type of applications from electric, water, natural gas, propane or underground construction. Some of the advantages are that knuckle crane have are:

i. Greater efficiencies since loads can be place directly wherever it is needed due to the ability to bend on the knuckle section of the boom.
ii. Increased in payload as a result of the knuckle crane being lighter and more compact.
iii. Ability to reach longer distances at a lower overall height.
iv. Ability to have long reaches that compete with telescopic crane.

Each crane built in today engineering society will be provided with a manual guideline for the operation condition. This guideline is known as load chart and it is use to determine the maximum lifting capability of the crane on different angle and height. The weight of the crane boom gives a significant contribution in the load chart. Reduction of the weight will see huge improvement in the chart. Figure 1.2 is the example of a load chart of a knuckle crane:

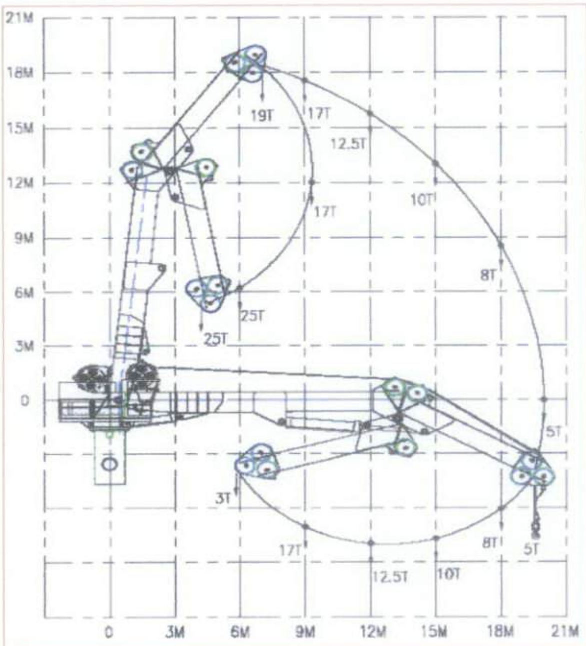


Figure 1.2: Load Chart of a Knuckle Crane

1.2 Problem Statement

Cranes are widely used in ports and on large ships for the loading/unloading of containers and bulk material. Due to recent industrial demand for the exploitation of ocean resources, these application fields of onshore cranes have been extended to offshore platform such as FPSO (Floating Production, Storage and Offloading unit) [1]. Generally, it is well recognized that detailed information of operation loads is considered for the design or construction of the crane for the offshore platform. However, a detailed stress analysis of a knuckle crane boom structure due to the operation load is not available yet. In this study, using on finite element method, analysis on the stress distribution of the boom structure will be done to evaluate the boom.

Knuckle crane play an important role in lifting machine due to the advantages that it provide. However, if we are going to enhance the capability of the knuckle crane, study on the boom of the crane need to be done. Effect of the different materials on the crane boom stress distribution should be studied. The result from the research will be used to increase the lifting ability of the knuckle crane and thus increase it efficiency.

Knuckle crane lifting capability is all decided by the stress distribution of the boom structure. Equally distributed stress on the boom structure of the crane will make the crane able to function on it highest efficiency. However, it is nearly impossible to create a uniform or equally distributed stress on the crane boom due to the crane design - knuckle and segment on the boom, and the crane operation - different lifting height and angle. As the crane is operating, stress produce will create buckling effect on the boom structure.

The stress distribution of the boom will definitely be different in location alongside the crane. One of reason for this differentiation is the nature of design of the boom structure itself. In order to support the statement above, finite element analysis will be done based on reference data which will be provided by the collaborator company, Favelle Favco Berhad.

By completing this project, data regarding the knuckle crane boom structure on stress distribution and effect of the different type of materials should be concluded and thus, efficiency of the knuckle crane boom structure can be increased.

1.3 Objectives

The objective of this project is to study the stress profile on the knuckle crane boom structure using finite element analysis method at different lifting angle. Effect of different materials on the maximum stress concentration on the crane under specific load will also be included in the project.

1.4 Scopes of Study

The scopes of study for this project are:

i. Limited to only one type of knuckle crane design - reference model.
ii. Focused on the stress distribution at the boom structure
iii. Investigate the effect of different material on the stress distribution

1.5 Relevancy of the project

From general perspectives, this topic is related to the mechanical engineering field of study as it covers both design and material. This project is collaboration with Favelle Favco Company which is an expert in knuckle crane manufacturing. Thus, result from this project should be a help for the company in designing new knuckle crane machine.

1.6 Feasibility of the Project

Allocated time for the project is two semesters which is about 8 months. The scope of the project is to study on the stress distribution both for normal loading condition and maximum loading condition as well as different material effect on the knuckle crane boom structure. Within the period of time, the project is supposed to be completed and result with conclusion should be presented. For the project to be able to finish accordingly Gantt chart is made and methodology for the project is prepared. Plus, collaboration with the Favelle Favco will assist the completion of the project with live data from the manufacturing sites.

CHAPTER 2

LITERATURE REVIEW

2.1 Boom Structure Analysis

The boom structure is for use with the lift supporting structure and consists of an outer boom which is pivotally mounted on the lift supporting structure for movement about a horizontal axis [2]. Boom structure usually includes a plurality of elongated sections which can be interconnected to permit formation of a boom structure of selected length [1]. Most of the crane boom structures have an elongated web plate of uniform thickness, which is shaped to provide increased stiffness to resist buckling. The boom structure main purposed is to expand and retract accordingly to the function of the crane. It is also designed to be able to withstand maximum load with correlation of its support system. Load supporting boom structures of the type used in equipment, such as crane, draglines, and the like, are generally pivotally mounted adjacent one end portion thereof to a stationary or mobile base support and are equipped with suitable rigging to alter the inclination of the boom structure with respect to the ground in order to perform the various functions for which the equipment is made [5]. As the boom structures lift some load, stress will be distributed trough out the structure. However, this stresses are so great that repeated telescoping of the boom structure may cause failure of certain parts of the boom or may cause failure of welds where they are utilized in the boom construction [2].

2.2 Stress Analysis

Stress analysis is an engineering method that determines the stress in material and structures subjected to static or dynamic forces or load [8]. The aim of the analysis is usually to determine whether the element or collection of elements, usually referred to as a structure, can safely withstand the specified forces [8,9]. To obtain the aim, the determined stress from the applied force must be less than the allowable strength of the material. One method to specifying the allowable load for the design or analysis is to use a number called the factor of safety. Factor of safety is a ration of the failure load divided by the allowable load [8,9].

$$Factor\ of\ safety = \frac{maximum\ stress}{maximum\ allowable\ stress}$$

The analysis of stress within a body implies the determination at each point of the body of the magnitudes of the nine stress components. In other words, it is the determination of the internal distribution of stresses [8]. There are two methods to apply the analysis [8] which are modelling and experimental testing procedures.

2.2.1 Modeling

To determine the distribution of stress in a structure it is necessary to solve a boundary-value problem by specifying the boundary conditions, i.e. displacements and/or forces on the boundary[8]. Constitutive equations, such as e.g. Hooke's Law for linear elastic materials, are used to describe the stress: strain relationship in these calculations. A boundary-value problem based on the theory of elasticity is applied to structures expected to deform elastically, i.e. infinitesimal strains, under design loads. When the loads applied to the structure induce plastic deformations, the theory of plasticity is implemented [9].

Approximate solutions for boundary-value problems can be obtained through the use of numerical methods such as the finite element method, the finite difference method, and the boundary element method, which are implemented in computer programs. Analytical or close-form solutions can be obtained for simple geometries, constitutive relations, and boundary conditions [8, 9].

2.2.2 Experimental testing

Stress analysis can be performed experimentally by applying forces to a test element or structure and then determining the resulting stress using sensors. In this case the process would more properly be known as *testing* (destructive or non-destructive). Experimental methods may be used in cases where mathematical approaches are cumbersome or inaccurate. Special equipment appropriate to the experimental method is used to apply the static or dynamic loading.

2.3 Finite Element Method (FEM)

The finite element method (FEM) is a powerful technique to simulate the linear and nonlinear behaviour of structures which in this case will be a knuckle crane boom [11]. Finite element analysis is based on the idea that more accurate and detail evaluation of a solution for a model can be gained by dividing the model into small part (finite element) [4]. By using the finite element model for analysis, behaviour of the structure such as stress distribution, thermal distribution, strength and stiffness of it can be known [10, 11]. Thus, due to the vast function of the method, lots of company are using this method as a medium of evaluation before the manufacturing process.

The most known programs for FEM used in structural analysis are Ansys, Autodesk Inventor, Abaqus, Nastran, Cosmos, Cosmos Works, FEMAP and so on [10].

In general, when using finite element analysis (FEA) software the next steps must be followed [10, 11]:

(1) Defining the geometry (by building the model in a modeling program for complex structures or directly in finite element method programs for simple ones),

(2) Defining the element type used for meshing, material proprieties (Young modulus, Poisson ratio and so on),

(3) Applying the meshing (auto mesh or mapped mesh),

(4) Defining the boundary conditions (displacements, loadings and so on),

(5) Solving the problem and analyzing the results.

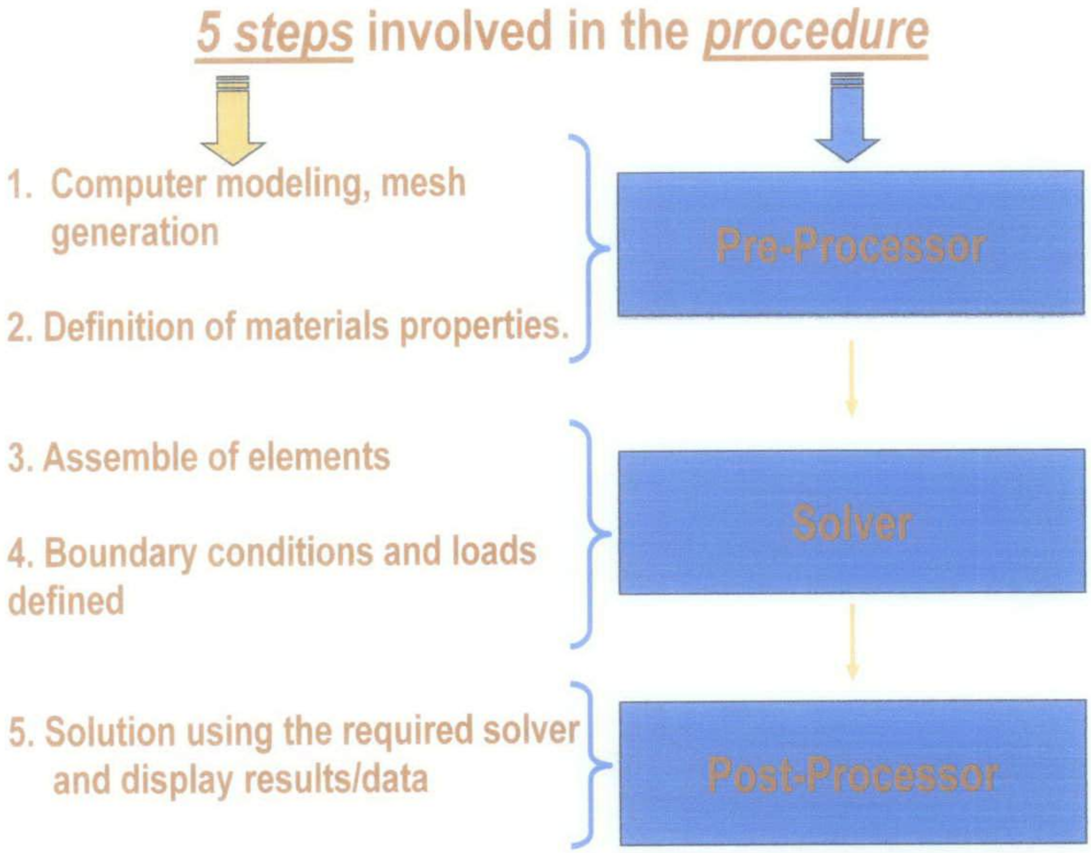


Figure 2.1: Steps using Finite Element Analysis program

In order to have this method to be able to correctly evaluate the boom structure of the crane, accurate design and boundary condition of the boom need to be acquired and used [4,10,11]. The most important parts when dealing with a FEA program are the meshing and elements type used [10]. We may obtain in some situations errors like very high or very low stresses, material discontinuities and so on because of the improper meshing and element type used. There are also presented the errors that may appear for each Finite Element Analysis variant used [10].

Based on [1], finite element method is used to understand the wind effect on the crane. This research paper is mainly focusing on the effect of the offshore crane which is subjected to dynamic forces as well as wind loads. Designing the offshore crane is a first step done for the study. The design of 100 tons was used to demonstrate the design process. The design is done by the CAD (computer aided design). However, this design is then simplified to make the analysis easier [1].

For the finite element analysis, detailed design is used on the study. The design is done separately as an individual part before it is assembled according to the real model [1]. Figure 2.2 show the 3D FEA design of the reference jib by part.

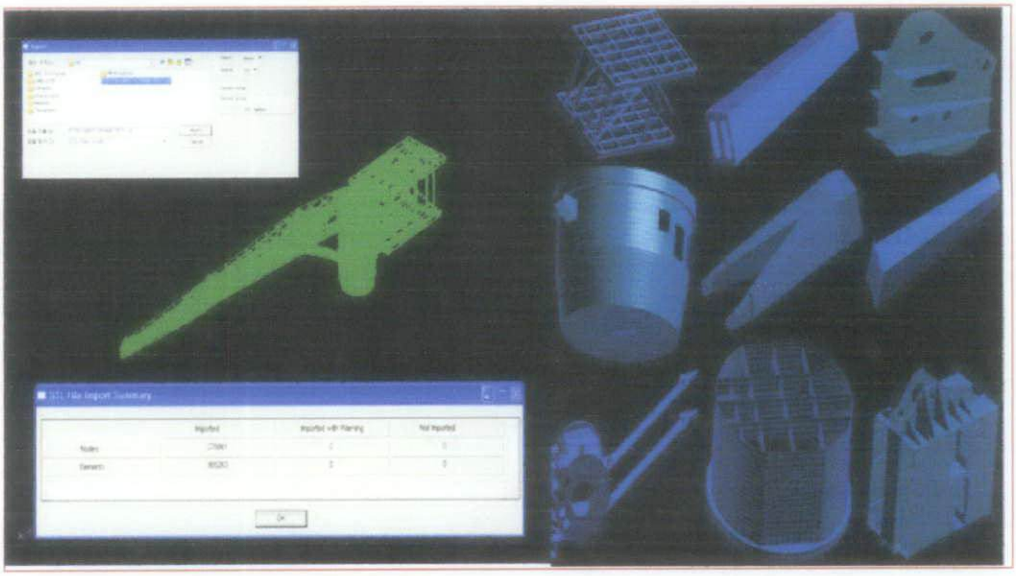
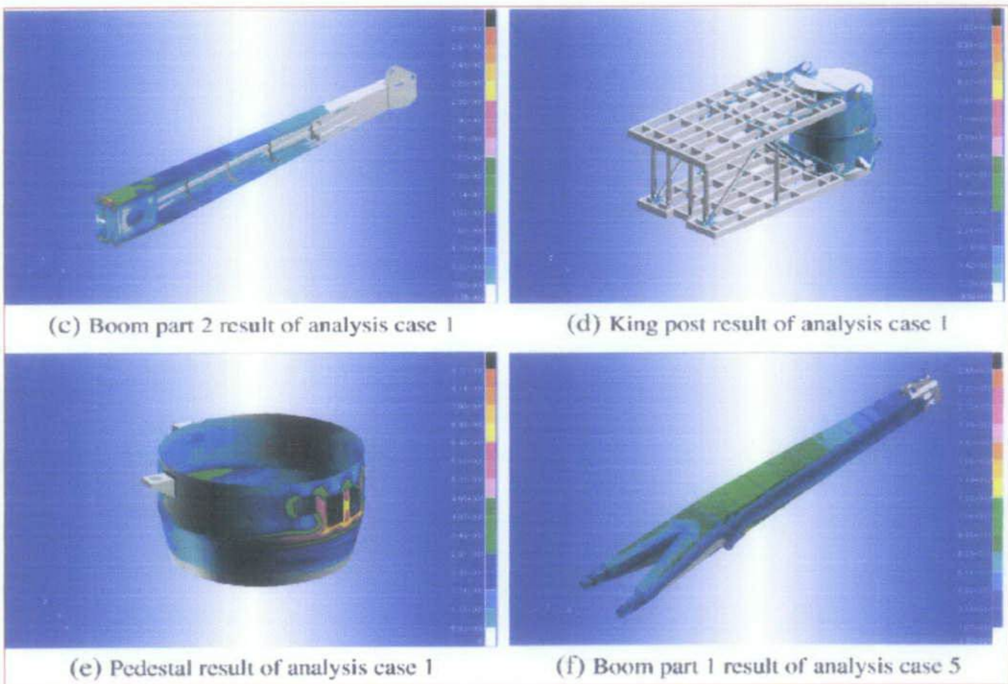


Figure 2.2: 3D FEA design for Jib (by part)

Result of the study is done separate for three types of analysis which are deformation, stress and fatigue estimation [1]. Stress analysis for the crane is done separately for each part of the crane. Figure 2.3 show the example of stress analysis done by separately by parts:



(c) Boom part 2 result of analysis case 1

(d) King post result of analysis case 1

(e) Pedestal result of analysis case 1

(f) Boom part 1 result of analysis case 5

Figure 2.3: 3D FEA simulation of stress distribution

CHAPTER 3
METHODOLOGY

3.1 Overall Project Methodology

Project methodology is generally a guideline for the project to ensure the completion of the project. Generally, the methodology of any project will start with some literature review. The literature review is hoped to be able to assist throughout the project. Other steps in methodology are usually unique for each project. Figure 3.1 shows the methodology of this project.

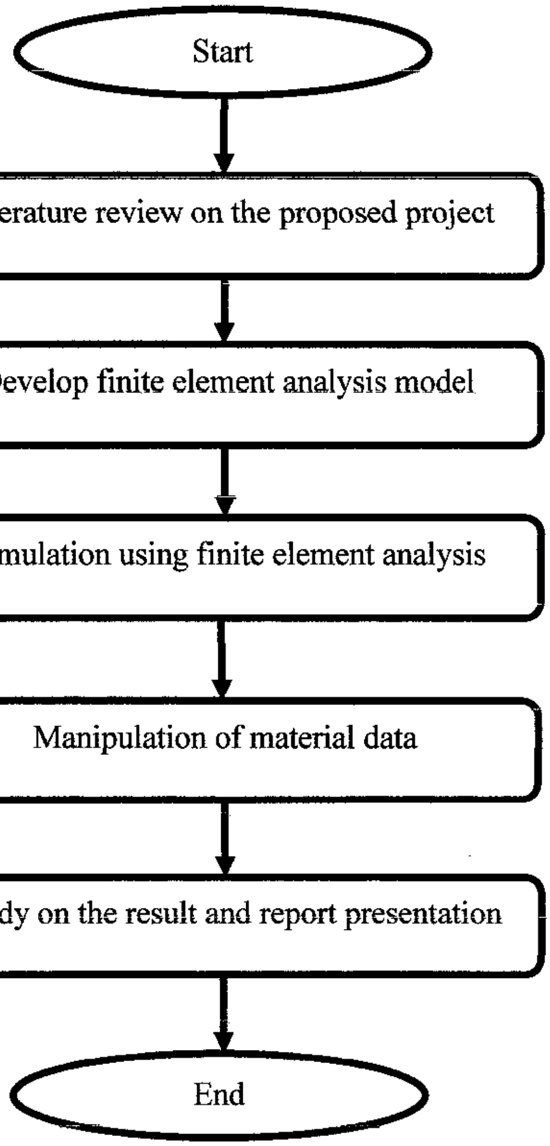


Figure 3.1: Methodology for the project

3.1.1 Literature review on the project

Basically, literature review is done to gain some information regarding the project. The purpose of literature review will be to assist in completing the project. For this project, several scope of topic will need to be review. The topics are:

- Study on design parameter, consideration and material.
- Loading function on the knuckle crane boom - type of force, moment, direction.
- Boundary condition on the crane boom structure.
- Current material used for the knuckle crane boom structure.

Some of the data above will be gain by requesting from collaborating company, Favelle Favco.

3.1.2 Developing model and run a simulation for finite element analysis

Developing model is done by designing software called Autodesk Inventor 2012. The model is design by referring to the reference model which is gained from the literature review. Selected model is from Coastal Hydraulic Crane Incorporation under the model CKB 25-40. The design is in form of 2D drawing complete with the load chart of the crane.

3.1.3 Boundary condition

Zero displacement constraints must be placed on some boundaries of the model to ensure an equilibrium solution. The constraints should be placed on nodes that are far away from the region of interest to prevent the stress or strain fields associated with reaction forces from overlapping [12]. For the project, all of the assembly point connecting the boom is a pin joint. Pin joint does not allow translational movement and it is assumed to be frictionless in order to allow rotation of the boom with respect to each other. However, to create the constraint for the equilibrium solution, a fix joint is assumed to be between the holder and the main boom. The fix joint will not allow any rotation or translation in any direction. Thus, with the existed fixed joint between the main boom and the holder, the crane can be assumed to be in the static condition and thus make the analysis possible.

For a fixed joint between the holder and the main boom, the boundary conditions are as follows;

$w(0)=0$. This boundary condition says that the end of the main boom will not move and will experience any deflection.

$w'(0)=0$. It is also assume that the boom will stay horizontal when no load applied, so that the derivative of the deflection function is zero at that point.

$w''(L)=0$. This means that there will be no bending moment at the pin joint between main boom and second boom.

$w'''(L)= -mg$ This is due to the lifting load that is applied at the end of the second boom

3.1.4 Loading condition

For the loading condition of the crane under different angle, data is referred from the load chart that is provided with the model 2D drawing. The load is applied directly at the end of the second boom which implies that the crane is lifting a load while it is analyze. The direction of the load will be in the -y direction. As the crane is analyzed with different lifting angle, the loading condition will also be changed by referring to the load chart.

3.1.5 Manipulation of material data

This step involve in comparing the current material used with the material selected for the purpose of the study. Based on the literature review, current material used is in the form of High-Strength Low-Alloy steel, which come in several standard as this material is not define by the quantity of the alloy inside but based on the strength it provide. On the purpose of the study, one type of High-Strength Low-Alloy steel is selected and made as the current material.

3.2 Research Methodology

3.2.1 The steps of research:

1. Gain information of the basic dimension and weight of knuckle crane boom for the references model. The information should be able to assist in developing the model in 3D using autodesk inventor 2012 program.

2. Understand the load chart of the selected knuckle crane.
   a. Maximum operating load with respect to boom angle
   b. Maximum operating angle of the knuckle crane boom

3. Identify direction of load act on the knuckle crane boom.
   a. Load transfer on the boom from the lifting action
   b. Load transfer during crane movement – different operating angle
   c. Reaction load from the supporting boom and hydraulic cylinder

4. Identify type of load act on the frame – torsion, axial, tension, compression and etc.

5. Simplify the static and dynamic concept of knuckle crane boom to apply on the analysis.

3.3 Software for the project

Figure 3.2: Inventor software program

Autodesk Inventor 2012 software is used to draw and create 3D model of the designed frame. Besides designing the frame, the software is utilized also to do finite element analysis.

3.4 Reference Model CKB 25-40

As stated in the methodology above, the reference model is built in 3D to conduct the stress analysis. Therefore, the dimension of the reference model is gained. Besides, all parameters including common material of the knuckle crane boom and the weight of the boom are also researched for further analysis. Tables 3.1 showed the specification of knuckle crane boom-model CKB 25-40. Figure 3.3 shows the 2D drawing of the reference model.

Table 3.1: Specification of Knuckle Crane Boom model CKB 25-40

Features	Specifications
Model Number	Coastal Hydraulic Crane Incorporation
Boom Length	12.192 M
Boom Width	0.6858 M
Boom Material	High-Strength Low-Alloy steel
Maximum Operation Angle (Boom)	80 degree
Maximum Overturning Moment	1628744 N.M
Maximum Vertical Reaction	55850 Kg
Total Boom Weight	10491 Kg
Maximum Load (Dynamic Motion)	4989 Kg
Minimum Load (Dynamic Motion)	3764 Kg
Maximum Load (Static Motion)	7257 Kg
Maximum Load (Static Motion)	5669 Kg

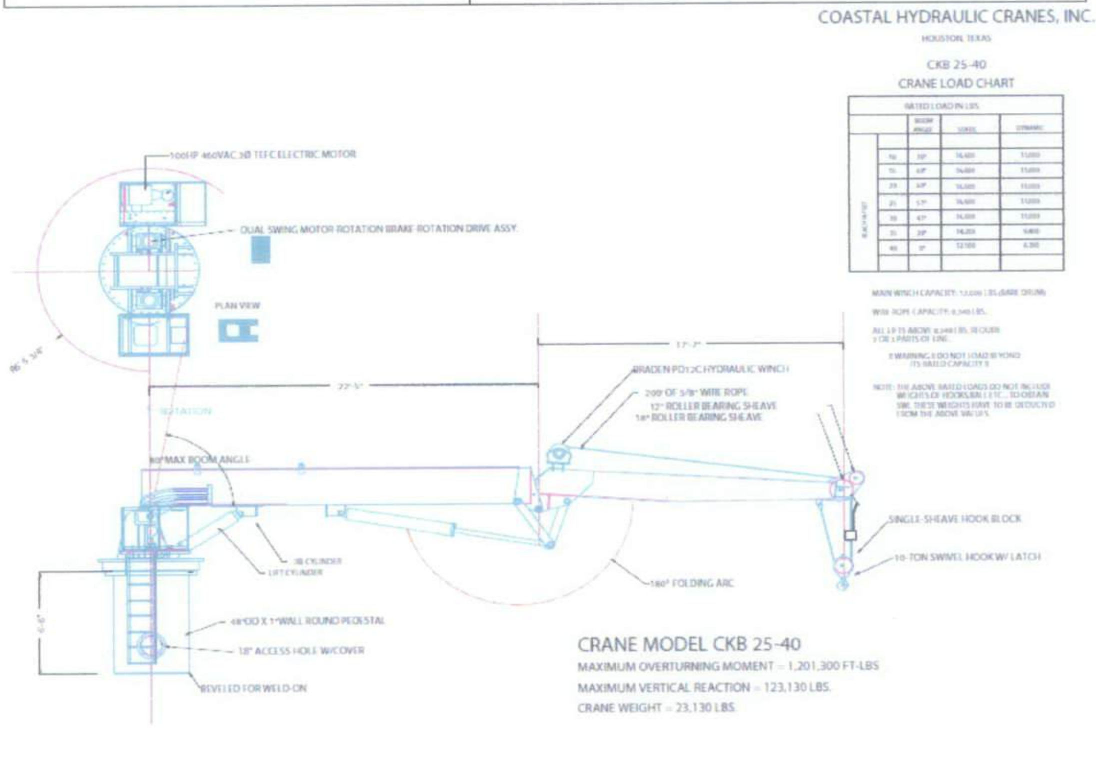


Figure 3.3: 2D drawing of reference model, CKB 25-40

3.5 High-strength low-alloy steel

Different material will be studied in this project in order to learn more on the effect of the material on the maximum stress concentration and stress distribution. Point to be noted that even though the material is changed, the maximum stress location will still be at the same point because the location is hugely affected by the geometry and design of the crane. Thus, expected result should be in the value of the maximum stress concentration rather than the stress distribution.

Commonly used material for knuckle crane boom is high-strength low-alloy steel. This material is currently the best suitable material that is used in the production of the crane. The uniqueness of the steel is that production of the material is base on specific material properties and not the material composition. The steel have small carbon content in it, 0.05% - 0.25% to retain formability and weldability and some manganese, copper, nickel, niobium, nitrogen, vanadium, chromium, molybdenum, titanium, calcium and zirconium. The advantage of the steel compared to carbon steel is that high-strength low-alloy steel is 20% - 30% lighter than carbon steel with the same strength. Plus, it is also more rust resistant compared to the carbon steel due to their lacking in pearlite. Table 3.2 show the mechanical properties of the current material used in crane manufacturing.

Table 3.2: Mechanical properties of High-strength Low-alloy steel

MECHANICAL PROPERTIES	HIGH STRENGTH LOW ALLOY STEEL
DENSITY (g/cm^3)	7.84
YOUNG MODULUS (GPa)	200
POISSON RATIO	0.287
YIELD STRENGTH (Mpa)	275.8
ULTIMATE TENSILE STRENGTH (Mpa)	448

3.6 Alternative material analysis

In order to propose new material to the design, several considerations need to be done. This is to ensure that correct material is chose as the alternative for the current material. The step for the consideration is done as below.

1. Determine the weight factor of the mechanical properties.

Table 3.3: Weight factor for mechanical properties

MECHANICAL PROPERTIES	WEIGHT FACTOR
DENSITY (g/cm^3)	3
YOUNG MODULUS (GPa)	2
POISSON RATIO	1
YIELD STRENGTH (Mpa)	3
ULTIMATE TENSILE STRENGTH (Mpa)	1
	10

2. The reasoning for the weight factor

Density of the material gives huge effect on the stress distribution and concentration of the knuckle crane boom. Other than that, higher the density, higher the mass and thus lower the weight lifting capability of the crane.

Density = Mass/Volume

From this equation, as the volume of the crane will stay constant for the entire project, density will give biggest role in determining the crane mass when different material is applied.

The yield strength is a point where a material begins to deform plastically. Prior to the yield point the material will deform elastically and will return to its original shape when the applied stress is removed. Once the yield point is passed, some fraction of the deformation will be permanent and non-reversible. Yield point is vital when designing a component since it generally represents an upper limit to the load that can be applied. Thus, higher yield strength means higher weight lifting capability due to higher load that can be applied. So, based on the reasoning above, it is reasonable for the density and the yield strength to be weighted 3.

Second highest weighted properties are the young modulus which is weighted 2. Young modulus which is also named modulus of elasticity is a measure the stiffness of an elastic material. It is defined as the ratio of the uni-axial stress over the uni-axial strain in the range of stress in which Hooke's Law holds. The higher the young modulus, the little the deformation that will occur on the material when it is applied with load. Thus, due to this reason, this mechanical property is weighted 2.

Ultimate tensile strength and poisson ration is weighted 1 which is the lowest weight among the properties poisson ratio is defined as the ration of transverse to longitudinal strains of a loaded specimen. Most of the material existed nowadays have the Poisson's ratio range from initially 0 to about 0.5. Generally, stiffer materials will have lower Poisson's ratios than softer materials. For the steel material, the ration should be exhibiting values around 0.3.

3. Propose alternative material

Table 3.4: Mechanical properties for selected alternative material

MATERIAL	DENSITY (g/cm^3)	YOUNG MODULUS (GPa)	POISSON RATIO	YIELD STRENGTH (Mpa)	ULTIMATE TENSILE STRENGTH (Mpa)
HSLA STEEL	7.84	200	0.287	275.8	448
TITANIUM	4.51	102.810	0.361	275.6	344.5
NICKEL-COOPER ALLOY 400	8.83	179.3	0.315	220	558
DUCTILE IRON	7.1	168	0.29	332	464
CARBON STEEL	7.870	200	0.29	350	420
STAINLESS STEEL 440C	7.75	206.7	0.27	689	861.250

4. Calculate the weight factor

Table 3.5: Weighting table for alternative material

		HSLA STEEL		STAINLESS STEEL 440C		CARBON STEEL		DUCTILE IRON		NICKEL-COOPER ALLOY 400		TITANIUM	
	Weight	Score	Weighted Score	Score	Weighted Score	Score	Weighted Score	Score	Weighted Score	Score	Weighted Score	Score	Weighted Score
YOUNG MODULUS	0.2	7	1.4	8	1.6	7	1.4	6	1.2	6	1.2	7	1.4
POISSON RATIO	0.1	7	0.7	8	0.8	6	0.6	6	0.6	5	0.5	5	0.5
YIELD STRENGTH	0.3	7	2.1	9	2.7	6	1.8	7	2.1	4	1.2	6	1.8
DENSITY	0.3	8	2.4	8	2.4	6	1.8	8	2.4	5	1.5	9	2.7
ULTIMATE TENSILE STRENGTH	0.1	6	0.6	9	0.9	5	0.5	7	0.7	8	0.8	5	0.5
		Sum	7.2		8.4		6.9		7.0		5.2		6.9

CHAPTER 4

RESULT AND DISCUSSION

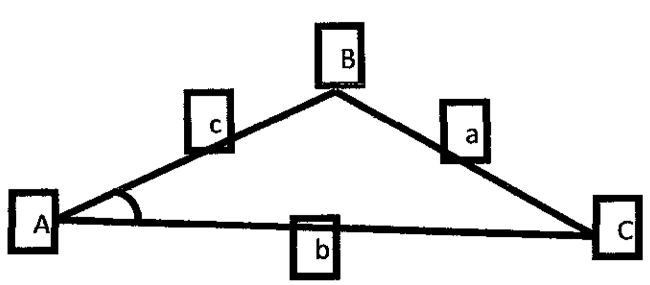
4.1 3D Drawing of Reference Model

Based on the dimension indicated, a 3D model of reference knuckle crane boom has been built for analysis process. The 3D model is constructed in detail in order to gain precise analysis result. Figure 4.1 shows the drawing of the crane boom according to the reference model.

Table 4.1: Load chart of reference knuckle crane

MAXIMUM BOOM REACH	MAIN BOOM ANGLE	STATIC LOAD (Kg)	DYNAMIC LOAD (Kg)
10	76^0	7257	4989
15	68^0	7257	4989
20	60^0	7257	4989
25	51^0	7257	4989
30	41^0	7257	4989
35	29^0	6441	4263
40	0^0	5669	3764

From the load chart:



b	Maximum reach length
a	Second boom length
c	Main boom length
A	Main boom angle

$b^2 = a^2 + c^2 - 2ac \cos B$

By using the equation, the second boom angle can be known and applied in the finite element analysis.

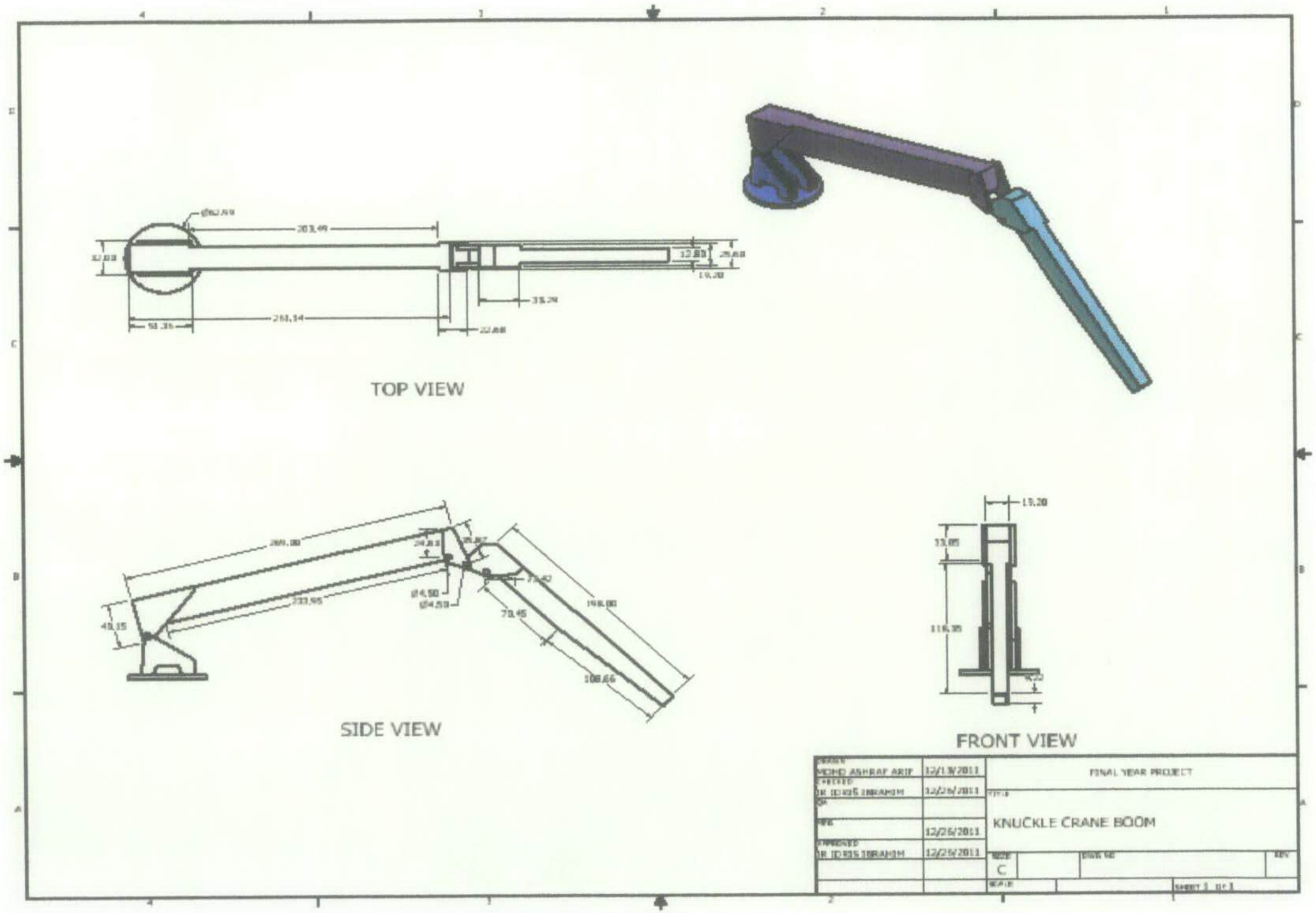


Figure 4.1: 3D model of knuckle crane boom based on the reference

4.2 Finite Element Analysis on Stress Profile

The knuckle crane boom is modeled as seen in 2D drawing. The design is done in 3 parts, main boom, second boom and the holder. However, we are only interested in the stress distribution alongside the boom and not the holder. Several constraints is applied to the boom in order to perform the finite element analysis. Table 4.2 shows the constraint location throughout the analysis process.

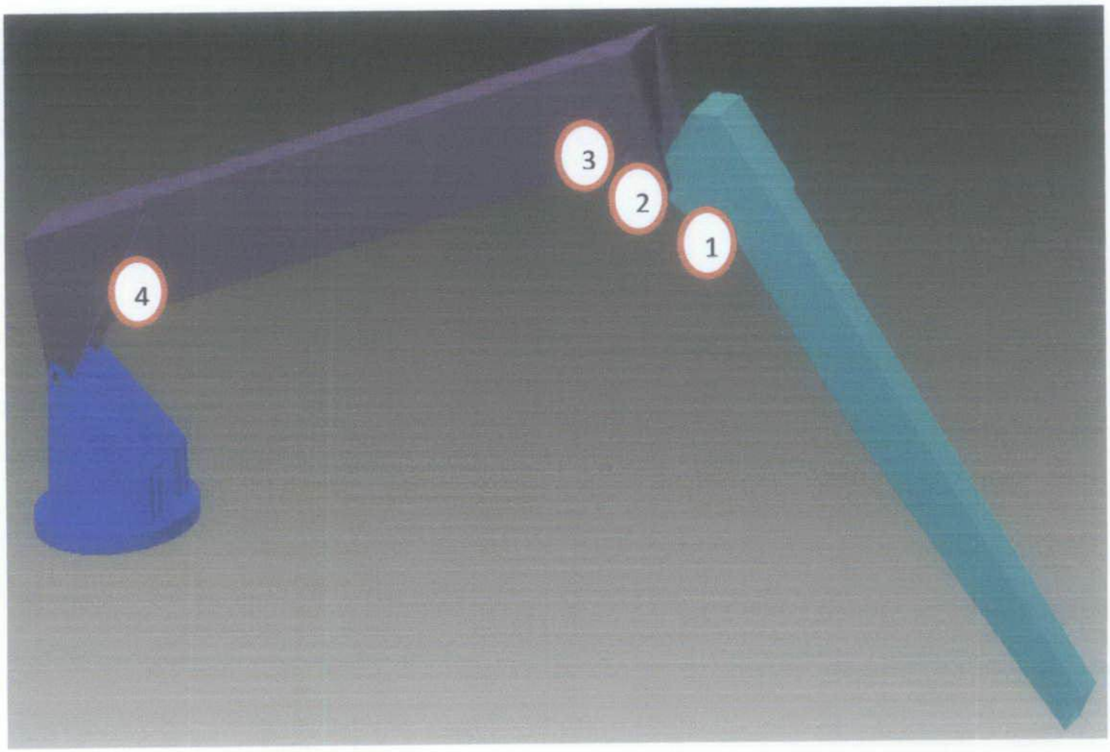


Figure 4.2: Constraints location for knuckle crane boom model

By knowing the maximum normal load that can be applied to the crane from the load chart, the loads are applied on the knuckle crane boom for the stress analysis.

4.2.1 Stress analysis result for high-strength low-alloy steel

The result of the forces applied is shown in figure. From Von Mises Stress, the maximum stress is 125.2 MPa which occur at the assembly point of the two booms. Force from the lifting action (load) is transmitted to the crane boom and concentrated between the assembly points as indicate by the shading of color. This stress is still not exceeding the maximum allowable stress of the design.

Using equation for maximum allowable stress with safety factor 2:

$$Factor\ of\ safety = \frac{maximum\ stress}{maximum\ allowable\ stress}$$

Maximum stress = 275.8 Mpa

Factor of safety = 2

Maximum allowable = 137.9 Mpa

Therefore, the crane boom is proven to be able to sustain the lifting load stated in the load chart. Figure 4.3 shows the finite element analysis result for maximum stress in specific angle.

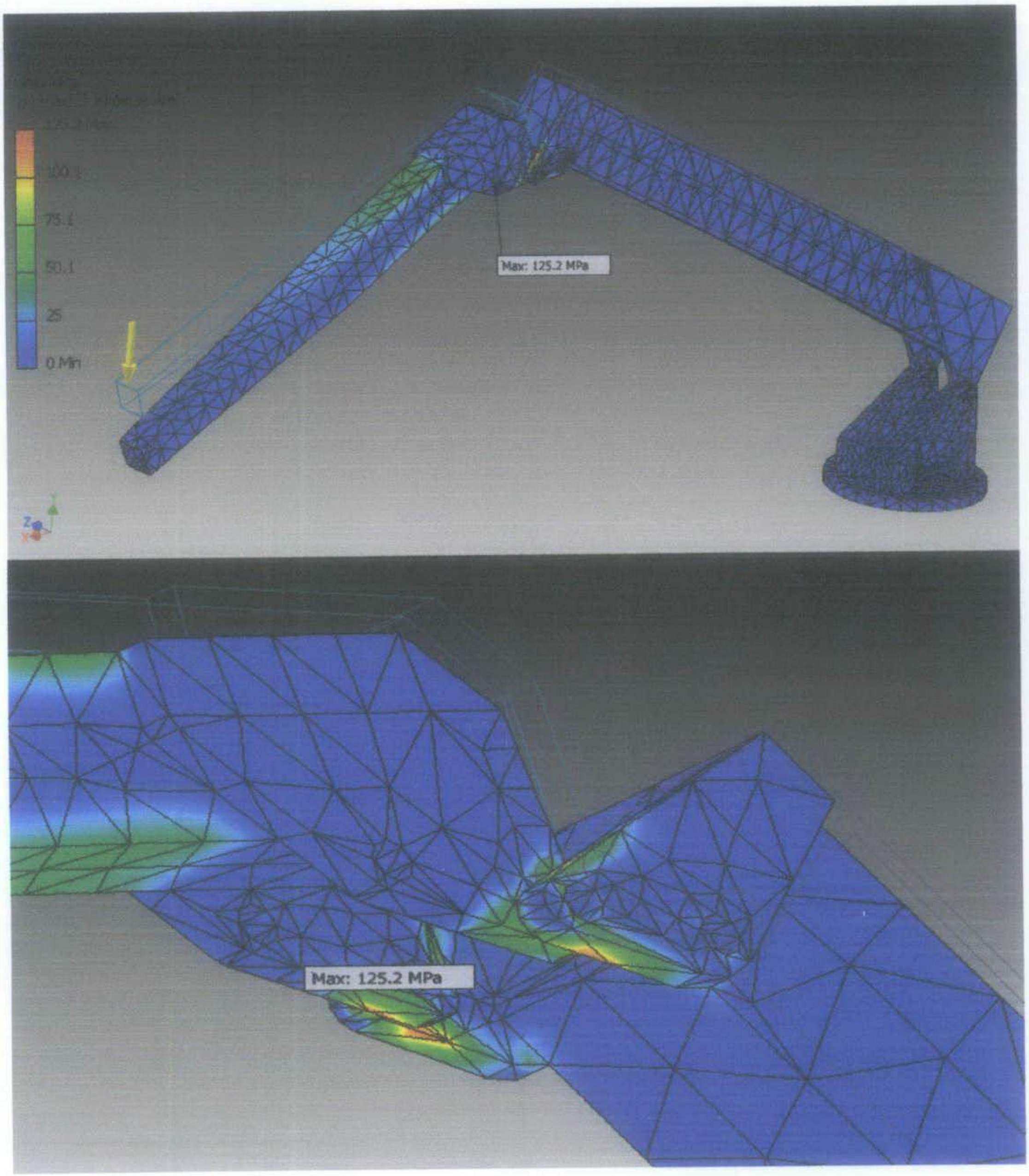


Figure 4.3: Knuckle crane boom with maximum load

From the result, it is also concluded that the crane lifting limitation is not entirely on the structure of the boom but it is more on the assembly section of the crane. This is due to the reaction force that is applied on the assembly section which is a pin joint.

The result of the forces applied for maximum deformation is shows in figure 4.4. From the deformation result, the maximum deformation is 36.98 mm which occur at the end of the boom where as the force is applied. Force from the lifting action (load) is transmitted to the crane boom and elongate the affected boom which is indicated by the shading of color. Deformation is mostly occurred alongside the second boom. This is an expected result because of the location of the load been applied is at the end of the boom itself. Due to the load, all the crane structure is forced to bend downward opposite with the load direction.

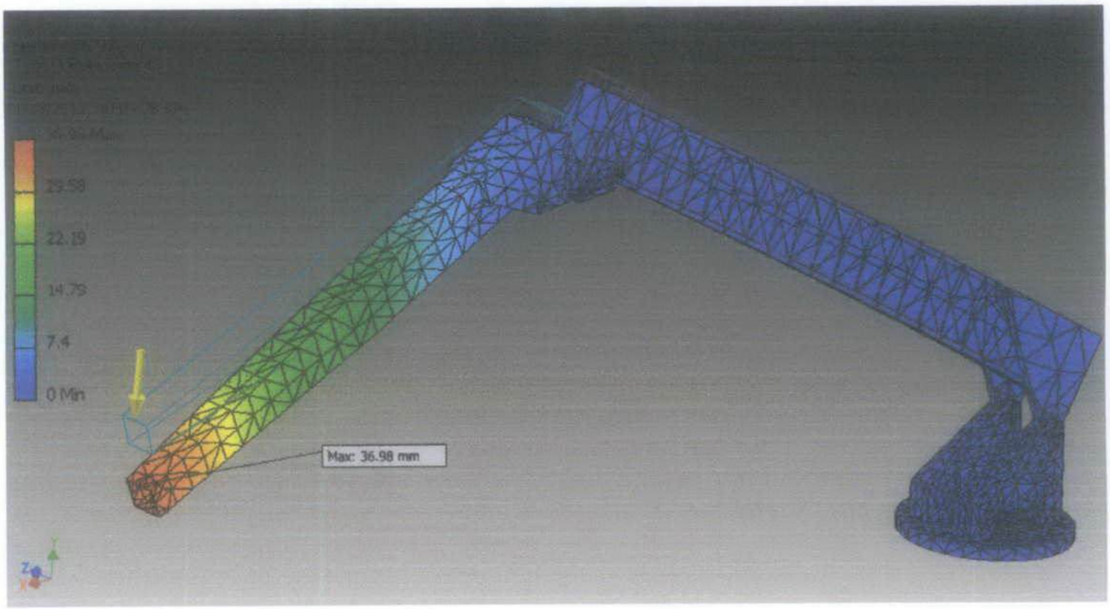


Figure 4.4: Maximum deformation of boom after load is applied

The stress analysis is done for all angles available in the load chart. This is to study the deformation and maximum stress distribution of the boom structure with referring to the load chart.

Working Condition 1

i. Main boom angle: 41^{o}

ii. Second boom angle: 100^{o}

iii. Load: 7257 kg

The analysis shows in figure 4.5 is the analysis for the crane under loading load of 7257 Kg, main boom angle 41^{o} and second boom angle 100^{o}. The maximum stress which is shown in the analysis is 131.7 Mpa which is still below the maximum allowable stress that is calculated, 137.9 Mpa. This indicates that the design of crane is workable as it is able to endure the amount of force that is stated in the load chart. The deformation of the crane is 28.8 mm. As expected, the deformation is mostly on the second boom due to the location of the applied load.

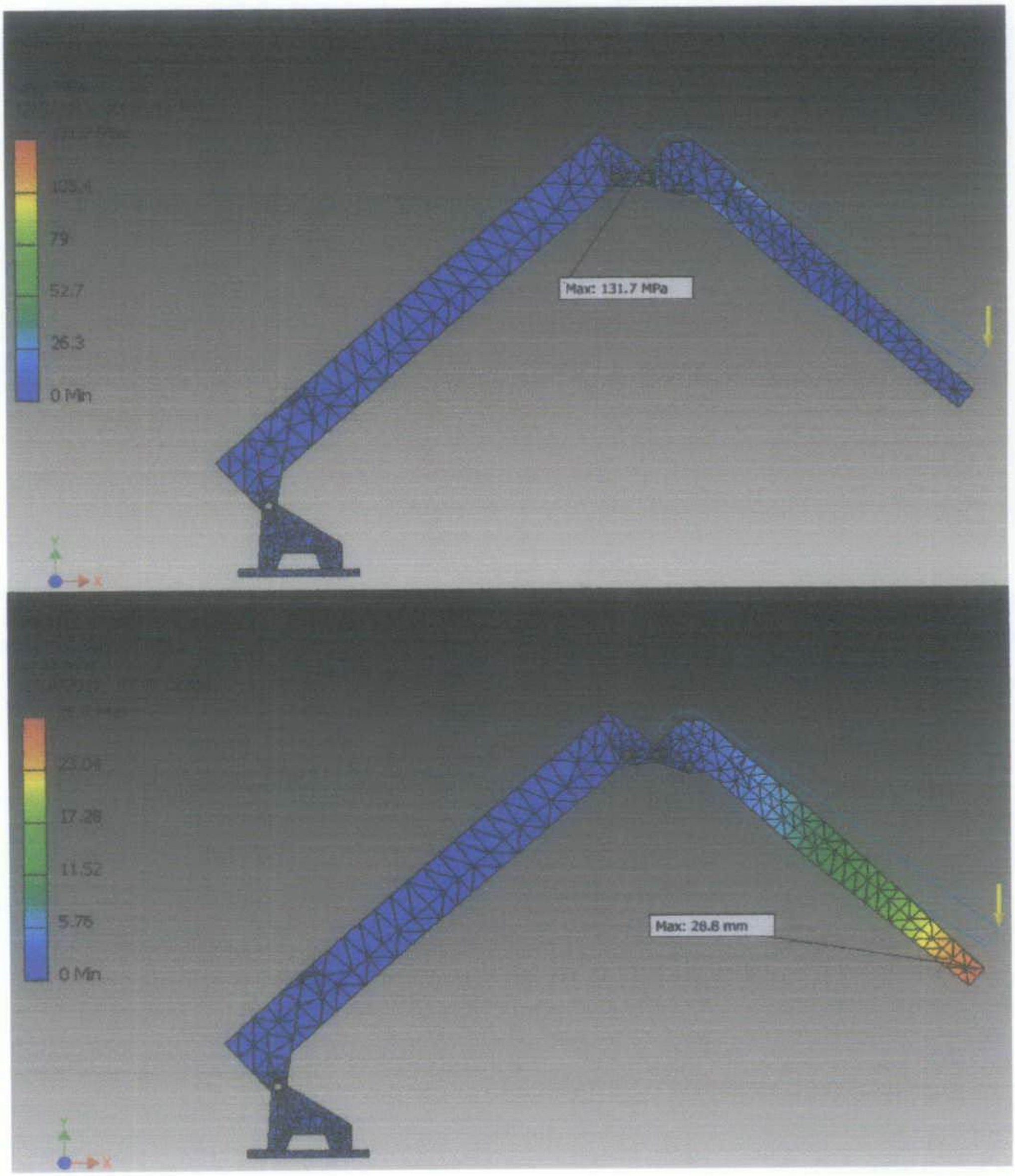


Figure 4.5: Maximum stress and deformation under working condition 1

Working Condition 2

i. Main boom angle: 51^{o}

ii. Second boom angle: 78^{o}

iii. Load: 7257 kg

Figure 4.6 shows the result of the finite element analysis using the Autodesk software for the working condition 2. Based on the result, it is found out that he maximum stress is 121.2 Mpa which is still below the maximum allowable stress 137.9 Mpa. This result again proved that the crane design is suitable for the assigned lifting weight which is 7257 kg. The maximum stress is found out to be at the assembly point between the booms. This proved that the pin joint which is connecting the crane is the most crucial location that should be well considered in crane design. The location can also be assumed as the limitation of the crane in lifting capability. The deformation of the crane is 23.25 mm.

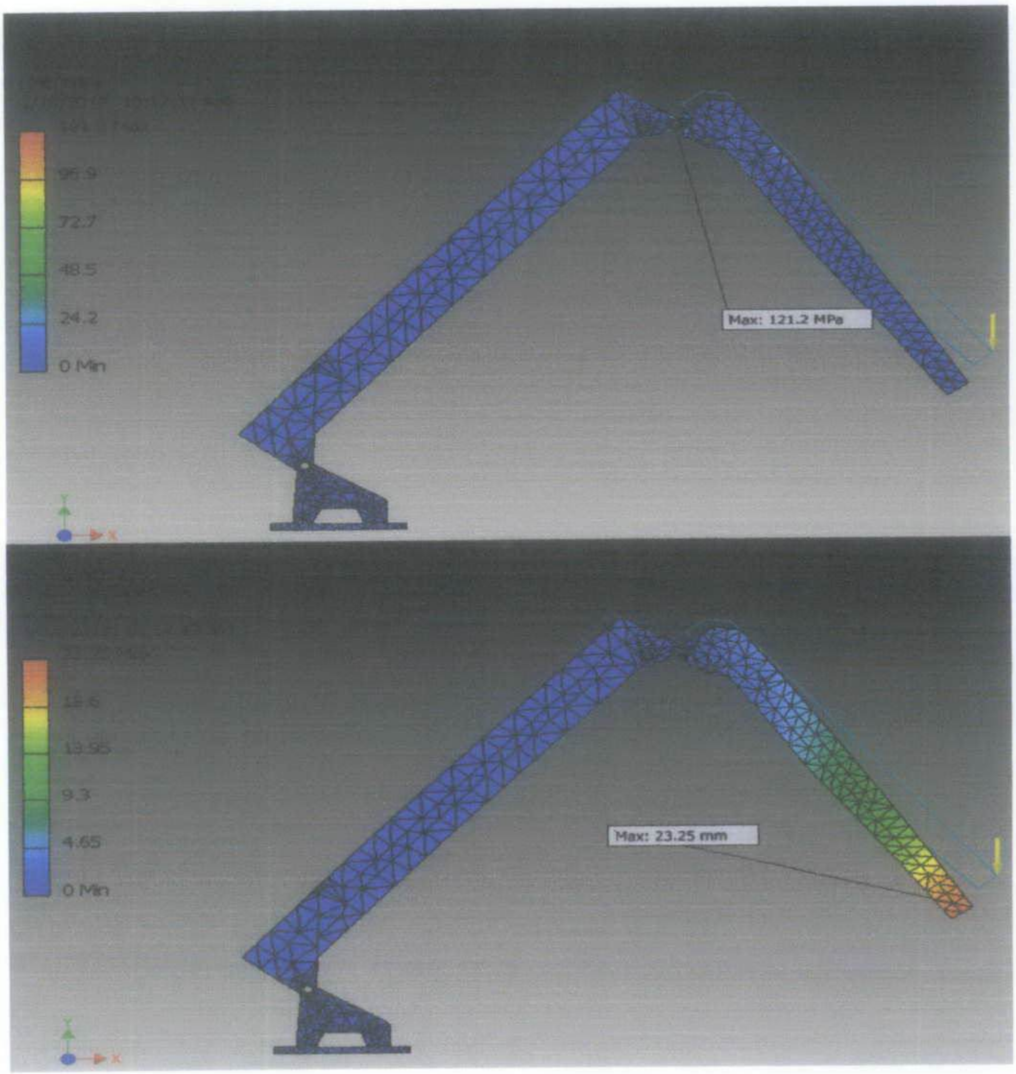


Figure 4.6: Maximum stress and deformation under working condition 2

Working Condition 3

i. Main boom angle: 60^{o}

ii. Second boom angle: 60^{o}

iii. Load: 7257 kg

Figure 4.7 shows the analysis result for both stress and deformation using Autodesk Inventor 2012. Referring to the figure, maximum stress is 113.8 Mpa which is under the maximum allowable stress. Thus, the design is proven to be able to endure the weight that is stated in the load chart. For the deformation result, 21.67 mm is the maximum deformation indicated after the analysis. Maximum deformation is shown to be at the end of the second boom where the load is applied. Maximum deformation which is indicated red shown that the boom is forced to bend downward opposite with the applied force. However, from the result, it is found out if the load is increased higher, the location where the crane will initially break will be at the pin joint connecting both booms.

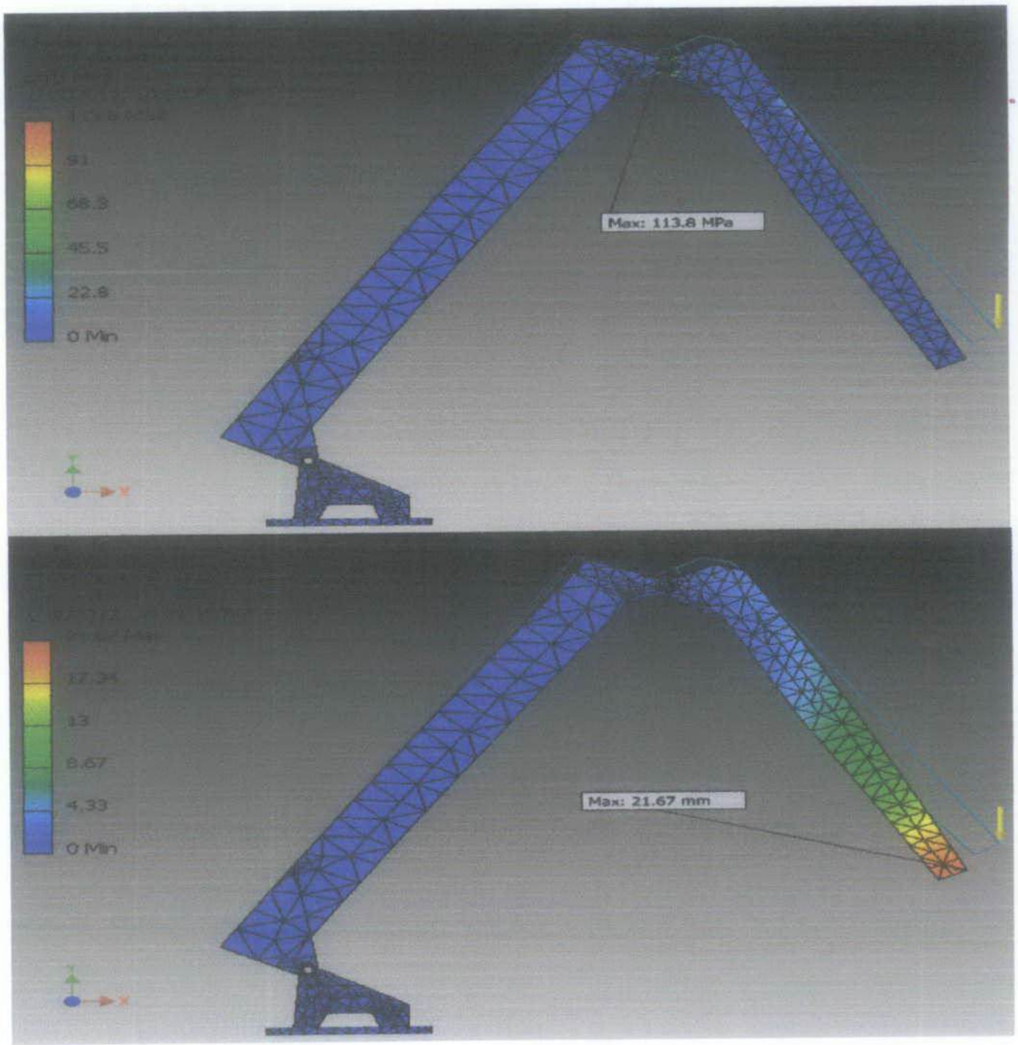


Figure 4.7: Maximum stress and deformation under working condition 3

Working Condition 4

i. Main boom angle: 68°

ii. Second boom angle: 43°

iii. Load: 7257 kg

The analysis shown in figure 4.8 is the analysis for the crane under loading load of 7257 Kg, main boom angle 68° and second boom angle 43°. The maximum stress which is shown in the analysis is 94.79 Mpa which is still below the maximum allowable stress that is calculated, 137.9 Mpa. This indicates that the design of crane is workable as it is able to endure the amount of force that is stated in the load chart. The deformation of the crane is 17.48 mm. As expected, the deformation is mostly on the second boom due to the location of the applied load.

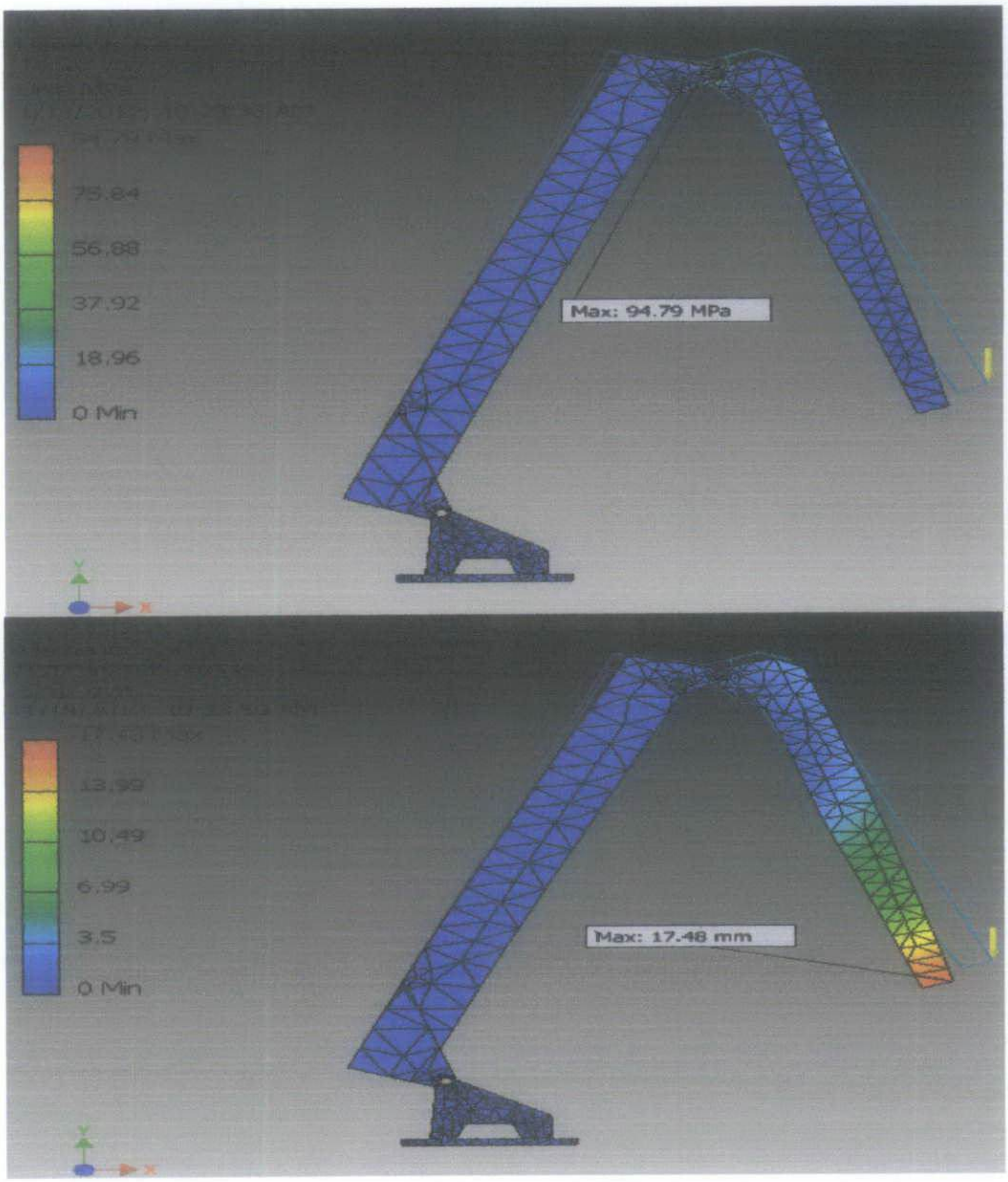


Figure 4.8: Maximum stress and deformation under working condition 4

Working Condition 5

i. Main boom angle: 76°

ii. Second boom angle: 26°

iii. Load: 7257 kg

Figure 4.9 shows the result of the finite element analysis using the Autodesk software for the working condition 2. Based on the result, it is found out that he maximum stress is 118.6 Mpa which is still below the maximum allowable stress 137.9 Mpa. This result again proved that the crane design is suitable for the assigned lifting weight which is 7257 kg. The maximum stress is found out to be at the assembly point between the booms. This proved that the pin joint which is connecting the crane is the most crucial location that should be well considered in crane design. The location can also be assumed as the limitation of the crane in lifting capability. The deformation of the crane is 20.75 mm.

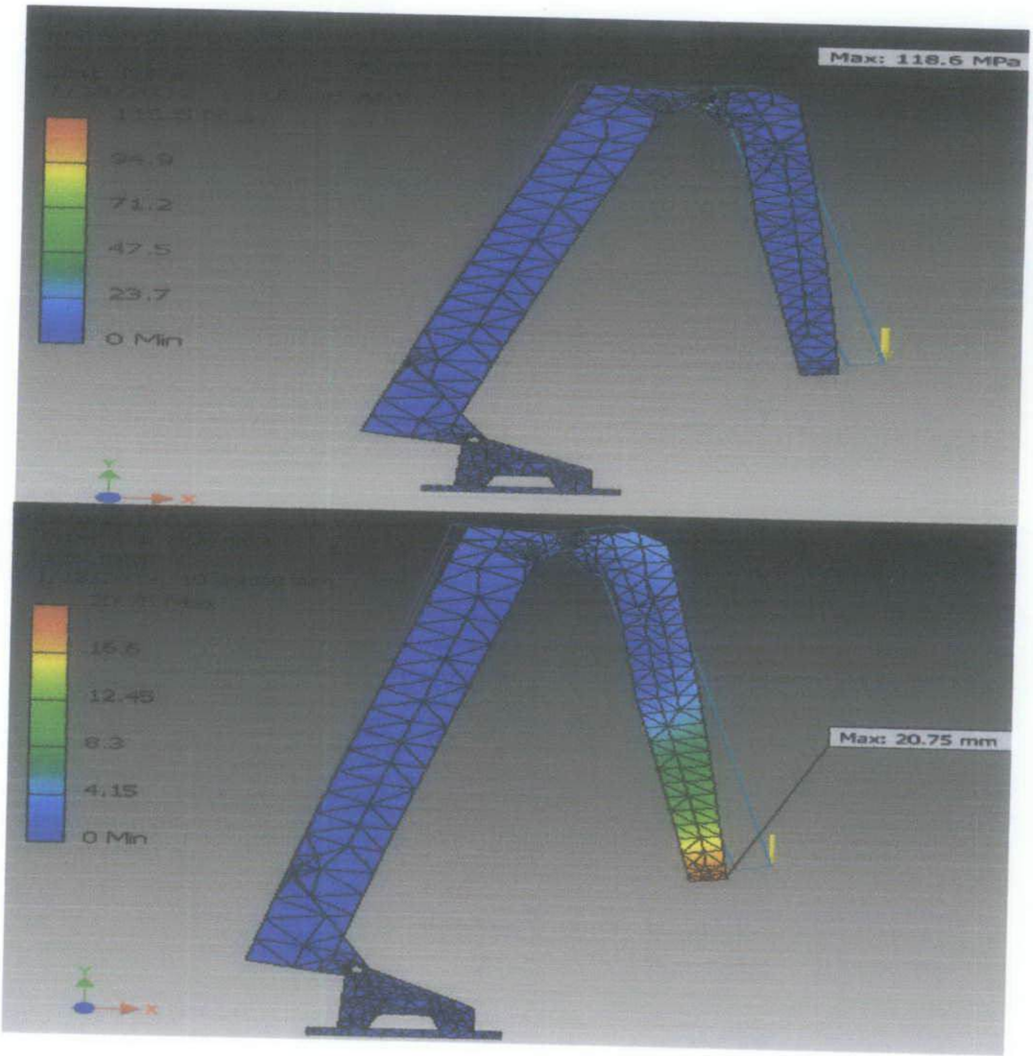


Figure 4.9: Maximum stress and deformation under working condition 5

Table 4.2: Summary of finite element analysis of knuckle crane boom

MATERIAL	MAIN BOOM ANGLE	SECOND BOOM ANGLE	MAX. CRANE REACH	MAX. VON MISES STRESS (Mpa)	MAX. DEFORMATION (mm)
HIGH STRENGTH LOW ALLOY STEEL	76	26	10	118.6	20.75
	68	43	15	94.79	17.48
	60	60	20	113.8	20.75
	51	78	25	121.2	23.25
	41	100	30	113.4	23.47
	29	127	35	128.9	27.12
	0	180	40	99.89	105.6

4.3 Finite Element Analysis for Alternative Material

Stress analysis is done for all alternative material to study the effect of the material on the deformation of the knuckle crane boom. The constraints, angles and load are all made to be similar as the previous stress analysis which is on the current material.

Titanium is a one of the low density metal but compared to the aluminium, titanium has higher strength and better corrosion-resistant. Referring to the table 14, titanium is the lightest material with yield strength nearly similar as the high-strength low-alloy steel. However, compared to the steel, titanium has the highest poisson ratio which is a determining factor in the deformation of a material. Common poisson ratio for steel is 0.3 and titanium has 0.361. Second weakness of titanium is that the material has lowest young modulus which means that it is the less stiff material among the four selected material. Thus, we can conclude that the titanium will result in higher deformation compared to other material. Table 4.3 shows the maximum deformation as the load is applied to the model. The deformation is the highest among the material and thus the result is in correlation with the data.

Table 4.3: Summary of finite element analysis for titanium

	MAIN BOOM ANGLE	SECOND BOOM ANGLE	MAXIMUM DEFORMATION (mm)
TITANIUM	76	26	13.11
	68	43	20.18
	60	60	29.25
	51	78	37.32
	41	100	45.46
	29	127	53.05
	0	180	102.25

Ductile iron as the name suggests is a ductile material which has 332 Mpa as yield strength and 7.1 g/cm^3 density. Both properties is better that the current material which is 275.8 Mpa for yield strength and 7.84 g/cm^3 for density. Due to the lighter weight of the ductile iron, it is expected that the crane capability will increase and thus higher load can be lifted than usual. According to the yield strength, it also indicated that ductile iron can endure more load compared to current material. Using the safety factor as 2, maximum allowable stress will be 166 Mpa which is 28.1Mpa higher than current material. However, same condition with titanium, ductile iron has lower young modulus and higher poisson ratio compared to the high-density low-alloy steel. This will result in higher deformation compared to the current material. Table 4.4 show the maximum deformation from the finite element analysis.

Table 4.4: Summary of finite element analysis for ductile iron

	MAIN BOOM ANGLE	SECOND BOOM ANGLE	MAXIMUM DEFORMATION (mm)
DUCTILE IRON	76	26	8.06
	68	43	12.41
	60	60	17.93
	51	78	22.87
	41	100	27.935
	29	127	32.69
	0	180	62.85

Carbon steel is second best among the four material due to its properties which has similar young modulus, lower poisson ratio value, higher yield strength and higher ultimate tensile strength compared to the high-strength low-alloy steel. From this, we can already conclude that carbon steel will sure have less maximum deformation compared to high-strength low-alloy steel. However, weakness of carbon steel is quite problematic in crane design as it density is quite high and thus making the total density of the crane increase. This will affect the maximum load that can be lifted by the crane as the overall crane weight is already high compared to the current material. Table 4.5 shows summary of finite element analysis for carbon steel.

Table 4.5: Summary of finite element analysis for carbon steel

	MAIN BOOM ANGLE	SECOND BOOM ANGLE	MAXIMUM DEFORMATION (mm)
CARBON STEEL	76	26	6.77
	68	43	10.425
	60	60	15.06
	51	78	19.21
	41	100	23.465
	29	127	27.46
	0	180	52.8

Stainless steel 440c is so far the best material that can be used in crane manufacturing. The steel has less weight compared to the current material, higher young modulus, less poisson ratio, highest yield strength and highest ultimate tensile strength. Due to all this advantages, it can be concluded that stainless steel 440c is so far the best material that can be used in crane manufacturing. Using the equation for maximum allowable stress with safety factor 2, it is found that the maximum stress the crane can endure with stainless steel 440c as material would be 344.5 Mpa. This will increase the load lifting capability of the crane and thus its efficiency. Table 4.6 below is the summary of the analysis for the stainless steel 440c.

Table 4.6: Summary of finite element analysis for stainless steel 440c

	MAIN BOOM ANGLE	SECOND BOOM ANGLE	MAXIMUM DEFORMATION (mm)
STAINLESS STEEL 440C	76	26	6.555
	68	43	10.09
	60	60	14.57
	51	78	18.58
	41	100	22.715
	29	127	26.59
	0	180	51.15

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 CONCLUSION

Stress analysis on the knuckle crane boom based on the load chart proves that the highest concentrated stress is at the assembly point where the area is small enough to develop high stress. Second reason for the stress distribution to behave as shown is due to the existed joint, pin joint, between the assembly points. Thus, from the data shown, it can be concluded that the strength of the joint is more as the contributed factor to the limitation of the crane lifting capability compared to the boom itself.

Among four materials that been studied, it is proven that stainless steel 440c is the best material for boom manufacturing. With high yield strength and lots more contributed advantages; the steel is believed to be one of the solutions for high end crane product. However, the process of implementation the material in current world might have some difficulty in term of cost and manufacturing ability.

5.2 RECOMMENDATION

As a recommendation, I would suggest that the crane manufacturer will focused more on the assembly point of the knuckle crane boom. This is due to the fact that the maximum stress location is at that point and thus becoming a huge contributor to the limitation of the crane capability in lifting weight. One of the solution that can be made is that to change the material only at the assembly point of the crane into stainless steel 440c which by far the best material that can be used in crane manufacturing process.

Knuckle crane technology is in constant evolution, and the industry is always looking for new designs and new technology. Understanding and mastering the behaviour of a knuckle crane is not an easy task and requires more fundamental and applied research.

REFERENCES

[1] Dae-Suk Han, Sung-Won Yoo, Hyun-Sik Yoon, Myung-Hyun Kim, Sang-Hyun Kim, Jae-Myung Lee, Coupling Analysis of finite element and finite volume method for the design and construction of FSPO crane, 2010

[2] Betram J. Leigh, Boom Structure, assignor to Telsta Corporation, April 17, 1967, Series Number. 631,206

[3] Joseph B. Tiffin, Daniel G. Quinn, Evart J. Vroonland, all of Cedar Rapids, Iowa, Extensible crane boom structure, May 27, 1975

[4] Barna Aladar Szabo, Ivo Babuska, Finite Element Analysis, Wiley -IEEE, 1991

[5] Donald C. Markwardt, Manitowoe, Wis., The Manitowoe Company, September 18, 1967

[6] Per Olog Oberg, Sanera Projecting Aktiebolag, Floatable Boom Structure, Mar 31, 1971

[7] Elsayed Mashaly, Mohamed El-Heweity. Handy Abou-Elfath, Mohamed Osman, Alexandria University, Alexandria, Egypt, Finite element analysis of beam-to-column joints in steel frames under cyclic loading, 2010

[8] Jaeger, John Conrad; Cook, N.G.W, & Zimmerman, R.W. (2007). "Fundamentals of rock mechanics (Fourth ed.)" Wiley-Blackwell. pp. 9–41. ISBN 0632057599

[9] R.C. Hibbeler, 2008, "Mechanics of Materials (7th Edition)", Pearson Education South Asia PTE LTD.

[10] C. Miriţoiu, D. Ilincioiu, 2011, "A Stress Analysis of a Metallic Structure using Finite Element Method", World Academy of Science, Engineering and Technology 2011

[11] Saeed Moaveni, 2008, "Finite Element Analysis: Theory and Application with ANSYS", Prentice Hall.

[12] Min-Lun Hsu, Chih-Ling Chang, "Application of Finite Element Analysis in Dentistry", National Yang-Ming University, Taipei, Taiwan