ANALYSIS OF CONNECTING ROD FRACTURE USING FINITE ELEMENT ANALYSIS

AHMAD RIDZUAN BIN IBRAHIM

BACHELOR OF ENGINEERING
UNIVERSITI MALAYSIA PAHANG

2010

ANALYSIS OF CONNECTING ROD FRACTURE USING FINITE ELEMENT ANALYSIS

AHMAD RIDZUAN BIN IBRAHIM

Thesis submitted in fulfilment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

DECEMBER 2010

UNIVERSITI MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

We certify that the project entitled "*Analysis of Connecting Rod Fracture Using Finite Element Analysis*" is written by *Ahmad Ridzuan bin Ibrahim*. We have examined the final copy of this project and in our opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. We herewith recommend that it be accepted in partial fulfilment of the requirements for the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

MR. MOHD AKRAMIN MOHD ROMLAY

Examiner Signature

SUPERVISOR DECLARATION

I hereby declare that I have read this project report and in my opinion this project report is sufficient in terms of scope and quality for the award of the Bachelor in Mechanical Engineering with Automotive Engineering

Signature	:
Name of Supervisor	: MRS. NORHAIDA BINTI AB RAZAK
Position	: LECTURER
Date	: 06 DECEMBER 2010

STUDENT'S DECLARATION

I hereby declare that the work in this project is my own except for quotations and summaries which have been duly acknowledged. The project has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature :

Name : AHMAD RIDZUAN BIN IBRAHIM

ID Number : MH08013

Date : 06 DECEMBER 2010

DEDICATION

First of all, I would like to show my expression of gratitude to Allah S.W.T whose guidance, help and grace was instrumental in making this humble work a reality. This dedication goes to my beloved mother Zabidah bt Hamzah and my beloved father Ibrahim bin Mamat. Also thank to my supervisor Mrs. Norhaida binti Ab Razak that graves me motivation and guide me to finish this project. Thanks a lot to my university (Universiti Malaysia Pahang) and my friends in their support and advice towards this project. Thanks to all for your enduring patience and continuous encouragement.

ANKNOWLEDGEMENTS

I am grateful and would like to express my sincere gratitude to my supervisor Mrs. Norhaida binti Ab Razak for her germinal ideas, invaluable guidance, continuous encouragement and constant support in making this research possible. She has always impressed me with his outstanding professional conduct, his strong conviction for science. I appreciate his consistent support from the first day I applied to graduate program to these concluding moments. I am truly grateful for her progressive vision about my training in science, his tolerance of my naive mistakes, and his commitment to my future career. I also sincerely thanks for the time spent proofreading and correcting my many mistakes.

My sincere thanks go to all my classmates and members of the staff of the Mechanical Engineering Department, UMP, who helped me in many ways and made my stay at UMP pleasant and unforgettable.

I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life. I acknowledge the sincerity of my parents-in-law, who consistently encouraged me to carry on my higher studies in Malaysia. I am also grateful to my family for their sacrifice, patience, and understanding that were inevitable to make this work possible. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to attain my goals. Special thanks should be given to my committee members. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this study.

ABSTRACT

This thesis presents an analysis of connecting rod fracture using Finite Element Analysis. The connecting rod is one important part in an engine. In this task, connecting rod of the Proton Wira 1.5 is used. After measurements were taken, connecting rods were drawn using SolidWorks software and save in 'IGES' format. Then, the drawing of connecting rod (IGES format) imported into ALGOR software. In this software, can makes analysis to connecting rod and thus to know the stress and strain that occur on the connecting rod. This analysis uses a different mesh of 100% and 80%. It aims to get more precise results. The smaller the percentage of mesh used the more accurate results will be. Failure of connecting rod occurs depend on compared value stress von mises and yield strength of materials. Failure occurs if value stress von mises higher compared yield strength of material. The result shows failure of connecting rod occurs while applied load at 26kN if used steel as material and 24kN if used aluminium alloy. The entire project involves various processes such as process design and analysis.

ABSTRAK

Analisis rod penghubung patah menggunakan Finite Element Analysis. Idea ini telah dikeluarkan oleh penyelia yang memberikan tugasan. Dalam tugasan ini, rod penghubung daripada kereta Proton Wira 1.5 digunakan. Selepas pengukuran diambil, rod penghubung dilukis menggunakan perisisan SolidWorks dan disimpan dalam bentuk ‘IGES’. Kemudian, lukisan rod penghubung (bentuk IGES) diimport ke dalam perisian ALGOR. Dalam perisian ini,dapat melakukan analisis ke atas rod penghubung dan seterusnya dapat mengetahui tekanan dan ketegangan yang terjadi pada rod penghubung tersebut. Analisis ini menggunakan mesh yang berbeza-beza iaitu 100 % dan 80%. Ini bertujuan untuk mendapatkan keputusan yang lebih tepat. Semakin kecil peratus mesh yang digunakan semakin tepat keputusan yang diperolehi. Kepatahan rod penghubung terjadi bergantung pada nilai tekanan von mises dibandingkan dengan hasil kekuatan bahan. Kepatahan terjadi jika nilai tekanan von mises lebih rendah berbanding nilai hasil kekuatan bahan yang digunakan. Keputusan kajian menunjukkan kepatahan rod penghubung terjadi ketika diterapkan beban di 26kN jika menggunakan steel sebagai bahan dan 24kN jika menggunakan aluminium sebagai bahan. Keseluruhan projek melibatkan pelbagai proses seperti proses merekabentuk dan juga analisis.

TABLE OF CONTENTS

LIST OF TABLE

LIST OF FIGURES

LIST OF SIMBOLS

D	Material Property
E	Modulus of Elasticity (Young's Modulus)
G	Shear Modulus
p_c	Tensile resultant load in the direction of the rod length
p_o	Normal Pressure
p_t	Compressive resultant load in the direction of the rod length
r	Radius of crank end
t	Width of crank end
σ	Stress
ε	Strain
ν	Poisson's Ratio

LIST OF APENDICES

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

Connecting rod is an important component in an engine. Connecting rod used to connect between piston and crankshaft. The purpose of the project was to analysis connecting rod fracture using Finite Element Analysis. The stress and thermal for connecting rod can analyzed with this software. With this software, can analysis the stress and thermal for connecting rod. Before that, draw the connecting rod using SolidWorks software, then exported to the finite element analysis (ALGOR software). This project focused on analysis. Overall, this project will acquire the analysis of fracture connecting rod. Figure 1.0 shows the breakdown that occurs in the connecting rod.

Figure 1.0: Failure of a connecting rod

Source: http://automobilelargestate.blogspot.com/2010/09/connecting-rod.html

1.2 PROBLEM STATEMENT

Connecting rod used to connect between the piston rod with crankshaft. Connecting rod is an important component in the engine. Failure at the connecting rod causes the engine cannot be functioned. This conditions not safe and can cause accidents while driving.

1.3 PROJECT OBJECTIVE

The objectives of this project are to:

- Developed a geometrical model for connecting rod using SolidWorks software.
- Investigate the stress analysis of steel and aluminum alloy of connecting rod using ALGOR software.
- Investigate the maximum stress of connecting rod using ALGOR software.

1.4 PROJECT SCOPE

The computational stress from finite element analysis will be carried out on a Proton Wira 1.5 connecting rod design. The two different materials are chosen for analysis is steel (AISI 4130) and aluminum alloy. The geometry model for the connecting rod had drawn using solid work software. The analysis was running using ALGOR software. The apply load for this analysis is at 90□ only.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

Connecting rod connects the piston to the crank or crankshaft. Together with the crank, form a simple mechanism that converts linear motion into rotating motion. Connecting rods may also convert rotating motion into linear motion. As a connecting rod is rigid, it may transmit either a push or a pull and so the rod may rotate the crank through both halves of a revolution, such as piston pushing and piston pulling. Connecting rods are best known through their use in internal combustion engines such as car engines.

The connecting rod is under tremendous stress from the reciprocating load represented by the piston, actually stretching and being compressed with every rotation, and the load increases to the third power with increasing engine speed. Failure of a connecting rod, usually called ‘throwing a rod’ is one of the most common causes of catastrophic engine failure in cars, frequently putting the broken rod through the side of the crankcase and thereby rendering the engine irreparable; it can result from fatigue near a physical defect in the rod, lubrication failure in a bearing due to faulty maintenance, or from failure of the rod bolts from a defect, improper tightening, or re-use of already used (stressed) bolts where not recommended (Nunney M.J. 2007).

Figure 2.1 shows the role of connecting rod from reciprocating motion into rotary motion. A four-stroke engine is the most common type. The four strokes are intake, compression, power, and exhaust. Each stroke requires approximately 180 degrees of crankshaft rotation, so the complete cycle would take 720 degrees. Each stroke plays a very important role in the combustion process as shown in Figure 2.1. In the intake cycle, while the piston moves downward, one of the valves open. This creates a vacuum, and an air-fuel mixture is sucked into the chamber, as shows Figure 2.1 (a) (tensile load). During the second stroke compression occurs. In compression both valves are closed, and the piston moves upward and thus creates a pressure on the piston. Indirectly, there is pressure on the connecting rod, as shows Figure 2.1 (b) (compress load). The next stroke is power. During this process the compressed air-fuel mixture is ignited with a spark, causing a tremendous pressure as the fuel burns. The forces exerted by piston transmitted through the connecting rod moves the crankshaft, as shows Figure 2.1 (c) (tensile load). Finally, the exhaust stroke occurs. In this stroke, the exhaust valve opens, as the piston moves back upwards, it forces all the air out of the chamber and thus which completes the cycle of crankshaft rotation, as shows Figure 2.1 (d) (compress load).

Connecting rod produced from alloy steel forging or aluminum alloy were used to a limited extend in early practice, the primary object being to reduce the loading on the big end and main bearing (Nunney M.J. 2007). Connecting rods used these days are the result of advancements in metallurgy and machining techniques. These rods are the critical part of an engine because even a small defect in it can lead to catastrophic engine failure in automobiles. Most of the automobile manufacturers are engaged in making superior and high performance connecting rods at economical prices. Figure 2.2 shows schematic of a typical connecting rod.

Roman Celin (2008) analyses the frequency of alternating loading increase rapidly with an increase in the engine RPM. In many cases a engine failure is caused by a connecting rod failure and sometimes the broken connecting rod shank may even be pushed through the side of the crank case. There are many reasons for such failure, such as the overheating of the engine, cracking, deficiency of the bearing lubrication, poor maintenance.

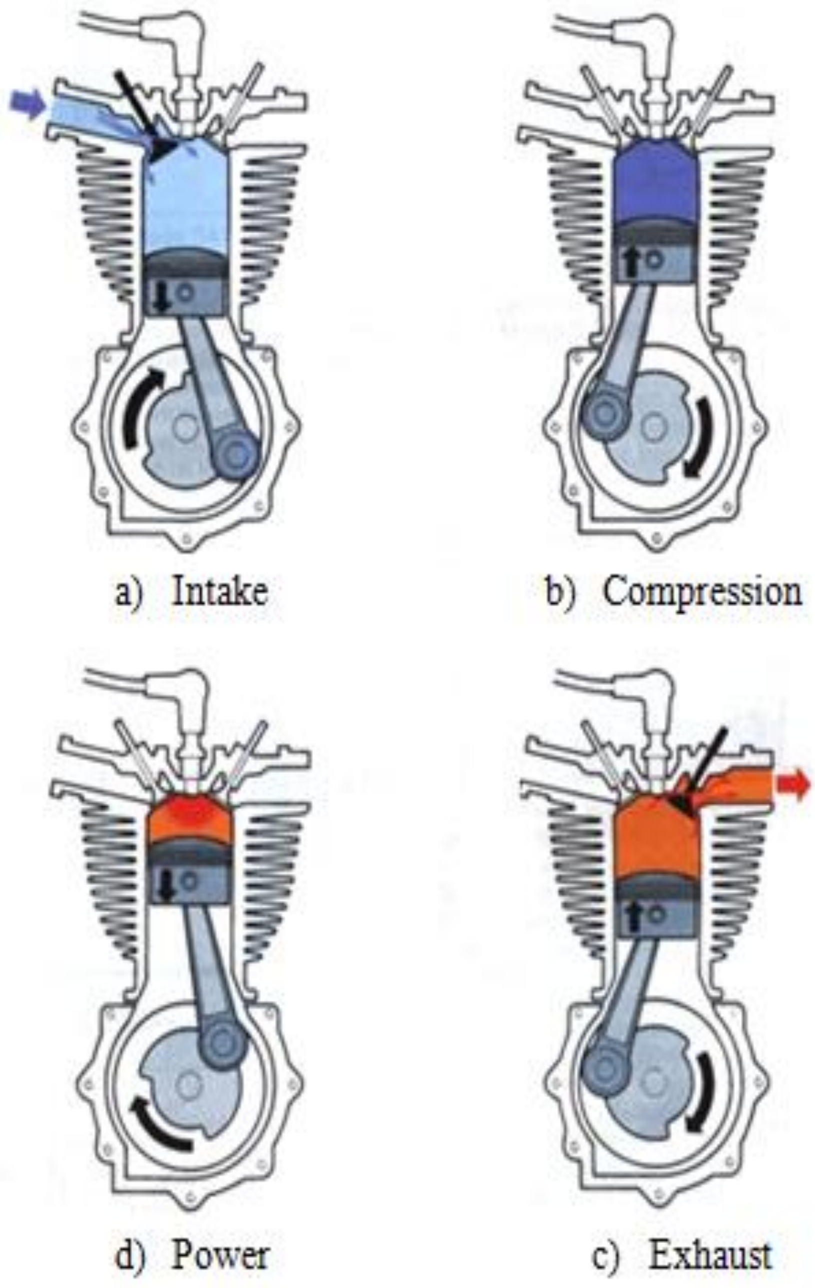


Figure 2.1: The four stroke in an engine

Source: http://www.whitedoglubes.com/4strokeengines.htm

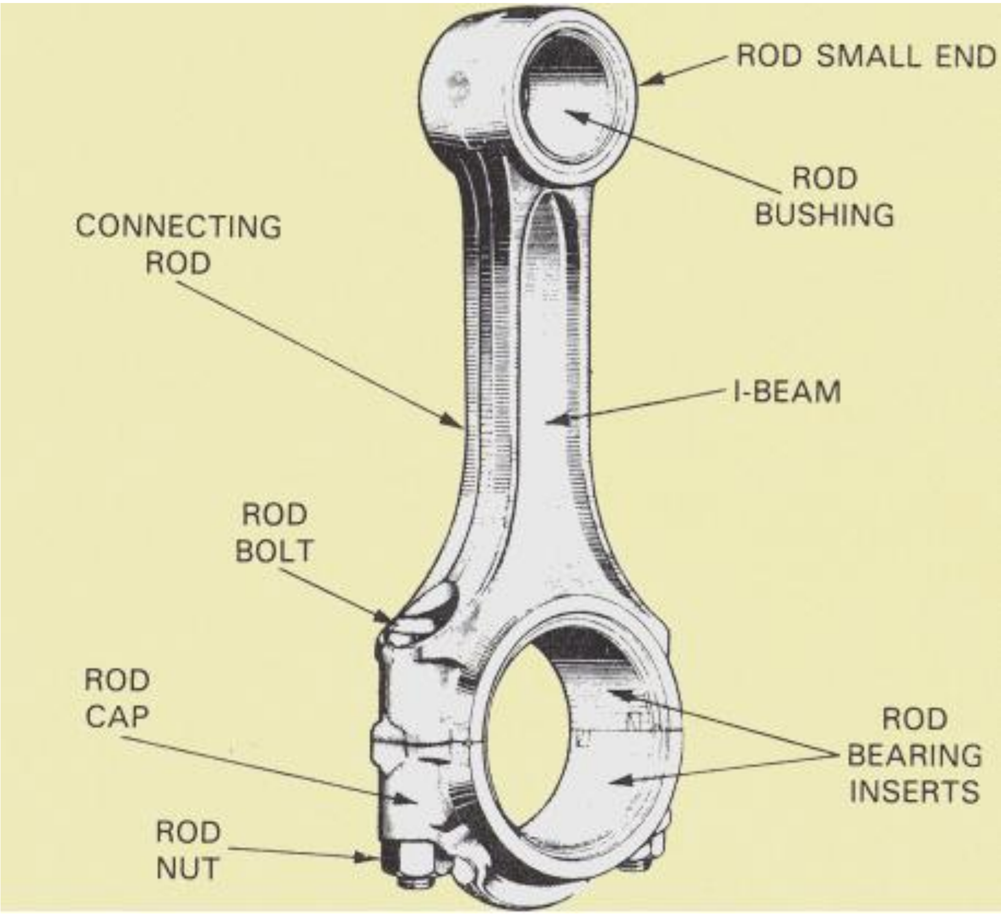


Figure 2.2: Connecting rod

Source: Luke Schreier (1999)

2.2 LOAD AND STRESS ANALYSIS

Sonsino (1996) analyzed the loading experienced by connecting rods. Connecting rods are submitted to mass and gas forces. The superposition of these two forces produces the axial force, which acts on the connecting rod. Connecting rods also experience bending moments due to eccentricities, crankshaft and rotational mass forces. Axial loading can be calculated by the knowledge of engine pressure and rotational speed, whereas bending moment can be determine by strain analysis in an engine. Stress analysis of critical areas and maximum local stresses of powder-forged connecting rods were also discussed by Sonsino (1996). The critical areas are the transition regions from the small end to the shank and from the shank to the big end.

Table 2.1 below shows the local stress amplitudes obtained for the critical areas. The R ratios were calculated for the most severe conditions with a gas pressure of p = 660 kPa, rotational speed of 6800 rpm, and an operational load amplitude of Fa = 19.2 kN. Because of mass distribution, different mean loads, and therefore R-ratios, for particular areas of the connecting rod were obtained, as listed in Table 2.1. The magnitude of piston side load forces and dynamics errors for the two-lump mass connecting rod model and actual rod model were discussed by Shiao and Moskwa (1993).

Table 2.1: The local stress amplitude obtained for the connecting rod critical areas

Area	**Kt**	**Maximum local stress amplitude**		**Local endurable stress amplitude in MPa, P_s =50%, N_E=2.10^6**		**Local allowable stress amplitude σ_a(R)/ jσ in MPa, P_f=10^{-5}**	
		σ_a in MPa	**R**	**σ_a (R= -1)**	**σ_a (R)**	**Jσ = 1.6**	**Jσ = 2.0**
Transition small end to stem	1.36	129	-1.80	238	273	171	137
Stem	1.00	95	-1.26	222	236	148	118
Transition stem to big end	1.10	104	-1.26	232	246	154	123
Oil bore	2.8	120	-1.26	286	303	189	152
Transition	1.07	102	-0.42	228	193	121	97

Material: Fe =1.5% Cu=0.6, R = 7.10g/cm^3, quenched and tempered

Source: Sonsino, (1996)

They concluded that an error in the moment of inertia of connecting rod using this model could predict incorrect engine forces and dynamics. With the actual connecting rod model, the inertia forces of the distributed mass model become more complicated to analyze. The small end of the rod has a reciprocating motion along the bore, whereas, the big end of the rod moves in rotational motion. However, the center of rod mass path describes an ellipse, which makes it difficult to analyze. Therefore, a two lump mass connecting rod model was introduced for simplification. The connecting rod is divided into reciprocating and rotating masses so that the distributed connecting rod mass is replaced by lumped masses at each end located at the bore center.

For a single cylinder engine, there are three moving parts for transmitting engine force: piston system, connecting rod system, and crankshaft system. 26.7 kN axial load applied at the crank end. In the analysis carried out the failure of connecting rod (Pravardhan S. Shenoy, 2004).

2.3 MATERIAL

The connecting rods are most usually made of steel or aluminum alloy for production engines. These materials have different properties and suitable for different engines (Khatiblou M.A. 1994).

2.3.1 Steel (AISI 4130)

Steel is an alloy that consists mostly of iron and has carbon content between 0.2% and 2.1% by weight, depending on the grade. Carbon is the most common alloying material for iron, but various other alloying elements are used, such as manganese, chromium, vanadium, and tungsten. Carbon and other elements act as a hardening agent, preventing dislocations in the iron atom crystal lattice from sliding past one another. Varying the amount of alloying elements and the form of their presence in the steel (solute elements, precipitated phase) controls qualities such as the hardness, ductility, and tensile strength of the resulting steel. Steel with increased carbon content can be made harder and stronger than iron, but such steel is also less ductile than iron.

AISI 4130 is one of steel used to manufacture connecting rod (eFunda, 2010). In the normalized state the maximum allowable tension stress is 95 ksi. At this value 4130 has good toughness and excellent elongation. Table 2.1 shows the mechanical properties of AISI 4130.

2.3.2 Aluminum Alloy

Aluminium alloys are alloys in which aluminium (Al) is the predominant metal. Aluminium alloy surfaces will keep their apparent shine in a dry environment due to the formation of a clear, protective layer of aluminum oxide. In a wet environment, galvanic corrosion can occur when an aluminium alloy is placed in electrical contact with other metals with more negative corrosion potentials than aluminium. Aluminium alloys are widely used in automotive engines, particularly in cylinder blocks and crankcases due to the weight savings that are possible. Since aluminium alloys are susceptible to warping at elevated temperatures, the cooling system of such engines is critical. Manufacturing techniques and metallurgical advancements have also been instrumental for the successful application in automotive engines. In the 1960s, the aluminium cylinder heads and crankcase of the Corvair earned a reputation for failure and stripping of threads, which is not seen in current aluminium cylinder heads. Table 2.3 shows the mechanical properties of aluminum alloy.

Table 2.2: Mechanical properties of steel AISI 4130

Property	**Value in metric unit**	
Density	$7.872\text{\^{}}10^3$	kg/m^3
Tensile strength (annealed)	561	N/mm^2
Yield strength (annealed)	361	N/mm^2
Elongation (annealed)	28.2	%

Source: eFunda (2010)

Table 2.3: Mechanical properties of aluminum alloy

Property	Value in metric unit	
Density	2.7^10³	kg/m³
Tensile strength (annealed)	400	N/mm^2
Yield strength (annealed)	340	N/mm^2
Elongation (annealed)	18	%

Source: eFunda (2010)

2.4 SOFTWARE

2.4.1 Finite Element Analysis

Finite Element Analysis or FEA has found many uses in mechanical engineering but has also found applications in electrical, civil, and chemical engineering. It is a numerical method for solving engineering problem and physics. It also a method to computationally reality model in a mathematical form.

Advantages of FEA

- Irregular geometries
- General loading
- It is capable of modeling and analyzing:
- Different material properties
- Various BCs
- Variable element types and size
- Easy modification
- Nonlinear and dynamics

2D FE models can be used to obtain rapid trend statements, and 3D FE models for more accurate investigation. The various individual loads acting on the connecting rod were used for performing simulation and actual stress distribution was obtained by superposition. The loads included inertia load, firing load, the press fit of the bearing shell, and the bolt forces (Balasubramaniam *et al*. 1991)

Modelled the inertia load in their finite element model interface software was developed to apply the acceleration load to elements on the connecting rod depending upon their location, since acceleration varies in magnitude and direction with location on the connecting rod. They fixed the ends of the connecting rod, to determine the deflection and stresses. This, however, may not be representative of the pin joints that exist in the connecting rod. The results of the detailed analysis were not discussed, rather, only the modeling technique was discussed. The connecting rod was separately analyzed for the tensile load due to the piston assembly mass (piston inertia), and for the compressive load due to the gas pressure (Athavale and Sajanpawar 1991). ALGOR software used for find the stress in this analysis.

In general, FEA consists of three main stages: preprocessing, processing and post-processing. Preprocessing consist of modeling the practical problem to be solved and include such aspect as creating the geometry, entering the input data for example material properties and meshing the model. Processing in the main stage and provides the numerical solution of the equation that represent the relationship among the stress, strain and other variables and produces the value of the fundamental variables. Post-processing in the stage , where the result are plotted and interpreted and additional variable calculated.

2.4.1.1 Algor

ALGOR is a complete FEA solution that offers a good combination of cost effectiveness, quality and features within ALGOR FEMPRO, an easy-to-use interface. ALGOR provides all the necessary features for directly capturing 3-D solid geometry from Autodesk Inventor, generating a high-quality solid FEA mesh, easily setting up loads and constraints, performing analyses quickly, evaluating results and presenting a final design.

Tension and compression loads were applied as pressure on the bearing surfaces of the connecting rod. Webster et al. (1983) found that under actual service conditions, the pin end experiences tension by the piston pin causing distribution of pressure along the upper half of the inner diameter (over 180^o of the contact area), which is approximated by the cosine function. In compression, the piston pin compresses the bearings against the pin end inner diameter (over 120^o contact arc), causing uniform distribution of pressure. The pressure acts normal to the contact surface area in a symmetric manner. Therefore, four cases were analyzed, two for tension loading (cosine pressure distribution over 180^o) and two for compression loading (uniform pressure distribution over 120^o), each with either loading the pin end while constraining the crank end, or loading the crank end while constraining the pin end. The constraints were applied for all six degrees of freedom over 180^o for the tension loading cases, and over 120^o for the compression loading cases. Figure 2.4 shows the boundary conditions used for the FEA analysis for the cases of tension as well as compression loading of the crank end, while the pin end is constrained.

2.4.2 SolidWorks

SolidWorks is a tool that helps design engineers harness their imagination and add creativity to their designs. There is no need to start from scratch to transition a 2D drawing to 3D model.

In assembly modeling, SolidWorks provides the tool to ‘get it right’ on screen to create assembly. The application is time saving and also saving cost of physical prototyping and remanufacturing. Mechanical relationships between component such as conduct interference, hole alignment checks and link the motion of pulley can also be created.

With the powerful design validation tools in SolidWorks software, the designed models can be easily subjected to the same condition that it will experience in the real world.

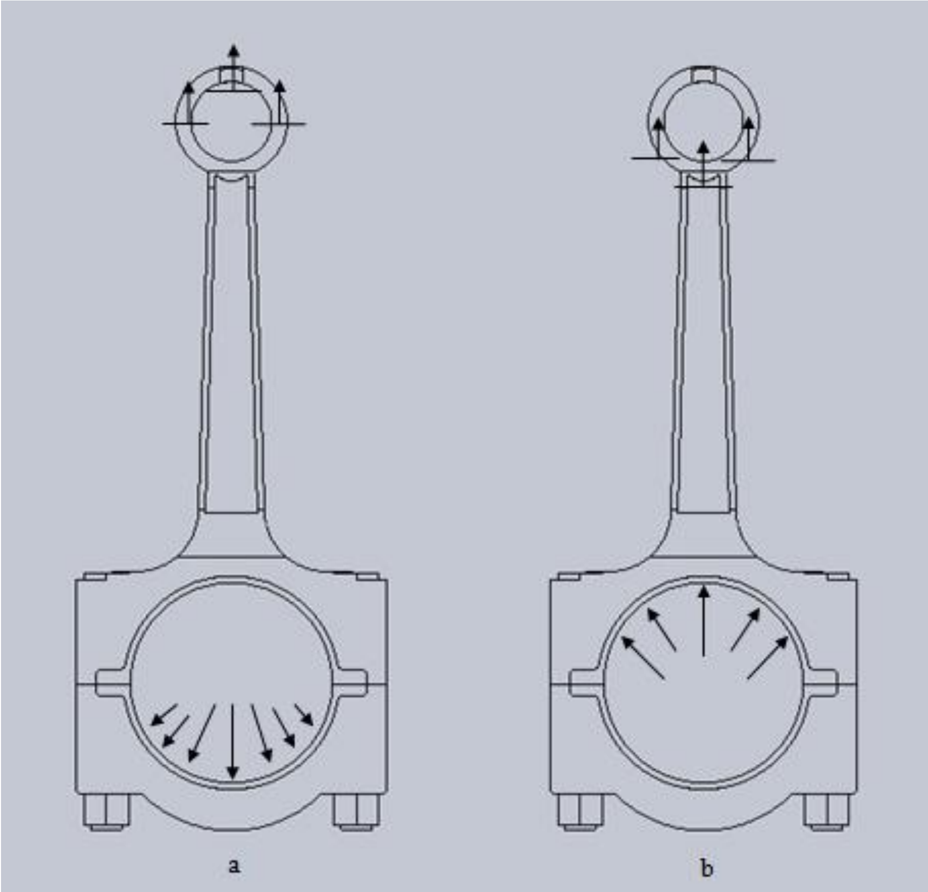


Figure 2.4: Loading and constraint used for FEA. (a) Tension loading at the crank end. (b) Compression loading at the crank end.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

In order to achieve the objective in this project, the overall of the methodology is explained briefly step by step from the initial progress until the project completion at the end of this semester. In this step, the methodology will go through the analysis of connecting rod using Finite Element Analysis (FEA) software, which is ALGOR.

3.2 PROJECT FLOW CHART

In analysis of connecting rod, there is a planning for overall progress to assure that project can be finish on schedule.

From the Figure 3.1, the project starts with find the literature review and research about the connecting rod fracture. This consist a review of the fracture of connecting rod. These tasks have been done through research on the internet, books and others sources.

After gathering all the relevant information, the project undergoes drawing process. The selected design sketched is the transfer to solid modeling and engineering drawing using SolidWorks software.

After the engineering drawing finished – include detail design and drawing and approved by supervisor, the drawing was used as a reference. After the design process, comes analysis process. The analysis is to investigate the failure for connecting rod.

After the process mentioned above is done. The report writing process will guided by the UMP thesis final year project report writing.

The project ended after the submission of the report and the slide presentation has been present.

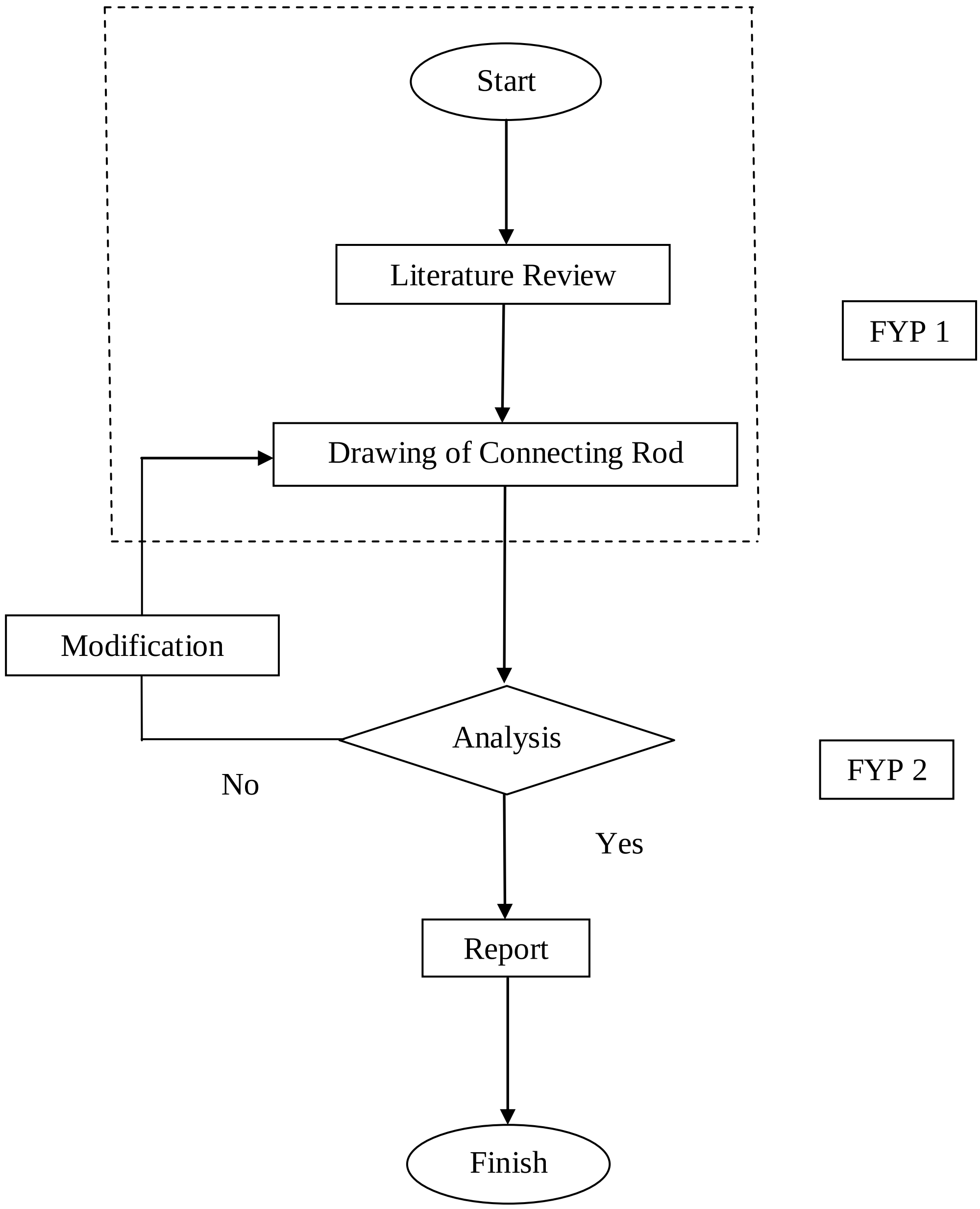


Figure 3.1: Project Flow Diagram

Figure 3.2 shows the flow diagram of analysis. The analysis stat with drawing the connecting rod and then saved in IGES file.

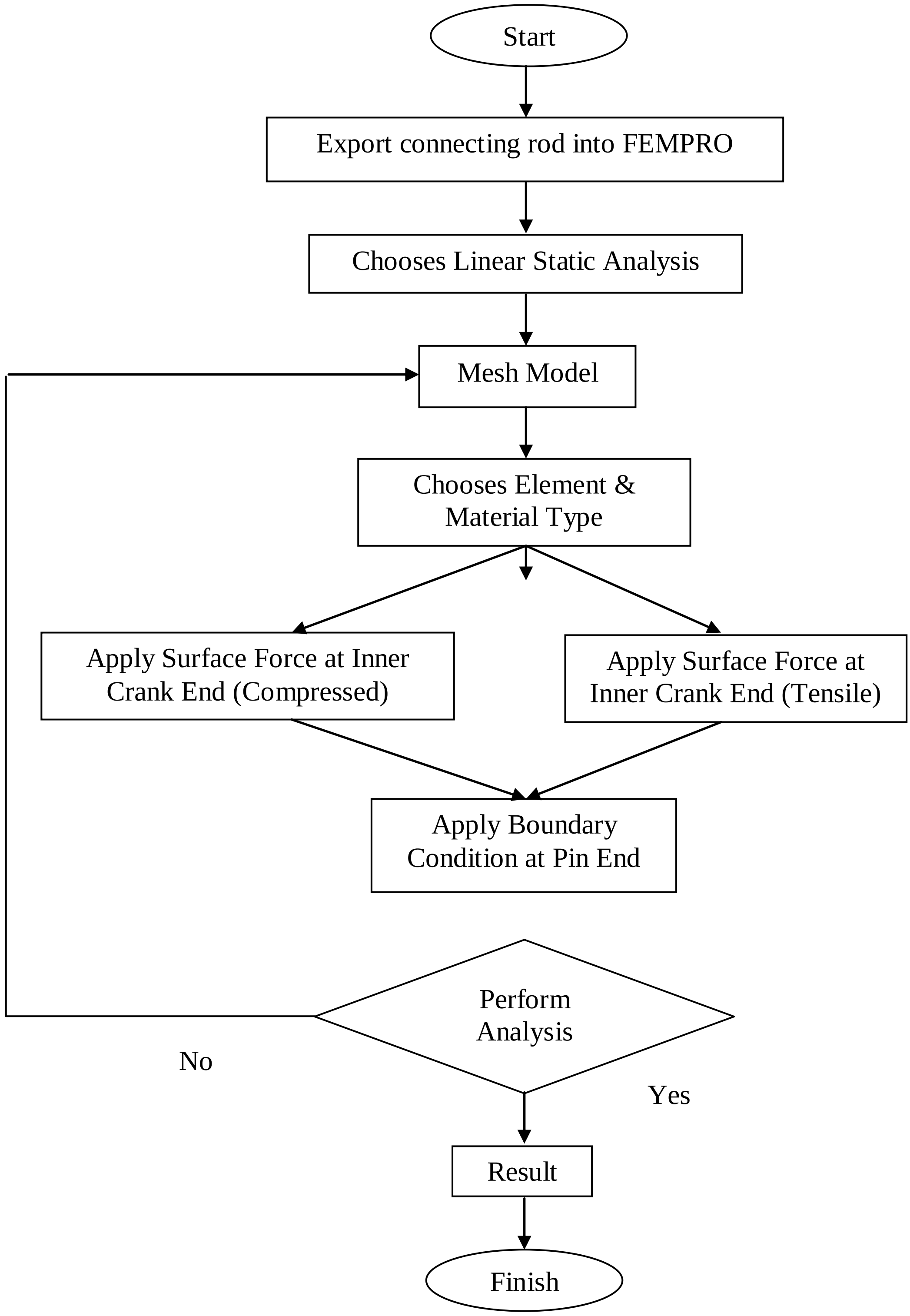


Figure 3.2: Flow Diagram of Analysis

3.3 MODEL DRAWING

The geometry model for the connecting rod had drawn using solid work software. For this drawing using dimension of connecting rod is using Proton Wira 1.5 engine specification. Figure 3.3 shows the dimension of connecting rod.

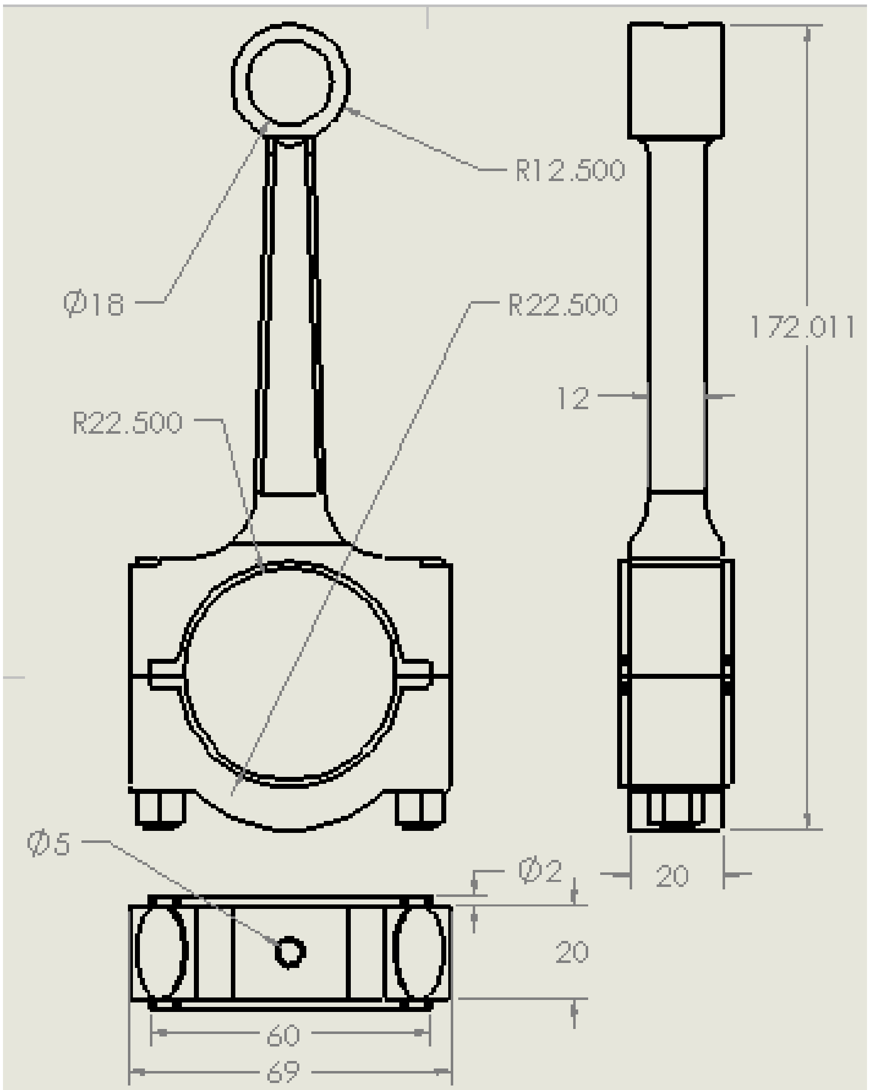


Figure 3.3: Dimension of connecting rod

3.4 SOFTWARE ANALYSIS

By using Finite Element, stress analysis was performed. Connecting rod would be the most suitable part for the analysis. The simple analysis will run at the first point of the connecting rod. This analysis will run by using three different meshes which 100% and 80%.

3.5 DESIGN

In design the connecting rod, the CAD software solid works 2007 is used. Finished with assembly all part, a completed model of connecting rod is ready. The model is then inspected for any mistake and several adjustments are made such as adding fillet to any sharp edges of the model. After finishing, the model is then saves and converts to IGES file and ready to export to CAE software for finite element analysis. IGES file type is the only file type that ALGOR could read when exporting any design from SolidWorks. Figure 3.4 shows the connecting rod from the SolidWorks software.

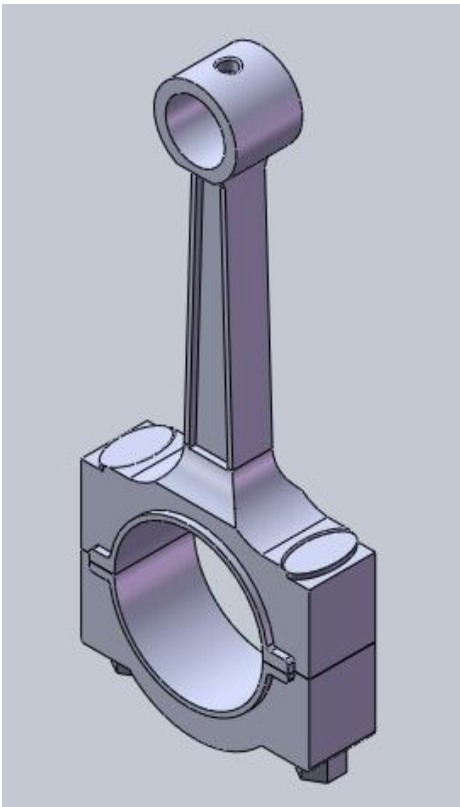


Figure 3.4: Connecting rod from SolidWorks software

3.6 FINITE ELEMENT ANALYSIS

Finite element analysis is a computational technique used to obtain approximate solution of boundary value problem in engineering. The simple state, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific condition on the boundary of the domain.

3.6.1 A General Procedure for Finite Element Analysis

Certain step in formulating a finite element analysis of a physical problem is common to all such analysis. These steps are embodied in commercial finite element software packages.

3.6.1.1 Preprocessing

The processing step is quite generally, described as defining the model and includes:

- Define the geometry domain of the problem
- Define the element type to be used
- Define the material properties of the element
- Define the geometry properties of the elements such as length, area and so on.
- Define the connectivity (mesh the model)
- Define the physical constraints (boundary conditions)

.

The preprocessing step is critical. A perfectly compute finite element solution is of absolutely no value if it corresponds to the wrong problem.

3.6.1.2 Solution

During the solution phase, finite element software assembles the governing algebraic equations in matrix form and computes the unknown values of the primary filed variables. The computed value is then used by back substitution to compute additional, derived variables such as reaction forces and element stresses. As it is not uncommon for a finite element model to be represented by tens of thousands of equations, special solution technique are used to reduce data storage requirements and computation time.

3.6.1.3 Post Processing

Analysis and evaluation of the solution results is referred to as post processing. Post processing software contains sophisticated routines used for sporting, printing and plotting selected result from a finite element solution. While solution data can be manipulated many ways in post processing, the most important objective is to apply sound engineering judgment in determining whether the solution result are physically reasonable.

3.6.2 Analysis Flow

3.6.2.1 Exports to CAE Software

The Computer Aided Engineering (CAE) software used for the finite element analysis is ALGOR V16. For import the design of the connecting rod from CAD environment is made possible when the model saved in IGES file in the CAD software. Figure 3.5 shows the connecting rod in IGES file.

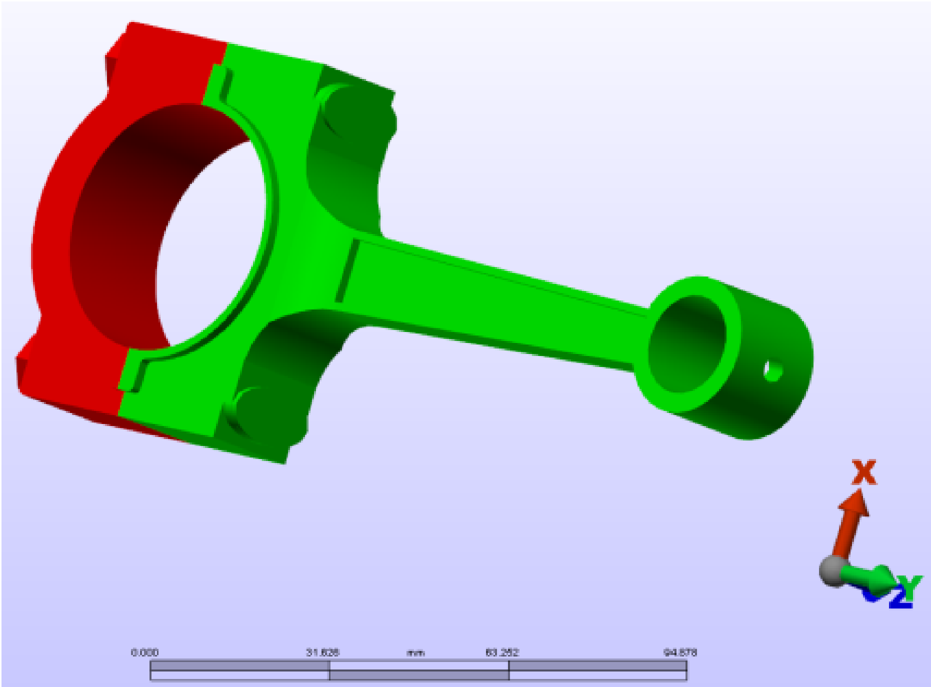


Figure 3.5: Connecting rod in IGES

3.6.2.2 Linear Static Analysis

In this step, linear step analysis allows design to test different load condition and their resulting stresses and deformation. Figure 3.6 shows the choose analysis type. Linear analysis allows design engineers and analysis expert to perform basic stress analysis of simple or complex model 3D assemblies. The program produces displacement, strain, stress, forces and error estimate as result of the analysis.

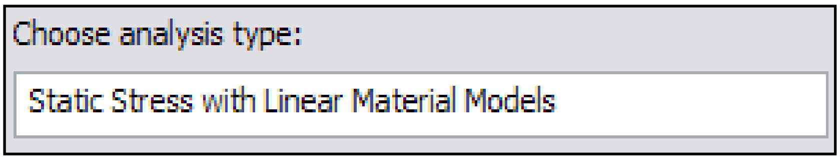


Figure 3.6: Chooses analysis type

3.6.2.3 Mesh Model

For analysis, every design must be meshed. From the CAD solid model, will go through FEMPRO surface and midplane meshing. After that, select solid meshing. In this process, the surface loaded previously area method into a number of elements with a certain number of nodes. If ‘coarse’ mesh size, have less element then analysis not accurate and otherwise if the ‘fine’ meshes size, have more elements and the analysis more accurate. Figure 3.7 shows the mesh model setting and Figure 3.8 shows the connecting rod after mesh.

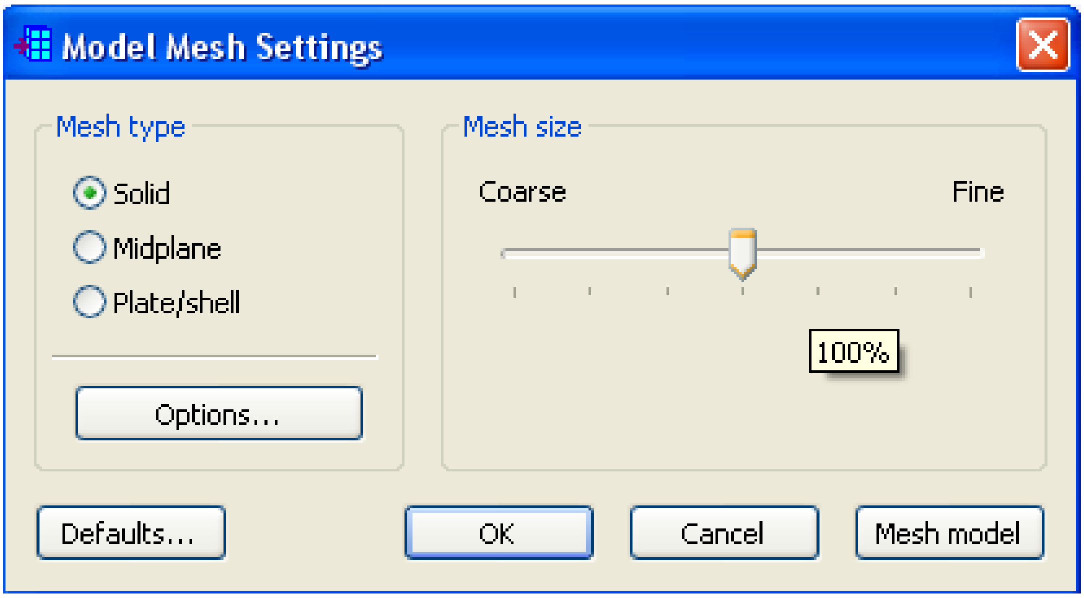


Figure 3.7: Mesh model setting

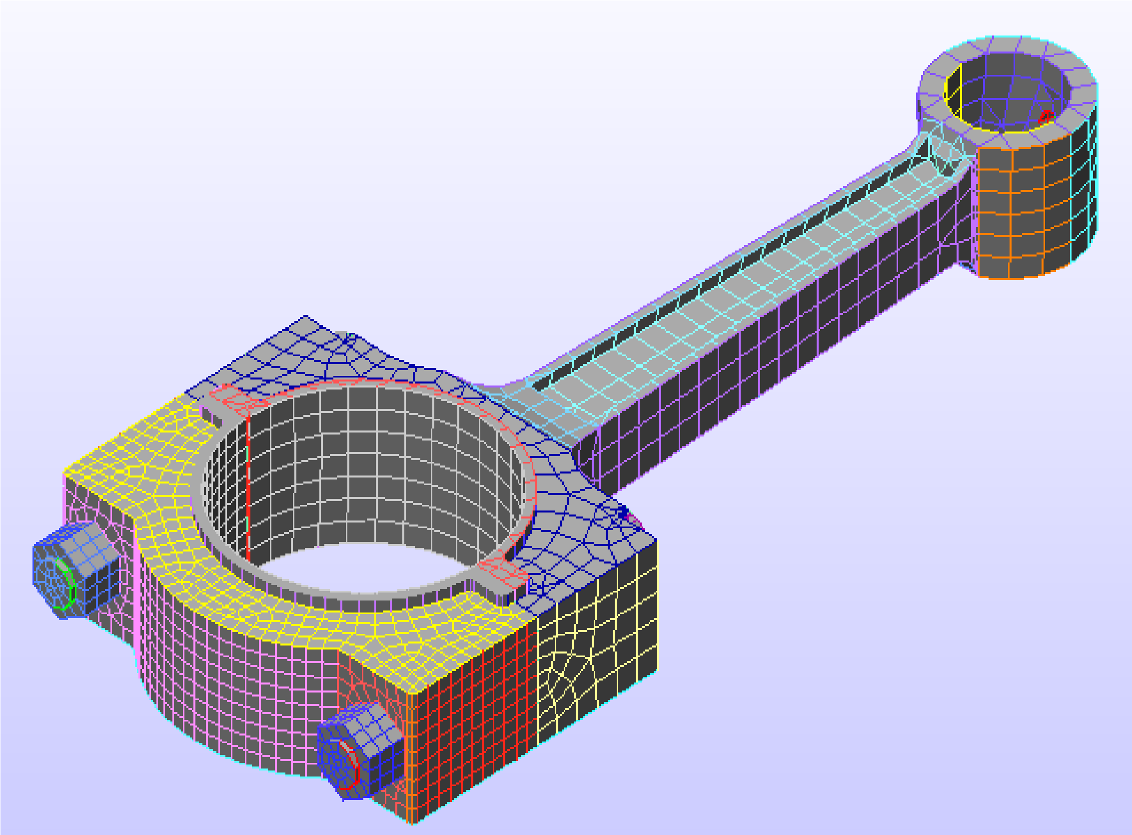


Figure 3.8: Meshing of connecting rod

3.6.2.4 Element and Material Type

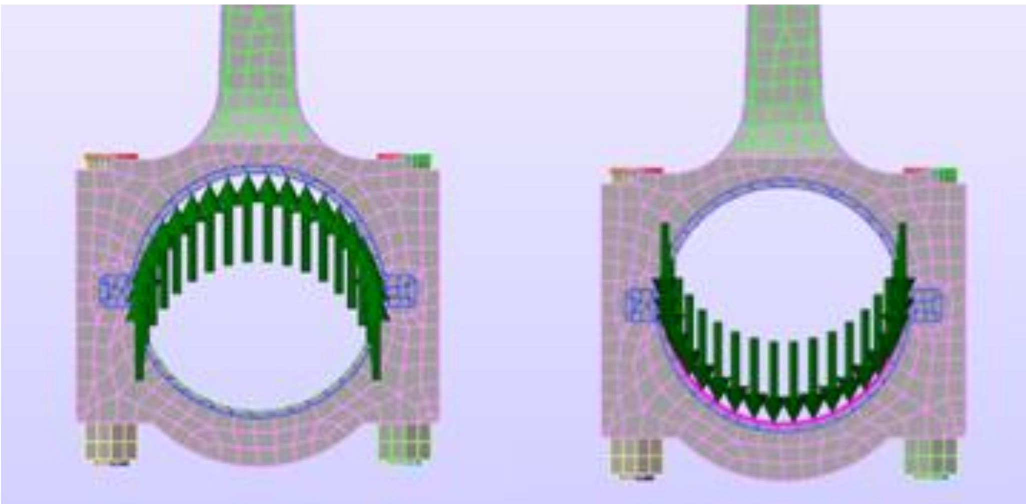
Element and material type in the ALGOR software is use to define all element and material group. Different finite element type such as brick, beam and tetrahedron can be effect to result analysis. The different material can be used to choose type of material such as aluminum, brass, plastic, glass, steel and so on. Every analysis used two different materials to makes comparison. In this analysis used steel (AISI 4130) and aluminum alloy.

3.6.2.5 Surface Force

The surface force at the crank end inner of the connecting rod is fixed and the load (magnitude) is applied. Y direction use for compressed analysis and -Y direction use for tensile analysis. Figure 3.9a show the surface force for compressed analysis and figure 3.9b show the surface force for tensile analysis. For this analysis, force is variables. Hence, force was used is 24kN, 26kN, 28kN, 30kN and 32kN to find failure of connecting rod.

3.6.2.6 Surface Boundary Condition

The surface boundary condition at the pin end of the connecting rod is fixed. Figure 3.10 shows the surface boundary condition for connecting rod.



a) Compressed b) Tensile

Figure 3.9: Surface force

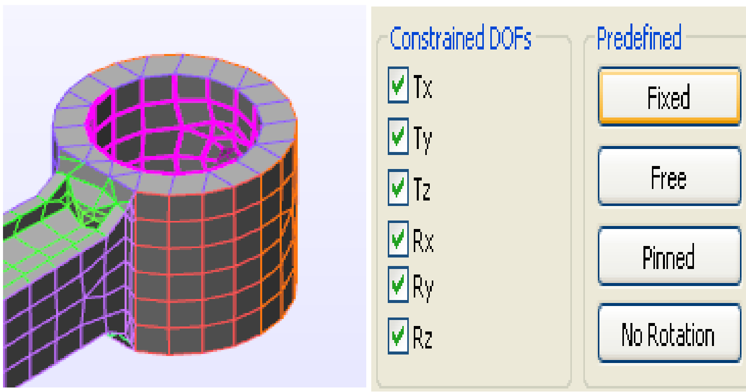


Figure 3.10: Surface boundary condition

3.6.2.7 Analysis Model

After the load applied to the crank end inner connecting rod, the next step is to perform the analysis. The model deformation displacement, stress and strain are viewable and the result can be easily saved in PDF format or word.

3.6.2.8 Von Misses Stress

After complete the analysis, the Von Misses Analysis value result will be compared to the material Yield Strength. The part will be considered as failed if the Von Misses Stress value is exceed the yield strength value. The analysis procedures then repeated by others parameter chosen such as material, mesh and load.

3.7 THREE-DIMENSIONAL ELEMENTS

Three-dimensional element based on extension of triangular element to tetrahedron and the other on extension of rectangular elements to rectangular parallelepipeds.

3.7.1 Four-Node Tetrahedral Element

A four-node tetrahedral element is depicted in Figure 3.11 in a relation to a global Cartesian coordinate system. The tetrahedral element definition for finite element models is so complex that it is almost always accomplished by auto meshing capabilities of specific software packages.

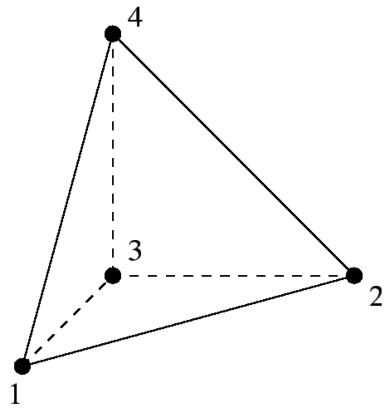


Figure 3.11: 4-Node tetrahedral element

Source: http://web.mit.edu/calculix_v1.6

3.8 STRESS STRAIN RELATIONS

The equations between stress and strain applicable to a particular material are known as the constitutive equations for the material. These two constants should be quite familiar from elementary strength of materials theory as the modulus of elasticity (Young's Modulus) and Poisson's ratio. The modulus of elasticity is defined as the slope of the stress strain curve in the elastic region or

$$E = \sigma_x/\varepsilon_x \quad (3.1)$$

Poisson's ratio is a measure of the well known phenomenon that an elastic body strained in one direction also experiences strain in mutually perpendicular directions. Poison's ratio is defined as

$$\nu = -\text{ unit lateral contraction / unit axial elongation} \quad (3.2)$$

The shear modulus or modulus of rigidly defined by:

$$G = E/2(1+\nu) \quad (3.3)$$

$$\{\sigma\} = [D]\{\varepsilon\} = [D][L]\{\varepsilon\} \quad (3.4)$$

3.9 BOUNDARY CONDITIONS

For a numerical simulation, it is impossible and unnecessary to simulate the whole universe. Generally we choose a region of interest in which we conduct a simulation. The interesting region has a certain boundary with the surrounding environment. Numerical simulations also have to consider the physical processes in the boundary region. In most cases, the boundary conditions are very important for the simulation region's physical processes. Different boundary conditions may cause quite different simulation results. Improper sets of boundary conditions may introduce nonphysical influences on the simulation system, while a proper set of boundary conditions can avoid that. So arranging the boundary conditions for different problems becomes very important. While at the same time, different variables in the environment may have different boundary conditions according to certain physical problems.

3.7.1 Static FEA Loading

The crank and piston pin ends are assumed to have a sinusoidal distributed loading over the contact surface area, under tensile loading (Webster *et al.* 1983). The normal pressure on the contact surface is given by:

$$p = p_o \, Cos\theta \quad (3.5)$$

The normal pressure constant p_o is, therefore given by:

$$p_o = P_t / (rt\pi/2) \quad (3.6)$$

For compressive loading of the connecting rod, the crank and the piston pin ends are assumed to have a uniformly distributed loading through 120_o contact surface (Webster *et al.* 1983). The normal pressure is given by:

$$p = p_o \quad (3.7)$$

The normal pressure constant is then given by:

$$p_o = P_c / (r\, t \sqrt{3}) \quad (3.8)$$

Since the analysis is linear elastic, for static analysis the stress, displacement and strain are proportional to the magnitude of the load. Therefore, the obtained results from FEA readily apply to other elastic load cases by using proportional scaling factor.

CHAPTER 4

RESULT AND DISCUSSION

4.1 INTRODUCTION

This chapter describes the result and discussion from the first step until the last step. The step is design and analysis connecting rod using ALGOR software. The analysis shows the result for stress and strain.

4.2 RESULT

For this analysis using different load such as 24 kN, 26 kN, 28 kN, 30 kN and 32 kN to find the failure at connecting rod. Figure 4.1 shows the result for compressed stress when using 26kN as force and steel as material. The maximum value of stress von mises is 304.125 N/mm^2 and 0.0265 for minimum value. Figure 4.2 shows the result for compressed stress when using 24kN as force and aluminum alloy as material. The maximum value of stress von mises is 279.976 N/mm^2 and 0.0309 for minimum value. This result gets with using 100% mesh. Maximum results do not lead to fracture the connecting rod for the maximum value does not exceed the yield strength of the material. Refer to Figures 4.1 and 4.2, the critical point at the shank end of connecting rod. This is cause by the shank design. The failure occurs at the critical point when stress value exceeds yield strength of the material used.

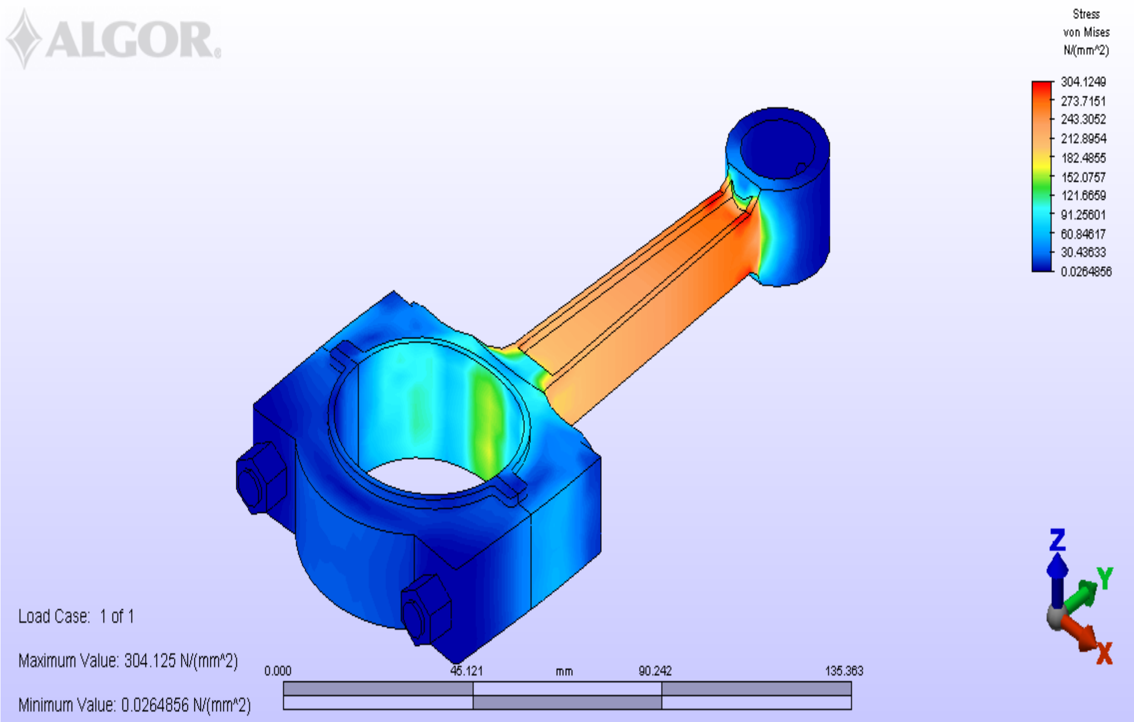


Figure 4.1: Compressed stress (steel)

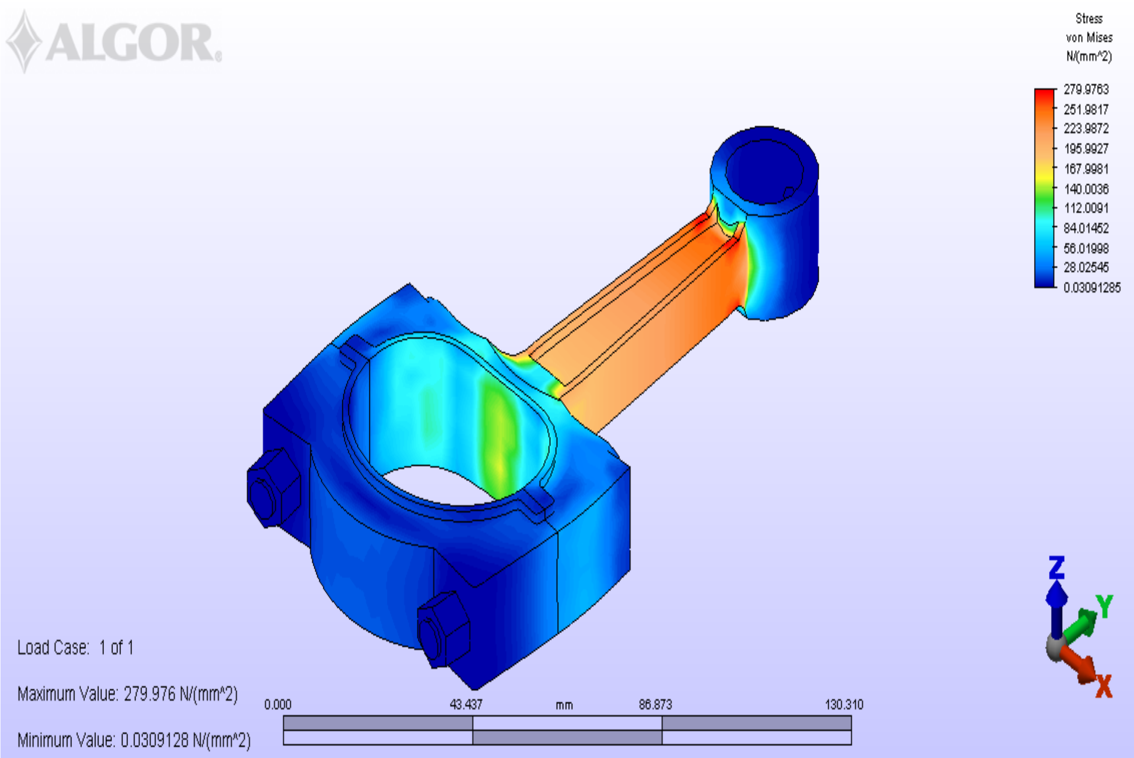


Figure 4.2: Compressed stress (aluminum alloy)

Figure 4.3 shows the result for tensile stress when using 26 kN as force and steel as material. The maximum value of stress von mises is 381.387 N/mm^2 and 0.0265 for minimum value. Figure 4.4 shows the result for tensile stress when using 24kN as force and aluminum allloy as material. The maximum value of stress von mises is 351.711 N/mm^2 and 0.0309 for minimum value. This result gets obtained has using 100% mesh. Maximum results do not lead to fracture the connecting rod for the maximum value does not exceed the yield strength of the material. Figure 4.3 and 4.4 show the critical point at inner crank end at the end of the shank connecting rod. This was due to design shank and assembles point of connecting rod.

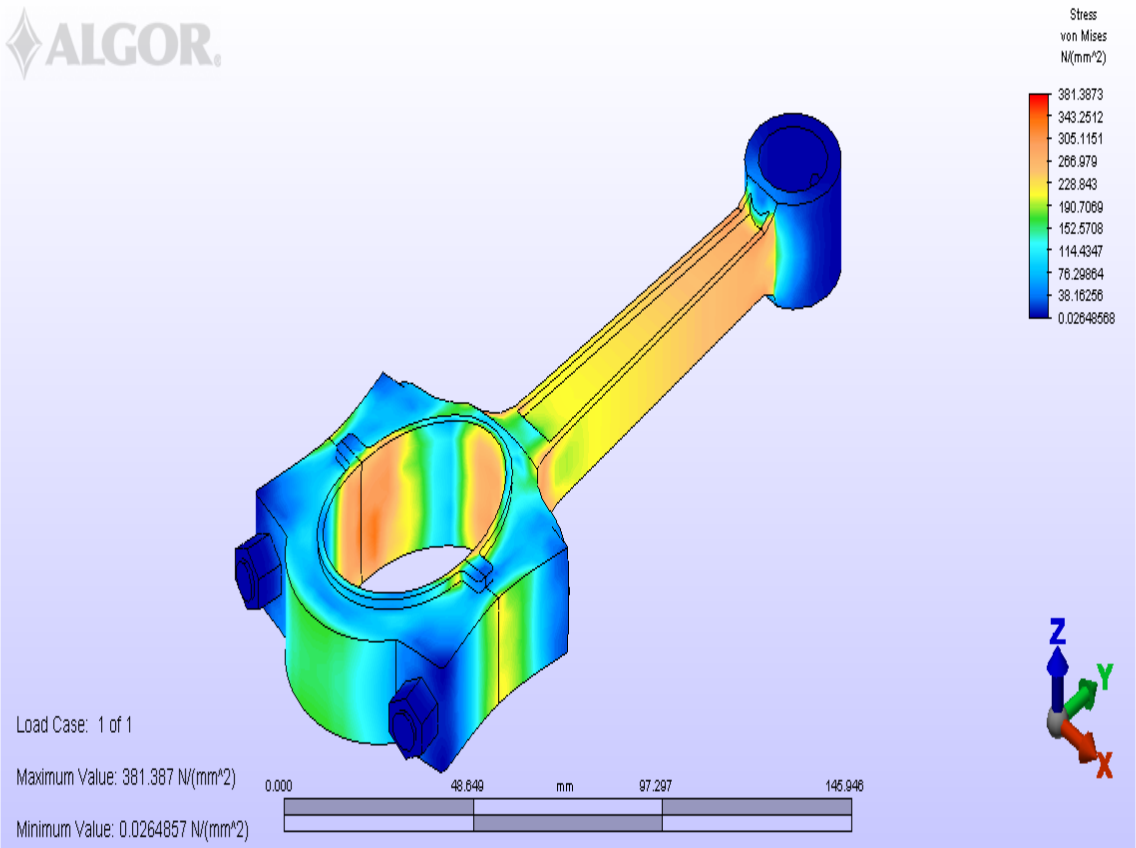


Figure 4.3: Tensile stress (steel)

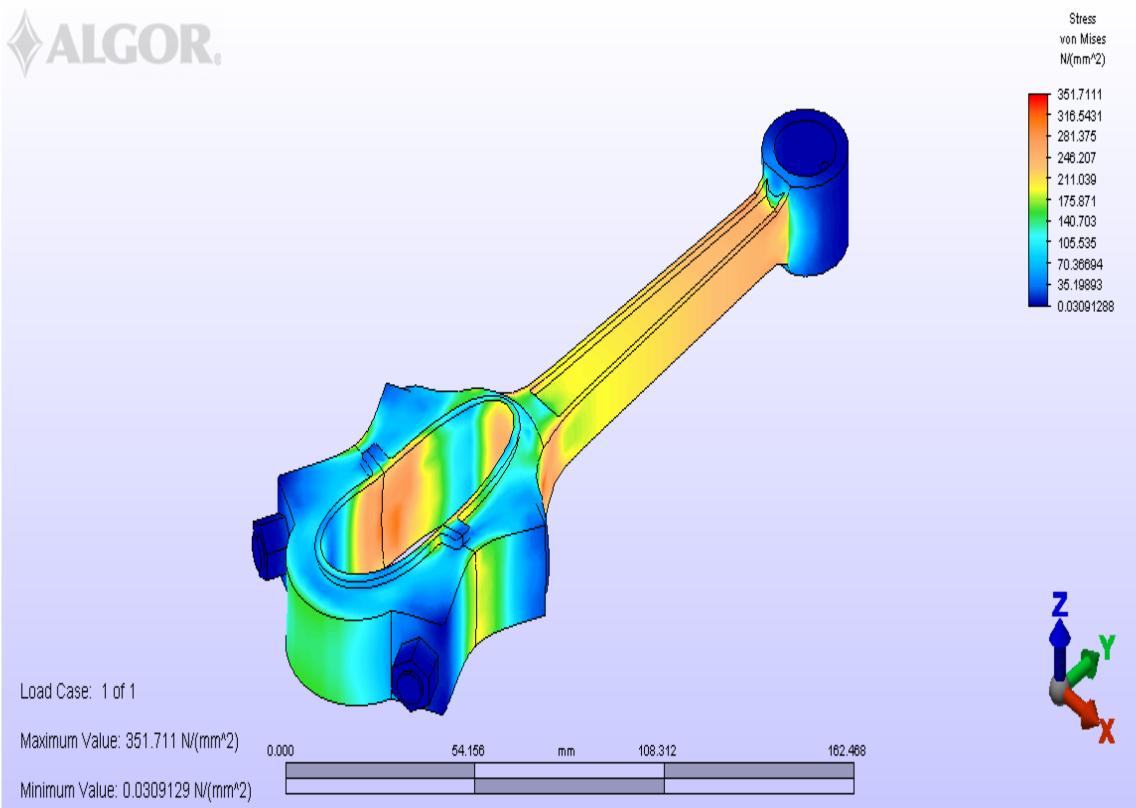


Figure 4.4: Tensile stress (aluminum alloy)

Analysis is performed several times using different force such as 26kN, 28 kN, 30 kN and 32 kN. Table 4.1 shows the result for compressed and tensile analysis using different mesh for steel AISI 4130. Table 4.2 shows the result for compressed and tensile analysis using different mesh for aluminum alloy. Table 4.3 shows the comparisons result for maximum compressed and maximum tensile analysis for different mesh and materials.

From the analysis, steel of connecting rod fracture occurs when the load are 26kN because tensile von mises stress exceed the yield strength of material (Yield strength steel= 360.6 N/mm^2) while aluminum alloy of connecting rod occurs when the load are 24kN because tensile von mises stress exceed the yield strength of material (Yield strength steel= 340 N/mm^2).

Figures 4.5 and 4.6 show the result for the maximum compress stress and maximum tensile stress using different mesh. The theoretically, tensile stress must be higher than compress stress because the elastic limit The material will not relax to its initial shape after the force is removed. The tensile strength where the material becomes plastic is called yield tensile strength. This is the point where the deformation (strain) of the material is unrecovered, and the work produced by external forces is not stored as elastic energy but will lead to cracks and ultimately failure of the construction.

Table 4.1: Result for compressed and tensile analysis using different meshfor steel AISI 4130.

Mesh (%)	Element	Load (kN)	Steel AISI 4130			
			Compress		Tensile	
			Stress Von Misses (N/mm^2)			
			Max	Min	Max	Min
100	12224	24	280.731	0.0244	352.05	0.0244
		26	304.125	0.0265	381.387	0.0265
		28	327.519	0.0285	410.722	0.0285
		30	350.914	0.0306	440.06	0.0306
		32	374.308	0.0326	469.398	0.0326
80	27168	24	309.634	0.0243	394.835	0.0268
		26	355.438	0.0263	427.738	0.029
		28	361.24	0.0283	460.64	0.0312
		30	387.042	0.0304	493.545	0.0335
		32	412.846	0.0324	526.448	0.0357

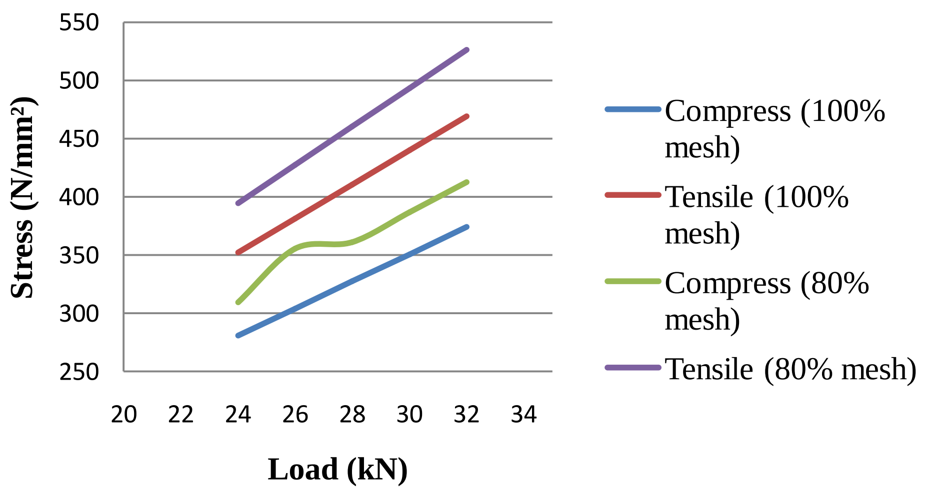


Figure 4.5: Stress vs Load for compress and tensile (Steel)

Table 4.2: Result for compressed and tensile analysis using different mesh for aluminum alloy

Mesh (%)	Element	Load (kN)	Aluminum Alloy Compress Max		Tensile	
			Stress Von Misses (N/mm^2)			
			Max	Min	Max	Min
100	12224	24	279.976	0.0309	351.711	0.0309
		26	303.307	0.0335	381.021	0.0335
		28	326.638	0.0361	410.328	0.0361
		30	349.971	0.0386	439.637	0.0386
		32	373.302	0.0412	468.946	0.0412
80	27168	24	310.208	0.01897	365.682	0.03
		26	336.06	0.0205	396.156	0.0326
		28	361.91	0.0221	426.629	0.0351
		30	387.759	0.0237	457.101	0.0376
		32	413.611	0.0253	487.575	0.0401

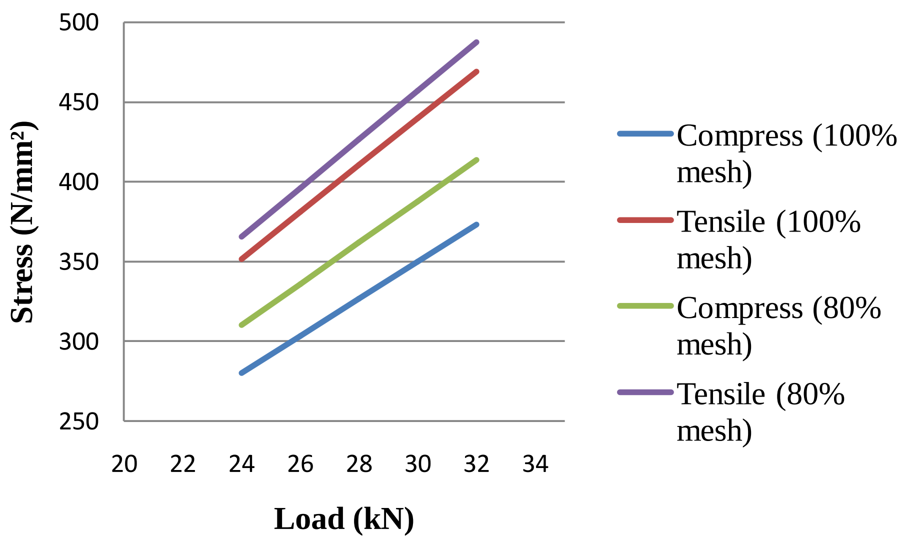


Figure 4.6: Stress vs Load for compress and tensile (Aluminum Alloy)

Table 4.3: Comparisons result for maximum compressed and maximum tensile analysis for different mesh and materials.

Mesh (%)	Load (kN)	Compress Mex. Stress Von Mises (N/mm^2)		Tensile Mex. Stress Von Mises (N/mm^2)	
		Steel AISI 4130	Aluminum Alloy	Steel AISI 4130	Aluminum Alloy
100	24	280.731	279.976	352.05	351.711
	26	304.125	303.307	381.387	381.021
	28	327.519	326.638	410.722	410.328
	30	350.914	349.971	440.06	439.637
	32	374.308	373.302	469.398	468.946
80	24	309.634	310.208	394.835	365.682
	26	355.438	336.06	427.738	396.156
	28	361.24	361.91	460.64	426.629
	30	387.042	387.759	493.545	457.101
	32	412.846	413.611	526.448	487.575

4.3 DISCUSSION

From the analysis, failures occur at the maximum stress area when maximum stress values exceed the yield strength of the material is used. Red area indicated the maximum value and the failure will be in this area. For the compressed analysis, failure will be at shank the connecting rod. For the tensile analysis, failure will be at inner crank end.

The result of the maximum deformation for the static quill analysis is shown in Table 4.3. Table 4.3 consists of maximum von mises stress using the different material and load such as steel and aluminum alloy for material, 24kN, 26kN, 28kN, 30kN, 32kN for the load. This integrated with different mesh size.

The range of mesh size used for the analysis is reduced from 100% to 80%. This is because a low percentage of mesh will cause the analysis of the model will take a longer time whereas we only use a standard laptop to perform the analysis. Meshing in the analysis can effect for the result. Smaller mesh is used the more accurate result can obtain. The relationship between maximum von mises stress and maximum with the number of elements used are shown in Table 4.1 for steel material and Table 4.2 for aluminum alloy material.

CHAPTER 5

CONCLUSION AND RECOMENDATION

5.1 CONCLUSION

Main objective of this project is analyzing fracture in connecting rod. Fracture of connecting rod measured using ALGOR software. In order to analyze, there are several type software was used such as SolidWorks and ALGOR. Drawing connecting rod using SolidWork software based on the actual connecting rod size used by proton wira 1.5 engines.

For this analysis, there are two material was selected, steel AISI 4130 and aluminum alloy. The stress von mises analysis actually was done. As a result, connecting rod made of aluminum alloy failure when apply load of 24kN which the maximum tensile stress von mises are 352.05 N/mm^2 while connecting rod made of steel AISI 4130 failure when apply load of 26kN which the maximum tensile stress von mises are 381.387 N/mm^2.. The failure measured when the maximum stress von mises exceed yield strength of material used where yield strength of aluminum alloy is 340 N/mm^2 and yield strength of steel are 360.6 N/mm^2. By comparison on the result, steel stress higher than aluminum alloy stress. Steel AISI 4130 has better endurance limit and better material for used in engines.

5.2 RECOMENDATION

For future study of this subject, there are few aspects that need to be considered and can improve for better and more accurate result.

Firstly, the connecting rod model should be meshed to finer mesh percentages. The mesh percentages use in this project is 100% and 80%. This value can be reduced again and it is recommended that for future analysis, the mesh percentages used should 20%- 40%. Although analysis time is increased and the simulation result obtain will be almost accurate to actual of the connecting rod model.

REFERENCES

Adila Afzal and Ali Ftemi(2003). A Comparative Study of Fatigue Behavior and Live Predictions of Forged Steel and PM Connecting Rods.the University of Toledo.

Athavale, S. and Sajanpawar, P. R., 1991, "Studies on Some Modelling Aspects in the Finite Element Analysis of Small Gasoline Engine Components," Small Engine Technology Conference Proceedings, Society of Automotive Engineers of Japan, Tokyo, pp. 379-389.

Balasubramaniam, B., Svoboda, M., and Bauer, W., 1991, "Structural optimization of I.C. engines subjected to mechanical and thermal loads," Computer Methods in Applied Mechanics and Engineering, Vol. 89, pp. 337-360.

David John Leeming, Reg Hartley (1989). Heavy Vehicle Technology. Tottenham Court Road. page 45

http://www.corusautomotive.com/file_source/automotive/Publications/Connecting_Rods.pf

http://www.mpif.org/designcenter/Conrod.pdf

James R Dale (2005). Connecting Rod Evaluation. Metal Powder Industries Federation.

Khatiblou M.A, Meadau Crest. Connecting Rod With Improve Fatigue Live.Publication number: US 1994/5485765A.

Kenji Hayama, Connecting Rod for Internal Combustion Engine, pattern 2008/0078353.

Nunney M.J. (2007).Light and Heavy Vehicle Technology, Elsevier Ltd. All right reserved

Peter Zernov, Milwaukee. Connecting Rod, pattern 2706263.

Rabb, R., 1996, "Fatigue failure of a connecting rod," Engineering Failure Analysis, Vol. 3, No. 1, pp. 13-28.

Roman Celin (2008), A Metallographic Examination of a Fractured Connecting Rod. Institute of Metals and Technology

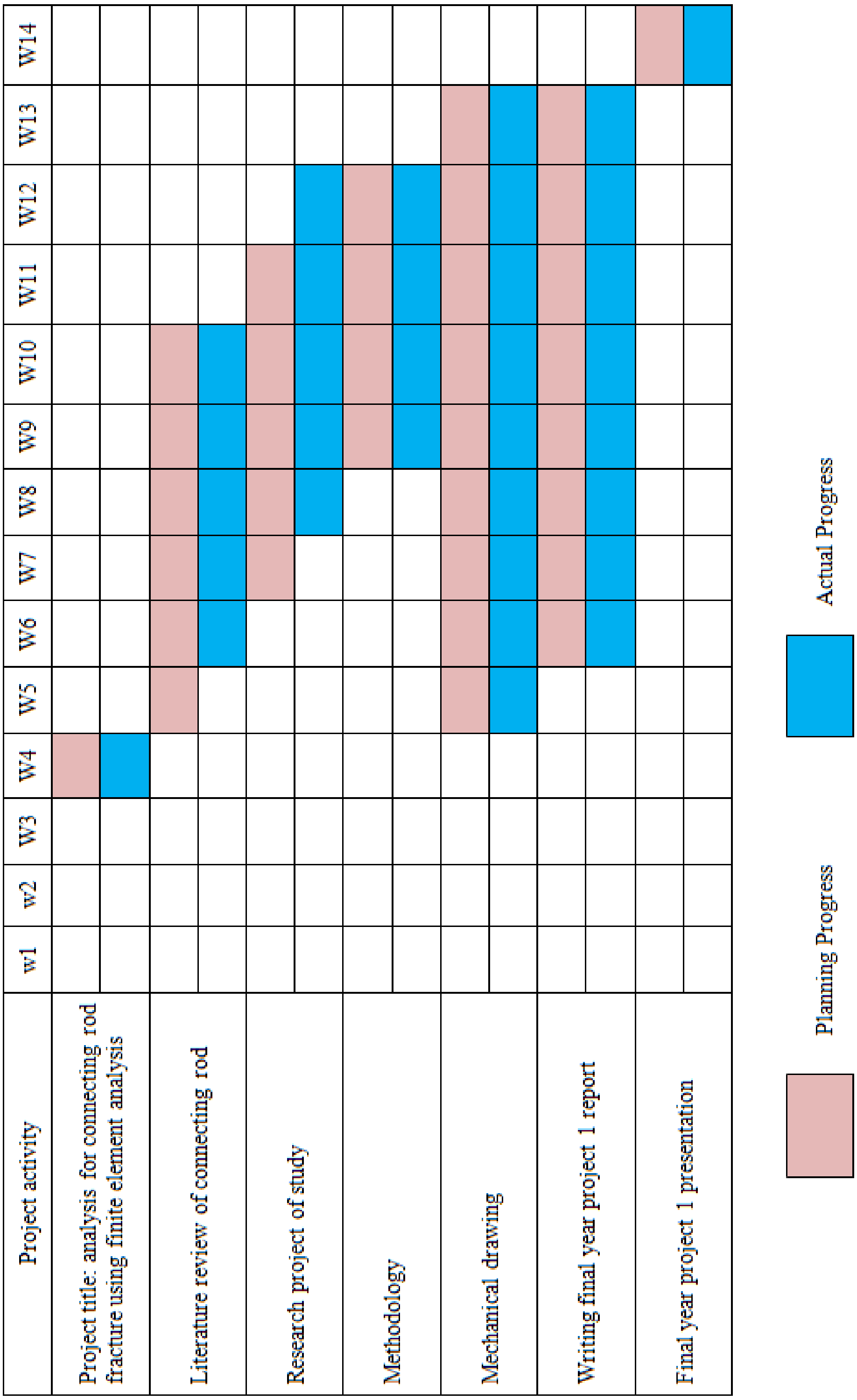
Sameh Hakim Andrawes Guirgis. Process to Fracture Connecting Rods and The Like With Resonance Fatigue.

Shenoy P.S.(2004). Dynamic Load Analysis and Optimization of Connecting Rod. The University of Toledo

Webster, W. D., Coffell, R., and Alfaro, D., 1983, “A three dimensional finite element analysis of a high speed diesel engine connecting rod,” SAE Technical Paper 831322, pp. 83-96.

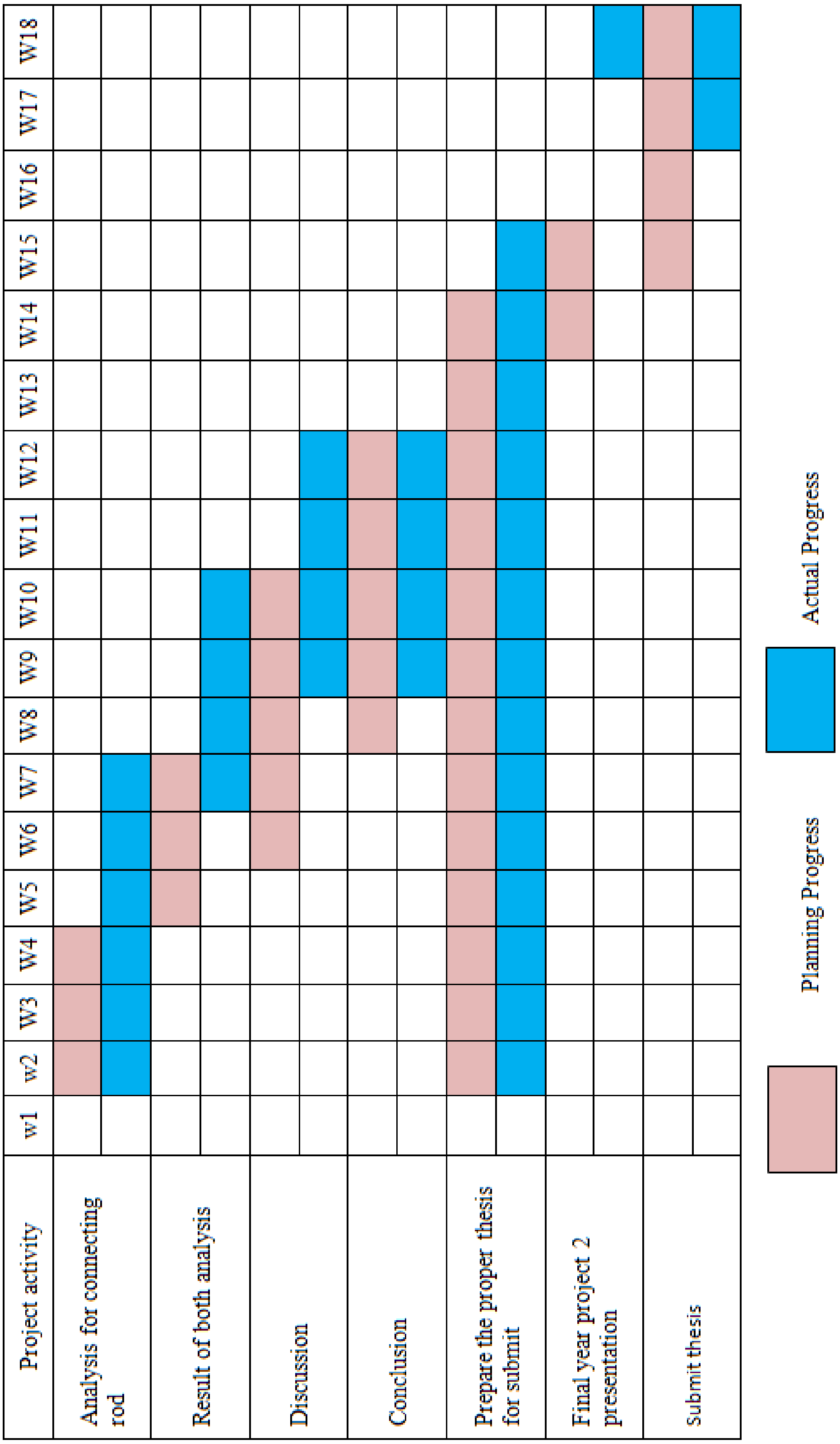
Xiao Lei Xu, Zhi Wei yu (2007). Failure Analysis of a Diesel Engine Connecting Rod

APPENDIX A
Gantt chart of Final Year Project 1

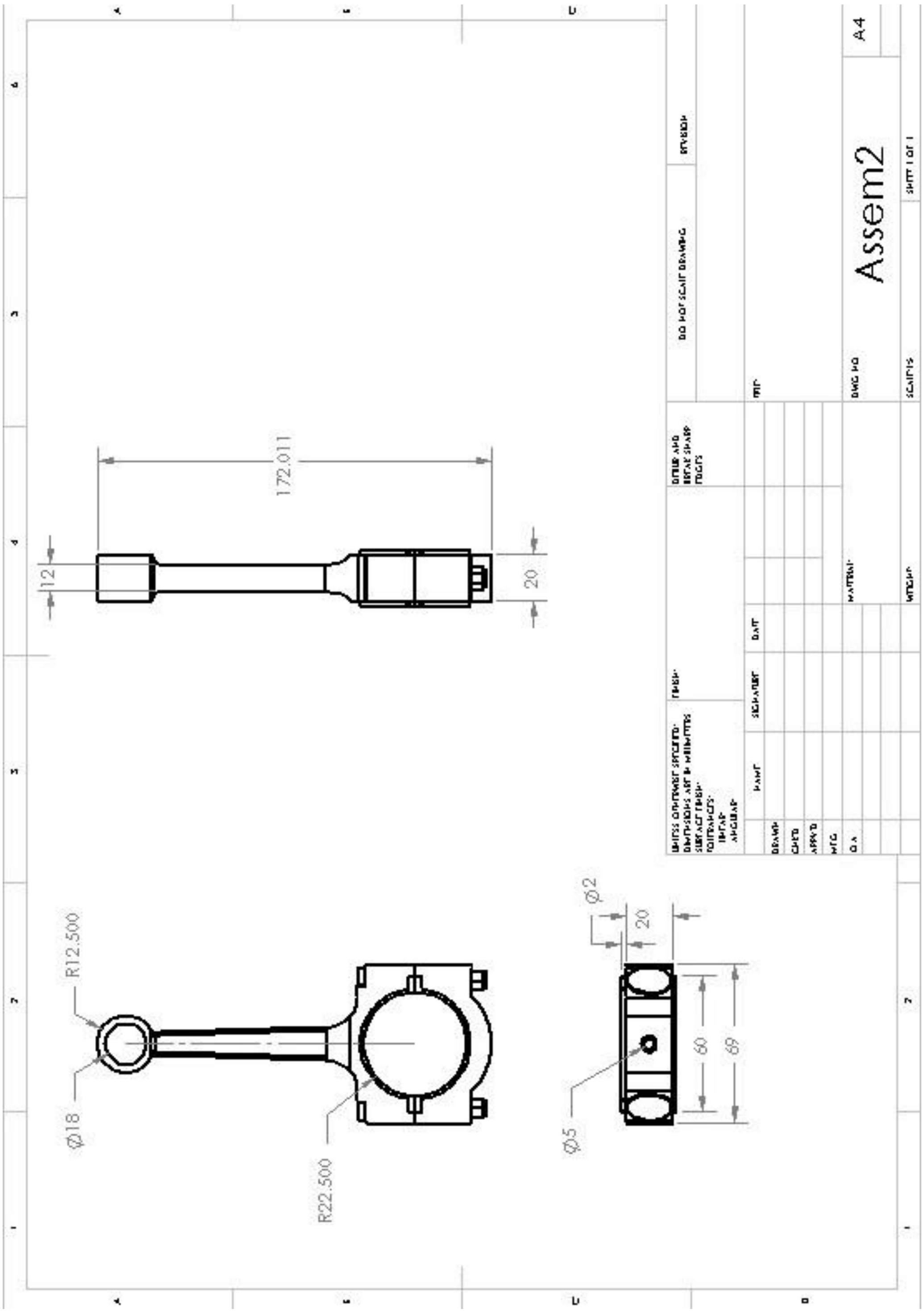


APPENDIX B

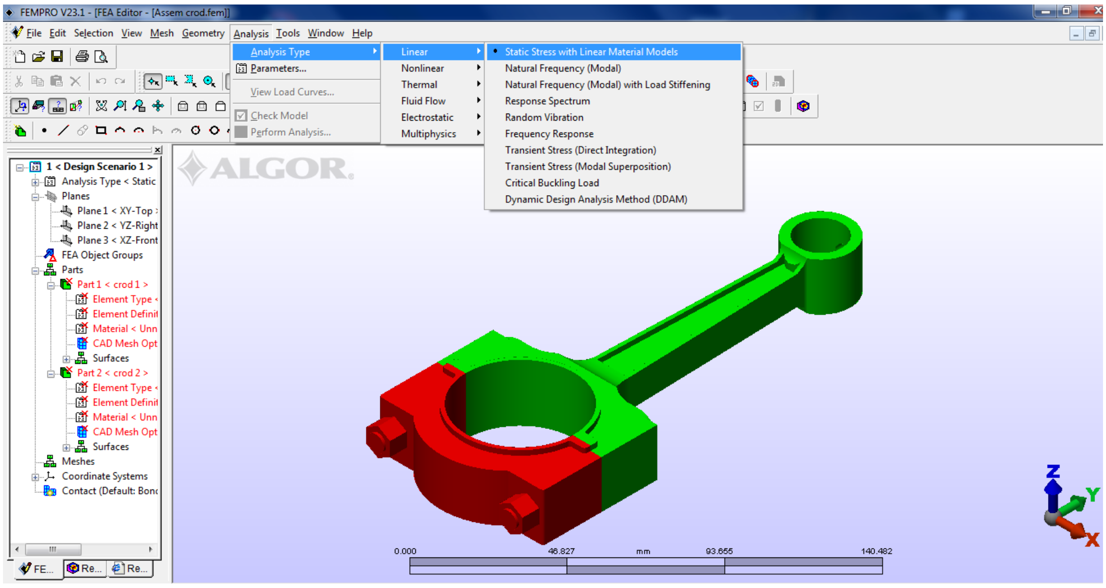
Gantt chart of Final Year Project 2

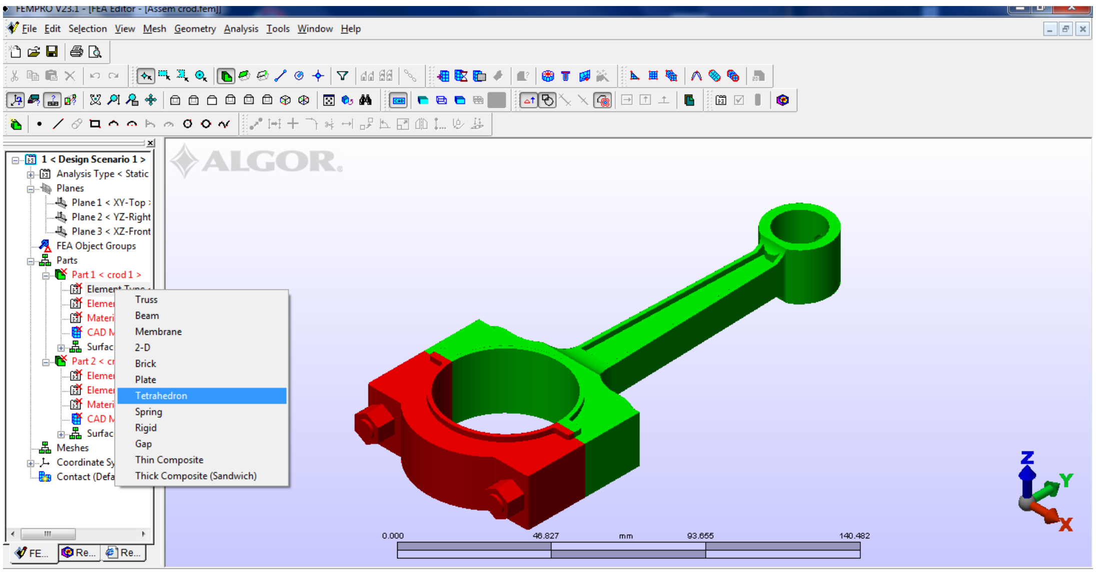


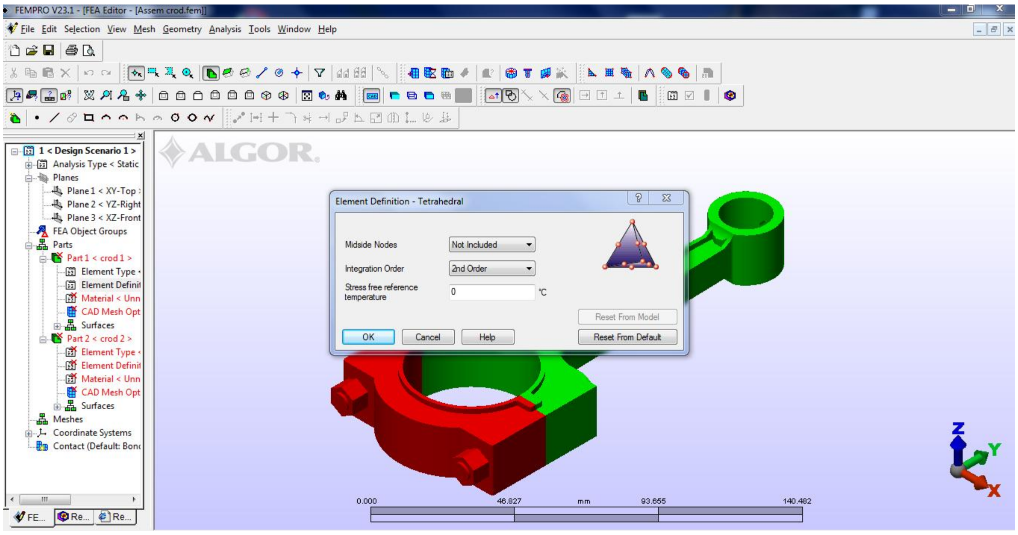
APPENDIX C
Dimension of Connecting Rod

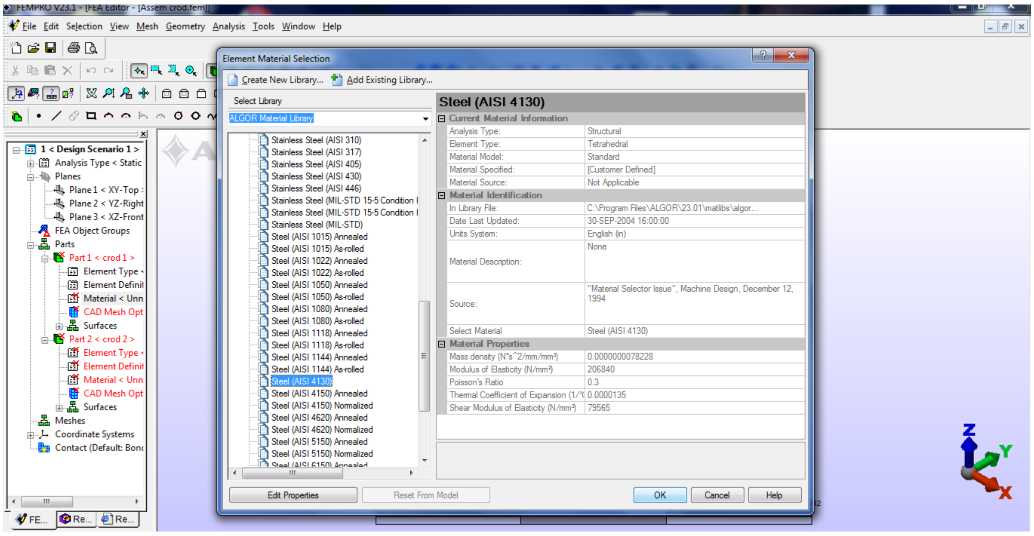


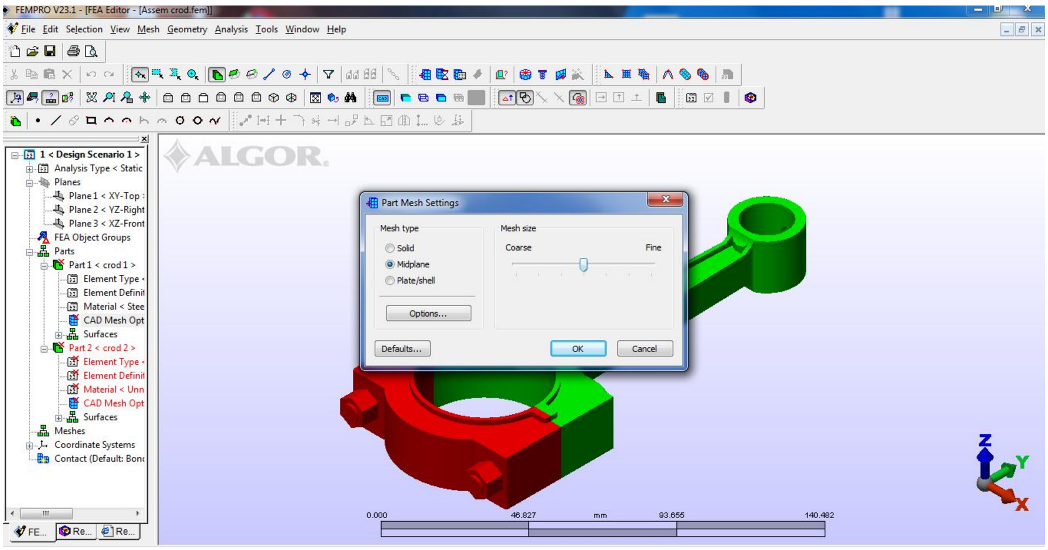
APPENDIC D
Analysis Flow

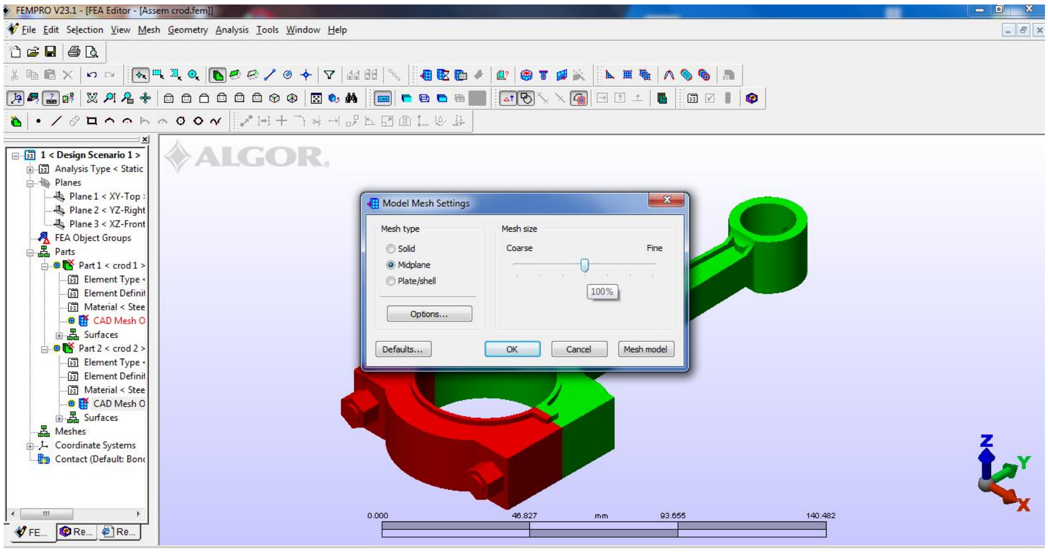


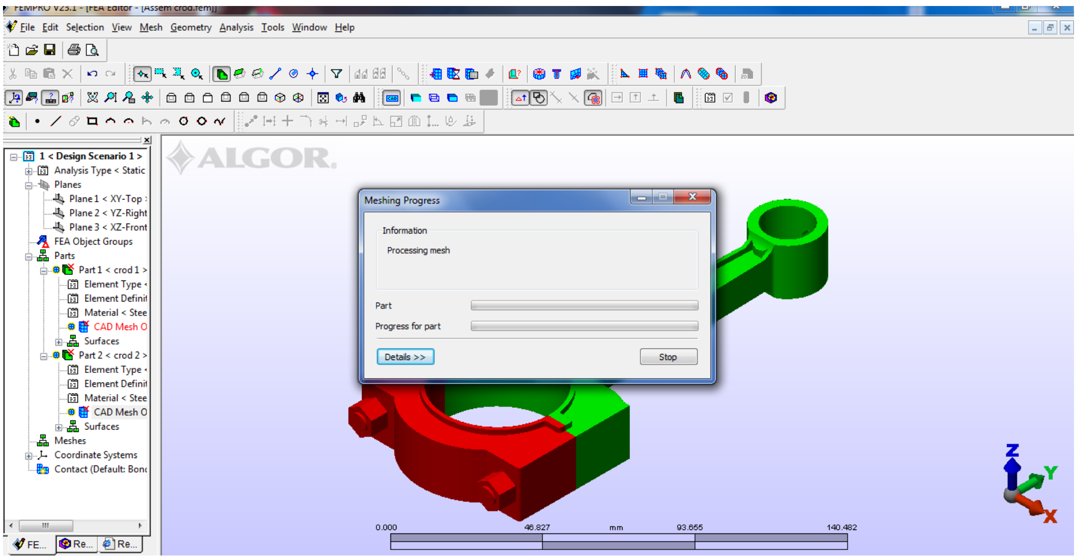


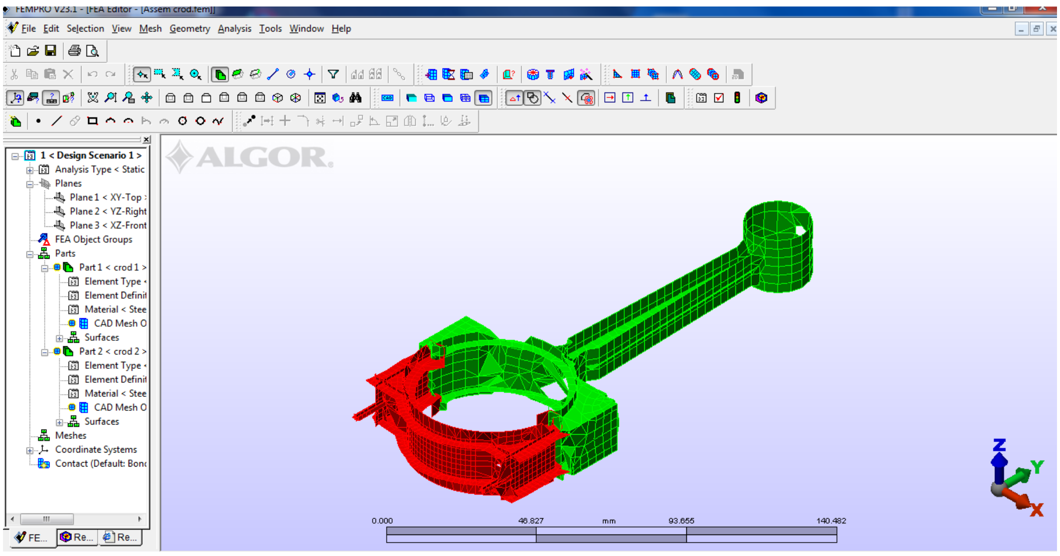

FEMPRO V23.1 - [FEA Editor - [Assem crod.fem]]
Element Definition - Tetrahedral
Midside Nodes
Not Included
Integration Order
2nd Order
Stress free reference temperature
Reset From Model
OK
Cancel
Help
Reset From Default
ALGOR

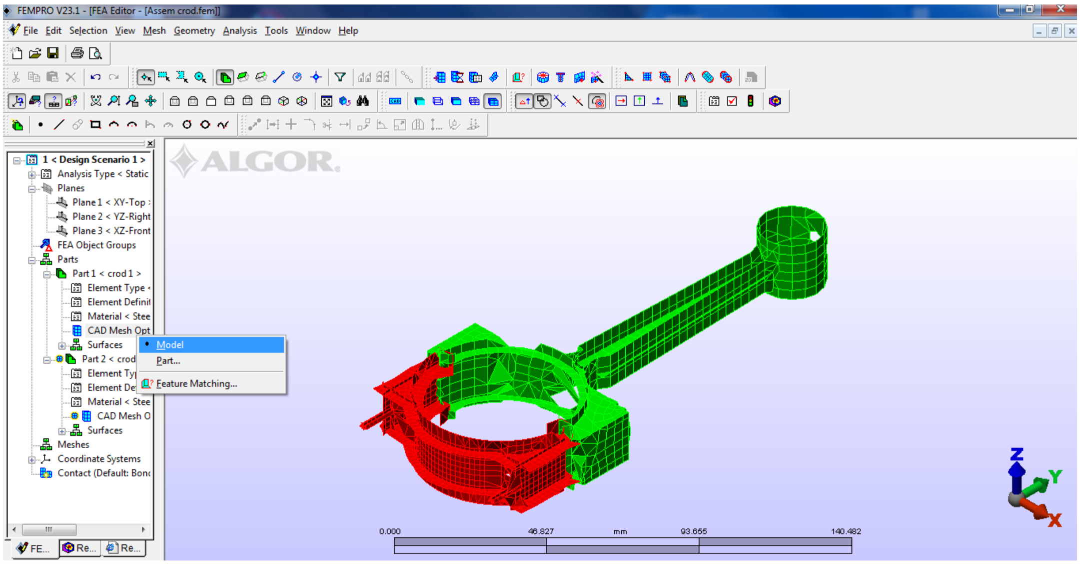

Element Material Selection
Create New Library...
Add Existing Library...
Select Library
ALGOR Material Library
Steel (AISI 4130)
Current Material Information
Material Identification
Material Properties
Edit Properties
Reset From Model
OK
Cancel
Help

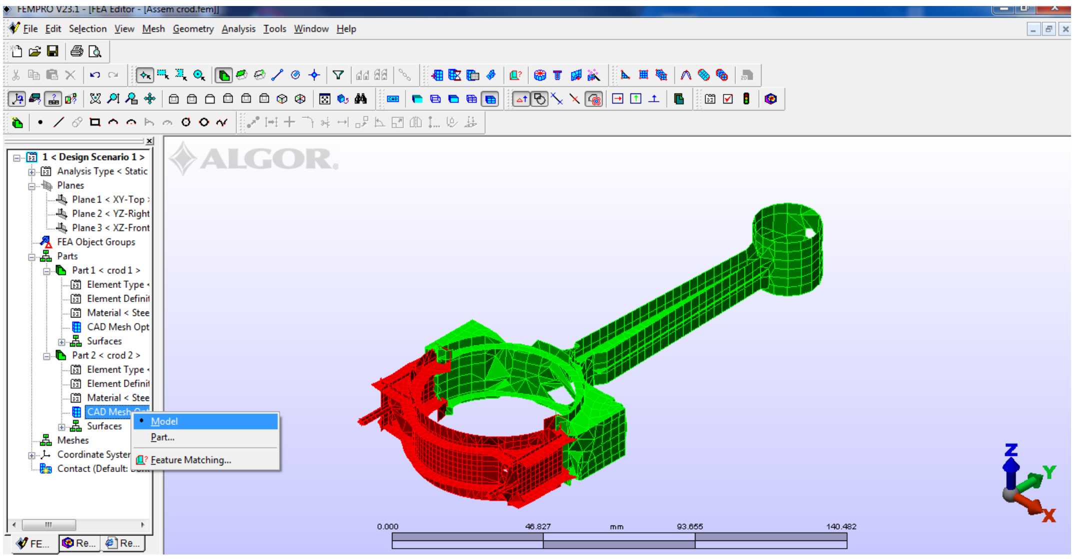

Part Mesh Settings
Mesh type
Solid
Midplane
Plate/shell
Options...
Mesh size
Coarse
Fine
Defaults...
OK
Cancel
ALGOR

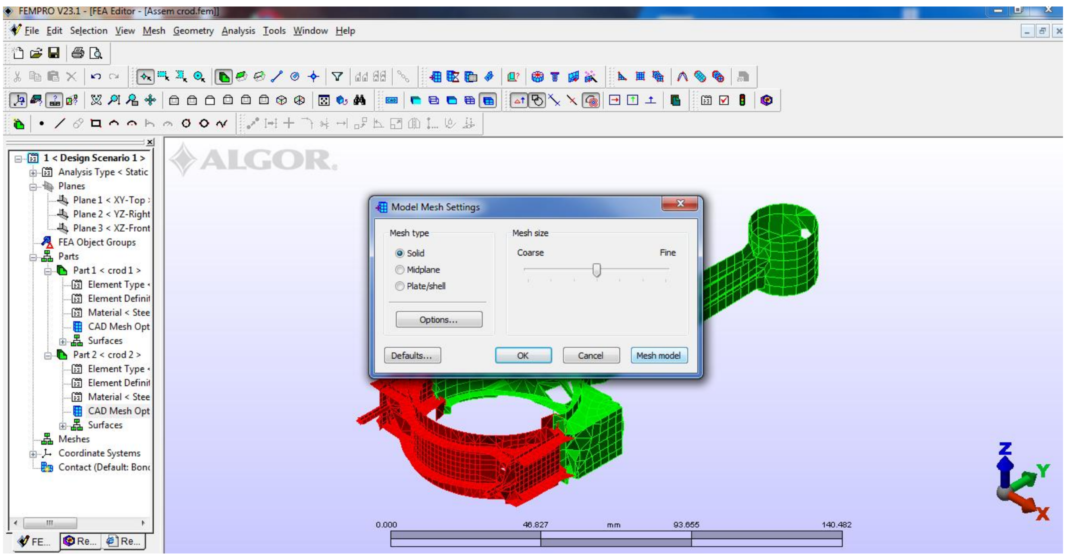

FEMPRO V23.1 - [FEA Editor - [Assem crod.fem]]
File Edit Selection View Mesh Geometry Analysis Tools Window Help
ALGOR
Model Mesh Settings
Mesh type
Solid
Midplane
Plate/shell
Options...
Mesh size
Coarse
Fine
100%
Defaults...
OK
Cancel
Mesh model
0.000
46.827
mm
93.655
140.482

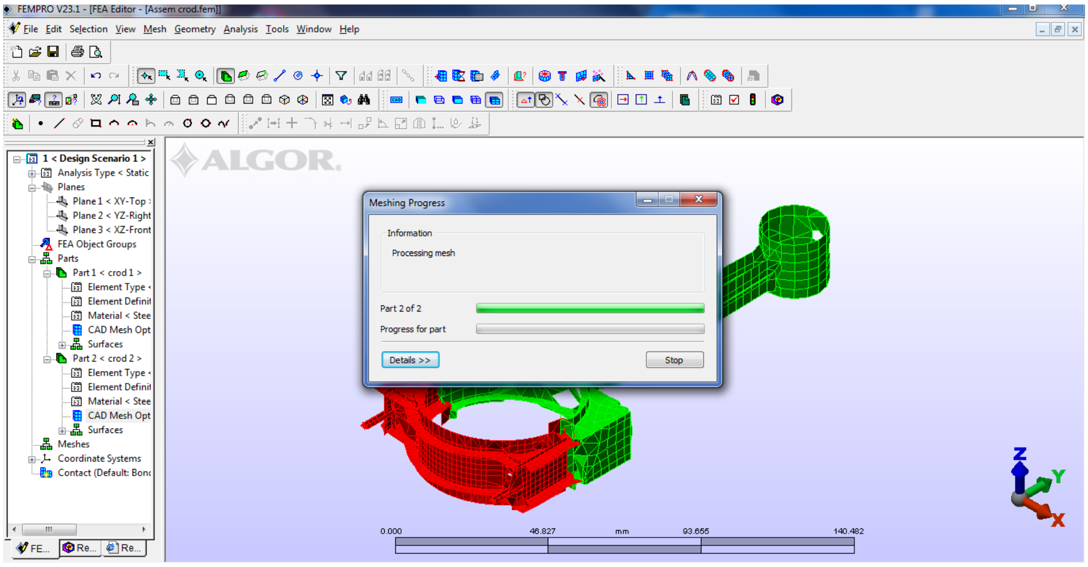

FEMPRO V23.1 - [FEA Editor - [Assem crod.fem]]
File Edit Selection View Mesh Geometry Analysis Tools Window Help
ALGOR
Meshing Progress
Information
Processing mesh
Part
Progress for part
Details >>
Stop
0.000
46.827
mm
93.655
140.482

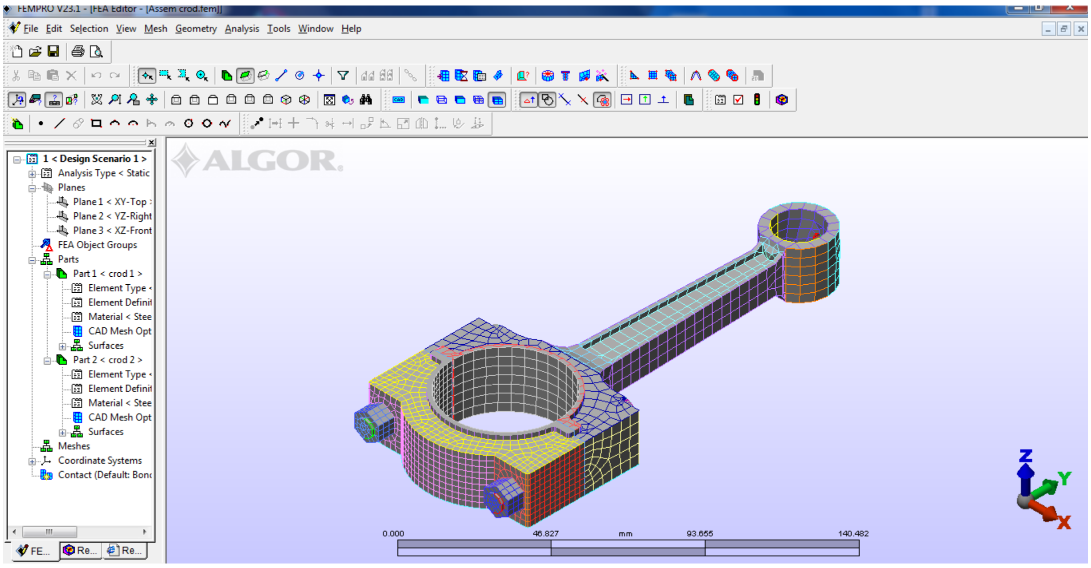

FEMPRO V23.1 - [FEA Editor - [Assem crod.fem]]
File Edit Selection View Mesh Geometry Analysis Tools Window Help
ALGOR
0.000
46.827
mm
93.655
140.482

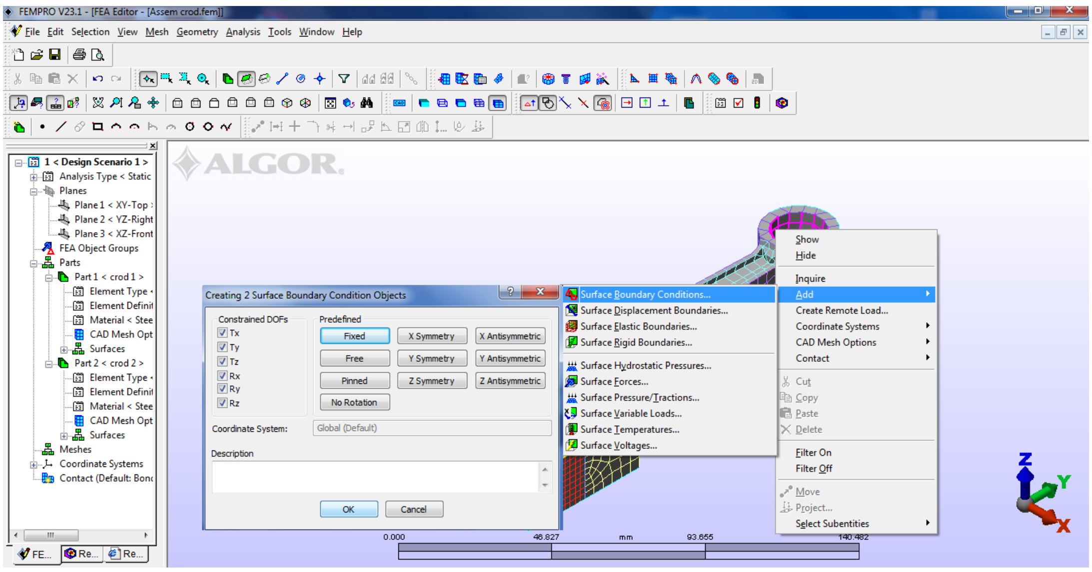


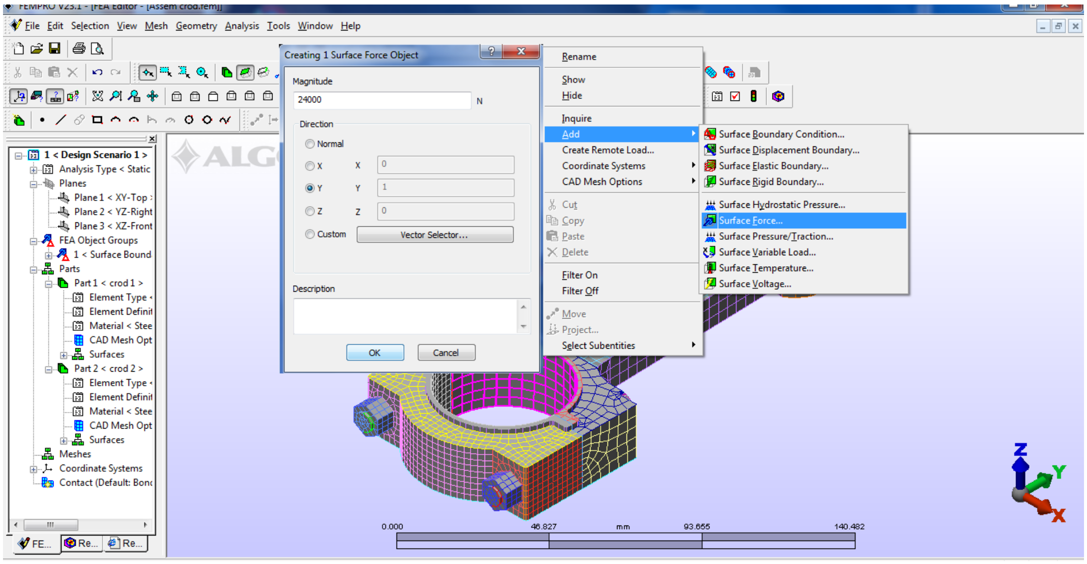


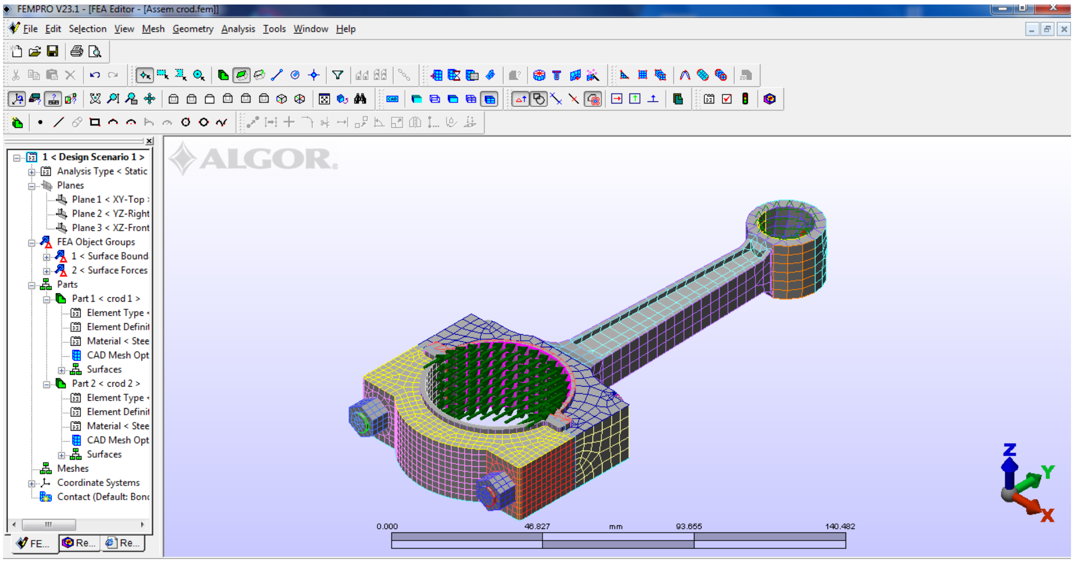


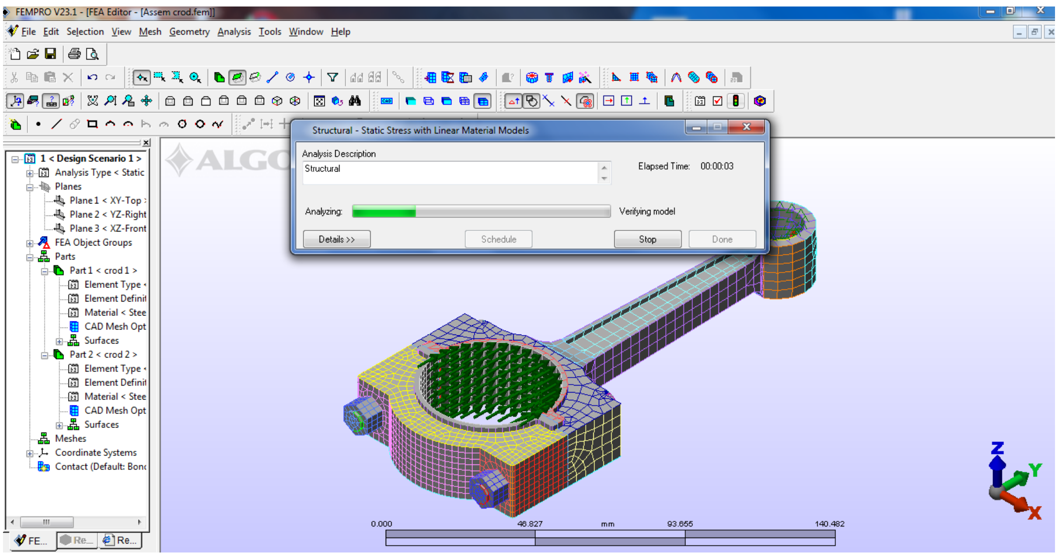

FEMPRO V23.1 - [FEA Editor - [Assem crod.fem]]
File Edit Selection View Mesh Geometry Analysis Tools Window Help
1 < Design Scenario 1 >
Analysis Type < Static
Planes
Plane 1 < XY-Top
Plane 2 < YZ-Right
Plane 3 < XZ-Front
FEA Object Groups
Parts
Part 1 < crod 1 >
Part 2 < crod 2 >
Meshes
Coordinate Systems
Contact (Default: Bon
ALGOR
Meshing Progress
Information
Processing mesh
Part 2 of 2
Progress for part
Details >>
Stop
0.000
46.827
mm
93.655
140.482

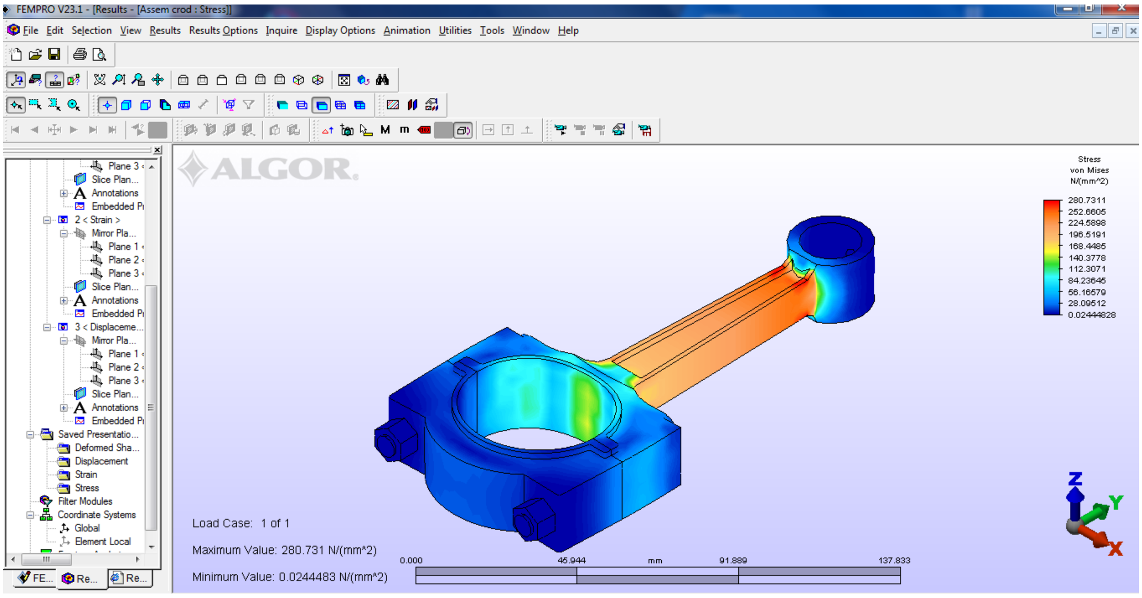

FEMPRO V23.1 - [FEA Editor - [Assem crod.fem]]
File Edit Selection View Mesh Geometry Analysis Tools Window Help
1 < Design Scenario 1 >
Analysis Type < Static
Planes
Parts
Part 1 < crod 1 >
Part 2 < crod 2 >
Meshes
Coordinate Systems
Contact (Default: Bon
ALGOR
0.000
46.827
mm
93.655
140.482

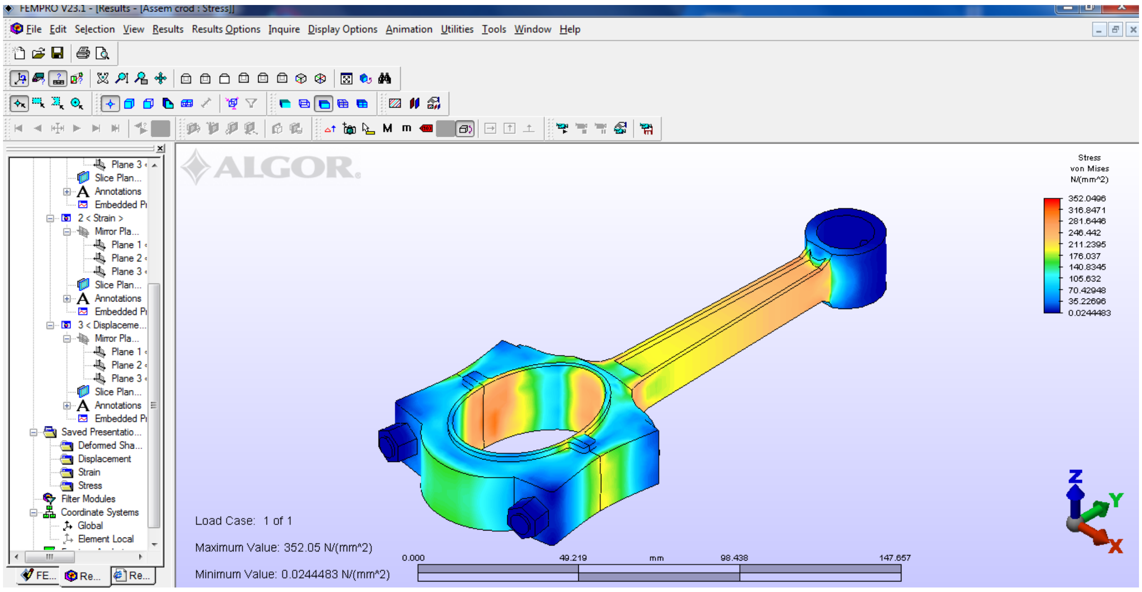

FEMPRO V23.1 - [FEA Editor - [Assem crod.fem]]
File Edit Selection View Mesh Geometry Analysis Tools Window Help
ALGOR
Creating 2 Surface Boundary Condition Objects
Constrained DOFs
Tx
Ty
Tz
Rx
Ry
Rz
Predefined
Fixed
X Symmetry
X Antisymmetric
Free
Y Symmetry
Y Antisymmetric
Pinned
Z Symmetry
Z Antisymmetric
No Rotation
Coordinate System:
Global (Default)
Description
OK
Cancel
Surface Boundary Conditions...
Surface Displacement Boundaries...
Surface Elastic Boundaries...
Surface Rigid Boundaries...
Surface Hydrostatic Pressures...
Surface Forces...
Surface Pressure/Tractions...
Surface Variable Loads...
Surface Temperatures...
Surface Voltages...
Show
Hide
Inquire
Add
Create Remote Load...
Coordinate Systems
CAD Mesh Options
Contact
Cut
Copy
Paste
Delete
Filter On
Filter Off
Move
Project...
Select Subentities


Creating 1 Surface Force Object
Magnitude
24000
N
Direction
Normal
X
Y
Z
Custom
Vector Selector...
Description
OK
Cancel
Rename
Show
Hide
Inquire
Add
Create Remote Load...
Coordinate Systems
CAD Mesh Options
Cut
Copy
Paste
Delete
Filter On
Filter Off
Move
Project...
Select Subentities
Surface Boundary Condition...
Surface Displacement Boundary...
Surface Elastic Boundary...
Surface Rigid Boundary...
Surface Hydrostatic Pressure...
Surface Force...
Surface Pressure/Traction...
Surface Variable Load...
Surface Temperature...
Surface Voltage...
1 < Design Scenario 1 >
Analysis Type < Static
Planes
Plane 1 < XY-Top
Plane 2 < YZ-Right
Plane 3 < XZ-Front
FEA Object Groups
1 < Surface Bound
Parts
Part 1 < crod 1 >
Part 2 < crod 2 >
Meshes
Coordinate Systems
Contact (Default: Bonc
0.000
46.827
mm
93.655
140.482


FEMPRO V23.1 - [FEA Editor - [Assem crod.fem]]
File Edit Selection View Mesh Geometry Analysis Tools Window Help
1 < Design Scenario 1 >
Analysis Type < Static
Planes
Plane 1 < XY-Top
Plane 2 < YZ-Right
Plane 3 < XZ-Front
FEA Object Groups
1 < Surface Bound
2 < Surface Forces
Parts
Part 1 < crod 1 >
Part 2 < crod 2 >
Meshes
Coordinate Systems
Contact (Default: Bonc
ALGOR.
0.000
46.827
mm
93.655
140.482


FEMPRO V23.1 - [FEA Editor - [Assem crod.fem]]
File Edit Selection View Mesh Geometry Analysis Tools Window Help
Structural - Static Stress with Linear Material Models
Analysis Description
Structural
Elapsed Time: 00:00:03
Analyzing:
Verifying model
Details >>
Schedule
Stop
Done
1 < Design Scenario 1 >
Analysis Type < Static
Planes
FEA Object Groups
Parts
Meshes
Coordinate Systems
Contact (Default: Bonc
0.000
46.827
mm
93.655
140.482


ALGOR
Stress
von Mises
N/(mm^2)
280.7311
252.6605
224.5898
196.5191
168.4485
140.3778
112.3071
84.23645
56.16579
28.09512
0.02444828
Load Case: 1 of 1
Maximum Value: 280.731 N/(mm^2)
Minimum Value: 0.0244483 N/(mm^2)
0.000
45.944
mm
91.889
137.833


ALGOR
Stress
von Mises
N/(mm^2)
352.0496
316.8471
281.6446
246.442
211.2395
176.037
140.8345
105.632
70.42948
35.22696
0.0244483
Load Case: 1 of 1
Maximum Value: 352.05 N/(mm^2)
Minimum Value: 0.0244483 N/(mm^2)
0.000
49.219
mm
98.438
147.657