DESIGN OF THE DETACHABLE HUB FOR TURBINE BLADE COOLING SYSTEM

By

MOHAMAD NURHALIM BIN MOHAMAD SAYUDI

Dissertation submitted in partial fulfillment of
the requirements for the
Bachelor of Engineering (Hons)
(Mechanical Engineering)

May 2011

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

DESIGN OF THE DETACHABLE HUB FOR TURBINE BLADE COOLING SYSTEM

By

Mohamad Nurhalim Bin Mohamad Sayudi

A project dissertation submitted to the
Mechanical Engineering Programmee
Universiti Teknologi PETRONAS
in partial fulfillment of the requirement for the
BACHELOR OF ENGINEERING (hons)
(MECHANICAL ENGINEERING)

Approved by:

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

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK
May 2011

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the originality 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.

M. NURHALIM BIN M. SAYUDI

ABSTRACT

This report basically presents the research that had been done based on the chosen topic, which is **Design of the Detachable Hub for Turbine Blade Cooling**. During the time given, research had been done regarding the patented design of hub blade to get the idea of how the hub of turbine blade look likes. In addition, the hub should have the air passage to allow the compressed air to flow through it and exit to the tip of blades. The objective of this project is to design the detachable hub in compliance with all the important factors that already been considered such as engineering requirement and fabrication. The aim of this project is to design the suitable detachable hub for the turbine blade for the test rig. After the design is done, the project also attempts to investigate the hub deformation profile and temperature distribution. The scope of this project covers the planning and designing, material specification, 3D design drawing and the stress distribution and deformation of the hub and blade subjected to a certain loading condition. Throughout this project, Inventor Professional 2009 had been used to design the hub because it more easy and already integrate with the ANSYS simulation software and so investigation of the stress analysis and deformation profile analysis can be done simultaneously. The number of bolt needed has been decide and the bolted joint to couple the hub with the shaft also considered in this project. Material chose is the Carbon steel Ansi 1045. Analysis regarding the deformation happen at the bolted joint, coupler and the fir-tree root blade as well as the thermal distribution happen on the hub also has been investigate. As the conclusion, the analysis done shows the hub design and material choose is fulfill the criteria needed and accepted to be the detachable hub for the internal cooling gas turbine test rig.

ACKNOWLEDGEMENT

I have taken efforts in this project. However, it would not have been possible without the kind support and help of many individuals and organizations. I would like to extend my sincere thanks to all of them.

I am highly indebted to **Mr. Ir. Idris B. Ibrahim** for his guidance and constant supervision as well as for providing necessary information regarding the project & also for their support in completing the project.

I would like to express my gratitude towards a certain individual & members from Mechanical Department namely **Mr Zailan** and **Mr Hafiz** of for their kind co-operation and encouragement which help me in completion of this project.

My thanks and appreciations also go to my colleague in developing the project and people who have willingly helped me out with their abilities.

I would also extend my heartfelt thanks to my family and well wishers. Who without their prayers and help, this project would have been a distant reality to be completed.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1
INTRODUCTION

1.1 Project background

As is known, generally gas turbine engines comprise a compressor section, a combustion chamber, and a turbine section. In general, the compressor section draws in air and compresses it. Fuel is then added to the compressed air in the combustion chamber, and the mixed fluid of fuel and compressed air is ignited. The fluid, which is at temperature in the range of about 1700-2600 Fahrenheit after ignition, is directed toward the turbine section where part of the energy in the fluid is extracted by the turbine blades which are mounted to a rotatable shaft. The rotating shaft in turn drives a compressor in the compressor section. The remainder of the energy is used for other functions: for example, the propulsive thrust of a jet craft. [1]

The gas turbine is a primary energy deliver not only for vehicular propulsion of such as air, land and water, but also for power generation. Several major factors affect thermal efficiency or specific fuel consumption of a gas turbine plant. These include:

- Increase in the turbine inlet temperature, namely firing temperature.
- Reduction of cooling air usage.
- Improving component efficiency.
- Cycle enhancement.

The thermal performance and specific thrust of gas turbines can only be improved significantly by increasing the turbine inlet temperature. [2] This approach is limited, however, by the availability of appropriate materials that withstand designed temperature. The higher temperature has jeopardized the integrity of the high-pressure turbine components and especially the turbine blades.

To overcome and prevent the failures of turbine blades in gas turbine engine resulting from these excessive operating, the film cooling solution has been incorporated into blades design. In film cooling, cool air from the compressor stage is ducted into the internal chambers of turbine blades, and discharged through small holes in the blades wall. This will provides a thin layer, cool, insulating blanket along the external surface of the turbine blade.

For this purpose, the internal cooling method using thin film is being studied in UTP by designing and fabricating the test rig which is capable of providing the experimental data and results. As there already many researches that had been done for the ideal blade material that used in the gas turbine and their related properties such as the stress distribution, blade deformation and the heat profile movement, this project is focusing on the hub that will be used for the designated test rig by the previous final year student. This project will be focus on the planning, designing and provide drawing, selection of material as to reduce the manufacturing cost, the stress distribution at the root slotting area, and the appropriate size of bolt that will be use that can prevent the crack before rotating the shaft and also provide the appropriate Finite Element Analysis to support the choose material.

1.2 Problem statement

Due to the absence of real gas turbine engine in UTP, the analysis of heat movement, temperature profile and the stress distribution cannot be done. So this project is about to build and develop the test rig which closely simulate the working environment of the real gas turbine engine so that we can study on it. As the test rig not actually limited to only study the cooling blade system, it also could be used more efficiently in the mechanical and energy field in investigating the behavior of gas turbine. It also can be used for other studies as it can be improved in the near future.

Design of the test rig has been partially completed by the previous final year student. But there is some part which is needed to improvise more and in this project, the author is focusing on the design of the hub blade for the test rig. The hub should meet the entire requirement needed in terms of engineering, economically cost and possibility of the fabrication. This is important because in terms of engineering, the hub should be running in the closed condition such as the actual gas turbine engine so that all the heat movement, temperature profile and the distribution of load will not be affected by other factors and can be determine exactly like the actual condition. In terms of economically cost, as this project is for academic purposes and not for gaining profit in return, the author should carefully consider the material that we will used for the hub. The design of the hub also should be easy to fabricate as we all know the more complicate the design, the higher the cost it will be.

The design also need to be analyzed using the Finite Element Analysis software so that all the information such as the mechanical damages, high temperature damage and exposure, and the creep and fatigue failures can be determine first before the design can be decide and move to further process such as fabrication

1.3 Objective and scope of study

The objective of the project:

1. To design the detachable hub turbine blade for the test rig.
2. To investigate the characteristics of the hub subjected to certain loading conditions by doing the finite element analysis.

The scope of this project covers

1. Planning and designing the hub.
2. Materials specifications.
3. Development of the 3D – engineering drawing.
4. Profile of stress distribution and deformation of the hub subjected to certain loading conditions.

CHAPTER 2
LITERATURE REVIEW

2.1 Introductions

The major function of the turbine is to extract energy from the hot gas flow to drive compressor, generator, and the accessory gearbox. Gas turbine disc work mostly at high temperature gradients and are subjected to high rotational velocity. High speed result in large centrifugal forces in discs and simultaneous high temperature reduces disc material strength.

The service life of critical gas turbine components is governed by the modes of degradation and failures such as: fatigue, fracture, yielding, creep, corrosion, erosion, wear, etc. Gas turbine disc are usually the most critical engine components, which must endure substantial mechanical and thermal loading. If a problem arises in the turbine section it will significantly affect the whole engine function and of course, safety of the gas turbine system. Blade loss can be contained within the engine casing, while the catastrophic failure of a turbine wheel could cause puncture of the engine casing by larger fragments of the disc. This kind of failure is often concerned with over speeding of the disc.

Aero engine turbine disc basically have three critical regions on which attention should be focused: the dovetail-rim area (fir-tree slots, serration fitting), the assembly holes and the hub zone. The joint between the turbine blade and the disc usually represents the most critical area from the point of view of the static and fatigue approaches. The loads associated with these regions are mainly the centrifugal forces and thermal stresses. [3]

The stress and failure analysis of the turbine engine has received the attention of several investigations. The problem of numerical evaluation of stress state in dovetail-rim area of disc and blade is described by Chan et al. [4] Meguid et al. [5] and Zboinski. [6]. In this study, attention is devoted to a analysis of the damage mechanism of the turbine disc subjected to both operational and over speed conditions and also to indicate critical areas, from the point of view of the stress analysis. The additional goal of this analysis is to improve the safety and reliability of the aircraft and different planes, powered by the same type of engine.

2.2 Detachable hub disc blades

Hub disc of gas turbine are subjected to a complex mechanical, thermal and chemical load during services. Life assessment of the hub disc is focused on the connection between blade and disc. High stress zone were found at the region of lower fir-tree slot, where the failure occurred. In particular, the disc may be damaged considerably due to fatigue and creep-fatigue loadings, which probably cause elastic-plastic deformation. [7]

Attention of this study is devoted to the mechanisms of damage of the turbine disc and also the critical high stress. As the design of hub disc blade finish, it will go through all the necessary experiment and Finite Element Analysis method before accepted for the test rig.

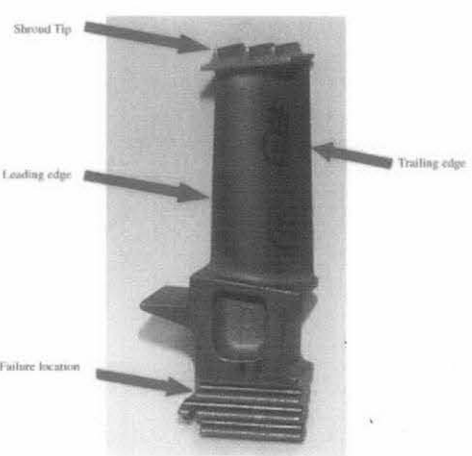


Figure 1.1: Example of turbine blade

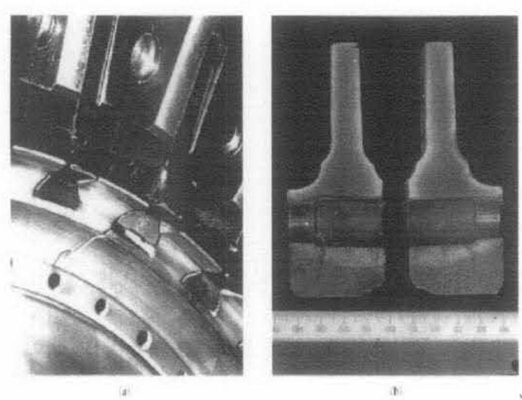


Figure 1.2: Crack propagation from the mounting slot of blade

The turbine disk is one of the most vital components of modern aero-engines. A typical loading spectrum experienced by an engine disk is characterized by low-frequency stress cycling resulting primarily from centrifugal forces, superposing dwell at peak load at high temperatures during aircraft's taking off and climbing. The service life of the turbine disks at elevated temperatures is mainly governed by the modes of failure at high temperature. [8]

Even if the design is strictly controlled, the structure during production and operation has significant scatter in dimensions; material properties and loading that degrade its safety and quality. Traditional design methods generally account for these uncertainties using experience-based design safety margins, which often leads to a conservative component. In contrast to deterministic design, a probabilistic assessment can quantify risk and thus identify areas of possible overdesign (conservatism). Moreover, the modes of failure for the new materials which potentially provide quantum improvements over conventional metallic alloys in some properties (strength, density, temperature capability) are not addressed by current design practices. Therefore, the need for cost-effective designs has resulted in the development of probabilistic design to quantify the effects of these uncertainties so as to improve the reliability of the component. [8]

The use of light alloys for the high temperature sections of the engine is not feasible since they cannot generally be designed to give acceptable creep properties at the high temperature needed for efficient turbine operation. In the case of aluminium alloys, the operating temperature is above the melting point. For the most part, nickel base alloys are used and the weight penalty is accepted. The use of hollow blades, sometimes with air ducted through the interior for cooling, reduced blade weight. The most common materials for turbine blade manufacture are nickel-based "super-alloy" materials. This alloy is able to withstand the very aggressive environment of high temperature and high stress found within the hot gas path of a turbine engine. Nickel-based is considered as a most suitable material for gas turbine blade since with aluminium additions, Al_2O_3, it will form a surface layer which is highly oxidation-resistant at very high temperature. [9]

In high performance gas turbine engines, the temperature of the hot gas stream generated exceeds the operating temperature capability of any practical material from which the turbine vanes, disc and blades could be fabricated. In order to reduce metal temperatures to a point where sufficient strength is maintained, it has become an accepted practice to duct lower temperature, pressurized air from the engine's compressor to these turbine components which operate in the hot gas stream environment.

The cooling air thus derived has been employed in various ways to reduce the metal temperatures of such components. Basically, there are two cooling mechanism generally involved as cooling air is introduced into hollow blade and vane and then discharged into the hot stream. The flow of cooler air through the hollow structures (or passageways therein) reduces the metal temperature by convection. The second cooling mechanism may be obtained by discharging the cooling air from a plurality of points around the surface of the hollow body to form a thin layer of air on its exterior surface. This mechanism, known as film cooling, further reduces metal temperatures permitting operation in higher temperature gas stream. [10]

CHAPTER 3
METHODOLOGY

3.1 Methodology

Studying and designing a system to manage blade cooling system is not a new topic. As reported from NASA there are hundreds of patent and simulation exist in industrial and academic institute focusing on blade cooling. Lots of patented system and advanced technology have been tried and used to improve the turbine blade cooling system which some of these are successful but involving and leading to the very high cost of money. Same goes to the disc hub of the blade. The hub actually one of the major critical part that we need to take into consideration because it is also subjected to a complex mission load cycles history.

The author of this work have focused his effort on designing a model of detachable hub gas turbine engine which will provide the placement of the blades in the end of the main drive shaft and rotating in high temperature condition. The detachable hub design is focus on to mount the 3 turbine blades total 10 kg in weight, as well as the position of the bolt so that the compress air will not leak out. The material are also taken into consideration because it will rotate in 650°C hot air housing, which are attached to a rotating shaft drive by electric motor.

In order to provide the detachable hub specification, the following steps and process need to be done:

- Finalized plan, design and details drawing in case of fabrication.
- Analysis on the forces distribution , stress and allowable load on the components
- Analysis on the safety and economic criteria

Continue from FYP I, FYP II focusing on the analysis that need to be done for this project.

Until this point of study, the critical analysis has been determined. Finite Element Analysis need to be done at three crucial areas which are:

1. Bolt
2. Coupler
3. Hub

3.1.1 Bolt

Bolt connection experienced crucial static loading due to the weight of the clamped components. Cracking of bolt before rotating gas turbine disc should be avoided to prevent any accidents. For static loading, the parameter involved is only the weight of all the clamped components. Total weight of the components calculated and used in the Inventor Professional 2009 Simulation software as the value of load.

Using this software;

1. 3D drawing design is finalized.
2. Mode of analysis is chosen which is the Static Loading Analysis.
3. In Static Loading Analysis mode, the value such as mass, volume and moment of inertia is inserted.
4. Various kinds of materials can be choosing to easiest the analysis.
5. Result of analysis such as deformation and maximum stress area can be obtained.

3.1.2 Coupler

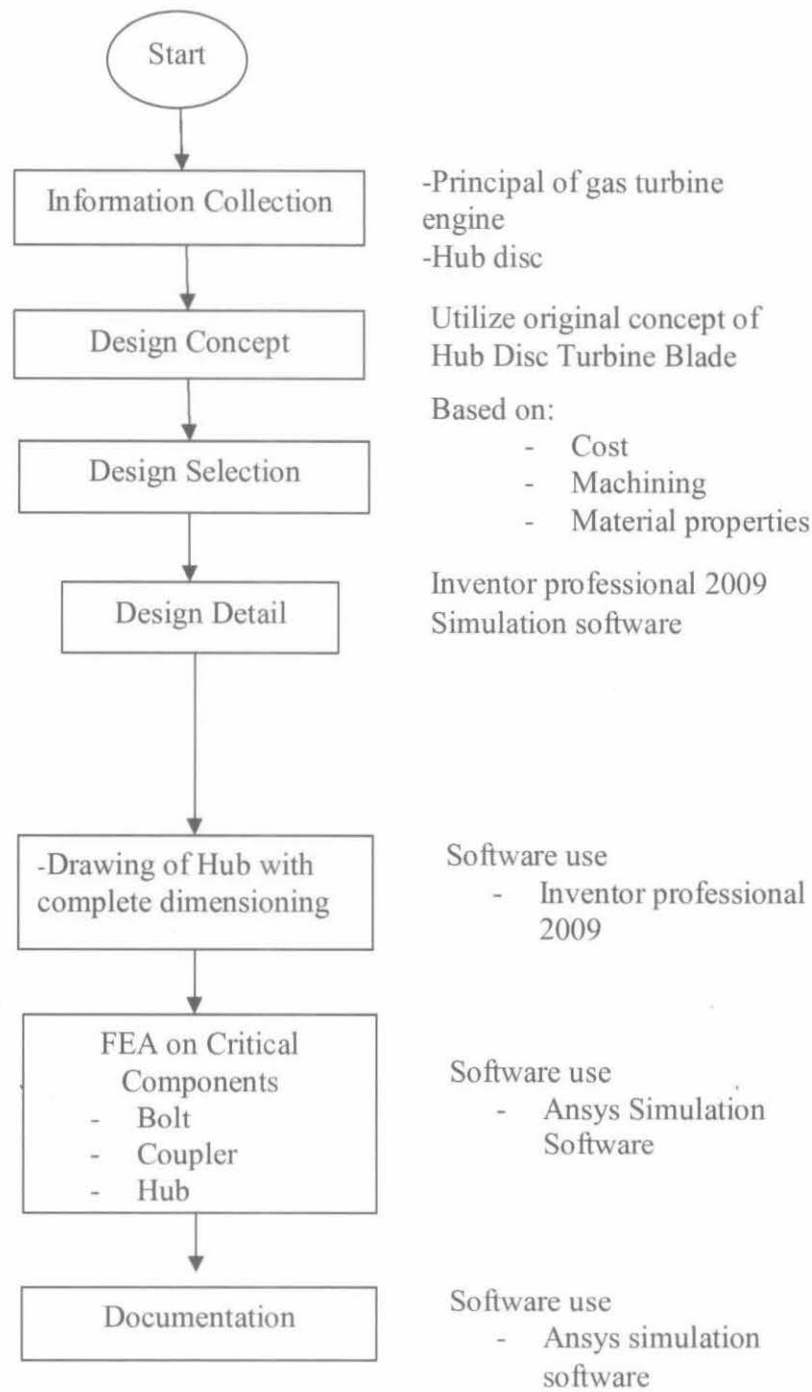
Same as the bolt, analysis of the coupler also can be done using the Inventor 2009 software. The steps taken are most likely same such as bolt analysis but the mode of analysis taken is different. For this project, the coupler need to support the weight of all components from its support point and in this case at fixed bearing point. So the coupler will experience bending moment.

3.1.3 Hub

The case of the hub is different because the hub is subjected also with the thermal load. Only the hub is analyzed assuming the blade design is ideal. While rotating, the hub will experience the centrifugal force due to weight of the three blades that also rotating with different load as function of different distance from centre. And at the same time, the hub also subjected to thermal load as high as 650 □C. As for the centrifugal load, the deformation and the maximum stress area, is obtained using the Inventor 2009 software. As for thermal distribution, the Ansys simulation software version 11 is used to obtained the result. 3D drawing of the hub is done using the Inventor and then imported to the Ansys. Before that, the heat transfer coefficient have to be find as we are dealing with force convection for the internal cooling and hot gases. After finding the heat transfer coefficient;

1. Import the 3D drawing from Inventor into Ansys.
2. Steady State Thermal Analysis mode is selected.
3. Surface that subjected to convection process is selected. In this case, force convection internal flow happen inside the hollow shaft and force convection external flow happen at the outer surface as it functioning such as combustion chamber.
4. The simulation is run after the temperature and heat transfer coefficient value is inserted.
5. Result is obtained.

3.2 Flow Chart of Detachable Hub Development



CHAPTER 4
RESULT AND DISCUSSION

The methodology used in designing the detachable hub for internal cooling blade turbine for this project is possibly to really solve the problem for the gas turbine test rig develops by the previous final year problem as all aspect of analysis for the turbine to be accepted already been studied. In addition, the designing criteria is also based on the most economical, safety and quality as the hub is subjected to running as an actual condition of working gas turbine.

As to get the accurate data for future studies, the hub gone through the finite element analysis before the design can be accepted. The material used which is Carbon Steel AISI 1045 also fulfills the requirement for the test rig. As for the heat transfer calculation, the value of Force Convection for internal flow is quite big because the value used as the mass flow rate is big and the diameter of the small hollow shaft that ducting the compressed air for the internal cooling is small. So this affected the value of the heat transfer. All calculation regarding the force convection heat transfer calculation for internal and external can be referring to Kirk D. Hagen (1999). [18]

In addition, the software used for this project was Ansys Simulation Software Version 11; use to do the finite element analysis regarding the thermal effect on the hub and Inventor Professional 2009 for all the related design of the 3D drawing. This software is selected because there are from the same company and already integrated by each other. The menu for 3D drawing is better in Inventor than Ansys. The drawing file format from the Inventor can be imported to Ansys for the finite element analysis since Ansys have better and user friendly for the analysis to be performed. For the finite element analysis part, result show the area where by most stress and deformation will concentrate.

Red area always marked as the most concentrated stress area can be too exaggerated so as can be confusing to the user. The result shown is exaggerate and only to show the most critical stress concentration area and the important is to examine the value that result from the analysis.

Followed by are the results that get from the performed analysis. Based on the maximum allowable yield stress criteria of Carbon Steel, all the perform analysis show no problem as the value of the result is all under the allowable stress. The design also shows significantly small deformation when subjected to centrifugal force and thermal force. The yield strength of Carbon Steel is 350MPa and the result shown after running the simulation show when the hub and bolt subjected to the load, the higher load is not even higher than 100MPa. It show Carbon Steel clearly is safe enough to be the material of the hub. In addition, the test rig design is not to running continuously. This will preserve the integrity of the hub. And at last the design show no problem after being done with the finite element analysis.

Last but not least, even though the fabrication process of the hub is not in the objective, the 3D drawing for the hub design is already been done and submit to the Mechanical Lab Technician in UTP. The hub is possible to fabricate using the Wire Electrical Discharge Machining available in the UTP. All the designing and fabrication of the test rig can be fabricated in UTP for future study.

4.1 Measurement and dimensioning of detachable hub

To produce the hub design, the complex shaped of the root area of turbine blade is measured using the Digitizer Machine available in UTP mechanical building. Problem occurred due to Digitizer Machine produce 2dimension drawing, while the need is 3drawing is solved using the Inventor Professional 2009 software. As the result, complete 3D drawing and dimension can be obtained and produce for the hub.

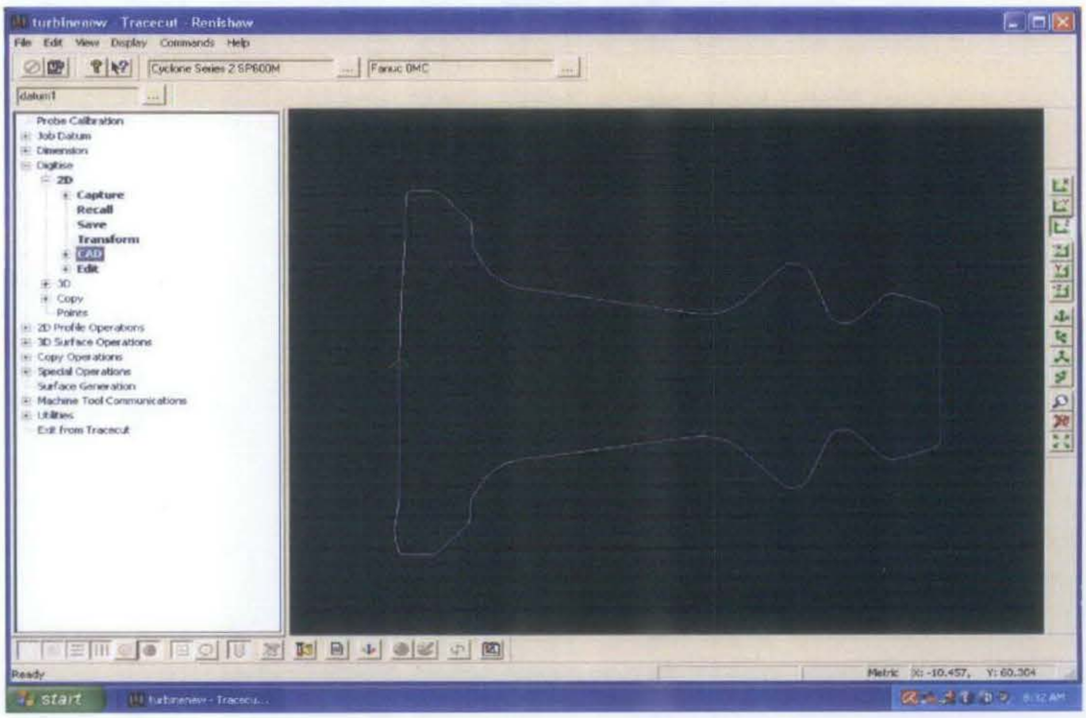


Figure 4.1 : Measurement of Turbine Root using Digitizer Machine

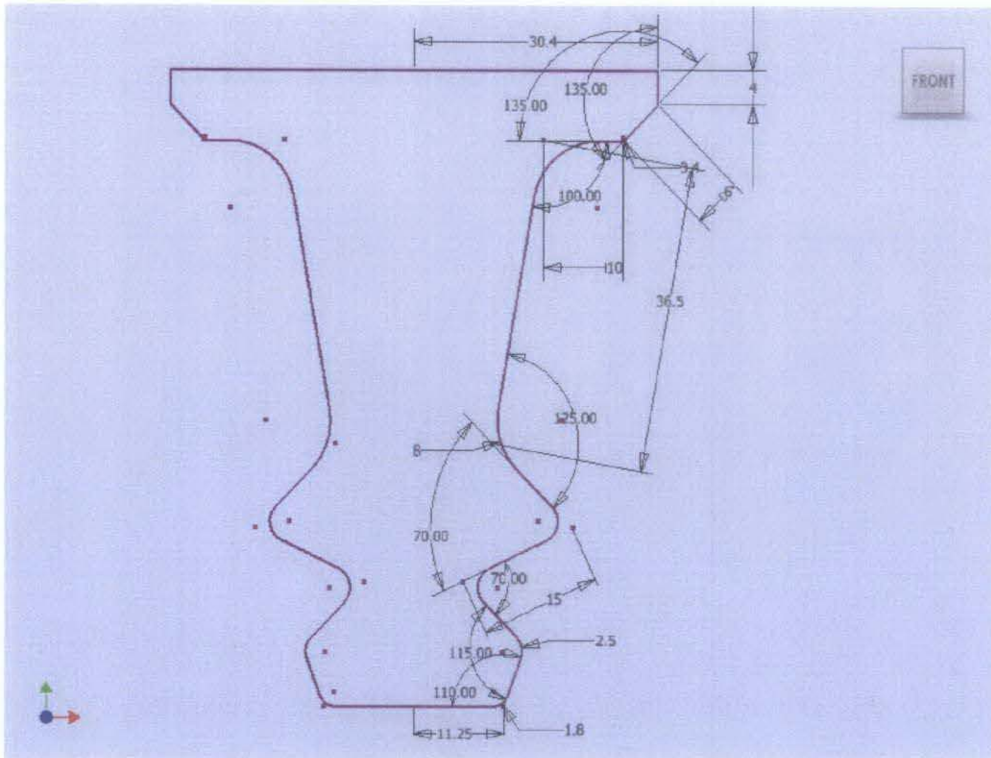


Figure 4.2: Blade root dimension using Inventor Professional 2009

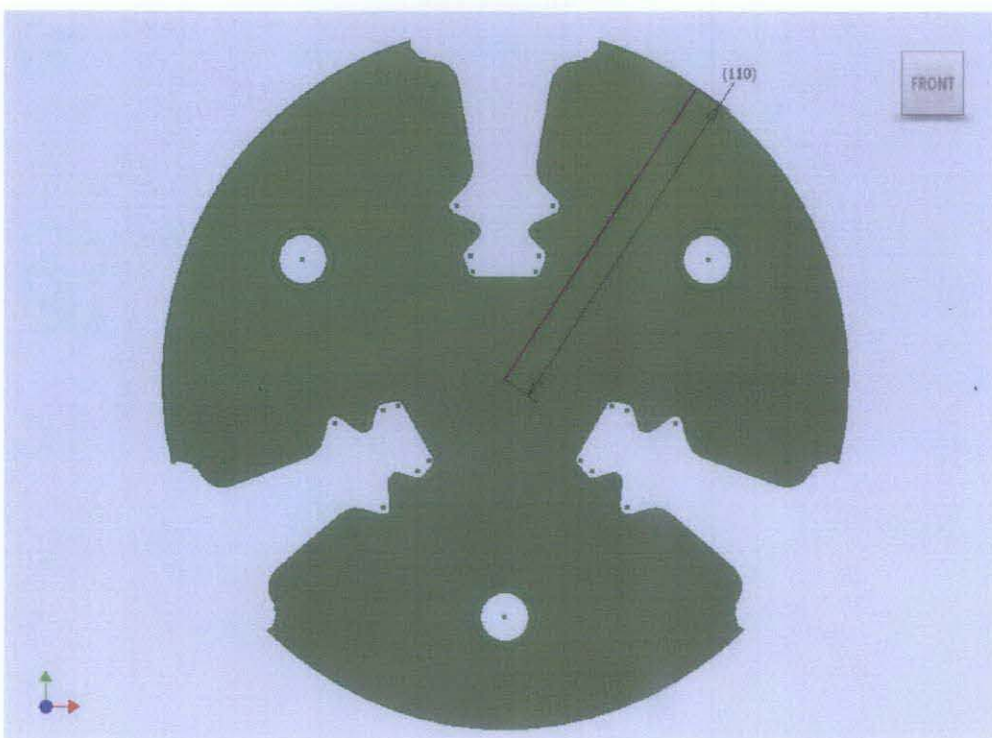


Figure 4.3: Hub dimension

4.2 Development of 3D-engineering drawing design

From the Digitizer Machine, the complete dimension of the blade can be obtained. Using the dimension of the root part, the hub is produced completely together with the complete dimension. Using the Inventor Professional software, the 3D drawing of all related parts and components is produced so that imaginary end product can be obtained. The dimension of all parts and design also is finalized.

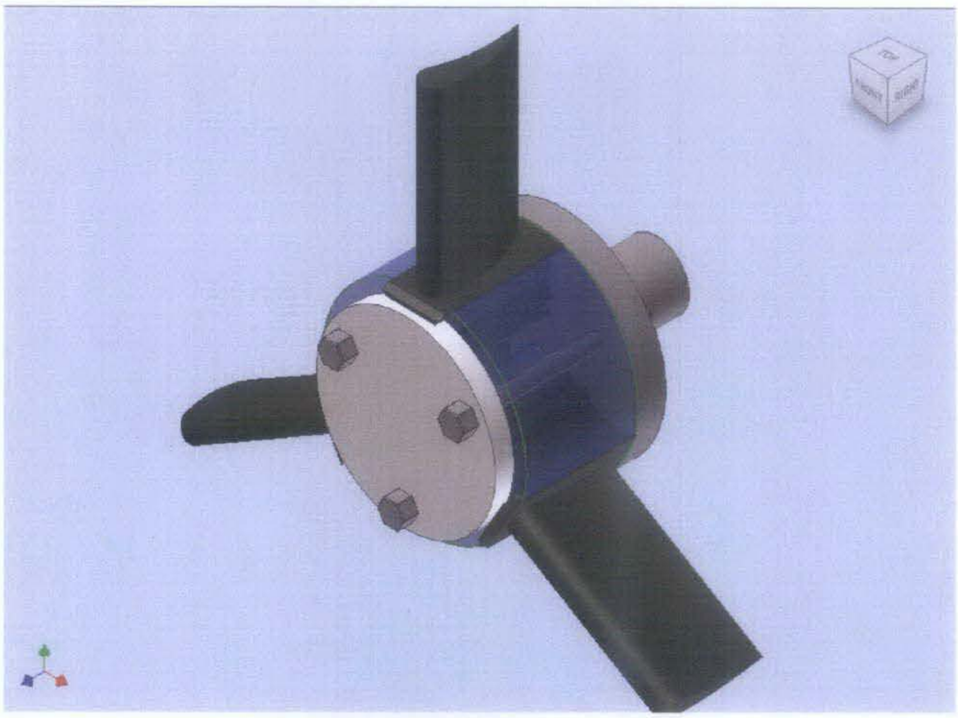
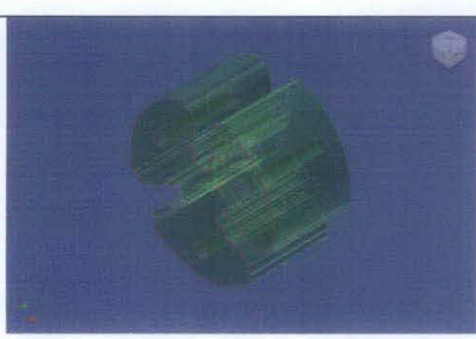
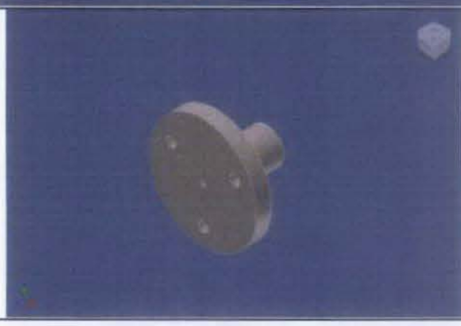
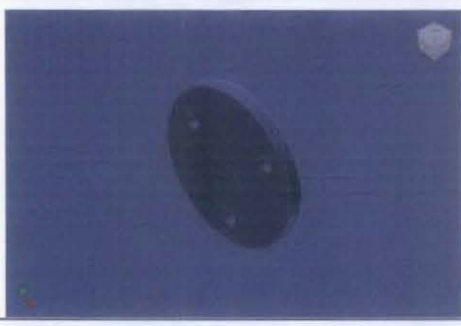
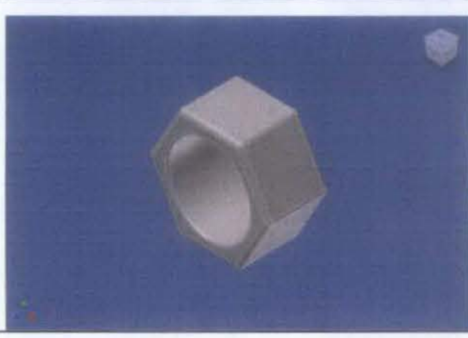


Figure 4.4: Finalized design of detachable hub with related parts

Table 4.1: Related parts of detachable

	**Detachable Hub** Material : Carbon steel Diameter : 220mm Length : 120mm Inner Diameter : 10mm Diameter of bolt : 16mm Weight : 28.4kg Quantity : 1

	**Turbine Blade** Material : Titanium Alloy Height : 275mm Length(root) : 120mm Wide : 60mm Weight : 3.62kg Quantity : 3
	**Coupler** Material : Carbon Steel Coupler diameter : 220mm Shaft diameter : 80mm Inner hollow diameter : 10mm Quantity : 1
	**Hub Cover** Material : Carbon Steel Diameter : 193mm Width : 20mm Quantity : 1
	**Bolt** Property Class : 8.8 (medium carbon, Q&T) M16 x 1.5 x 210 Quantity : 3
	**Hexagonal Nut** M16 x 1.0 x 10 Quantity : 3

4.3 Material Selection

1. Until this point of study, for the selected material, currently I am using the Carbon Steel as the material.
2. This is because the material has high yield strength and the high temperature of melting to sustain the torsion and thermal load.

Table 4.2: Properties of Carbon Steel AISI1045 [11]

Mechanical Properties

Properties		Conditions	
		T (°C)	Treatment
Density (×1000 kg/m^3)	7.7-8.03	25	
Poisson's Ratio	0.27-0.30	25	
Elastic Modulus (GPa)	190-210	25	
Tensile Strength (Mpa)	585	25	cold drawn, annealed (round bar (16-22 mm)) more
Yield Strength (Mpa)	505		
Elongation (%)	12		
Reduction in Area (%)	45		
Hardness (HB)	170	25	cold drawn, annealed (round bar (19-32 mm)) more

Thermal Properties

Properties		Conditions	
		T (°C)	Treatment
Thermal Expansion (10^{-6}/°C)	15.1	0-700 more	annealed

4.4 Cost of material

Table 4.3: Cost comparison of Carbon Steel AISI 1045 and Titanium Alloy [12]

Material/condition	Cost ($ USD/Tonne)	Relative Cost
Carbon Steel 1045	864.867	1.0
Titanium Alloy	26736.03	17.4

Based on Table 2, the cost of titanium alloy per tonne is much higher than cost of carbon steel per tonne. The Carbon Steel AISI 1045 is selected because it have the cost is much lower while meet the requirement for the gas turbine test rig condition This is not including the fabrication and manufacturing cost.

Table 4.4: melting point for some metals and alloys [13]

Metal	Melting Point	
	(^oC)	(^oF)
Sodium	97.83	208
Carbon Steel	1425 - 1540	2600 - 2800
Stainless Steel	1510	2750
Tantalum	2980	5400
Thorium	1750	3180
Tin	232	449.4
Titanium	1670	3040

Based on table 3, it shows that Carbone Steel have melting point in the range of 1425-1540 degree Celsius. The requirement for the gas turbine test rig is 650 degree Celsius. This clearly showed that Carbon Steel AISI 1045 can withstand the working condition of 650 degree Celsius for the gas turbine test rig.

4.5 Possibility of manufacturing and fabrication

1. The detachable hub is possible to be fabricating using the Wire Electrical Discharge Machining available in UTP.
2. The Wire EDM work with AutoCAD 2004 drawing format as the inserted command.
3. Using the Inventor Professional 2009, the original format of 3D drawing can be converted into the AutoCAD 2004 format drawing.
4. Price of the Carbon Steel is less expensive than other steel type and it still can serve it purpose within the range as actual condition.
5. In addition, since the hub can be fabricating using the Wire EDM in UTP, the fabrication cost can be avoided.

4.6 Torque Calculation

For this project, the value of torque provided by the compressor motor need to be determined as it will be the value of load that the bolts need to withstand when it start to rotate. As for the bolt, the stress analysis regarding the force and moment is done. First of all, the torque of the motor is calculated such as below;

Motor Torque [14]

Rpm: 1500rpm

Power: 10 kW

Max torque: 100N.m

Using the formulae, Torque = $9.55 \frac{H}{n}$

Where n = rpm

H = Power, kW

We get, T = $\frac{9.55 \times 10000\ W}{1500}$

= 63.67N.m

= 63670N.mm

As for this project, the most critical and crucial analysis that must be done has been determined. There are three areas that need to be focused on which are:

1. Bolt
2. Coupler
3. Hub

4.7 Analysis of bolt

From the figures below, the blades, hub and hub cover is attached to the coupler by using the bolt. So, the bolt will experience the force due to the weight of the related components. The bolt will experience deformation due to the weight and at the same time will experience moment stress while rotating.

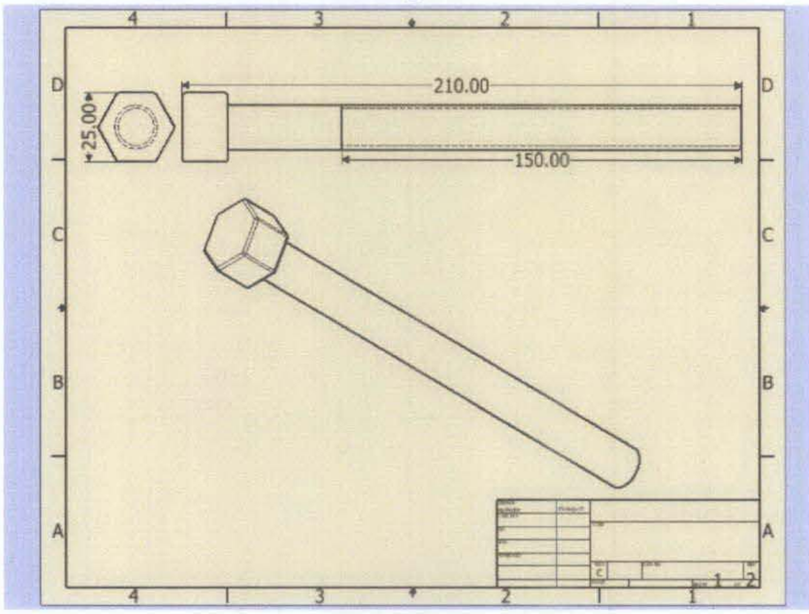


Figure 4.5: Section view of bolt

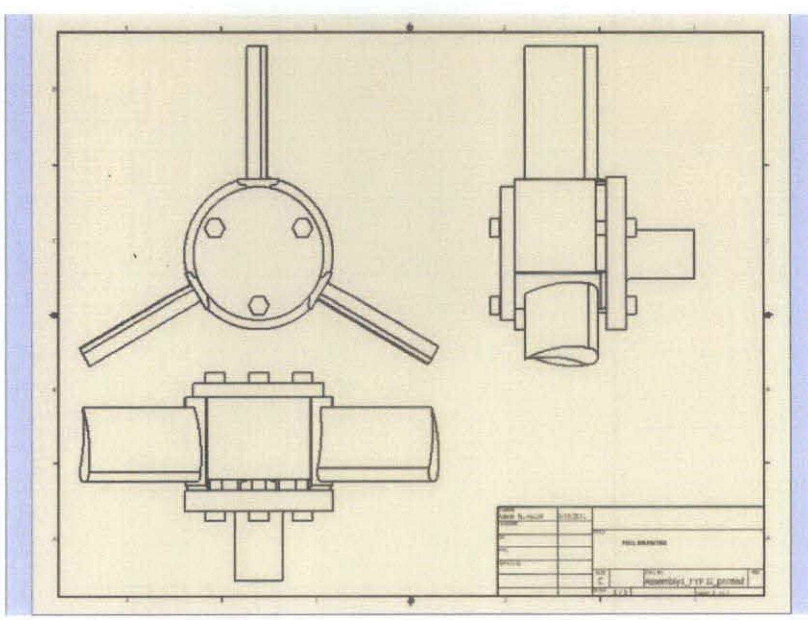


Figure 4.6: Section view of hub and parts

Figure 4.7: Meshing process done on the bolt

Bearing load on the bolt,

Using the Inventor 2009 software;

Total mass = $Mass_{blade}$ + $Mass_{hub}$ + $Mass_{cover}$ + $Mass_{nut}$ + $Mass_{bolt}$

= (3.62kg x 3)+(28.4kg)+(5.8kg)+(0.032 x 3)+(0.551 x 3)

= 46.8kg

= (46.8kg x 10)/3 [have 3 bolt clamping components]

= 156N each

This value will be used in simulation as the force value that the bolts need to withstand in static loading.

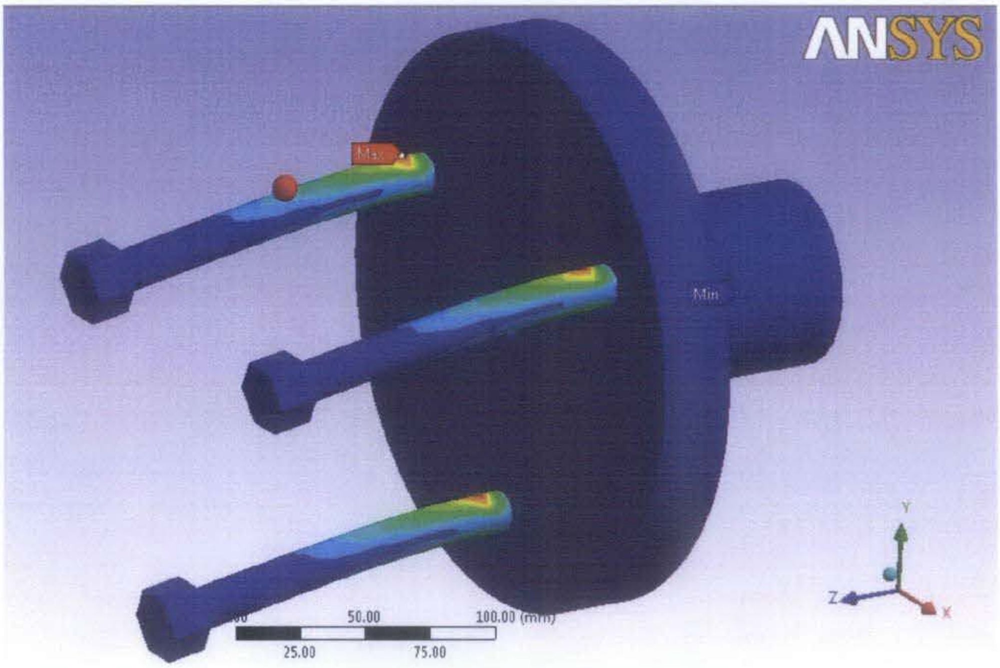


Figure 4.8: Area of maximum stress on the bolt from bearing load analysis

Table 4.5: properties of Carbon steel in Inventor 2009 software

Carbon Steel

Young's Modulus	2.e+005 MPa
Poisson's Ratio	0.29
Mass Density	7.87e-006 kg/mm³
Tensile Yield Strength	350.0 MPa
Tensile Ultimate Strength	420.0 MPa

Table 4.6 : Result from bearing load analysis on the bolt

Structural Results

Name	Minimum	Maximum
Equivalent Stress	3.401e-007 MPa	2.88 MPa
Maximum Principal Stress	-0.4586 MPa	2.285 MPa
Minimum Principal Stress	-2.289 MPa	0.478 MPa
Deformation	0.0 mm	9.649e-004 mm
Safety Factor	15.0	N/A

Table 4 showed the properties of the Carbon Steel which already been integrated in the software. The material for the bolt is also used the Carbon Steel. There are also other various types of steels available in the software. All the simulation regarding loads and stresses analysis is depending on these properties. At the start of the simulation, there are menu for users to select the appropriate material so this easy the user and no need to insert the value of material properties manually.

Based on the result in table 5, the maximum value of stress happen on the bolt is 2.285MPa and the deformation is 9.649e-004 mm. this value is within the allowable stress for the Carbon Steel which is in the range of 350MPa.

Shear stress develops in the bolt;

By using the torque of the motor compressor, the shear stress in the bolt is analyzed and determines whether the bolt can withstand the torque from the compressor when the shaft starts to rotate.

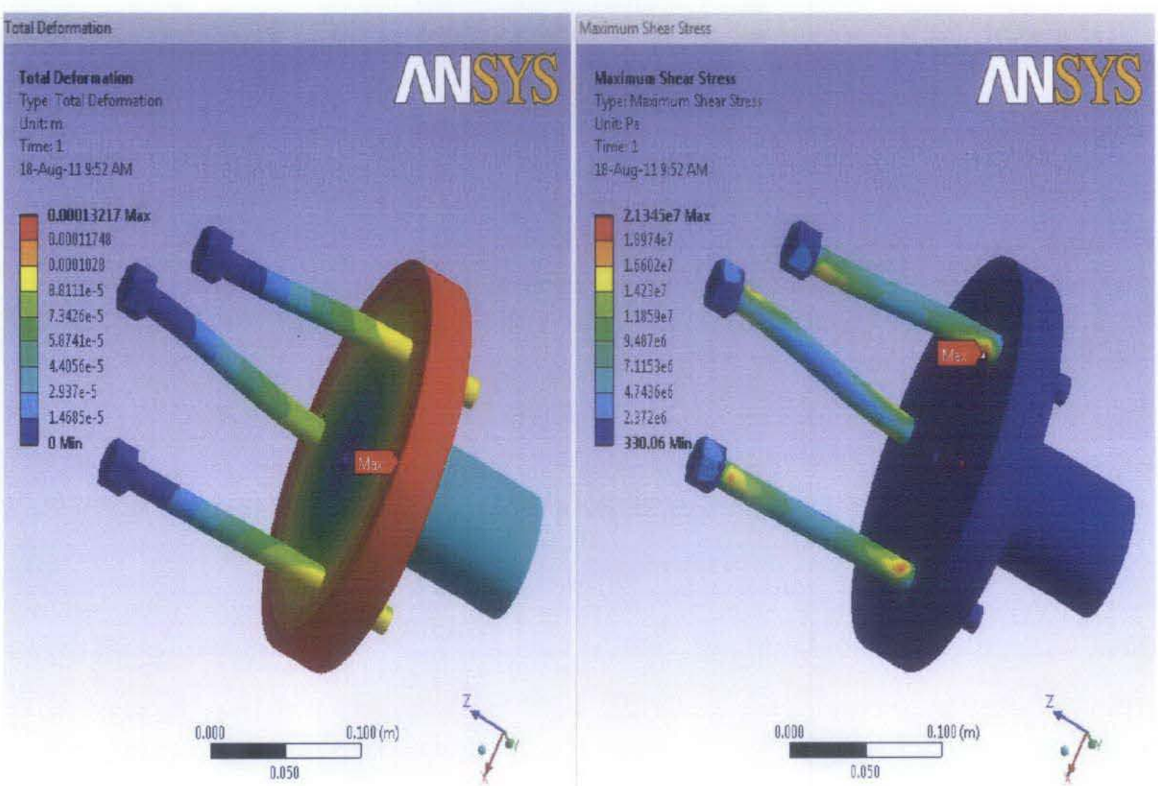


Figure 4.9: Deformation on bolt subjected to shear stress

Table 4.7: Result of total deformation on bolt

Model > Static Structural > Solution > Results

Object Name	*Total Deformation*
State	Solved
Scope	
Geometry	All Bodies
Definition	
Type	Total Deformation
Display Time	End Time
Results	
Minimum	0. m
Maximum	1.3217e-004 m
Minimum Occurs On	bolt baru 16mm:1
Maximum Occurs On	Part6_New Coupler:1
Information	
Time	1. s
Load Step	1
Substep	1
Iteration Number	1

Table 4.8: Result of maximum shear stress on bolt

Model > Static Structural > Solution > Results

Object Name	*Directional Deformation*	*Maximum Shear Elastic Strain*	*Maximum Shear Stress*	*Shear Stress*
State	Solved			
Scope				
Geometry	All Bodies			
Definition				
Type	Directional Deformation	Maximum Shear Elastic Strain	Maximum Shear Stress	Shear Stress
Orientation	X Axis			XY Plane
Display Time	End Time			
Results				
Minimum	-1.3204e-004 m	4.1137e-009 m/m	330.06 Pa	-3.469e+006 Pa
Maximum	1.3155e-004 m	2.6604e-004 m/m	2.1345e+007 Pa	5.1888e+006 Pa
Minimum Occurs On	Part6_New Coupler:1			bolt baru 16mm:1
Maximum Occurs On	Part6_New Coupler:1	bolt baru 16mm:2		
Information				
Time	1. s			
Load Step	1			
Substep	1			
Iteration Number	1			

Based on result, the maximum shear stress happen on the bolt is 21.345MPa and the maximum deformation is 1.3217e-7 mm. these value is relatively small compared to the maximum allowable stress for Carbon Steel.

4.8 Analysis of the Coupler

The coupler experience bending moment due to weight of the entire component clamped to it. The coupler also made of Carbon Steel and has same properties as the hub. Bending moment of the coupler has a value of 43571Nmm.

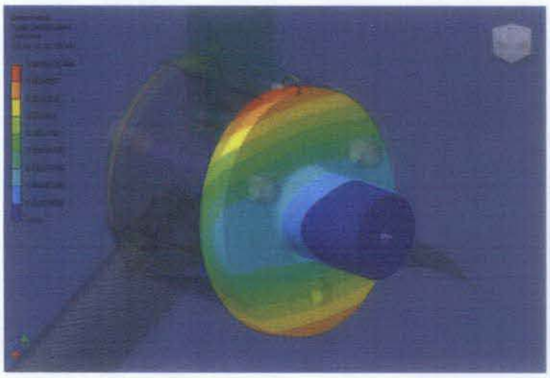


Figure 4.10 : Deformation occur at the coupler

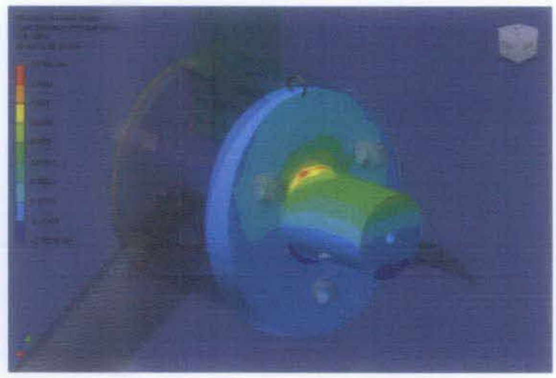


Figure 4.11: Area of maximum stress

Table 4.9: Result for coupler bending moment analysis

Name	Minimum	Maximum
Equivalent Stress	2.458e-003 MPa	1.578 MPa
Maximum Principal Stress	-0.4554 MPa	2.075 MPa
Minimum Principal Stress	-1.886 MPa	0.5216 MPa
Deformation	0.0 mm	2.123e-003 mm
Safety Factor	15.0	N/A

Based on the table, result shown that the maximum stress is only 2.075MPa and the deformation is 2.123e-3mm. These values relatively small compared to the maximum allowable stress for Carbon Steel. The higher stress happens at the fillet connection area between the coupler and shaft.

4.9 Analysis of the detachable hub

There are two type of analysis that need to be done which are stress distribution regarding the centrifugal force and also the thermal distribution regarding the convection happen due to the streaming of hot gases pass through it. Assumption made for this section is;

i. The blade design already ideal for internal blade cooling and does not need to be include in analysis and just for graphically use only.
ii. Constant angular acceleration.

4.9.1 Centrifugal force

The formulae used to calculate the centrifugal force is;

$$\mathbf{F_{centrifugal} = Mr\square^2}$$

Where M = mass R = radius □= rpm

[15]

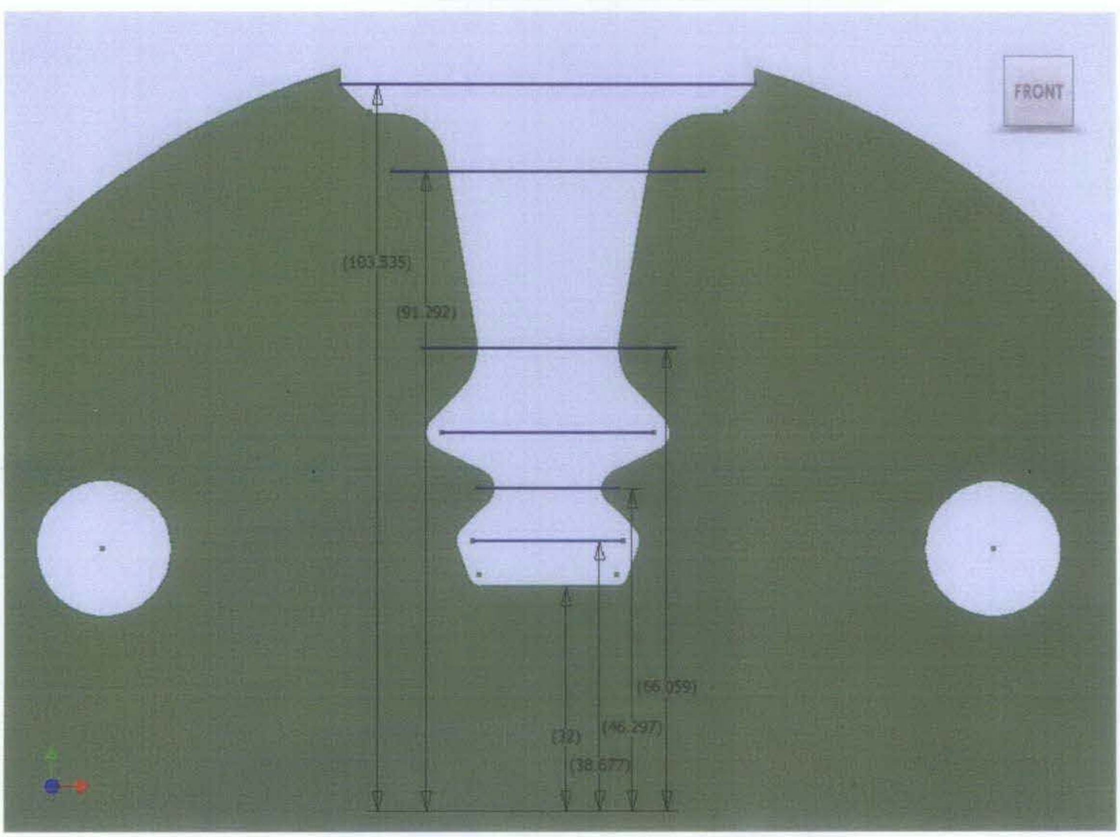


Figure 4.12: Radius of blade from centre

Using the formulae above and value of radius calculated, the value of centrifugal force is:

$f_1 = 2630N$	$f_5 = 5430N$
$f_2 = 3179N$	$f_6 = 7503N$
$f_3 = 3805N$	$f_7 = 8509N$
$f_4 = 5000N$	

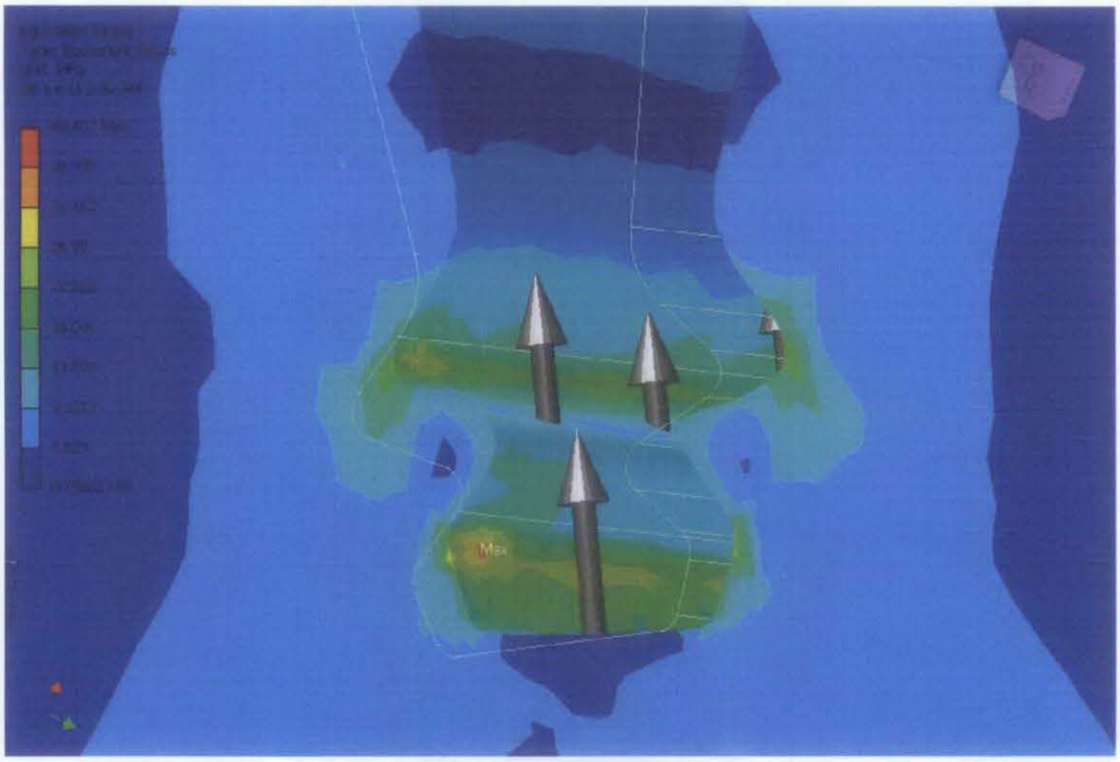


Figure 4.13: Maximum stress area of centrifugal analysis

Table 4.10 : Structural result of centrifugal force

Name	Minimum	Maximum
Equivalent Stress	0.1568 MPa	40.41 MPa
Maximum Principal Stress	-2.924 MPa	44.74 MPa
Minimum Principal Stress	-15.26 MPa	8.139 MPa
Deformation	0.0 mm	1.168e-002 mm
Safety Factor	8.662	N/A

Based on the figure, the arrows show the direction of centrifugal force which tend to go outer from the hub as center. The arrow also defining the area which subjected to the load calculated before depending on the distance from the center of hub.

Based on result, the maximum stress has a value of 44.74MPa. The value is likely a bit higher than other analysis results. This is because the centrifugal force is depending on the weight of the blade. Heavier the blades, the greater the centrifugal force. Tendency of the blade to fly out from it center while rotating showed that the most concentrated area of stress is at the root of the blade area.

The result is the same as the journal according to Lucjan Witek(2004) [3], most concentrated area of stress is at the bottom of the fir-tree root of blade. He perform the analysis using the ABAQUS v.6.4 [16] on the turbine disc and got the same result which indicate that the slots of the dovetail-rim area of turbine blade is the critical and higher area of stress due to centrifugal force of excessive rotational speed.

4.9.2 Thermal distribution

This is one of the crucial analyses for this project. The hub must withstand the temperature of hot gas as high as 650 □C. The source of internal cooling air comes from the compressor and the hot gas is from the combustion chamber. The hub experience both type of force convection which is internal force convection and external force convection.

Finding the heat transfer coefficient

Force Convection – Internal Flow (inside the hollow shaft through to the hub)

$$h = \frac{Kw}{D_H} \times Nu$$

Where, h = heat transfer coefficient
K_w = Thermal conductivity of air
D_H = Hydraulic diameter
Nu = Nusselt Number

Temperature = 33 □C

Convert to Kelvin = 33 + 273

= 306 K

From the **Thermal Properties of Gases at Atmospheric Pressure Table**,

$\rho = 1.1414\ kg/m^3$, $K_w = 26.74 \times 10^{-3}$ W/m.K , $\mu = 187.43$ kg/m.s ,

□ = $16.49 \times 10^{-6}\ m^2/s$, $D_{internal} = 0.01m$, Prandtl Number = 0.706

Volume flowrate, V = 16.65 m^3/s – compute from compressor flowrate

$$v_m = \frac{V}{A}$$

$$= \frac{16.65}{\frac{\pi \times 0.01^2}{4}}$$

$$= 211994.4 \text{ m/s}$$

$$\text{Reynold's Number, Re} = \frac{V_m D_H}{}$$

$$= \frac{211994.4 \times 0.01}{16.49 \times 10^{-6}}$$

$$= 1.3 \times 10^8 > 5 \times 10^5 \Rightarrow \text{turbulent flow}$$

$$Nu = 0.023 \times Re^{0.8} \times Pr^{0.33}$$

$$= 0.023 \times (1.3 \times 10^8)^{0.8} \times 0.706^{0.33}$$

$$= 63531.3$$

$$\text{So, } h = \frac{63531.3 \times 26.74 \times 10^{-3}}{0.01}$$

$$= 169883 \text{ W/m.K}$$

Force Convection – External Flow (inside housing of the blade)

Temperature = 650 □C

Convert to Kelvin = 650 + 273

= 923 K

From the thermal properties of gases at atmospheric pressure table,

$K_w = 63.058 \times 10^{-3}$ W/m.K , □ = 107.178×10^{-6} m^2/s , Pr = 0.721 , ρ = 0.3775 kg/m^3 ,

D = 0.42m

The mass flowrate, □: 10.08 Kg/s (Actual value real working gas turbine)

$$U\infty = \frac{}{\rho\, A}$$

$$= \frac{10.08}{0.3775 \times \frac{\pi \times 0.42^2}{4}}$$

$$= 192.7 \text{ m/s}$$

$$Re_L = \frac{U\infty L}{v}$$

$$= \frac{192.7\frac{m}{s}\times 0.42m}{107.178 \times 10^{-6}\frac{m^2}{s}}$$

$$= 7.6 \times 10^5 < 5 \times 10^5 \Rightarrow \text{turbulent flow}$$

$$Nu_x = 0.0296\, Re_x^{4/5}\, Pr^{1/3}$$

$$= 0.0296 \times (7.6 \times 10^5)^{4/5} \times 0.721^{1/3}$$

$$= 1344.6$$

$$h = \frac{Nu_x K}{L}$$

$$h = \frac{1344.6 \times 63.058 \times 10^{-3}\frac{W}{m.K}}{0.42}$$

$$= 201.9 \text{ W/m}^2\text{.K}$$

All calculation to get the heat transfer coefficient for internal and external force convection is according to the Kirk D. Hagen (1999) [17]

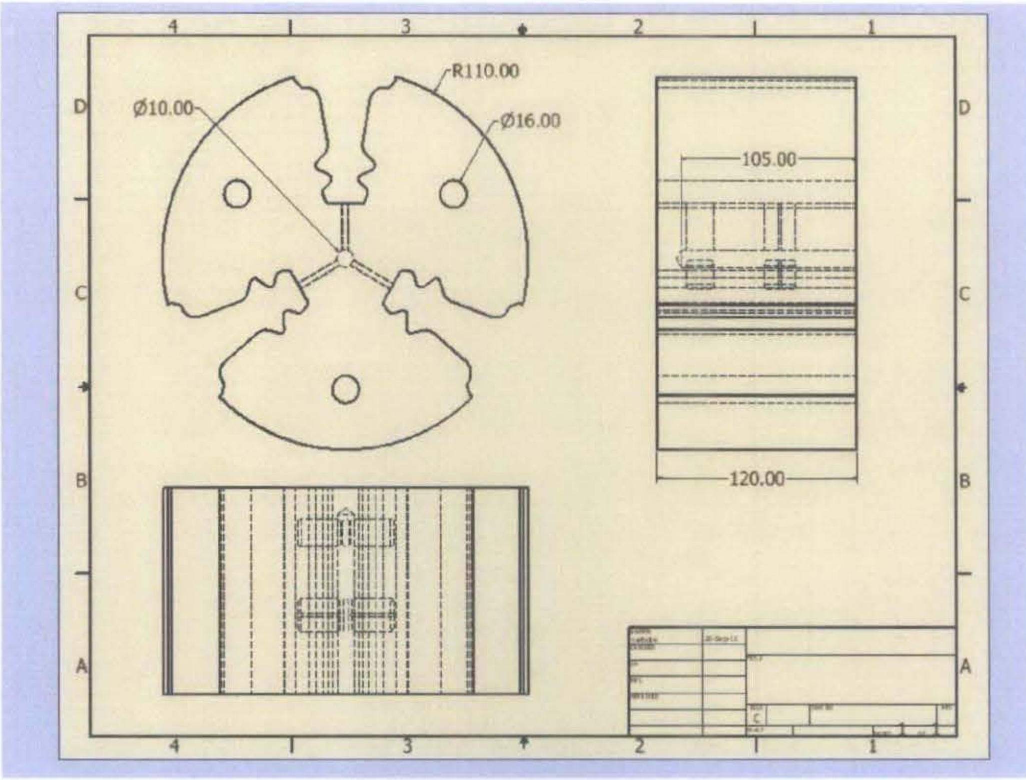


Figure 4.14: Section view of the detachable hub

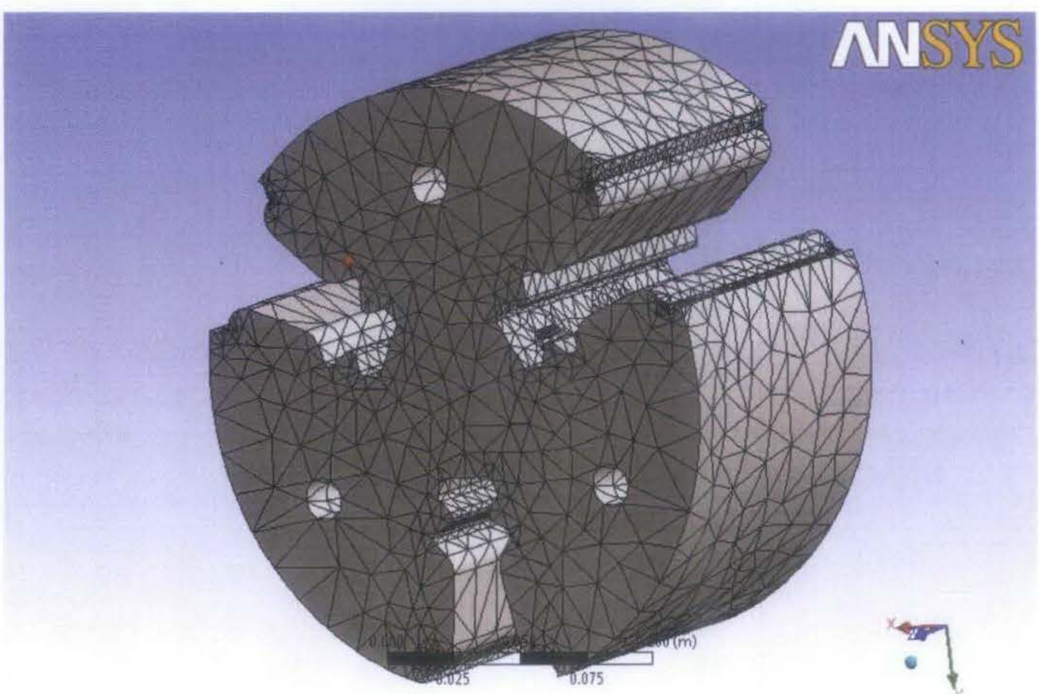


Figure 4.15: Meshing done by Ansys v.11 on hub

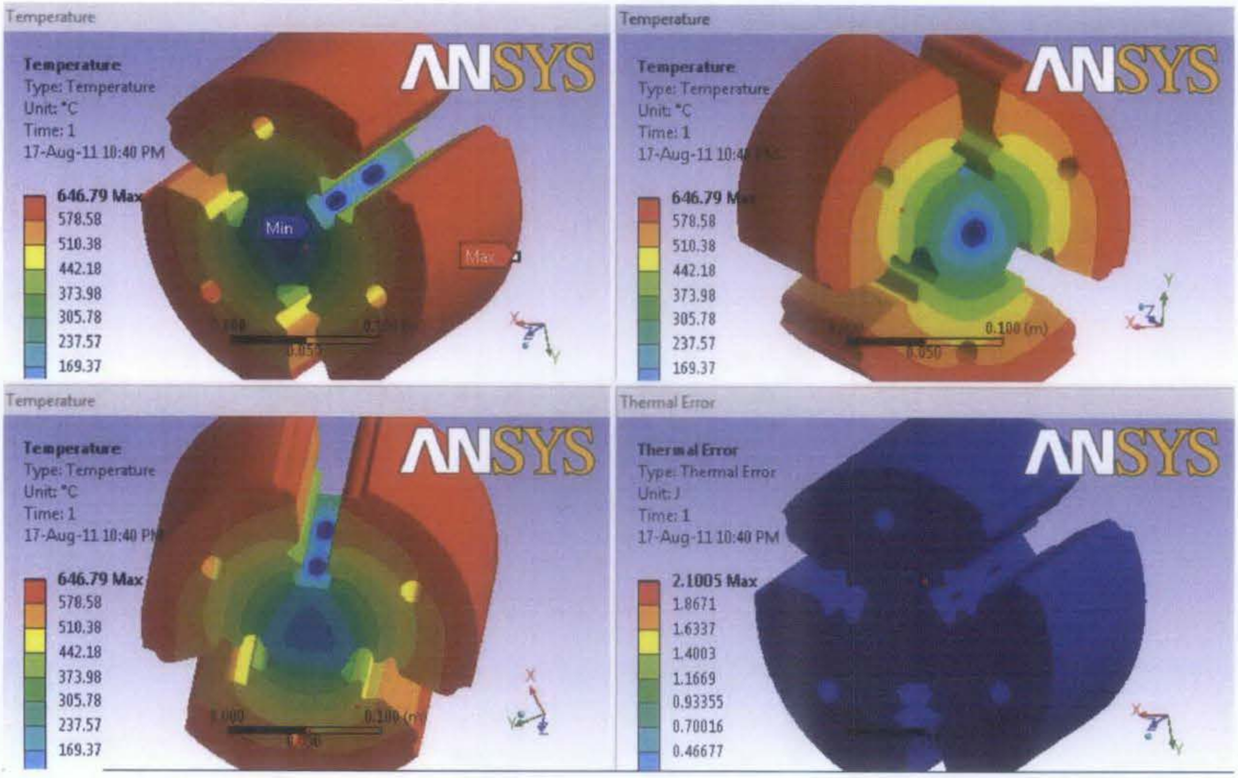


Figure 4.16: Thermal distribution on component

Table 4.11: Thermal load on components

Model > Steady-State Thermal > Loads

Object Name	Convection	Convection 2
State	Fully Defined	
Scope		
Scoping Method	Geometry Selection	
Geometry	38 Faces	3 Faces
Definition		
Type	Convection	
Film Coefficient	1.6988e+005 W/m²·°C (ramped)	201.9 W/m²·°C (ramped)
Ambient Temperature	33. °C (ramped)	650. °C (ramped)
Suppressed	No	

Table 4.12: Result of thermal distribution

Model > Steady-State Thermal > Solution > Results

Object Name	*Temperature*	*Total Heat Flux*	*Directional Heat Flux*	*Thermal Error*
State	Solved			
Scope				
Geometry	All Bodies			
Definition				
Type	Temperature	Total Heat Flux	Directional Heat Flux	Thermal Error
Display Time	End Time			
Orientation			X Axis	
Results				
Minimum	32.967 °C	20.958 W/m²	-10799 W/m²	2.1193e-008 J
Maximum	646.79 °C	12872 W/m²	9806.6 W/m²	2.1005 J
Information				
Time	1. s			
Load Step	1			
Substep	1			
Iteration Number	1			

Figure 4.14 shows the cross section view of the detachable hub. The inner sides of the detachable hub consist of small ducting hollow to permit the streaming of compress air for internal cooling. The compress air will exit to the small hole on the blades.

Figure 4.15 shows the meshing process for the detachable hub done using Ansys Simulation. Meshing must be done so that all part of body will experience the same certain loading condition. Figure 4.16 shows the thermal analysis done on the hub using the Ansys software.

The result obtained based on Table 4.12 shows that the minimum temperature is at the area of internal convection by compress air which is 32.967 °C. The highest value of temperature is at the outer part which is 646.79 °C. The outer part of hub experienced the force convection by the flow of hot gas comes from the combustion chamber. For this analysis, the outer surface of the detachable hub experienced the temperature value of 650 °C. Based on result, internal cooling from compressed air has lowered the value of outer surface temperature.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

In this project, the designing and dimensioning of the detachable hub is done using the Inventor Professional 2009. To get the final view of the product, there is also some related part that was developed for this project. This includes:

- Coupler and shaft
- Hub cover
- Bolts
- Turbine blades
- Nuts

In purpose to conclude weather the design of the detachable hub can be accepted for the gas turbine test rig, the finite element analysis is done using the Ansys simulation software.

For this project, the analysis is done for the bolts, coupler part and also the detachable hub. Based on the result, the maximum stress is 2.285MPa subjected to the static loading from all the clamped parts. The maximum shear stress that occurred at the bolt when the hub started to rotate is 21.345MPa. For the coupler, the highest stress happen at the connection fillet area between the coupler and the shaft.

As for the detachable hub, the stress analysis to investigate the effect of centrifugal force is done using the Inventor software. Result obtained shows the high stress area is at the bottom of the dovetail-rim area. These results comply with the result obtained by Lucjan Witek [3] in his experiment regarding the failure analysis of turbine disc.

As the conclusion, all the objectives have been achieved. By using the Inventor Professional and Ansys simulation software, the design of the detachable hub is completed and the characteristic of the detachable hub subjected to certain loading condition has been investigated.

5.2 Recommendation

As for the future studies of this gas turbine test rig, there are several recommendation that should be taken into consideration to get the accurate result from the gas turbine test rig.

The mass flow rate of air should be determined first. In this project, value of the mass flow rate of air is based on the actual mass flow rate provided by the real power plant operation. This makes the calculation of the convection coefficient of heat transfer is not fully accurate.

The area of fillet connection between the coupler and the shaft are also worth to be taken into consideration for this project. The recommendation is to increase the radius of the fillet area so that the shaft will rotate smoothly and without much vibration. Increasing the fillet area also can reduce the stress concentration due to the total weight of clamped parts.

Last but not least, it is recommended that for the future studies of this project, all the related part for drawing the 3D design and development of part model will be using the Inventor Professional software and the Ansys simulation software for the finite element analysis. The 3D drawing is easy to be developed by using the Inventor. Then the format file of drawing can be import directly to the Ansys simulation for finite element analysis. This is because both of the software is already integrated which each other.

REFERENCES

[1] Mark C. Morris, Pheonix:Nnawuihe "Gas Turbine With aft internal Cooling" 1996

[2] Sukhvinder Kaur Bhatti, Shyamala Kumari, M L Neelapu, C Kedarinath, Dr. In Niranjan kumar. "Transient State Stress Analysis on an axial Flow Gas Tubine Blades and Disk Using Finite Element Procedure" 2006

[3] Lucjan Witek, 2005. "Failure analysis of Turbine Disc of an Aero Engine"

[4] Chan SK, A. 1971. "Finite Element Method for Contact Problem of Solid Bodies – Part II: Applications to Turbine Blade Fastenings" Int J Mech Sci 1971; 627 – 39

[5] Meguid SA, "Finite Element Analysis of Fir-tree Region in Turbine Disc" Finite Element Analysis, Des 2000; 35:305 – 17

[6] Zboinski G, "Physical and Geometrical Non-Linearity in Contact Problems of Elastic Turbine Blade Attachments. J Mech Eng Sci 1995; 209:273 – 86

[7] Stephen issler, Ebernard Roos. "Numerical and Experimental Investigation into Life Assesment of Blade-disc Connections of Gas Turbine". 2003

[8] Dianyin Hu, Rongqiao Wang, Zhi Tao, (2011) "Probabilistic design for turbine disk at high temperature", Aircraft Engineering and Aerospace Technology, Vol. 83 Iss: 4, pp.199 - 207

[9] Tim J Carter. "Common Failures in Gas Turbine Blades". 2004

[10] Joseph W. Savage, Henry J. Brands, Richard H. Anderson. "Cooled Turbine Blade".1970

[11] http://www.efunda.com/materials/alloys/carbon_steels/show_carbon.cfm?id=aisi_1045&prop=all&page_title=aisi%201045>

[12] http://www.roymech.co.uk/Useful_Tables/Matter/Costs.html

[13] http://www.engineeringtoolbox.com/melting-temperature-metals-d_860.html

[14] Richard G. Budynas, J. Keith Nisbett, *"Shigley's Mechanical Engineering Design"*, Eight Edition in SI Unit.

[15] http://en.wikipedia.org/wiki/Centripetal_force

[16] ABAQUS User's Manual, ver 6.4, Abaqus Inc.; 2003

[17] Kirk D. Hagen, 1999, "Heat Transfer with Applications" (252,296)

APPENDICES

FYP I

No.	Activities	Week														
		1	2	3	4	5	6		7	8	9	10	11	12	13	14
1	Selection of project topic							Mid-semester break								
2	Information Collection - Gas turbine principal - Hub design and specification															
3	First draft of hub dimension															
4	Project work -Study and finalize the design specification -material selections -Familiarization to ANSYS and Inventor Professional software - measurement of turbine blade dimension															
5	Torque Analysis - Find the preferred bolt size															
6	Seminar															
7	Stress Analysis															
8	Submission of interim Report Final Draft															
9	Oral presentation															

FYP II

No.	Activities	Week														
		1	2	3	4	5	6		7	8	9	10	11	12	13	14
1	Review FYP I progress works							Mid-semester break								
2	1. Finalize design specifications and dimension. 2.FEA on bolt and fir-tree slot															
3	Ansys progress – 1.Thermal Load • Thermal deformation • Thermal stress 2.Dynamic load • Stress distribution Deformation															
4	Documentation of progress report															
5	Completing the documentation for hub disc design and technical report															
6	Draft report and poster															
7	Preparation for final report and final presentation															