CHAPTER 1
INTRODUCTION

1.1 Background of Study

Impellers are rotating devices that force liquids, gases and vapors in a desired direction. They are widely used in rotating equipment such as pumps, blowers, compressors and mixing applications. This area gives the ability to search for impellers for pumping and mixing of media and allows selection of type of impeller and size. Impeller applications, specifications and features, types, and materials are all important parameters to consider when searching for impellers. In single-stage centrifugal compressor, the types of impellers that usually be used are the single-entry type since it is designed for centrifugal-flow fluid compression. As it is being mostly used in power plants and oil and gas industries, the impeller are experiencing heavy load environment in terms of pressures and temperatures. Thus, the failure of the impeller are expected and also possibly failed upon its service life.

Figure 1.1: Typical centrifugal impeller in centrifugal compressor

(http:/www.leitemlane.com/)

Finite Element Analysis, or popularly termed as FEA has become commonplace in recent years, and is now the basis of a multibillion dollar per year industry. Numerical solutions to even very complicated stress problems can now be obtained routinely using FEA. The Finite Element method is a good choice for solving partial differential equations over complex domains, when the domain changes (as during a solid state reaction with a moving boundary), when the desired precision varies over the entire domain, or when the solution lacks smoothness.

1.2 Problem Statement

Impeller can be stated as the most critical part or component in compressor. It is because the impeller is used to manipulate the pressure of the fluid from lower pressure to higher pressure as it called a compression process of the fluids. Therefore, it is being used mostly in heavy environment process plants such as power plants and oil and gas industries. Under its service life, it tends to fail due to crack. The nature of complex geometry of the impeller itself required the numerical method to solve the potential problem of the design. Thus, it is required to predict by Computer Aided Engineering using Finite Element Analysis to predict the corresponding potential failure initiation by undergo a study of stress distribution of the impeller.

1.3 Objective and Scope of Study

The main objectives of this research are:

- To undergo a Finite Element Analysis of the single-entry impeller of single stage centrifugal compressor
- To find the stress distribution and concentration of the impeller

The scope of study for this project is to analyse the stress concentration and distribution by undergo a Finite Element Analysis of the impeller by using ANSYS® software which is undoubtly one of the best Finite Element software available. This will require modelling and simulation of the impeller to identify the stress distribution and concentration and also identify the possible locations of the crack that tends to initiate. In addition, the modelling process of the impeller will be done by using CATIA® Computer Aided Design software. The study will confront some analysis such as high stress levels resulting from the critical fluid properties that are necessary to achieve high compression which will yield the possibilities of crack initiation.

CHAPTER 2

LITERATURE REVIEW

2.1 Centrifugal Compressor

Centrifugal compressors are fluid-flow machines for the compression of gases according to the dynamic principle. The bladed impeller with its continual through flow transfers the mechanical shaft energy into enthalpy, i.e. gas energy. Thus, the pressure, temperature and velocity of gas leaving the impeller are higher than at the impeller inlet. The annular diffuser downstream of the impeller delays the gas velocity, thus providing a further pressure and temperature increase. Figure 2.1 shows part of a centrifugal compressor. It consists of a stationary casing containing an impeller, which rotates and imparts kinetic energy to the air and a number of diverging passages in which the air decelerates. The deceleration converts kinetic energy into static pressure. This process is known as diffusion, and the part of the centrifugal compressor containing the diverging passages is known as the diffuser.

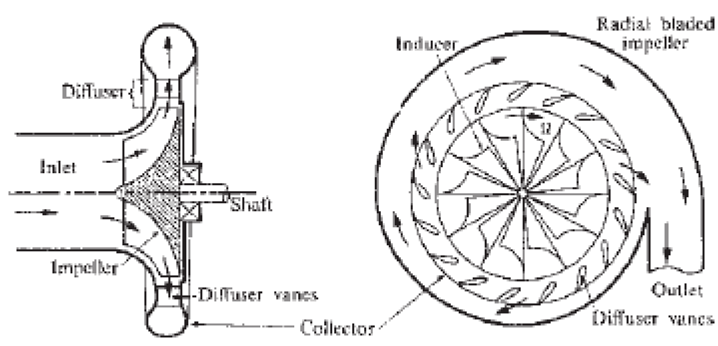


Figure 2.1: Typical centrifugal compressor

(https://engineering.purdue.edu/AAE/Research/Propulsion/Info/jets/basics/comp)

Air enters the impeller eye and is whirled around at high speed by the vanes on the impeller disc. After leaving the impeller, the air passes through a diffuser in which kinetic energy is exchanged with pressure. Energy is imparted to the air by the rotating blades, thereby increasing the static pressure as it moves from eye radius r_1 to tip radius r_2. The remainder of the static pressure rise is achieved in the diffuser. The normal practice is to design the compressor so that about half the pressure rise occurs in the impeller and half in the diffuser. The air leaving the diffuser is collected and delivered to the outlet.

In practice nearly half the total pressure is achieved in impeller and remaining half in the diffuser. A pressure ratio of 4:1 can be achieved with single stage centrifugal compressor. For higher ratios, multistage compressors are used. In multistage compressors, the outlet of the first stage is passed to the second stage and so on. A pressure ratio of 12:1 is possible with multistage centrifugal compressors.

2.2 Impeller

Impeller can be defined as rotating devices that force liquids, gases and vapours in a desired direction with increasing of pressure. Various types of single-entry impeller have been designed with different materials of construction such as Stainless Steel, Titanium alloys, Nickel alloys, ceramic composites and many more. This depends on the application and environmental area of the impeller itself like compressor, pumps and gas turbines. The impeller consists of forged disk complete with radially arranged vanes and it may be double sided, having an eye on either side of the compressor, so that air is drawn in on both sides, as shown in Fig. 2.2.

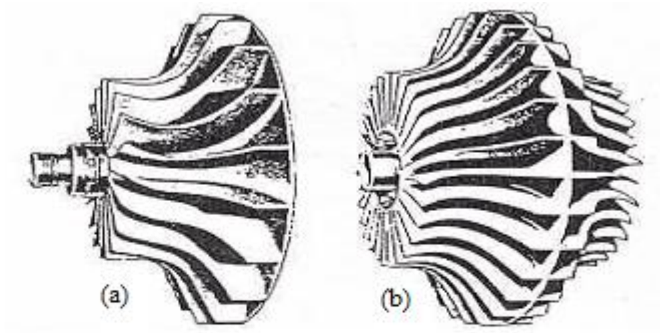


Figure 2.2 Types of centrifugal impeller (a) single-entry, (b) double entry

(http://www.compressorworld.com.au/archives/34-Centrifugal-Compressors.html)

For these types of impeller, it is specifically designed for axial-flow centrifugal compressors. The choice of types is determined by the engine design requirements but it is claimed that the single entry ducting allows the fluid to be fed in at the best all-round efficiency. The principal differences between the single entry and dual entry are the size of the impeller and the ducting arrangement. The single entry impeller permits ducting directly to the inducer vanes, as opposed to the more complicated ducting needed to reach the rear side of the dual-entry type. Although slightly more efficient in receiving air, single-entry impellers must be of greater diameter to provide sufficient air. An impeller in centrifugal compressors imparts energy to a fluid. The impeller consists of two basic components; an inducer like an axial-flow rotor and the radial blades or vanes where energy is imparted by centrifugal force as in Figure 2.3.

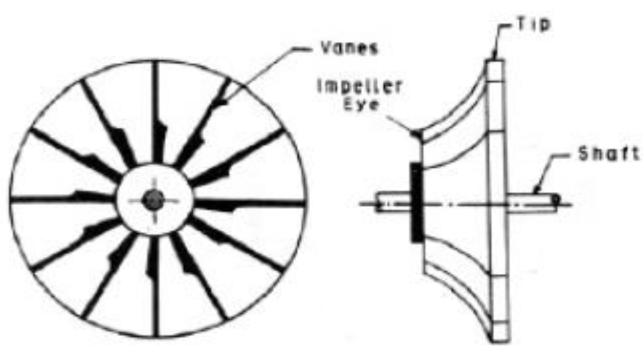


Figure 2.3: Single-entry impeller

(http://www.compressorworld.com.au/archives/34-Centrifugal-Compressors.html)

The flow can enter the impeller axially, with a positive rotation (rotation of the flow in the direction of the impeller), or with a negative rotation (rotation of the flow in the direction opposite to the rotation of the impeller). It then flows into an inducer with a minimal incidence angle, and its flow direction is changed from axial to radial.

Then, air is drawn in near the hub, called the impeller eye, and is whirled round at high speed by the vanes on the impeller as the impeller rotates at high rotational speed. The static pressure of the air increases from the eye to the tip of the impeller in order to provide the centrifugal force on the air. As the air leaves the impeller tip it is passed through diffuser passages that convert most of the kinetic energy of the air into an increase in enthalpy and hence the pressure of the air is further increased.

There are three impeller vane types, as shown in Figure 2.4 along with the velocity triangles in the radial plane for the outlet of each type of vane. These are defined according to the exit blade angles. Impeller with exit blade angle $\beta_2 = 90^0$ are radial vanes. Impellers with $\beta_2 < 90^0$ are backward-curved or backward-swept vanes, and for $\beta_2 > 90^0$, the vanes are forward-curved or forward-swept. They have different characteristics of theoretical head-flow relationship to each other as shown in Figure 2.5.

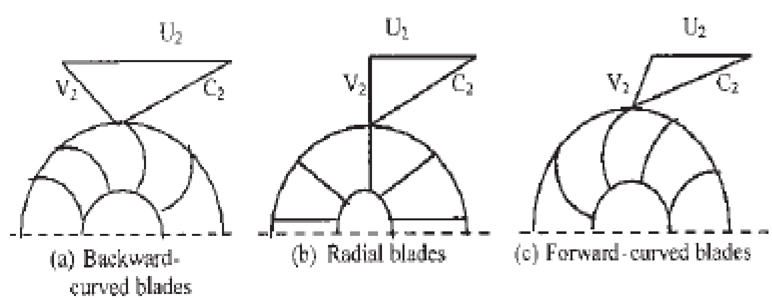


Figure 2.4: Shapes of centrifugal impeller blades: (a) backward-curved blades, (b) radial blades, (c) forward-curved blades *(Rama S. R. Gorla, Aijaz A. Khan, 2003)*

Figure 2.5 represents the relative performance of these types of blades. It is clear that increased mass flow decreases the pressure on the backward blade, exerts the same pressure on the radial blade, and increases the pressure on the forward blade. For a given tip speed, the forward-curved blade impeller transfers maximum energy, the radial blade less, and the least energy is transferred by the backward-curved blades. Hence with forward-blade impellers, a given pressure ratio can be achieved from a smaller-sized machine than those with radial or backward-curved blades.

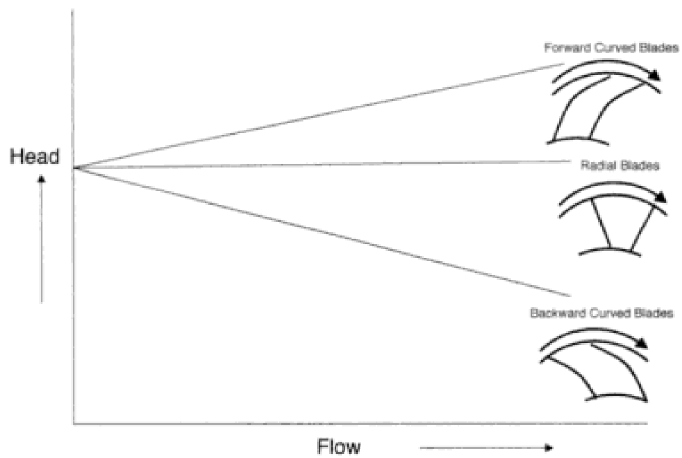


Figure 2.5: Theoretical head characteristics as a function of the flow in a centrifugal compressor *(Meherwan P. Boyce, 2002)*

The velocity of the fluid is converted to pressure, partially in the impeller and partially in the stationary diffusers. Most of the velocity leaving the impeller is converted into pressure energy in the diffuser as shown in Figure 2.7. The air enters the compressor in axial direction and exits in a radial direction into a diffuser. This combination of impeller and diffuser comprises a single stage. The diffuser consists essentially of vanes, which are tangential to the impeller. These vane passages diverge to convert the velocity head into pressure energy. The inner edge of the vanes is in line with the direction of the resultant airflow from the impeller, as shown in Figure 2.8.

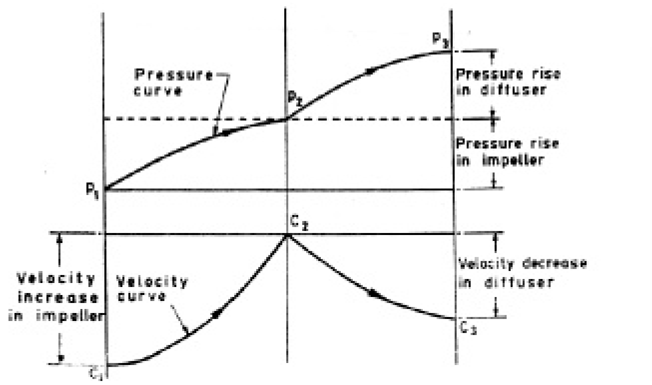


Figure: 2.6 Variations of pressure and velocity of air passing through impeller and diffuser *(Meherwan P. Boyce, 2002)*

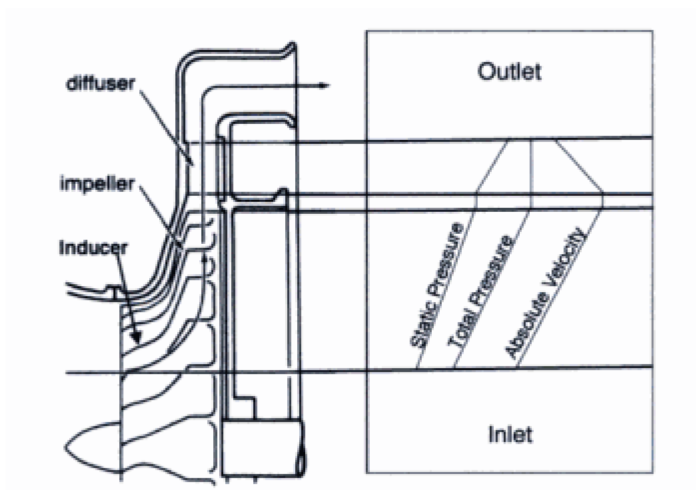


Figure 2.7 Aerodynamic and thermodynamic properties in a centrifugal compressor stage *(Meherwan P. Boyce, 2002)*

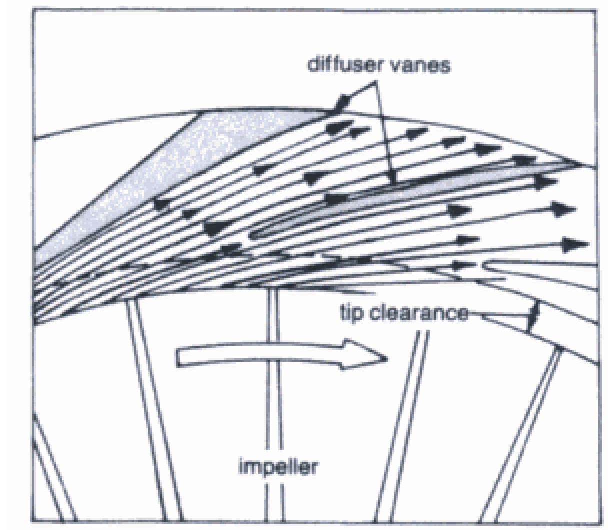


Figure 2.8 Flow in a vaned diffuser *(Meherwan P. Boyce, 2002)*

2.3 Finite Element Analysis

Finite Element Analysis (FEA) is a numerical method which provides solutions to problems that would otherwise be difficult to obtain. In terms of fracture, FEA most often involves the determination of stress intensity factors. FEA, however, has applications in a much broader range of areas; for example, fluid flow and heat transfer. While this range is growing, one thing will remain the same: the theory of how the method works. Finite Element Analysis (FEA) is particularly well suited to dealing with complex problems, which in reality are normally a composite of continuous fields of displacements, strains, stresses, temperatures, state variables, and etcetera. The problem can thus be solved by using an approximate, discrete element based solution. In actual practice, the continuum is subdivided into small pieces or elements. Each element is in itself continuous, however it also assumes a given distribution pattern of the fields, based on values at a selected number of control points or nodes.

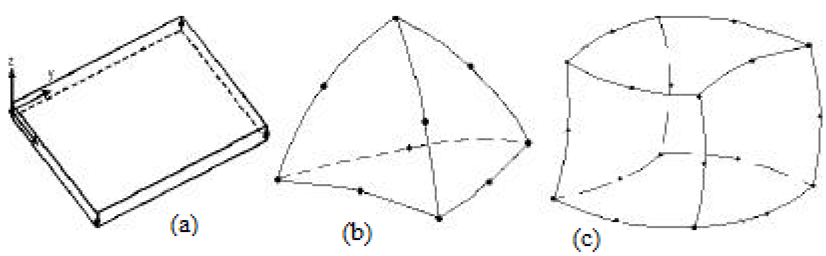
The purpose of FEA is to expose and solve, to the extent possible, design flaws before a product is produced. This includes the capability to perform failure mode prediction such as fatigue studies on materials. It does not normally preclude the manufacture of prototypes but it should greatly reduce the number of prototype iterations required and in the best case will reduce the number of prototypes to one. The modeling of the finite element can be done by using finite element software that available on the market such as ANSYS®, ABAQUS™, MSC Nastran™ and many more. Specifically, this project will be using ANSYS® as the finite element modeling software since it is the most widely used worldwide. ANSYS, Inc. is an engineering simulation software provider. It develops general-purpose finite element analysis and computational fluid dynamics software. While ANSYS has developed a range of computer-aided engineering (CAE) products, it is perhaps best known for its ANSYS Mechanical and ANSYS Multiphysics products.

Finite Element Analysis can be pictured as a mathematical representation of a physical system comprising a part/assembly (model), material properties, and applicable boundary conditions (collectively referred to as *pre-processing*), the solution of that mathematical representation (*solving*), and the study of results of that solution (*post-processing*). Simple shapes and simple problems can be, and often are, done by hand. Most real world parts and assemblies are far too complex to do accurately, let alone quickly, without use of a computer and appropriate analysis software. The analysis that are covered in ANSYS® are such as Static Structural, Flexible Dynamic, Rigid Dynamic, Modal, Random Vibration, Fatigue, Linear Buckling and many more.

2.3.1 Pre-Processing

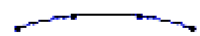
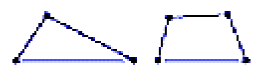
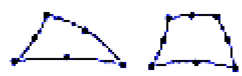
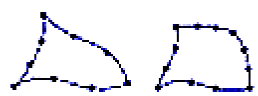
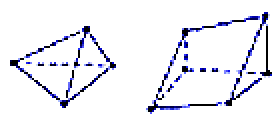
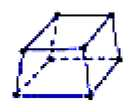
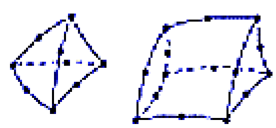
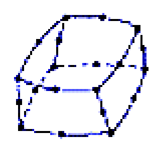
FEA software typically uses a CAD representation of the physical model and breaks it down into small pieces called finite “elements”. This process is called “meshing”. The higher the quality of the mesh(collection of elements), the better the mathematical representation of the physical model. The primary purpose of an element is to connect nodes with predictable mathematical equations based on stiffness between nodes; the type of element used often depends upon the problem to be solved. The behaviour of each element, by itself, is very well understood. By combining the behaviours of each element using simultaneous equations, one can predict the behaviour of shapes that would otherwise not be understood using basic "closed form" calculations found in typical engineering handbooks.

There are many different types and classes of elements, most created for specialized purposes (cable, piping, beams, truss structures, e-mag, etc.). A one-dimensional element represents line shapes, such as beams or springs. A 2D element, also known as a quadrilateral element, will represent triangles and squares. 3D elements represent solid shapes and are usually in 2 basic shapes: brick (hexahedrons or "hex") and pyramids (tetrahedrons or "tets"). Many thin shapes can, and are, meshed with 3D solid elements but at the cost of increased processing time and sometimes a loss in accuracy because of the special formulation of 2D shell elements. The tradeoff is that, in order to mesh with 2D shell elements, there is often significant modification and preparation required to the CAD geometry in order to obtain a meshable surface model, or models in the case of an assembly. In other words, the pre-processing requirement increases substantially.



·Figure 2.9: Example of nodes (a) 4-Noded Shell (b) 10-Noded Tetrahedral (c) 20-Noded Hexahedral

Table 2.1: Other common element types

Dimension	Degree	Element Shape	Element Type
1D (Line)	Linear		Beam, Truss
	Quadratic		Beam
	Cubic		Beam
2D (Area)	Linear		
	Quadratic		Plane stress Plane strain Plate, Shell
	Cubic		
3D (Volume)	Linear		
	Quadratic		

Each element is comprised of 2 or more "nodes" which help define its shape as well as to convey physical reactions from one element to the next. The "finite" in FEA comes from the fact that there are a known number of elements in a finite element model. The solver adds up the individual behaviours of each element to predict the behaviour of the entire physical system. Other aspects of the pre-processing phase involve identifying material properties and environmental conditions the design will be subject to. These conditions include various forms of physical forces (loads, pressures, moments, etc.), thermal loads and conditions (temperature, conductivity, convection, etc.), and constraints (fixed, pinned, frictionless/symmetrical, etc.).

2.3.2 Post-Processing (Interpretation of Results)

The output of a solver is generally a very substantial quantity of raw data. This quantity of raw data would normally be difficult and tedious to interpret without the data sorting and graphical representation referred to as post-processing. Post-processing is used to create graphical displays that show the distribution of stresses, strains, deformations, temperatures, and other aspects of the model. Interpretation of these post-processed results is the key to identifying areas of potential concern (weak areas in a model), areas of material waste (areas of the model bearing little or no load), or valuable information on other model performance characteristics (thermal, modal) that otherwise would not be known until a physical model were built and tested (*prototype*).

The post-processing phase of FEA is where the most critical thinking must take place, where the user looks at the results (the numbers vs. colour contours, movements, etc.), and compares results with what might be expected. It cannot be stressed enough that it is up to the user to determine if the results make sense, to be able to explain the results based upon engineering "common sense." If the results are other than expected, one must search until an explanation can be found before the results can be fully trusted.

2.4 Static Stress Analysis

The static stress analysis can be conducted for a single sector or a full impeller finite element model. Supported loads include centrifugal load, aerodynamic forces and thermal loads. If a single sector model is used, the cyclic constraint boundary condition and cyclic load are applied.

A static structural analysis determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time. The types of loading that can be applied in a static analysis include:

- Externally applied forces and pressures
- Steady-state inertial forces (such as gravity or rotational velocity)
- Imposed (nonzero) displacements
- Temperatures (for thermal strain)

The parameter that is being observed for stress distribution is the Von-Mises Stress or also known as Equivalent Stress. In material science and engineering, the von Mises yield criterion can be formulated in terms of the von-Mises stress or equivalent tensile stress, σ_v, a scalar stress value that can be computed from the stress tensor. In this case, a material is said to start yielding when its von-Mises stress reaches a critical value known as the yield strength, σ_y. The von-Mises stress is used to predict yielding of materials under any loading condition from results of simple uniaxial tensile tests. The von-Mises stress satisfies the property that two stress states with equal distortion energy have equal von-Mises stress.

2.5 CAD Modeling

Computer-aided design (CAD) is the use of computer technology for the design of objects, real or virtual. The design of geometric models for object shapes. Modeling of the impeller must be done in order to integrate or simulate the impeller in ANSYS® Workbench software. The design of the impeller will be done using Computer Aided Design software CATIA®. CATIA (Computer Aided Three Dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software suite developed by French company Dassault Systemes and marketed world-wide by IBM. Written in the C++ programming language, CATIA® is the cornerstone of the Dassault Systemes PLM software suite. The software commonly referred to as 3D Product Lifecycle Management software suite, CATIA® supports multiple stages of product development. The stages range from conceptualization, through design (CAD) and manufacturing (CAM), until analysis (CAE). CATIA® is widely used throughout the engineering industry, especially in the automotive and aerospace sectors. The advantages of modeling the impeller using CATIA® is that the drawing can be integrated with ANSYS® as for the finite element analysis that will be done.

CHAPTER 3
METHODOLOGY

3.1 Project Flow

To ensure the project is going well, smoothly and to complete the project in the timeframe given, a proper plan of the project flow is made. The project is done step by step, phase by phase as the information and data gathering or simply called literature review is done as the first step as shown in Figure 3.1. The information is obtained through research medium such as papers, journals, books and some online papers available on the internet. All findings are basically about the keywords of the project like centrifugal compressors, impellers, finite element and static stress analysis. The variety methods, techniques, parameters and boundary conditions of impeller are highlighted in order to perform the accurate analysis of the impeller in the centrifugal compressor.

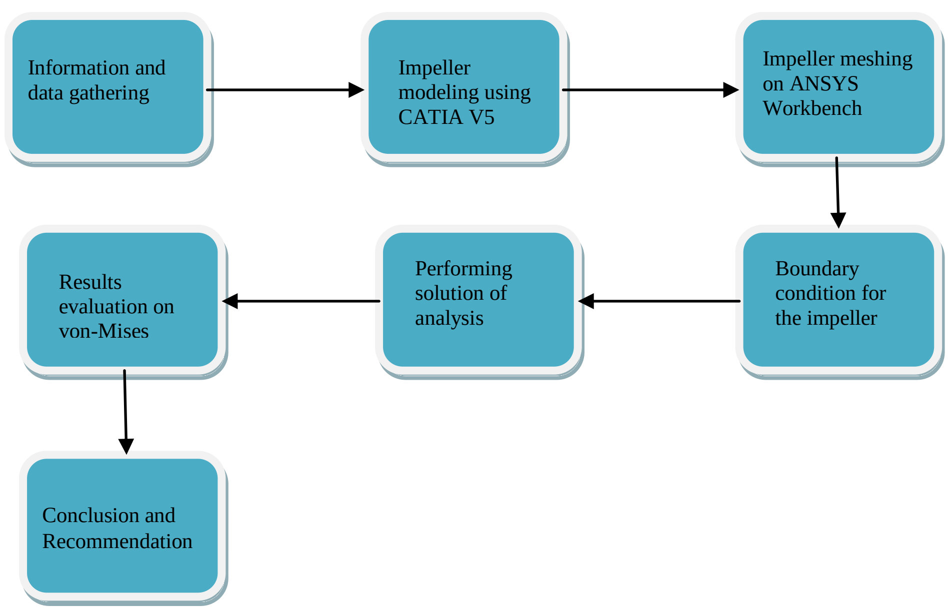


Figure 3.1: Project flow planning

For the second phase of the project, the progress of modeling the CAD model of the impeller is done. The modeling is done on CATIA® V5, a well-know CAD software available and very popular among mechanical engineers in creating 3D solid models. Lots of tutorials and notes are consumed in order to be able to model the impeller in CATIA® V5. The next process of the project is to perform Finite Element Analysis on the centrifugal impeller. The analysis begin with pre-processing module in ANSYS® Workbench, meshing then followed-up by boundary condition and finally, retrieving the results of analysis from ANSYS® solver output.

After completing with impeller simulation in ANSYS®, a deep study is conducted regarding the results the stress distribution and concentration on the impeller. Finally, as the last stage of the project, the conclusion and recommendation are made to the project results. Below is the Gantt chart for both semesters as shown in Table 3.1 and Table 3.2.

Table 3.1: Gantt chart for Semester 1

No	Activities / Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Selection of Project Topic														
2	Information and Data Gathering on Topic														
3	Submission of Preliminary Report														
4	Modeling of The Impeller														
5	Submission of Progress Report														
6	Results Gathering														
7	Submission of Interim Report														
8	Oral Presentation														

Table 3.2: Gantt chart for Semester 2

No	Activities / Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
2	Finite Element Analysis														
3	Submission of Progress Report 1														
4	Results Gathering & Discussion														
5	Submission of Preliminary Report 2														
6	Seminar														
7	Conclusion and Recommendation														
8	Poster Exhibition														
9	Submission of Dissertation (softbound)														
10	Oral Presentation														
11	Submission of Dissertation (hardbound)														

3.2 Impeller Modeling

As one of the important earlier stage in this project, it is required to model an open type single-entry impeller before the project continues with the computerized finite element analysis using ANSYS® Workbench software. To model the impeller, popular CAD simulation software, namely CATIA® V5 were used to model the impeller. The reason of not modeling the impeller by using ANSYS® Workbench software is due to the nature complex design of the impeller itself and not very well practiced by the author. As being recommended, the CATIA® V5 however has the future of abilities to be imported or integrated with ANSYS® FEA software. Modeling of the impeller takes time because it is need a lot of practices and efforts to simulate the impeller very well.

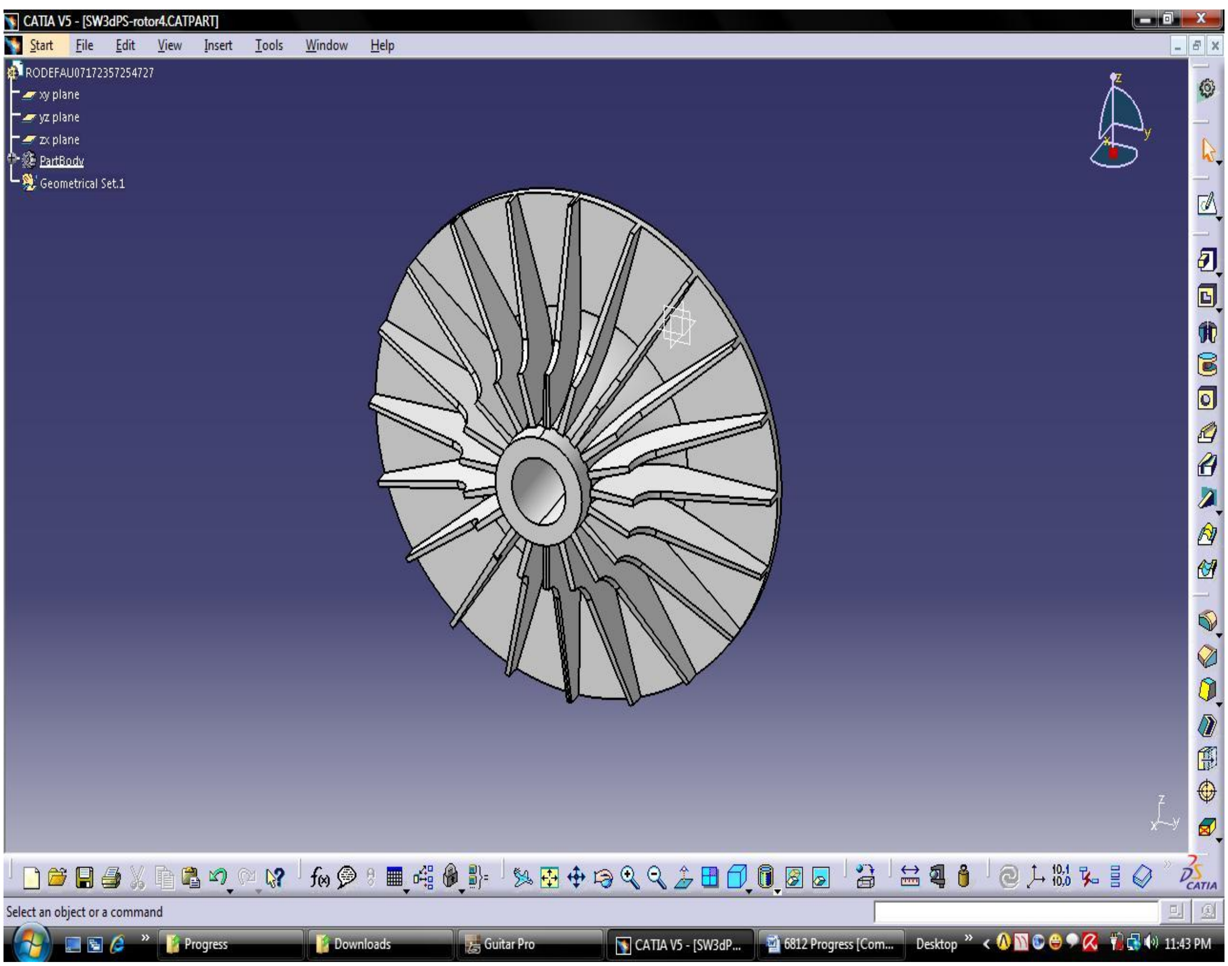


Figure 3.2 Screenshot of impeller modeling

The figure above shows the screenshot of completed impeller modeling in CATIA® V5 and below are some specifications of the impeller properties that are used in analysis as shown in Table 3.3.

Table 3.3: Impeller data specifications

Overall diameter	0.060 m
No. of blades	20
Blade type	Radial

3.3 Finite Element Analysis

Finite element method or analysis will be done by using ANSYS® software. The analysis will determine the appropriate stresses such as stress-intensity factor and also von-Mises stress which is a parameter for crack initiation. Finite element analysis uses a complex system of points called nodes which make a grid called a mesh. This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. Regions which will receive large amounts of stress usually have a higher node density than those which experience little or no stress. Points of interest may consist of: fracture point of previously tested material, fillets, corners, complex detail, and high stress areas. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes. This web of vectors is what carries the material properties to the object, creating many elements. As the vanes are symmetrical and identical in design to the other, the analysis will be done in single vane modeling and further continue with the full model analysis for failure mode predictions.

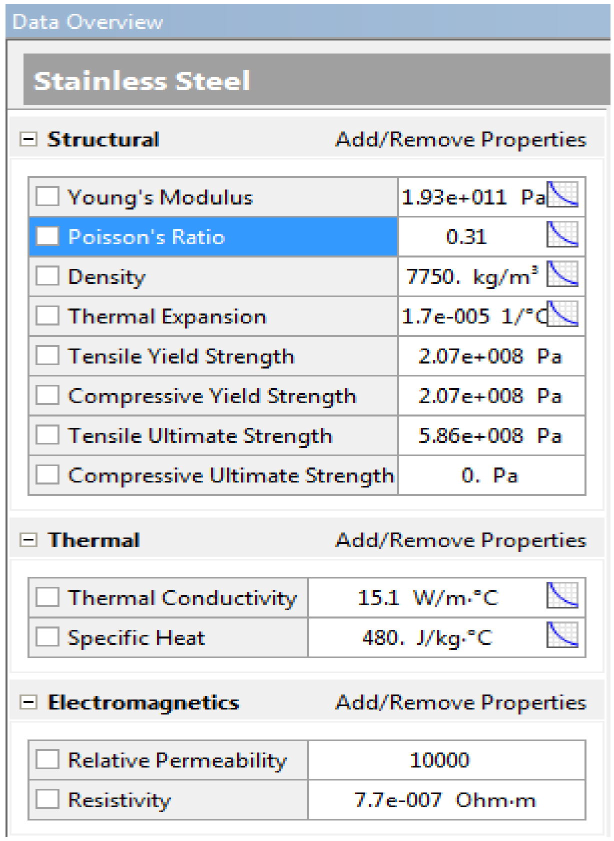
Considering the complex geometry of the gas compressor components, diffuser, and the complex loading cases, the numerical methods only are appropriate to analyse the static and dynamic behavior compared to finite difference and boundary element method.

Therefore widely used numerical finite element method (FEM) was used to determine deformation and stress distribution of the impeller.

3.3.2 Meshing

Refinement in meshing or simply called as mesh refinement was used in this analysis. Mesh refinement is a postprocess in the mesh generation process in which the elements on the selected topology are split. This is useful for local mesh sizing control. This approach will reduce the elements size of the nodes on the impeller. The types of elements from the meshing that were used are Tet-10 (10-Noded Tetrahedral) elements (SOLID 187 in ANSYS® Classic). There are arguments about Tet-10 and Hex-20 about which one provides better meshing and stress results accuracy but in ANSYS® Workbench Simulation, it uses several primary element types and will default to high-order (10 node quadratic) tetrahedral elements for solid model geometries if they are not sweepable. Moreover, ANSYS® Workbench Simulation applies these various element types automatically. However, the mesh refinement was not at its best due to the lack of computer memory. To increase the level of refinement, a high-end computer specification for hardware is needed.

Finer meshing must be used in regions of expected high stress gradients (usually occur at discontinuities) and mesh refinement must be gradual with adjacent elements of not too dissimilar size. For information, the element of material being used for this impeller in the analysis is Stainless Steel provided in ANSYS® Workbench.



Data Overview

Stainless Steel

Structural	Add/Remove Properties
Young's Modulus	1.93e+011 Pa
Poisson's Ratio	0.31
Density	7750. kg/m³
Thermal Expansion	1.7e-005 1/°C
Tensile Yield Strength	2.07e+008 Pa
Compressive Yield Strength	2.07e+008 Pa
Tensile Ultimate Strength	5.86e+008 Pa
Compressive Ultimate Strength	0. Pa

Thermal	Add/Remove Properties
Thermal Conductivity	15.1 W/m·°C
Specific Heat	480. J/kg·°C

Electromagnetics	Add/Remove Properties
Relative Permeability	10000
Resistivity	7.7e-007 Ohm·m

Figure 3.3: Material data overview of Stainless Steel

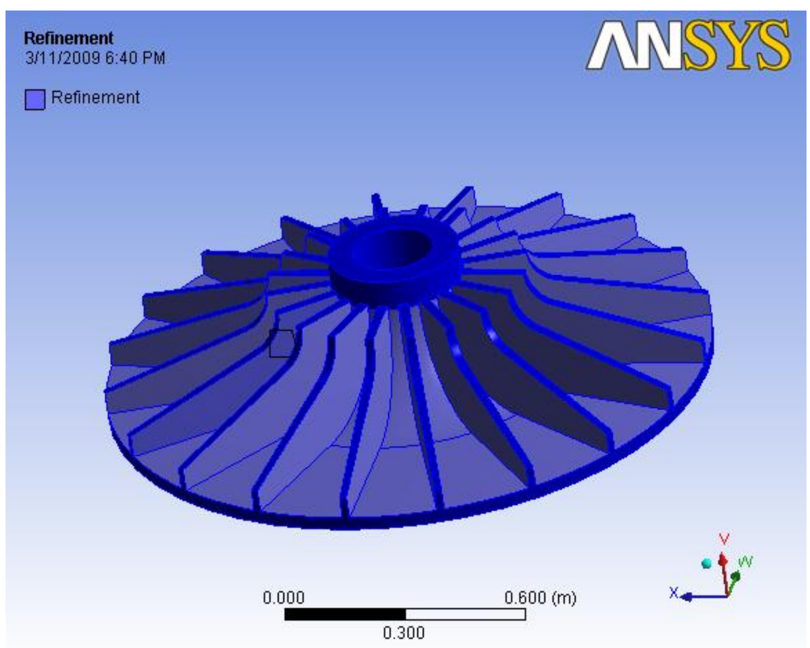


Figure 3.4: Selected faces for mesh refinement

Figure 3.4 shows the selected faces for mesh refinement process. The selected faces are where the high-stress regions are concentrated or high pressures are exerted on the area. The refinement has to be done before meshing the impeller. Then, the mesh generation process is continued and thus, the analysis creates 548580 nodes and 355680 elements.

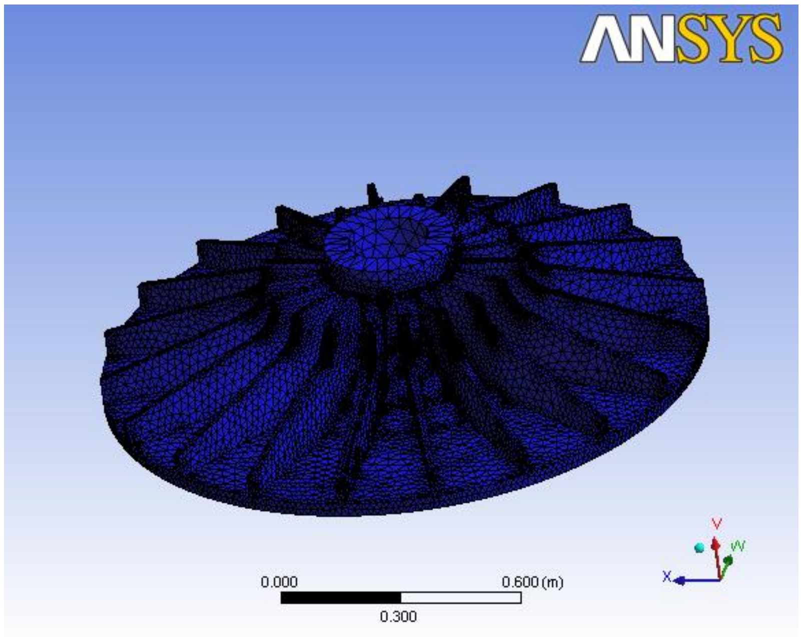


Figure 3.5: Meshing distribution on the impeller

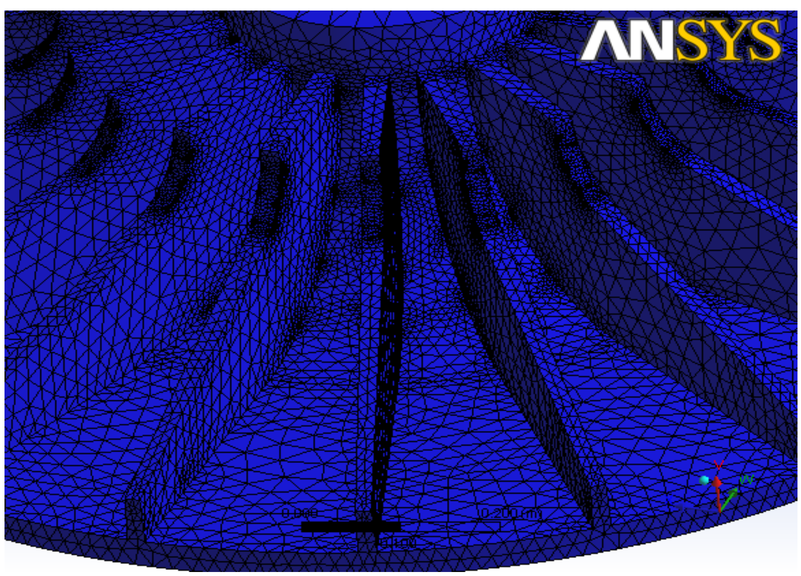


Figure 3.6: Closer view of meshing on the impeller

3.3.3 Boundary Conditions

After finish with meshing, the analysis continues with boundary condition or environments which are called in ANSYS® Workbench. As for static stress analysis, the boundary condition used is to represent the real operation environment of the impeller inside the compressor. In ANSYS® Workbench, there are four environment modules available which are Inertial, Loads, Supports and Condition.

A full load was applied to the whole impeller of pressure and temperature. The pressure was distributed by 20 blades on the impeller thus the load exerted on every single blade is 3.5e+005 Pa since the maximum working pressure is 7e+006Pa (70 bar), which the value are based on maximum working pressure of the sample studied compressor, LMC 313 Single-Stage Centrifugal Compressor manufactured by Sundyne Corporation. While for Conditions, the Thermal Condition is set to 120°C. Finally, the Supports for the impeller are selected using combination of 157 faces between base plate and blades. All of these analysis environment settings are shown in Figure 3.7 below.

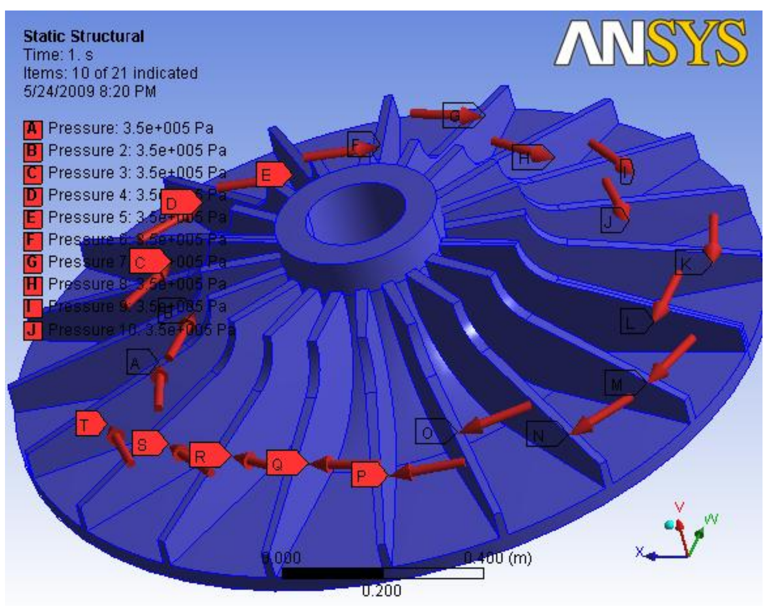


Figure 3.7: Boundary condition of the analysis on the impeller

CHAPTER 4
RESULTS AND DISCUSSION

4.1 Stress Distribution

In the solution part, the process is mainly on simulating the results or stress distribution on the impeller. The Equivalent Stress or also known as von-Mises Stress is the key to be observed on the stress distribution. The results yield maximum value of 1.9979e+009 Pa and minimum value of 30.767 Pa. From Figure 4.1, obviously the stress is more distributed on the blades and the base plate of the impeller. Clearly as it seen, the stress yielded on the impeller is not critical which most of the stress are considered as low-stress. The contour in blue and light blue are majoring the distribution while some mid-stress (contoured in green) are distributed on the base plate of the impeller, within blades (8.8797e+008 Pa).

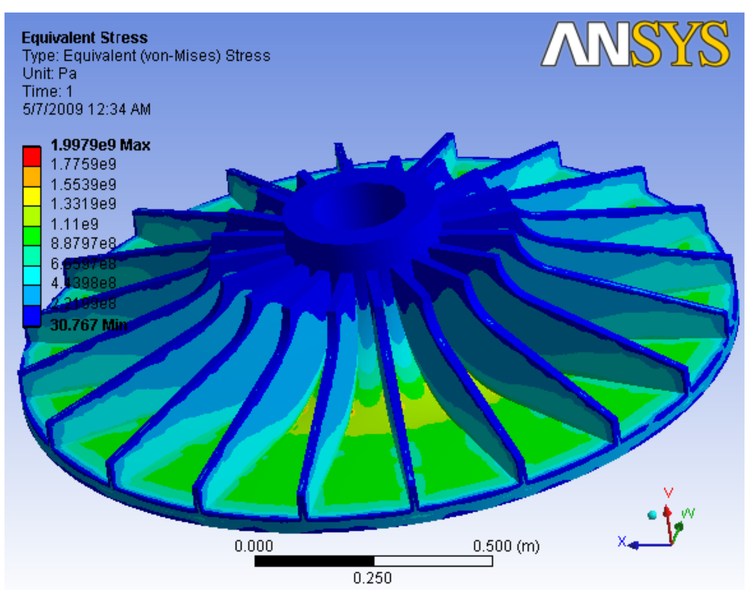


Figure 4.1 Equivalent (von-Mises) Stress distribution on the impeller

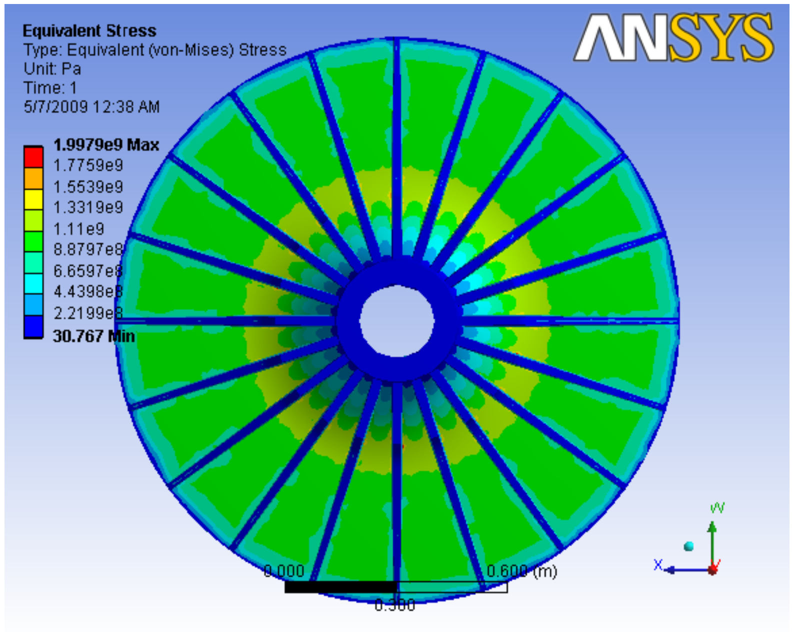


Figure 4.2: Top view of stress distribution on the impeller

4.2 Stress Concentration

From Figure 4.3, it is shown that there is a very small region of high-stress (maximum) area concentrating at the right between the joint of blades and base plate which are contoured in red colour. The value for maximum von-Mises stress is 1.9979e+009 Pa. It is covered up with also small region of yellow contoured stress around the maximum stress area. Take note, the high-stress area are concentrating where the blade region area are higher which means in the middle of the blades where and also the blades and base plate curvature is critical. Finally, the localized areas of maximum stress can be said as the possible sites to start crack initiation.

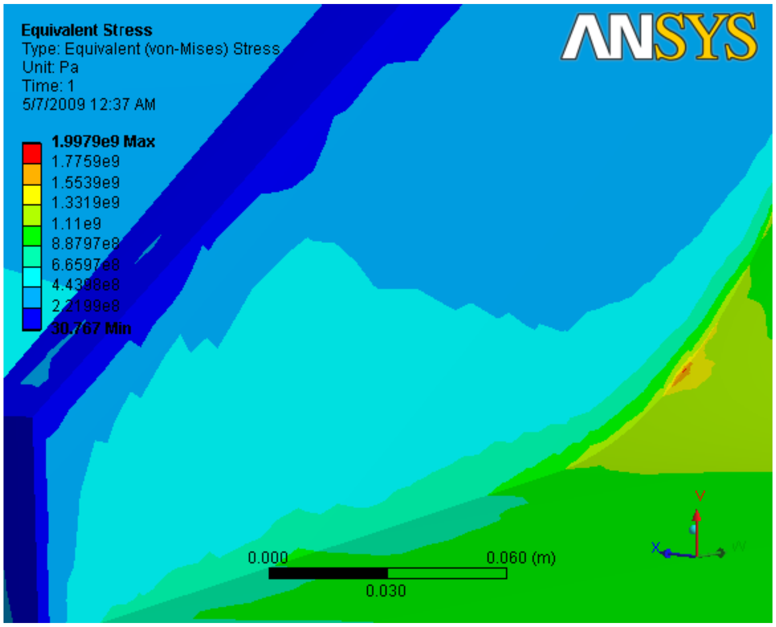


Figure 4.3: Concentration of Equivalent (von-Mises) Stress on the impeller (front side)

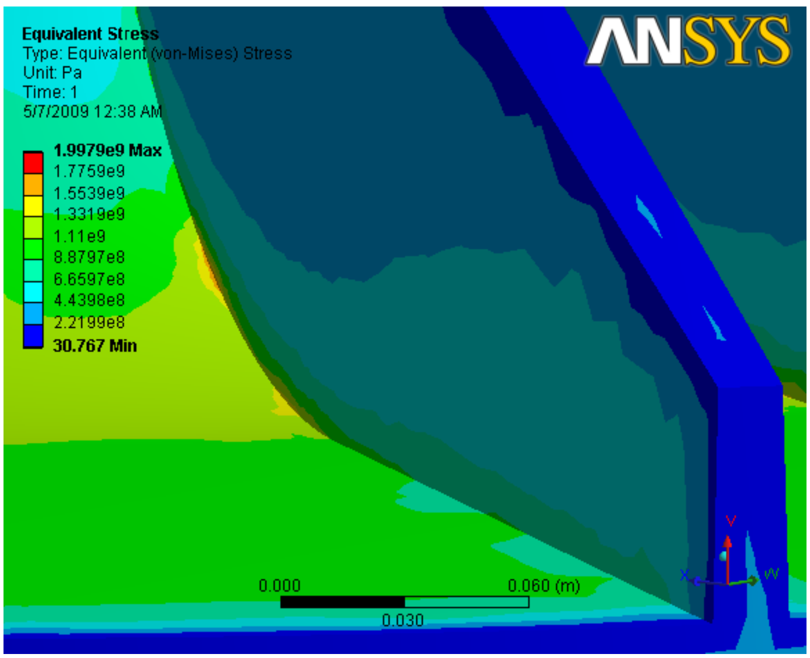


Figure 4.4: Concentration of Equivalent (von-Mises) Stress on the impeller (back side)

CHAPTER 5
CONCLUSION AND RECOMMENDATION

5.1 Conclusion

Impeller can be considered as the critical part in every compressor. It is the key of any gas compression in compressor. Thus, it is required to conduct any studies regarding the impeller. The stress distributions on the impeller are more likely concentrated on the middle section of the impeller, where the curved shapes of the body are crucial. The high stress region obviously concentrated at the joint of blade and body or base plate. The difficulties during the analysis was the lack of computer memory resources since it is affecting the results especially in mesh refinement since the ANSYS® Workbench 11 was run just on personal notebook.

5.2 Recommendation

As for recommendation, the further study should be carried to analyse the stress distribution and concentration on the other two types of centrifugal impeller vanes, forward-curved and backward-curved vanes. It also recommended that a further detailed theoretical study need to be carried on to investigate the behaviour of the stress distribution and stress concentration of the impeller such the effects of the vanes on the stress behaviour. The analysis can be improved theoretically by using Boolean method on the blades for applying the load since the load (pressure) is not constant along the blades. It is the limitation of ANSYS® Workbench. The analysis also can be improved by using cyclic symmetry method which can reduce the workload of CPU and time.

REFERENCES

1. Robert A Adey, John M. W. Baynham, Sharon Mellings, Tom Curtin, 2003 "**Fatigue Life and Crack Growth Prediction Using FEM Data**"
2. Rama S. R. Gorla, Aijaz A. Khan, 2003, "**Turbomachinery: Design and Theory**" ISBN: 0-8247-0980-2, Marcel Dekker, Inc
3. Mike Stanko, Jeffrey M. Steele, 2008, " **Ansys Advantage**", Volume II, Issue 1, Rotating Machinery
4. Christian H. Roduner, Hans-Peter Dickmann, Dietmar Filsinger, 2004, "**Unsteady Flow in a Turbocharger Centrifugal Compressor: Three-Dimensional CFD Simulation and Numerical and Experimental Analysis of Impeller Blade Vibration**"
5. Benjamin D. Craig, 2005, "**Material Failure Mode, Part I**"
6. Kenneth H. Huebner, Donald L. Dewhirst, Douglas E. Smith, Ted G. Byrom, 2001, "**The Finite Element Method for Engineers**"
7. Henri Champliaud, L Van Ngan, 2002, "**Finite Element Analysis of Crowning Sealing Caps**"
8. E. Ayder, R.A Van Den Braembuscche,.J Brasz, 1993, "**Experimental and Theoretical Analysis of The Flow In a Centrifugal Compressor Volute**". ASME Journal of Turbomachinery, Vol 115, pp. 582-589
9. John R. Brauer, 1993, "**What Every Engineer Should Know About Finite Element Analysis**"
10. Teo Han Fui, Roslan Abd. Rahman, 2007, "**Statics And Dynamics Structural Analysis Of A 4.5 Ton Truck Chasis",** No. 24, 56 – 67
11. Meherwan P. Boyce, 2002, "**Centrifugal Compressor: A Basic Guide**", ISBN: 9780878148011, Pennwell Corp.

LIST OF FIGURES

LIST OF TABLES