COMPUTATIONAL METHOD TO ESTIMATE FATIGUE LIFE AND DURABILITY OF SPUR GEAR BY USING STRESS-LIFE (SN) AND MEASUREMENT OF ROOT STRESS DATA

FINAL REPORT RDU150336

PROJECT LEADER : DR DAING MOHAMAD NAFIZ DAING IDRIS

FACULTY OF MECHANICAL ENGINEERING
UNIVERSITI MALAYSIA PAHANG

ACKNOWLEDGEMENTS

I am grateful and would like to express my sincere gratitude to my project members for their germinal ideas, invaluable guidance, continuous encouragement and constant support in making this research possible. My sincere thanks go to all my students and members of the staff of the Mechanical Engineering Department, UMP, who helped me in many ways and made the research target achieved. Many special thanks go to Automotive Excellence Centre and Department of Research and Innovation, Universiti Malaysia Pahang for their monetary contribution, cooperation, inspiration and support during conducting this research.

ABSTRAK

ABSTRACT

Kajian ini merupakan ilmu asas tentang tekanan pada gigi gear taji nipis berbingkai dalam "finite element" analisis dan kajian teori. Kebanyakkan orang industri kurang mempunyai kesedaran tentang had tekanan pada gigi gear lebih-lebih lagi tentang gear taji nipis berbingkai. Kebanyakan aplikasi dalam dunia kejuruteraan memerlukan komponen yang ringan atas tujuan yang tertentu. Kajian ini bertujuan untuk mengesahkan ralat peratusan antara Kaedah "Finite Element" dengan kajian teori, dengan menggunakan persamaan "AGMA Design". Perisian Abaqus telah digunakan dalam simulasi ini untuk mendapatkan hasil kajian dalam FEA. Untuk melengkapkan ini kajian, model simetri dan model tidak simetri telah dibina dalam perisian CAD, dan kemudian kedua-dua model tersebut telah diimport ke dalam perisian Abaqus. Terdapat dua belas set untuk kedua-dua model dengan dua ketebalan web yang berbeza dan tiga ketebalan rim yang berbeza telah disimulasikan dengan jayanya dalam perisian Abaqus. Mengikut kaedah analisis dan berangka, nilai tekanan pada gigi gear meningkat dengan kerap apabila ketebalan rim telah dikurangkan dari 8mm untuk 4mm. Kesilapan peratusan bagi model simetri dan model tidak simetri telah dikira dan disahkan dengan keputusan FEA.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

P	**Diametral pitch**
z	**Number of teeth**
d	**Pitch circle diameter**
m	**Module (mm)**
p	**Circular pitch (mm)**
a	**Addendum**
b	**Dedendum**
c	**Clearance = b – a**
d_o	**Diameter of addendum circle or Outside diameter = (d + a)**

LIST OF ABBREVIATIONS

ISO	International Organization for Standardization
LTCA	Loaded Tooth Contact Analyses
HCRG	High Contact Ratio Gears
AGMA	American Gear Manufacturing Associations

CHAPTER 1

INTRODUCTION

1.1 Project background

In this chapter 1 will discuss about the project background, problem statement, objective of the research and scope of research. For information, this research is about to investigate the tooth root stresses of thin-rimmed spur gears based on Finite Element Dynamics Analysis.

Gears are already well-known as one of the oldest items in the engineering world. They have existed since the development of rotating machinery. Because of their force-multiplying properties, previous engineers used them for lifting huge and weighted loads, example construction materials.

The previous gears were made up from wood with cylindrical pegs for cogs and were often lubricated with animal fat lubricant. They were also been applied in wind and water wheel machinery for decreasing or increasing the provided rotational speed for application to pumps and other powered machines. An early gear adjustment used to function up the textile machinery is illustrated in figure 1.1. Basically, the rotational speed of a water wheel was too slow to use. As the solution to speed up the rotation, a set of wooden gears has to been use.

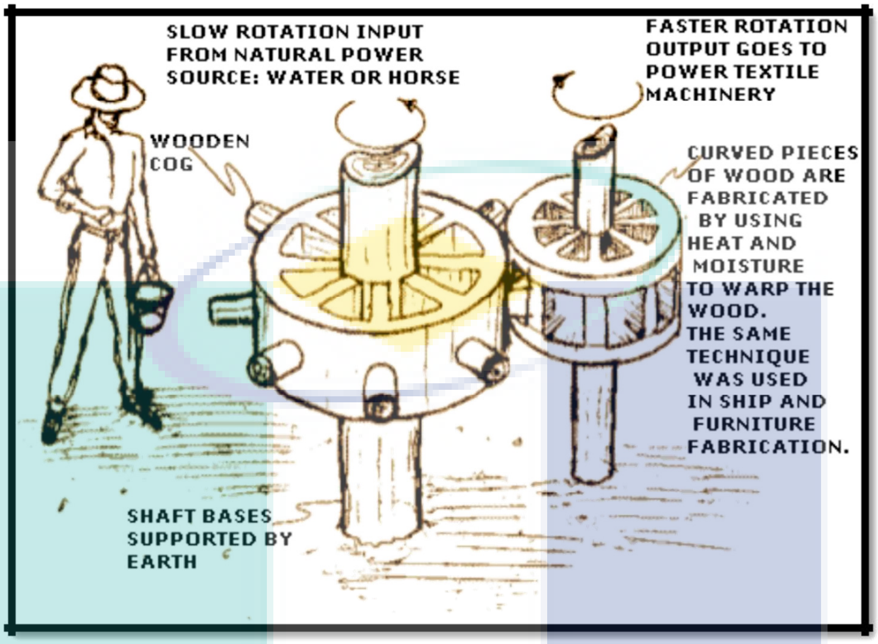


Figure 1.1: An early gear adjustment

Source:http://www.efunda.com/designstandards/gears/gears_history.cfm
(retrieved on 27 April 2017)

Today, the most important new gear inventions are in the area of materials. Modern design has greatly increased the useful life of industrial and automotive gears, and consumer electronics has driven plastic gearing to new levels of grease-free reliability and quiet operation.

A gear can be defined as a machine element that was used to transmit motion and power between two parallel rotating shafts. They are the most common means of transmitting power to the current mechanical engineering world. Most of the gears applications are used in automotive, aircraft and machinery world. Furthermore, these gears are flexible in tiny size application such as in watch. There are four principal types of gears: spur, helical, bevel and worm gears(Richard G. Budynas, 2011).

The position of the shaft in worm gear is located not parallel to each other. There is a circular spiral gear in a worm and a wheel worm. As analogy, it is like a spring in spiral binding. The wheel may have two types; they are either tooth angles or straight form. The power input has been supplied to worm and at wheel can get the output based on the necessary. Figure 1.2 shows the worm gear.

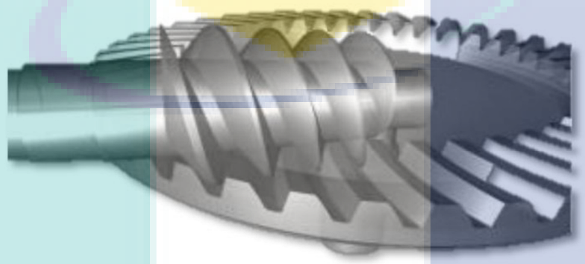


Figure 1.2: Worm Gear

Bevel gears, as shown in figure 1.3 are used to carry the power from one direction to another direction. To understand how the bevel gear work easily, where the power created by one shaft is in X-axis and the power needs to be transferred into 90°, for example to Y-axis. In these gears are having some slope and angled position.

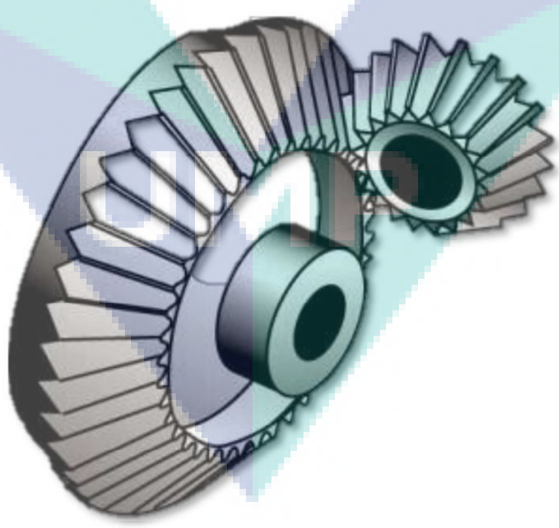


Figure 1.3: Bevel Gear

Helical gears as shown in figure 1.4. Tooth of these gear have some angles with respect to the axis of the shaft. The degree of this angle must be in the range of 30-50°. If the angle is not in the range, they cannot meet to each other's and cannot transmit the power effectively. This condition can affect the lifetime of the gear.

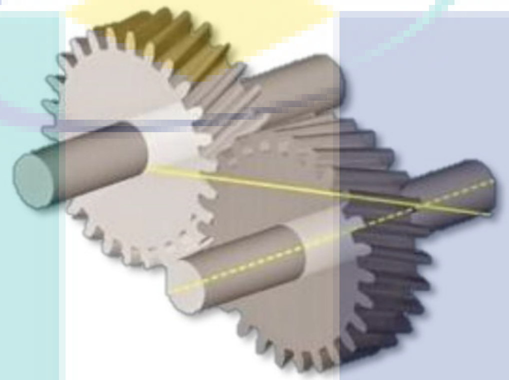


Figure 1.4: Helical Gear

The last one is spur gear, as shown in figure 1.5. Spur gears have straight tooth and they parallel to the axis of the shaft. An example application of these gears is used to transmit the power in the gearbox of automobiles. It is used to transmit more power compare to others type.

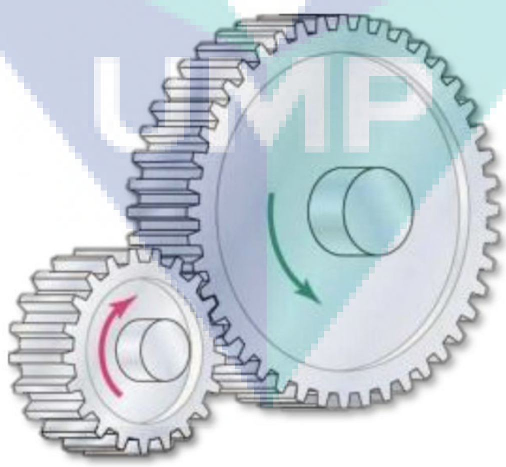


Figure 1.5: Spur Gear

Presently on this research work is focused on the spur gear. People also called spur gear as straight gear. Basically, it will be used to develop the primary kinematic relationship of tooth form. That is why, the spur gear is the simplest design among the others gear. Figure 1.6 shows the solid spur gear, which are widely used in industry.

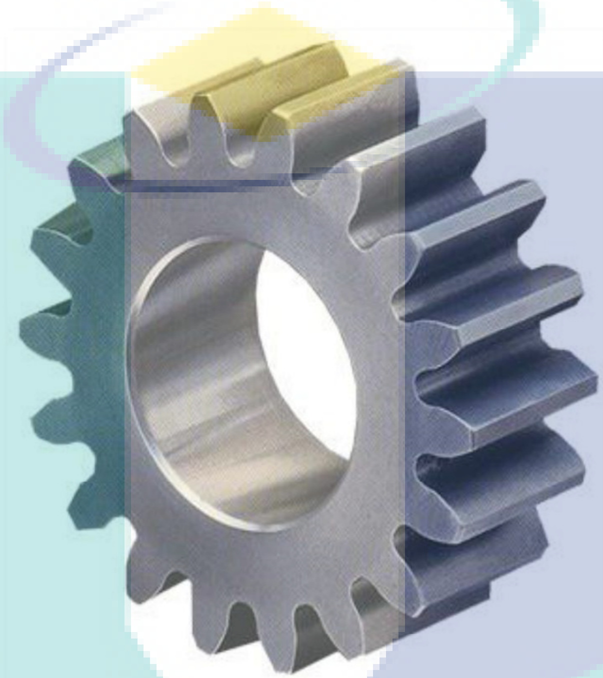


Figure 1.6: Solid Spur Gear

1.2 Problem statement

In recent times, a complicated and comprehensive gear has been designed. Nowadays, there are very few thin-rimmed spur gears in an industry. Normally, people in the industry are using solid spur gear, which is heavy in weight as shown in the previous page. Consequently, the main focus of this research is on thin-rimmed spur gear. Present day gears must be designed in thin-rimmed for a certain aspiration. In automotive, aircraft and machinery required lightweight part to optimize the usage of fuel and the efficiency of the engine can be increased. Furthermore, the manufacturing cost can be minimized for thin-rimmed gear. The bending strength can be affected by: the gear size, described by the diametral pitch; the shape of the tooth, described by the number of teeth on the gear; the highest location of the full load, described by the number of teeth on the mating gear; and the fillet geometry of the gear tooth. For thin rim gears, the thickness of the rim is another powerful cause which influences the bending strength of the gear. Therefore, the thickness of the rim is optimized to achieve light weight in aircraft purpose.

1.3 Objective of the research

To validate the simulation result of tooth root stresses of spur gears based on Finite Element Analysis in Abaqus software with the theoretical studies of AGMA Design Equation.

1.4 Scope of research

In this section is to summarize the scope of research. Previously, the historical knowledge, definition of gear, problem statement and objective of the research has been explained. In this research, to simulate the thin-rimmed spur gear model in Abaqus software to verify the error percentage compare with the theoretical studies to show the limitation of applied stress on tooth of thin-rimmed spur gear.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

This chapter reviews about previous research that related with objective and scopes of research. The literature study conferred current research about tooth root stress of gear. From the studies, it showed that over the years, there are huge numbers of simulation and experimental research about this topic.

2.2 Basic Fundamental about Spur Gear

Spur gear is a gear that having involutes teeth either straight or helical on a cylindrical surface. These gears work in pairing, because they use to transmit power, between two parallel shafts. Spur gear has wide range of application in engineering world. Normally, they are used in metal cutting machines, power plants, marine engines, washing machines, material handling equipment, automobiles gear boxes and so on. Modern gears are a clarification of the wheel and axle. Gear wheels have projections called teeth that are designed to intersect the teeth of another gear. They are in mesh, when gear teeth fit together or interlock in this manner. They are capable to transmit force and motion alternately from one gear to another. The drive gear is the gear transmitting the force or motion and known also as pinion. While the driven gear is the gear connected to the drive gear. Figure 2.1 shows the basic geometry of spur gear(Richard G. Budynas, 2011).

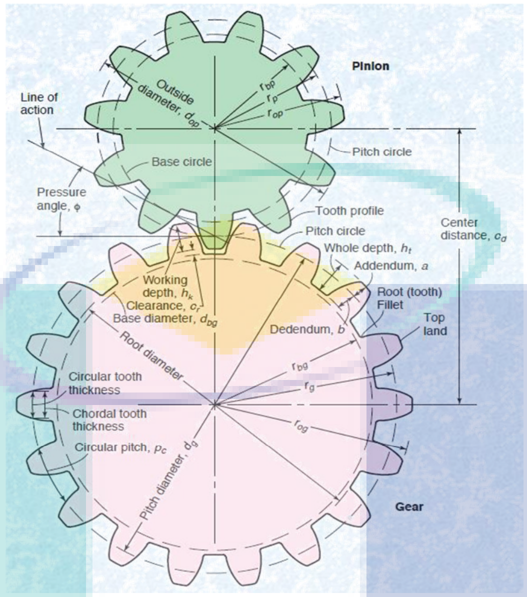


Figure 2.1: Geometry of gear

The nomenclature of spur gear is illustrated in figure 2.2. The pitch circles of meshing gears have to be tangent to each other. The circular pitch, p correlates to the distance, measured on the pitch circle, from one point on the tooth to a corresponding point on the other adjacent tooth. In simple words, the circular pitch is summation between the thickness of a tooth and the space between two adjacent teeth measured along the pitch circle(Gear, Force, & Fig, n.d.). The following notations are used in spur gears(as shown in box):

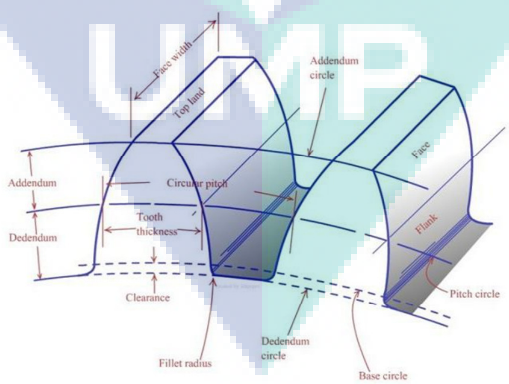


Figure 2.2: The nomenclature of spur gear

P = Diametral pitch	z = Number of teeth
d = Pitch circle diameter	m = Module (mm)
p = Circular pitch (mm)	a = Addendum
b = Dedendum	c = Clearance = b - a
d_o = Diameter of addendum circle or Outside diameter = (d + a)	

Normally, internal spur gear teeth are stronger compared to pinion teeth with same pitch and face width, since at the base the external teeth are tiny in size. However, ring gears, where made from a material with a lower strength than that of the meshing pinion may be loaded more desperately in bending stress. In this research study, a model for the bending strength is constructed for an internal gear tooth. It could be as a function of the applied load pressure angle. The model was based on the inscribed Lewis fixed strength parabolic beam. The geometry of the Lewis factor determination is presented, the iteration to identify the factor is illustrated and the bending strength J factor is matched to that of an external gear tooth. This strength model will help excellent design efforts for different addendum gears and gears with various type of materials(Savage et al., 1995).

2.3 Application of Spur Gear in Industry

The first thing to have before design gears is having technical knowledge and it is necessary as mechanical element. Even though use the designed standard gears. The basic knowledge such as formulas to calculate the strength and size of gear, types of gear are commonly used in application, properties of gear (mechanical, physical) and the most important part is mechanical drawing of gear, where to know special geometric symbols and geometric tolerances(Pippan et al., n.d.). Normally, people in industry used special software in designing certain gear. It is because the supplier has to fulfil various orders with different parameter from the contractor and customer. By using the software, it can make the system more efficient and systematic. The most common gears to visualize easily are spur gears, where they are used to transmit motion between two parallel shafts. For extra information, there is no thrust force produced in the axial direction, since the axes of the mounted shafts are parallel to the tooth surfaces of the gears. The spur gears can be created to high degree of precision, because they are easy to produce(Conrado & Davoli, 2007). As specified by ISO, the unit display the sizes of spur gear and named as "module". Usually, the pressure angle is set to 20 degrees in these recent years. To use the portion of an involute curve as the tooth profile is the most common thing in commercial machinery. There are two conditions that require the usage of the profile. Firstly, when the center distance slightly changes or secondly, when the teeth of gear not strengthen. So, it is used to shift the gear. To produce the profile tooth is by manipulating the distance between cutting tool of gear (hobbing tool) and gear in production phase. The tooth root stress increases, when the shift is positive. Otherwise happen when the center distance being reduces and the shift is negative. When two gears are meshed, the backlash plays between the teeth and it also needs for smoothing gears rotation. The vibration on gears leads to increase, if the backlash is huge. It also can create problem if the backlash too small, because it will lead to tooth failure due to the lack of lubrication(Kohara Gear Industry Co., 2015). The figure 2.3 shows the common gears that used in industry.

Figure 2.3: Gear in industry

2.4 Experimental of Spur Gear

This previous research is about the design and analysis of a prototype of tooth impact test rig for spur gear. The main focus is to study the limitation and capabilities of spur gear. To achieve the stated objective, a test rig was fabricated. From the designed rig, an analysis was conducted to ensure that there is no problems occur during run the test. The responding variable like safety factor and stresses on rig was recorded, when the maximum load was applied on it. These two responding variables are most significant factors for design consideration of test rig. This test rig is 1.25 meters in height as shown in figure 2.4. The gear specification as stated in Table 2.1. Among the three materials as stated in table, AISI 1040 is the proposed material for this test rig because AISI 3215 and AISI 4140 are expensive compared to AISI 1040. After did the analysis, there were some limitations that exceed the specification in this test and it started from in design stage. In future, some improvement needs to do in designing of test rig, so then it will work properly(Ghazali, Aziz, Idris, Ismail, & Sofian, 2016).

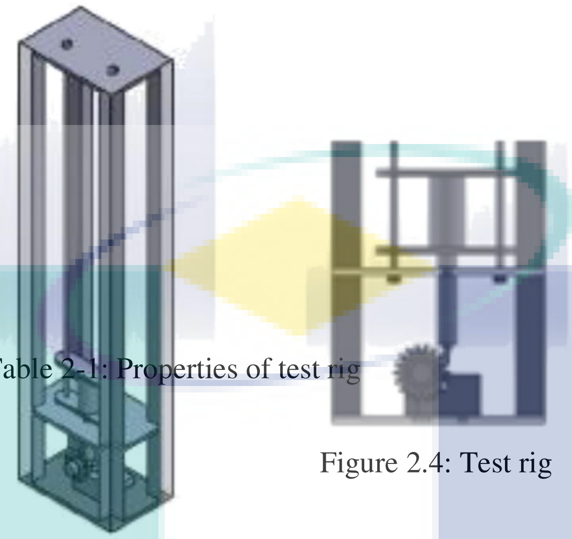


Table 2-1: Properties of test rig

Figure 2.4: Test rig

Table 2-1: Properties of test rig

Parameters	Dimension
Module, m	4 mm
Pressure angle, α_0	20°
Number of teeth, z	18
Face width	10 mm
Materials	AISI: 1040, 3215, 4140

This researcher conducted an experiment of 3D, thin-walled spur gears and did an analysis of finite element. The aim of this study is to analyse the resonance frequency behaviour of spur gear. There were three models of spur gears with different wall thickness in this experiment. SCM415 was assigned as material of these three models. To measure the vibration on the models, a power-circulating-form like figure below was constructed and the speed range is about 500-3000rpm. When the thin-walled spur gears were running in a resonance state completely, the resonance mode shapes must be identified by presenting in strain phase method. An equivalent model system of lump-mass was built, to analyse the torsional vibration frequencies of the rig system. The analysis was about mode shapes and natural frequencies of the model system. Finite Element Method calculation investigated the web thickness, rim thickness, gear hub, effect of tooth module, and face width on natural frequencies. Among the others parameter, the web and rim thickness of thin-walled spur gear are the biggest effect on structural frequencies(Li, 2008b).

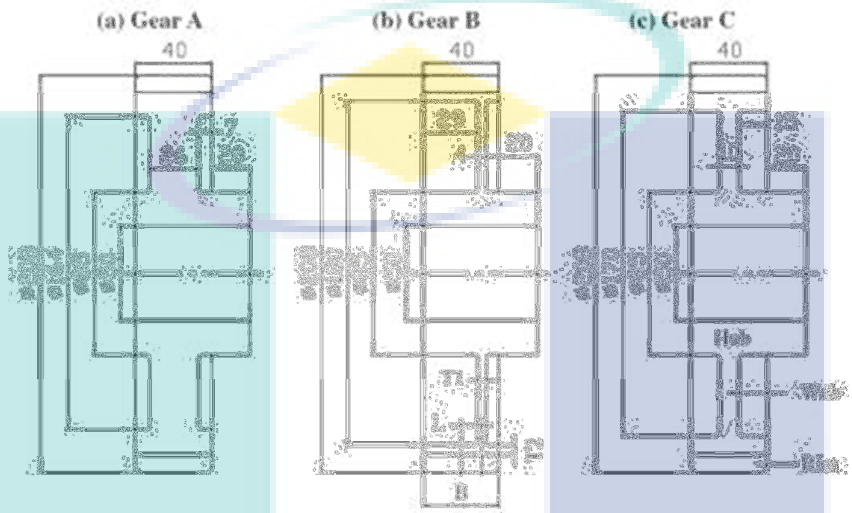


Figure 2.5: Types of gear

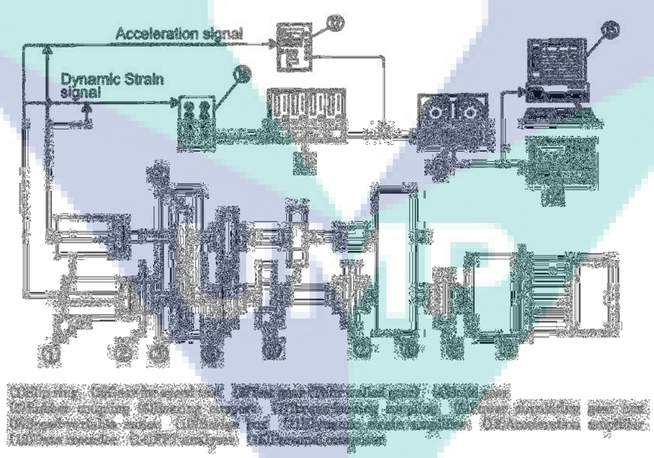


Figure 2.6: Power-circulating-form

2.5 ABAQUS Software

The previous researcher investigated about bone and did finite element analysis by using two different softwares. ABAQUS is a new software tool that can assign the material properties and it compares with a similar tool, Ansys finite element models. Software tool of this researcher (py_bonemat_abaqus) was written in Python, where this language can be read by ABAQUS. The main focus of this investigation was to compare the software packages in terms of the material assignment calculation and processing speed. There were three element types were assigned to be compared; linear hexahedral (C3D8), linear tetrahedral (C3D4) and quadratic tetrahedral elements (C3D10). Processing speeds were evaluated for the three different element types. Even though ABAQUS was slower to complete the simulation of the models, but will provide the users with a useful and helpful method to assign or create material properties of bone to finite element models. This software also come with fewer limitations, an easier workflow, and as it is written in Python, it can be directly integrated into scripts written to interact with ABAQUS(Pegg & Gill, 2016).

This study investigated the functions and features of a software tool called Fatigue Prediction Utility for Abaqus/CAE (FPU). It is constructed as a plugin for the Abaqus commercial Finite Element (FE) code allowing for fatigue estimation based on the results of Abaqus FE analyses. It also has an interface for data transfer between Abaqus and PragTic, a standalone fatigue post-processor. The program has a simple and intuitive graphical interface, which makes arrangement of a fatigue analysis, easy to understand. The FPU fatigue solver may be performed on multiple CPU cores independently from Abaqus/CAE, which allows huge-scale problems to be solved on server machines in tolerate able time. As an overview, ABAQUS is a software that can help the researcher, because it has a lot of helpful main features(Neslàdek & Španiel, 2017).

2.6 Thin-Rimmed Spur Gear

Basically, application like aircraft transmission, lightweight and high-power is used thin rim spur gears. The main focus of this research study was to obtain the limit of bending stresses on thin-rimmed spur gear compared with solid gear. When spur gear deformed from solid to thin-rim, the bending stresses on the tooth fillet and root areas will be differ. For this research study, rim thickness is the significant parameter for the gears. Normally, a gear unit keys used to protect the gears and shafts, so the moment on the shafts can be transmitted among them. As well as the thickness of the rim become thinner, the strength of the keyway become more important in its application(Kametani & Umezawa, 1984). The finite element analysis was only done on a segment of the gear. The magnitude and location of maximum bending stresses on tooth gear are changed when the rim thickness was being reduced. This study can conclude as, by decreasing the backup ratio of the rim can increase the maximum alternating and maximum compressive bending stress on tooth gear(Bibel, Reddy, Savage, & Handschuh, 1994).

Mechanical elements subjected to cyclic loading have to be designed against fatigue. The focus of this study is to analyse the bending fatigue life of truck gearboxes. It is because in truck gearboxes are used thin-rimmed spur gear. The gear service life is divided into two conditions. The first one is the initiation stage of the damage accumulation and the second one is the crack growth. There are two methods to show the result, which are the finite element method and the boundary element method. The continuum mechanics based approach is used for the estimation of the fatigue process initiation stage. The linear-elastic fracture mechanics is being used to evaluate the remaining life of gear with an initial crack(Kramberger, Šraml, Potrč, & Flašker, 2004).

This researcher conduct a study, to investigate the effect of addendum on tooth contact strength, bending strength and basic performance parameters of spur gears. To conduct loaded tooth contact analyses (LTCA), deformation and stress calculations of spur gears with different addendums and contact ratios, the researcher used together the face-contact model of teeth, mathematical programming method (MPM) and three-dimensional (3D), finite element method (FEM) in this study. To achieve the objective, analysis of the tooth load, load-sharing rate, contact stress, root bending stress, transmission error and mesh stiffness of the spur gears must be done. Not to forget, the effects of addendum and contact ratio on gear strength and basic performance parameters are also have to discuss. A standard gear tooth without shifting will look like figure 2.7. (k_m) and (k_m + ck_m) are stand for addendum and dedendum of the gear. Module is expressed as m, coefficient of addendum is k and Here, m is module, k is called addendum coefficient and ck is symbol for tooth clearance coefficient (by referring figure 2.7). Calculation of tooth contact stress is done by using Hertz formula and this formula can only calculate it approximately. While to conduct the tooth contact stress analysis, can use FEM. The increment of addendum can increase number of contact teeth, then root bending stress and tooth contact stress can be reduce by this increment. Unfortunately, it make also the depth of tooth become longer and lets the tooth easy to deform, scoring strength calculations of the tooth tip and root becomes necessary when to use high contact ratio gears (HCRG) with long addendum(Li, 2008a).

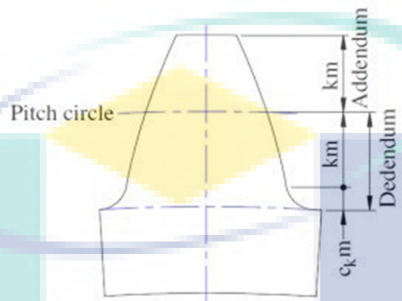


Figure 2.7: Standard gear tooth without shifting

When spur gear deformed from solid to thin-rim, the bending stresses on the tooth fillet and root areas will be differ. For this research study, rim thickness is the significant parameter for the gears. A gear, where made with a rim and spokes is used in this conditions, as the rim thickness factor. Normally, failure can happen on the rim rather than on the tooth root for these cases. The focus of this research study is to predict the critical area when the pressure angles and backup ratios are manipulated by using simulation and compare the obtained results by previous researchers. ANSYS is used to obtain asymmetric and symmetric model of tooth spur gear in finite element model. From that, the effect of bending stress at the critical area was studied for different backup ratios. The magnitude and location of maximum bending stresses on tooth gear are changed when the rim thickness was being reduced. Then, the obtained result is validated with the Lewis Bending Equation(Mallesh, Math, P, & K, 2009).

In this study, the effect of centrifugal deformation and stresses on thin-rimmed spur gears being analyzed with inclined webs and at very high speeds run the gears in the Finite Element Method. From the method, showed that at high speeds, the centrifugal load could effect on tooth root bending stresses of gears. When the gear speed is greater than 10,000 min− 1, need to assign it as high speed. To conduct Loaded Tooth Contact Analyses (LTCA) and calculate tooth load distribution, a mathematical programming method needs to use. Mathematical programming method is used to conduct LTCA and calculate tooth load distributions of a pair of spur gears. Frictions between the teeth surfaces are assume as frictionless, since the transmission of gear has a very good efficiency, it is about 95–98%. The root bending stresses and tooth contact stresses and of the thin-rimmed gears are computed under a moment load. (Li, 2008a) found that the centrifugal deformation of the thin-rimmed straight web gears could influence the root bending stresses and tooth contact stresses and of the contact teeth(Li, 2013).

This previous study is presented about a three tooth sector of thin-rimmed spur gear in two-dimensional (2D) finite element stress analysis as shown figure 2.8 and at the radial end of the rim teeth was set up the boundary conditions. To obtain the stress variation, the diametric ratio must be manipulated from 1.02 to 2. From this analysis, the von misses stresses and Maximum Principle stresses are considered at the fillet and the root of the gear. The aim of this research is to figure out the lowest limit of spur gear's thickness, when the diametric ratio Md was being changed. Lewis equation will give analytical results and stress analysis in finite element will obtain from package MSC PATRAN and NASTRAN. The geometrical construction was done on the CATIA software. Thus, the minimum thickness of thin rim spur gear is 1.2, as instructed by the strength criteria. From the obtained result, both Von Misses and Maximum Principle stresses (the variation of stresses) is significant(Yenarkar, 2013).

Figure 2.8: Two-dimensional (2D) finite element

Shear stress cycles and repeated compression can cause contact fatigue, where it is one of the main failure modes on gear tooth flanks. Hertzian theory was being used in this study to investigate the surface and subsurface stresses of gear teeth and analyze using the finite element method. To predict fatigue crack initiation, the number of loading cycles is needed. Based on the multi-axial fatigue mechanism, it can be predicted by using the Smith–Watson–Topper method. The main objective of this research study is to figure out either the high stresses or strain during meshing process can cause the fatigue crack on tooth gear. Friction shifted the distributions of the von Misses stresses, increased the von Misses stresses on the surface, and changed the extreme values of the shear stress cycles, where it influences the contact stresses. When the corresponding position reaches the surface as friction increases, the minimal fatigue life will decrease. So then, the tip relief of the teeth is proposed to decline the stresses on the points and improve the initiation fatigue life on tooth gear(Qin & Guan, 2014).

To reduce weight in particular applications in engineering world, webbed and thin rimmed gears are used. An accurate and fails safe design is needed in this type of gears. The cracks may reproduce either through the rim or through the tooth. When the wheel geometry parameters were above all the rim thickness, it will influence the direction of crack propagation. The three ranges in literature are emphasized in studies for values of backup ratio by differentiate the behaviours. The ranges were through the rim, through the tooth and in an unpredictable way, where relevant to the crack propagation directions. Other parameters such as both geometry and loading conditions can influence the last doubt zone. The relationship between bending stress and the effect of wheel speed has been investigated in this study. The model of four teeth thin-rimmed spur gear is constructed in two-dimensional by MSC Partan/Nastran software and being extended finite element models (XFEM). In order to minimize the simulation period, the gear tooth model had to construct in 2D as shown in figure 2.9. This study found that, centrifugal load can cause crack propagation direction and initiate the crack point on the gear tooth(Curà, Mura, & Rosso, 2015).

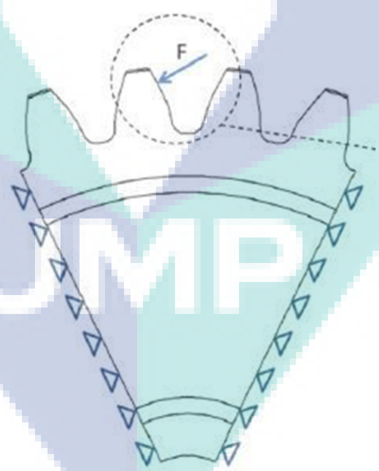


Figure 2.9: Two-dimensional of gear tooth

The International Organisation for Standardisation ISO 6336:2006 and the American Gear Manufacturers Association AGMA 2101-D04 are the two most predominant empirically based analytical gear stress analysis methods for establishing contact and root bending stress in involute spur and helical gears. These two method were also being used in the previous research to improve the fillet capacity in bending for asymmetric model(Senthil Kumar, Muni, & Muthuveerappan, 2007). Although based on the same fundamental principles, they have evolved to such an extent that, for various reasons, they are not necessarily in agreement. The use of commercial universal numerical finite element software has become an increasingly popular alternative for gear stress analysis and this research com- pares the root bending stresses in external spur gears established using ISO 6336:2006, AGMA 2101-D04 and numerical finite element analysis (ANSYS) with experimental strain gauge validation as shown in figure 2.10(Lisle, Shaw, & Frazer, 2017).

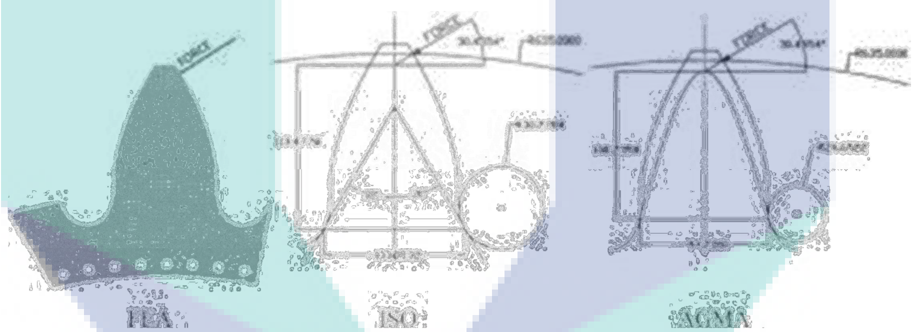


Figure 2.10: Gear tooth in three different methods

CHAPTER 3

METHODOLOGY

3.1 Introduction

As an overview, to investigate the bending stress on tooth thin-rimmed spur gears, there are three methods need to be explored. First method is by calculating the root stress using theoretical studies, which is American Gear Manufacturing Association (AGMA) Design Equation. Imported the thin-rimmed spur gear from CAD software into ABAQUS software, adjust and create the setting and run the computational simulation, where this is the second method. Last method is by designing an impact test rig of thin-rimmed spur gear and conduct the experiment. The comparison between these three methods can validate the percentage errors among them.

In less than six months of duration, only two methods were capable and available to finish up. The methods are computational simulation and theoretical studies. The finite element result of tooth root stress of thin-rimmed spur gear will obtain from ABAQUS software. Then, the FEA result is validated with the AGMA Design Equation. The percentage error between these methods can give early prediction for experimental.

3.2 Finite Element Method

The finite element method (FEM) is a powerful numerical technique for obtaining approximate solutions to a wide variety of problems in engineering and science. It is also known as finite element analysis (FEA). FEM separates a huge problem into smaller, simpler, parts called finite elements. Applications range from deformation and stress analysis of automotive, aircraft, spacecraft, building and bridge structures. It includes the use of mesh generation techniques for dividing a complicated problem into tiny in size of elements, as well as the use of software program coded with FEM algorithm. Some general function of finite element software available in the market includes: NASTRAN, PANSTRAN, ABAQUS, ANSYS, ALGOR, LS-DYNA and much more.

There are two types of approaches that were used to solve the finite element problem, which is Implicit and Explicit. The Implicit approaches use in problems in which time dependency of the solution is not an important factor, such as static structural, harmonic. Explicit Dynamics approach uses to help most in solving high deformation time-dependent problems like crash, blast and impact.

The use of FEA mainly aims to improve the design of a product or a component by analysing the effect of the loading such as mechanical, thermal or other, the structure will be subjected to during use. The result from FEA can be used to improve the design. The purpose of this study is to investigate the tooth root stress on thin-rimmed spur gears.

3.3 Abaqus

ABAQUS is one of the finite element analysis modules that offer effective and entire solutions for both routine and sophisticated engineering problems covering wide in range of industrial applications as shown in figure 3.1. The analysis modules which are completing and integrated analysis tools are called as the heart of ABAQUS. It is a general function FEM program that can solve a variety of problems. For example, in the automotive industry engineering, the work groups are capable of considering full vehicle loads, dynamic vibration, contact stress, impact or crash.

Theoretically, ABAQUS comes with two solvers, which are Standard and Explicit; which can be used to run difference of simulations. Simulations are set up either sketch the model in ABAQUS itself or import model from others software (in format .Parasolid) by using keywords that define the functioning of simulation. ABAQUS also allows users to create the materials using a variety of models. The user also can control over the meshing and the element types that used in the model. The huge advantage of ABAQUS is to allow the modelling at a high level of detail. Moreover, the software is instruction-line approachable and supports scripting functionality.

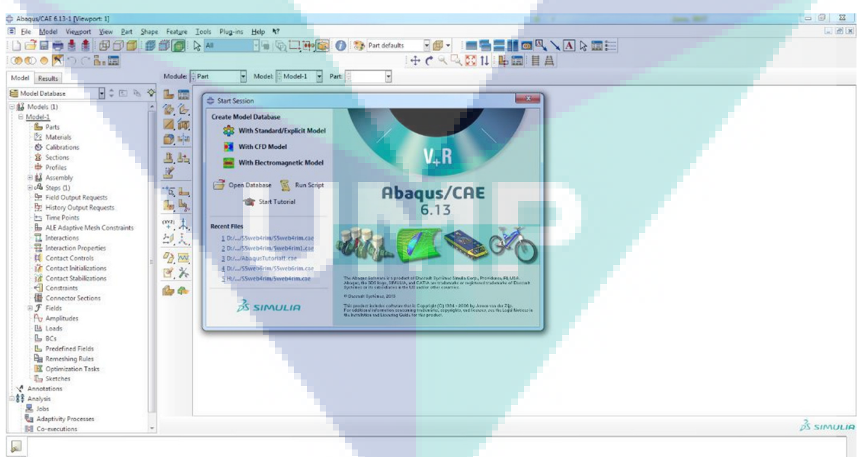


Figure 3.1: Abaqus 6.13 Software

Throughout this study, complete models of thin-rimmed spur gear were analysed using ABAQUS/Explicit V6.13. In general, there are three phases in any computer aided engineering task which includes pre-processing or modelling, processing or finite element analysis and post processing or generating a report, image, animation and so on.

3.4 Gear Design

As an overview for this research study, there are two designs were created for this study, which are symmetric and asymmetric. For each design, there are six sets of model, where the thickness of the rim and web for each set are different. Firstly, the models were designed in CAD software, then the models being imported into ABAQUS software.

Before proceeding with simulation, the design of the gear must be settled up first. The thin-rimmed spur gear was designed based on basic involutes profile due to relative simplicity and the both gear teeth use the same tooth profile. This fundamental knowledge is stated in the textbook.

3.4.1 Gear Pair Design

A different number of teeth and pitch circle between the driven gear and driving gear have been decided for these models. The material for these models is AISI 3215 carbon steel. Table 3.1 shows the detail of design geometry and material properties for gear pair.

Table 3-1: Gear geometry and material properties

Parameter	Driven Gear (Thin-rimmed gear)	Driving Gear (Solid spur gear)
Number of teeth	18	24
Module (mm) m_n	4	
Pressure angle (degree) ϕ	20	
Face width (mm) F	30	
Pitch circle (mm)	72	96
Addendum circle (mm)	80	104
Dedendum circle (mm)	62	86
Material	AISI 3215	
Density (Kg/m^3) ρ	7700	
Young's Modulus (Gpa) E	207	
Poisson Ratio v	0.29	

3.4.2 Thin-rimmed design

For this study, two designs created in this simulation, which is symmetric and asymmetric. Figure 3-2 shows the arrangement of the web between two designs a) symmetric and b) asymmetric.

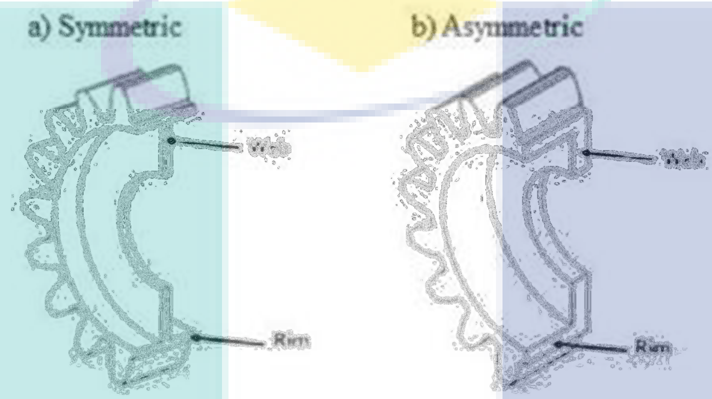


Figure 3.2: Arrangement of the web between two designs

3.4.3 Dimension of Rim and Web

As mentioned above, there are two designs for this simulation and table 3.2 and 3.3 show the dimension for the both designs.

Table 3-2: Dimension of rim and web thickness for symmetric

Gear Structure	Solid	Symmetric					
Web thickness	-	5mm			2.5mm		
Rim thickness	-	8mm	6mm	4mm	8mm	6mm	8mm

Table 3-3: Dimension of rim and web thickness for asymmetric

Gear Structure	Solid	Asymmetric					
Web thickness	-	5mm			2.5mm		
Rim thickness	-	8mm	6mm	4mm	8mm	6mm	8mm

3.5 Computer Simulation

During the course of this research, the development of the computer simulation is the most critical stage in investigating the tooth root stress of thin-rimmed spur gear. By doing the setting for this computer simulation, the user must be very careful by inserting the value for each parameter. Even a small error can affect the duration of the simulation. Sometimes, the simulation took four days to complete running. Figure 3-3 shows the simple steps by setting parameter in the computer simulation.

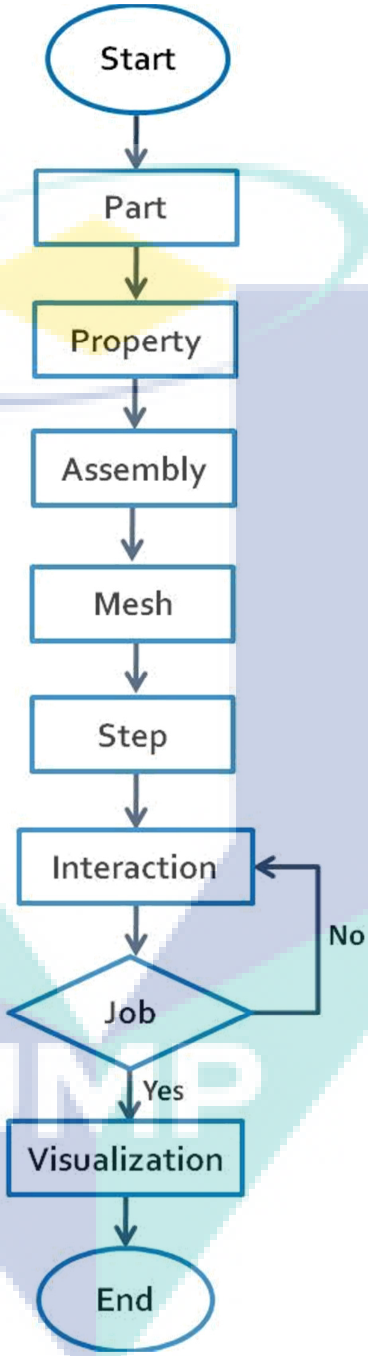


Figure 3.3: Flow chart for computer simulation

3.5.1 Part Module

In Part module, the user either can do the model's sketching directly in ABAQUS or importing model from CAD software. For this study, the designs have already designed in CAD software and then being imported into ABAQUS software. When opened this software, the first task is by completing the part module. In this module, all parameter of material value has to be inserted first. All general and mechanical behaviours such as name of material, mass density, Young's modulus and Poisson's ratio have to be assigned correctly with careful consideration on the consistency of unit(Pippan et al., n.d.). All these values can refer in Table 3.1. Table 3.4 shows the consistent unit in ABAQUS and this simulation use SI (mm).

Table 3-4: Consistent unit in ABAQUS

Quantity	SI	SI (mm)	US Unit (ft)	US Unit (inch)
Length	m	mm	ft	in
Force	N	N	lbf	lbf
Mass	kg	tonne (10^3 kg)	slug	lbf s^2/in
Time	s	s	s	s
Stress	Pa (N/m^2)	MPa (N/mm^2)	lbf/ft^2	psi (lbf/in^2)
Energy	J	mJ (10^{-3} J)	ft lbf	in lbf
Density	kg/m^3	tonne/mm^3	slug/ft^3	lbf s^2/in^4

3.5.2 Property Module

Material properties have been accredited in Part module and these properties will be assigned to respective parts accordingly in property module. For create Section, must click AISI 3215 as material, the category is solid and the type is Homogeneous. To assign the section, right click on "Section Assignment", and click create as shown in figure 3.4.

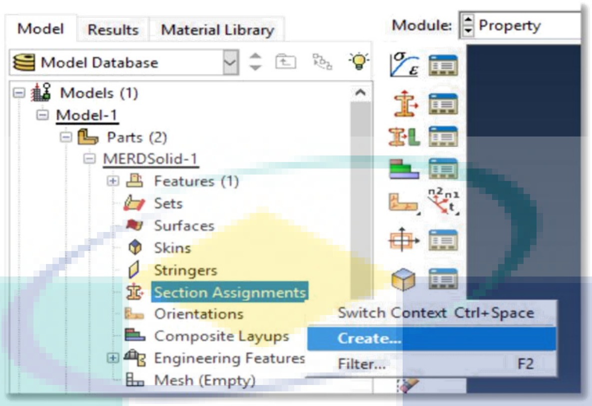


Figure 3.4: Create Section Assignment

3.5.3 Assembly Module

Once the both parts have been assigned to its respected properties, the instance parts are then being imported into the assembly module to assemble the parts. Create the instances by clicking right click, select the both parts and click "OK". The figures 3-5, 3-6 and 3-7 show the window before and after assembly.

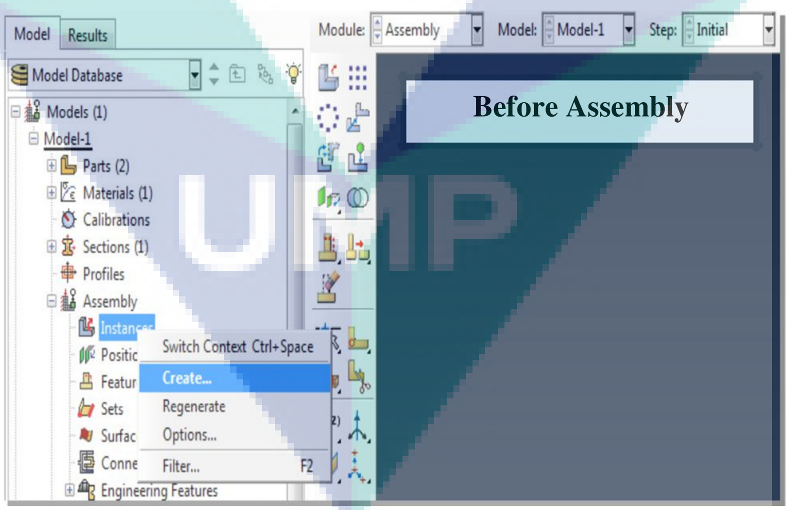


Figure 3.5: Create instances

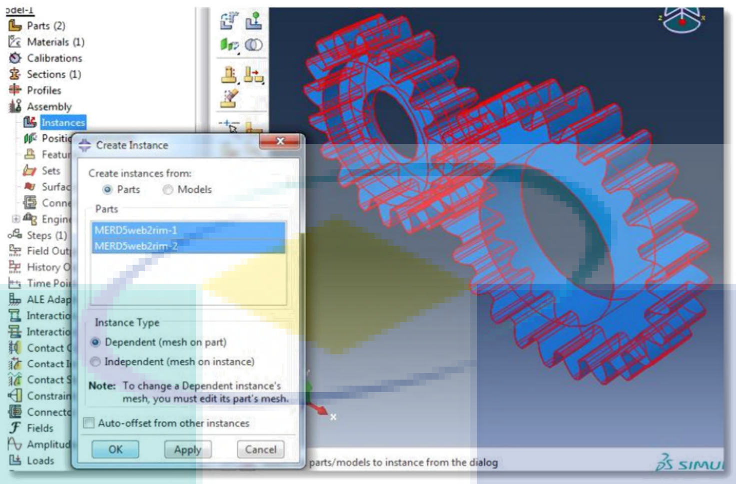


Figure 3.6: Select the both parts

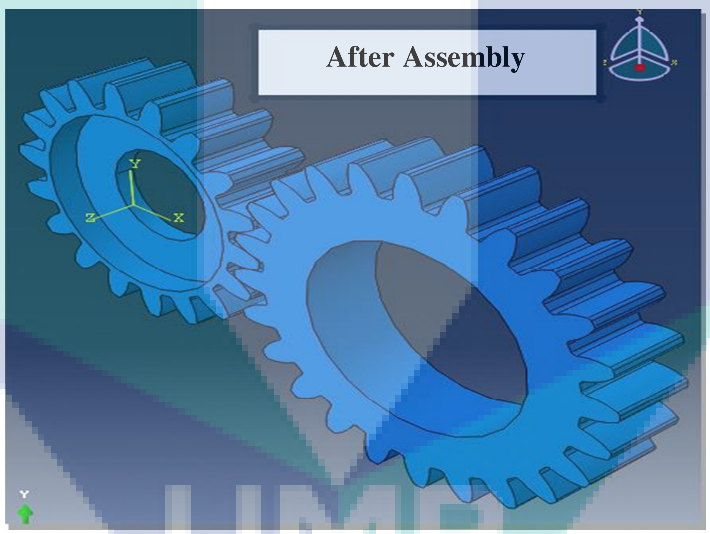


Figure 3.7: Done create instances

3.5.4 Mesh Module

Before proceed by doing meshing, the partition on the teeth must be sketched first. In this simulation, four teeth on thin-rimmed spur gear have been assigned as critical part. The purpose of the partition is differentiating between critical part and not critical part. Hence, the user can easily observe, where the maximum stress occurred. The figure 3-8 shows the four teeth have been assigned as critical part (in red circle).

In order to generate the mesh to every single part, the mesh characteristics of the parts are firstly defined such as mesh density, element shape and element type in the mesh control tool. A global size seed is defined according to the appropriate number of nodes and elements before meshing is generated. Normally, meshing on critical part is refiner compared to not critical part as shown in figure 3-9.

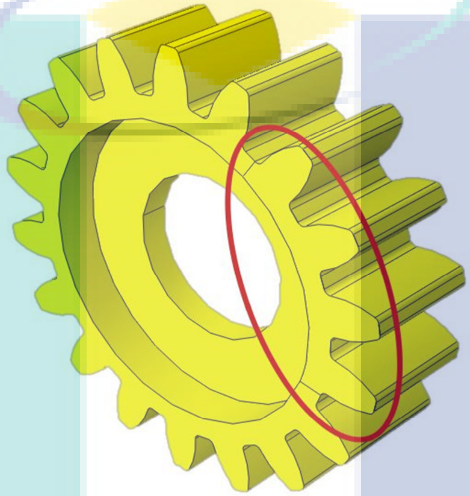


Figure 3.8: Partition on thin-rimmed spur gear.

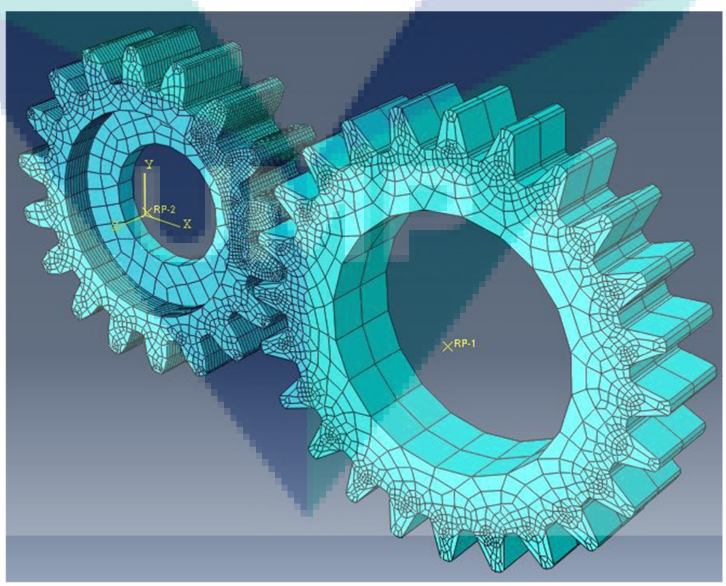


Figure 3.9: Meshing

3.5.5 Step and Output Definitions

By investigating the tooth root stress of thin-rimmed spur gears, a step of Dynamic, Explicit was created. The procedure named as Step-1 and the time period was set as 0.22s. There are two types of output, which are Field Output (FO) and History Output (HO). The entire outputs request for the whole model at every x unit of time is 0.001s. In Field Output, misses, stress and displacement are requested, while in History Output, defaults are set up.

Basically, FO needs Visualization module to view its data by using deformed shape, contour or symbol plot. Otherwise, HO used Visualization module to display result by using X-Y plots.

3.5.6 Interaction Module

Interaction module plays as the main role in this computer simulation. There are a few important setting in this module. If the simulation suddenly aborted while running, for sure there is wrong setting have been set up.

3.5.6.1 Interaction properties

To set-up the contacts between thin-rimmed and solid spur gear, the Interaction properties must be created. Tangential behaviour with frictionless and normal behaviour with "Hard" contact was set in contact property as shown on figure 3-10.

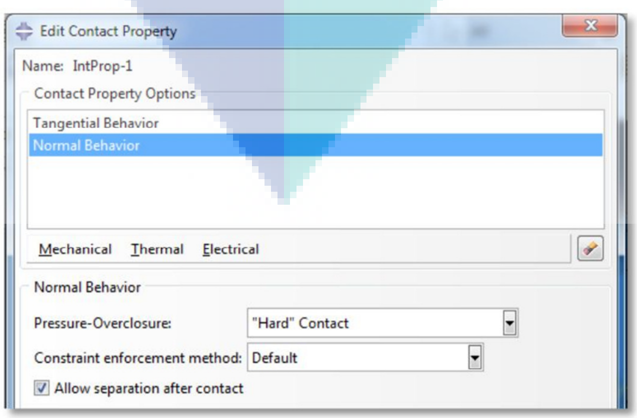


Figure 3.10: Interaction properties created.

3.5.6.2 Interaction

In this analysis, a General contact (Explicit) is used for the contact interaction which the ABAQUS will automatically define contact between teeth of the spur gears. This contact domain will be used for all exteriors surfaces of the parts and make a simple way in defining the contact interaction for the whole model.

3.5.6.3 Reference point

After settled set the both interactions, reference point (RP) on the gears must be assigned. To assign the RP, click a point on the axis of each spur gear and to make easy, select a point on the middle of the gears. They will automatically be named as RP-1 and RP-2 as shown in figure 3.11.

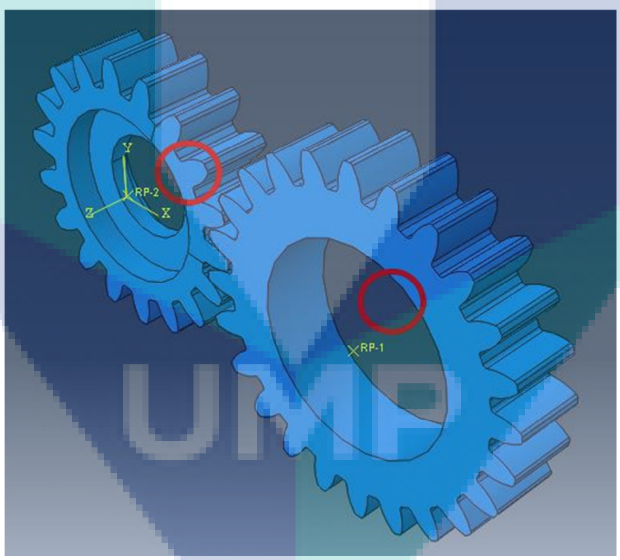


Figure 3.11: Reference point on both gears.

3.5.6.4 Constraint

In the root stress simulation, there might have several components need to be assigned some constraints in order to ensure that the simulation is similar to the real cases. The purpose is to constraint the degree of freedom of some interaction during the analysis. In this research study, the type of constraint is coupling. The both parts must be assigned as coupling constraint. The figure 3-12 shows how the spur gears look after created the constraints.

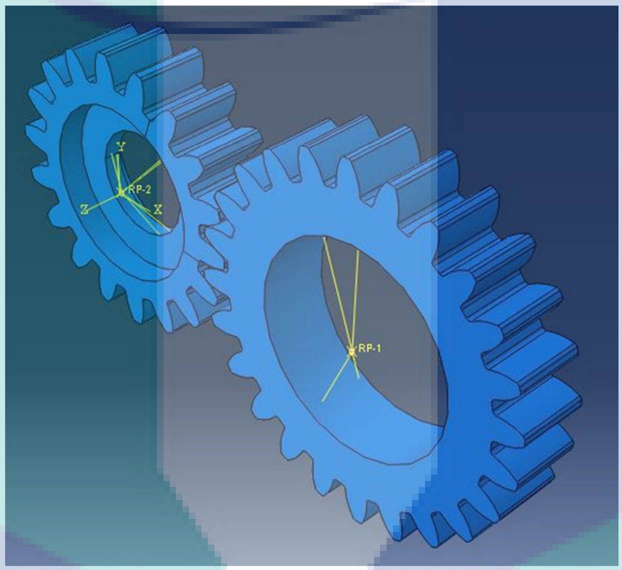


Figure 3.12: Constraints created

3.5.6.5 Load and Boundary Condition

As known, the solid spur gear act as driving gear and thin-rimmed spur gear is as driven gear. For driving gear, type of boundary condition (BC) is Velocity/Angular Velocity, the step is Step-1 and named it as BC-1. The values of V1-V3 are 0 and VR1-VR2 are 0 rad/s. As mention before, only four teeth are involved in this simulation, even though a complete model is designed. Angle for four teeth is 80° and unit for an angle in ABAQUS is in radian. When converted 80° into radian, it will be 1.39rad/s, where it is as angular velocity VR3 about the Z axis. BC-2 was assigned on driven gear, where its type is Displacement/Rotation. Do tick all displacements with value of 0, except UR3. It is because to allow the rotation about the Z axis.

3.5.7 Job Module

This module is the last part of the computer simulation. In this module will decide either the simulation is running smoothly or there is error occurred in it. When the simulation aborted while submitting the job, the user needs to make troubleshooting on Interaction module. Otherwise, the user also can monitor its progress. Then, the results of the completed analysis can be obtained by using ABAQUS in format .cae under Visualization module.

3.6 AGMA Equation Design

AGMA stress equation stands for American Gear Manufacturing Associations. In AGMA methodology, there are two fundamental stress equations are used, one for bending stress and the other one is for contact stress. For this research study, are focusing on bending stress. The AGMA formula for bending stress (1) is given in SI unit by

$$= \left(\frac{1}{\quad}\right)\left(\frac{\quad}{\quad}\right) \tag{1}$$

Where for U.S customary units (SI units),

W^t is the transmitted load; N

K_o is the overload factor

K_v is the dynamic factor

K_s is the size factor

b is the face width, m

m_t is the transverse module

K_H is the load distributor factor

Y_J is the bending strength geometry factor

K_T is the temperature factor

K_R is the reliability factor

K_B is the rim thickness factor

For case not solid gear, priority was given to the rim thickness factor KB [1] and this factor is influence by the value of the backup ratio, mB as stated in equation (2). The others factors in AGMA equation will remain constant and most of the factors will assume as 1.

$$= \begin{cases} 1.6 \ln \frac{2.242}{\quad} & < 1.2 \\ 1 & > 1.2 \end{cases} \qquad (2)$$

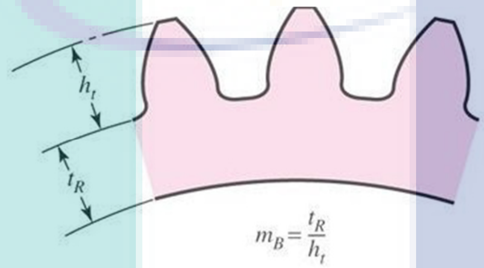


Figure 3.13: Backup ratio

CHAPTER 4

RESULT AND DISCUSSION

4.1 Introduction

This chapter presents the result based on the computational simulation of tooth root stress (bending stress) using finite element software. In this research, two designs of thin-rimmed spur gears, which are symmetric and asymmetric, have been simulated with different rim thickness and web thickness. As mention in previous chapter, the reading of bending stress is taken on the second tooth at critical part.

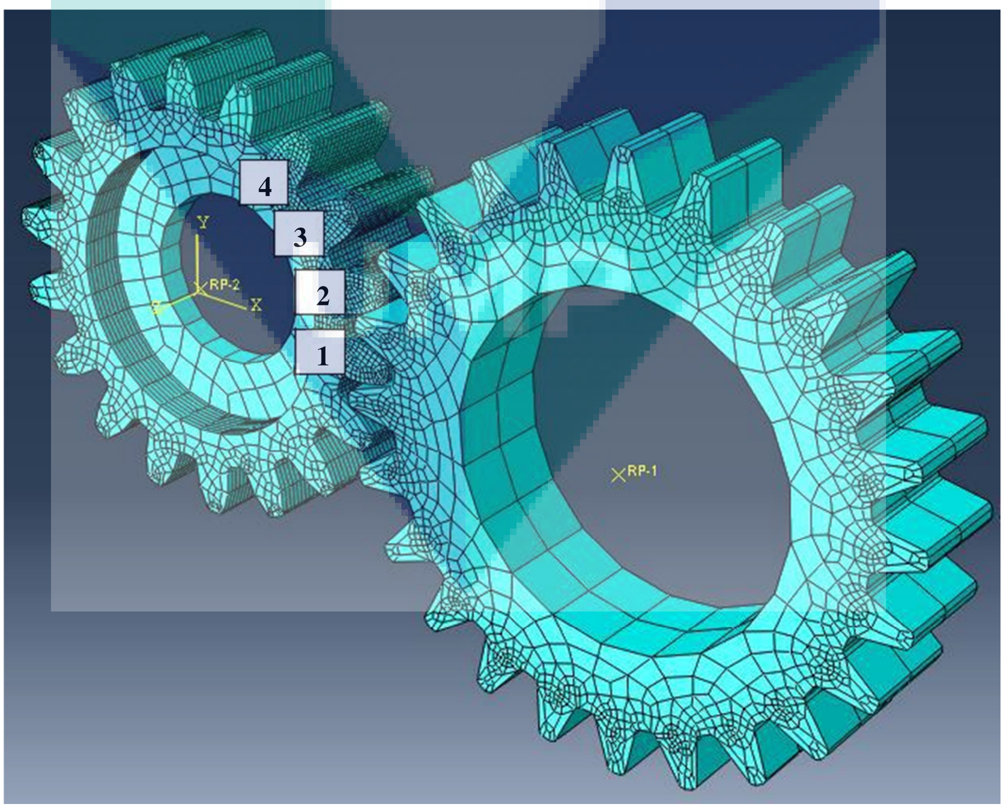


Figure 4.1: Position of taking reading

4.2 Tabulated Data for Symmetric Thin-Rimmed Spur Gear

Figure 4.2 shows the finite element analysis for symmetric model. The arrangement of the web for this model is on the middle. So, the critical bending stress is located on the middle too.

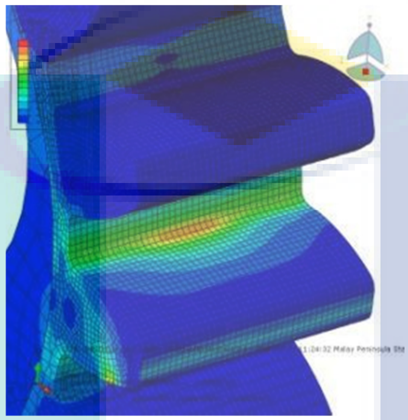


Figure 4.2: Finite element for symmetric model

4.2.1 Simulation in ABAQUS Software

Tabulated tables below show the value of bending stress that obtained from ABAQUS software. Table 4.1 shows the reading for 5mm web thickness, while table 4.2 shows the reading for 2.5mm web thickness.

Table 4-1: Bending stress for 5mm web thickness

Symmetric	Web Thickness (5mm)		
Rim Thickness	500Nm	375Nm	250Nm
solid	461.506	346.129	230.752
8	693.257	550.23	385.45
6	885.023	685.148	488.257
4	1003.23	903.398	622.265

Table 4-2: Bending stress for 2.5mm web thickness

Symmetric	Web Thickness (2.5mm)		
Rim Thickness	500Nm	375Nm	250Nm
solid	461.506	346.129	230.752
8	675.235	530.367	368.85
6	825.78	593.778	478.254
4	945.478	800.37	600.587

4.2.2 Theoretical Studies (AGMA)

Theoretical studies that being used in this research project is American Gear Manufacturing Association (AGMA) Design Equation. Table 4.3 shows the obtained values from the equation.

Table 4-3: Calculated values of bending stress

Symmetric	Transmitted Load		
Rim Thickness	500Nm	375Nm	250Nm
solid	468.417	351.312	234.208
8	693.257	519.942	346.628
6	904.044	678.033	452.022
4	1222.567	916.926	611.284

4.2.3 Percentage Error between Simulation and Theoretical Studies

After obtained the result from the both methods, a percentage error for web thickness 5mm and 2.5mm have been calculated. All values of percentage error are tabulated in table 4.4 and 4.5.

Table 4-4: Percentage error for symmetric (5mm)

Symmetric	Transmitted Load		
Rim Thickness	500Nm (5mm)	375Nm (5mm)	250Nm (5mm)
solid	1.475394787	1.475326775	1.475611422
8	0	5.825265126	11.19990307
6	2.103990514	1.049358955	8.016202751
4	17.94069364	1.475364424	1.796382696

Table 4-5: Percentage error for symmetric (2.5mm)

Symmetric	Transmitted Load		
Rim Thickness	500Nm (2.5mm)	375Nm (2.5mm)	250Nm (2.5mm)
solid	1.475394787	1.475326775	1.475611422
8	2.59961313	2.00503133	6.41090737
6	8.657100761	12.42638633	5.803257364
4	22.66452473	12.71160377	1.749923113

4.3 Tabulated Data for Asymmetric Thin-Rimmed Spur Gear

Figure 4.3 shows the result from finite element analysis that obtained in ABAQUS software. The arrangement of the web for this model is on the end of the model. It shows that the maximum reading of bending stress is at the end of the model (right side).

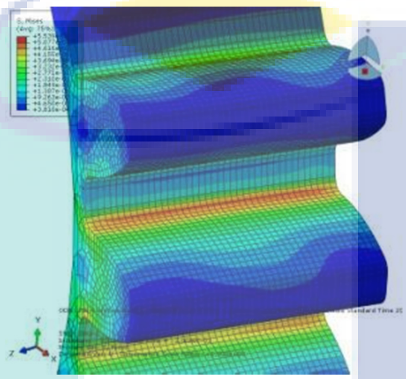


Figure 4.3: Finite Element for asymmetric model

4.3.1 Simulation in ABAQUS Software

Tabulated tables below show the value of bending stress that obtained from ABAQUS software. Table 4.6 shows the reading for 5mm web thickness, while table 4.7 shows the reading for 2.5mm web thickness.

Table 4-6: Bending stress for 5mm web thickness

Asymmetric Rim Thickness	Web Thickness (5mm)		
	500Nm	375Nm	250Nm
solid	461.506	346.129	230.752
8	683.257	555.368	400.875
6	858.587	605.148	470.257
4	958.65	915.358	602.265

Table 4-7: Bending stress for 2.5mm web thickness

Asymmetric Rim Thickness	Web Thickness (2.5mm)		
	500Nm	375Nm	250Nm
solid	461.506	346.129	230.752
8	668.245	540.367	393.258
6	840.559	587.25	578.254
4	905.478	805.268	620.47

4.3.2 Theoretical Studies (AGMA)

Theoretical studies that being used in this research project is American Gear Manufacturing Association (AGMA) Design Equation. Table 4.8 shows the obtained values from the equation.

Table 4-8: Calculated values of bending stress

Asymmetric Rim Thickness	Transmitted Load		
	500Nm	375Nm	250Nm
solid	468.417	351.312	234.208
8	693.257	519.942	346.628
6	904.044	678.033	452.022
4	1222.567	916.926	611.284

4.3.3 Percentage Error between Simulation and Theoretical Studies

After obtained the result from the both methods, a percentage error for web thickness 5mm and 2.5mm have been calculated. All values of percentage error are tabulated in table 4.9 and 4.10.

Table 4-9: Percentage error for asymmetric (5mm)

Asymmetric Rim Thickness	Transmitted Load		
	500Nm (5mm)	375Nm (5mm)	250Nm (5mm)
solid	1.475394787	1.475326775	1.475611422
8	1.442466502	6.81345227	15.64991864
6	5.028184469	10.7494768	4.034095686
4	21.58711956	0.171006166	1.475418954

Table 4-10: Percentage error for asymmetric (2.5mm)

Asymmetric Rim Thickness	Transmitted Load		
	500Nm (2.5mm)	375Nm (2.5mm)	250Nm (2.5mm)
solid	1.475394787	1.475326775	1.475611422
8	3.607897216	3.928322774	13.45246201
6	7.022335196	13.38917132	27.92607439
4	25.93632905	12.17742762	1.502738498

4.4 Graph of Bending Stress for Symmetric and Asymmetric

Result from Abaqus simulation was obtained in Finite Element Analysis (FEA) and focusing on Von Misses stress value at Hofer's critical area. This critical area is on tensile side at single tooth root, with 30° of tangential angle. Figure 4.4 shows the comparison of root stress between FEA results and AGMA standards for symmetric model with 5mm web thickness, while figure 4.5 shows the result for 2.5mm web thickness. Figure 4.6 shows the difference bending stress values for asymmetric with 5mm web thickness and figure 4.7 shows the reading for web thickness 2.5mm.

4.4.1 Symmetric

There are three different values of transmitted load, W_t have been decided for simulation and theory, which are 500Nm, 375Nm and 250Nm. For symmetric model, the root stress values are slightly differ between FEA and AGMA, even though the same load was transmitted on them. The highest FEA value of root stress with 5mm of web thickness is at 4mm rim thickness, where 500Nm load applied and the bending stress is 1003.23MPa. By referring figure 4.4 at rim thickness 8mm, the FEA result shows higher bending stress value compared to AGMA result when these three loads transmitted. Tooth root stresses at 4mm rim thickness had no big gap when 375Nm and 250Nm load applied. When the web thickness being shorten to 2.5mm, the root stress value declined by 57.75MPa at rim thickness 4mm.

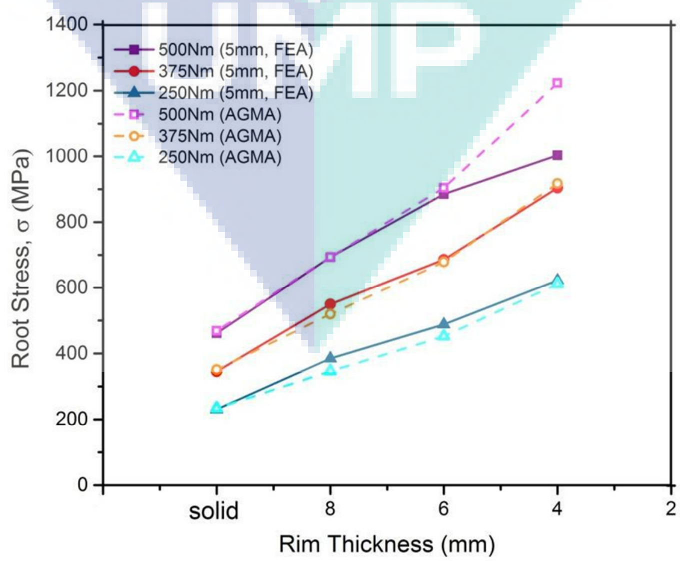


Figure 4.4: Graph of symmetric model with 5mm web thickness

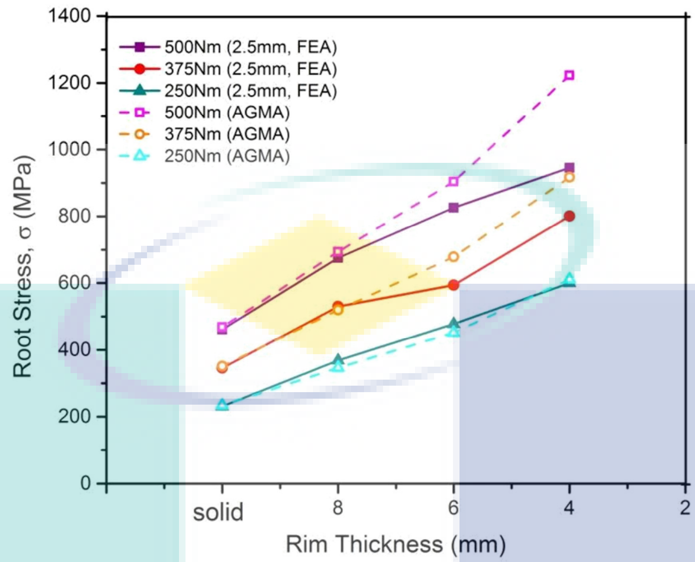


Figure 4.5: Graph of symmetric model with 2.5mm web thickness

4.4.2 Asymmetric

The same situation happened for asymmetric model. When thickness of the rim was being shortened, the root stress on the tooth will increase consistently. The highest FEA value of root stress with 5mm of web thickness is at 4mm rim thickness, where 500Nm load applied and the value is 958.65MPa. By referring the both graphs, the reading of tooth root stress at 4mm was significantly same between FEA result and AGMA result, when 250Nm load was applied. The value of bending stress at 4mm rim thickness experienced huge gap between these two methods. By doing calculation using AGMA equation, the tooth of thin-rimmed spur gear still can stand high bending stress, where can exceed 1000MPa, when 500Nm load was applied on it, but in computer simulation, the tooth only can stand for 900MPa at 4mm rim thickness. It became worst when the web thickness was being reduced, even though the same amount of load applied and at the same rim thickness. The tooth was only can stand for 800MPa. After did the analysis, the trend for asymmetric model are quick difficult to explain, because of that the design of asymmetric need some improvements, to make it more stable to simulate in software.

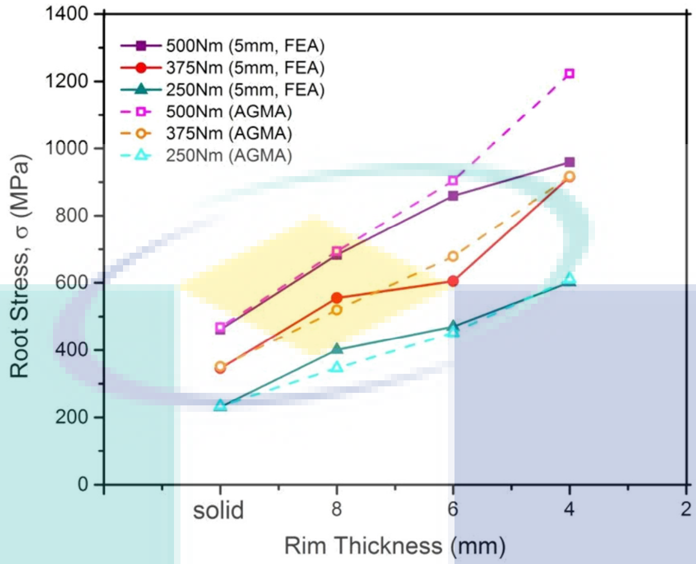


Figure 4.6: Graph of asymmetric model with 5mm web thickness

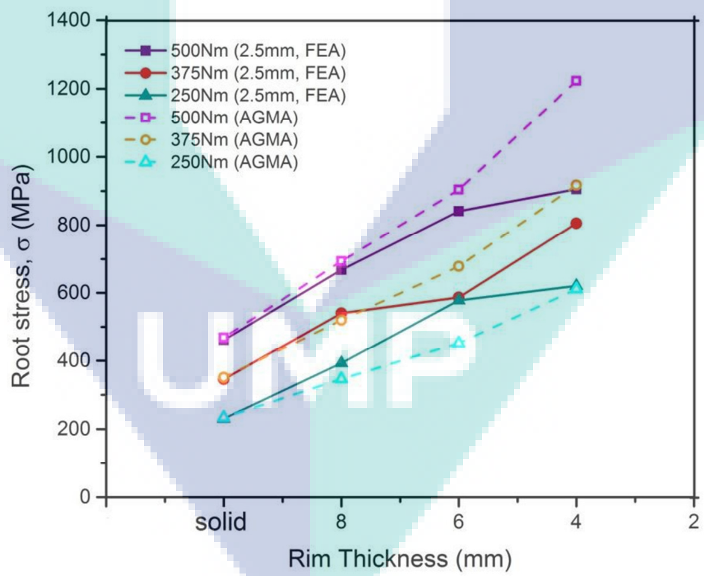


Figure 4.7: Graph of asymmetric model with 2.5mm web thickness

4.5 Graph of Percentage Error

Previous section explained about comparing FEA and AGMA result for each model. The figure 4.8 and figure 4.9 show the differences between percentage error of symmetric and asymmetric model. By referring the both graphs, it shows that, asymmetric model have huge percentage error compared to symmetric model. The highest value of percentage error for symmetric model was 22.66% at 4mm rim thickness, when 500Nm was applied on 2.5mm web thickness. A large percentage error for asymmetric model with 2.5mm web thickness is at 6mm rim thickness, where 250Nm load applied. The percentage error is about 27.93% as shown in figure 4.9.

Uniform beam theory is the guideline for AGMA standard in gear studies. Gear geometry of thin-rimmed was more complicated compared to uniform beam. It is because the profile of gear teeth was an involute curve. When the thickness of the rim manipulated, the value of root stress will respond constantly. To decrease the value of root stress on gear tooth, the rim thickness need to be expand. Otherwise, the root stress value being decreasing, when reduce the thickness of the web. From the result, it shows clearly that thickness of the rim and web can influence the value of the stresses.

Obtained result from FEA shows that tooth root stress can be influenced by the thickness of the rim. The thinner the rim thickness, the higher the root stress that tooth can stand. The value of Von Misses stress constantly increases with further reductions of rim thickness. When the web thickness was being manipulated, it will cause the reading pattern of bending stress value became more irregular. From this study, it shows that the web thickness and rim thickness are significant factors that can influenced the root stress on the gear.

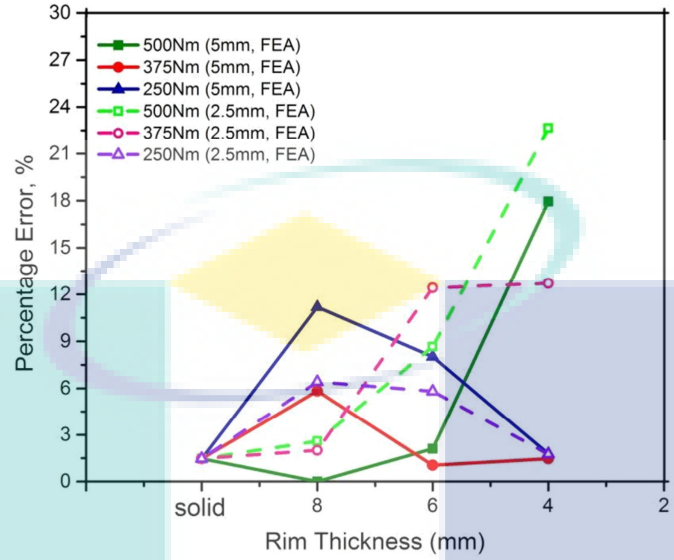


Figure 4.8: Percentage error for symmetric model

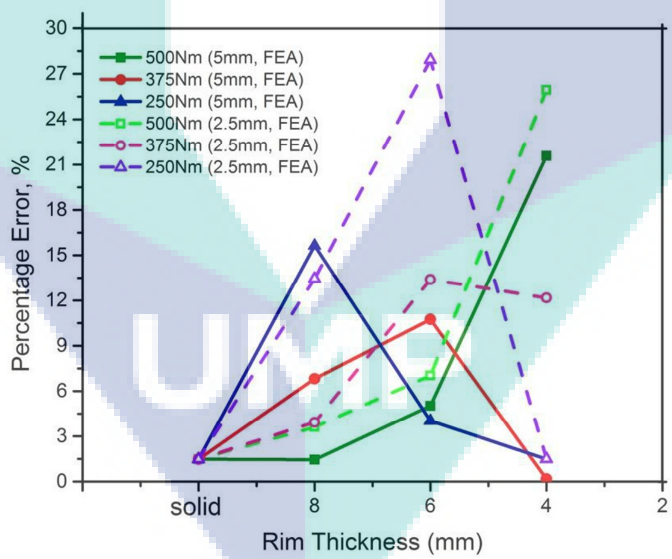


Figure 4.9: Percentage error for asymmetric model

CHAPTER 5

CONCLUSION

5.1 Conclusion

A numerical analysis of tooth root stress of thin-rimmed spur gear has been made using Finite Element Analysis software, which is ABAQUS/Explicit V6.13. Two different models of thin-rimmed spur were considered in this research work such as symmetric and asymmetric.

The theoretical study also has been made in this study by using AGMA Design Equation. Because of the various values of transmitted load, a system to calculate the bending stress was created in Microsoft Excel. The both methods are presented in the report and the obtained results have been analysed.

By reflecting the research objective, a good and an organized work could be creating. To achieve the objective completely, there were two scopes must be done. Firstly, by doing simulation of thin-rimmed spur gear model in ABAQUS software and before start the simulation, the model has to be imported from CAD software. Secondly, the obtained result from FEA need to be verify the error percentage with theoretical studies, which is using AGMA Design Equation.

Conferring to numerical and analytical studies, with the different rim thickness, simulated FEA could show the effect of root stress on the tooth thin-rimmed spur gear. The obtained readings have a similar pattern with calculated value from AGMA Design Equation. From this analysis, it shows that the reliability of FEA and could be used in determining the tooth root stress.

As conclusion, the value of the tooth root stress on gear can be affected by the thickness of the rim and web. These two parameters are the important aspect in gear design and application. By knowing the critical factor in gear, it will increase the awareness of limitation gear among industry's people.

5.2 Recommendation

The asymmetric model need to do some improvements, to make sure the model is stable when the simulation runs.

To validate and compare the obtained result from analytical and numerical part to actual experiment and the observation of any agreement in term of failure modes can be made.

REFERENCES

Bibel, G. D., Reddy, S. K., Savage, M., & Handschuh, R. F. (1994). Effects of rim thickness on spur gear bending stress. *Journal of Mechanical Design*, *116*(4), 1157. http://doi.org/10.1115/1.2919500

Conrado, E., & Davoli, P. (2007). The "True" Bending Stress in Spur Gears. *Gear Technology*, (August).

Curà, F., Mura, A., & Rosso, C. (2015). Effect of centrifugal load on crack path in thin-rimmed and webbed gears. *Frattura Ed Integrita Strutturale*, *9*(34), 447–455. http://doi.org/10.3221/IGF-ESIS.34.50

Gear, S., Force, T., & Fig, A. (n.d.). Module 2 - GEARS Lecture 7 - SPUR GEAR DESIGN. *Machine Design*, *6000*.

Ghazali, W. B. M., Aziz, I. A. B. A., Idris, D. M. N. B. D., Ismail, N. B., & Sofian, A. H. Bin. (2016). Design and Analysis of Tooth Impact Test Rig for Spur Gear. *IOP Conference Series: Materials Science and Engineering*, *114*, 12058. http://doi.org/10.1088/1757-899X/114/1/012058

Kametani, T., & Umezawa, O. (1984). Effects of Key on Bending Fatigue Breakage of Thin-Rimmed Spur Gear. *Chem. Pharm. Bull.*, *27*, 369–375. http://doi.org/10.1248/cpb.37.3229

Kohara Gear Industry Co., L. (2015). KHK Gear Stock. Retrieved June 14, 2017, from http://khkgears.net/

Kramberger, J., Šraml, M., Potrč, I., & Flašker, J. (2004). Numerical calculation of bending fatigue life of thin-rim spur gears. *Engineering Fracture Mechanics*, *71*(4–6), 647–656. http://doi.org/10.1016/S0013-7944(03)00024-9

Li, S. (2008a). Effect of addendum on contact strength, bending strength and basic performance parameters of a pair of spur gears. *Mechanism and Machine Theory*, *43*(12), 1557–1584. http://doi.org/10.1016/j.mechmachtheory.2007.12.010

Li, S. (2008b). Experimental investigation and FEM analysis of resonance frequency behavior of three-dimensional, thin-walled spur gears with a power-circulating test rig. *Mechanism and Machine Theory*, *43*(8), 934–963. http://doi.org/10.1016/j.mechmachtheory.2007.07.009

Li, S. (2013). Effects of centrifugal load on tooth contact stresses and bending stresses of thin-rimmed spur gears with inclined webs. *Mechanism and Machine Theory*, *59*, 34–47. http://doi.org/10.1016/j.mechmachtheory.2012.08.011

Lisle, T. J., Shaw, B. A., & Frazer, R. C. (2017). External spur gear root bending stress: A comparison of ISO 6336:2006, AGMA 2101-D04, ANSYS finite element analysis and strain gauge techniques. *Mechanism and Machine Theory*, *111*(October 2016), 1–9. http://doi.org/10.1016/j.mechmachtheory.2017.01.006

Mallesh, G., Math, V. B., P, K. P. B., & K, P. K. M. (2009). Effect of Rim Thickness on Symmetric and Asymmetric Spur Gear Tooth Bending Stress, 113–118.

Neslàdek, M., & Španiel, M. (2017). An Abaqus plugin for fatigue predictions. *Advances in Engineering Software*, *103*, 1–11. http://doi.org/10.1016/j.advengsoft.2016.10.008

Pegg, E. C., & Gill, H. S. (2016). An open source software tool to assign the material properties of bone for ABAQUS finite element simulations. *Journal of Biomechanics*, *49*(13), 3116–3121. http://doi.org/10.1016/j.jbiomech.2016.07.037

Pippan, R., Motz, C., Kriszt, B., Zettl, B., Mayer, H., Stanzl-Tschegg, S., … Kovacik, J. (n.d.). Material Properties. *Handbook of Cellular Metals*, 179–241. http://doi.org/10.1002/3527600558.ch5

Qin, W. J., & Guan, C. Y. (2014). An investigation of contact stresses and crack initiation in spur gears based on finite element dynamics analysis. *International Journal of Mechanical Sciences*, *83*, 96–103. http://doi.org/10.1016/j.ijmecsci.2014.03.035

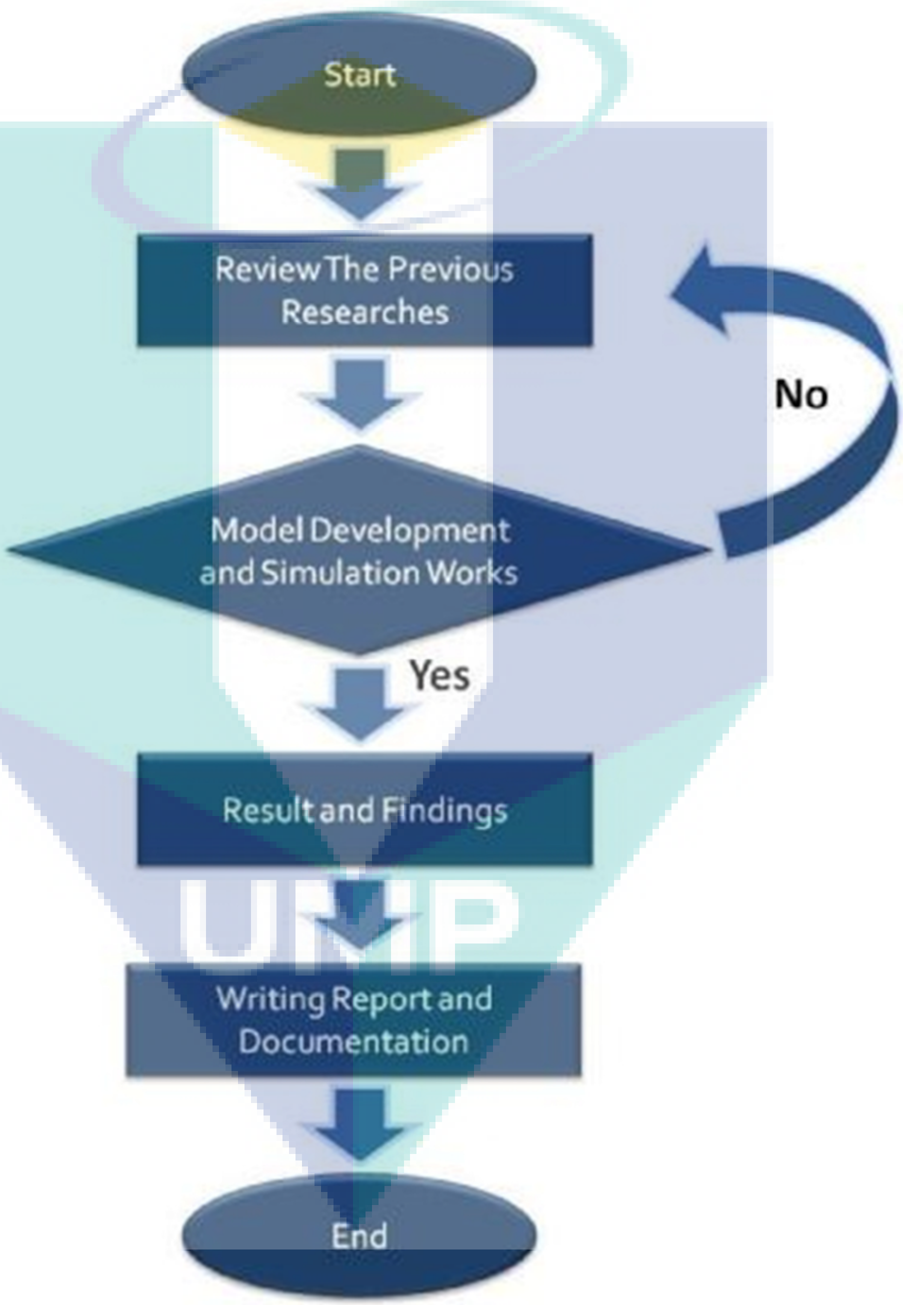
Richard G. Budynas, K. J. N. (2011). *Shigley's Mechanical Engineering Design*. United States: McGraw-Hill.

Savage, M., Savage, M., Rubadeux, K. L., Rubadeux, K. L., Coe, H. H., & Coe, H. H. (1995). Bending Strength Model for Internal Spur Gear Teeth. *31st Joint Propulsion Conference and Exhibit*.

Senthil Kumar, V., Muni, D. V., & Muthuveerappan, G. (2007). Optimization of asymmetric spur gear drives to improve the bending load capacity. *Mechanism and Machine Theory*, *43*(7), 829–858. http://doi.org/10.1016/j.mechmachtheory.2007.06.006

Yenarkar, Y. L. (2013). Investigation of Stresses in the Thin Rimmed Spur Tooth using FEM, 437–441.

APPENDIX A
RESEARCH FLOW CHART



APPENDIX B
LIST OF PUBLICATIONS

No	Title and author
1	Design and Analysis of Tooth Impact Test Rig for Spur Gear, Wafiuddin Bin Md Ghazali, Ismail Ali Bin Abdul Aziz, Daing Mohamad Nafiz Bin Daing Idris, Nurazima Binti Ismail and Azizul Helmi Bin Sofian, IOP Conf. Series: Materials Science and Engineering 114 (2016) 012058 doi:10.1088/1757-899X/114/1/012058
2	Investigating Bending Strength of Spur Gear: A Review, Ismail Ali Abdul Aziz, Daing Mohamad Nafiz Daing Idris, and Wafiuddin Mohd Ghazali, MATEC Web of Conferences 90, 01037 (2017) doi:10.1051/matecconf/20179001037