

Bending Stress Analysis for Spur Gear (2D Analysis)

by

NoorFurqan Binti Irfan

Dissertation submitted in partial fulfilment of

the requirements for the

Bachelor of Engineering (Hons)

(Mechanical Engineering)

SEPTEMEBR 2012

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Bending Stress Analysis for Spur Gear (2D Analysis)

Dissertation

by

NoorFurqan Binti Irfan

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfilment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(MECHANICAL ENGINEERING)

Approved by,

Prof.Dr. Nagarajan Thirumalaiswamy

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

SEPTEMBER 2012

CERTIFICATION OF ORIGINALITY

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

NOORFURQAN BINTI IRFAN

ABSTRACT

This report is to study the characteristics of involute gear system including bending stress and transmission errors of gears in mesh. Gearing is the most critical components in mechanical power transmission systems. Transmission error measurement has become a popular area of research on gears as it is related with a possible method for quality control. Transmission error is considered to be one of the main factors that contribute to noise and vibration in a gear set. Hence, to estimate the transmission error in a gear system, the characteristic of involute spur gears were analyzed by using the finite element method. Before proceed with simulation, the gears are drew by using CATIA P3 V5R12. Fives model of spur gear are design by into 3D and 2D models. The bending stress in the tooth root was analyzed by using 2-D FEM models. The simulation of 2-D FEM models was done by using ANSYS 14.0. In order to enable the investigation of contact problems with FEM, the stiffness relationship between the two contact areas is established by inserting a contact element placed in between two areas where contact occurred. Besides that, current analytical method of calculating gear bending stresses use Lewis equations, which were originally derived for bending stress of the contact between a pair of tooth. However for this project, we will vary the number of module of gear with fix pressure angle and number of teeth. Then, the result with different number of module will be related with the bending stress. The results of the two dimensional FEM analysis from ANSYS are presented. The bending stresses were compared with different number of gear module system. In the past, the bending stress sensitivity of a gear tooth has been calculated using photo elasticity or relatively coarse FEM meshes. However, with present computer developments we can make significant improvements for more accurate FEM simulations.

In this report the effect of module number of gear design is considered and how the contact stress results vary with these parameters is studied. The gear design is optimized based on FEM analysis.

ACKNOWLEDGEMENT

Firstly, I would like to thank to my supervisor, Professor Dr. Nagarajan Thirumalaiswamy for his guidance throughout this project. He always shows the high co-operation whenever I want to discuss matters regarding my project. He also supports me from the beginning until the end of the project by advising me the steps for completing this project.

My most sincere thanks also go to Mr. Santosh S. Patil for assisting me in getting the much needed information on gears and ANSYS software in order to complete this project. He never give up to teach me on how to deal with ANSYS software and always available to discuss if I got problem related with my project.

Thank you also to my family and friends who always support me in finishing my project successfully.

Lastly, to all that have helped me in completing this project directly or indirectly, my thanks go out to all of you.

TABLE OF CONTENTS

CERTIFICATION OF ORIGINALITY	i
ABSTRACT.....	ii
ACKNOWLEDGEMENT	iii
LIST OF FIGURES	v
LIST OF TABLES	vi
NOMENCLATURE	vi
CHAPTER 1	1
INTRODUCTION	1
1.1 Project Background.....	1
1.2 Problem Statement	2
1.3 Objective and Scope of Study	3
1.4 The Relevancy of the Project.....	3
1.5 Feasibility of the Project	4
CHAPTER 2	5
LITERATURE REVIEW	5
CHAPTER 3	9
METHODOLOGY	9
3.1 Overview.....	9
3.2 Gantt Chart.....	10
3.3 Spur Gear Modelling.....	12
3.4 Spur Gear Bending Stress Analysis	20
CHAPTER 4	24
RESULTS AND DISCUSSIONS.....	24
4.1 Theoretical Analysis	24
4.2 Bending Analysis Results	26
4.3 Result Analysis	28
CHAPTER 5	30
CONCLUSIONS AND RECOMMENDATIONS	30
5.1 Conclusions.....	30
5.2 Recommendations.....	31
REFERENCES	32
APPENDICES	34
APPENDIX A: MESHED PART OF SPUR GEAR	34
APPENDIX B: VON MISESS (BENDING) STRESS.....	37

LIST OF FIGURES

Figure 1.1: Tooth breakage	3
Figure 2.1: Point-to-surface contact element	7
Figure 3.1: Process flow chart	9
Figure 3.2: Gantt chart for FYP 1	10
Figure 3.3: Gantt chart for FYP 2	11
Figure 3.4: Generative Shape Design	13
Figure 3.5: Basic Parameters are entered	14
Figure 3.6: Involute Profile Gear Formulas	15
Figure 3.7: Add Laws to define the involute	15
Figure 3.8: Starting Point for Involute	16
Figure 3.9: The Involute Points	17
Figure 3.10: Create Base Circle	18
Figure 3.11: Draw the involute of Gear Profile	19
Figure 3.12: Circular Patterns to Draw the Gear Teeth	19
Figure 3.13: Gear Translation	21
Figure 3.14: Defining Contact	22
Figure 3.15: Mesh Edge Sizing	22
Figure 3.16: Types of Load Support	23
Figure 4.1: Length dimensions used in determining bending tooth stress	24
Figure 4.2: Maximum Bending Stress versus Module Number	28
Figure 4.3: Maximum Bending Stress versus Velocity Factor for Theoretical	29
Figure 4.4: Maximum Bending Stress versus Velocity Factor for Simulation	29
Figure A.1: Mesh Part for module 1mm	34
Figure A.2: Mesh Part for module 2mm	34
Figure A.3: Mesh Part for module 3mm	35
Figure A.4: Mesh Part for module 4mm	35
Figure A.5: Mesh Part for module 5mm	36
Figure B.1: Von Misses (Bending) Stress for Module 1mm	37
Figure B.2: Von Misses (Bending) Stress for Module 2mm	37
Figure B.3: Von Misses (Bending) Stress for Module 3mm	38
Figure B.4: Von Misses (Bending) Stress for Module 4mm	38
Figure B.5: Von Misses (Bending) Stress for Module 5mm	39

LIST OF TABLES

Table 3.1 Spur Gear Parameters	12
Table 3.2: Data for Analysis	20
Table 4.1: Comparison of Maximum Bending Stress for Theoretical and Simulation	26
Table 4.2 Comparison of Velocity Factor for Theoretical and Simulation	27

NOMENCLATURE

d_o	Outer Diameter
b	Face Width
d	Pitch Diameter
d_b	Base Diameter
h_a	Addendum
h_d	Deddendum
h	Tooth Depth
d_f	Root Diameter
s	Tooth Thickness
ϕ	Angular of Thickness
D	Centre of Distance
m	Module Number
z	Number of Teeth
α	Pressure Angle
W_t	Transmitted Load
I	Area of Moment Inertia
K_v	Velocity Factor
V	Pitch Line Speed
Y	Lewis Factor
n	Gear Speed
σ	Maximum Bending Stress

CHAPTER 1

INTRODUCTION

1.1 Project Background

In manufacturing, power transmission is the most important. This is due to the efficiency of every machine depends on the amount of power loss during the process. Hence, the inventions of gears are the best methods of transmitting power between the shafts. Usually, gears are used to transmit torque and angular velocity.

Gearing is one of the most critical components in a mechanical power transmission system and in most industrial rotating machinery. The fast shift in the industry from heavy industries such as automobile manufacture and office automation tools will necessitate a refined application of gear technology. In addition, it is possible that gears will predominate as the most effective means of transmitting power in future machines due to their high degree of reliability and compactness.

The increasing demand for quiet power transmission in machines, vehicles, elevators and generators has create a growing demand for a more precise analysis of the characteristics of gear system. In the automobile industry, the largest manufacturer of gears, higher reliability and lighter weight gears are needed as lighter automobiles continue to be in demand.

Spur gears are the simplest type of gear. They consist of a cylinder or disk with the teeth projecting radially, the edge of each tooth is straight without angle and aligned parallel to the axis of rotation. In spur gears the design parameters play a major role in determination of stresses. The American Gear Manufacturing Association set the AGMA standard for spur gear design. The simplest motion of two external spur gears can be seen by an example of two external rotating cylinders, if sufficient friction is present at the rolling interface. The main disadvantage of these rotating is the possibility of slip at interface which is avoided by adding meshing teeth to rolling cylinders.

Designing highly loaded spur gears for power transmission system that are both strong and quiet requires analysis methods that can easily implemented and also provide information on contact and bending stresses.

In this report, the parameters of different module number play a key role in gear design are studied. This parameter is varied and their effects on the final stress are observed at the tooth and mating regions of the gear.

A gear was mating with similar kind of gear and then FEM Model was built in CATIA P3 V5R12. There are five models of spur gears with different module number were designed by using this software.

The theoretical method of estimating the bending stresses in a gear tooth is by calculating the bending stress using the Lewis equation. Furthermore, the default formula that has been provided by ANSYS software is Lewis Equation. It models a gear tooth taking the full load at its tip as a simple cantilever beam. The excessive of bending stress can cause tooth breakage to the gear.

1.2 Problem Statement

There are many types of gear failure but generally they can be classified into two general groups. One is failure of the root of the teeth because of bending strength is inadequate and the other type is created on the surfaces of the gears. This failure is related with the module number of gear. The failure starts as a crack which is usually at the root of the tooth and proceeds across the base of the tooth until the complete tooth breaks away from the gear. This is because the weakest point of the gear is at the root of the tooth or at the fillet where high stress concentration exists together with highest tensile stress from bending or from the surface defects. The continuing bending failure in gears can cause gear tooth breakage as shown in Figure 1.1.

Therefore, to prevent tooth breakage, the tooth bending stress is calculated by using analytical method and Finite Element Method (FEM). This stress should be calculated first before we proceed with designing the real gear set.

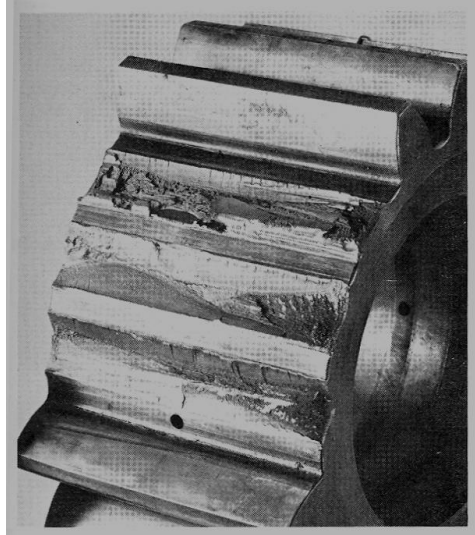


Figure 1.1 Tooth breakage

1.3 Objective and Scope of Study

The objectives of this project are to use a numerical method to develop theoretical models of the behaviour of spur gears in mesh and to help to predict the effect of gear tooth stress. The main focuses of this project are:

- To identify the relationship between module number of gear with the maximum bending stress.
- To perform 2D FEM analysis by using ANSYS software and compare with the theoretical result (Lewis Equation).
- To study the relationship of spur gear velocity factor and maximum bending stress.

1.4 The Relevancy of the Project

In studying cases of gear wear and failure, it is important that the correct analysis be made. Frequently the cause of failure will be something quite different from the amount of transmitted load. An incorrect analysis can lead a designer to make a new gear set larger than it ought to be, and yet the new set may still fail because the real cause of trouble is still uncorrected. In a general sense, failure should not be thought of as just inability to operate. But, failure should be thought of as some unsatisfactory condition which either threatens that a geared system will become inoperable, or poses an environmental disturbance that is considered improper.

By using different number of module, we can vary the involute profile of gear. A true involute design provides no clearance at the first point of contact. If a

pair of teeth was spaced too close together, there is a bump as the teeth come into mesh. With some modification on the involute profile gear like increasing the number of module, there is a little relief at the first point of contact. This make the teeth come into mesh smoothly even if an occasional pair of teeth are too close together.

1.5 Feasibility of the Project

This project no needs to take a long time to finish. This is because nowadays computers are becoming more and more powerful, and that is the reason why people tend to use numerical approach to develop theoretical models to predict the effects. Numerical methods can potentially provide more accurate solution since they require much less restrictive assumption. However, in order to get the accurate results and reasonable computational time, the correct model and the solution method should be chosen.

In this project, first step is the solid model of the spur gear is made with relations and equations modelling option in CATIA P3 V5R12. After the modelling of spur gear, the file is imported into ANSYS 14.0 software for bending stress analysis. The results of ANSYS 14.0 are then compared with the calculation result by using Lewis Equation. There are some assumptions should be made for Lewis Equation. The assumptions are the load is static and does not take the dynamics of meshing teeth into account. The stress formula must be modified to account different situations like stress concentration and geometry of the tooth. Hence, the Lewis Equation is modified to account for variety of conditions that can be encountered in service.

CHAPTER 2

LITERATURE REVIEW

Darle W.Dudley [1] write in his book that the best design of gear is practical and economical to manufacture and it it well enough thought out to meet all the hazards of service in the field. His main aim is to show a practical gear design must be based on the limitations and availability to machine tools and tooling setup. This mean the design should follow the right standard of gear design. Spur gears are simple in design and in machinery used to manufacture or maintenance. Most of gear designers prefer to use them whenever design requirements permit. He also mentioned that spur gears also are slow-speed gears.

Cockerham presents a computer design that can be used for 20-degrees pressure angle which ignores gear-tooth-tip scoring. The uniqueness of this program are we can vary the parameters of the gear like module number, face width and gear ratio to obtain an acceptable design [2]. Tucker [3] and Estrin [4] consider the gear mesh parameters such as addendum ratios and pressure angles and outline the procedures for varying a standard gear mesh to obtain a more favourable gear set. The basic approach is important to ensure the given data is acceptable [5]. According to American Gear Manufacturing Association (AGMA), the gear strengths must be considered including fatigue [6].

In 1980, Savage and Coy said that the design of standard gear mesh should minimize the gear size for a given ratio, pinion torque, and allowable tooth strength. They also optimized the gear tooth numbers for compact standard spur gear set. They define design space by considering the terms of the number of teeth on the pinion and the module number. This space is then combined with the objective function of minimum centre distance to obtain an optimal design region. This region defines the number of pinion teeth for the most compact design [7].

Xu Rixin [8], there is the development of analytical approach and modelling procedure to evaluate stress distributions and quenching process under applied velocity and moment. Finite element stimulation of spur gear was developed and used to predict distribution of stress and other material properties. The quenching result in the stimulation proved the theory and ensured product quality.

In another finding by Patchigolla [9] a finite element modelling approach was developed to determine the effect of gear rim thickness on tooth bending stresses in large spur gears. Wei defined stresses and deformations in involute spur gear by finite element method. He examined Lewis bending stresses using 2-D and 3-D models. He also considered the variations of the whole gear body stiffness arising from the gear body rotation due to bending deflection, shearing displacement and contact deformation [10].

Saxena [11] analyzed the stress distribution in spur gear teeth using FEM program. He studied a single teeth rather than whole spur gear because due to time limitation. Spur gear profile was created in ANSYS. In his study also he analyzed geometry factor for different type of gear profile. In order to get a gear set with good quality, Richard G.Budynas and J.Keith Nisbett said in their book that the contact ratio should be somewhat greater than unity. If in fact, the gears are cut with sufficient accuracy, the tip-load condition is not the worst, because another pair of teeth will be in contact when this condition occurs. By examining the run-in teeth, it will show that the heaviest loads occur near the middle of the tooth. Therefore, the maximum stress probably occurs while a single pair of teeth is carrying the full load, at a point where another pair of teeth is just on the verge of coming into contact [12].

A recent study by Kahraman [13] considers each gear as a deformable body and meshes them to predict load, stress and deformations. Based on the result presented, a deformable body analysis with a thin rim is necessary. Deng used tooth contact analysis, loaded tooth contact analysis and finite element method to analyze the meshing behaviour, tooth surface contact stress, maximum tensile, bending stress and maximum compressive bending stress. The modified pitch cone method is first presented and verified in the gear research centre of Dong Feng vehicle-bridge Co. Ltd. [14]

As for Rufang Li, he developed 3D static and dynamic contact or impact analysis of gear drives. The tooth load allocation and result are derived under the static load. His paper contain the stress distribution of gear system under dynamic loading conditions and stimulates the stress of gears under conditions of initial speed and a sudden load being applied [15].

Mao developed an advanced non-linear finite element method, which has been successfully used to simulate gear contact behaviour under real load condition accurately. For a gear set, the stresses were firstly computed by the 2D FEM and the formulae were drawn allowing a simple calculation of maximum tooth root stresses [16].

In general, there are three basic types of contact modelling application as far as ANSYS use is concerned [17]:

1) Point-to-point contact

The exact location of contact should be known beforehand. These types of contact problem usually only allow small amounts of relative sliding deformation between contact surfaces.

2) Point-to-surface

The exact location of the contacting area may not be known beforehand. These types of contact problems allow large amounts of deformation and relative sliding. Furthermore, opposing meshes do not need to have the same discretisation or a compatible mesh.

3) Surface-to-surface

This type of contact is typically used to model surface-to-surface contact applications of the rigid-to-flexible classification.

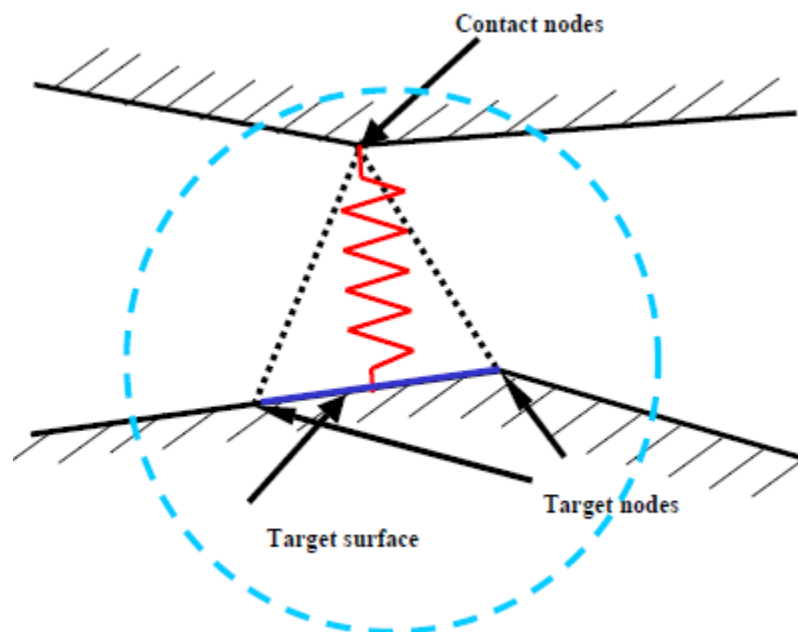


Figure 2.1 Point-to-surface contact element

In the thesis final version by Zeping Wei, due to the simplicity of their formulation, the advantages of using contact elements are easy to use, simple to formulate and easily accommodated into existing FE code. However, using contact elements poses some difficulties such as the fact that their performance, in term of convergence and accuracy, depends on user defined parameters. All FEA involves a trade-off between expense and accuracy. More detail and finer mesh usually leads to a more accurate solution, but requires more time and system resources. Nonlinear analyses add an extra factor, the number of load increments, which affect both accuracy and expense. Other nonlinear parameters, such as contact stiffness can also affect both accuracy and expense [18].

Ognyan Alipiev stated that when gears of different teeth number mesh, the radius of the lantern circle of the small gear is larger than the one of the large gear. Besides, by increasing the difference between numbers of teeth the proposed meshing cannot exist because the teeth of the larger gear are sharpened and a part of its involute profile is cut. In these cases it is appropriate to use modified involute-lantern meshing that will be considered in the next publications by the authors [19].

In 1999, Kelenz [20] investigated a spur gear set using FEM. The contact stresses were tested using two dimensional FEM model. The bending stress analysis was performed on different thin rimmed gears. The contact stress and bending stress comparisons were given in his studies.

CHAPTER 3

METHODOLOGY

3.1 Overview

This project is done step by step. It is important part to ensure the correct steps are taken in order to avoid procrastination. Figure 3.1 illustrates the flow chart diagram for this project.

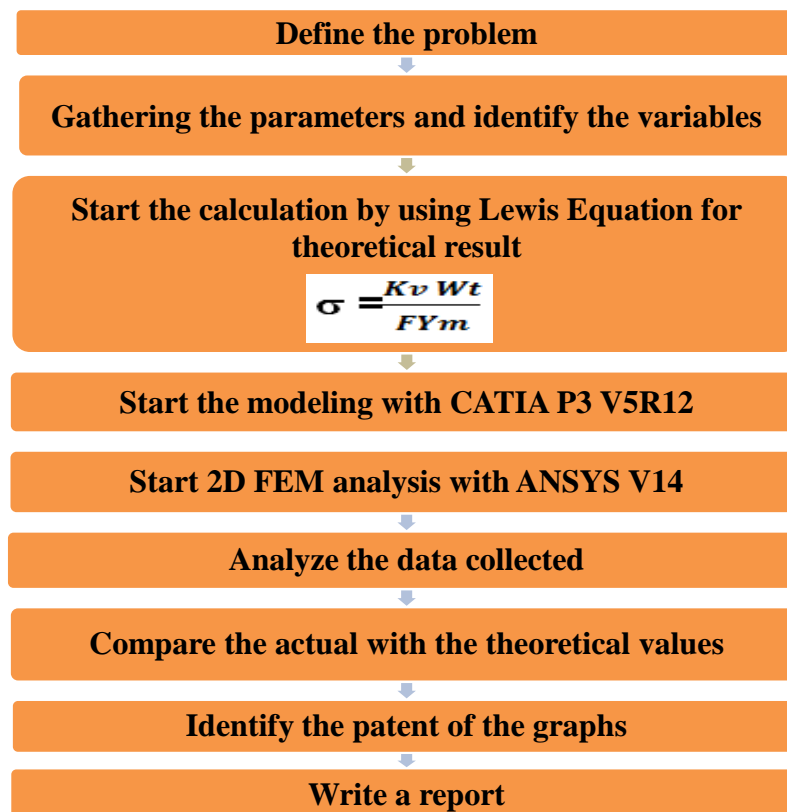




Figure 3.1 Process Flow Chart

3.2 Gantt Chart

The project is divided into two semesters. For the first semester, the author begins by attending briefing on Final Year Project and selecting title. After the title has been approved, he gathered information pertaining to the project, while selecting the appropriate method and tools or software to be used. In the second half of the first semester, the author performed data analysis and started on software programming. The author is deemed to submit extended proposal in Week 6 as well as interim report in Week 14. Figure 3.2 shows the Gantt chart of the project for the first semester.

No.	Activities / Week	1	2	3	4	5	6	7		8	9	10	11	12	13	14	
1	Project topic selection.	Process	Process						Mid-semester break								
2	Preliminary research work.		Process	Process	Process	Process											
3	Extended Proposal Defence Submission.						Key milestone										
4	Proposal Defence.										Process	Process					
5	Detailed Design											Process	Process				
6	Material Procurement.													Process			
7	Interim draft report submission.															Key milestone	
8	Interim report submission.																Key milestone



Figure 3.2 Gantt chart for FYP 1

 - Key milestone  - Process

In second semester of the project, the author has completed her spur gear modelling and bending stress analysis by end of Week 14. Based on Figure 3.3, the author is required to submit progress report in Week 8, technical paper in Week 13, and dissertation in Week 15. Besides that, he is also required to deliver a presentation in Week 11 and also in Week 14.

No.	Activities / Week	1	2	3	4	5	6	7		8	9	10	11	12	13	14	15	
1	Simulation of spur gear	Process	Process	Process					Mid-semester break									
2	Analysis of stimulation result				Process	Process	Process	Process										
3	Progress report submission									Key milestone								
4	Provide discussion and conclusion									Process	Process							
5	Prepare final report											Process	Process	Process				
6	Pre - EDX													Key milestone				
7	Draft report submission														Key milestone			
8	Dissertation submission															Key milestone		
9	Technical paper submission																Key milestone	
10	Oral Presentation																	Key milestone
11	Project Dissertation submission																	Key milestone

Figure 3.3: Gantt chart for FYP 2

 - Key milestone  - Process

3.3 Spur Gear Modelling

Before the spur gear can be analyzed by ANSYS 14.0 software, the spur gear should be modelled first by using CATIA software. The important data are gathered and some of the value is calculated by using gear formula. The formulas using followed the module system in metric system. Some values are fixed such as number of teeth and pressure angle. The values that varied are the module number with 1mm, 2mm, 3mm, 4mm and 5mm. The Table 3.1 shows the gear parameters.

Number of teeth, $z = 20$ Pressure angle, $\alpha = 20^\circ$

Parameters	Formula	Number of Module, m (mm)				
		1	2	3	4	5
Outer Diameter, d_o	$(z+2)m$	22	44	66	88	110
Pitch Diameter, d	zm	20	40	60	80	100
Base Diameter, d_b	$D \cos \alpha$	18.7939	37.5877	56.3816	75.1754	93.9693
Addendum, h_a	$1.00m$	1	2	3	4	5
Deddendum, h_d	$h-h_a$	1.25	2.5	3.75	5	6.25
Tooth Depth, h	$2.25m$	2.25	4.5	6.75	9	11.25
Root Diameter, d_f	$d-2.5m$	17.5	35	52.5	70	87.5
Tooth thickness, s	$\Pi m/2$	1.5708	3.1416	4.7124	6.2832	7.8540
Angular of thickness, ϕ	$360^\circ / z$	18°	18°	18°	18°	18°
Centre of distance, D	$(d_1+d_2) / 2$	20	40	60	80	100

Table 3.1 Spur Gear Parameters

The steps of drawing spur gear by using CATIA software is described as per below:

- 1) Start CATIA, go to **TOOLS->OPTIONS->Infrastructure->Part Infrastructure** and in **Display** select **Parameters and Relations**
- 2) Then in **Options->General** in **Parameters and Measures** select **With value**
- 3) Go to **Generative Shape Design**

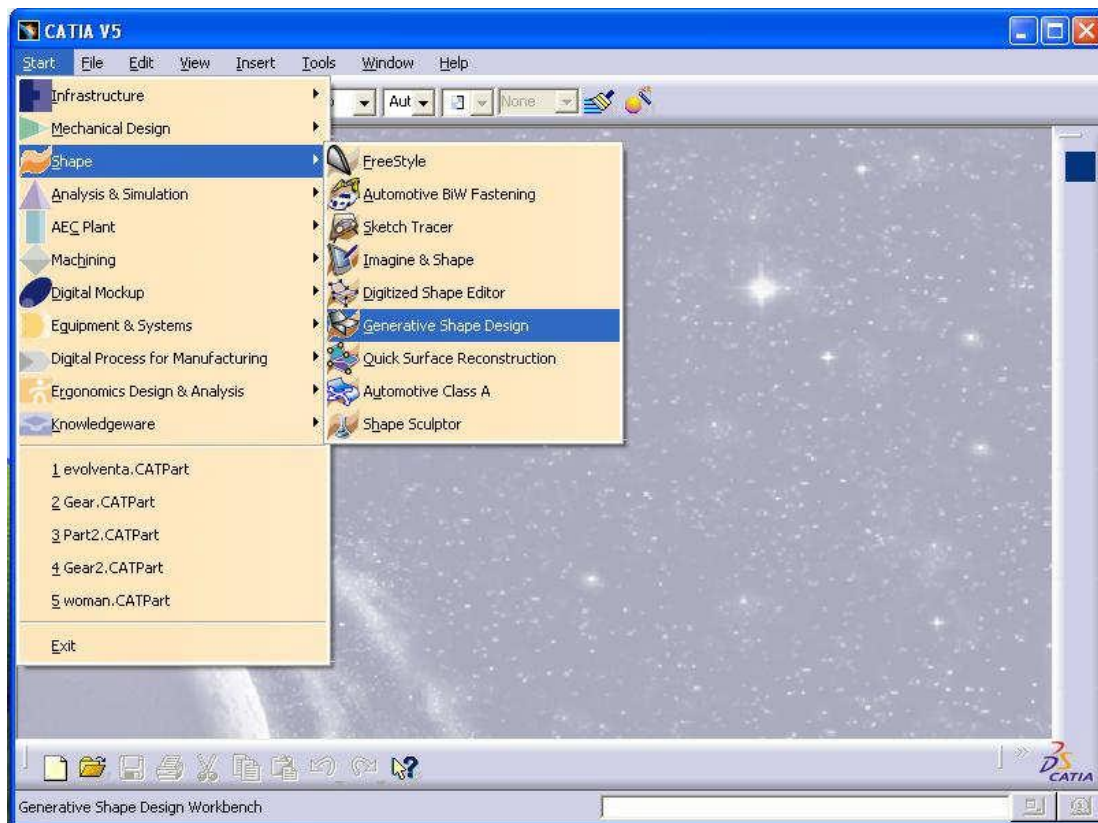


Figure 3.4 Generative Shape Design

- 4) Then by clicking on arrow pointed down near table icon: **Fog and f(x)** are two most important things you will use for gear design.
- 5) Now it is time to enter some basic parameters that define gear. This is done by clicking at f(x) icon.

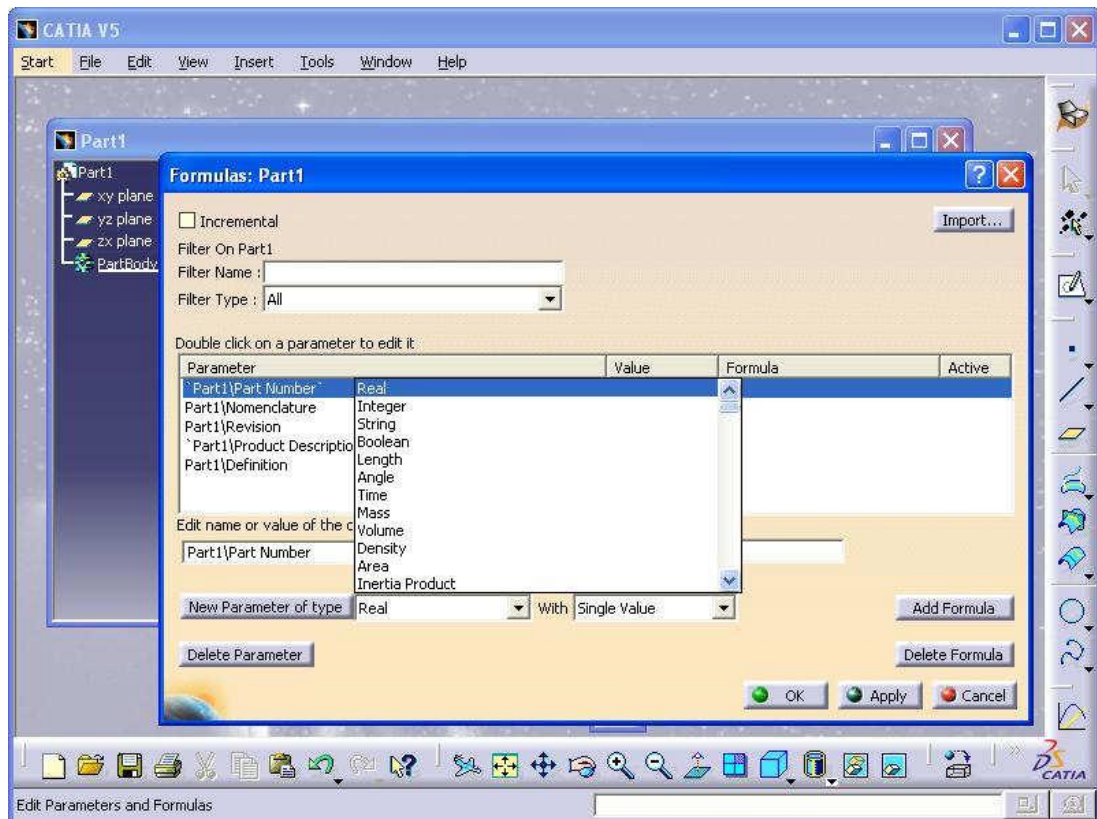


Figure 3.5 Basic Parameters are entered

- 6) And then when see dialog box: Formulas: Part1 first select Parameter type (real, length or angle) click new Parameter of type and then edit value. Do this step until all parameters are entered.
- 7) After entering the parameters it is time to enter some formulas. Enter the formulas by naming them and by clicking Add Formula. Formula editor will appear.

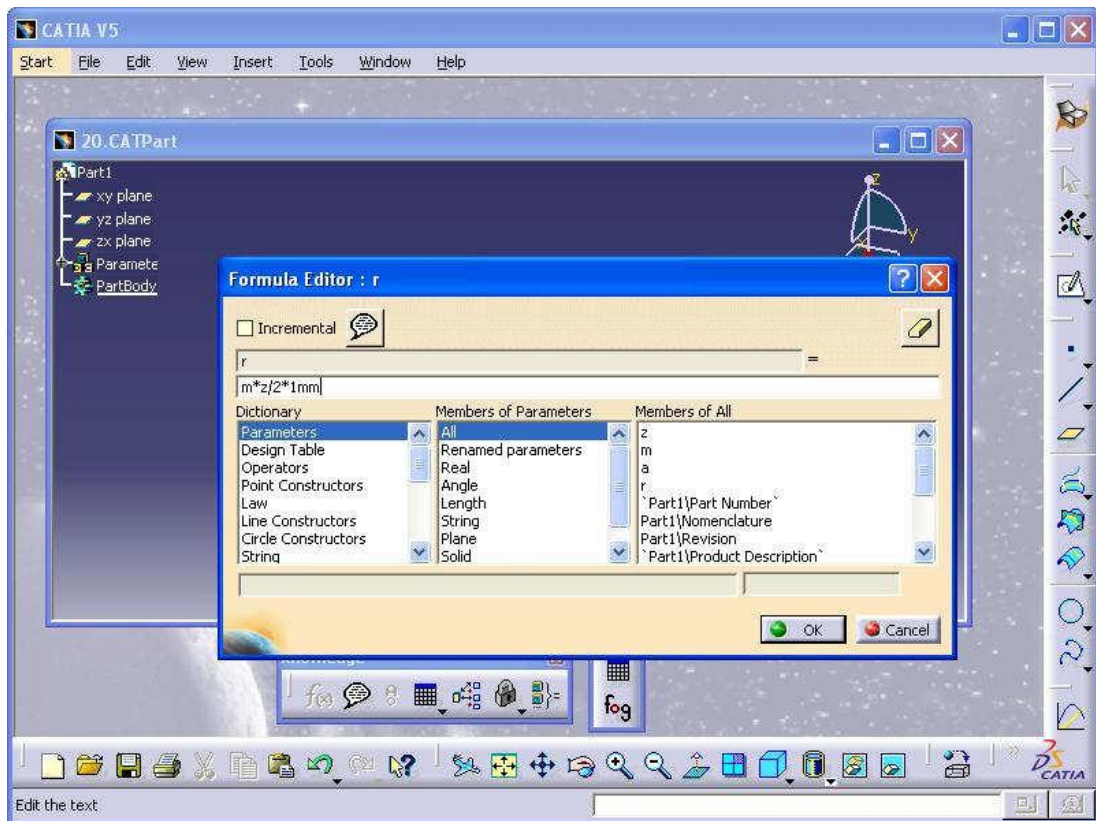


Figure 3.6 Involute Profile Gear Formulas

- 8) It is time to add laws that will define the involute. Click on **fog icon**, name law x add parameters, t and x select their types and add law.

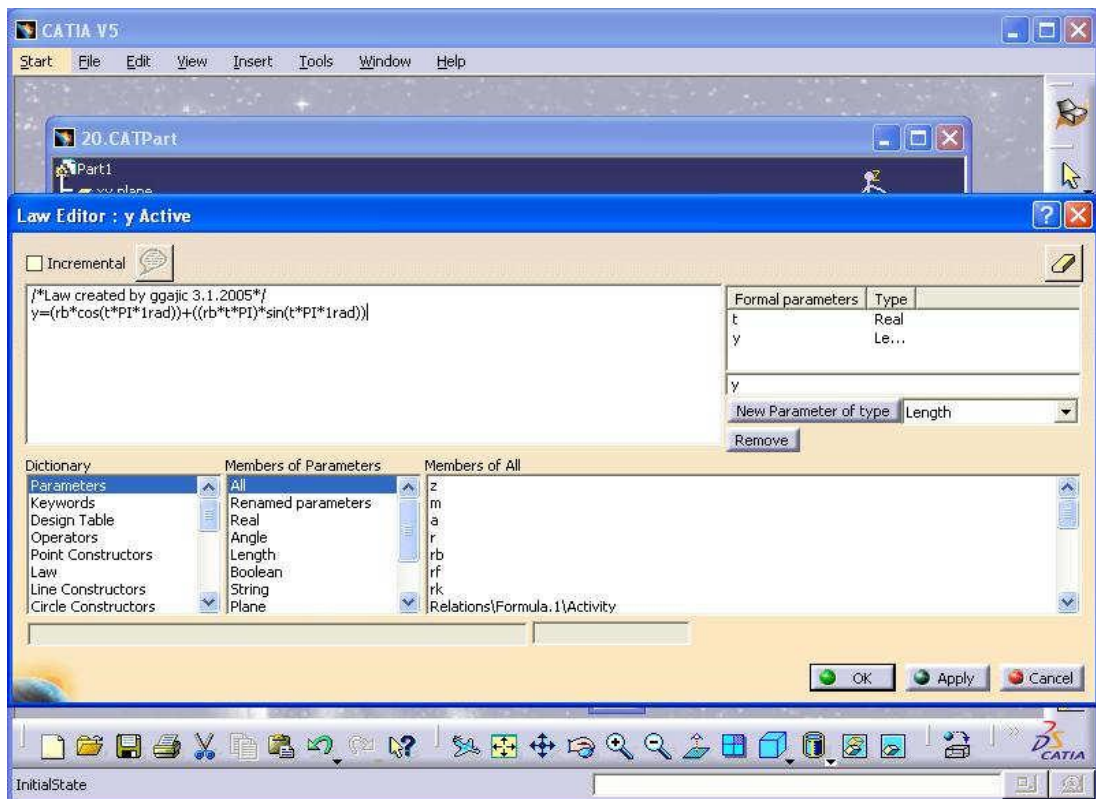


Figure 3.7 Add Laws to define the involute

- 9) Same should be done for y. This law will help us to create points that define spline for the involute. Involute are line that is trajectory of point belonging to line that is always tangent to base gear cylinder. It is used for tooth profile. If gears had profiles formed by straight lines they wouldn't work.
- 10) Start creating the points for involute spline. Click on point icon, select xy plane for support and when asked to enter H and V coordinates right button click should bring menu where you should chose edit formula.
- 11) When edit formula dialog prompted, type in: Relations\x. Evaluate (0).
- 12) Repeat the same step for V except if it should use Relations\y Evaluate (0).
Then get the starting point for involute.

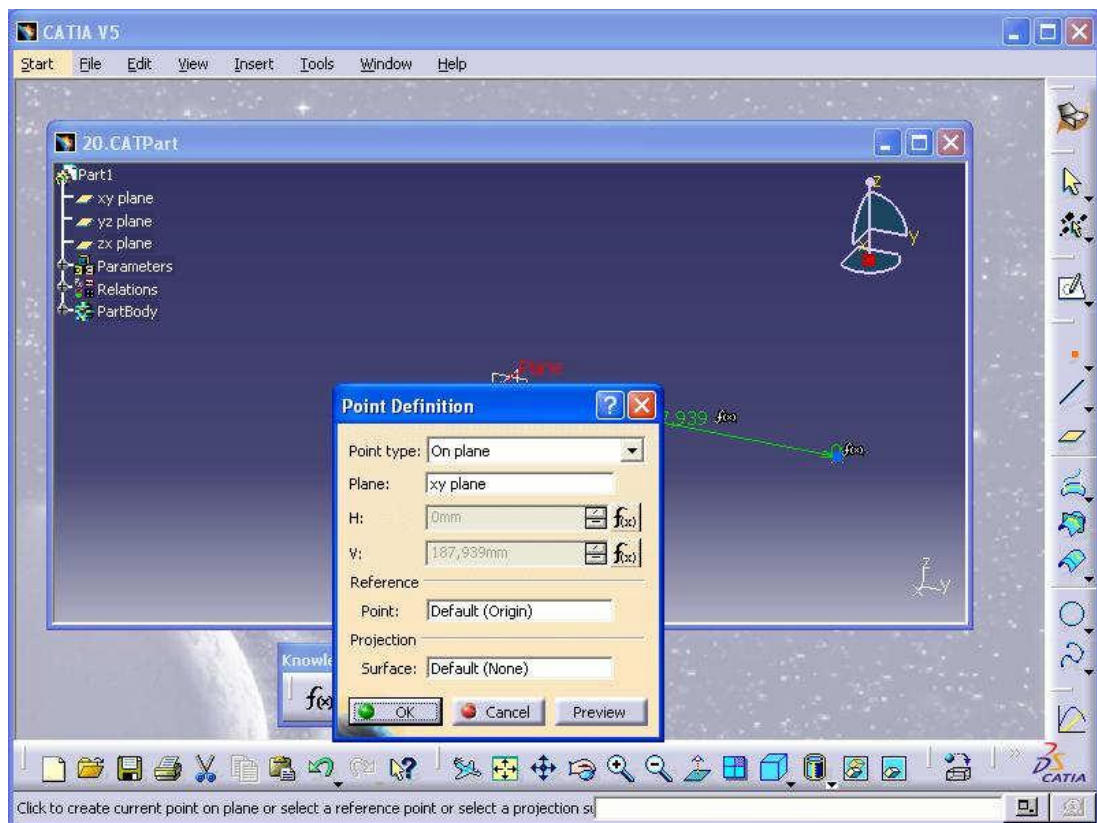


Figure 3.8 Starting Point for Involute

13) After repeating this step for .Evaluate(0.1)-.Evaluate(0.5), the result will be as per below:

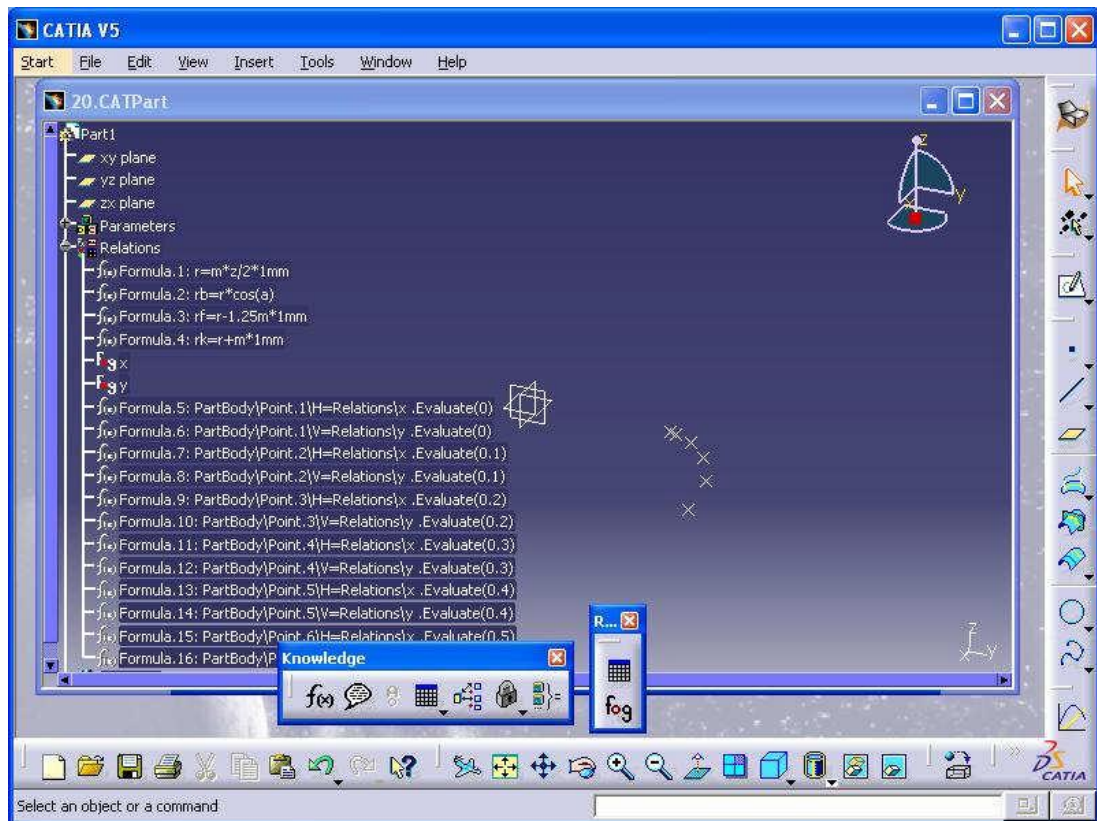


Figure 3.9 the Involute Points

- 14) Click on spline icon and chose all 6 points:
- 15) Create base circle by clicking on circle icon. Right click on radius and chose rb from formula editor.

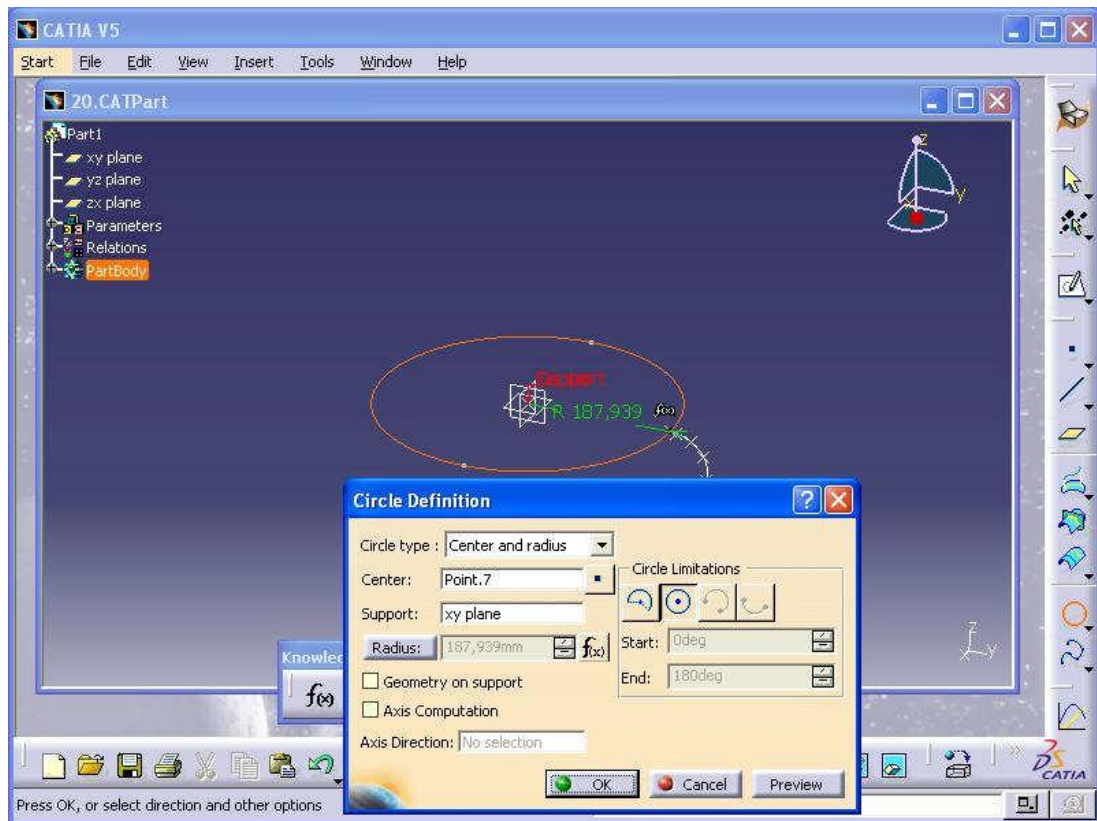


Figure 3.10 Create Base Circle

- 16) Now it is time to extrapolate our involute. For length also right click and chose formula $(rb-rf)*1.5$
- 17) Create plane. Use formula: $\text{inv}(360/z)/4$. You will get -4.5deg angle offset from ZY plane.
- 18) After this create rf circle and from insert menu use corner to create corner. Corner dialog will appear.
- 19) It is time for symmetry. Tooth starts to get shape. Create rk circle and use trims to get tooth shape.

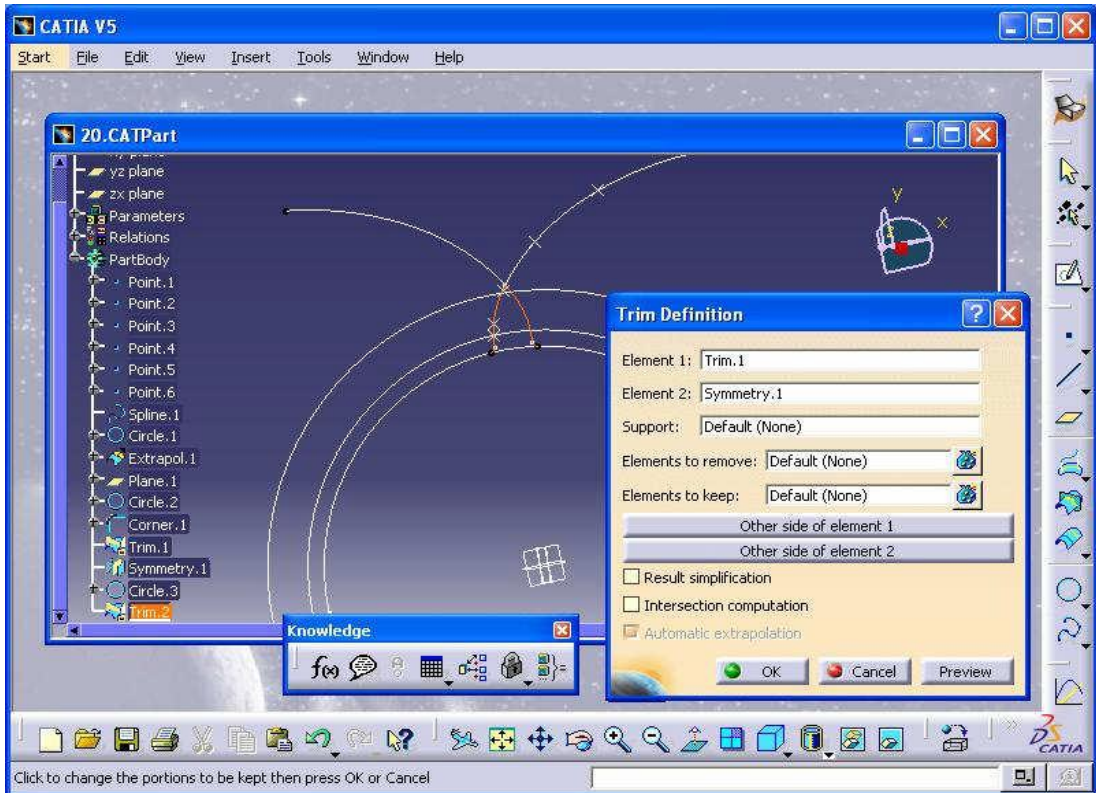


Figure 3.11 Draw the involute of Gear Profile

20) Create a circular pattern.

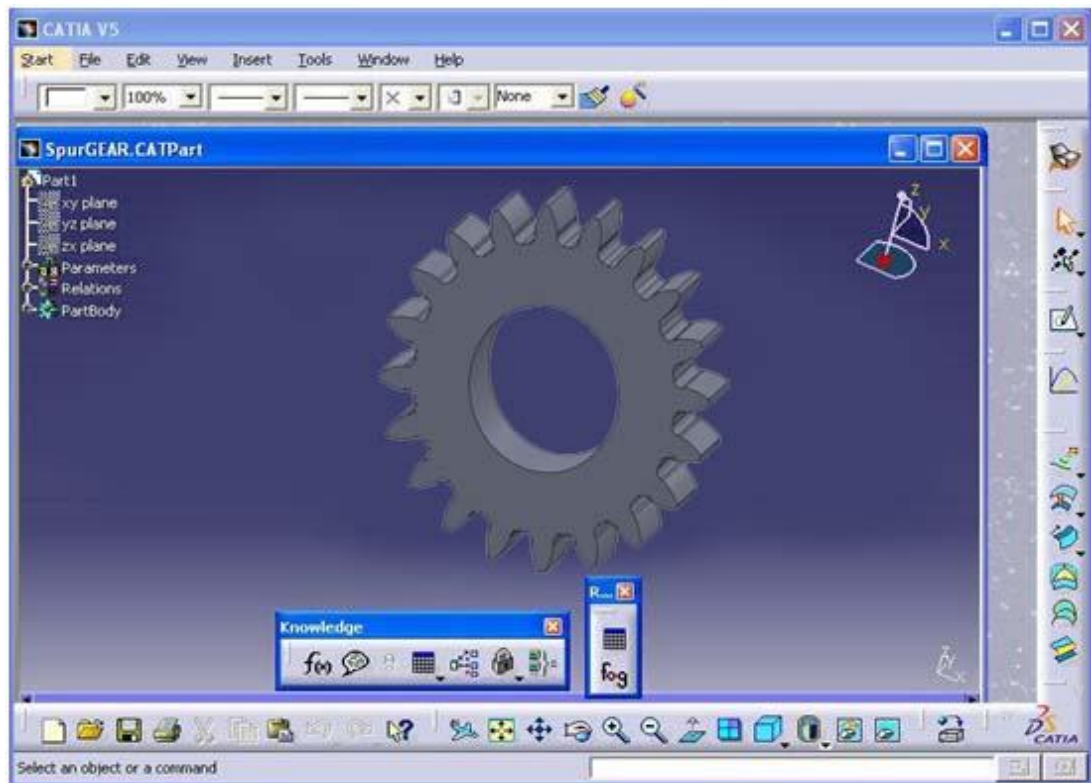


Figure 3.12 Circular Patterns to Draw the Gear Teeth

21) Extrude the drawing for 3D model and save as surface planar for 2D model.

3.4 Spur Gear Bending Stress Analysis

After that, the sketch is performed with surface operation. Since this is for 2-D model, we just save the file as surface planar and no need to be extruded. Then, the designs in CATIA software were saved in file type .IGS so that it can be imported into ANSYS 14.0.

Then the structural analysis was performed by using ANSYS 14.0. The body parts were imported into ANSYS and translate it with gap of centre distance. The Table 3.2 shows the data that should be prepared before analysis are as below:

Modulus of Elasticity	200 GPa
Poisson's Ratio	0.3
Type of Gear	Standard Involute, Full depth
Transmitted Load (W)	3000 N
Revolution per minute (rpm)	3000
Moment of inertia	5000 Nm

Table 3.2 Data for Analysis

After the assembly is imported in ANSYS Workbench 14, assembly is subjected to the boundary conditions. The mating of gear is using the same module number of gear. Translation is done while one of the gear body is preserved and the other one is translated by centre distance. For this project, it is assumed that the one gear is fixed support and the other gears are given torque along its axis. Adjust the pair of gear until three pair of gear teeth are touching to each other. The Y-coordinate is the same value with centre distance. The X-coordinate is needed to adjust too in order to get the contact body. The picture below illustrates the translation of the gear bodies.

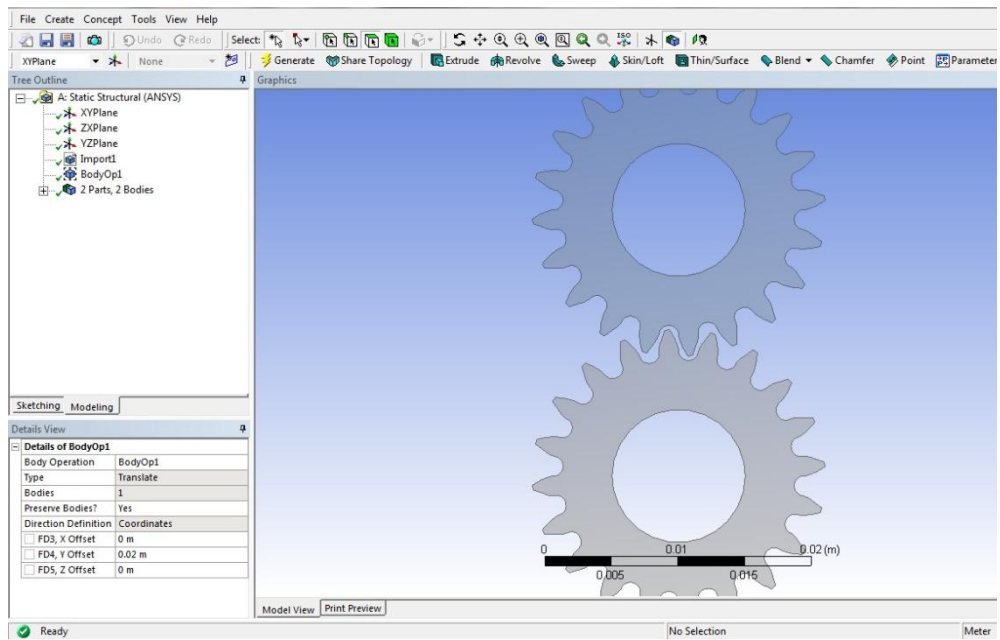


Figure 3.13 Gear Translation

ANSYS has many types of analysis, so it is important to select the correct type of analysis from the option menu bar. As the imported geometry is 2-Dimensional, select 2D and Static Structural Analysis and connect the geometry to analysis tab. This can be done by selecting the Engineering Data from the analysis tab and inserting the corresponding values. The default values are used for the other information needed.

Next thing to do after the geometry is attached with Static Structural Analysis tab is to define the contact between the two involute teeth. The special of ANSYS software is it has a built option, which automatically reads the attached geometry for any predefined contacts or other boundary definitions. The contacts between the pairs of teeth are assumed to be frictionless. Then change the 'Interface Treatment' to 'Adjust to touch' for the type of contact between the selected bodies. The Figure 3.14 shows the image from ANSYS showing the contact defined for the two spur gear teeth in mesh.

Details of "Frictionless - Lower Body To Upper ..."	
Scope	
Scoping Method	Geometry Selection
Contact	5 Edges
Target	5 Edges
Contact Bodies	Lower Body
Target Bodies	Upper Body
Definition	
Type	Frictionless
Scope Mode	Manual
Behavior	Symmetric
Suppressed	No
Advanced	
Formulation	Augmented Lagrange
Interface Treatment	Adjust to Touch
Normal Stiffness	Program Controlled
Update Stiffness	Never
Pinball Region	Program Controlled
Time Step Controls	None

Figure 3.14 Defining Contact

If mesh is performed with the default settings, it is not adequate to get the accurate results. For this analysis both the gear were finely meshed with ‘Sizing’ option in menu. The element size was compute to be 0.000254 m and it was then refined at the bending surfaces to get the finer mesh and continuous stress values. The Figure 3.15 shows the mesh part.

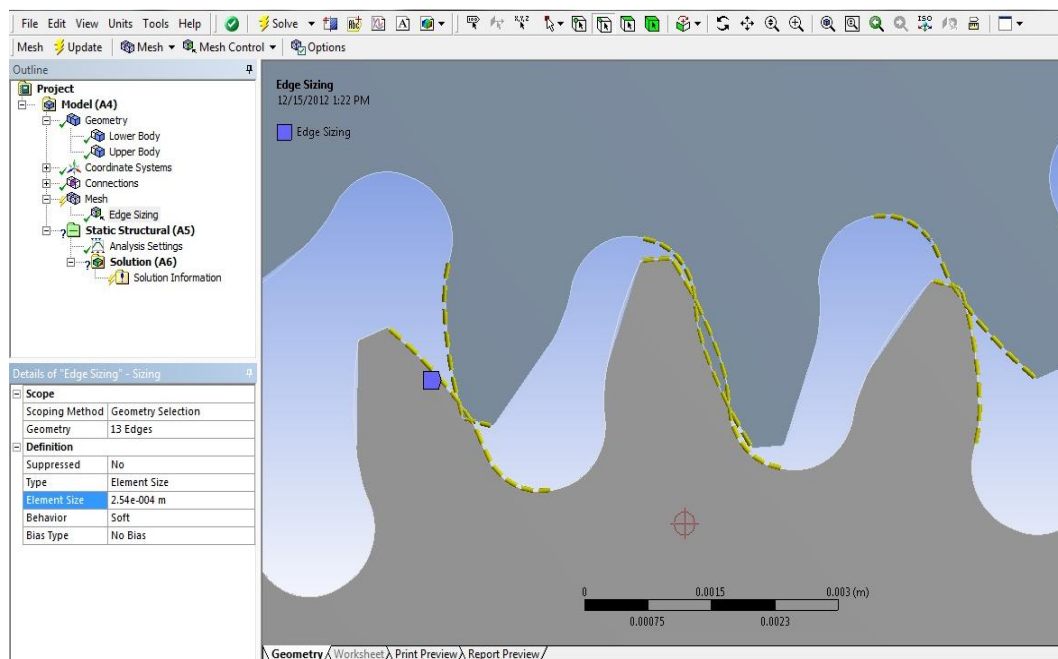


Figure 3.15 Mesh Edge Sizing

For supports and loads, the lower gear is defined as a fixed support and the upper gear is the frictionless support. The upper gear is also given a torque or moment in clockwise direction (-5000 Nm). The image below shows how the supports and loads were applied to the gear model.

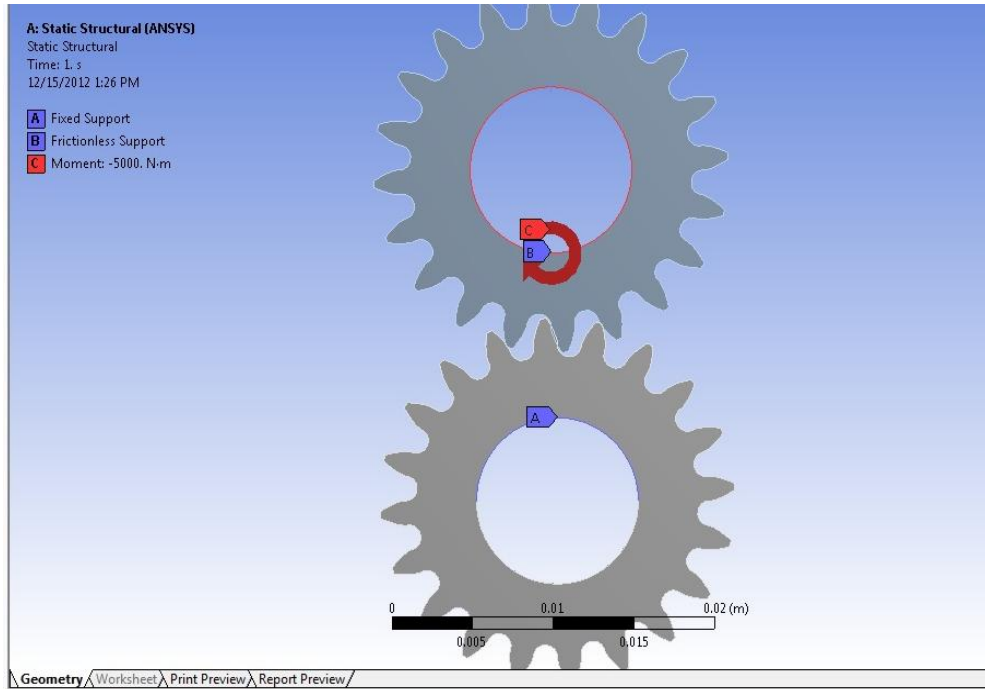


Figure 3.16 Types of Load Support

The last step will be defined the stress. For this project, the type of solution used is equivalent stress (Von Misses Stress). Then, solve all of the solvers for the contact part.

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 Theoretical Analysis

For this project, the bending failure in gears is predicted by comparing the calculated bending stress to simulation that is determined by ANSYS software. The bending stress equation was derived from the Lewis Formula. Wilfred Lewis [] was the first person who derive the formula for bending stress in gear teeth using the bending of a cantilevered beam to simulate stresses acting on a gear tooth shown in Figure 4.1 are length = l, load = F, cross section = b*t, uniform across the face. For a rectangular section, the area moment of inertia is $I = \frac{bh^3}{12}$

section, the area moment of inertia is $I = \frac{bh^3}{12}$

$M = Fl$ and $c = t/2$, stress then is

$$\sigma = \frac{M}{I} = \frac{F_t l (\frac{t}{2})}{\frac{bt^3}{12}} = \frac{6F_t l}{bt^2}$$

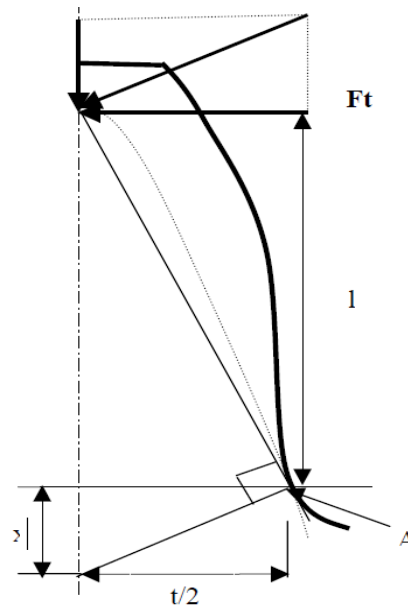


Figure 4.1 Length dimensions used in determining bending tooth stress

Where b = the face width of the gear. For a gear tooth, the maximum stress is expected at point A, which is a tangential point where the parabola curve is tangent to the curve of the tooth root fillet called parabola tangential method. Two points can be identified at each side of the tooth root fillet. The stress on the area connecting those two points is thought to be the worst case. The crack will likely to start from the point A.

The Lewis Equation assumes only static loading and does not take the dynamics of meshing teeth into account. The Lewis form factor is given for various numbers of teeth while assuming a pressure angle of 20° and a full – depth involute.

The Lewis form factor is dimensionless and also independent of tooth size but only a function of the shape. The above stress formula must be modified to account for the velocity factor K_v .

Hence, the modified Lewis Equation is

$$\sigma = \frac{K_v W_t}{bmY}$$

Where,

K_v = Velocity Factor

W_t = Transmitted Load

b = Face Width

m = Module Number

Y = Lewis Factor

For this project, the equation K_v is assumed as hobbed or shaped profile which is;

$$K_v = \frac{3.56 + \sqrt{v}}{3.56}$$

As the value of pitch line velocity, K_v is calculated using formula

$$V = \pi d_{\text{pitch}} n$$

Where,

n = speed in rps

d_{pitch} = diameter of pitch in m

For module number, $m = 1\text{mm}$,

$$\text{Pitch line velocity, } V = \pi * 0.02 * 50$$

$$= 3.14 \text{ m/s}$$

$$\text{Hence, the velocity factor, } K_v = \frac{3.56 + \sqrt{3.14}}{3.56}$$

$$= 1.50$$

$$\text{After that, the maximum bending stress, } \sigma = \frac{1.50 * 3000}{0.04 * 0.001 * 0.322}$$

$$= 349 \text{ MPa}$$

4.2 Bending Analysis Results

Module Number (mm)	Maximum Bending Stress (MPa)		Difference Error (%)
	Theoretical	Simulation	
1	349	251	28.08
2	198	183	7.58
3	144	112	22.22
4	116	78	32.76
5	98	49	50.00

Table 4.1 Comparison of Maximum Bending Stress for Theoretical and Simulation

The table above shows the result obtained from both calculation (theoretical) and ANSYS (simulation). The maximum bending stress is decreasing as the module number increasing. The highest maximum bending stress is 349 MPa for theoretical and 251 MPa for simulation which is belong to module number 1mm. The percentage difference for this module number is 28.08%.

While for the lowest maximum bending stress is 98 MPa (theoretical) and 49 MPa (simulation) which produced by gear module number 5mm. The percentage difference is 50.00 % which is slightly higher.

However, the better analysis will be module number 2mm. This is because the percentage error is less difference between theoretical value and simulation value which is only 7.58%.

There is a difference between theoretical value and simulation value because in ANSYS, all of factors are considered in the bending stress calculation like contact region, meshing part, and others. While for calculation by using Lewis Equation, the calculation on restricted with the fixed equation only.

The good result is when the percentage error is less because it shows that calculation is match with the ANSYS calculation.

Module Number (mm)	Velocity Factor (Kv)	
	Theoretical	Simulation
1	1.50	1.08
2	1.70	1.57
3	1.86	1.44
4	2.00	1.34
5	2.11	1.05

Table 4.2 Comparison of Velocity Factor for Theoretical and Simulation

From the table, the velocity factor increased as the module number increased. Velocity factor is the important factor to be considered in the Lewis Equation calculation. It indicates the speed of pitch line of gear that should be run during gear mating. It is logically to state that gear with highest module number can withstand the highest velocity such as module number 5mm with velocity factor of 2.11 (theoretical) and 1.05 (simulation).

4.3 Result Analysis

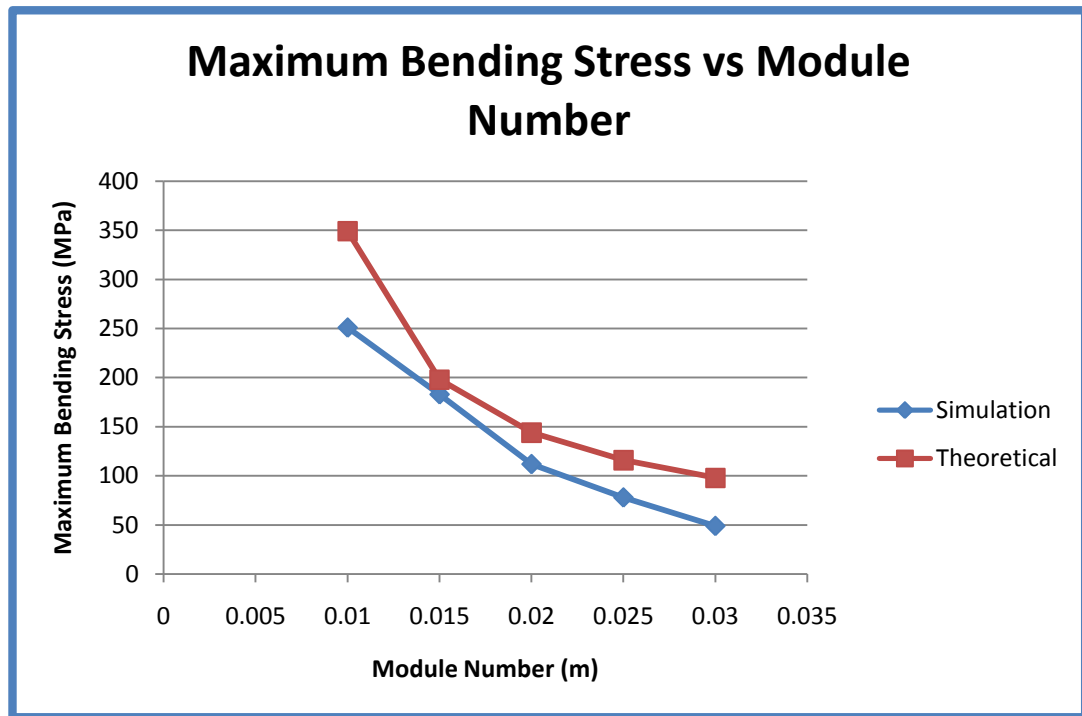


Figure 4.2 Maximum Bending Stress versus Module Number

The graph above shows that the inclination of theoretical value is steeper than the simulation value. The theoretical value is higher than the simulation value. This is due to the highest module number will generate the lowest maximum bending stress. Technically, the size of gear tooth is greater as the module number increases. Hence, the size of fillet at gear root tooth will increase as well. Since the possibility of bending failure to occur at the root of gear tooth, the larger gear fillet size will obtain less maximum bending stress.

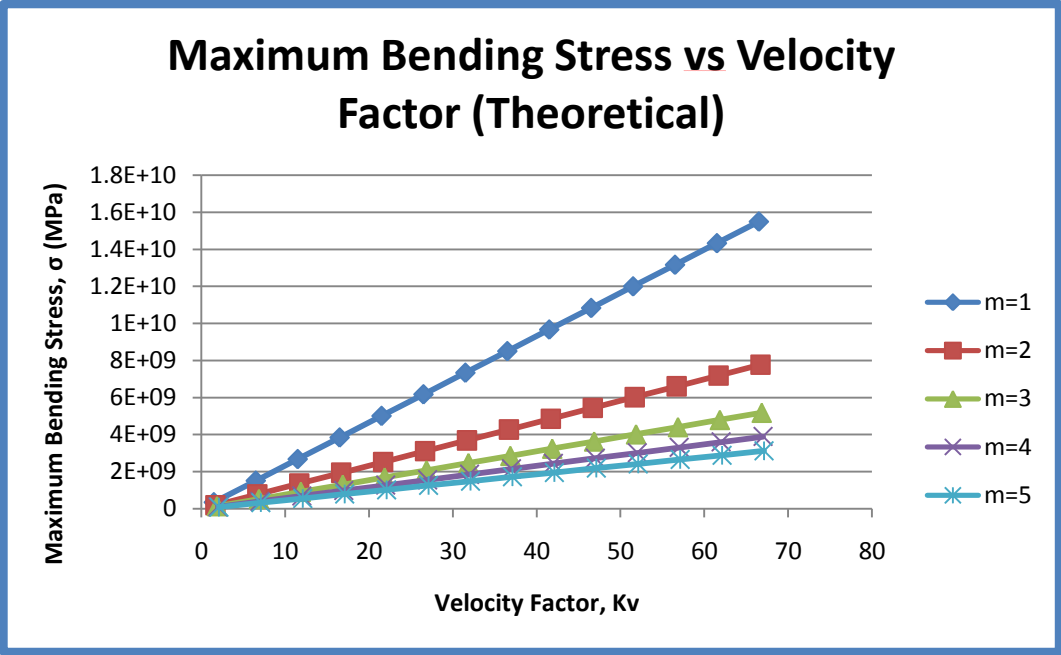


Figure 4.3 Maximum Bending Stress versus Velocity Factor for Theoretical

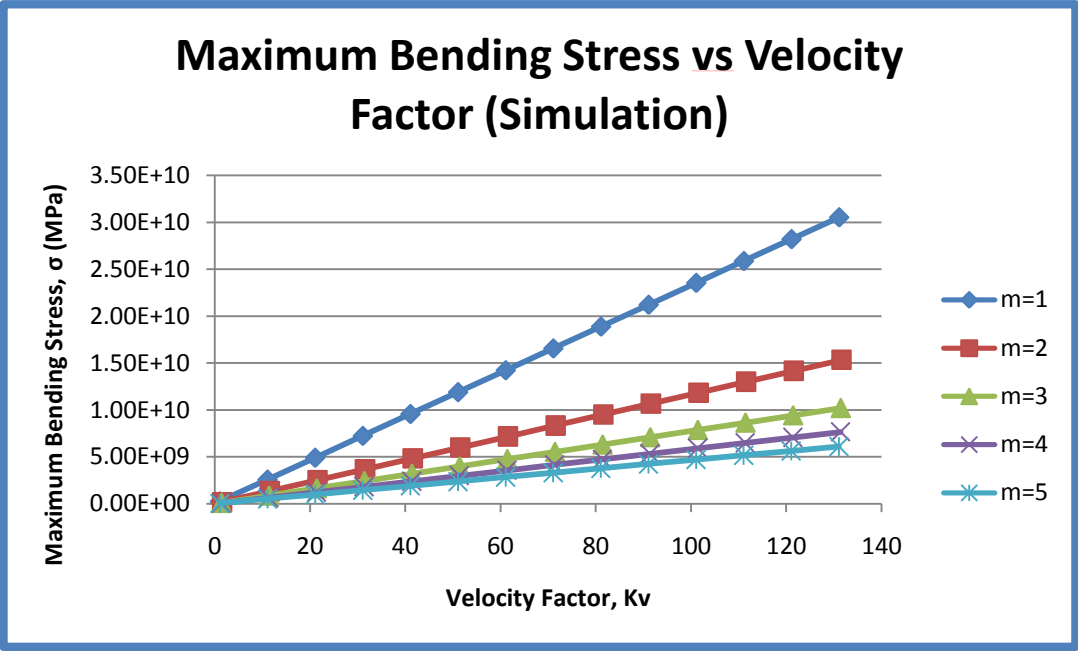


Figure 4.4 Maximum Bending Stress versus Velocity Factor for Simulation

The comparison of both graphs above is that the simulation value is higher than the theoretical value. This is because the velocity factor is related with maximum bending stress as well in the calculation. As the maximum bending stress increases, the velocity factor will increase as well. Hence, the maximum bending stress is directly proportional to the velocity factor for both theoretical and simulation results.

CHAPTER 5

CONCLUSIONS AND RECOMMENDATIONS

5.1 Conclusions

In this project a two dimensional deformable-body model of spur gear was developed. The results obtained were then compared with the Lewis Equation theoretical values. Moreover, the results are in good congruence with the theoretical values, which implies that the model designed is correct. The module is important geometrical parameter during the design of gear. As it is expected, in this project that the maximum bending stress decreases with increasing module number and it be higher at pitch point. As a result, based on this finding if the bending stress minimization is the primary concern and if the large power is to be transmitted then spur gear with higher module is preferred.

By using a relational equation modelling in CATIA software, one can accurately design complicated parts like involute tooth gears. This process is really helpful in this type of analysis as it needs model with high accuracy.

Besides that, the Finite Element Method results matched well with the theoretical results. The percentage error of both theoretical value and simulation value for maximum bending stress are not in huge difference since it is not more that 50%. Thus, this method turns out to be fast and accurate method of computing stress problem of the involute tooth gear system.

5.2 Recommendations

The gear study area is worthy of further research as computer capabilities increase. Hence, there are some future work could be done within this area. The first one is the further numerical method analysis should be conducted on three dimensionally meshed simulations for spur gears.

The second one is the other type of analysis like contact stress analysis that related with Hertz's Equation. This type of analysis can solve the gear failure of pitting or scoring. Besides, this type of analysis also can be performed on other types of gear like helical gears, spiral bevel gears, worm gear and others. The using of other types of gear can vary the results obtained. Apart from that, an analysis can also be conducted on a whole gearbox with all elements in the system such as the bearing and the gear casing. The last recommendation is perform the simulation of an oil film in contact some of any types of gear.

REFERENCES

- [1] Darle W.Dudley, 2002, "Handbook of Practical Gear Design", Gear Design Trends, Chap.1,pp. 1.1-1.19.
- [2] Cockerham, G., 1967, "Computer-Aided Design of Spur or Helical Gear Train", Computer Aided Design, Vol.8 No. 2, pp. 84-88.
- [3] Tucker,A. I., 1980 "The Gear Design Process", ASME Paper 80-c2/DET-13
- [4] Estrin, M., 1980, "Optimization of Tooth Proportion for a Gear Mesh", ASME paper 80 C2/DET-101.
- [5] Anon., 1966, "Design Procedure for Aircraft Engine and Power Take-Off Spur and Helical Gears", AGMA Standard No. 411.02
- [6] Anon, 1966, "Rating the Strength of Spur Teeth", AGMA Standard 220.02.
- [7] Savage, M., Coy, J.J., 1980, "Optimal Tooth Number for Compact Standard Spur Gear Sets", by Journal of Mechanical Design, vol.104 749-777
- [8] Rixin, Xu., 2008, "Finite Element modeling and simulation on the quenching effect for Spur Gear design optimization", M.Sc. Thesis, The University of Akron
- [9] R, Patchigolla., Y, Singh.,2006,"Finite Element Analysis of Large Spur Gear Tooth and Rim with and without Web Effects-Part I", M.Sc. Thesis, The University of Texas at San Antonio
- [10] Zeiping Wei', 2004, "Stress and Deformations in Involute Spur gears by Finite Element Method", M.Sc. Thesis, University of Saskatchewan.
- [11] Saxena, Rajul., 2004,"Finite Element Stress Analysis of Spur Gear Teeth" M.Sc. Thesis, The University of Texas at Arlington.
- [12] Richard G.Budynas, J. Keith Nisbett, 2008, "Mechanical Engineering Design", Spur and Helical Gears, Chapter 14, pp. 714-715.
- [13] Kahraman, A, Kharazi, A.A, Umrani, M, A "Deformable body dynamic analysis of planetary gears with thin rims"- *Journal of sound and vibration* 262(2003) 752-768
- [14] Jinliang Zhang, Zongde Fang, Xuemei Cao, Xiaozheong Deng, "The modified pitch cone design of the hypoid gear: Manufacture, stress analysis and experiment tests"- *Mechanism and Machine Theory* 42(2007) 147-158
- [15] Lin, Tengjiao, Ou, H, Li, Runfang, "A finite element method for 3D static and dynamic contact/impact analysis of gear drives"- *Computer Methods in Applied*

Mechanics and Engineering, Volume 196, issue 9- 12 (February 1, 2007), p. 1716-1728

[16] K.Mao, “gear tooth contact analysis and its application in the reduction of fatigue wear”*262(2007) 1281-1288*

[17] Shreyash D Patel, 2010, “Finite Element Analysis of Stresses in Involute Spur & Helical Gear”, M.Sc. Thesis, The University of Texas at Arlington.

[18] Zeiping Wei', 2004, "Stress and Deformations in involute Spur gears by Finite Element Method", M.Sc. Thesis, University of Saskatchewan.

[19] Ognyan Alipiev, 2011, “Method for Calculations of Spur Gear Drives with Asymmetric Involute-Latern Meshing” .

[20] Klensz, S. R., 1999, “Finite Element Analyses of A Spur Gear Set”, M.Sc. Thesis, Dept. of Mechanical Engineering, University of Saskatchewan.

[22] Gay, C.E., 1970, “How to Design to Minimize Wear in Gears” , Machine Design, Vol.42, Nov.26, pp.92-97.

[22] *Spur and Helical Gear Mechanical Drawing Specifications*. Retrieved on 25th April 2012 from

http://www.engineersedge.com/gears/engineering_drawing_gear_dimensioning.htm

[23] *Spur Gear Terms & Calculations*. Retrieved on 5th May 2012 from <http://shopswarf.orconhosting.net.nz/spur.html>

[24] *Introduction to Spur Gear*. Retrieved on 6th June 2012 from http://www.roymech.co.uk/Useful_Tables/Drive/Gears.html

[25] *Involute Gear and Spline Software*. Retrieved on 24th July 2012 from <http://quickgear.bizland.com/id11.html>

[26] *Finite Element Simulations with ANSYS Workbench 13*. Retrieved on 3rd August 2012 from

http://myweb.ncku.edu.tw/~hhlee/Myweb_at_NCKU/ANSYS13.html

[27] *American Gear Inc*. Retrieved on 5th August 2012 from <http://www.americangearinc.com/gears.html>

APPENDICES

APPENDIX A: MESHED PART OF SPUR GEAR

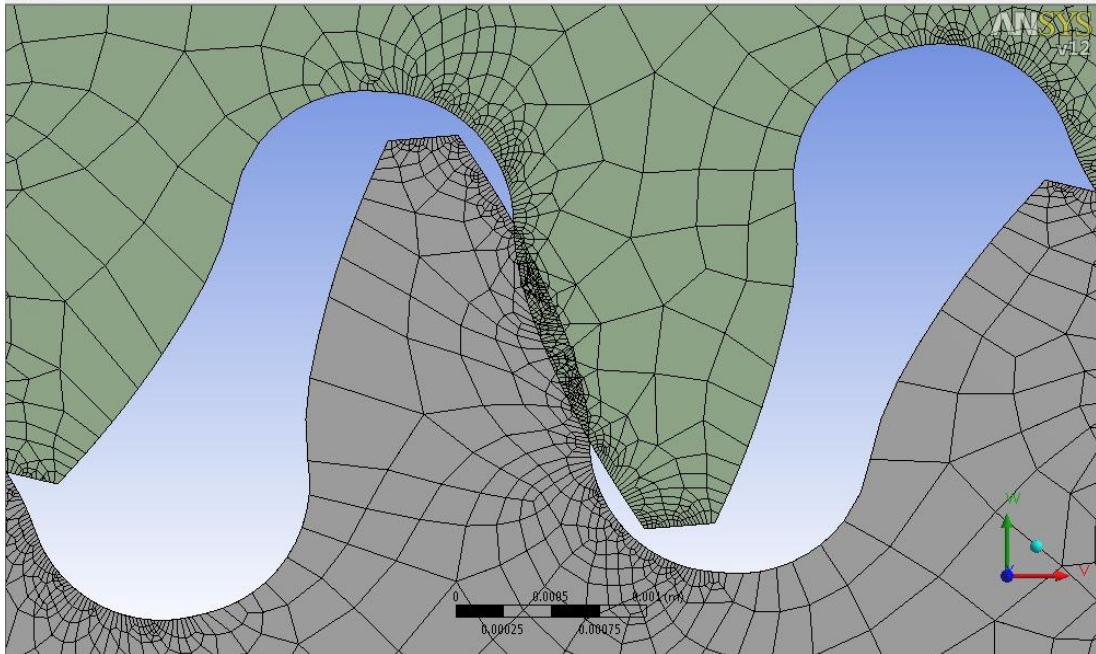


Figure A.1 Mesh Part for module 1mm

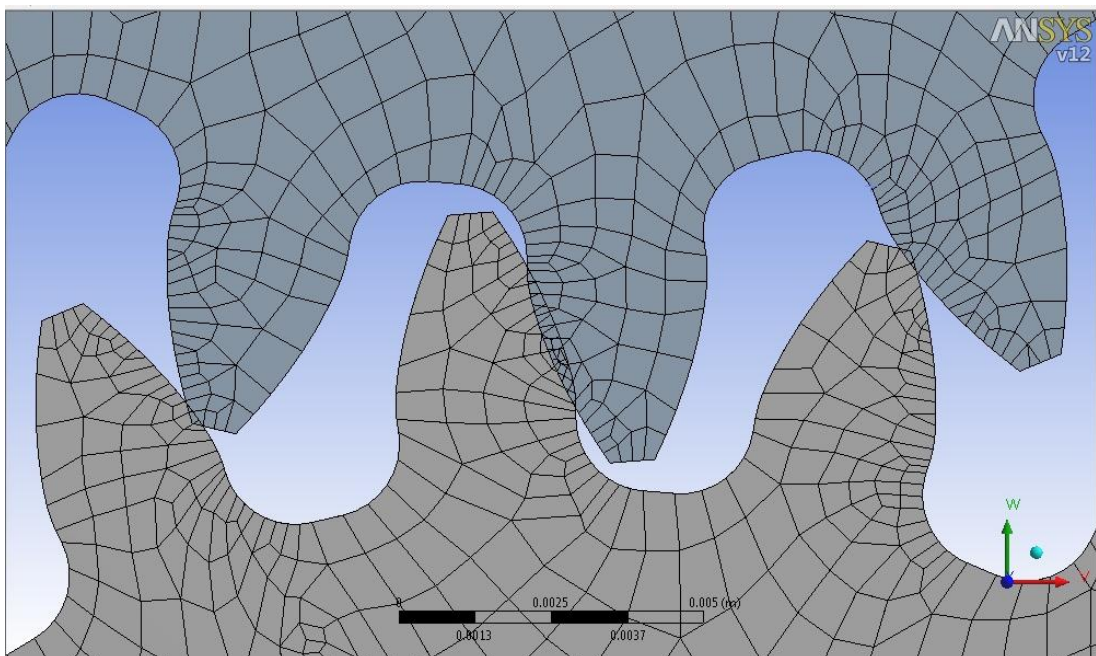


Figure A.2 Mesh Part for module 2mm

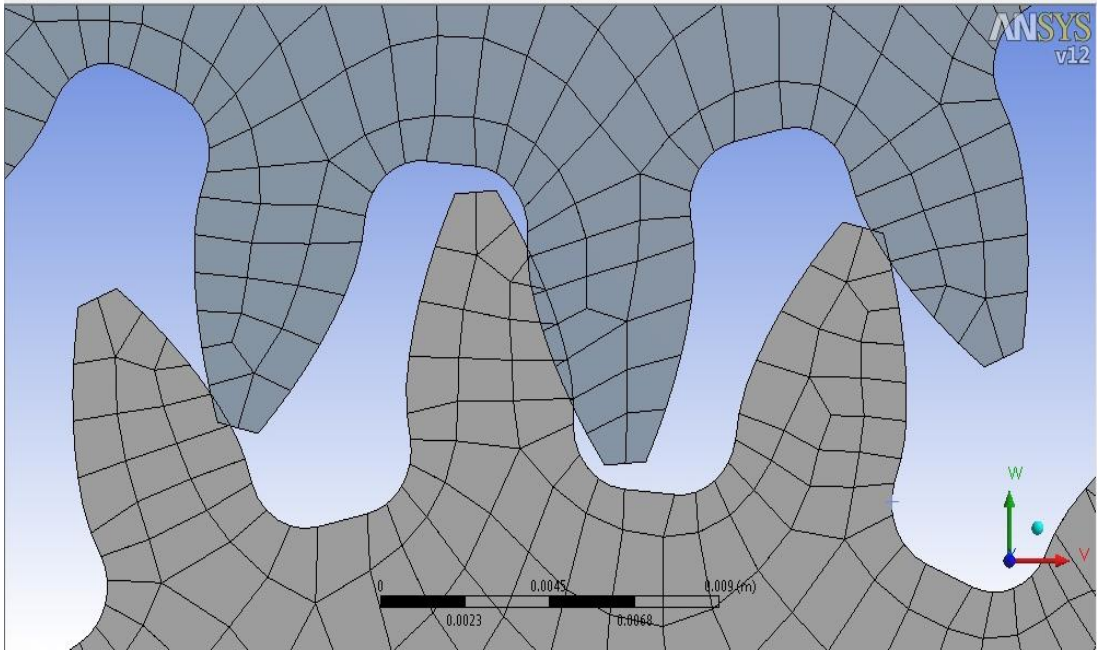


Figure A.3 Mesh Part for module 3mm

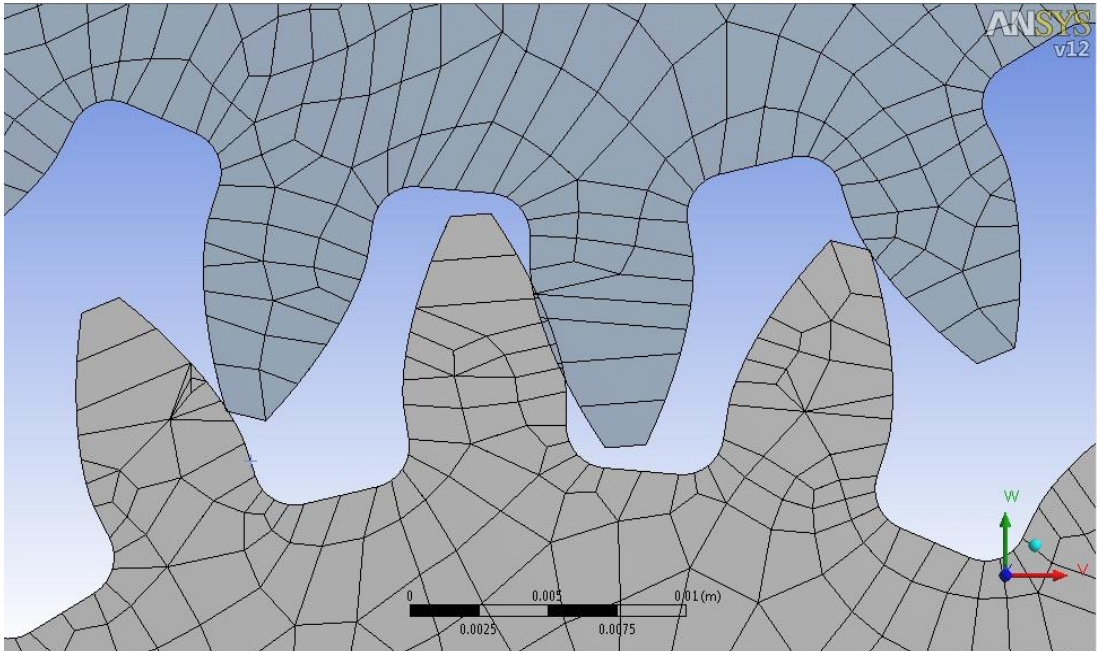


Figure A.4 Mesh Part for module 4mm

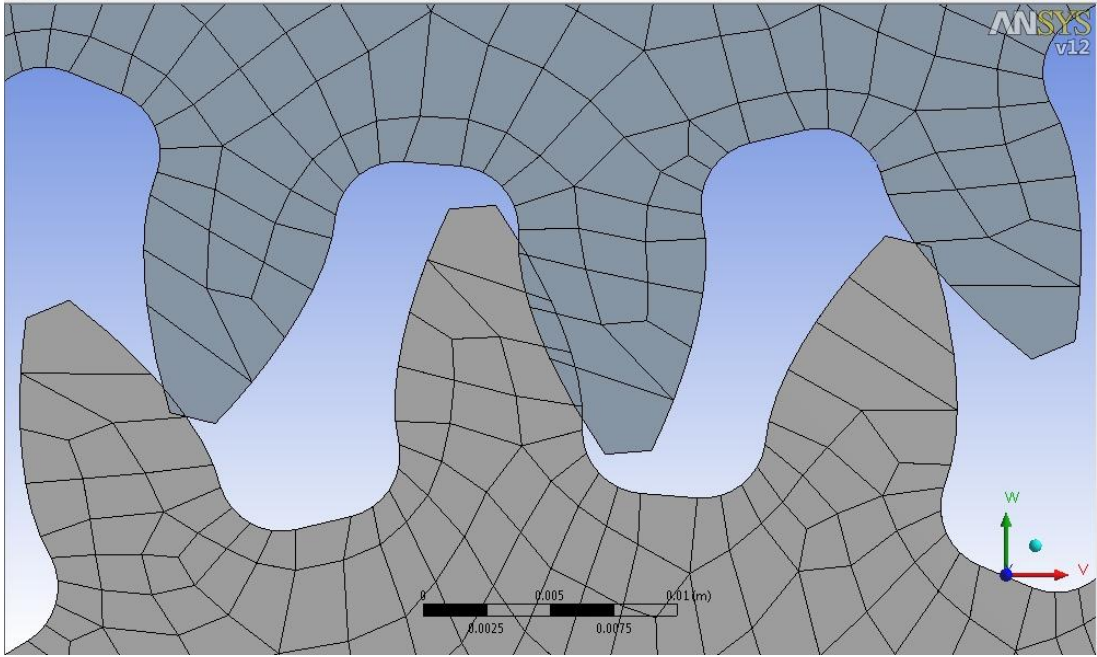


Figure A.5 Mesh Part for module 5mm

APPENDIX B: VON MISESS (BENDING) STRESS

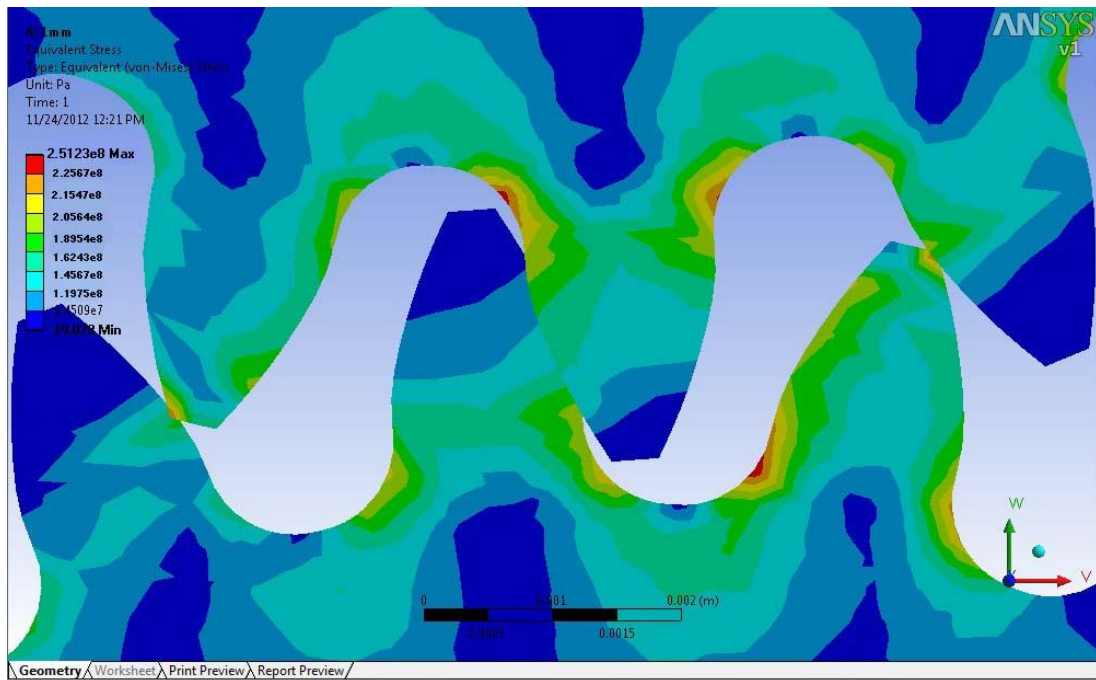


Figure B.1 Von Misses (Bending) Stress for Module 1mm

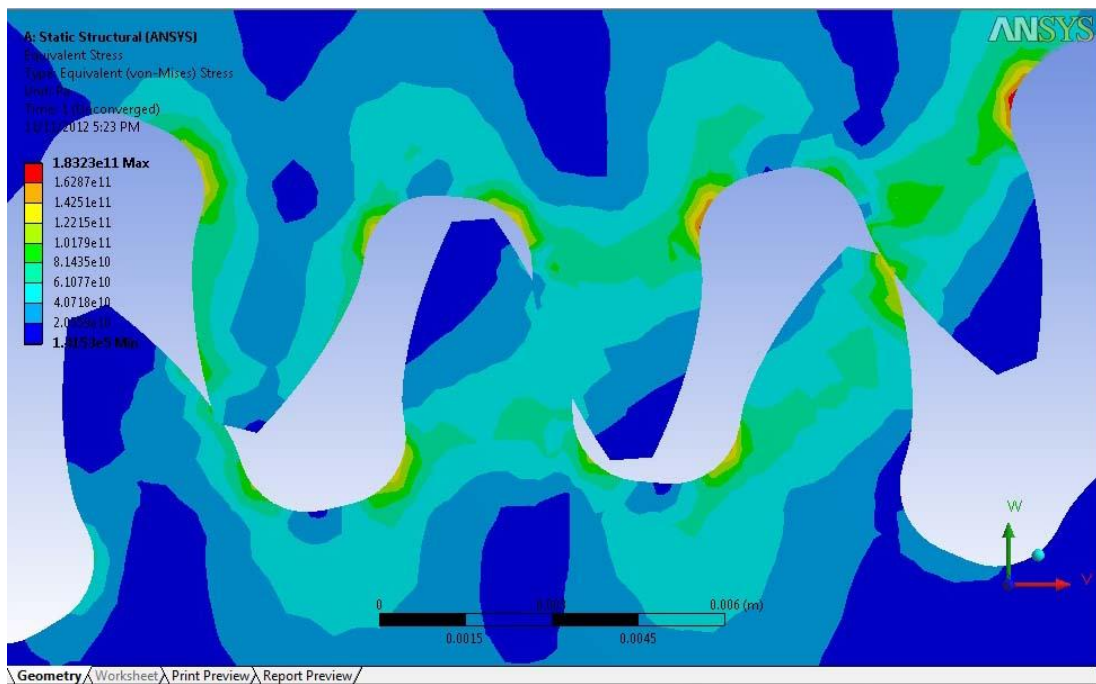


Figure B.2 Von Misses (Bending) Stress for Module 2mm

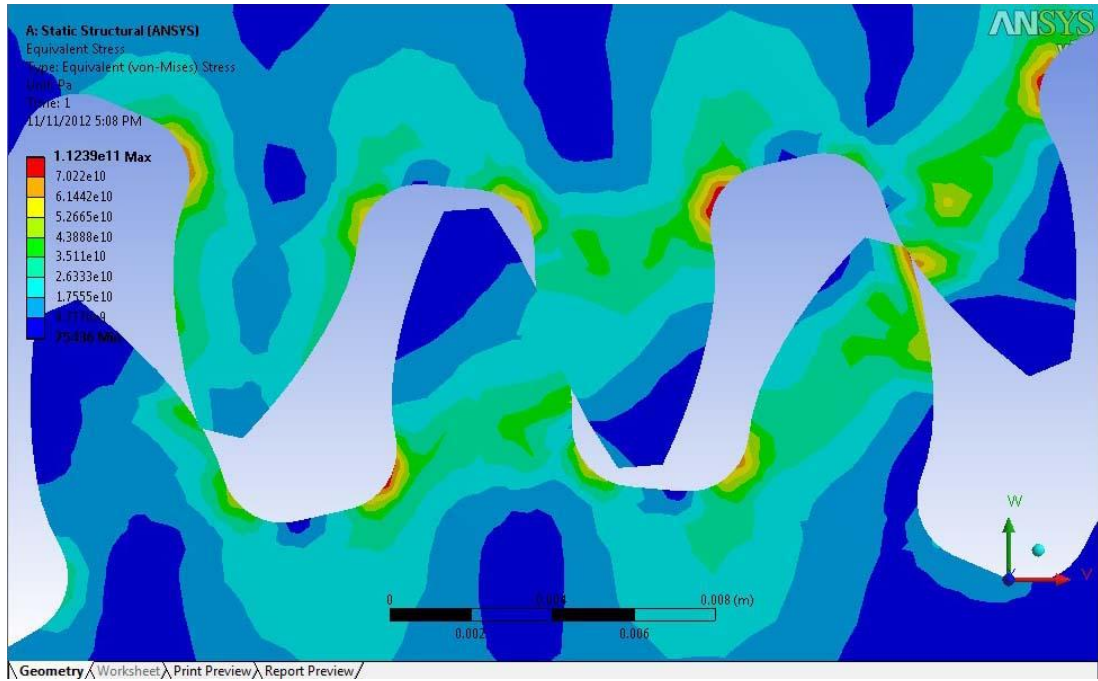


Figure B.3 Von Misses (Bending) Stress for Module 3mm

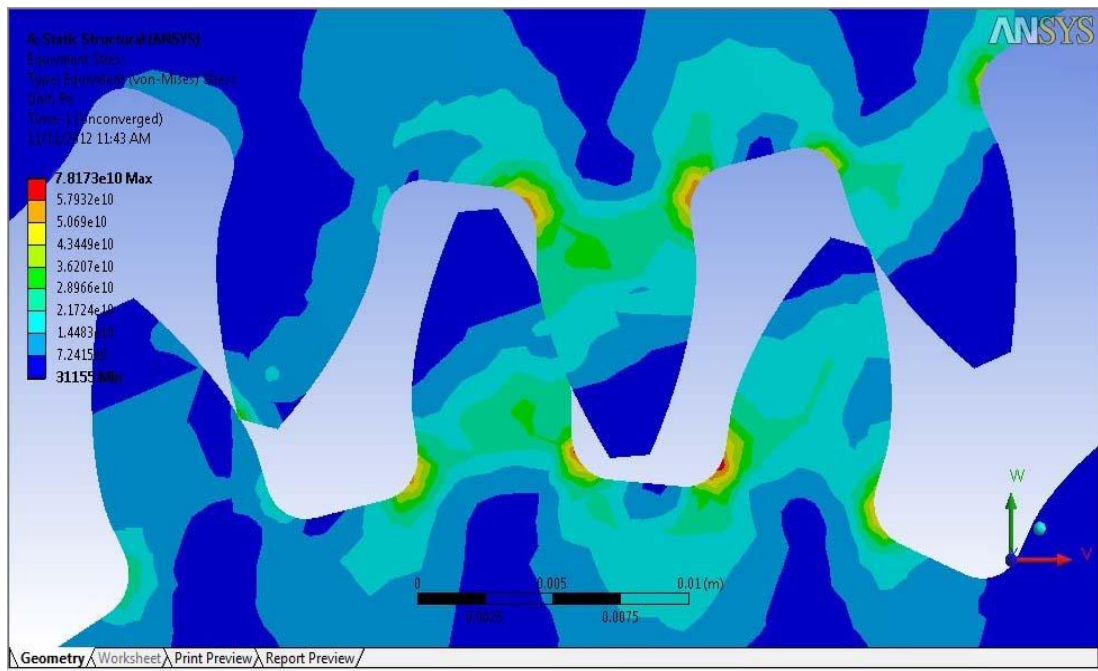


Figure B.4 Von Misses (Bending) Stress for Module 4mm

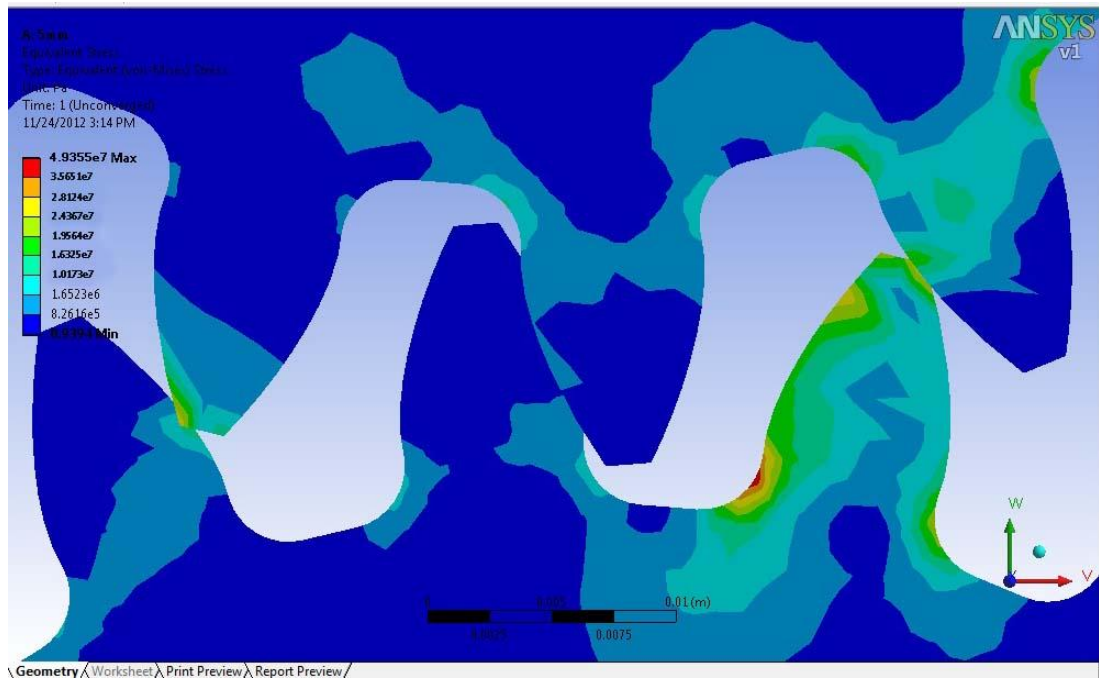


Figure B.5 Von Misses (Bending) Stress for Module 5mm