2012

Finite Element Analysis of Spur Gear Set

V.S.N. Karthik Bommisetty
Cleveland State University

Recommended Citation

Bommisetty, V.S.N. Karthik, "Finite Element Analysis of Spur Gear Set" (2012). *ETD Archive*. 630.
https://engagedscholarship.csuohio.edu/etdarchive/630

FINITE ELEMENT ANALYSIS OF SPUR GEAR SET

V.S.N KARTHIK BOMMISETTY

Bachelors of Technology in Mechanical Engineering

Acharya Nagarjuna University, India

May 2009

Submitted in partial fulfillment of requirements for the degree

MASTER OF SCIENCE IN MECHANICAL ENGINEERING

at the

CLEVELAND STATE UNIVERSITY

MAY 2012

This thesis has be approved for the Department of MECHANICAL

ENGINEERING and the college of Graduate Studies

_____

Thesis Chairperson, Dr. Majid Rashidi

_____

Department & Date

_____

Dr. Rama S.R Gorla

_____

Department & Date

_____

Dr. Asuquo B Ebiana

_____

Department & Date

ACKNOWLEDGEMENT

First and foremost I offer my sincerest gratitude to my supervisor, Dr. Majid Rashidi, who has supported me throughout my thesis with his patience and knowledge whilst allowing me the room to work in my own way. I attribute the level of my Master's degree to his encouragement and effort and without him this thesis, too, would not have been completed or written. One simply could not wish for a better or friendlier supervisor.

I would like to thank Department of Mechanical Engineering for providing me the support, labs and assistance

Finally, I would like to thank my parents for their continuous support and encouragement.

FINITE ELEMENT ANALYSIS OF SPUR GEAR SET

V.S.N KARTHIK BOMMISETTY

ABSTRACT

A Finite Element procedure has been developed in this work to determine the load distribution factor, Km, of the AGMA formula for a set of spur gear. At first, a spur gear with perfect involute is modeled using a 3-D CAD software. The model is them is assembled with shafts having 1, 2, and 3 degree misalignments. The generated 3-D models were in turn imported to ANSYS workbench to calculate the maximum bending and contact stresses using finite element method. The results generated were then compared with the maximum bending stress results obtained for parallel shafts to estimate the Load Distribution Factor Km. This study resulted in Km values of 1.03, 1.11, and 1.14.

TABLE OF CONTENTS

CHAPTER

LIST OF TABLES

Table

LIST OF FIGURES

Figures

CHAPTER I

INTRODUCTION

Gears are mechanical components used for transmitting motion and torque from one shaft to another. Ever since invention of rotating machines, gears existed. Early records states that around 2600 BC Chinese used gears to measure the speeds of chariots. In 250 B.C Archimedes used a screw to drive toothed wheels which were used in engines of war. In 4 century B.C., Aristotle used gears to simulate astronomical ratios. Greek and Roman literatures mention the extensive use of gears in clocks of cathedrals and ecclesiastical buildings [1].

During early centuries gears were made of either wood or stone teeth set in wood. Later during metal ages Iron, Bronze or tin were used instead of stone. There was no standard procedure for gear manufacturing until 1835 when English inventor Whitworth

patented the first gear hobbing process [2]. The Pfauter of Germany patented the first gear hobbing machine capable of cutting both spur and helical gearing in 1897, they introduced the first NC hobbing machine and in 1975 and they introduced the first all 6 axis gear hobbing machine in 1982.

Although gear manufacturing has achieved lots of advancement during its evolution, however the failure of gear due to bending and contact stress still remained a challenge for designers and manufacturers until 1892. In 1892 the Philadelphia Engineers club first recognized Wilfred Lewis presentation of stresses on the gear tooth and it still serves as the basis to determine the gear stress [3].

The Lewis bending equation has a lot of draw backs which include

1. Load on gear tooth is dynamic and is influenced by pitch-line velocity.

2. The entire load is carried on single tooth.

3. The location of application of load is not true as the load is shared by the tooth.

4. The Stress concentration factor at tooth fillet is not considered.

In order to overcome all these factors AGMA (American Gear Manufactures Association) came out with several factors which influence bending stress on the gear tooth which were discussed in detail in Chapter II [4].

Although bending stress on an involute spur gear can be calculated using AGMA bending stress number but the contact stress on spur gear are approximated using Buckingham contact stress equation. The AGMA contact stress equation assumes the

meshed gear tooth as two cylinders with parallel axis and predicts the contact pressure using Hertz Contact Stress equation.

For practical considerations the contact stress on involute spur gear can be better approximated using Finite Element Method [5]. This Method can be used in approximating any kinds of stress, strains and deformations in single parts and assemblies.

Finite Element Method is a numerical method [6] to obtain approximate solutions to partial differential equations and integral equations. This method originated for solving complex elastic and structural analysis problems. The first people to develop this method were Alexander Hrennikoff and Richard Courant [7]. In 1947 Olgierd Zienkiewicz coined the term Finite element Analysis by gathering these methods. In 1952 Boeing made a great effort to analysis the aircraft structures using Finite element Methods and in 1964 NASA developed a software in Fortran language called Nastran to analysis the aircraft structures. In mid-1970 due to advancement in computer technology many software's capable of performing Finite element analysis were available.

Among the stress prediction factor in AGMA stress prediction formula, there is a factor, K_m, which accounts for the load distribution across the face of a typical gear. The non-uniform stress distribution is mainly caused by misaligned of the shaft and distortion of the gear hub. It is the purpose of this thesis to create a F.E.A method which can be used to postulate intentional shaft misalignments and predict the resulting stresses in a typical spur gear set. The results can then be analyzed to predict the load- distribution factor, K_m, of the AGMA formulation.

CHAPTER II

BENDING AND CONTACT STRESS IN GEAR TOOTH

2.1 AGMA Bending Stress of Gear Tooth

Bending Stress on a typical gear tooth is mainly resulted from the tangential force components acting on the tooth. Based on this, Wilfred Lewis derived an equation to determine the bending stress on a gear tooth. Assuming gear tooth as a cantilever beam and the entire load to be transmitted through one tooth, an equation for the bending stress is defined as

$$\sigma = \frac{W_t.P_d}{b.Y} \qquad (2.1)$$

Where

- W_t is Tangential load acting on the gear tooth
- P_d is Diametrical Pitch of Gear

- Y is Lewis Form Factor

- σ is Bending Stress

- b is Face Width

In the above equation stress concentration at tooth fillet is not considered, so AGMA derived a new bending stress equation by introducing stress concentration factor to reduce the impact of stresses near tooth fillet which is given by

$$\sigma = \frac{W_t.P_d.K_t}{b.Y} \tag{2.2}$$

Where

- W_t is Tangential load acting on the gear tooth

- P_d is Diametrical Pitch of Gear

- b is Face Width

- Y is Lewis Form

- K_t is Stress concentration factor

- σ is Bending Stress

For Practical design considerations AGMA suggests more design factors to estimate bending stress on gear tooth. The modified bending stress is given by the following equation

$$\sigma_s = \frac{W_t.P_d}{b.J}.K_o.K_v.K_m.K_s.K_B \tag{2.4}$$

Where

- W_t is Tangential force acting on the gear tooth

- P_d is Diametrical Pitch

- b is Face Width

- K_o is Overload factor

- K_v is Dynamic factor

- K_m is Load distribution factor

- K_s is Size distribution factor

- K_B is Rim thickness factor

The definitions of these factors are presented in the following section

2.2 Definition of AGMA Bending Stress Factors

2.2.1 Overload Factor (K_o)

This factor represents the behavior of the gear on application of the external loads. These factors mainly include the vibrations, shocks, speed variations and other specific loading conditions.

2.2.2 Dynamic Factor (K_v)

Dynamic Factor mainly depends on the tooth profile inaccuracies, speed of approach of tooth, vibrations of tooth during meshing, tooth friction, Dynamic imbalance of rotating members, and Gear shaft misalignment and elastic property of the tooth. Dynamic factor is generally given by the following formula

$$K_v = \left(\frac{A + \sqrt{V}}{A} \right)^B \qquad (2.5)$$

$$(V_t)_{max} = [A + (Q_v - 3)]^2 \qquad (2.6)$$

Where

- A is 50+56(1-B),

- $B = 0.25(12\text{-}Q_v)^{2/3}$

- Q_v is Transmission accuracy level

- V_t is Pitch line velocity

2.2.3 Load Distribution Factor (K_m)

Determination of load distribution factor depends on various factors which include the design of gear as well as design of shaft, bearings, housing and structure on which gear drive is mounted. The main objective of the load distribution factor is to reflect the non-uniform load distribution across the line of contact. The load distribution factor can either be defined by AGMA 2001/2101 or can be defined directly which is given by

$$K_m = 1.0 + C_{pf} + C_{ma} \qquad (2.7)$$

Where

- C_{pf} is Proportion Factor

- C_{ma} is Mesh alignment Factor

2.2.4 Size Distribution Factor

Size Distribution factors resonates the material non-uniform properties due to its size. The main factors include

- Tooth Size

- Diameter of part

- Ratio of tooth size to diameter of part

- Face Width

- Area of Stress Pattern

However AGMA suggests the value of K_s to be 1 for diametrical pitch greater than or equal to 5 and following table suggests the size factor for diametrical pitches less than 5.

I: AGMA Suggested Size Factors

Diametrical Pitch	Size Factor K_s
4	1.05
3	1.15
2	1.25
1.25	1.40

2.2.5 Rim Thickness Factor

The Lewis Equation assumes gear tooth as a cantilever beams attached perfectly to a rigid base. If the rim is thin, it deforms and causes the point of maximum stress to

shift from fillet to some point on the rim. This factor estimates the influence of stress on the rim which is given by

$$K_B = \begin{cases} 1.6 \ln \dfrac{2.242}{m_b} & ; \quad m_b < 1.2 \\ 1 \\ & ; m_B \geq 1.2 \end{cases} \tag{2.8}$$

Where

- m_b is equal to t_r / h_r

- t_r is Rim thickness below the tooth

- h_r is tooth height.

2.3 Hertz Contact Stress

The study of deformation of solids which are in contact at one or more points is called Contact Mechanics. Pressures and Adhesion acting perpendicular to the contact surfaces are its central aspects. The present study deals with the cylinders with two parallel axes which are in contact. The contact in between is a Non-Adhesive contact.

1: Cylinders
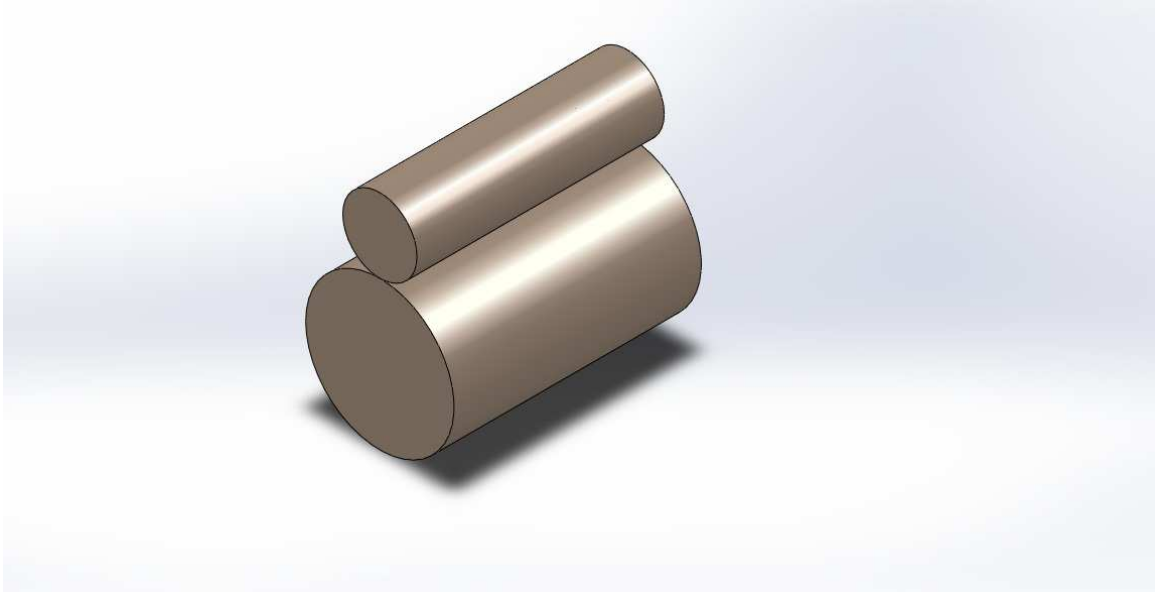
The Cylinders used in this study are having a length L, d_1 and d_2 as diameters. On application of Force F, on the surfaces of the cylinders a narrow rectangle of width 2b and length L is generated at the contact surfaces. A pressure is developed in the rectangle area in an elliptical shape. The half width of the ellipse is given by

$$c = \sqrt{\frac{2F}{\pi L} \cdot \frac{\dfrac{(1 - v_1^2)}{E_1} + \dfrac{(1 - v_2^2)}{E_2}}{1/d_1 + 1/d_2}} \qquad (2.9)$$

Where

- c is ellipse half width

- F is Force acting on surface of Cylinder

- L is length of Cylinder

- d_1 and d_2 are diameters of Cylinder

- v_1, v_2 represent the Poisson's ratio of cylinder material

- E_1, E_2 represent Young's Modulus of the materials.

The maximum contact pressure in between the cylinders acts along a longitudinal line at the center of the rectangular contact area, which is given by:

$$P_{max} = \frac{2.F}{\pi.c.L}$$

(2.10)

Where

- F is force acting on the Cylinders

- C is half width of the ellipse

- L is length of Cylinder

- P_{max} is the maximum pressure generated

2.4 F.E.A Modeling and Design of Gear Tooth for Bending Stress

Involute spur gears are the most common form of gears which are used to transfer the motion between the parallel shafts. The main concerns while designing an involute spur gear include generation of involute. In earlier days to design an involute spur gear there are many theoretical procedures to draw an approximate involute but no procedure was present to draw a perfect involute for performing Analysis. In the present day with the 3-D modeling software's it is easy to generate the involute spur gear with exact involute. For the current project, the involute spur gear with perfect involute is generated from Solid Works and is imported to Ansys Workbench.

2.4.1 Design of Gear Tooth

The involute spur gear used for the current analysis has the following specifications

1. 36 Tooth

2. Diametrical Pitch of 10

3. Pressure Angle of 20°

The gear is generated using following equations as expressions in solid works.

"N"= 36 '   Number Of Teeth

"Pd"= 10 '   Diametrical Pitch

"Pc"= "N"/"Pd" '   Pitch Circle Diameter

"Phi"= 20 '   Pressure Angle

"A"= 1/"Pd" '   Addendum

"D"= 1.157/"Pd" '   Dedendum

"Od"= ("N"+2)/"Pd" '   Outer Circle Diameter

"Bd"= "Pc"/cos("Phi") '   Base Circle Diameter

"Id"= ("N"-2)/"Pd" '   Internal Diameter

"b"= 0.4 '   Face Width

In addition to the above equations to generate a perfect involute we need the following parametric equations with parameters described
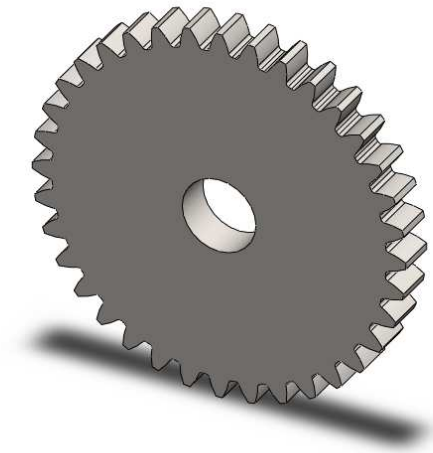
$$X_t = 0.7 * (\cos(t) + t * \sin(t)) \tag{2.10}$$

$$Y_t = 1.7 * (\sin(t) - t * \cos(t)) \tag{2.11}$$

Where

- t is angle parameter in radians

- X_t is the X function with respect to parameter t

- Y_t is the Y function with respect to Parameter t

Using the above equations a 2-D sketch of the spur gear is generated; it is now extruded to length equal to Face Width to obtain 3-D Involute Spur Gear.

2: Image showing the final view of generated gear

2.4.2 Analysis of Gear Tooth

Finite Element analysis for any imported 3-D model is performed in three main steps

- Pre-Processing
- Solution
- Post-Processing

Pre-Processing

The Pre-Processing mainly involves the modeling of the 3-D part. The following are the main steps in Pre-Processing

- Engineering Data

- Geometry

- Discretization

Engineering Data [8]:

In an analysis system the main resource for material properties is engineering data, they can either be experimental or user defined. In this analysis density and linear elastic properties like Young's Modulus and Poisson's ratio for structural steel are declared.

Geometry:

The gear geometry generated from solidworks is imported in to Ansys Workbench through Design Modeler.

Discretization:

Discretization is the method of converting continuous models to discrete parts. The goal is to select and locate finite element nodes and element types so that the associated analysis is sufficiently accurate. Element Aspect ratio must be near unity to obtain accurate results. For the current analysis average aspect ratio is obtained as 1.85 by setting the mesh relevance to fine and smoothing to medium and span angle center to coarse.

Tetrahedral elements [9] are used since stress is present all alone the thickness of the part. Patch independent algorithm is used since a finer mesh is required around edges and corner. For rest of the body a normal mesh is sufficient and in advanced meshing features the proximity and curvature feature needs to turned on since the curvature size function examines the curvature on the faces and edges and computes the element sizes so that the element size doesn't exceed the maximum size of curvature angle which are either defined by the user or taken automatically. The proximity size function allows defining the minimum number of element layers in the region that constitute gaps. The minimum size limit is defined as 0.1 in.

2: Image showing the final mesh obtained for the Involute gear
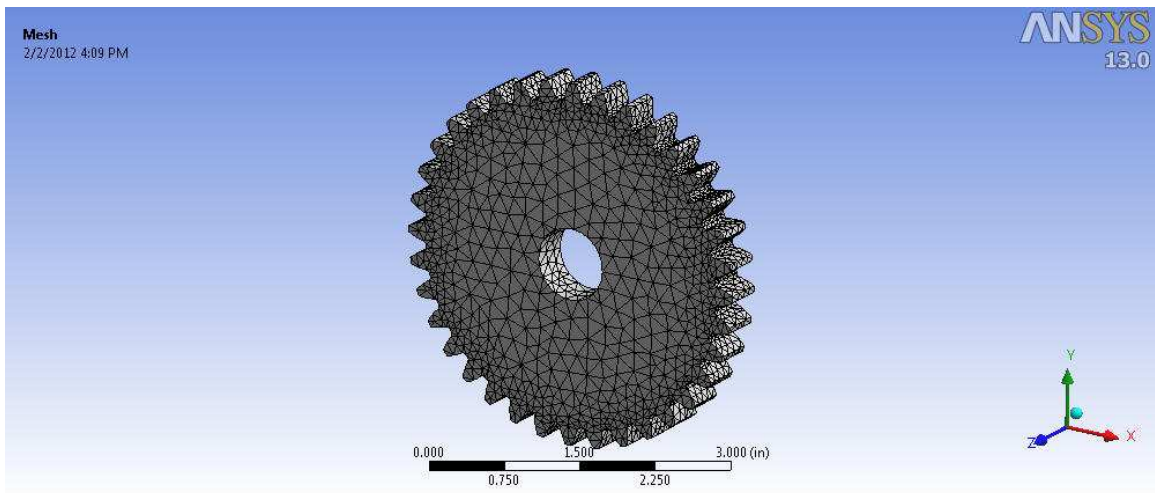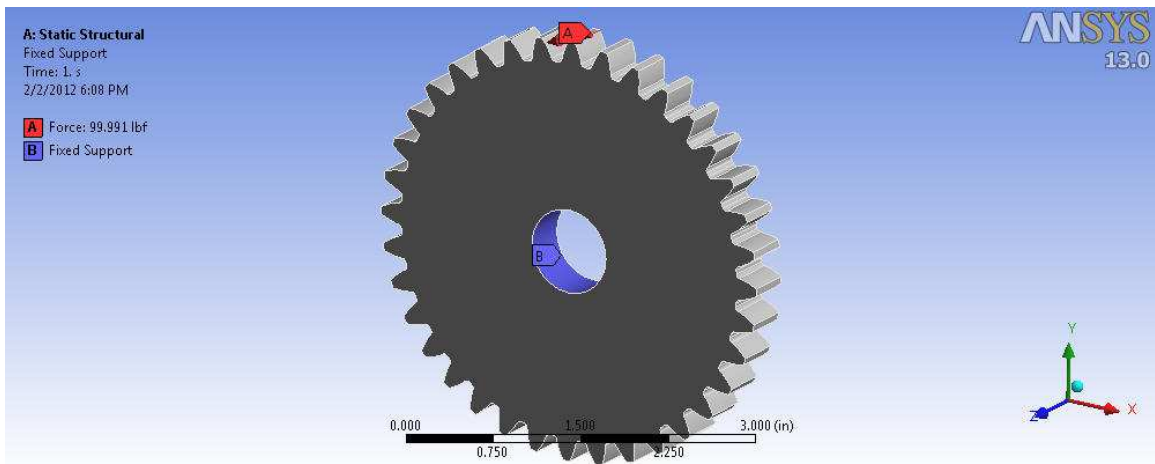

Solution:

Solution Part involves declaration of the Analysis type, location of forces and fixation of part. For the present analysis, Static structural type is used with no large deflection and inertia relief features. The solver type and weak spring's features are set to program controlled and all the nonlinear controls are turned off.

The gear is fixed at the center by the fixed support tool. Forces can be applied to the gear by selecting the edge from the graphics window and forces are defined in the component form.

II : Force Components

Co-ordinate System	Force (lbs.)
X- Component	-93.96 lbs. (Ramped)
Y- Component	-34.20 lbs. (Ramped)
Z- Component	0 lbs.

3: Location of Force applied and Fixed support

Post Processing

The post processing stage involves viewing of data files generated by the software during the solution phase. Bending stresses for the gear can be obtained from the Normal stresses menu by choosing the normal stress in Y-direction. The following figure displays the stress contour of the gear tooth.

4: Bending Stress of Gear Tooth
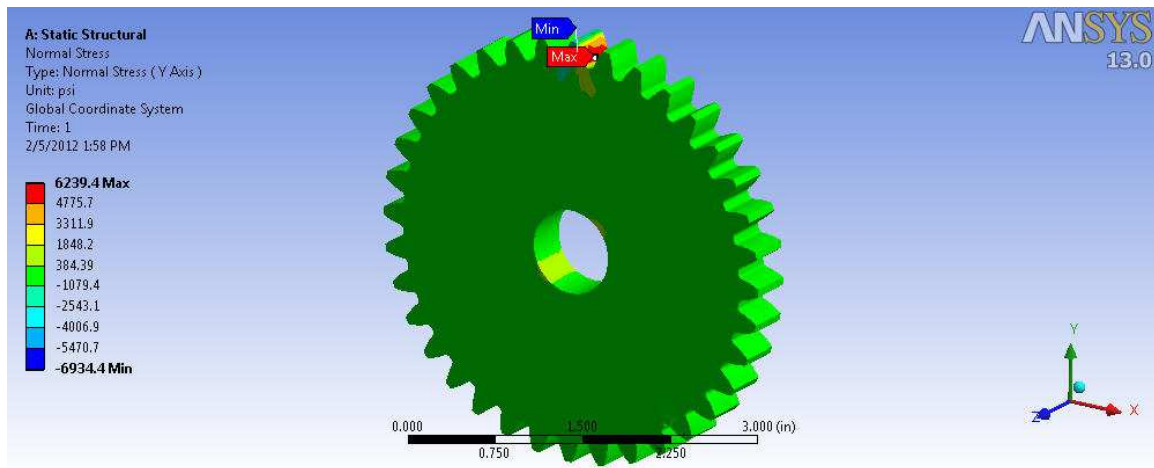
2.5 F.E.A Modeling and Design of Cylinders for Hertz Contact Stress

Gear tooth may not only break due to bending stress during its life time but develops pits on tooth surface due to high contact stresses fatiguing the surface by compression. This is intensified near the pitch circle where contact is pure rolling with zero sliding velocity. A close form solution with two cylinders which are in contact is made to determine the contact stresses. In this section these contact stresses in the cylinders are determined using finite element method.

2.5.1 Design of Cylinders

The current design has the two cylinders with 1 inch diameter and height equal to that of face width of the involute gear. The following are the specifications of the cylinder

1. Diameter is 1 in
2. Height is 0.4 in

Cylinders are designed as two separate entities and are assembled together to perform analysis. The following figure displays the final view of the generated cylinder.

5: Generated Cylinders
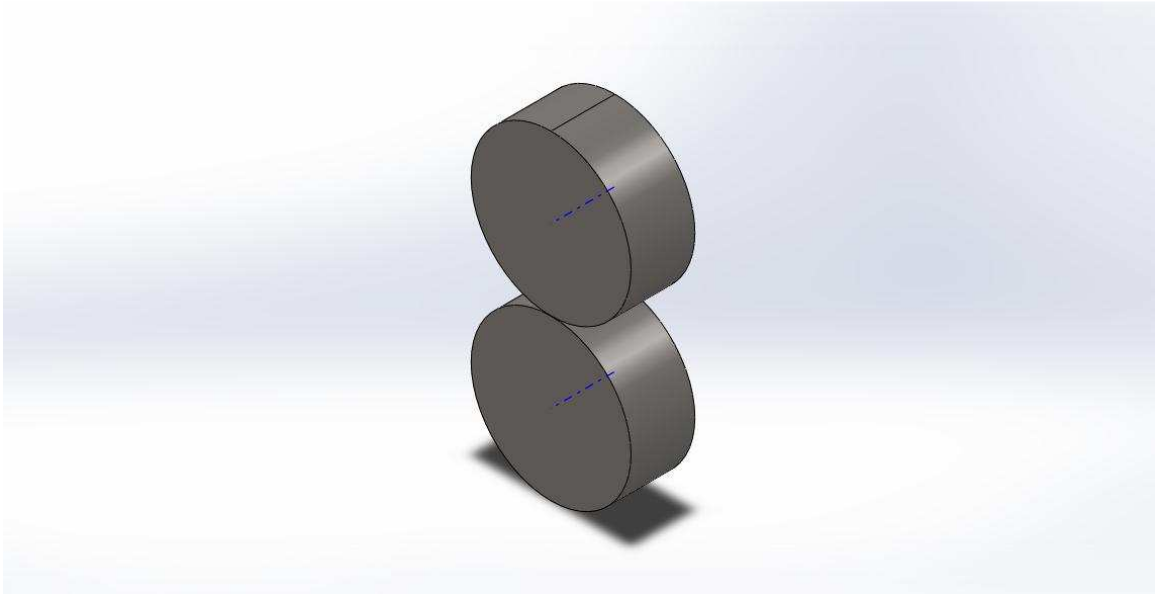
2.5.2 Analysis of Cylinder

Analysis is performed in three main steps

1. Pre-Processing
2. Solution
3. Post-Processing

Pre – Processing

Pre-Processing involves three steps.

1. Engineering Data
2. Geometry
3. Discretization

Engineering Data

For current analysis, an alloy with Poisson's ratio of 0.28 and Young's Modulus of 30E6 psi is used.

Geometry

The geometry for the current model is to be done carefully since it involves the connections. For the current model a "No Separation" contact is used in between the cylinders since the cylinders should always be in contact.

Pure Penalty formulation [10] is used for the above contact analysis because of single contact and absence of large deformation. The Normal stiffness is controlled by program. The following image displays the location of No – Separation contact.

6: No separation Contact
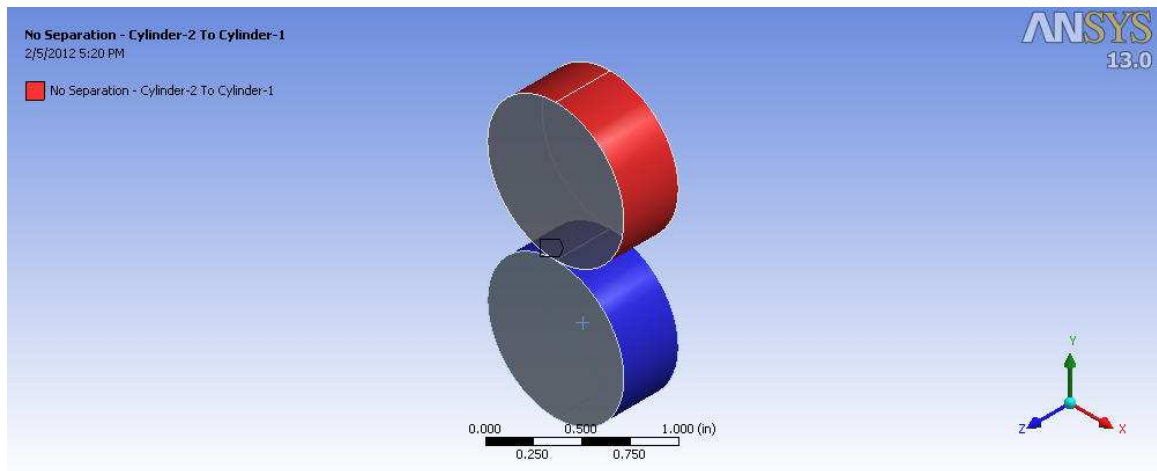
Discretization

Discretization for an assembly requires complex meshing since it involves contacts. For the present analysis a mesh with an aspect ratio of 2.00 is required. This can be obtained by setting relevance to fine, initial smoothing to medium and span angle

center to coarse. Individual meshing is preferred. Contact sizing should also be used to obtain a more uniform fine density mesh and to obtain a better distribution of contact pressure.

The refinements in the mesh can be controlled by adding the refinement control tool. To obtain a minimum refinement the refinement control tool is set to 1. Patch Conforming Algorithm with tetrahedral elements is used in the mesh control method. The following image shows the final view of the refinement.

7: Mesh Refinement
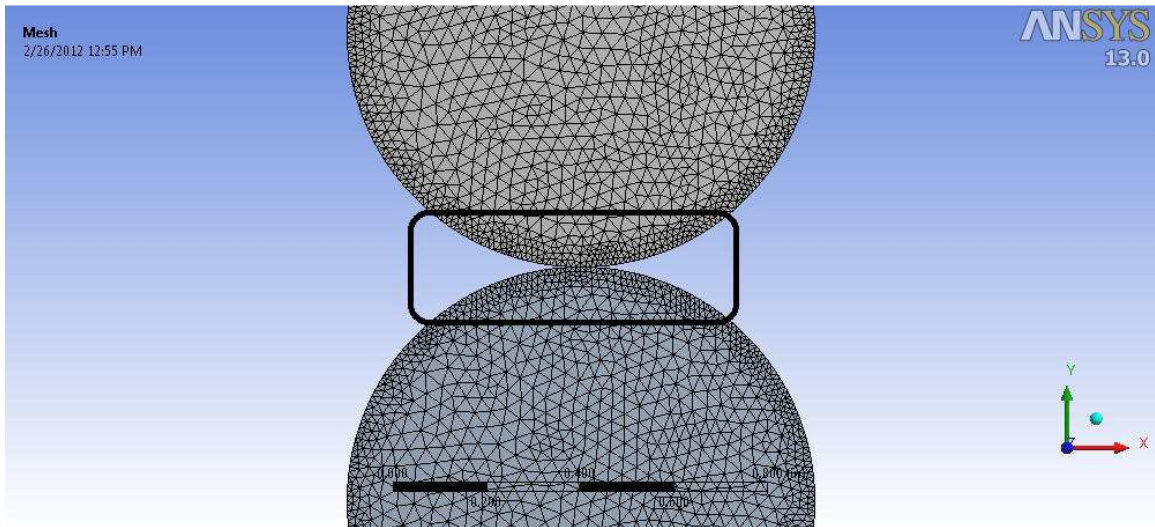
Solution

Analysis is performed as a static structural analysis with no large deflections and weak springs are turned on and the spring stiffness is controlled by program. All the non-linear functions are turned off and the solving method is set to Direct. Loads are applied in the Y-direction.

Post – Processing

To obtain Contact pressures a contact tool must be used. The contact surfaces on which the pressures to be determined are selected and then evaluated for contact pressures.

8: Contact Pressure
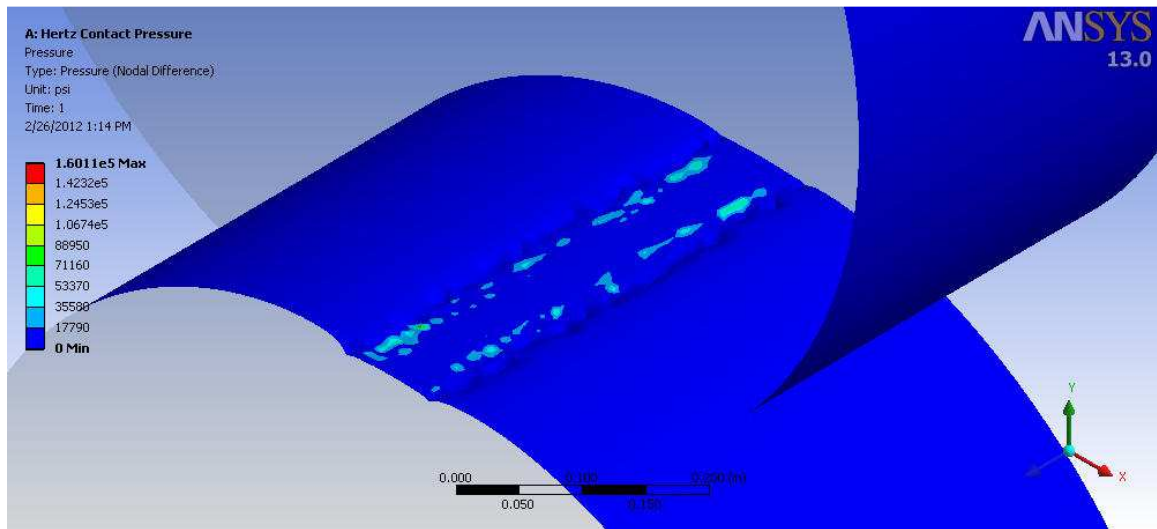
2.6 Validation of Bending and Contact Stress Results Predicted by FEA

The results obtained from FEA analysis are verified by comparing them with the standard theoretical procedures. FEA bending stresses are validated by comparing them with standard Lewis bending stress equations and FEA contact stresses are compared with the Hertz contact stresses. A table at the end of the chapter demonstrates the deviation of the FEA results with the standard theoretical results.

2.6.1 Bending Stress Validation

As mentioned in the above section standard bending stress is compared with the Lewis bending stress equation. These values can be calculated from equation (2.1)

Gear Specifications

Number of Teeth   N: 36

Diametrical Pitch P_d: 10

Pressure Angle   φ: 20

Face Width   b: 0.4

Lewis Form Factor   Y: 0.38

Now from the above gear specifications, Lewis bending stress from equation (2.1) is obtained as

$$\sigma_{Lewis} = \frac{93.96 \times 10}{0.4 \times 0.38}$$

$$\sigma_{Lewis} = 6181.57 \text{ psi}$$

Bending stress obtained from FE Analysis is given by

$$\sigma_{FEA} = 6239.4 \text{ psi}$$

2.6.2 Validation of Contact Stress

As mentioned above contact stresses are verified by comparing them to Hertz Contact stress. The following are the specifications and other factors used for calculation of the hertz contact stress

Modulus of Elasticity   E: 30E6

Poisson's Ratio   v: 0.28

Load W_t : 500lb

Length   L: 0.4

Diameter   d: 1

From above parameters the contact stress are given by the equation (2.8)

$$\sigma_c = \sqrt{\frac{500}{\pi.l} \cdot \frac{(1/1) + (1/1)}{[1 - 0.28^2/30 \times 10^6] + [1 - 0.28^2/30 \times 10^6]}}$$

$$\sigma_{Hertz} = 1,61,087.99 \text{ Psi}$$

Contact stresses obtained from FE Analysis are given by

$$\sigma_{FEA} = 1,60,110 \text{ Psi}$$

III: Validation of FEA Results

Stress	Theoretical values	FE Results	%Error
Bending Stress	6181.57 psi	6239.4 psi	0.9%
Contact Stress	1,61,087.99 psi	1,60,110 psi	0.6%

CHAPTER III

USE OF SOLID WORKS AND ANSYS SOFTWARE FOR BENDING STRESS ANALYSIS OF GEAR TOOTH

There are several theoretical procedures to measure the bending stress on single gear tooth but for an assembly, bending stress is under the influence of many factors. AGMA derived some standards to determine these factors which are explained in chapter II. In this chapter a finite element procedure is used to determine the bending stress and is compared with standard AGMA procedure to determine the bending stress.

A virtual assembly is created with the 3-D parts using CAD software and finite element analysis is performed on the assembly to determine the bending stress on the gear tooth. The following sections describe the procedure to perform the finite element analysis.

3.1 Assembly Specifications

3.1.1 Bearing Specifications

Bore: 0.75

Outer Diameter: 1.6250

Thickness: 0.3125

Number of Balls: 10

3.1.2 Bearing Holder Specifications

Bearing holder is designed based on bearing specifications. The diameter of the outer race of the bearing is equal to the inner diameter of the bearing holder and thickness of the bearing holder is equal to thickness of bearing.

3.1.3 Spur Gear Specifications

Number of Teeth   N: 36

Diametrical Pitch P_d : 10

Pressure Angle φ: 20

Face Width   b: 0.4

3.2 Solidworks to Generate Geometry

The geometry of the required parts is developed using Solidworks except bearings. Bearings are taken from Solidworks Toolbox.

An assembly is developed using the above parts. These parts are linked together using the coincident, concentric and gear mates. The gear ratio is defined as the gear mate constraint. The following image displays the exploded view of the assembly.

10: Exploded View

3.3 Modeling of Assembly in Ansys

Assembly modeling in FE analysis is complex and time taking. Accuracy of the FE solution mainly depends on the Pre-processing.

3.1.1 Pre – Processing

The main Pre-processing steps include

1. Engineering Data

2. Geometry

3. Discretization

Engineering Data

Grey Cast Iron is used for this Analysis. The main material properties like the Young's modulus, Poisson's Ratio and Density that are required to perform a static analysis are defined in this section.

Geometry

The 3-Dimensional part generated in Solidworks is imported in to Ansys through Design Modeler. The geometry is then configured by adding the contacts and joints to the imported model. The current assembly is configured with two different contacts and two different joints.

The BONDED contact is used in between the bearing holders and the bearings since the movement of the outer race of bearing is restricted and for other contacts NO SEPARATION is used since the bodies have movements with respect to their axis. Pure Penalty formulation is used to solve the contacts. The following images display the location of the BONDED and the NO SEPARATION contacts. The following figure displays the location of Bonded Contact.

11: Bonded Contacts
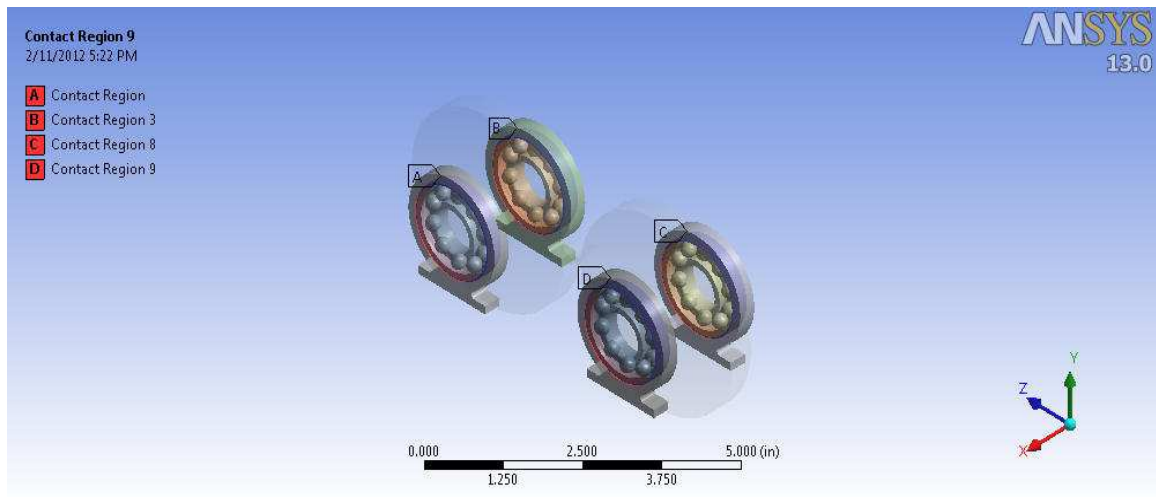
The location of the No-separation contact is displayed in the following figure

12: No Separation Contact Location
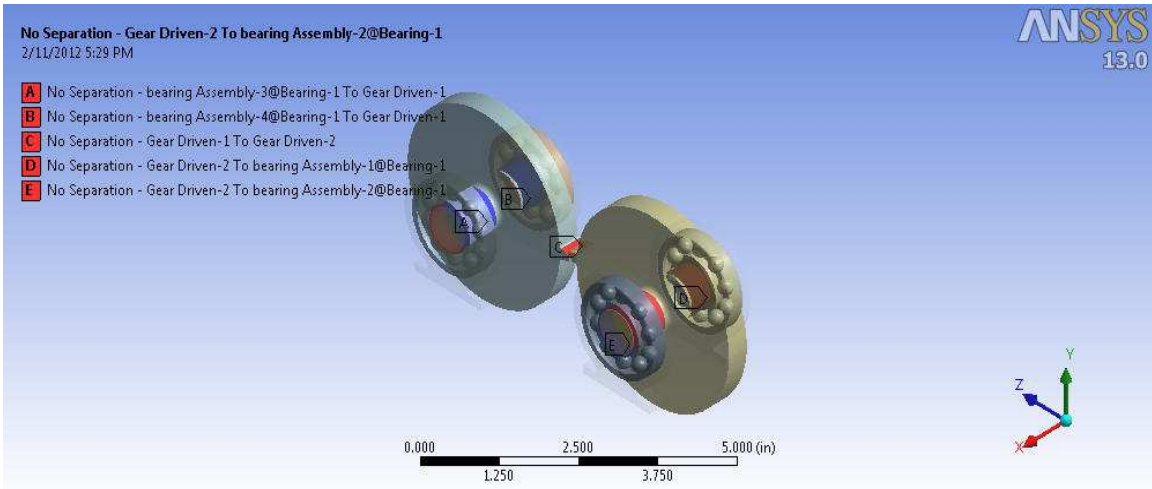
Joints are used to constrain the DEGREES OF FREEDOM of parts in an assembly. For the current analysis shaft should only have the rotational movement around the X-axis so a revolute joint must be used. The bearing holder should be fixed so a fixed joint is used. The following image displays the location of fixed joints.

13: Location of Fixed Joint

28
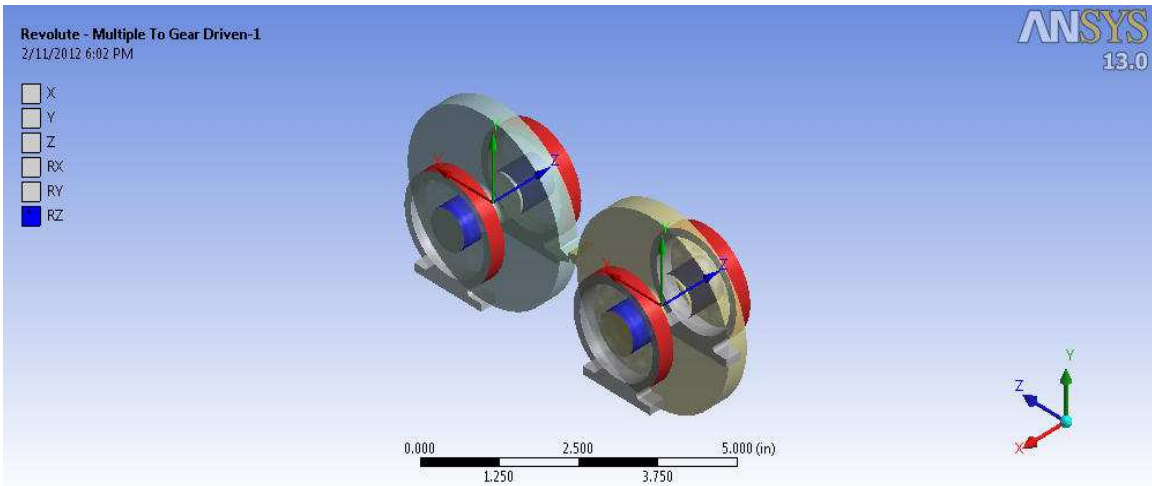
The following image displays the location of revolute joint.

14: Location of Revolute Joint
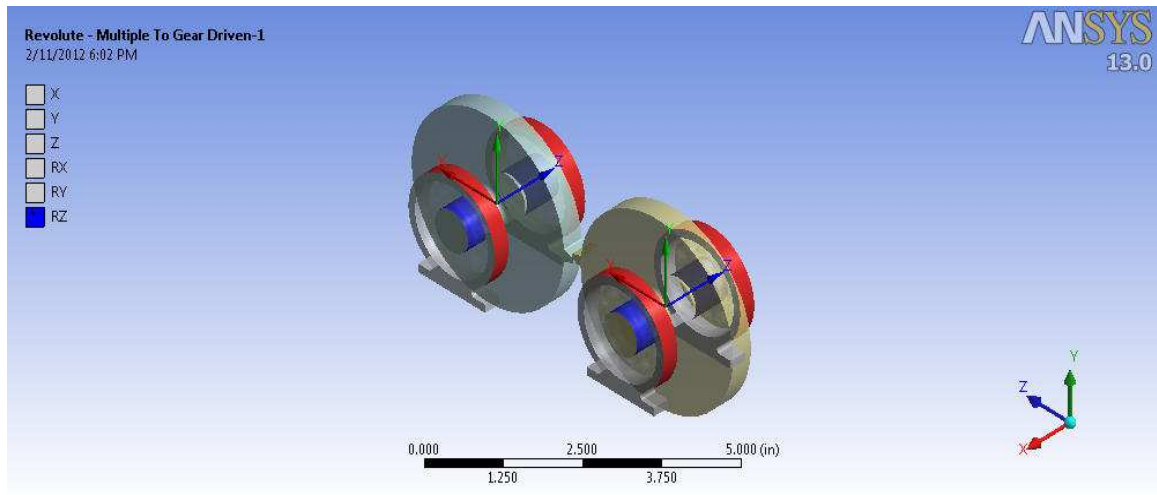
Discretization

The accuracy of results in an assembly depends mainly on the meshing methods. For the current analysis a part should be meshed independent so the initial seed size is set to part and meshing size should be set to fine with smoothing set to medium and initial span angle set to medium. Computation for the analysis goes complex as the smoothing and initial span angle are set to fine. The following image shows the generated mesh view

15: Meshing
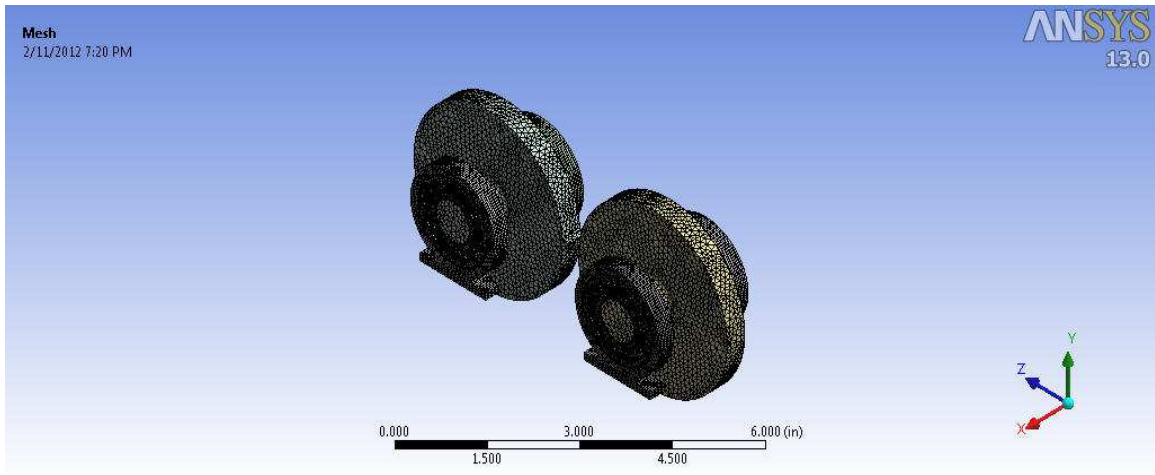
Mesh Controls should be included to obtain more refined solution. For the current assembly tetrahedral elements with patch independent algorithm is used since the algorithm ensures the refined mesh where ever necessary but maintains larger elements where possible allowing faster computation. CONTA 174 and TARGE 170 type contact elements are used in this study. CONTA 174 represents sliding contacts in between 3 – D target and deformable surfaces. It has the same geometric characteristics as solid and shell elements face which is connected. The following images displays element view of the model

16: Element View
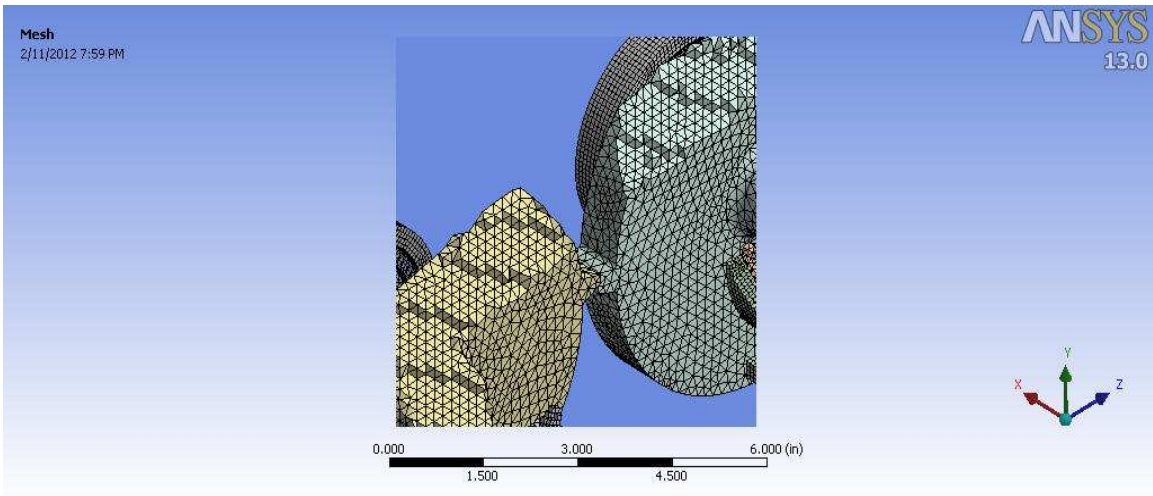
Contact sizing should be included since it creates elements of relatively same size on the bodies from faces of a face to face contact region. This control generates a sphere of influence with automatic determination of shape and size. Element size is set back to 0.1in. The following image displays the element shapes and the sphere of Influence.

17: Contact Sizing Location
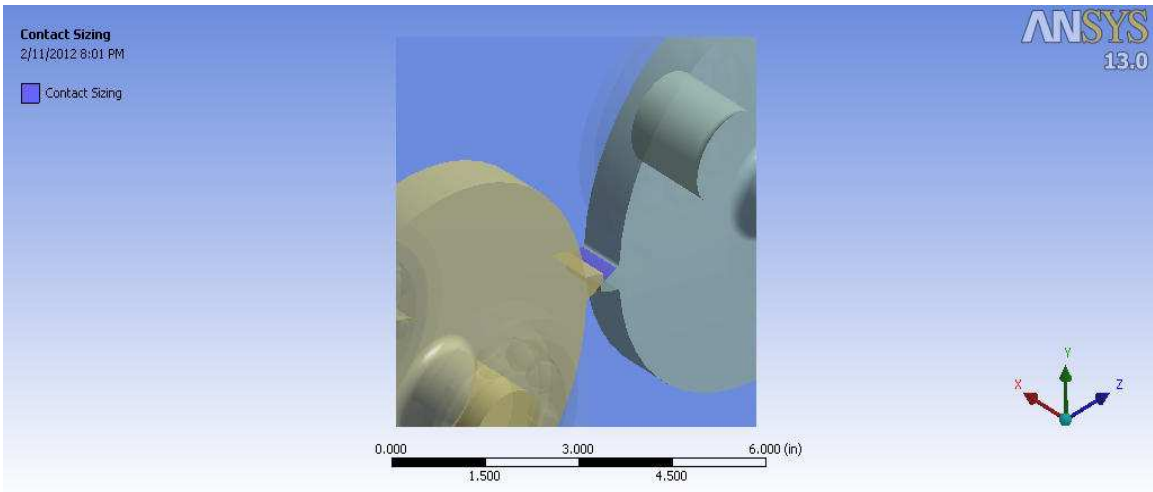
3.1.2 Solution

Static structural analysis determines the stress, strain and displacements in a body that are caused due to loads. In this analysis steady loading is used. Ansys supports the following types of loading for the static analysis

1. External loads

2. Steady state inertial forces

3. Imposed displacement's

4. Temperatures

Iterative solver is best used with the thick bulky structures and weak spring's option set to on position and their spring stiffness is set to program controlled. Since no hyper elastic materials are used, Large Deflection option and Inertia relief is turned off.

Two loads are now added to the assembly in the form of moments of magnitude 178.52 lbf-in. These two moments are added in such a way that one moment opposes the moment of another. The following image displays the location of the moments.

18: Location of Loads
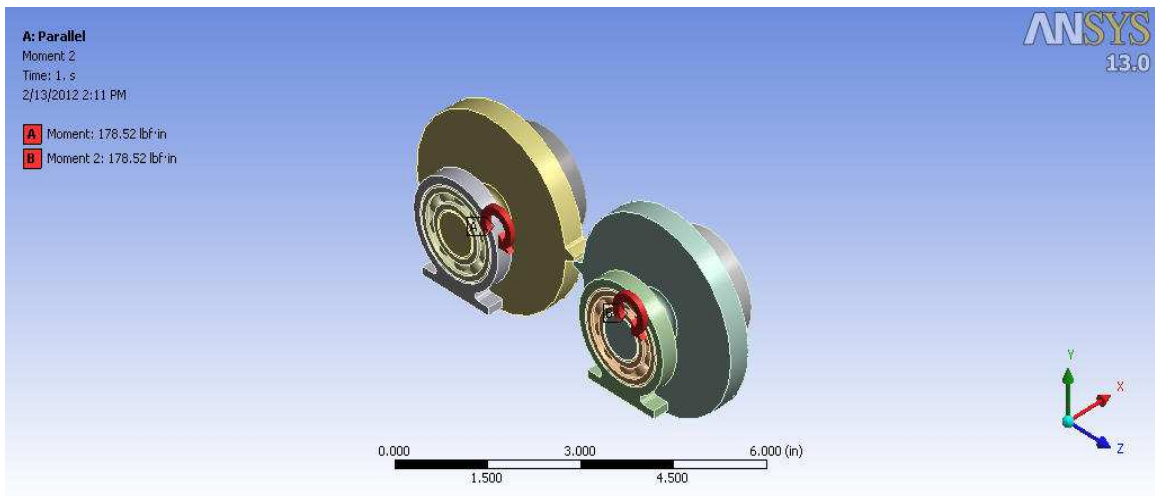
On performing the simulations solver generates data files which are to be processed to determine the results. The bending stress to be calculated can be obtained from the normal stresses along the Z- axis direction. On evaluating the results the Maximum Bending stresses is obtained as 6181.7 Psi. The following image displays the contour of the distributed bending stresses.

19: Stress Distribution along Z-axis
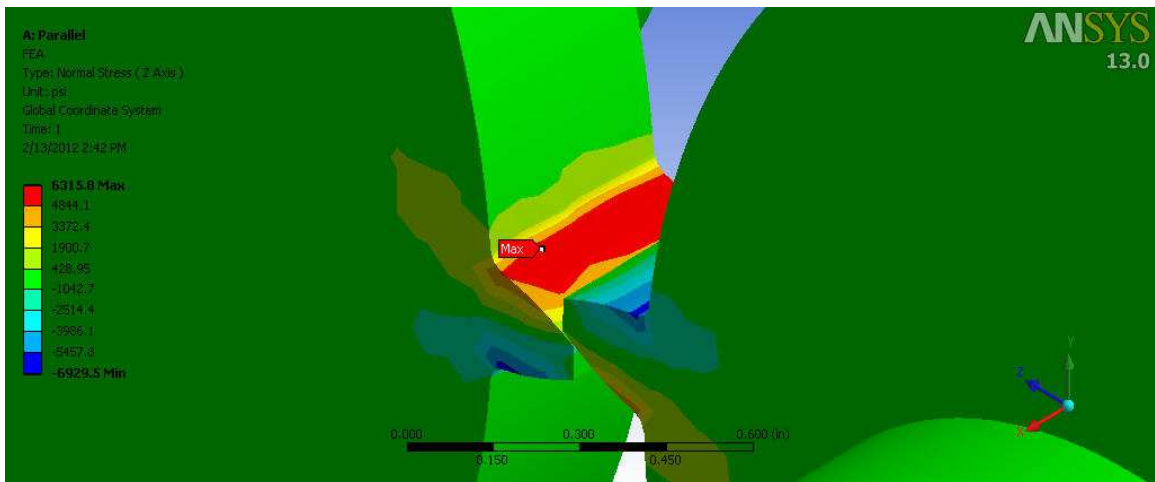
3.4 Validation of FEA Bending Stress with AGMA Bending Stress

The FEA bending stress generated can be validated by comparing them with AGMA bending stress. The following are the values are used for the AGMA bending stresses. These values are substituted in the AGMA equation from equation (2.4)

3.4.1 AGMA Factors

Tangential Force w^t: 93.96 lbf.

Overload Factor k_0: 1

Dynamic Factor k_v: 1

Size Factor k_s: 1

Rim thickness Factor k_m: 1

Reliability Factor k_R: 1

Geometric Factor   J: 0.38

3.4.2 Gear Specifications

Number of Teeth   N: 36

Diametrical Pitch P_d : 10

Pressure Angle φ: 20

Face Width   b: 0.4

Now from the above factors and gear specifications, AGMA bending stress from equation (2.4) is obtained as

$$\sigma_{AGMA} = 93.96 \times 1 \times 1 \times 1 \times 1 \times \frac{10}{0.4} \times \frac{1 \times 1}{0.38}$$

$$\sigma_{AGMA} = 6181.57 \text{ psi}$$

FEA generated Bending stress gives the following values

$$\sigma_{FEA} = 6315.8 \text{ psi}$$

On comparing both the values the FEA generated results deviate at 2% from the AGMA

Bending stress.

CHAPTER IV

USE OF SOLID WORKS AND ANSYS SOFTWARE FOR CONTACT STRESS ANALYSIS OF GEAR TOOTH

There is no theoretical procedure to determine contact stress in between a pair of involute spur gears. AGMA has derived a near approximate procedure to determine the contact stresses by comparing them with Hertzian contact stress of two cylinders with parallel axis. In this chapter the contact stress in-between two involute gears is determined using a Finite Element Method.

The assembly geometry required for the finite element method is developed using solidworks. The assembly consists of bearings, bearing holders and involute spur gear. The following section describes the specification of the parts.

4.1 Assembly Specifications

4.1.1 Bearing Specifications

Bore: 0.75

Outer Diameter: 1.6250

Thickness: 0.3125

Number of Balls: 10

4.1.2 Bearing Specification

Bearing holder is designed based on bearing specifications. The diameter of the outer race of the bearing is equal to the inner diameter of the bearing holder and thickness of the bearing holder is equal to thickness of bearing.

4.1.3 Spur Gear Specification

Number of Teeth   N: 36

Diametrical Pitch P_d : 10

Pressure Angle   φ: 20

Face Width   b: 0.4

4.2 Solidworks to Generate Geometry

The assembly required to perform the FE analysis is made in Solidworks. Bearing holder and spur gear are developed as part files and are assembled. Constraints for the assembly are same as once in chapter II. The following image displays the exploded view of the assembly

20: Exploded view

4.3 Modeling of assembly in Ansys

Pre-processing phase for the analysis is critical since it requires more memory resources and tools. Pre-processing phase involves the following steps

4.3.1 Pre- Processing

The main steps involved in Pre-Processing are

- Engineering Data

- Geometry

- Discretization

Engineering Data

For the present assembly we use an alloy with Young's modulus of 30E6 psi and Poisson's ratio of 0.28.

Geometry

      Assembly developed in Solidworks is imported into Ansys through Design Modeler and parts are constrained using joints. For present analysis a revolute joint is used in between bearing holder and shaft, as it constrains all the motions of the shaft except the rotational DOF along the Z-axis. A fixed constrain is used for bearing holder since it has to be fixed rigidly to the base. The following images displays the location of the Revolute and fixed joints

21: Location of Revolute Joint
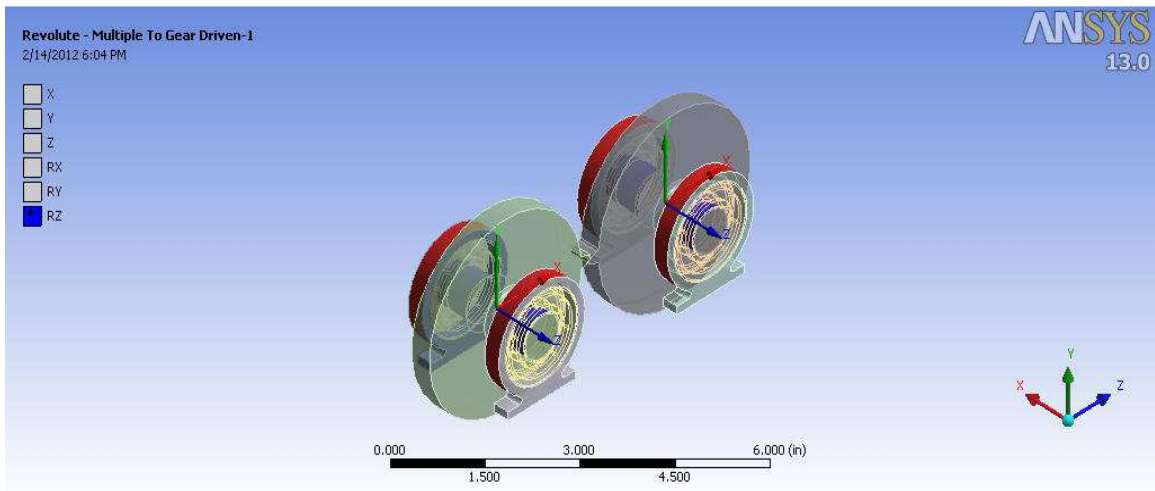
22: Location of Fixed Joint
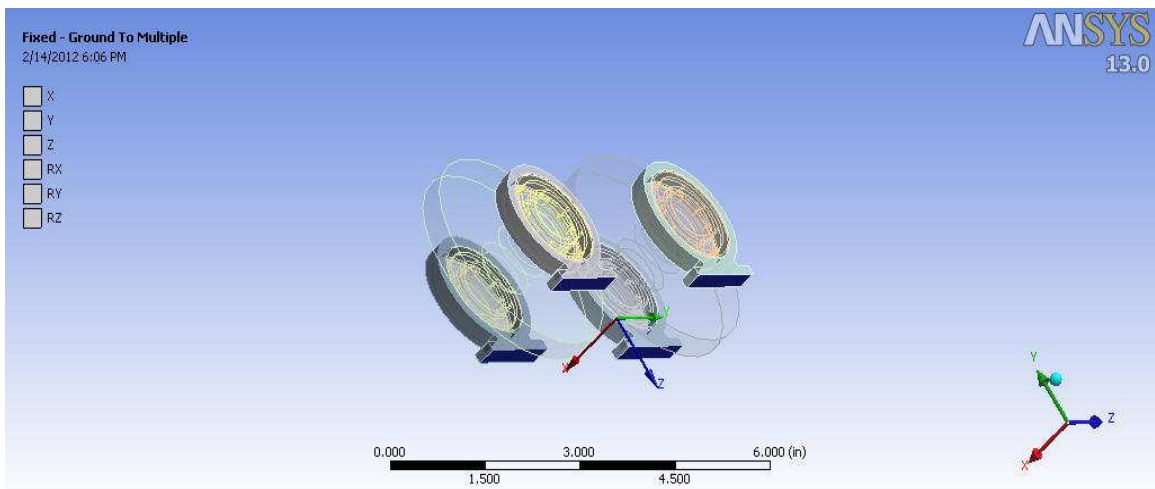
Contacts define the nature of contact in between the surfaces. For the current analysis Bonded and No-separation contacts are used. The movement in-between the outer race of the bearing and the inner surface of the bearing holder is restricted so bonded contact is used, and for rest of contacts separation in-between surfaces is restricted so No-Separation contact is used. The following images show the location of Bonded and No-separate contacts.

23: Bonded Contact
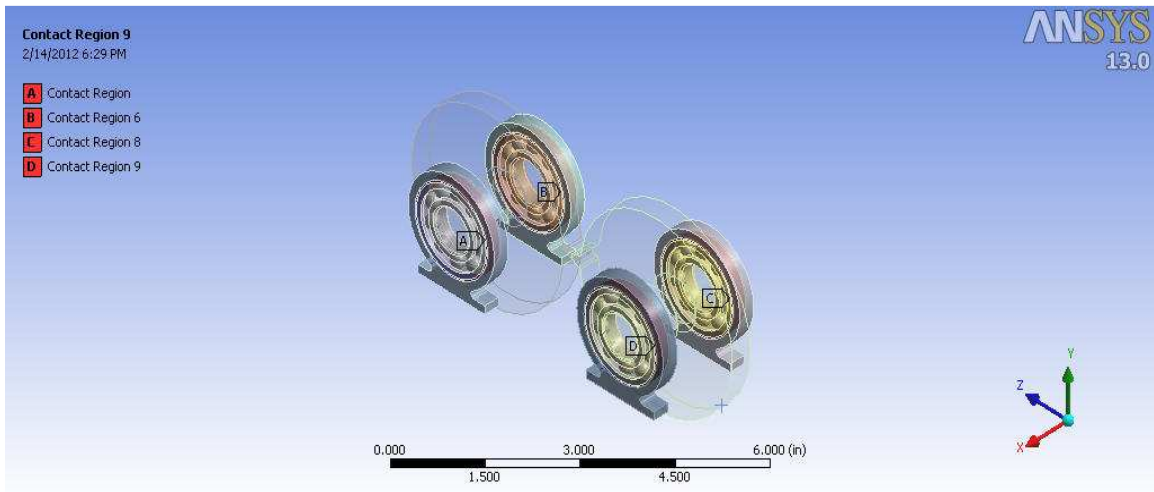
24: Location of No-Separation Contact
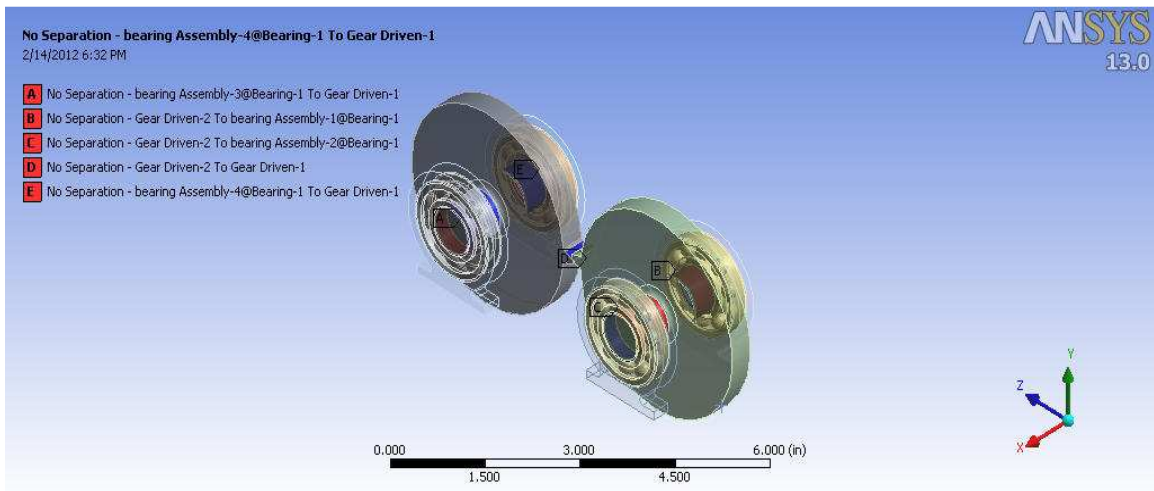
Discretization

Meshing for contact analysis is complex and requires more refined meshing tools for accurate solution. For the current analysis a fine mesh is used with smoothing and initial span angle kept medium. Initial seed size should be kept part since we need the mesh to be distinct and separate from each part. The following image displays the generated mesh

25: Fine Mesh
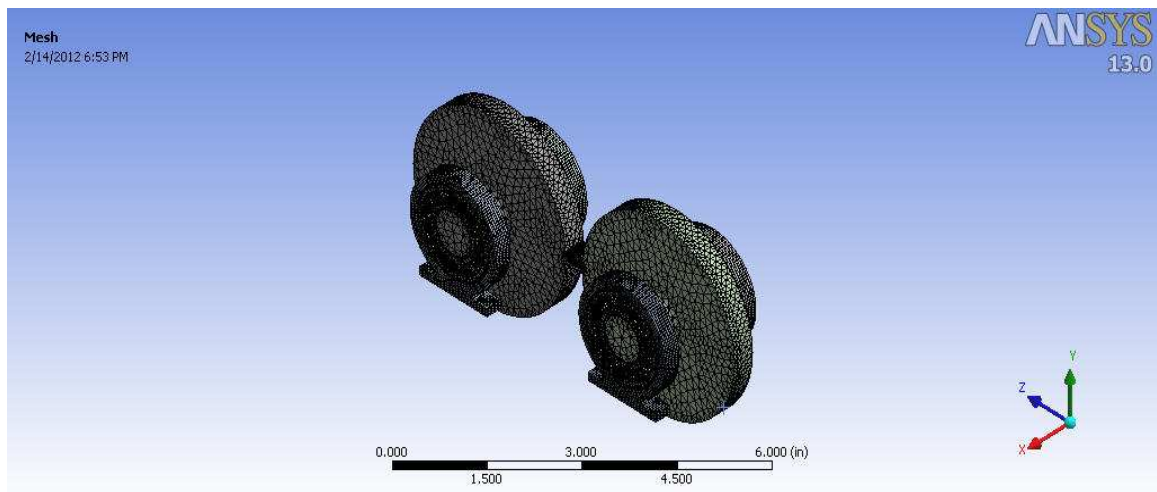
Tetrahedral elements are used as meshing elements with patch independent algorithm. This algorithm ensures refined mesh where ever necessary and maintains larger elements where possible to reduce computational time. The following image displays the mesh elements.

CONTA 174 [11] and TARGE 170 type contact elements are used in this study. CONTA 174 represents sliding contacts in between 3 – D target and deformable surfaces. It has the same geometric characteristics as solid and shell elements face which are connected.

41

26: Tetrahedral Mesh Elements
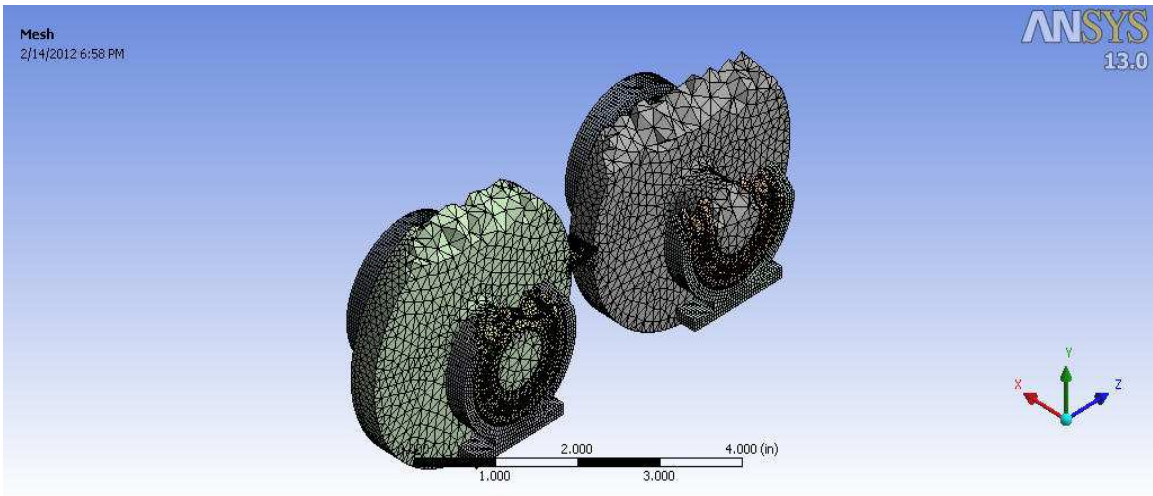
Contact sizing tool is used to ensure that all the generated elements around the tooth contact are 0.003 in. Refinement control is set to 1 to obtain minimum refinement. Following image displays the refinement of mesh near gear tooth contact.

27: Mesh Refinement
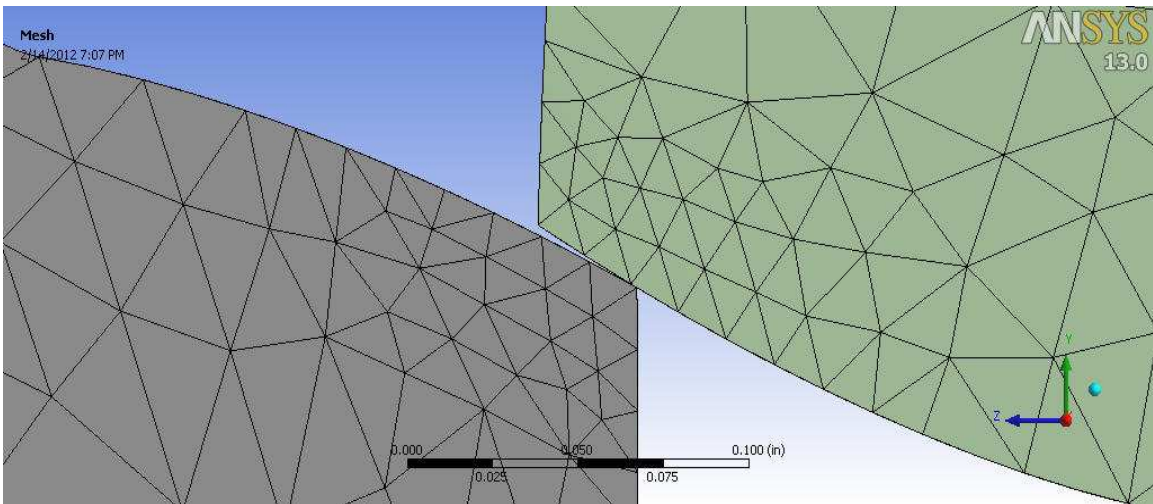
4.1.2 Solution

Static structural analysis determines the stress, strain and displacements in a body that are caused due to loads. In this analysis steady loading is used. Ansys supports the following types of loading for the static analysis

5. External loads

6. Steady state inertial forces

7. Imposed displacement's

8. Temperatures

Iterative solver is best used with the thick bulky structures and weak spring's option set to on position and their spring stiffness is set to program controlled. Since no hyper elastic materials are used, Large Deflection option and Inertia relief is turned off.

Loads are now added to the assembly parts. For the current assembly a moment of magnitude 178.52 lbf.in is applied on the shaft and counter moment of magnitude 178.52 is applied to the other shaft. The following image shows the location of the moments applied.

28: Location of Moment application
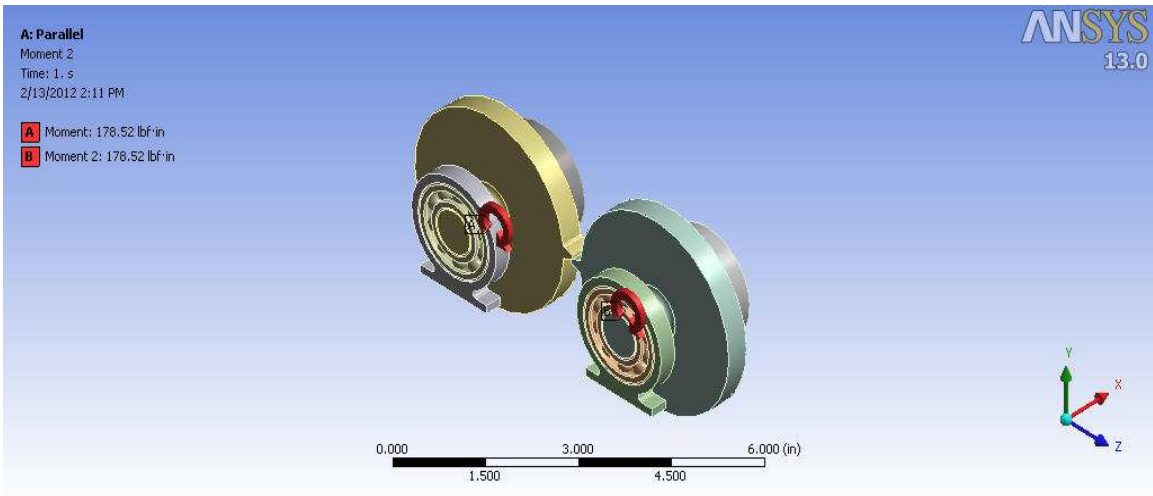
4.1.3 Post -Processing

Contact pressure is measured using a pressure tool. The nodal difference values of the pressure are taken from the solution. The maximum contact pressure is obtained as 1.6751E5 Psi. The following image displays the contours of the pressure distribution along the face width.

29: Maximum Contact Pressure

44
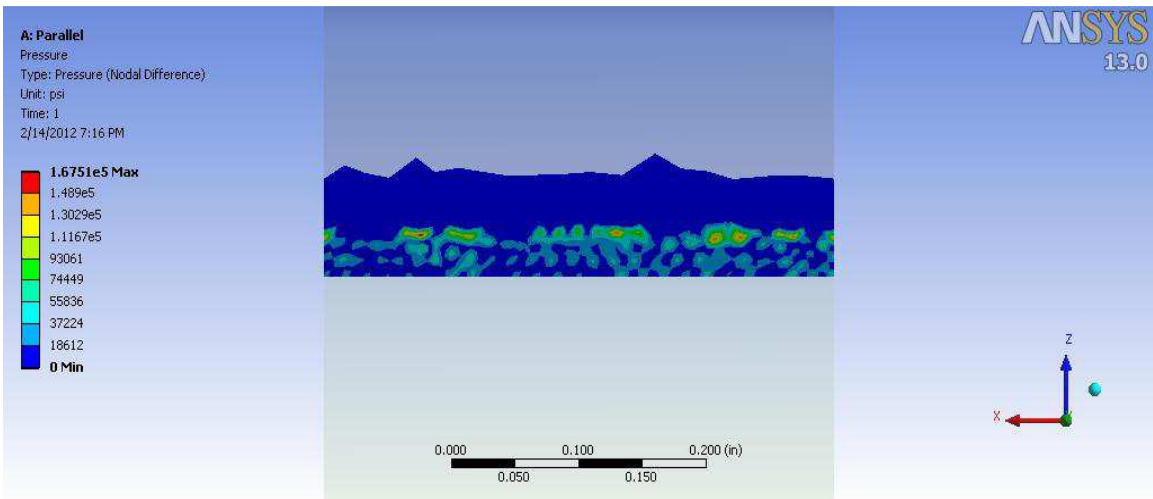
CHAPTER V

INFLUENCE OF SHAFT MISALIGNMENT ON GEAR TOOTH

The most common problem in the rotating mechanisms is shaft misalignments. Misalignment in the meshing gear results in uneven load distribution resulting in shift of peak bending stress to the edge of face width. Misalignment in meshing alters the location of contact on tooth flank and may lead to large stresses and increase noise of gear pair. These Mesh misalignments can be classified in to three types

1. Parallel Misalignment
2. Angular Misalignment parallel to the plain of action
3. Angular Misalignment perpendicular to the plain of action

5.1 Parallel Misalignment

Parallel Misalignment results in change in central distance of shafts. A change in center distance will result in a slight change in the intersection of the outside diameters with the plane, thus slightly altering the profile contact ratio of the gear pair.

5.2 Angular Misalignment Parallel to the Plain of Action

This type of misalignment tends to shift the load to the side of the tooth by increasing the separation at one side of the tooth and reducing the separation at the other side of the tooth. In this case, the shape and area of the theoretical active contact plane remains the same as the ideal shape.

5.3 Angular Misalignment perpendicular to the plain of action

For the angular misalignment perpendicular to plane of action the outside diameters of the respective gears rotate as in such a way that the shape of the contact zone becomes skewed and the contact area is reduced. This reduction in contact area results in a reduction of total contact ratio.

In the present chapter the variation of the bending stress with various angular misalignments parallel to the plain of action is studied. At first FE analysis for maximum bending stress is performed for parallel shafts and compared it to the maximum bending stress for 1^0, 2^0 and 3^0.

The study shows that the maximum bending stress increases with the increase in angular misalignment and the load concentration is more on the edge of the gear tooth as the angle increases. The following table displays the maximum bending stress for various misalignments.

IV: Maximum Bending Stress for various angular alignments

Angular Alignment	Maximum Bending Stress
Parallel	6315.8 psi
1^0	6550 psi
2^0	7002.3 psi
3^0	7214.8bpsi

5.4 Calculation of AGMA Stress Distribution Factor

The load distribution factor can be defined or can be calculated from the empirical method of AGMA 2001/2101. Empirical method is recommended for normal, relatively stiff gear designs which is defined as

$$K_m = 1.0 + C_{pf} + C_{ma} \qquad (5.1)$$

Where C_{pf} is Pinion Proportion factor and C_{ma} is Mesh alignment factor. In the present study the load distribution factor for parallel gear is defined as one and the stress distribution factor for the various angular alignments can be calculated as

$$\sigma_{FEA} = K_m \times \sigma_{AGMA} \times X \qquad (5.2)$$

$$K_m = \frac{\sigma_{FEA}}{\sigma_{AGMA} \cdot X} \qquad (5.3)$$

Where X is multiplication factor and K_m is load distribution factor. The following table displays the obtained load distribution factors for various angular misalignments.

5: Table displaying Load Distribution factors for various Alignments

Alignment	Load Distribution Factor
1^0	1.03
2^0	1.11
3^0	1.14

REFERENCES

1. http://nptel.iitm.ac.in/courses/IIT-MADRAS/Machine_Design_II/pdf/2_1.pdf

2. NASA Reference Publication 1152, AVS COM technical report 84-C-15

3. Design of Machine Elements by C.S Sharma and Kamlesh Purohit

4. Shigley's Mechanical Engineering Design by Budynas–Nisbett

5. Finite element Method – its basics and fundamentals by Zienkiewicz, O.C.; Taylor, R.L.; Zhu, J.Z.

6. Finite Element Analysis by S.S Bhavikatti

7. History of Rotating Dynamics by J.S Rao

8. Ansys Workbench User's Guide

9. Ansys Meshing User's Guide

10. Ansys Mechanical APDL and Mechanical Applications Theory Reference

11. Ansys Mechanical APDL Help

APPENDIX

APPENDIX I

GENERATION OF INVOLUTE GEAR

Step 1: Create a new part in a part file
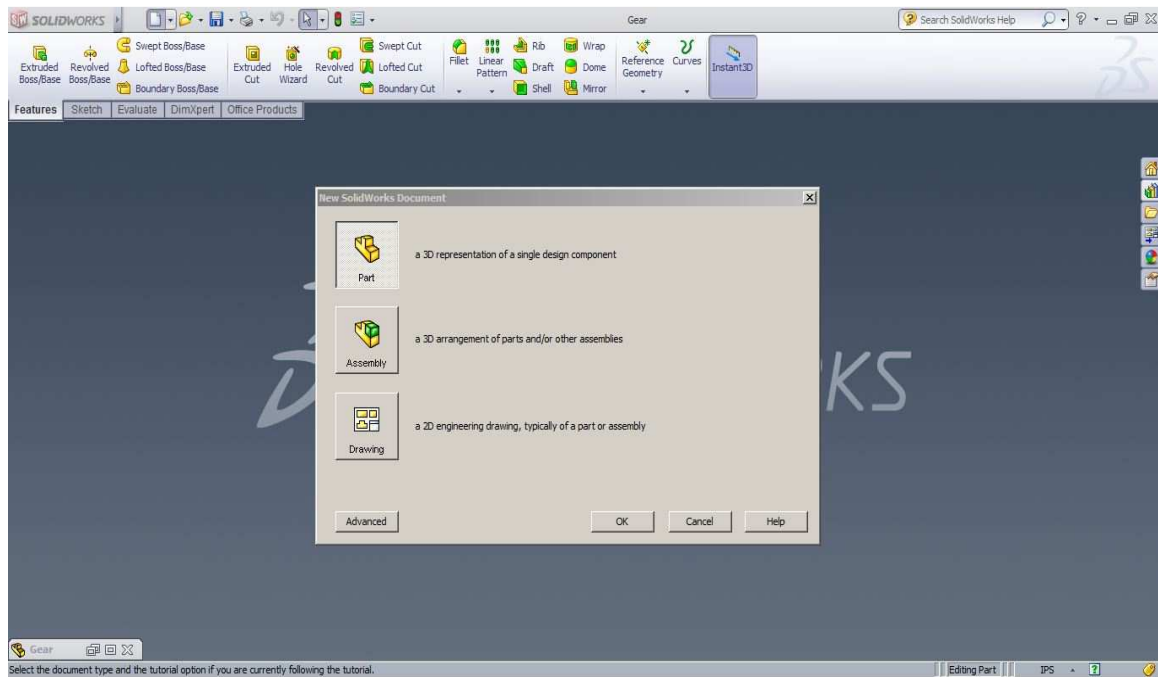
Step 2: Write the Gear Nomenclature as Expressions in the Expressions tool bar

"N"= 36 '   Number Of Teeth

"Pd"= 10 '   Diametrical Pitch

"Pc"= "N"/"Pd" '   Pitch Circle Diameter

"Phi"= 20 '   Pressure Angle

"A"= 1/"Pd" '   Addendum

"D"= 1.157/"Pd" '   Dedendum

"Od"= ("N"+2)/"Pd" '   Outer Circle Diameter

"Bd"= "Pc"/cos("Phi") '   Base Circle Diameter

"Id"= ("N"-2)/"Pd" '   Internal Diameter

"b"= 0.4 '   Face Width
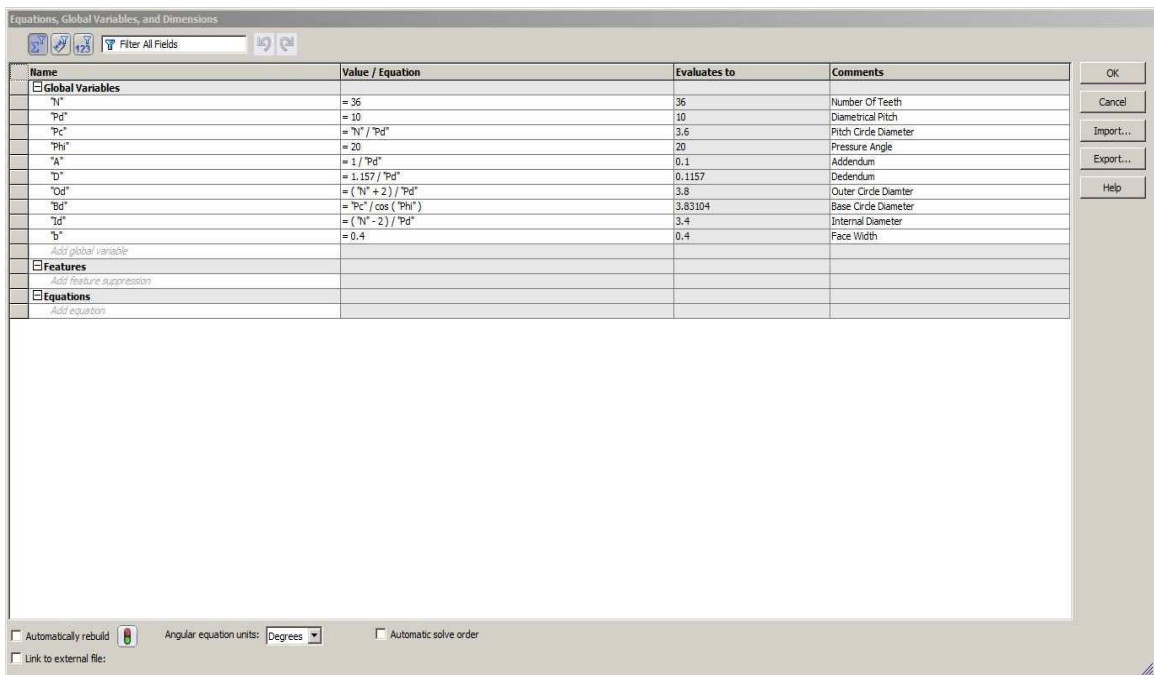
Step 3: Select the front plane and start the sketch
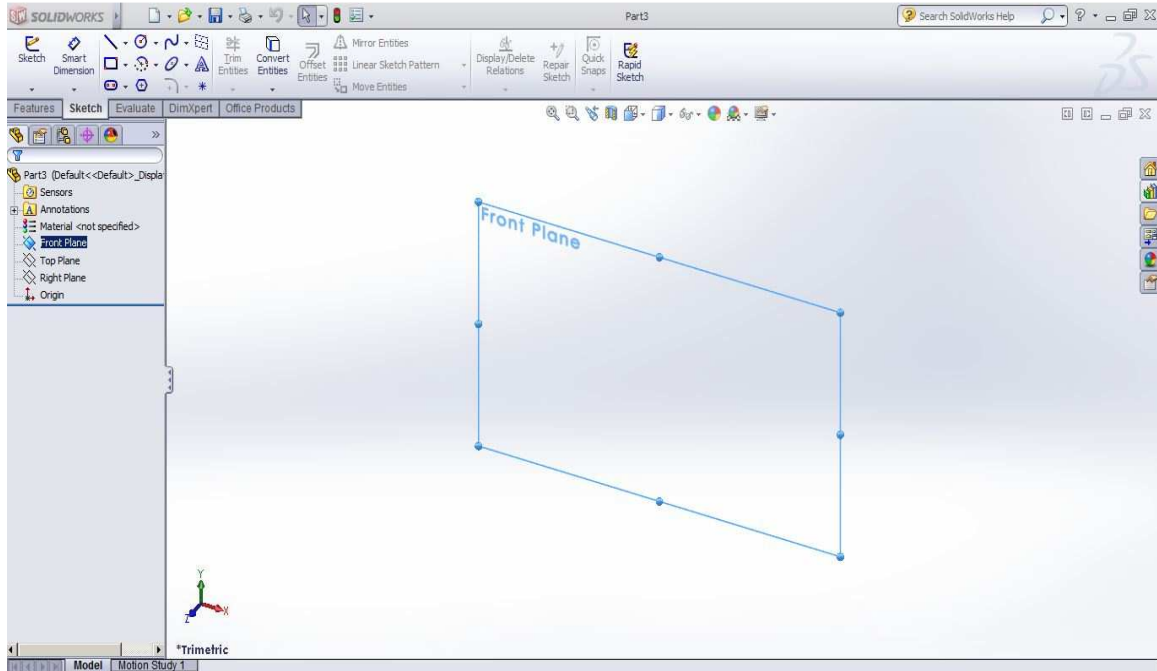
Step 4: Now using the circle tool draw three circles with Origin as center. Link the dimensions of circles with the Pitch Circle Diameter, Base Circle Diameter and Outer Circle Diameter expressions in the Smart Dimension Tool

Step 5: Draw two lines symmetric about the horizontal axis line which makes an angle of 5°

Step 6: In order to get an involute in the sketching plane we need to use the Equation Driven Tool. In the Equation driven curve tool select the parametric type and in the equation column enter the parametric equation of the involute curve. Parameter vary from 0.17807282 to 0.50455305

$$X_t = 0.7 * (\cos(t) + t * \sin(t))$$

$$Y_t = 1.7 * (\sin(t) - t * \cos(t))$$

Step 7: Mirror this line along the horizontal axis and close it using the circular Arc tool.
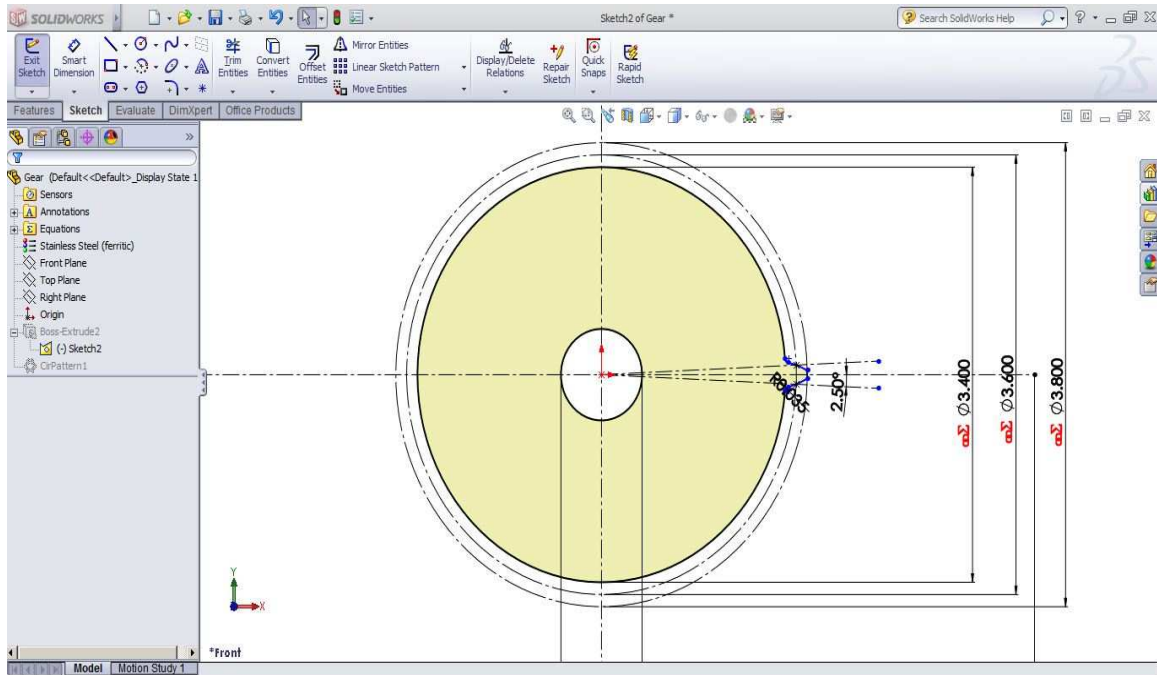
Step 8: Exit Sketcher mode and in the feature tab select the extrude tool. The following pop-out window appears. Now in the Direction 1 menu from fly over menu, select mid plane and in the dimension menu enter the face width value to be 0.4 in.
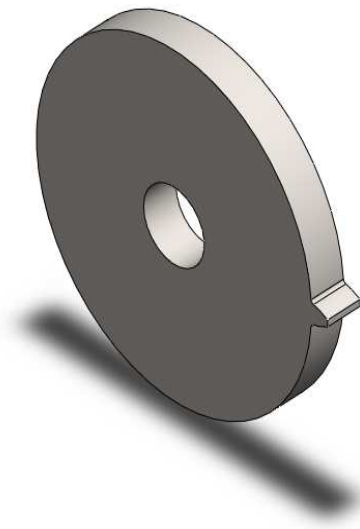
Step 9: Pattern the extruded tooth with 36 instances since the gear has 36 tooth
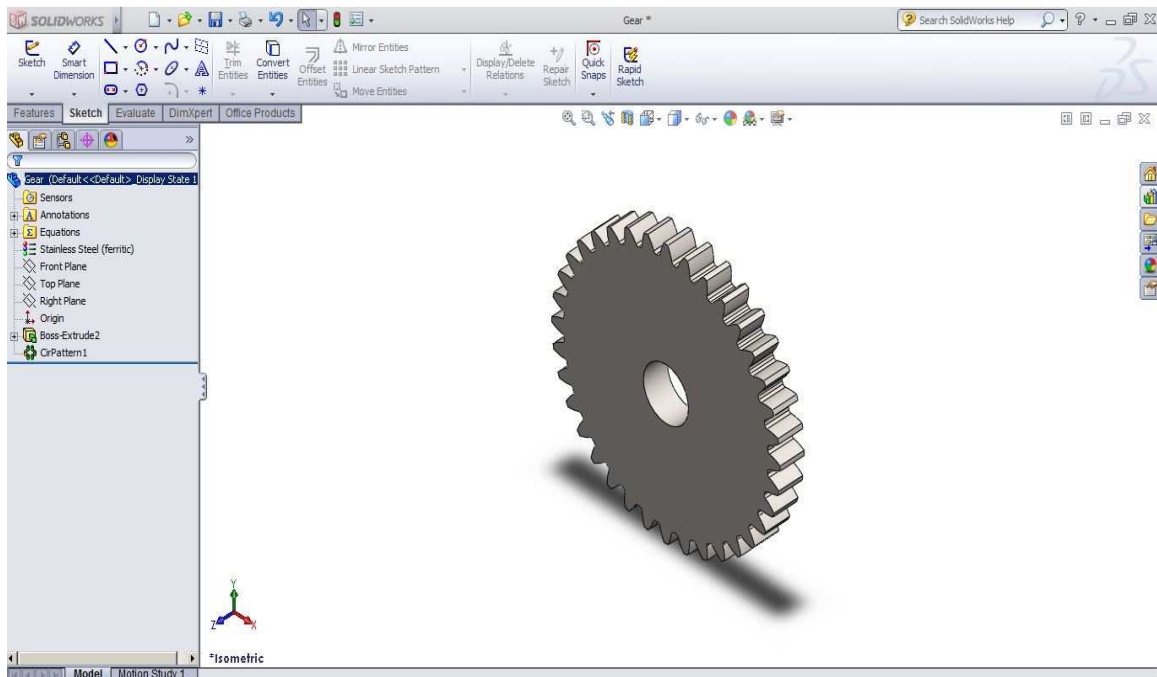
APPENDIX II

GEAR TOOTH MODELING

Step 1: Create a New Work Bench Project and select the Static Structural Analysis
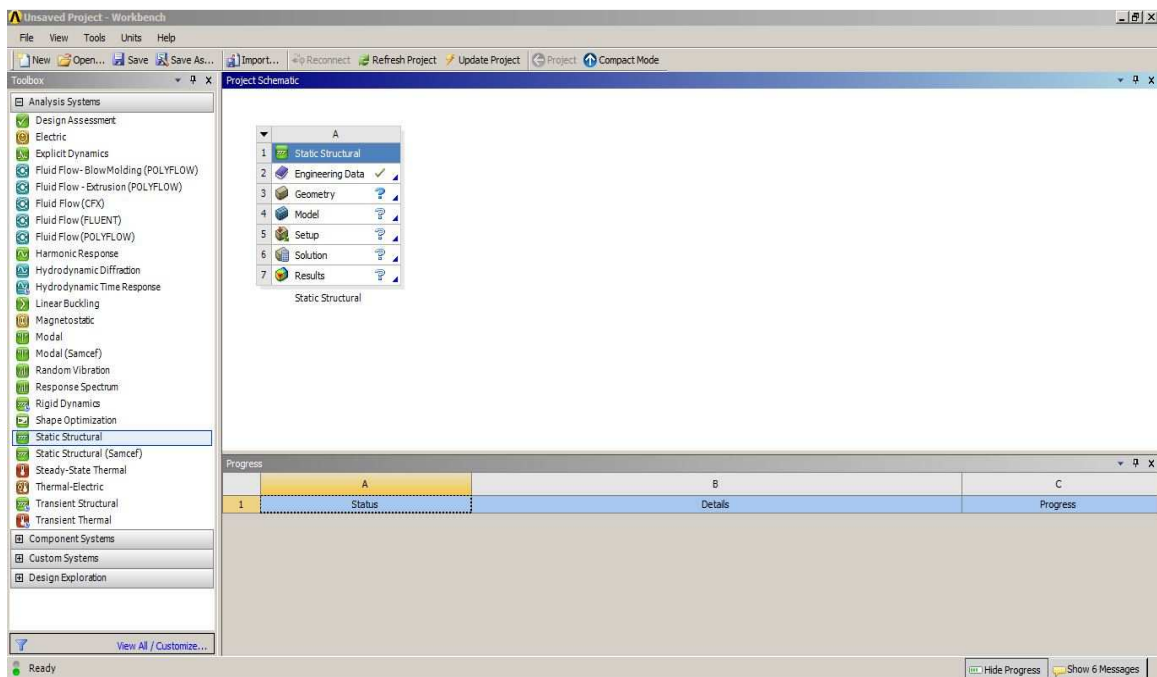
Step 2: Select the Engineering data, create a new material and name it as Stainless Steel
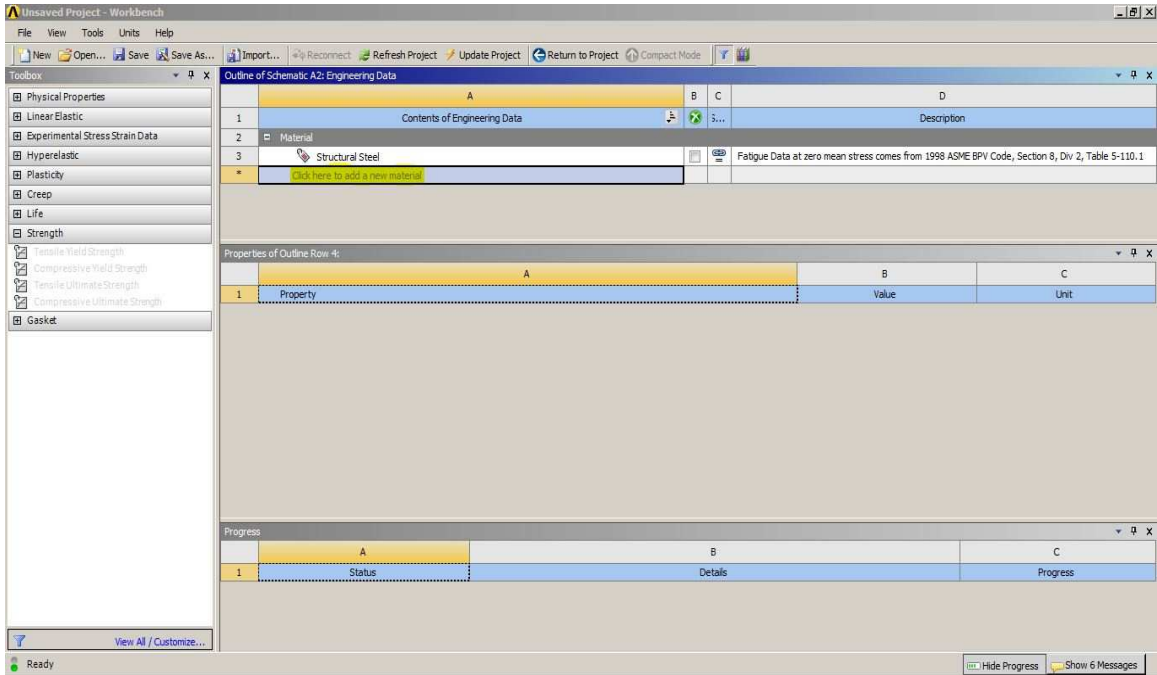
Step 3: Expand the physical properties and add the Density properties to the material
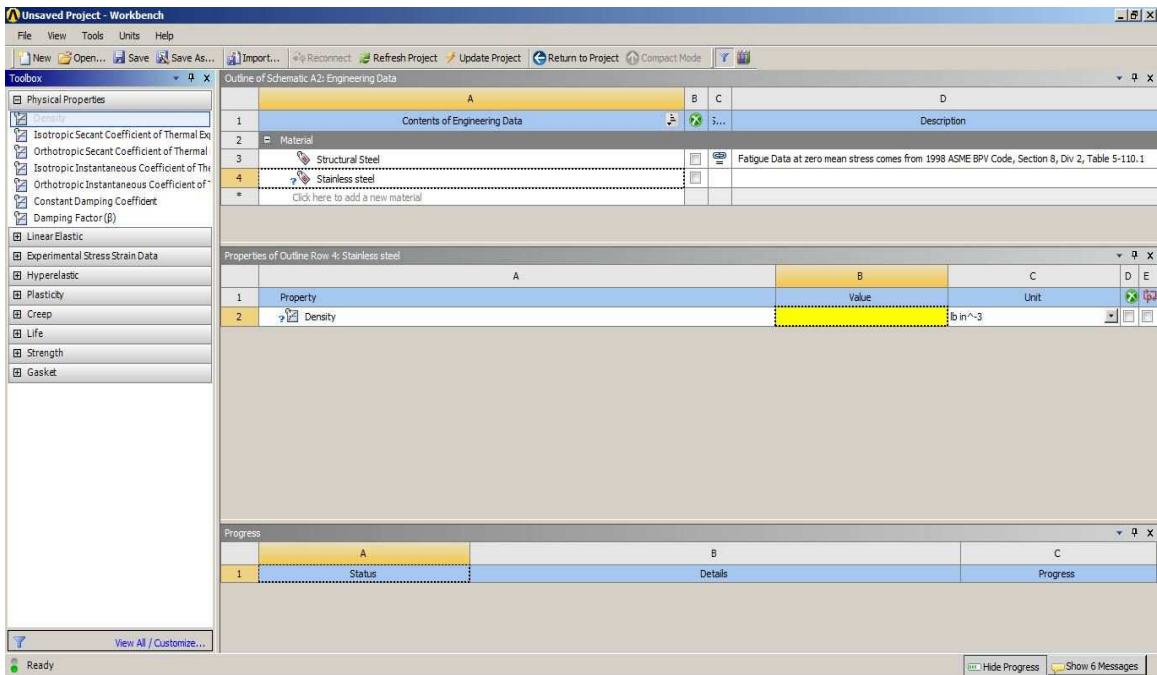
Step 4: Similarly Select the Young's Modulus and Poisson's ratio from the Linear Elastic

   Properties
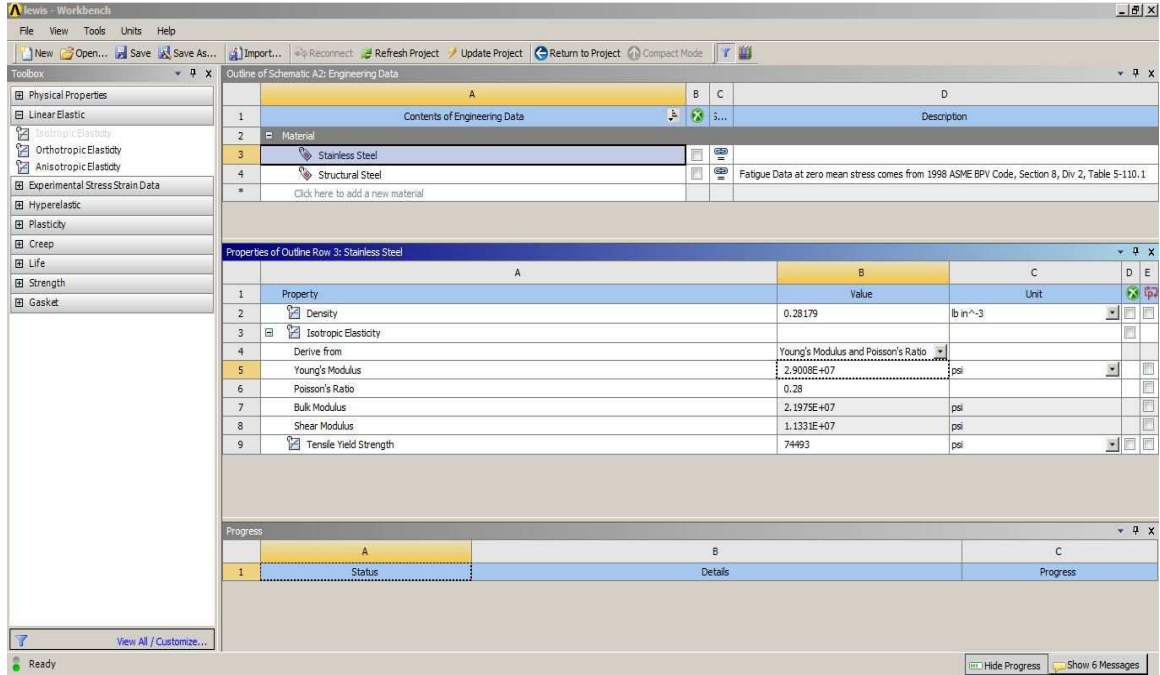
Step 5: Return to project.

Step 6: Select the Design Modeler from workbench home screen
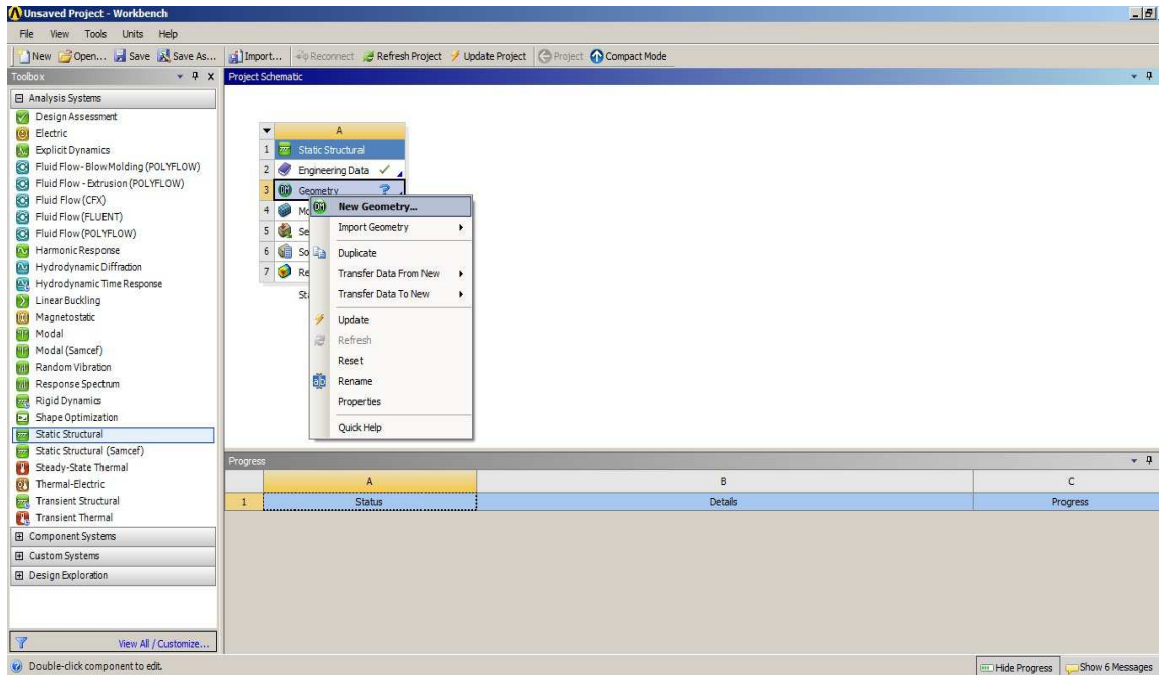
Step 7: In the File menu select the Import External Geometry option
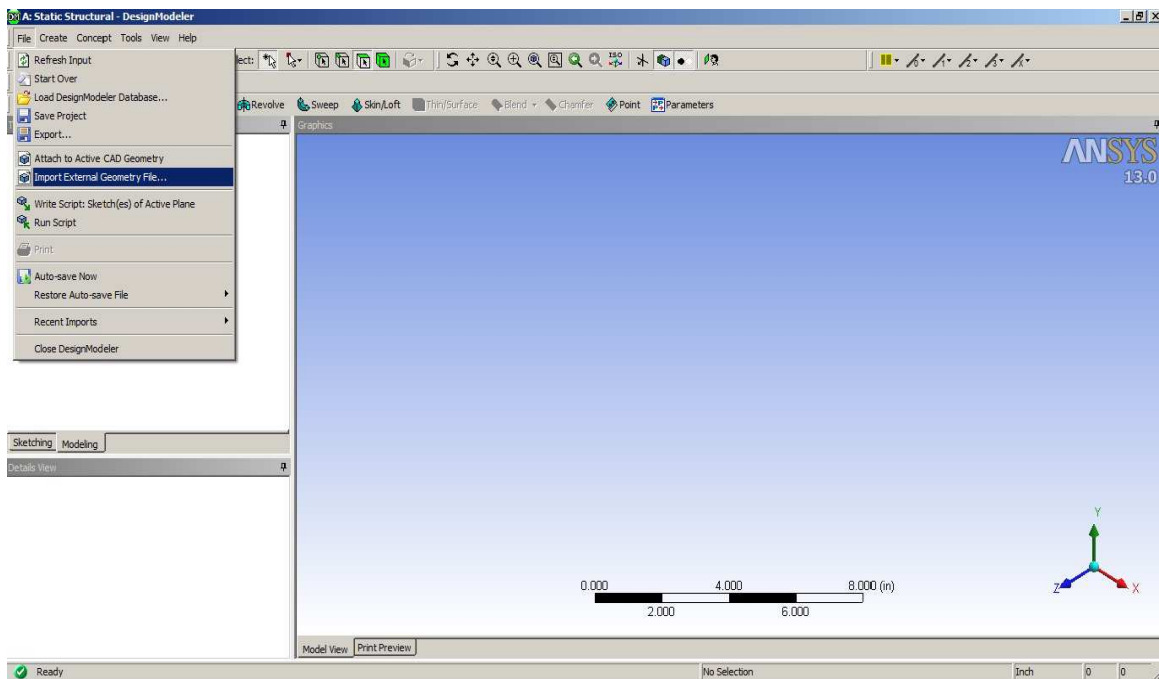
Step 8: After selecting the option now hit generate button. The final Geometry is

   generated
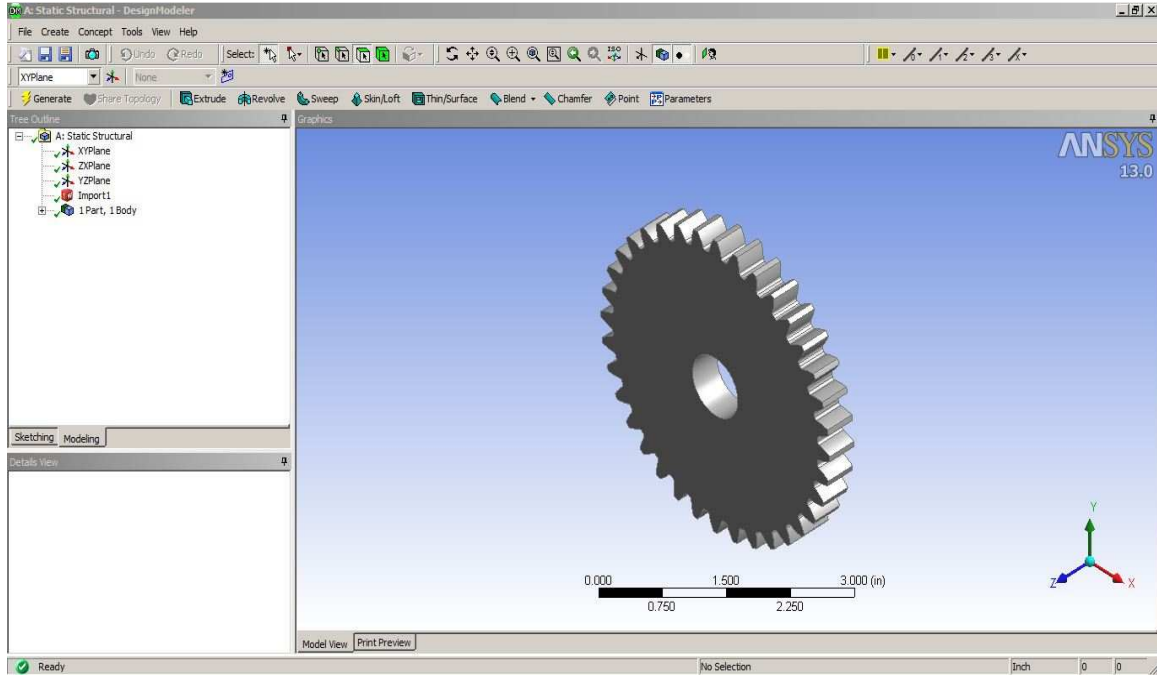
Discretization:

Step 1: On select mesh in the outline window a mesh tool bar appears on the top

Step 2: On Select the Mesh Control button from the tool bar a drop down menu appears,

   Now select the patch independent option from it. The following options appear in

   the details windows.

Step 3: In the scope menu, select the part in the graphics window for geometry column.

Step 4: Select the Patch Independent for the algorithm column, tetrahedron for the

   Method and enter a value of 0.1 for Min Size Limit Column.

Step 5: Now generate the mesh by hitting the Generate mesh button from mesh menu

Step 6: The Following screen shot displays the final preview of the generated mesh
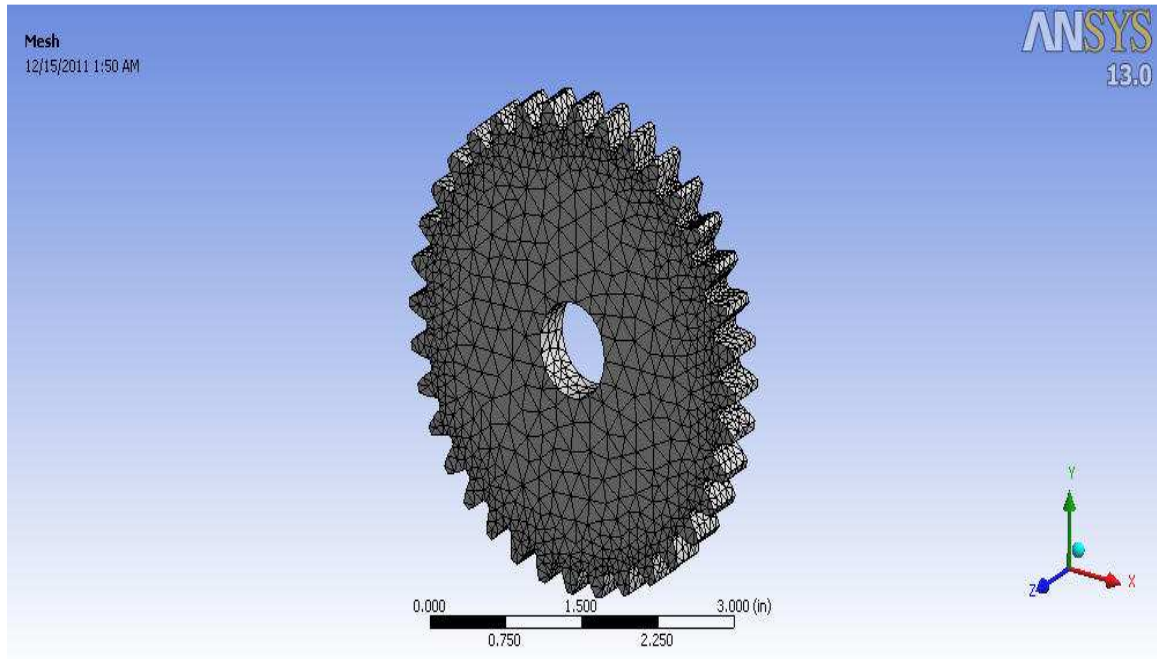
Load and Supports:

Step 1: On selecting the Static Structural analysis from the outline window, an

environment toolbar appears on the top.

Step 2: Select the Force icon from the loads menu.

Step 3: For the geometry column select the edge where you want to apply the load from

the graphics window.

Step 4: Define the force in the form of components and apply a load of 93.96 lb. force

Step 5: Select the fixed support icon from the support menu in the environment toolbar a

details window appears as follows

Step 6: Select the face to the fixed from the graphics window for the geometry column

Step 7: The Final view of the part is shown as screenshot
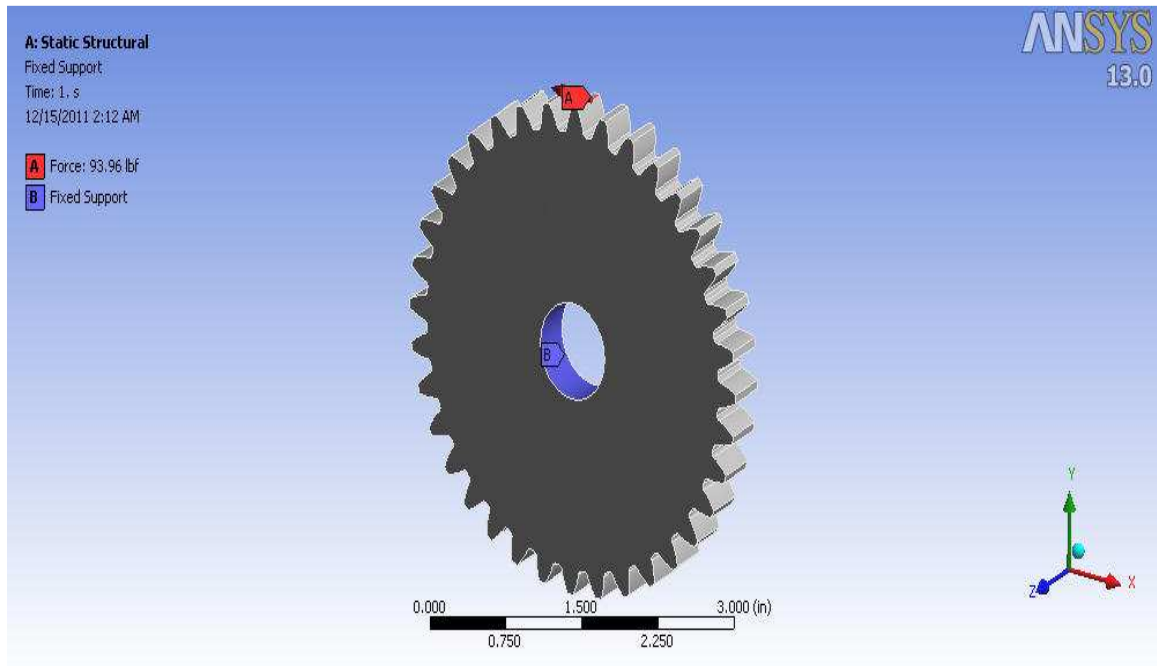
Solution:

Step 1: On selecting the solution in the outline menu solution tool bar appears

Step 2: Select the Normal stress option from the stress menu

Step 3: The following details windows appears

Step 4: Now select the y-axis in the orientation menu (Since we need the bending stress

for the tooth)

Step 5: The Minimum and Maximum normal stress are shown the Results menu.

Step 6: The following screen shot displays the final solution view

Post Processing:

Step 1: On highlighting the solution, Solution Tool bar appears on the top.

Step 2: From the Tool bar select the Stress Option and select the normal Stress Option.

Step 3: Details window appears, Select the orientation to Y- Axis

Step 4: In the integration point results menu select the display option to Average

Step 5: Hit the evaluate Results button

Step 6: Final Normal Stress view appears as follows
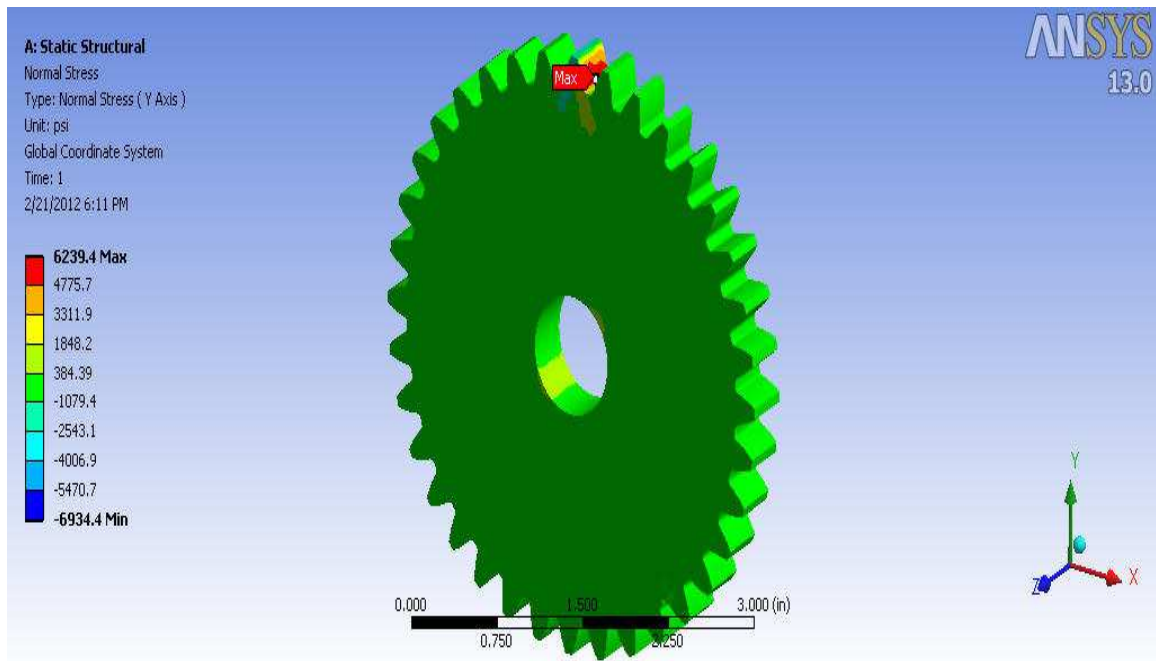
APPENDIX III

DESIGN OF CYLINDERS

Step 1: Select the front plane and start a new sketch
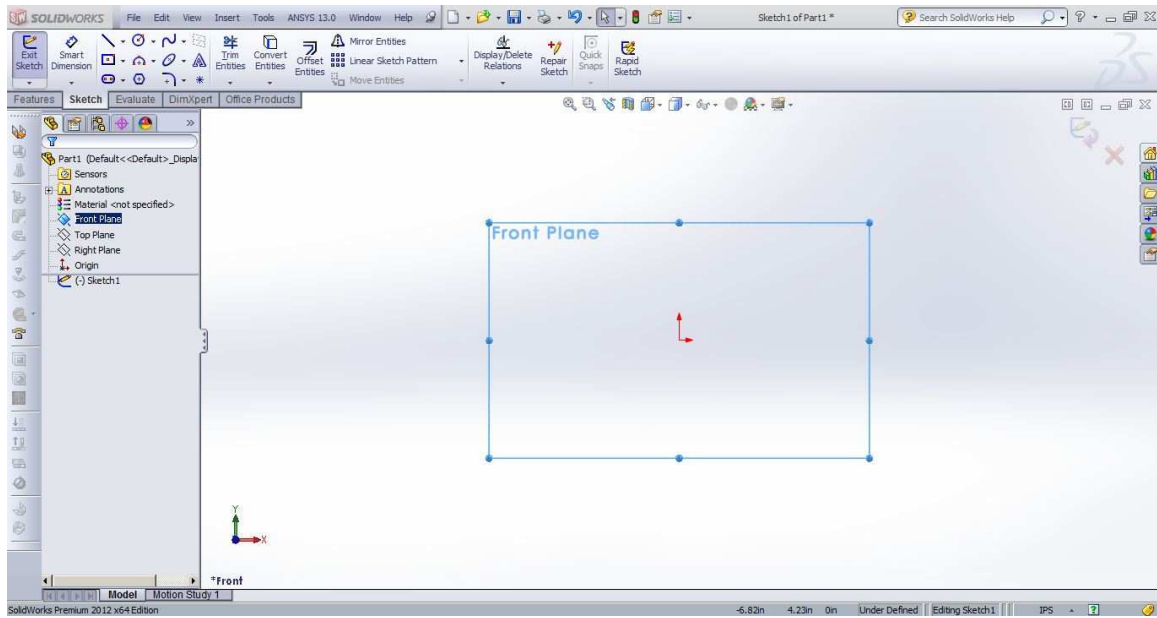
Step 2: Draw a circle of diameter 1 inch
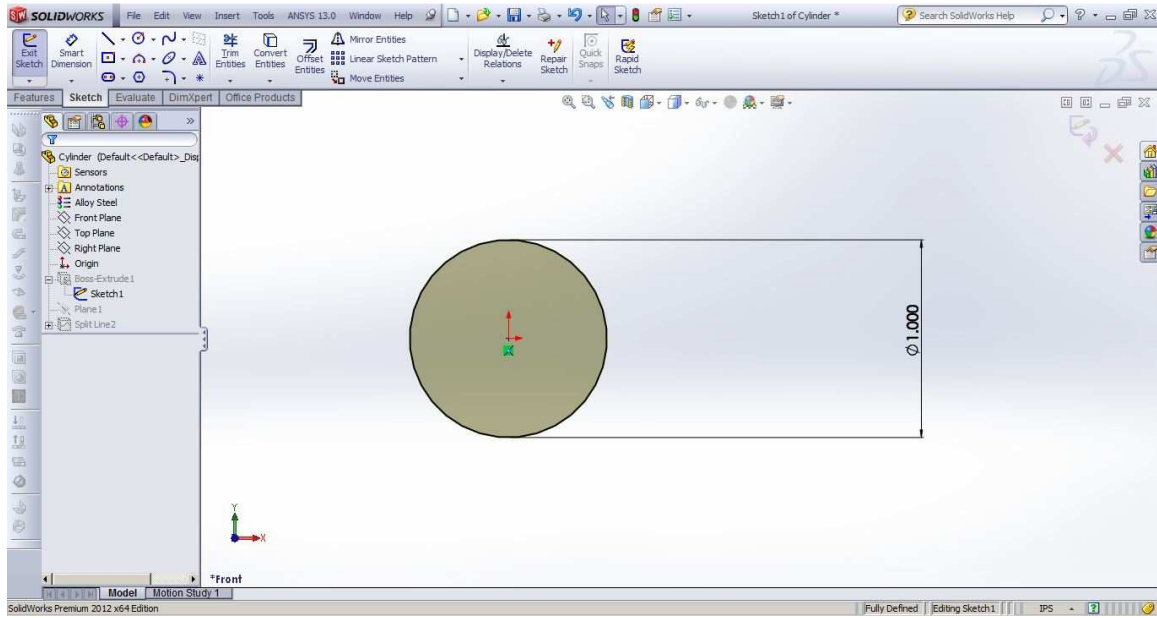
Step 3: Extrude the Sketch to a height equal to the face width of the gear tooth i.e., 0.4 in
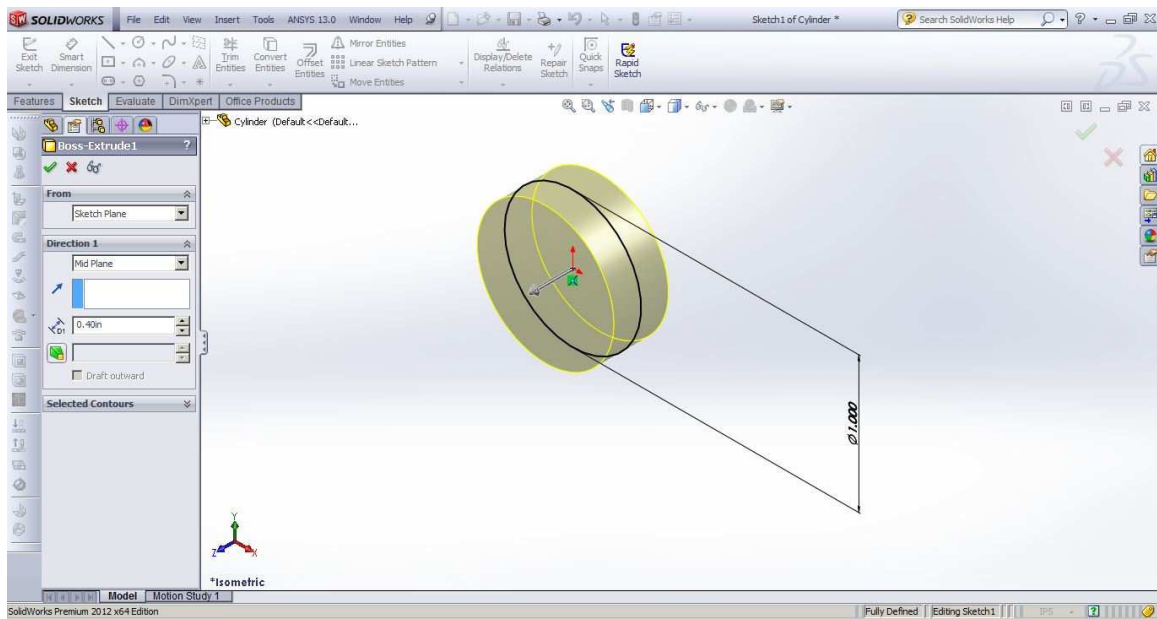
Step 4: Start a new assembly part
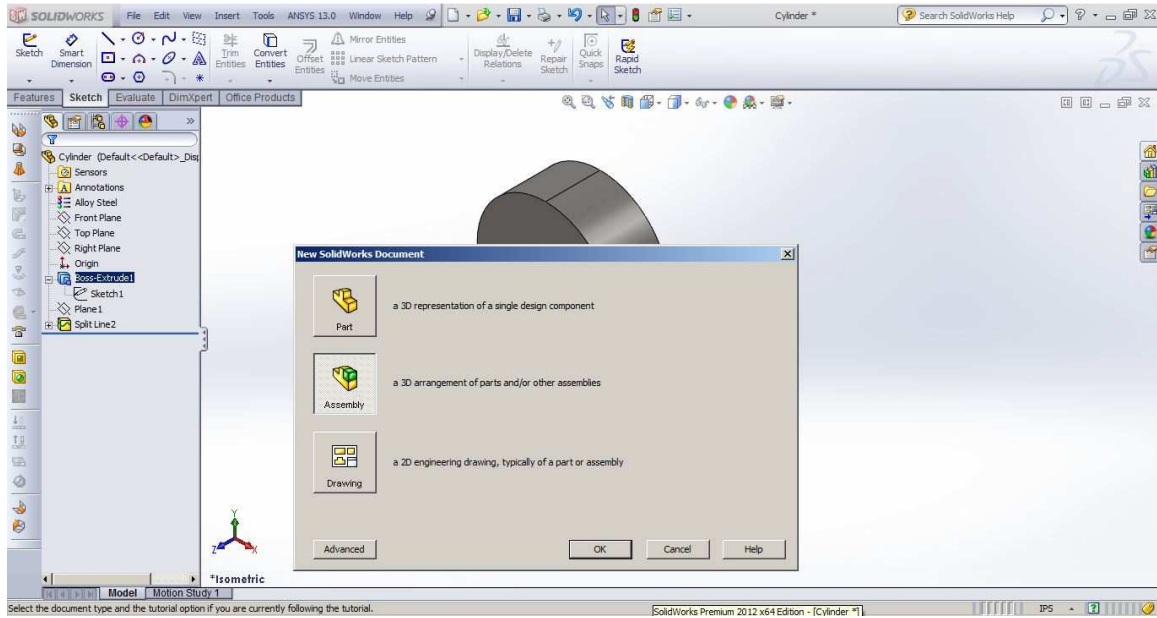
Step 5: Constrain one of axis of the cylinder so that it intersect the top and right planes
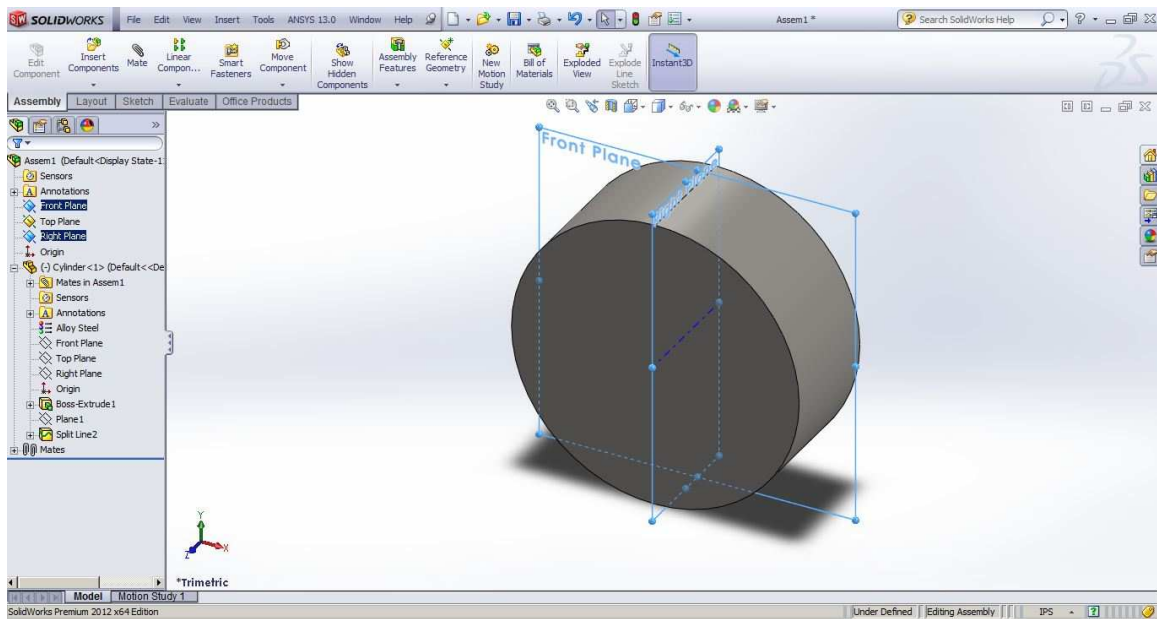
Step 6: Insert the same Cylinder again and constrain it so that axis is coincident on the right plane, both the faces of cylinder align parallel and the axis of both the cylinders is at a distance of 1 in
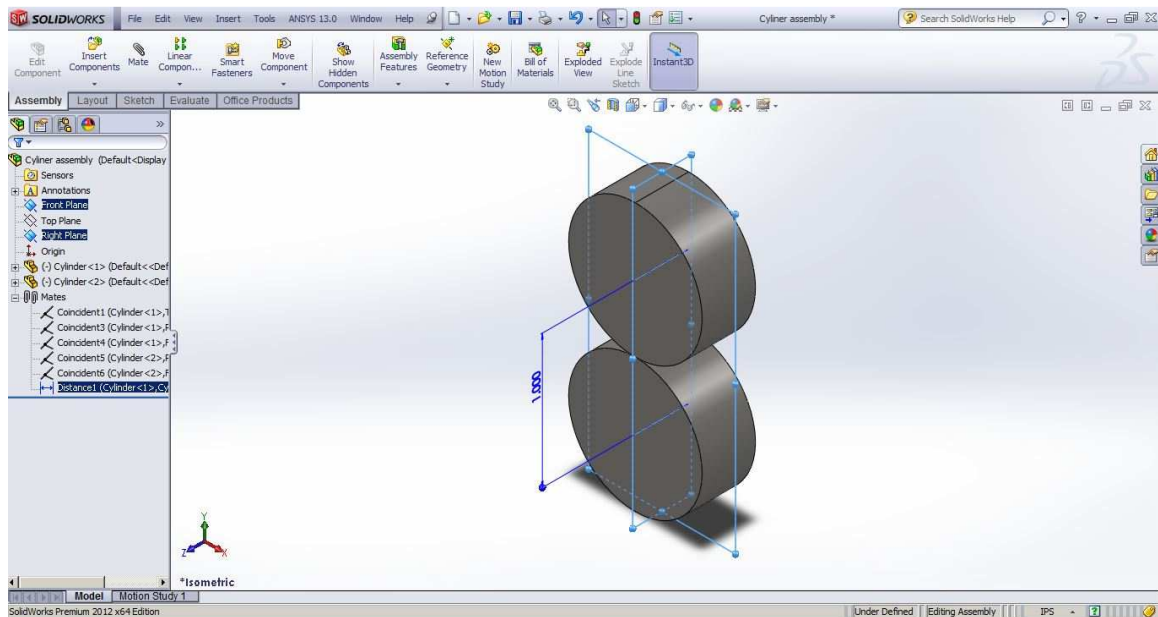
APPENDIX IV

CONTACT STRESS MODELING

Step 1: Create a Workbench project to perform Static Structural Analysis
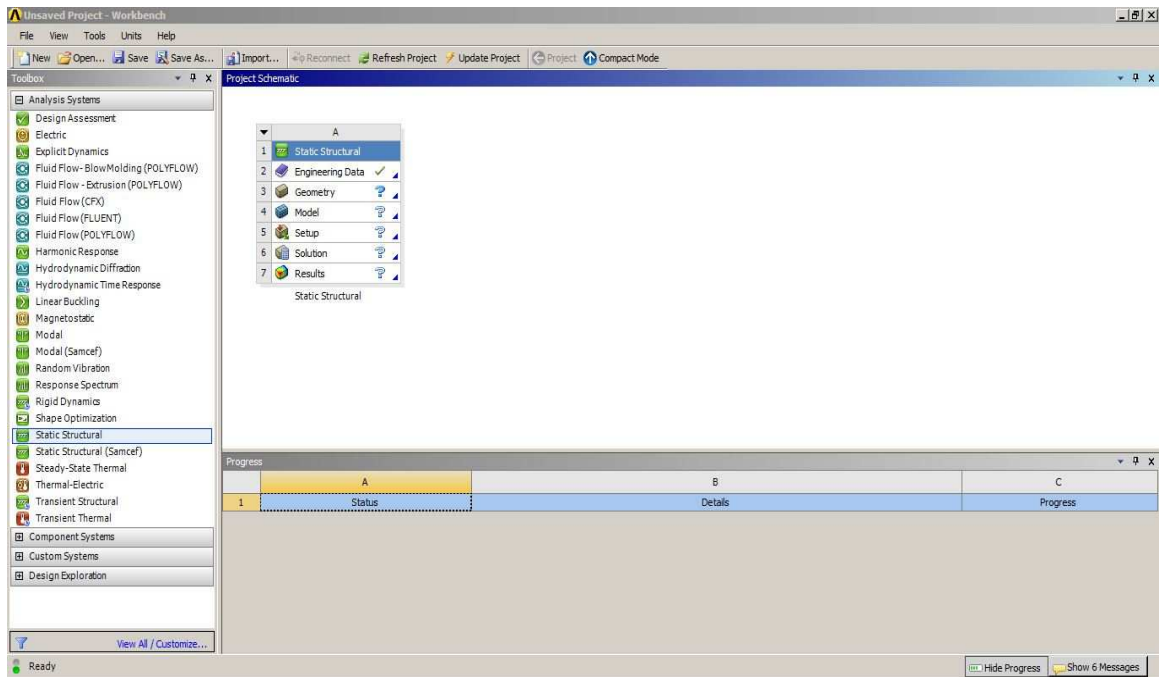
Step 2: Select the engineering data and create an alloy
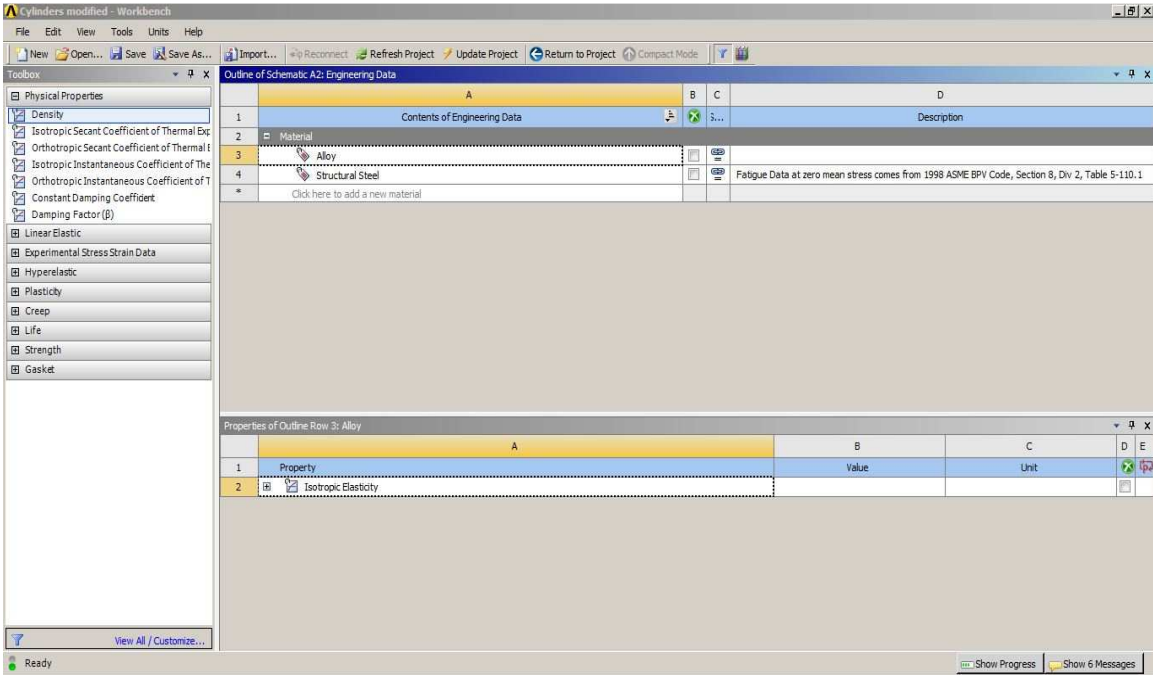
Step 3: Add density from Physical Properties menu
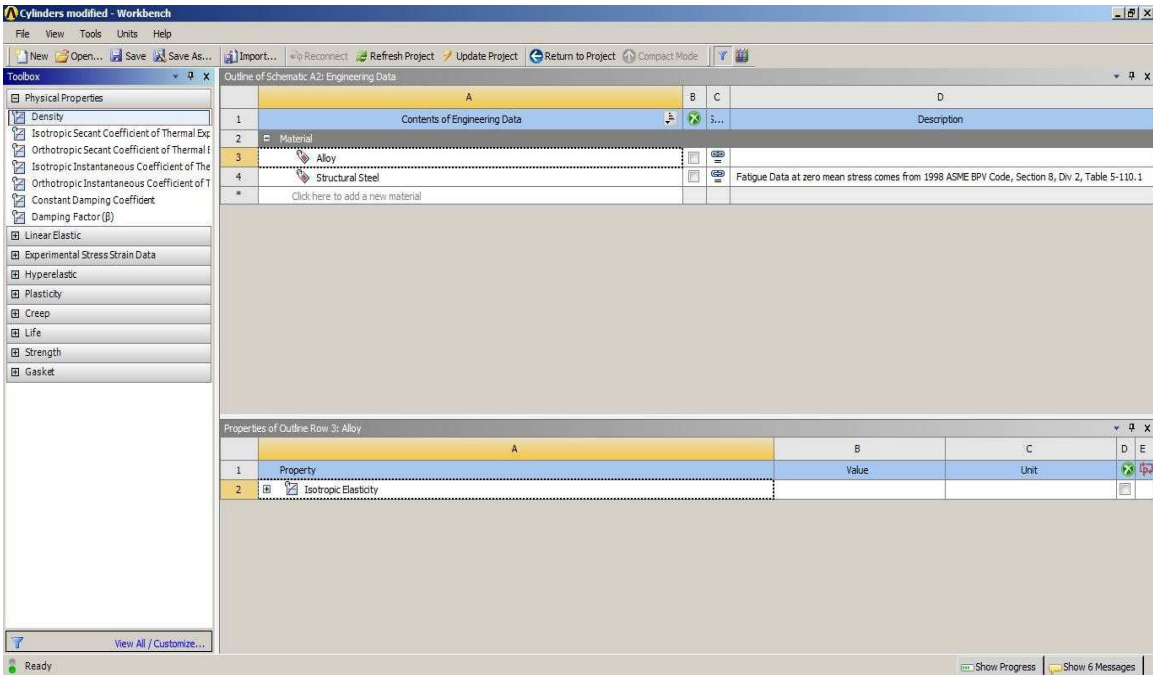
Step 4: Add Young's Modulus and Poisson's ratio from Linear Elastic menu
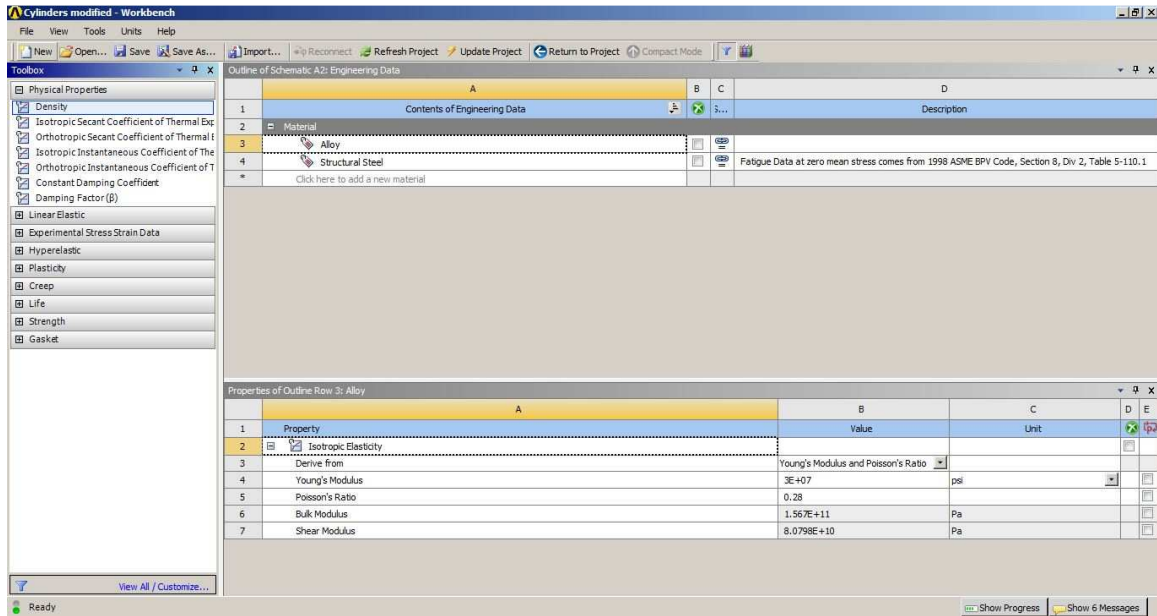
Step 5: Enter the values of Young's Modulus and Poisson ratio in their columns

Step 6: Return to the Project

Step 7: Open the Design Modeler from the workbench menu and import the solid works
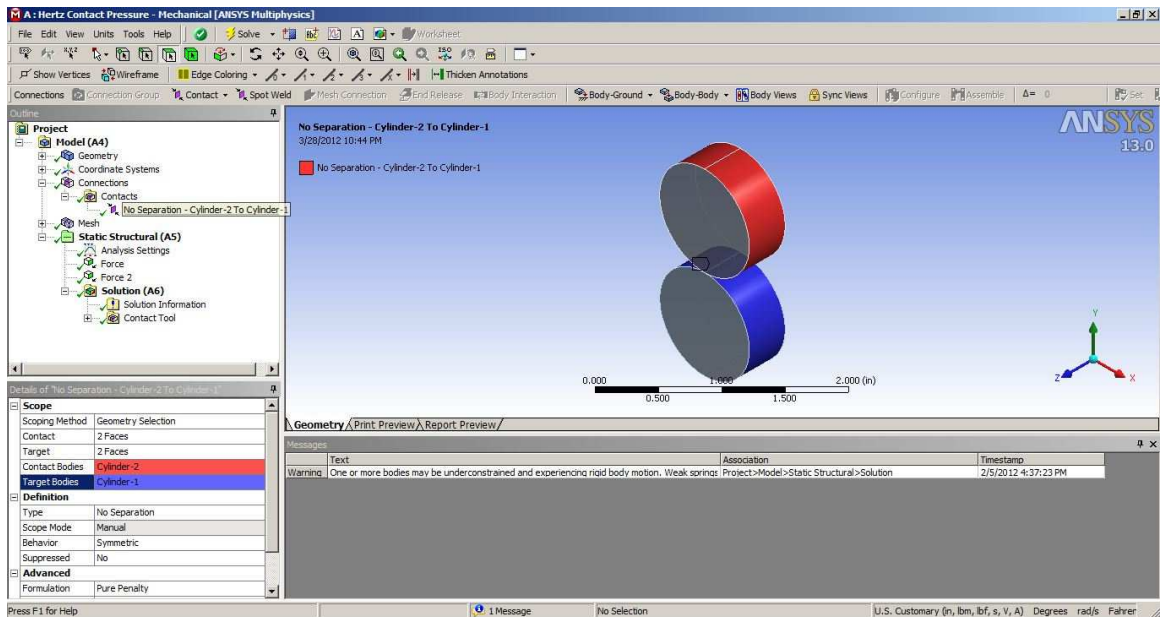
assembly.
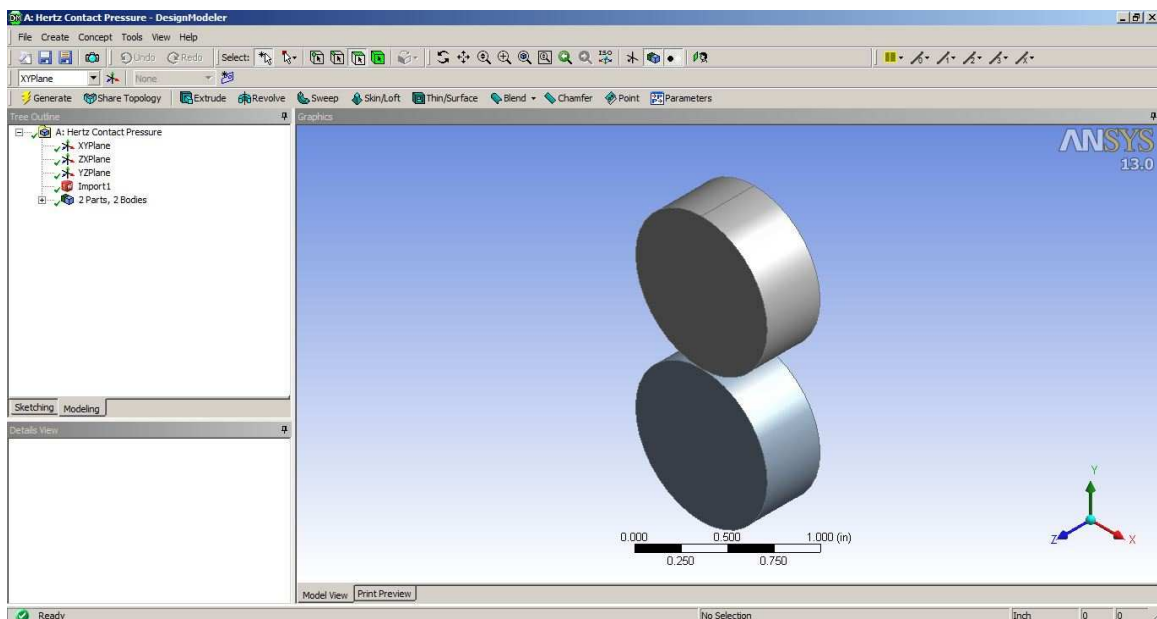
Step 8: Close Design Modeler

Discretization:

Step 1: In the Outline toolbar expand the Geometry section change the material to alloy

Step 2: In the Connections menu select the contacts and add a No – Separation contact to it.
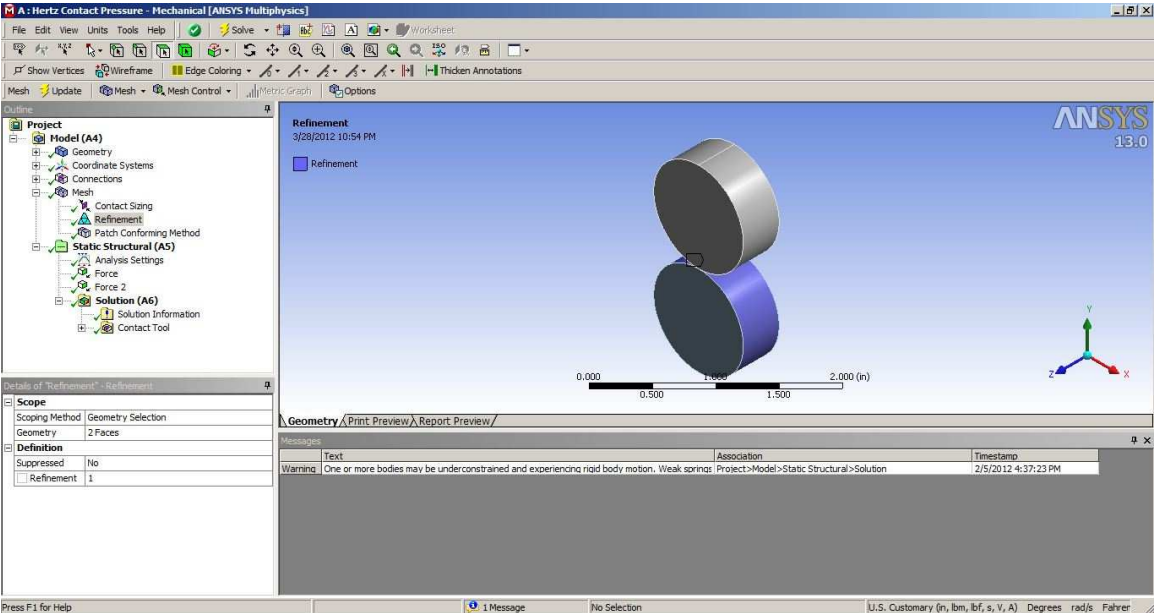
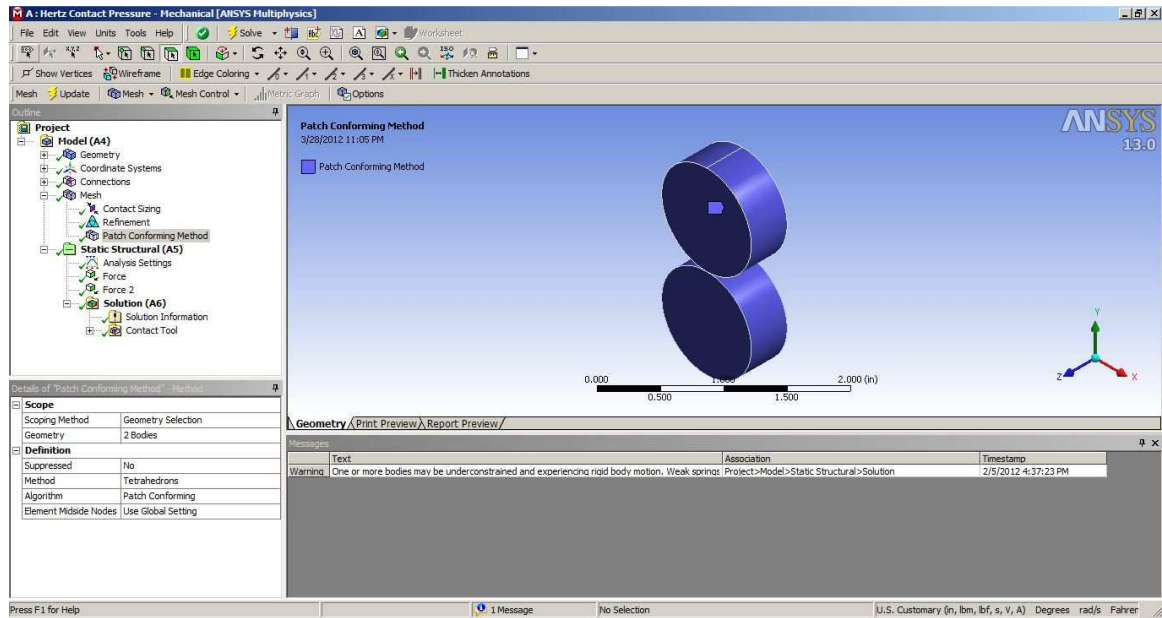Step 3: Select the Contact Surfaces from the geometry window

Step 4: Add a Contact sizing tool to the assembly, select the contact region and define

element size as 0.03 inch.
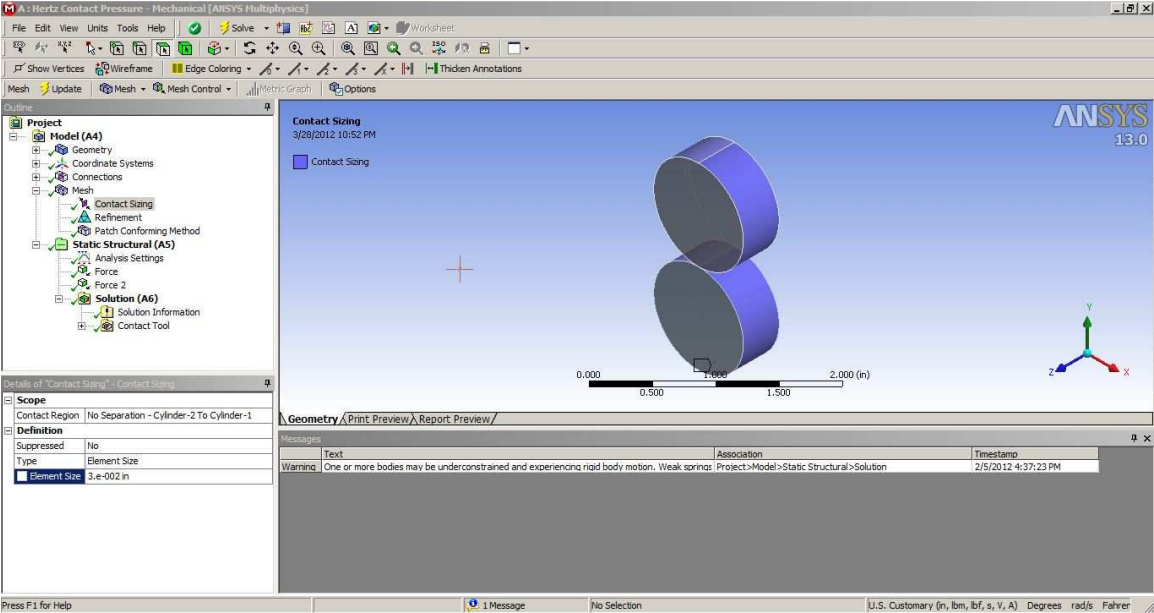
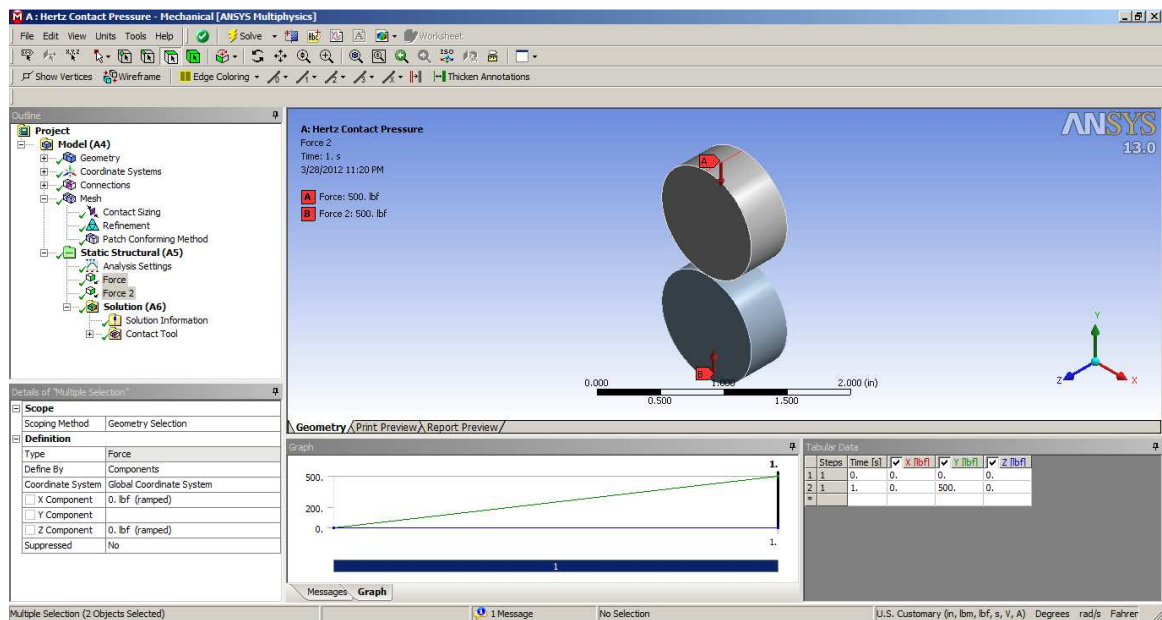Step 5: Add a refinement tool and set the refinement value to 1

Step 6: Add a meshing method and set method to tetrahedron and Algorithm to Patch conforming method

Loads and Supports:

Step 1: Add forces of magnitude 500 lbf in the Y-direction

Step 2: Run the solution

Step 3: Insert a pressure tool from contact tools in the solution and set the display option

to nodal difference