STATIC AND DYNAMIC ANALYSIS OF HCR SPUR GEAR DRIVE USING FINITE ELEMENT ANALYSIS

A THESIS SUBMITTED BY

PANKAJ KUMAR JENA
ROLL NO: 10503007

Bachelor of Technology
In
Mechanical Engineering

Under the Guidance of

Prof. S.C. MOHANTY

Department of Mechanical Engineering
National Institute of Technology
Rourkela

2008-2009

National Institute of Technology
ROURKELA

CERTIFICATE

This is to certify that the PROJECT entitled, ***"STATIC AND DYNAMIC ANALYSIS OF HCR SPUR GEAR DRIVE USING FINITE ELEMENT ANALYSIS"*** submitted by Mr**. Pankaj Kumar Jena** in partial fulfillment of the requirements for the award of Bachelor of Technology in Mechanical Engineering at the National Institute of Technology, Rourkela (Deemed University) is an authentic work carried out by him under my supervision and guidance.

To the best of my knowledge, the matter embroiled in the project has not been submitted to any other University/ Institute for the award of any Degree or Diploma.

Date: May 13, 2009

Prof. **S.C.Mohanty**
Department of Mechanical Engineering
National Institute of Technology,
Rourkela-769008

ACKNOWLEDGEMENT

I deem it a privilege to have been the students of Mechanical Engineering stream in National Institute of Technology, ROURKELA.

My heartfelt thanks to Prof S.C.Mohanty, our project guide who helped me to bring out this project in good manner with his precious suggestion and rich experience. Without his guidance and support this work would have been impossible.

I take this opportunity to express our sincere thanks to our project guide for co-operation and to reach a satisfactory conclusion.

PANKAJ KUMAR JENA

ROLL NO: 10503007

ABSTRACT

This thesis investigates the characteristics of a gear system including contact stresses, bending stresses, and the transmission errors of gears in mesh. Gearing is one of the most critical components in mechanical power transmission systems. The contact stresses were examined using 2-D FEM models. The bending stresses in the tooth root were examined using a 3-D FEM model.

Current methods of calculating gear contact stresses use Hertz's equations, which were originally derived for contact between two cylinders. To enable the investigation of contact problems with FEM, the stiffness relationship between the two contact areas is usually established through a spring placed between the two contacting areas. This can be achieved by inserting a contact element placed in between the two areas where contact occurs. The results of the two dimensional FEM analyses from ANSYS are presented. These stresses were compared with the theoretical values. Both results agree very well. This indicates that the FEM model is accurate.

This thesis also considers the variations of the whole gear body stiffness arising from the gear body rotation due to bending deflection, shearing displacement and contact deformation. Many different positions within the meshing cycle were investigated. Investigation of contact and bending stress characteristic of spur gears continues to be of immense attention to both engineers and researchers in spite of many studies in the past. This is because of the advances in the engineering technology that demands for gears with ever increasing load capacities and speeds with high reliability, the designers need to be able to accurately predict the stresses experienced the stresses experienced by the loaded gears.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

NOMENCLATURE

K	Structural stiffness
u	Displacement vector
F	Applied load vector
P_{max}	Maximum contact stress
d_1	Pinion pitch diameter
d_2	Gear pitch diameter
F_i	Load per unit width
R_i	Radius of cylinder i
Φ	Pressure angle
ν_i	Poisson's ratio for cylinder i
E_i	Young's modulus for cylinder i
σ_H	Maximum Hertz stress.
a	Contact width
r	Any radius to involute curve
r_b	Radius of base circle
θ	Vectorial angle at the pitch circle
ξ	Vectorial angle at the top of tooth
φ	Pressure angle at the pitch circle
φ_1	Pressure angle at radius r

B_p	Tooth displacement vectors caused by bending and shearing of the pinion
B_g	Tooth displacement vectors caused by bending and shearing of the gear
H_p	Contact deformation vectors of tooth pair B for the pinion
H_g	Contact deformation vectors of tooth pair B for the gear
θ_p	Transverse plane angular rotation of the pinion body
θ_g	Transverse plane angular rotation of the gear body
p_d	Diametral pitch
Y	Lewis form factor
K_a	Application factor
K_s	Size factor
Km	Load distribution factor
K_v	Dynamic factor
F_t	Normal tangential load
Y_j	Geometry factor
θ_g	Angular rotation of the output gear
θ_p	Angular rotation of the input gear

CHAPTER 1 INTRODUCTION

1.1 Research overview

Gearing is one of the most critical components in a mechanical power transmission system, and in most industrial rotating machinery. 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. In addition, the rapid shift in the industry from heavy industries such as shipbuilding to industries such as automobile manufacture and office automation tools will necessitate a refined application of gear technology.

A pair of teeth in action is generally subjected to two types of cyclic stresses: bending stresses inducing bending fatigue and contact stress causing contact fatigue. Both these types of stresses may not attain their maximum values at the same point of contact. However, combined action of both of them is the reason of failure of gear tooth leading to fracture at the root of a tooth under bending fatigue and surface failure, like pitting or flaking due to contact fatigue. In addition there may be surface damage associated seizure of surfaces due to poor lubrication and overloading. The seizure of surfaces leading to welding is usually prevented by proper lubrication so that there is always a very thin film of lubricant between a pair of teeth in motion. However the fracture failure at the root due to bending stress and pitting and flaking of the surfaces due to contact stress cannot be fully avoided. Theses types of failures can be minimized by careful analysis of the problem during the design stage and creating proper tooth surface profile with proper manufacturing methods. In spite of all the cares, these stresses are sometimes very high either due to overloading or wear of surfaces with use and need proper investigation to accurately predict them under stabilized working conditioned so that unforeseen failure of gear tooth can be minimized.

Gears are usually used in the transmission system is also called a speed reducer, gear head, gear reducer etc., which consists of a set of gears, shafts and bearings that are factory mounted in an enclosed lubricated housing. Speed reducers are available in a broad range of sizes, capacities and speed ratios. Their job is to convert the input provided by a prime mover (usually an electric motor) into an output with lower speed and correspondingly higher torque. In this thesis, analysis of the characteristics of spur gears in a gearbox was studied using nonlinear FEM.

The increasing demand for quiet power transmission in machines, vehicles, elevators and generators, has created a growing demand for a more precise analysis of the characteristics of gear systems. In the automobile industry, the largest manufacturer of gears, higher reliability and lighter weight gears are necessary as lighter automobiles continue to be in demand. In addition, the success in engine noise reduction promotes the production of quieter gear pairs for further noise reduction. Noise reduction in gear pairs is especially critical in the rapidly growing field of office-automation equipment as the office environment is adversely affected by noise, and machines are playing an ever widening role in that environment. Ultimately, the only effective way to achieve gear noise reduction is to reduce the vibration associated with them. The reduction of noise through vibration control can only be achieved through research efforts by specialists in the field. However, a shortage of these specialists exists in the newer, lightweight industries in Japan mainly because fewer young people are specializing in gear technology today and traditionally the specialists employed in heavy industries tend to stay where they are.

Designing highly loaded spur gears for power transmission systems that are both strong and quiet requires analysis methods that can easily be implemented and also provide information on contact and bending stresses, along with transmission errors. The finite element method is capable of providing this information, but the time needed to create such a model is large. In order to reduce the modeling time, a preprocessor method that creates the geometry needed for a finite element analysis may be used, such as that provided by CATIA. CATIA can generate models of three-dimensional gears easily. In CATIA, the geometry is saved as a file and then it can be transferred from CATIA to ANSYS. In ANSYS, one can click File > Import > IGES > and check No defeaturing and Merge coincident key points.

Gears analyses in the past were performed using analytical methods, which required a number of assumptions and simplifications. In general, gear analyses are multidisciplinary, including calculations related to the tooth stresses and to tribological failures such as like wear or scoring. In this thesis, static contact and bending stress analyses were performed, while trying to design spur gears to resist bending failure and pitting of the teeth, as both affect transmission error.

As computers have become more and more powerful, people have tended to use numerical approaches to develop theoretical models to predict the effect of whatever are studied. This has improved gear analyses and computer simulations. Numerical methods can potentially provide more accurate solutions since they normally require much less restrictive assumptions. The model and the solution methods, however, must be chosen carefully to ensure that the results are accurate and that the computational time is reasonable. The finite element method is very often used to analyze the stress state of an elastic body with complicated geometry, such as a gear. There have been numerous research studies in the area.

In this thesis, first, the finite element models and solution methods needed for the accurate calculation of two dimensional spur gear contact stresses and gear bending stresses were determined. Then, the contact and bending stresses calculated using ANSYS 7.1 were compared to the results obtained from existing methods. The purpose of this thesis is to develop a model to study and predict the contact stresses, and the torsional mesh stiffness of gears in mesh using the ANSYS 7.1 software package based on numerical method.

1.2 Objectives of the Research

In spite of the number of investigations devoted to gear research and analysis there still remains to be developed, a general numerical approach capable of predicting the effects of variations in gear geometry, Hertz contact stresses,bending stresses and Von Mises stresses. The objectives of this thesis are to use a numerical approach to develop theoretical models of the behavior of spur gears in mesh, to help to predict the effect of gear tooth stresses and transmission error. The main focus of the current research as developed here is:

- To develop and to determine appropriate models of contact elements, to calculate contact stresses using ANSYS and compare the results with Hertzian theory.
- To generate the profile of spur gear teeth and to predict the effect of gear bending using a three dimensional model and two dimensional model and compare the results with those of the Lewis equation.
- To determine the static transmission errors of whole gear bodies in mesh.

The objectives in the modeling of gears in the past by other researchers have varied from vibration analysis and noise control, to transmission error during the last five decades. The goals in gear modeling may be summarized as follows:

- Stress analysis such as prediction of contact stress and bending stress.
- Prediction of transmission efficiency.
- Finding the natural frequencies of the system before making the gears.
- Performing vibration analyses of gear systems.
- Evaluating condition monitoring, fault detection, diagnosis, prognosis,reliability and fatigue life.

Different analysis models will be described in chapter 2. For gears, there are many types of gear failures but they can be classified into two general groups. One is failure of the root of the teeth because the bending strength is inadequate. The other is created on the surfaces of the gears. There are two theoretical formulas, which deal with these two fatigue failure mechanisms. One is the Hertzian equation, which can be used to calculate the contact stresses. The other is the Lewis formula, which can be used to calculate the bending stresses. The surface pitting and scoring shown in Figure 1.1 is an example of failure which resulted in the fatigue failure of tooth surface. The Hertzian equation will be used to investigate surface pitting and scoring by obtaining the magnitude of the contact stresses.

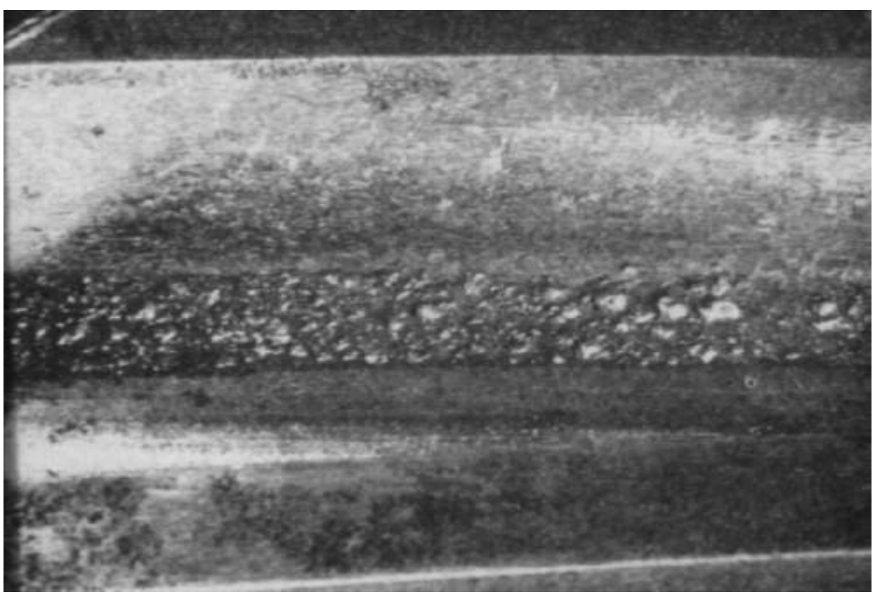


Figure 1.1 Fatigue failure of the tooth surface

Gear load capacity may be limited either by tooth contact conditioned or by the strength of the teeth themselves. In general as the material properties are improved, the load capacity of a given design of tooth will increase. However the contact stress will increase at a lower rate than the toot stresses. The deflection of the gear tooth at medium and heavy loads provides errors in both the pitch and profile of the gear tooth. This deflection-induced profile changes produce variations in angular velocity and sudden tooth engagement and disengagement, which results in dynamic loading of gear teeth, vibration, noise and reduction in the life of gears. With increased requirements for high speed, heavy load and lightweight in gear design, the fatigue damage analysis of modern gears has become important. Such analysis requires accurate determination of dynamic gear tooth loads and stresses. High contact ratio (HCRG) at least two tooth pair always shares the instantaneous load. A dynamic load analysis is required t determine the operating load sharing among the two or three tooth pairs in contact. Root bending stress is to be accurately estimated for economical design of the gear tooth dimensions.

Pitting and scoring is a phenomenon in which small particles are removed from the surface of the tooth due to the high contact stresses that are present between mating teeth. Pitting is actually the fatigue failure of the tooth surface. Hardness is the primary property of the gear tooth that provides resistance to pitting. In other words, pitting is a surface fatigue failure due to many repetitions of high contact stress, which occurs on gear tooth surfaces when a pair of teeth is transmitting power.

The literature available on the contact stress problems is extensive. But that available on the gear tooth contact stress problem is small. Klenz examined the spur gear contact and bending stresses using two dimensional FEM. Coy and Chao studied the effect of the finite element grid size on Hertzian deflection in order to obtain the optimum aspect ratio at the loading point for the finite element grid. Gatcombe and Prowell studied the Hertzian contact stresses and duration of contact for a very specific case, namely a particular rocket motor gear tooth. Tsay has studied the bending and contact stresses in helical gears using the finite element method with the tooth contact analysis technique.

1.3 Layout of Thesis

This thesis is comprised of a total of five chapters. Chapter 1 presents a general introduction, and the objectives to be achieved. Finally the layout of the thesis is described. Chapter 2 is a literature review and gives background of characteristics of spur gears for different types of modeling. Chapter 3 describes why the contact problem is difficult. A contact problem classification was done which as well as provides a discussion of the advantages and disadvantages of contact elements. Finally a discussion on how to overcome some of the disadvantages is presented. In Chapter 5 the contact stress model between two cylinders was then developed. Many graphical results from ANSYS are shown. Chapter 5 begins with presentation of a gear tooth contact stress analysis model from ANSYS, and then presents the bending stresses from 3-D models and 2-D models for the different numbers of teeth. The results are compared with the results from the Lewis Formula. Chapter 6 gives the conclusions of this thesis, and suggests future work.

CHAPTER2 LITERATURE REVIEW AND BACKGROUND

There has been a great deal of research on gear analysis, and a large body of literature on gear modeling has been published. The gear stress analysis, the transmission errors, and the prediction of gear dynamic loads, gear noise, and the optimal design for gear sets are always major concerns in gear design. Errichello and Ozguven and Houser survey a great deal of literature on the development of a variety of simulation models for both static and dynamic analysis of different types of gears. The first study of transmission error was done by Harris. He showed that the behavior of spur gears at low speeds can be summarized in a set of static transmission error curves. In later years, Mark and analyzed the vibratory excitation of gear systems theoretically. He derived an expression for static transmission error and used it to predict the various components of the static transmission error spectrum from a set of measurements made on mating pair of spur gears. Kohler and Regan discussed the derivation of gear transmission error from pitch error transformed to the frequency domain. Kubo et al estimated the transmission error of cylindrical gears using a tooth contact pattern. The current literature reviews also attempt to classify gear model into groupings with particular relevance to the research. The following classification seems appropriate:

- Models with Tooth Compliance
- Models of Gear system Dynamics
- Models of A Whole Gearbox
- Models for Optimal Design of Gear Sets

2.1 Model with Tooth Compliance

These models only include the tooth deformation as the potential energy storing element in the system. There are studies of both single tooth and tooth pair models. For single tooth models, a method of stress analysis was developed. For the models with paired teeth, the contact stresses and meshing stiffness analysis usually were emphasized. The system is often modeled as a single degree of freedom spring-mass system. The basic characteristic in this group is that the only compliance considered is due to the gear tooth deflection and that all other elements have assumed to be perfectly rigid.

Harris made an important contribution to this area. The importance of transmission error in gear trains was discussed and photo-elastic gear models were used in his work. Due to the loss of contact he considered manufacturing errors, variation in the tooth stiffness and non-linearity in tooth stiffness as three internal sources of vibration. Harris was the first investigator who pointed out the importance of transmission error by showing the behavior of spur gears at low speeds. His work can be summarized in a set of static transmission error curves. In 1969, Aida presented other examples of studies in this area. He modeled the vibration characteristics of gears by considering tooth profile errors and pitch errors, and by including the variation of teeth mesh stiffness. In 1967, Tordion first constructed a torsional multi-degree of freedom model with a gear mesh for a general rotational system. The transmission error was suggested as a new concept for determining the gear quality, rather than individual errors.

In 1981, Cornell obtained a relationship between compliance and stress sensitivity of spur gear teeth. The magnitude and variation of the tooth pair compliance with load position affects the dynamics and loading significantly. The tooth root stresses versus load varies significantly with load positions. With improved fillet/foundation compliance analysis the compliance analysis was made based on work by Weber and O’Donell. The stress sensitivity analysis is a modified version of the Heywood method. These improved compliance and stress sensitivity analyses were presented along with their evaluation using tests, finite element analysis, and analytic transformation results, which indicated good agreement.

In 1988, Umezawa developed a new method to predict the vibration of a helical gear pair. The developed simulator was created through theoretical analysis on the vibration of a narrow face width helical gear pair. The studies on a helical gear are very different from the ones on a spur gear. A simple outline of the theoretical analysis on the vibration of a helical gear is given below. The length of path of contact is on the plane of action of helical gear pairs in Figure 2.1. A pair of mated teeth starts meshing at point S on the plane of action. The meshing of the pair proceeds on the plane with the movement of the contact line. It finishes at point E, the position of the line-of contact is represented by the coordinate Y along the line-of-action of the helical gear (hereafter simply stated as the line-of-action) which is considered to be the middle of the face width. That is, the starting point of meshing S is substituted by S', the position of line-of-contact $_{c}CC$ by C' and the ending point E by E' on the line-of-action. A vibration model was built there. When the rotational vibration of a power transmitting helical gear pair is considered along the line-of-action model similar to the case of a spur-gear pair, in which the tooth is replaced by a spring and the gear blank is replaced by a mass.

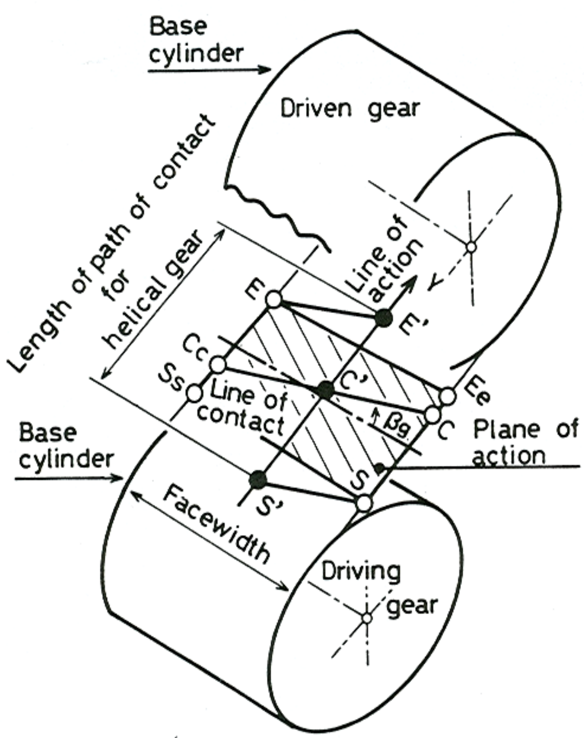


Figure 2-1 Meshing of a helical pair

In 1992, Vijayarangan and Ganesan [58] studied static contact stresses including the effect of friction between the mating gear teeth. Using the conventional finite element method the element stiffness matrices and the global stiffness matrix $[K]$of the two gears in mesh were obtained. If the external forces at the various nodes are known, then the system of equations is written as:

$$[K]\{U\} = \{F\} \tag{2.1}$$

where $\{U\}$ is the nodal displacement vector and $\{F\}$is the nodal force vector. The system of equations is solved and $\{U\}$ is obtained. Then the stress can be calculated. Each gear is divided into a number of elements such that in the assumed region of contact there is equal number of nodes on each gear. These contact nodes are all grouped together.

In 2001, David and Handschuh investigated the effect of this moving load on crack trajectories. The objective of this work was to study the effect of the moving gear tooth load on crack propagation predictions. A finite element model of a single tooth was used to analyze the stress, deformation and fracture in gear teeth when subjected to dynamic loading. At different points on the tooth surface impulsive loads were applied. Moving loads normal to the tooth profile were studied. Even effective designs have the possibility of gear cracks due to fatigue. In addition, truly robust designs consider not only crack initiation, but crack propagation trajectories. As an example, crack trajectories that propagate through the gear tooth are the preferred mode of failure compared to propagation through the gear rim. Rim failures would lead to catastrophic events and should be avoided. Analysis tools that predict crack propagation paths can be a valuable aid to the designer to prevent such catastrophic failures. Using weighting function techniques to estimate gear tooth stress intensity factors, analytical methods have been developed. Numerical techniques such as the boundary element method and the finite element method have also been studied. Based on stress intensity factors, and fatigue crack growth, gear life predictions have been investigated. The gear crack trajectory predictions have been addressed in a few studies. From publications on gear crack trajectory predictions, the analytical methods have been used in numerical form (finite or boundary element methods) while solving a static stress problem.

2.2 Models of Gear System Dynamics

The current models can predict shaft torsional vibration, shaft bending stiffness, gear tooth bending stiffness, bearings stiffness, etc. The models of gear system dynamics include the flexibility of the other parts as well as the tooth compliance. The flexibility of shafts and the bearings along the line of action are discussed. In these models, the torsional vibration of the system is usually considered.

In 1971, Kasuba determined dynamic load factors for gears that were heavily loaded based on one and two degree of freedom models. Using a torsional vibratory model, he considered the torsional stiffness of the shaft. In 1981, he published another paper. An interactive method was developed to calculate directly variable gear mesh stiffness as a function of transmitted load, gear profile errors, gear tooth deflections and gear hub torsional deformation, and position of contacting profile points. These methods are applicable to both normal and high contact ratio gearing. Certain types of simulated sinusoidal profile errors and pitting can cause interruptions of the normal gear mesh stiffness function, and thus, increase the dynamic loads. In his research, the gear mesh stiffness is the key element in the analysis of gear train dynamics. The gear mesh stiffness and the contact ratio are affected by many factors such as the transmitted loads, load sharing, gear tooth errors, profile modifications, gear tooth deflections, and the position of contacting points.

In 1979 Mark analyzed the vibration excitation of gear systems. In his papers, formulation of the equations of motion of a generic gear system in the frequency domain is shown to require the Fourier-series coefficients of the components of vibration excitation. These components are the static transmission errors of the individual pairs in the system. A general expression for the static transmission error is derived and decomposed into components attributable to elastic tooth deformations and to deviations of tooth faces from perfect surfaces with uniform lead and spacing.

In the 1980s although more and more advanced models were developed in order to obtain more accurate predictions, some simple models were developed for the purpose of simplifying dynamic load prediction for standard gears. In 1980, the coupled torsional flexural vibration of a shaft in a spur geared system was investigated by some researchers. That the output shaft was flexible in bearing and the input shaft was rigid in bearing was assumed. Researchers derived equations of motion for a 6-degree-offreedom (DOF) system. The tooth contact was maintained during the rotation and the mesh was rigid in those models. Four years later, other researchers presented another model that consists of three shafts, rather than two shafts, one of them being a counter shaft.

In 1992, Kahraman developed a finite element model of a geared rotor system on flexible bearings. The gear mesh was modeled by a pair of rigid disks connected by a spring and a damper with a constant value which represented the average mesh value. Coupling between the torsional and transverse vibrations of the gear was considered in the model, and applied the transmission error as the excitation at the mesh point to simulate the variable mesh stiffness.

In 1996, Sweeney developed a systematic method of calculating the static transmission error of a gear set, based on the effects of geometric parameter variation on the transmission error. He assumed that the tooth (pair) stiffness is constant along the line of action (thin-slice model) and that the contact radius for calculation of Hertzian deformation is the average radius of the two profiles in contact. Sweeney’s model is applicable to cases where the dominant source of transmission error is geometric imperfections, and is particularly suited to automotive quality gear analysis. The results of his model gave very good agreement with measurements on automotive quality gears.

Randall and Kelley modifications have been made to Sweeney’s basic model to extend it to higher quality gears where the tooth deflection component is more important. The tooth deflection compliance matrix and the contact compliance vector have been derived using finite element models. The effects on the transmission error of the variation of the tooth body stiffness with the load application point have been investigated, and a simulation program for transmission error (TE) computation with varying stiffness has been developed. In order to study the case where the tooth deflection component is the dominant source of the transmission error nylon gears were used. All the simulation results have been compared with the measured transmission errors from a single-stage gearbox.

In 1999, Kelenz investigated a spur gear set using FEM. The contact stresses were examined using a 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.

In 2001, Howard simplified the dynamic gear model to explore the effect of friction on the resultant gear case vibration. The model which incorporates the effect of variation in gear tooth torsional mesh stiffness was developed using finite element analysis, as the gears mesh together. The method of introducing the frictional force between teeth into the dynamic equations is given in his paper. The comparison between the results with friction and without friction was investigated using Matlab and Simulink models developed from the differential equations.

In 2003, Wang surveyed the nonlinear vibration of gear transmission systems. The progress in nonlinear dynamics of gear driven system is reviewed, especially the gear dynamic behavior by considering the backlash and time-varying mesh stiffness of teeth. The basic concepts, the mathematical models and the solution methods for non-linear dynamics of geared systems were all reviewed in his paper.

2.3 Models of a Whole Gearbox

The studies in this group may be thought of as advanced studies. Traditional analysis approaches mentioned previously in the gear dynamic area have concentrated on the internal rotating system and have excluded dynamic effects of the casing and flexible mounts. All elements in the system including the gear casing are considered in the recent models. The studies of this group is to focus on the dynamic analysis including the gear pair, shafts, rolling element bearings, a motor, a load, a casing and a flexible or rigid mount. The gearbox may be single stage or multi-stage.

In 1991, Lim and Singh presented study of the vibration analysis for complete gearboxes. Three example cases were given there: a single-stage rotor system with a rigid casing and flexible mounts, a spur gear drive system with a rigid casing and flexible mounts, and a high-precision spur gear drive system with a flexible casing and rigid mounts. In 1994, Sabot and Perret-Liaudet presented another study for noise analysis of gearboxes. A troublesome part of the noise within the car or truck cab could be attributed by the transmission error which gives rise to dynamic loads on teeth, shafts, bearings and the casing. During the same year, a simulation method by integrating finite element vibration analysis was developed by others. Each shaft was modeled as a lumped mass and added to the shaft in their model. Each of the rolling element bearings was represented as a spring and damper. The casing of the gearbox was modeled by a thin shell element in the finite element package program.

2.4 Models for Optimal Design of Gear Sets

Several approaches to the models for optimum design of gears have been presented in the recent literature. Cockerham presents a computer design program for 20-degree pressure angle gearing, which ignores gear-tooth-tip scoring. This program varies the diametral pitch, face width, and gear ratio to obtain an acceptable design. Tucker and Estrin look at 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 favorable gear set. Gay considers gear tip scoring and shows how to modify a standard gear set to bring this mode of failure into balance with the pitting fatigue mode. In order to obtain an optimal design he adjusts the addendum ratios of the gear and pinion. The basic approach available is to check a given design to verify its acceptability to determine the optimal size of a standard gear mesh. With the object of minimizing size and weight, optimization methods are presented for the gearbox design. The gear strengths must be considered including fatigue as treated by the AGMA (American Gear Manufacturing Association). Surface pitting of the gear teeth in the full load region must also be handled with as scoring at the tip of the gear tooth.

In 1980, Savage and Coy optimized tooth numbers for compact standard spur gear sets. The design of a standard gear mesh was treated with the objective of minimizing the gear size for a given gear ratio, pinion torque, and the allowable tooth strength. Scoring, pitting fatigue, bending fatigue, and interference are considered. A design space is defined in terms of the number of teeth on the pinion and the diametric pitch. This space is then combined with the objective function of minimum center distance to obtain an optimal design region. This region defines the number of pinion teeth for the most compact design.

Many engineering design problems are multi objective as they often involve more than one design goal to be optimized. These design goals impose potentially conflicting requirements on the technical and cost reduction performances of system design. To study the trade-offs that exist between these conflicting design goals and to explore design options, one needs to formulate the optimization problem with multiple objectives. The optimization algorithms seek an optimum design, which attains the multiple objectives as closely as possible while strictly satisfying constraints.. Tappeta and Hwang and Masud summarized the progress in the field of multi-criteria optimization. A comprehensive survey of multi objective optimization methods is also given. The most traditional methods involve converting a multi objective problem into a single objective problem for a compromise solution is also presented. This scalarization was usually achieved using either weights or targets that the designers have to specify for each objective a priori. Some of the disadvantages of traditional methods are listed there.

In 2001, Chong and Bar demonstrated a multi objective optimal design of cylindrical gear pairs for the reduction of gear size and meshing vibration. The results of the relation between the geometrical volume and the vibration of a gear pair were analyzed, in addition a design method for cylindrical gear pairs to balance the conflicting objectives by using a goal programming formulation was proposed. The design method reduces both the geometrical volume and the meshing vibration of cylindrical gear pairs while satisfying strength and geometric constraints.

CHAPTER 3 STATIC ANALYSIS OF SPUR GEARS IN MESH

3.1 FINITE ELEMENT ANALYSIS:

In this finite element analysis the continuum is divided into a finite numbers of elements, having finite dimensions and reducing the continuum having infinite degrees of freedom to 'finite' degrees of unknowns. It is assumed that the elements are connected only at the nodal points.
The accuracy of solution increases with the number of elements taken. However, more number of elements will result in increased computer cost. Hence optimum number of divisions should be taken.
In the element method the problem is formulated in two stages:

The element formulation:
It involves the derivation of the element stiffness matrix which yields a relationship between nodal point forces and nodal point displacements.

The system formulation:
It is the formulation of the stiffness and loads of the entire structure.

BASIC STEPS IN THE FINITE ELEMENT METHOD:

1. **Discretisation of the domain**
   The continuum is divided into a no. of finite elements by imaginary lines or surfaces. The interconnected elements may have different sizes and shapes .The success of this idealization lies in how closely this discretised continuum represents the actual continuum. The choice of the simple elements or higher order elements, straight or curved, its shape, refinement are to be decided before the mathematical formulation starts.

2. **Identification of variables**
   The elements are assumed to be connected at their intersecting points referred to as nodal points. At each node, unknown displacements are to be prescribed. They are dependent on the problem at hand. The problem may be identified in such a way that in addition to the displacement which occurs at the nodes depending on the physical nature of the problem, certain other quantities such as strain may need to be specified as nodal unknowns for the element, which however, may not have a corresponding physical quantity in the generalized forces. The value of these quantities can however be obtained from variation principles.

3. **Choice of approximating functions.**
   After the variables and local coordinates have been chosen, the next step is the choice of displacement function, which is the starting point of mathematical analysis. The function represents the variation of the displacement within the element. The function can be approximated in many ways. A convenient way of expressing it is by polynomial expressions.

The shape of the element or the geometry may also approximate. The coordinates of corner nodes define the element shape accurately if the element is actually made of straight lines or planes. The weightage to be given to the geometry and displacements also needs to be decided for a particular problem.

4. Formation of element stiffness matrix

After the continuum is discretised with desired element shapes, the element stiffness matrix is formulated. Basically it is a minimization procedure. The element stiffness matrix for majority of elements is not available in explicit form. They require numerical integration for this evaluation. The geometry of the element is defined in reference to the global frame.

5. Formation of the overall stiffness matrix

After the element stiffness matrix in global coordinates is formed, they are assembled to form the overall stiffness matrix. This is done through the nodes which are common to adjacent elements. At the nodes the continuity of the displacement functions and their derivatives are established. The overall stiffness matrix is symmetric and banded.

6. Incorporation of boundary conditions

The boundary restraint conditions are to be imposed in the stiffness matrix. There are various techniques available to satisfy the boundary conditions.

7. Formation of the element loading matrix.

The loading inside an element is transferred at the nodal points and consistent element loading matrix is formed.

8. Formation of the overall loading matrix

The element loading matrix is combined to form the overall loading matrix. This matrix has one column per loading case and it is either a column vector or a rectangular matrix depending on the no. of loading conditions.

9. Solution of simultaneous equations

All the equations required for the solution of the problem is now developed. In the displacement method, the unknowns are the nodal displacement. The Gauss elimination and Choleky's factorization are most commonly used methods.

10. Calculation of stresses or stress resultants

The nodal displacement values are utilized for calculation of stresses. This may be done for all elements of the continuum or may be limited only to some predetermined elements.

3.2 LIMITATIONS OF FEM

Due to the requirement of large computer memory and time, computer program based on FEM can be run only in high speed digital computers.

For some problems, there may be considerable amount of input data. Errors may creep up in their preparation and the results thus obtained may also appear to be acceptable which indicates deceptive state of affairs.

In the FEM, the size of problem is relatively large. Many problems lead to round off errors.

3.3 FINITE ELEMENT MODELS

1) The two dimensional models

Fatigue or yielding of a gear tooth due to excessive bending stress is two important gear design considerations. In order to predict fatigue and yielding, the maximum stresses on the tensile and compressive sides of the tooth, respectively, are required. 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 developments for more accurate FEM simulations. When meshing the teeth in ANSYS, if "SMART SIZE" is used the number of elements near the roots of the teeth are automatically are much greater than in other places. The maximum tensile stress on the tensile side and maximum compressive stress on the other side of the tooth respectively. It also indicates that only one tooth is enough for the bending stress analysis for the 3-D model or a 2-D model.

2) Three dimensional models

In this section the tooth root stresses and the tooth deflection of one tooth of a spur gear is calculated using an ANSYS model. For the bending stress, the numerical result is compared with the values given by the draft proposal of the standards of AGMA. The element type "SOLID TETRAHEDRAL 10 NODES 187" was chosen. Because "SMART SET" was chosen on the tool bar there are many more elements near the root of the tooth than in other places. There are middle side nodes on the each side of each element. So a large number of degrees of freedom in this 3D model take a longer time to finish running. From the stress distribution on the model, the large concentrated stresses are at the root of the tooth. There are also large Von Mises stresses on the root of the tooth. They are equal to the tensile stresses. The tensile stresses are the tensile side. From the Lewis equation if the diameters of the pinion and gear are always kept the same and the number of teeth was changed, the diametral pitch will be changed or the module of gears will be changed. That means that there are different bending strengths between the different teeth numbers.

3) Gear contact stress

One of the predominant modes of gear tooth failure is pitting. Pitting is a surface fatigue failure due to many repetitions of high contact stress occurring on the gear tooth surface while a pair of teeth is transmitting power. In other words, contact stress exceeding surface endurance strength with no endurance limits or a finite life causes this kind of failure. The AGMA has prediction methods in common use. Contact failure in gears is currently predicted by comparing the calculated Hertz stress to experimentally determine allowable values for the given material. The details of the subsurface stress field usually are ignored. This approach is used because the contact stress field is complex and its interaction with subsurface discontinuities is difficult to predict. However, all of this information can be obtained from ANSYS model. Since a spur can be considered as a two-dimensional component, without loss of generality, a plane strain analysis can be used. The nodes in the model were used for the analysis. The nodes on the bottom surface of the gear fixed. A total load is applied on the model.

3.4 PROCEDURE FOR FINITE ELEMENT ANALYSIS

The examination of the structural integrity of large speed reduction gear systems requires a proper assessment of the tooth contact and fillet stresses. In calculating these stresses, it is important to consider gears as complete structures rather than as pairs of teeth in mesh, through a two-dimensional analysis of an epicyclic gear stage. In large offset speed reduction gear systems, the problem is, in addition, three-dimensional in nature. The tooth contact load distribution is influenced significantly by the stiffness of the foundations — the rim, web, and shaft. The formulation of the finite-element method using the substructure concept is suited ideally to include all aspects of the static problem. Easy generation of the required finite-element network is important to study the effects of different gear parameters in a cost-effective manner. Such a study can be used to attempt weight reduction.

The transfer of torque from the pinion (driver) to the gear (driven) takes place along a differing number of tooth pairs. This number changes as the gears roll in and out of mesh. At any instant, it depends on whether it is a high- or low-contact ratio spur or helical gear pair. In the "just-loaded" state, the line of contact along the tooth face can be calculated using involute geometry relations. In the loaded state, the contact is over surfaces around these "lines" of contact. The variation of the contact load intensity along these surfaces depends on the distribution of the combined stiffness in the normal direction of the gear and pinion. This stiffness is a result of several parameters related to the teeth pressure and helix angle, chordal thickness, radius of contact etc. and to the foundation—rim, web, shaft, etc. The finite-element method with three-dimensional representation of the pinion and gear structures can internally compute the stiffness variations.

The finite-element method with three-dimensional representation of the pinion and gear structures can internally compute the stiffness variations. If nodes are identified on the pinion, and gear tooth contact faces and displacement boundary conditions in the normal direction are given, the contact stresses can be calculated by the analysis program. However, there are two problems:

1) Surface of contact is over a narrow area around the lineof contact; this area changes with applied torque, making the problem nonlinear.

2) As is evident from Hertz formulas for bearing stresses between non conformal cylinders, the contact area around the line of contact is very "narrow," requiring very refined mesh density.

The method that is found practical is to assume the contact to be along the "line" derived from involute geometry. Displacement boundary conditions are given along the direction of tangency of the pinion and gear-base circles at the nodes identified along these "lines." The reactive forces at these nodes from the finite-element analysis can be converted to distributed line load intensities. The contact stresses then can be computed using standard Hertz formulas for spur or helical gears. This procedure does not take into account the effect of the Hertzian contact deformation on the distributed line load intensity. For flexible gear systems used in aerospace applications, this effect is not considered significant.

High contact ratio gears have been demonstrated to provide significant advantages for decreasing tooth root and contact stresses with potential flow-on benefits for increased load carrying capacity. Previous investigations with high contact ratio gears have involved analytical, numerical and experimental aspects. Much of the earlier numerical work using FEA was limited in its usefulness due to several factors; (i) the difficulty in predicting load sharing over roll angles covering two or three teeth simultaneously in mesh, (ii) the difficulty for the analysis to obtain quality results when modelling Hertzian contact deflection simultaneously with the bending, shear and angular deflections, and (iii) the problem of primary unconstrained body motion when (long) profile modifications were applied.

A gear wheel is made of substructures which have identical geometry and material properties and are connected to each other in the circumferential direction. Thus a gear may be considered rotationally periodic structure. If cyclic symmetry concept is used in the analysis of such structures, a signifcant reduction in computational effort can be effected. In the present analysis, only one substructure (tooth) is assumed to have a contact line load. A line load on one substructure and zero loads on all other substructures give rise to an asymmetric loading system. A cyclic symmetric structure subjected to asymmetric loading can be analyzed as discussed by Ramamurti. This loading system is decomposed into finite Fourier series with the number of harmonics being equal to the number of teeth. One substructure of spur gear teeth is discretised as shown in Fig 1. The details of the spur gear wheel are given in Tables 1. One substructure is treated as a three-dimensional stress problem with 570 nodes and 84 twenty-noded elements (Fig. 1). Due to computer core limitations a finer mesh could not be used in the fillet area. The material is assumed to be isotropic and homogeneous. Element stiffness and load matrices are assembled to obtain the matrices of one repeatable substructure.

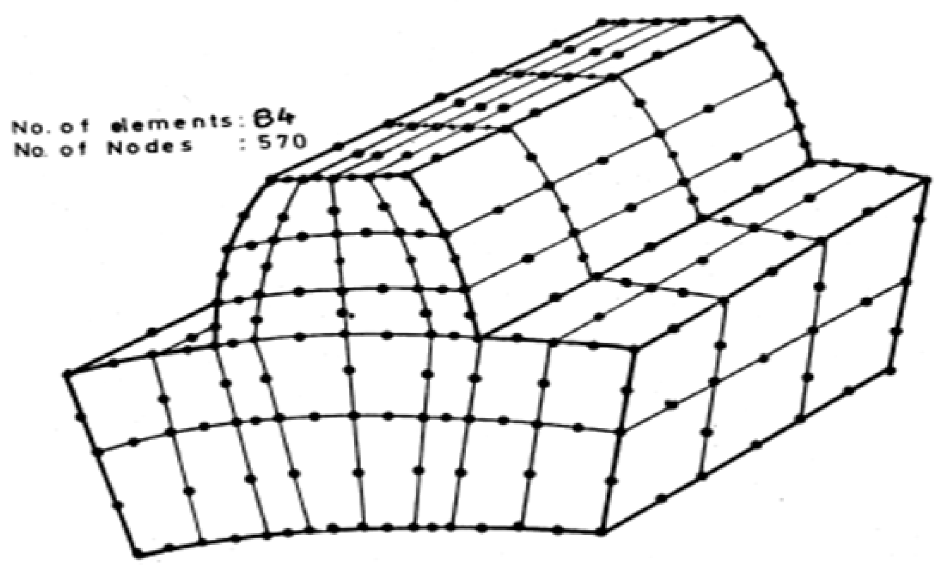


Fig 3.1 Discretisation of spur gear tooth (20 noded element)

Fourier expansion of forces

The gear wheel is a rotationally periodic structure with N identical substructures, where N is the number of teeth of the wheel as mentioned earlier. The force is assumed to act in one sub-structure. In general, the force acting on the k^{th} DOF of the j^{th} substructure of a rotationally periodic structure can be expanded in the form of a finite Fourier series given by

$$a_{jk} = \sum_{p=1}^{N} f_{pk}\ e^{i(j-1)p¥} \qquad \text{,where } ¥ = 2\pi/N \tag{3.1}$$

and f_{pk} is the pth Fourier harmonic of the force corresponding to a spatial phase difference of $2\pi p/N$ between adjacent substructures and may be complex. Equation (1) is applicable to any arbitrary distribution of forces acting on the structure.

Expanding Equation (1),

$$\begin{Bmatrix} a_{1k} \\ a_{2k} \\ \dots \\ a_{Nk} \end{Bmatrix} = \begin{bmatrix} 1 & 1 & \dots 1 \\ e^{i\psi} & e^{2i\psi} & \dots 1 \\ e^{2i\psi} & e^{4i\psi} & \dots 1 \\ e^{i(N-1)\psi} & \dots & \dots 1 \end{bmatrix} \begin{Bmatrix} f_{1k} \\ \dots \\ \dots \\ f_{Nk} \end{Bmatrix} \tag{3.2}$$

Hence the Fourier harmonic of the force on the k^{th} degree of freedom will be given by,

$$\begin{Bmatrix} f_{1k} \\ f_{2k} \\ \dots \\ f_{NK} \end{Bmatrix} = \frac{1}{N} \begin{bmatrix} e^{-i\psi} & e^{-2i\psi} & e^{-i(N-1)\psi} & 1 \\ e^{-i2\psi} & e^{-4i\psi} & e^{-i2(N-1)\psi} & 1 \\ & & & \\ 1 & 1 & 1 & 1 \end{bmatrix} \begin{Bmatrix} a_{1k} \\ a_{2k} \\ \\ a_{NK} \end{Bmatrix} \tag{3.3}$$

As a special case, when only one substructure experiences the force (as in the case of gear wheel) and all other substructures experience no forces, Equation (1) can be written as

$$\begin{Bmatrix} a_{1k} \\ 0 \\ \dots \\ \dots \\ 0 \end{Bmatrix} = \begin{bmatrix} 1 & 1 & 1 \\ e^{i\psi} & e^{i2\psi} & 1 \\ \dots & \dots & 1 \\ \dots & \dots & 1 \\ e^{i(N-1)\psi} & \dots & 1 \end{bmatrix} \begin{Bmatrix} f_{1k} \\ f_{2k} \\ \dots \\ \dots \\ f_{Nk} \end{Bmatrix} \tag{3.4}$$

Hence

$$f_{1k} = f_{2k} = \dots\dots\dots = f_{Nk} = a_{1\ k/n} \tag{3.5}$$

The equation of equilibrium of the full structure obtained from finite element analysis is given by

$$[K]\{\delta\} = \{f\} \tag{3.6}$$

[K] is of size (Nm,Nm) and $\{\delta\}$ and {f} are of size (Nm,1) where N is the number of substructures and m the number of degrees of freedom per substructure.

The resulting system of equations is to be solved by expressing the force system in terms of N finite Fourier components. In other words, each term in $\{\delta\}$ of equation (6) is a finite series having N terms. Let each one of the terms is given by $\{\delta_p\}$ of size (Nm,1).

Since all the substructures are identical and are subjected to forces which differ from those on the substructure by the same phase multiplier exp(i2πp/N), the effect on the neighboring substructures must be related in the same way.

Thus $\{\delta\}$ can be expressed as,

$$\{\delta\} = \left\{ \begin{array}{c} \delta'_p \\ \delta'_p e^{i(2\pi p/N)} \\ \cdots \\ \cdots \\ \delta'_p e^{i(N-1)(2\pi p/N)} \end{array} \right\} \tag{3.7}$$

Where $\{\delta'_p\}$ is the deformation of the first substructure. Hence for the first substructure the matrix equation can be simplified and can be written in the form

$$[K']\{\delta'_{1p}\} = \{f'_p\} \tag{3.8}$$

For any Fourier index p. Here $[K']$ will be of size (m, m) and $\{\delta'_p\}$ and $\{f'_p\}$ of size (m,1).

Let us assume that we have solved Equation 8for each Fourier harmonic p; the resultant displacement vector of the first substructure is given by

$$\{\delta_1\} = \sum_{p=1}^{n} \{\delta'_p\} \tag{3.9}$$

The deflection of any other substructure j is therefore given by

$$\{\delta_j\} = \sum_{p=1}^{n} \{\delta'_p\} e^{i(j-1)(2\pi p/N)}$$

(3.10)

Frequency analysis

The mathematical approach to the concept of cyclic symmetry used in the frequency analysis of rotationally periodic structures has been discussed. The use of cyclic symmetry concept effects a large saving in memory and reduction in computational effort. The method proposed by Balasubramanian and Ramamurti is extended to the free vibration of the spur gear tooth. The simultaneous iteration scheme is modified to compute the Hermitian eigen value problem. An out of core technique is adopted. Lumped mass approach is used in the analysis. This approach not only results in faster convergence of Eigen values, but also reduces the computational effort. The element mass is assumed to be lumped at the nodes of the brick elements. The rotary inertia terms are neglected. Thus, a diagonal mass matrix is obtained for the entire substructure.

CHAPTER 4 CONTACT STRESS SIMULATIONS OF TWO CYLINDERS

4.1 PROBLEMS IN SOLVING CONTACT PROBLEMS

Despite the importance of contact in the mechanics of solids and its engineering applications, contact effects are rarely seriously taken into account in conventional engineering analysis, because of the extreme complexity involved. Mechanical problems involving contacts are inherently nonlinear. Why is it “nonlinear” behavior? Usually the loading causes significant changes in stiffness, which results in a structure that is nonlinear. Nonlinear structural behavior arises for a number of reasons, which can be reduced to three main categories: (1) Geometric Nonlinearities (Large Strains, Large Deflections) (2) Material Nonlinearities (Plasticity) (3) Change in Status Nonlinearities (Contact). So the contact between two bodies belongs to the case (3).

Why is the contact problem significantly difficult? Contact problems present many difficulties. First, the actual region of contact between deformable bodies in contact is not known until the solution has been obtained. Depending on the loads, materials, and boundary conditions, along with other factors, surfaces can come into and go out of contact with each other in a largely unpredictable manner. Secondly, most contact problems need to account for friction. The modeling of friction is very difficult as the friction depends on the surface smoothness, the physical and chemical properties of the material, the properties of any lubricant that might be present in the motion, and the temperature of the contacting surfaces. There are several friction laws and models to choose from, and all are nonlinear. Frictional response can be chaotic, making solution convergence difficult (ANSYS). In addition to those difficulties, many contact problems must also address multi-field effects, such as the conductance of heat and electrical currents in the areas of contact. Bodies in contact may have complicated geometries and material properties and may deform in a seemingly arbitrary way.

With the rapid development of computational mechanics, however, great progress has been made in numerical analysis of the problem. Using the finite element method, many contact problems, ranging from relatively simple ones to quite complicated ones, can be solved with high accuracy. The Finite Element Method can be considered the favorite method to treat contact problems, because of its proven success in treating a wide range of engineering problem in areas of solid mechanics, fluid flow, heat transfer, and for electromagnetic field and coupled field problems.

4.2 HOW TO SOLVE THE CONTACT PROBLEM

4.2.1 Contact Problem Classification

There are many types of contact problems that may be encountered, including contact stress, dynamic impacts, metal forming, bolted joints, crash dynamics, assemblies of components with interference fits, etc. All of these contact problems, as well as other types of contact analysis, can be split into two general classes (ANSYS),

1. Rigid - to - flexible bodies in contact,
2. Flexible - to - flexible bodies in contact.

In rigid - to - flexible contact problems, one or more of the contacting surfaces are treated as being rigid material, which has a much higher stiffness relative to the deformable body it contacts. Many metal forming problems fall into this category. Flexible-to-flexible is where both contacting bodies are deformable. Examples of a flexible-to-flexible analysis include gears in mesh, bolted joints, and interference fits.

4.2.2 Types of Contact Models

In general, there are three basic types of contact modeling application as far as ANSYS use is concerned.

1) Point-to-point contact: the exact location of contact should be known beforehand. These types of contact problems usually only allow small amounts of relative sliding deformation between contact surfaces.

2) Point-to-surface contact: 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. Also, opposing meshes do not have to have the same discretisation or a compatible mesh. Point to surface contact was used in this chapter.

3) Surface-to-surface contact is typically used to model surface-to-surface contact applications of the rigid-to-flexible classification. It will use in chapter 6.

4.2.3 How to Solve the Contact Problem

In order to handle contact problems in meshing gears with the finite element method, the stiffness relationship between the two contact areas is usually established through a spring that is placed between the two contacting areas. This can be achieved by inserting a contact element placed in between the two areas where contact occurs.

There are two methods of satisfying contact compatibility: (i) a penalty method, and (ii) a combined penalty plus a Lagrange multiplier method. The penalty method enforces approximate compatibility by means of contact stiffness. The combined penalty plus Lagrange multiplier approach satisfies compatibility to a user-defined precision by the generation of additional contact forces that are referred to as Lagrange forces.

It is essential to prevent the two areas from passing through each other. This method of enforcing contact compatibility is called the penalty method. The penalty allows surface penetrations, which can be controlled by changing the penalty parameter of the combined normal contact stiffness. If the combined normal contact stiffness is too small, the surface penetration may be too large, which may cause unacceptable errors. Thus the stiffness must be big enough to keep the surface penetrations below a certain level. On the other hand, if the penalty parameter is too large, then the combined normal contact stiffness may produce severe numerical problems in the solution process or simply make a solution impossible to achieve. For most contact analyses of huge solid models, the value of the combined normal contact stiffness may be estimated [ANSYS] as,

$$K_n = fEh \tag{4.1}$$

where f is a factor that controls contact compatibility. This factor is usually be between 0.01 and 100,

E = smallest value of Young's Modulus of the contacting materials
h = the contact length

The contact stiffness is the penalty parameter, which is a real constant of the contact element. There are two kinds of contact stiffness, the combined normal contact stiffness and the combined tangential or sticking contact stiffness. The element is based on two stiffness values. They are the combined normal contact stiffness k_n and the combined tangential contact stiffness k_t. The combined normal contact stiffness k_n is used to penalize interpenetration between the two bodies, while the combined tangential contact stiffness $_t k$ is used to approximate the sudden jump in the tangential force, as represented by the Coulomb friction when sliding is detected between two contacting nodes. However, serious convergence difficulties may exist during the vertical loading process and application of the tangential load often results in divergence. A detailed examination of the model's nodal force during the vertical loading may indicate the problem. Not only are friction forces developing but they develop in random directions. This is due to Poisson's effect causing small transverse deflections of the nodes in the contact zone. These deflections are enough to activate the friction forces of the contact elements. The friction forces are developing in various directions because the generation of a tangential friction force facing right on one node would tend to pull the node on its left to the right. This would generate a friction force facing left on this node, pulling back on the other node. This continual tug-of-war causes the poor convergence. This problem was eliminated by applying a small rotation to the above cylinder model forces as it was displaced and loaded vertically. This rotation ensured that the friction forces would develop in the proper direction.

Because of the simplicity of their formulation, the advantages of using contact elements are:

- They are easy to use
- They are simple to formulate, and
- They are 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.

In overcoming convergence difficulties, usually the biggest challenge is that the solution must start within the radius of convergence. However, there is no simple way to determine the radius of convergence. If the solution converges, the start was within the radius. If solution fails to converge, the start was outside the radius. Trial-and-error must be used to obtain convergence. In order to get convergence in ANSYS, difficult problems might require many load increments, and if many iterations are required, the overall solution time increases. Balancing expense versus accuracy: All FEA involves a trade-off between expense and accuracy. More detail and a finer mesh generally 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. One must use their own engineering judgment to determine how much accuracy is needed versus how much expense can be afforded.

4.2.5 Numerical Example - Contact Problem of Two Circular Discs

First, to investigate the accuracy of the present method, two circular elastic discs under two-dimensional contact are analyzed, and the numerical solutions are compared with that of the Hertz theory. The calculation is carried out under a plane strain condition with a Poisson's ratio of 0.3 using eight-node iso-parametric elements.

Consider two circular discs, A and B, with a radius of R1= 3mm. and R2= 3mm. as shown in Figure 3.2. To reduce the number of nodes and elements and to save more computer memory space, half of the discs are partitioned to the finite element mesh, the number of elements and nodes for each disc is 1766 and 1281, respectively.

In this problem, two steel cylinders are pressed against each other. This model was built based on the Hertz contact stress theoretical problem. The radii were calculated from the pitch diameters of the pinion and gear and other parameters shown in Table 3.1and Figure 3.2. The contact stress of this model should represent the contact stress between two gears. In the input file, first, the geometry of two half cylinders, must be described. Then the geometry areas were meshed. In contact areas a fine mesh was built.The boundary conditions were applied in this model. The loads also were applied four times as four steps. In each step there are a lot of sub-steps. In each sub-step the number of equilibrium iterations was set. The steel material properties have an elastic Young's modulus of 30,000,000 psi and the Poisson's ratio was 0.30.

Table 3.1 Specifications of spur gears used

Number of teeth	25
Normal Module (M)	6 mm
Addendum Modification coefficient	0
Normal Pressure Angle	20 degrees
Face Width (mm)	0.015 M
Addendum (mm)	1.00 M
Dedendum (mm)	1.25 M

4.3 Hertz Contact Stress Equations

Usually, the current methods of calculating gear contact stresses use Hertz's equations, which were originally derived for contact between two cylinders. Contact stresses between two cylinders were shown in Figure 4.2. An ellipsoidal-prism pressure distribution is generated between the two contact areas.

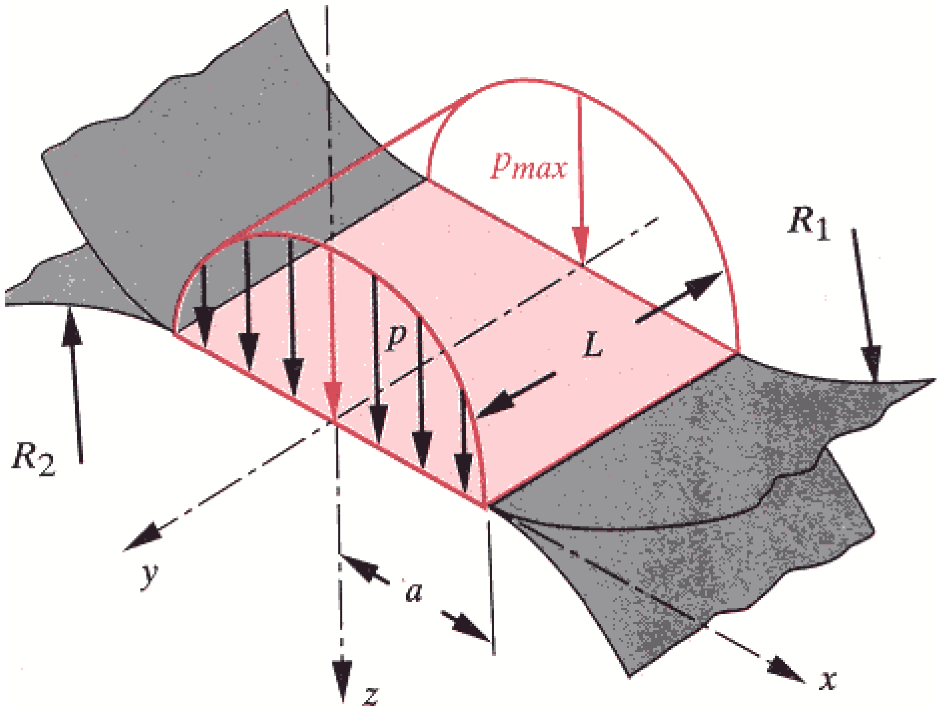


Figure 4.1 Ellipsoidal-prism pressure distributions

From Figure 4.2 the width of the contact zone is 2a. If total contact force is F and contact pressure is p(x), there is a formula, which shows the relationship between the force F and the pressure p(x):

$$F = 2\,L \int_0^a p\,(x)dx \tag{4.2}$$

Contact width
$$a = \frac{\sqrt{\left(\frac{2F}{\pi L}\right)\left(\frac{1-v_1^2}{E_1}+\frac{1-v_2^2}{E_2}\right)}}{\sqrt{\left(\frac{1}{d_1}+\frac{1}{d_2}\right)}} \tag{4.3}$$

The maximum contact stress
$$p_{max} = \frac{2F}{\pi a L} \tag{4.4}$$

d_1 and d_2 represent the pinion and gear pitch diameters.

The maximum surface (Hertz) stress:

$$p_{max}=\ \sigma_H=0.564\ \frac{\sqrt{F(\frac{1}{R_1}+\frac{1}{R_2}}}{\sqrt{(\frac{1-v_1^2}{E_1}+\frac{1-v_2^2}{E_2}}} \tag{4.5}$$

F is the load per unit width,

R_i is the radius of cylinder i , $R_i = d_i \sin \varphi/2$ for the gear teeth,

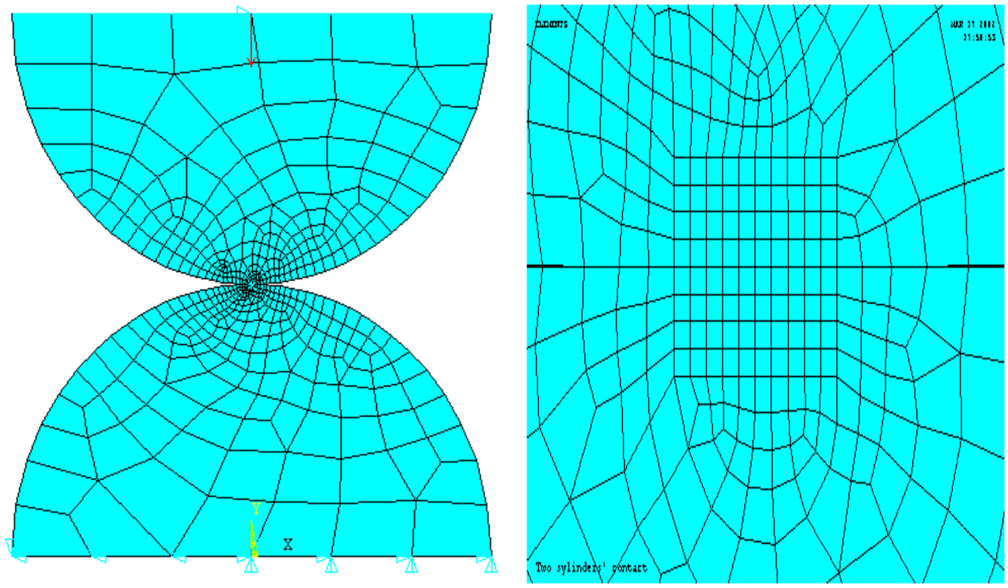
φ is pressure angle

v_i is Poisson's ratio for cylinder i ,

E_i is Young's modulus for cylinder i .

4.4 The Result of the Contact Stress Analysis

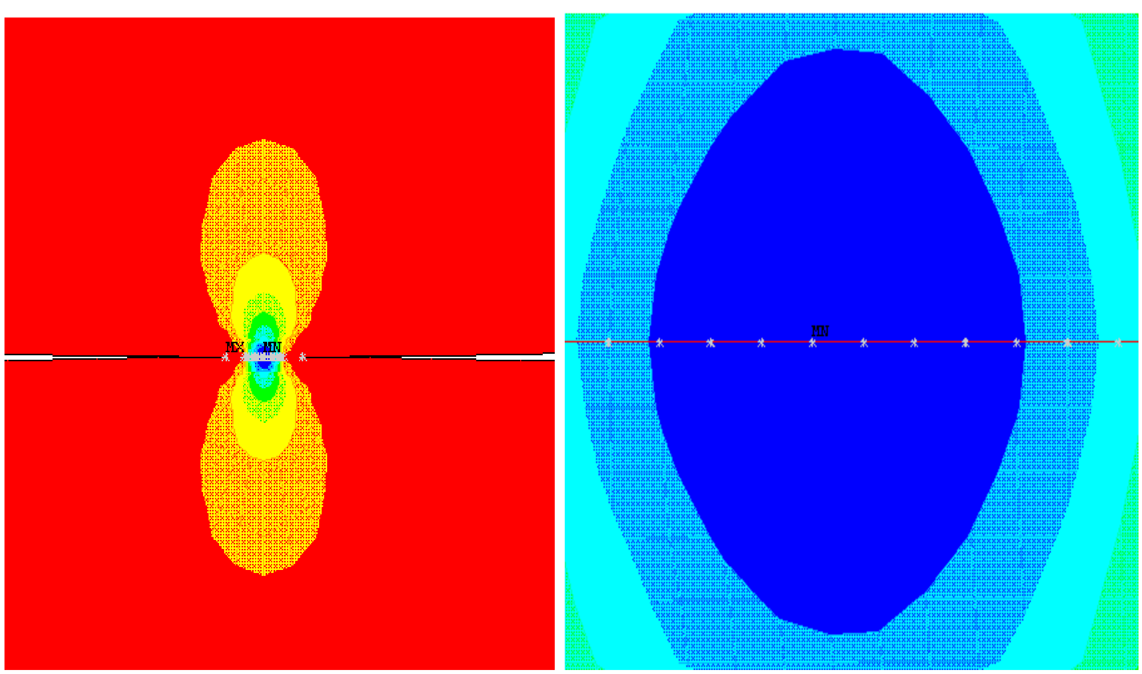
The objective of the contact stress analyses was to gain an understanding of the modeling and solution difficulties in contact problems and examine the contact stresses in the gears. In order to verify the FEM contact model procedure, contact between two cylinders was modeled. To reduce computer time, only half cylinders were meshed in the model as shown in Figure 4.3(a). The fine meshed rectangular shaped elements were generated near contact areas shown as 4.3 (b). The dimensions of the elements are based on the half width of the contact area. The contact conditions are sensitive to the geometry of the contacting surfaces, which means that the finite element mesh near the contact zone needs to be highly refined. Finer meshing generally leads to a more accurate solution, but requires more time and system resources. It is recommended not to have a fine mesh everywhere in the model to reduce the computational requirements. The edge length dx of the rectangular shaped fine mesh elements: $dx = 2 * a / Num$. Num is the number of elements in the contact zone as specified in the input file. It was hoped that the number of elements in the Hertz contact zone could be related to the solution accuracy, independent of the specific force or cylinder sizes considered. *Num* is equal to 10 here.



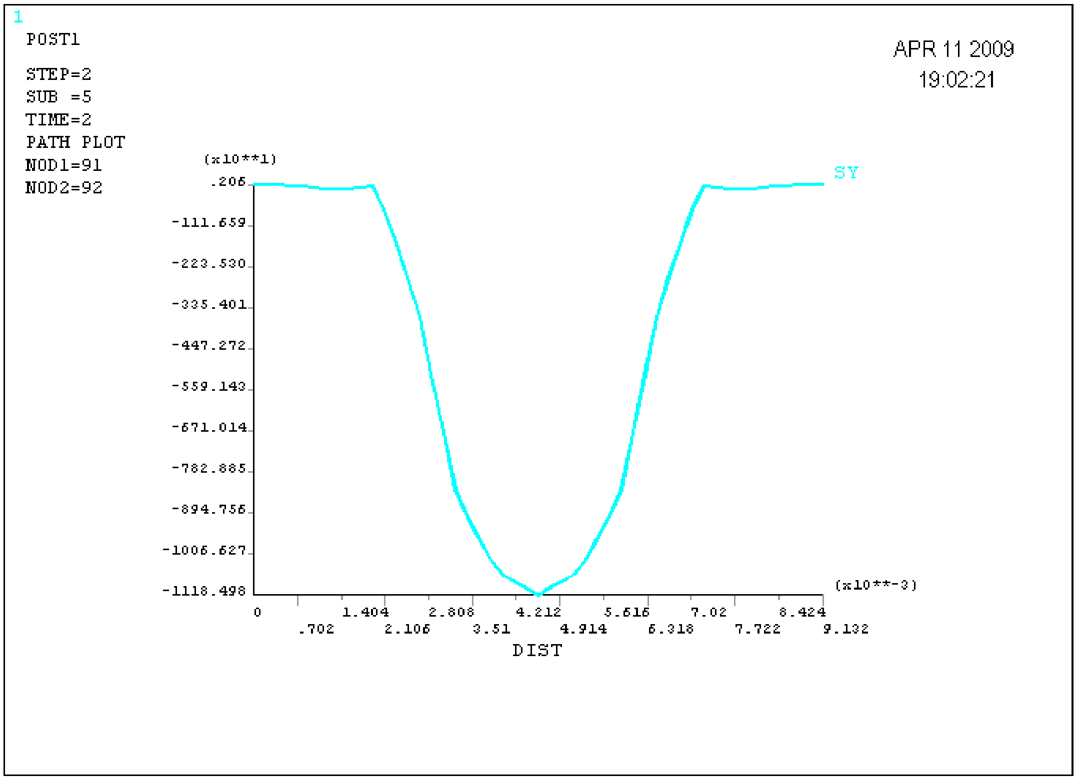
(a) (b)

Figure 4.2 Rectangular shaped elements were generated near contact areas

The normal contact stress along the contact surface from the ANSYS solution is shown in Figure 4.4. Figure 4.4 (a) and (b) show the distributions of the contact stress along the contact area, and Figure 4.4 (c) shows the magnitude directly from ANSYS.



(a) (b)



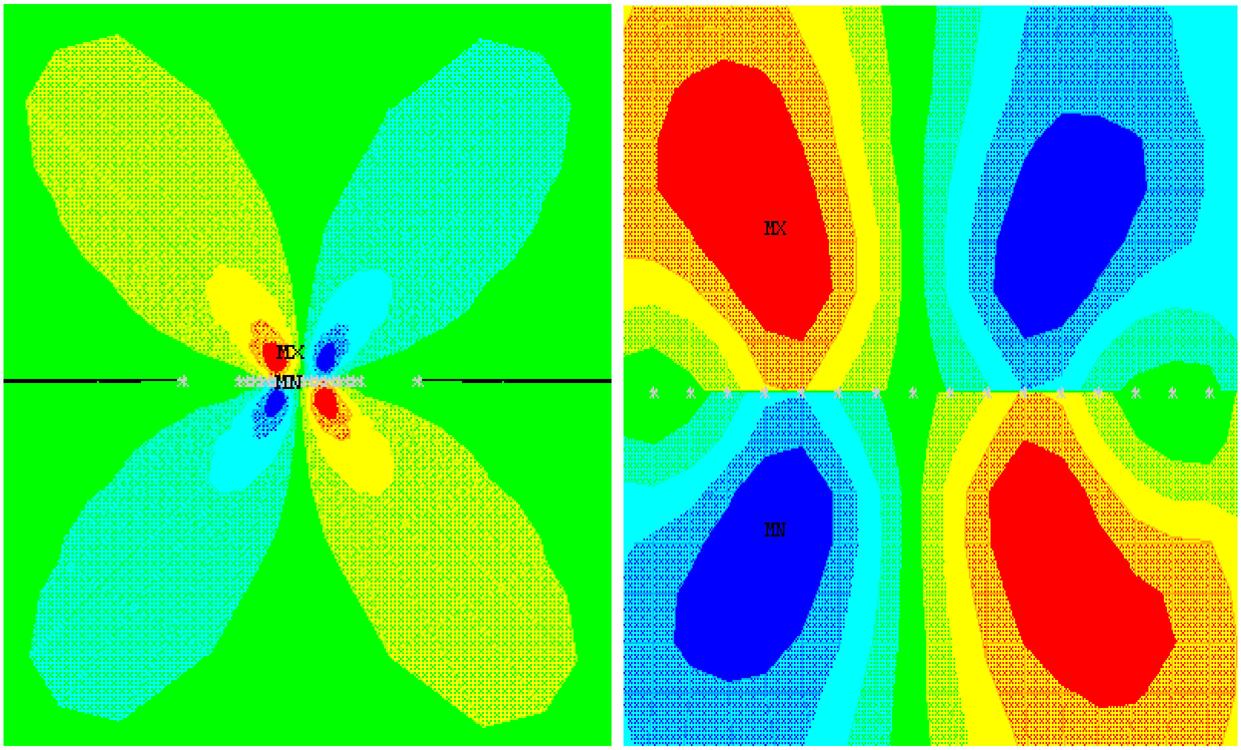
(c)

Figure 4.3 Normal contact stresses along the contact surface

The peak values of the equivalent stress using the Von Mises criterion, the maximum shear stress, and the maximum orthogonal shear stress can be calculated from the maximum Hertz stress (4.5) as follows:

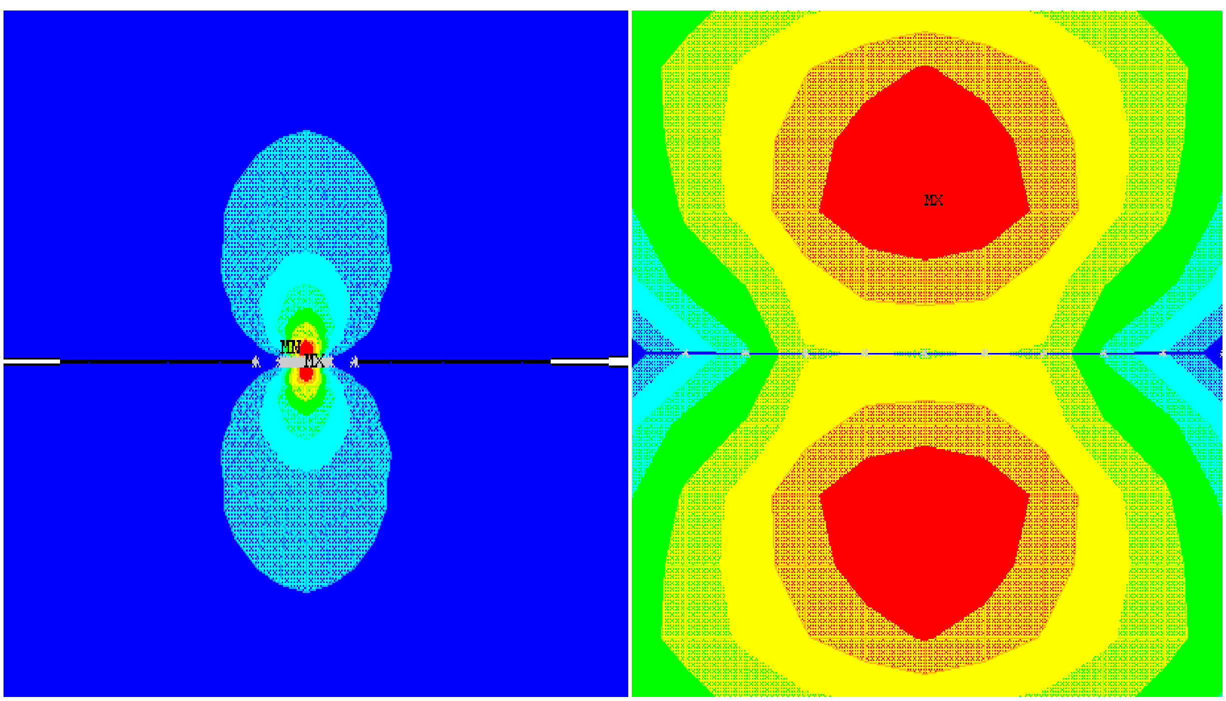
$$\begin{aligned} \sigma_{\text{Von Mises}} &= 0.57\ \sigma_{H} \\ \sigma_{\text{Max Shear}} &= 0.30\ \sigma_{H} \\ \sigma_{\text{Ortho Shear}} &= 0.25\ \sigma_{H} \end{aligned} \qquad (4.6)$$

where σ_H is the maximum Hertz stress.



(a) (b)

Figure 4.4 orthogonal shear stress magnitudes



(a) (b)

Figure 4.5 Maximum shear stress from ANSYS

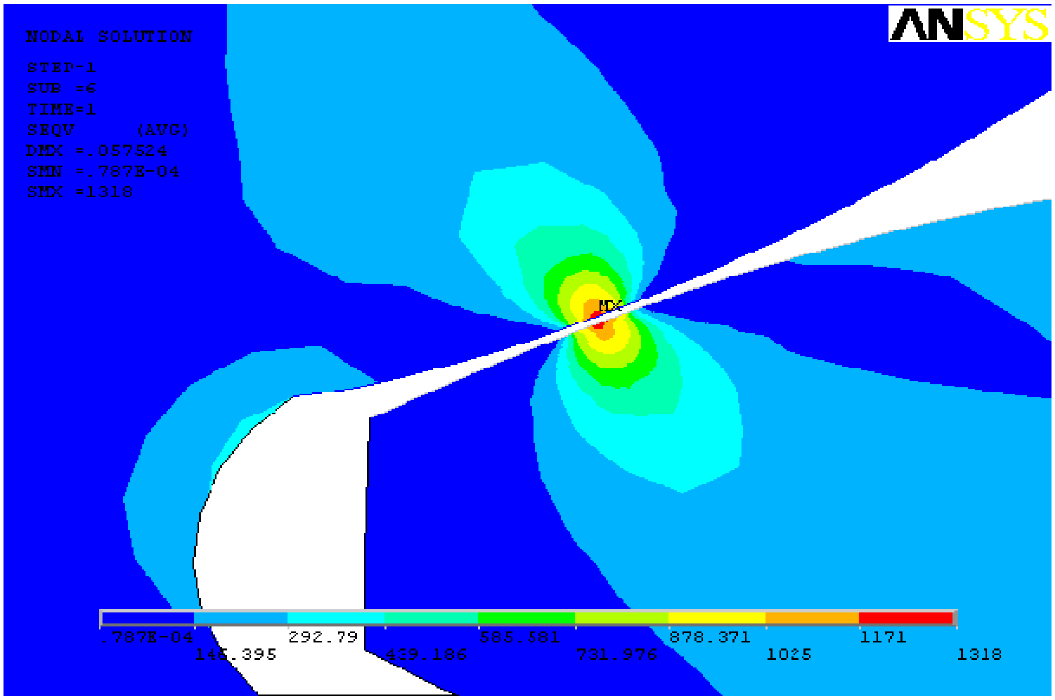


Figure 4-6 Von Mises stresses in spur gears from ANSYS

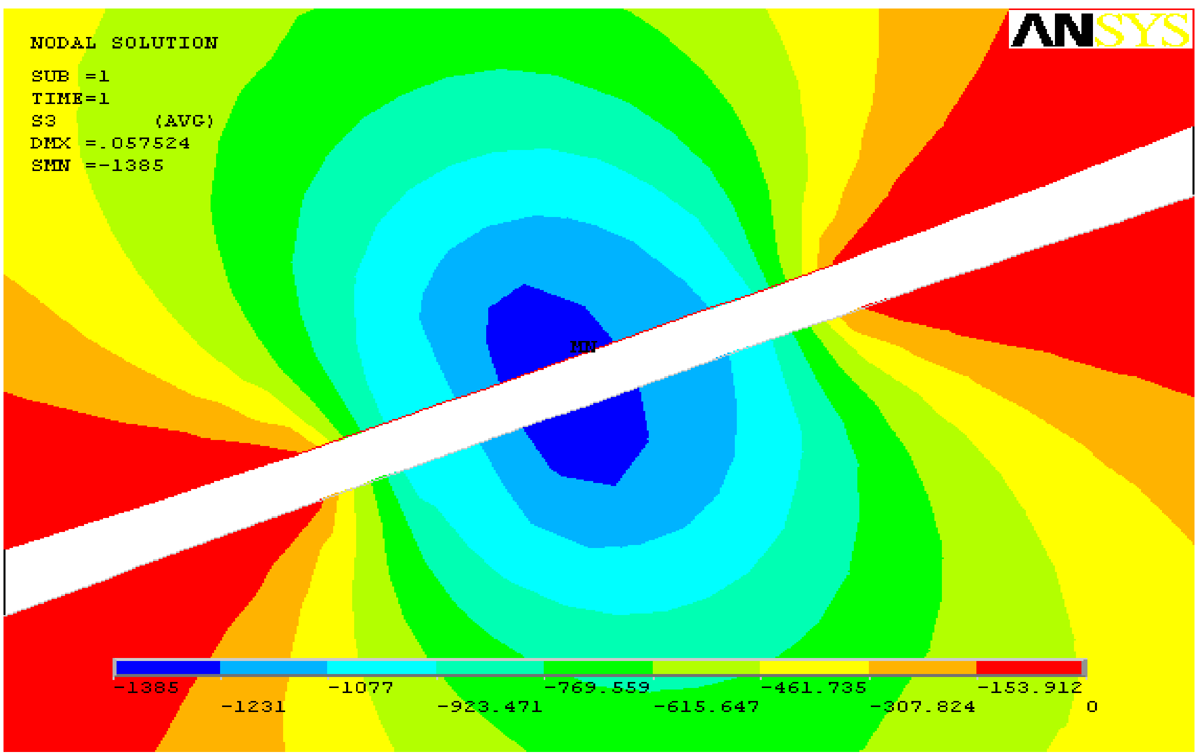


Figure 4-7 the distribution of contact stresses between two teeth

Figures 4.5 (a) and (b) show orthogonal shear stress. Figures 4.6 (a) and (b) show maximum shear stresses under the contact areas between two cylinders. The largest orthogonal shear stress lies below the surface at the edge of the contact zone. This was shown in Figure 4.5 (b). The subsurface location of the maximum shear stress can also be seen lying below the surface at the center of the contact zone shown in Figure 4.6 (b). If both materials are steel, it occurs at a depth of about 0.63 *a* where a is half of the contact length shown in Figure 4.2 and its magnitude is about 0.30 P_{max} . The shear stress is about 0.11 P_{max} at the surface on the *z* axis. The subsurface location of the maximum shear stress is believed to be a significant factor in surface-fatigue failure. The theory indicates that cracks that begin below the surface eventually grow to the point that the material above the crack breaks out to form a pit.

Von Mises stresses and the contact stresses just for one position between two teeth are shown below in Figure 4.7 and Figure 4.8. For the gears the contact stress was compared with the results from the Hertz equations, and the two results agree with each other well. In this model, there are 4751 elements and 5163 nodes. For the contact surfaces there are more than eight nodes on each contact side. So the distribution of contact stresses is resonable. In this chapter the transmission error is emphasized and contact is a nonlinear problem so the solution will likely be done after a greater time compared with the time in linear analysis. It is much simpler to use “WIZARD BAR” and to create contact pair between the contact surfaces from “Preprocessor>Modeling>Create>Contact Pair”.

4.5 Conclusion

Finite element modelling of the contact between two cylinders was examined in detail. The finite element method with special techniques, such as the incremental technique of applying the external load in the input file, the deformation of the stiffness matrix, and the introduction of the contact element were used. It was found that initial loading using displacements as inputs was helpful in reducing numerical instabilities.

CHAPTER 5 BENDING STRESS ANALYSIS

5.1 INTRODUCTION

When one investigates actual gears in service, the conditions of the surface and bending failure are two of the most important features to be considered. The finite element method is very often used to analyze the stress states of elastic bodies with complicated geometries, such as gears. There are published papers, which have calculated the elastic stress distributions in gears. In these works, various calculation methods for the analysis of elastic contact problems have been presented. The finite element method for two-dimensional analysis is used very often. It is essential to use a three-dimensional analysis if gear pairs are under partial and non uniform contact. However, in the three dimensional calculation, a problem is created due to the large computer memory space that is necessary. In this chapter to get the gear contact stress a 2-D model was used. Because it is a nonlinear problem it is better to keep the number of nodes and elements as low as possible. In the bending stress analysis the 3-D model and 2-D models are used for simulation.

5.2 ANALYTICAL PROCEDURE

From the results obtained in chapter 3 the present method is an effective and accurate method, which is proposed to estimate the tooth contact stresses of a gear pair. Special techniques of the finite element method were used to solve contact problems in chapter 4. Using the present method, the tooth contact stresses and the tooth deflections of a pair of spur gears analyzed by ANSYS 7.1 are given in section 5.4. Since the present method is a general one, it is applicable to many types of gears. In early works, the following conditions were assumed in advance:

- There is no sliding in the contact zone between the two bodies
- The contact surface is continuous and smooth.

Using the present method ANSYS can solve the contact problem and not be limited by the above two conditions. A two-dimensional and an asymmetric contact model were built. First, parameter definitions were given and then many points of the profile of the pinion and gear were calculated to plot a profile using a cylindrical system. The equations of a curve below were taken from Buckingham

$$r = r_b * (1+\beta^2)^{1/2} \tag{5.1}$$

$$\psi = \theta + \pi/2n_1 - \xi \tag{5.2}$$

$$\theta = \tan\varphi - \varphi = \text{inv}\,\varphi \tag{5.3}$$

Where r = radius to the involute form, r_b = radius of the base circle

$$\beta = \xi + \varphi$$

θ = vectorial angle at the pitch circle

ξ = vectorial angle at the top of the tooth

φ = pressure angle at the pitch circle

One spur tooth profile was created using equation 5.1, shown in Figure 5.1, as are the outside diameter circle, the dedendum circle, and base circle of the gear.

Secondly, in ANSYS from the tool bars using “CREATE”, “COPY”, “MOVE”, and “MESH” and so on, any number of teeth can be created and then kept as the pair of gear teeth in contact along the line of the action. The contact conditions of gear teeth are sensitive to the geometry of the contacting surfaces, which means that the element near the contact zone needs to be refined. It is not recommended to have a fine mesh everywhere in the model, in order to reduce the computational requirements. There are two ways to build the fine mesh near the contact surfaces. One is the same method as presented in chapter 4, a fine mesh of rectangular shapes were constructed only in the contact areas. The other one, “SMART SIZE” in ANSYS, was chosen and the fine mesh near the contact area was automatically created. A FEM gear contact model was generated as shown in Figure 5.1.

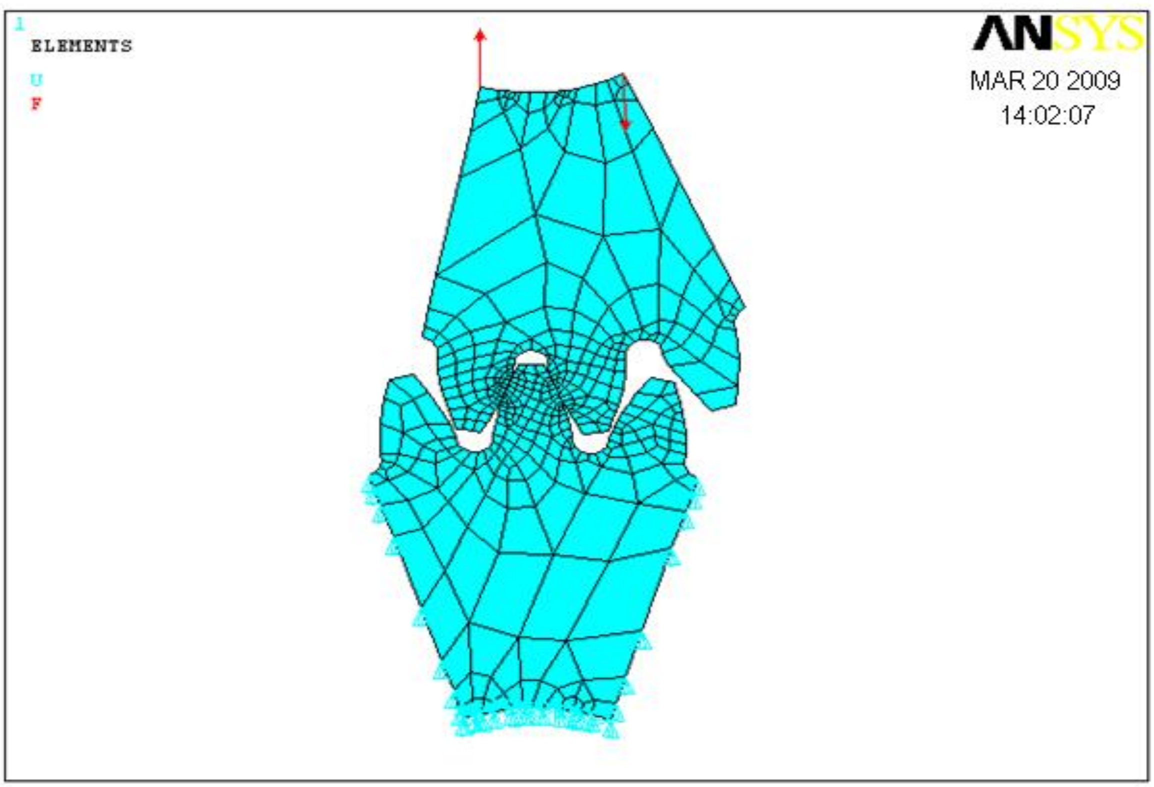


Figure 5.1 Gear contact stress model

Thirdly, proper constraints on the nodes were given. The contact pair was inserted between the involute profiles, the external loads were applied on the model from ANSYS "SOLUTION > DEFINE LOAD > FORCE / MOMENT", and finally, ANSYS was run to get the solution.

5.3 ROTATION COMPATIBILITY OF THE GEAR BODY

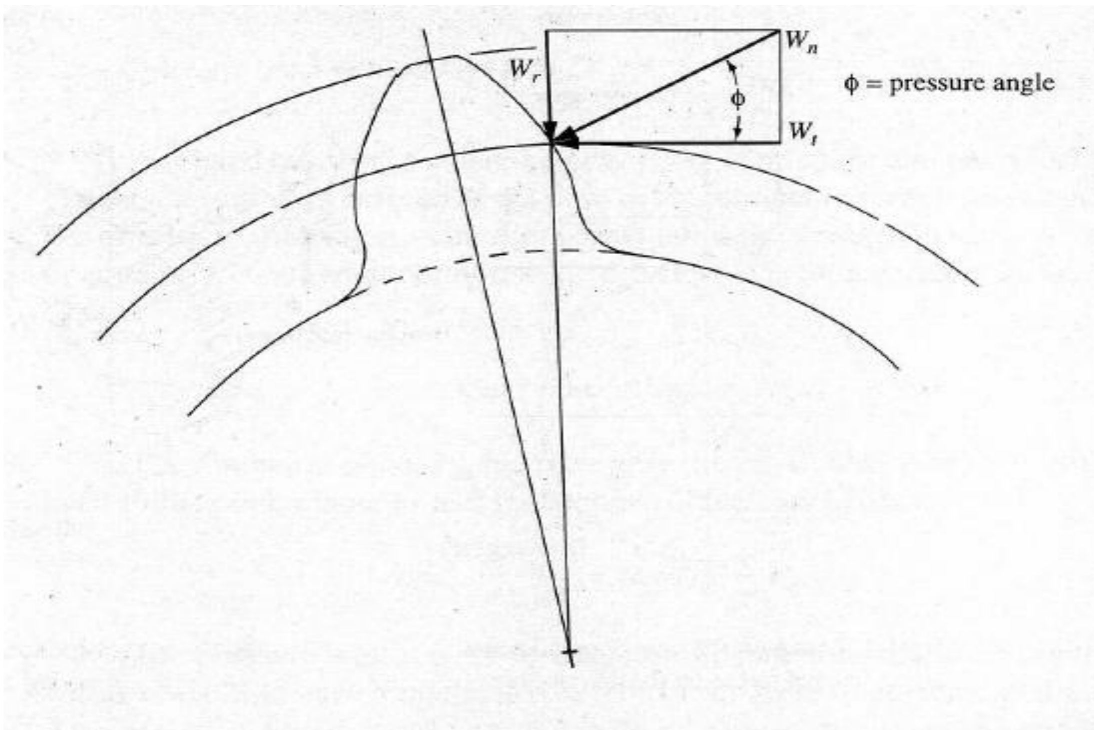


Figure 5.2 Forces on Spur Gear Tooth

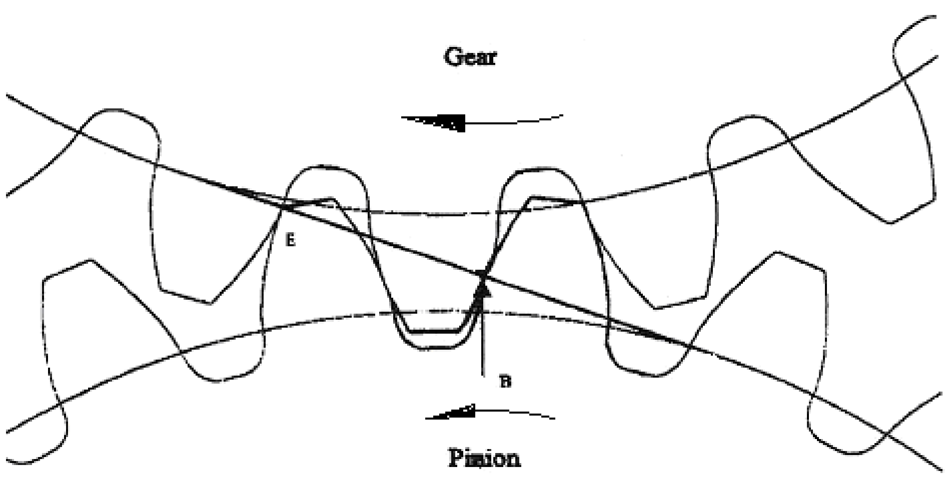


Figure 5.3 Different positions for one complete tooth meshing cycle

In order to know how much load is applied on the contact stress model and the bending stress model, evaluating load sharing between meshing gears is necessary. It is also an important concept for transmission error. It is a complex process when more than one-tooth pair is simultaneously in contact taking into account the composite tooth deflections due to bending, shearing and contact deformation. This section presents a general approach as to how the load is shared between the meshing teeth in spur gear pairs. To simplify the complexity of the problem, the load sharing compatibility condition is based on the assumption that the sum of the torque contributions of each meshing tooth pair must equal the total applied torque.

Analytical equations can also be developed for the rotation of the gear and pinion hubs, including the effects of tooth bending deflection and shearing displacement and contact deformation. In the pinion reference frame, it is assumed that the pinion hub remains stationary, while the gear rotates due to an applied torque.

Considering the single pair contact zone, the condition of angular rotation of the gear body will then be given by

For the pinion,
$$\theta_p = \frac{B_P + H_p}{R_P} \quad (5.4)$$

And for the gear,
$$\theta_g = \frac{B_g + H_g}{R_g} \quad (5.5)$$

where B_p and B_g are the tooth displacement vectors caused by bending and shearing of the pinion and gear respectively, H_p and H_g are the contact deformation vectors of the pinion and gear respectively. θ_P denotes the transverse plane angular rotation of the pinion body caused by bending deflection, shearing displacement and contact deformation of the tooth pair while the gear is stationary. Conversely, for the gear rotation while the pinion is stationary, θ_g gives the transverse plane angular rotations of the gear body.

5.4 GEAR CONTACT STRESS

One of the predominant modes of gear tooth failure is pitting. Pitting is a surface fatigue failure due to many repetitions of high contact stress occurring on the gear tooth surface while a pair of teeth is transmitting power. In other words, contact stress exceeding surface endurance strength with no endurance limits or a finite life causes this kind of failure. The AGMA has prediction methods in common use. Contact failure in gears is currently predicted by comparing the calculated Hertz stress to experimentally determine allowable values for the given material. The details of the subsurface stress field usually are ignored. This approach is used because the contact stress field is complex and its interaction with subsurface discontinuities is difficult to predict. However, all of this information can be obtained from the ANSYS model.

Since a spur gear can be considered as a two-dimensional component, without loss of generality, a plane strain analysis can be used. The nodes in the model were used for the analysis. The nodes on the bottom surface of the gear were fixed. A total load is applied on the model. It was assumed to act on the two points shown in Figure 5.1 and three points in Figure 5.4.

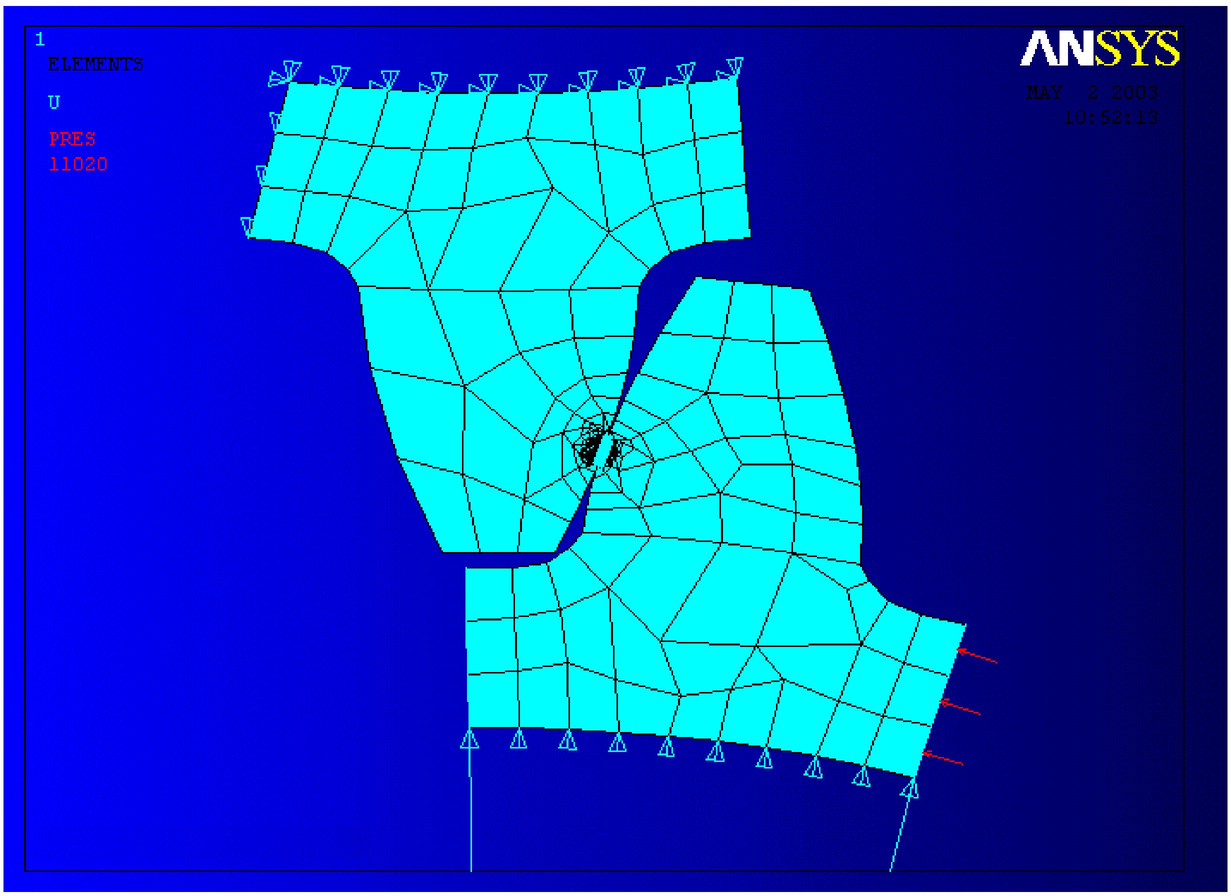


Figure 5.4 FEM Model of the gear tooth pair in contact

There are two ways to get the contact stress from ANSYS. Figure 5.4 shows the first one, which is the same method as one in chapter 4 to create the contact element COCNTA 48 and the rectangular shape fine mesh beneath the contact surfaces between the contact areas.

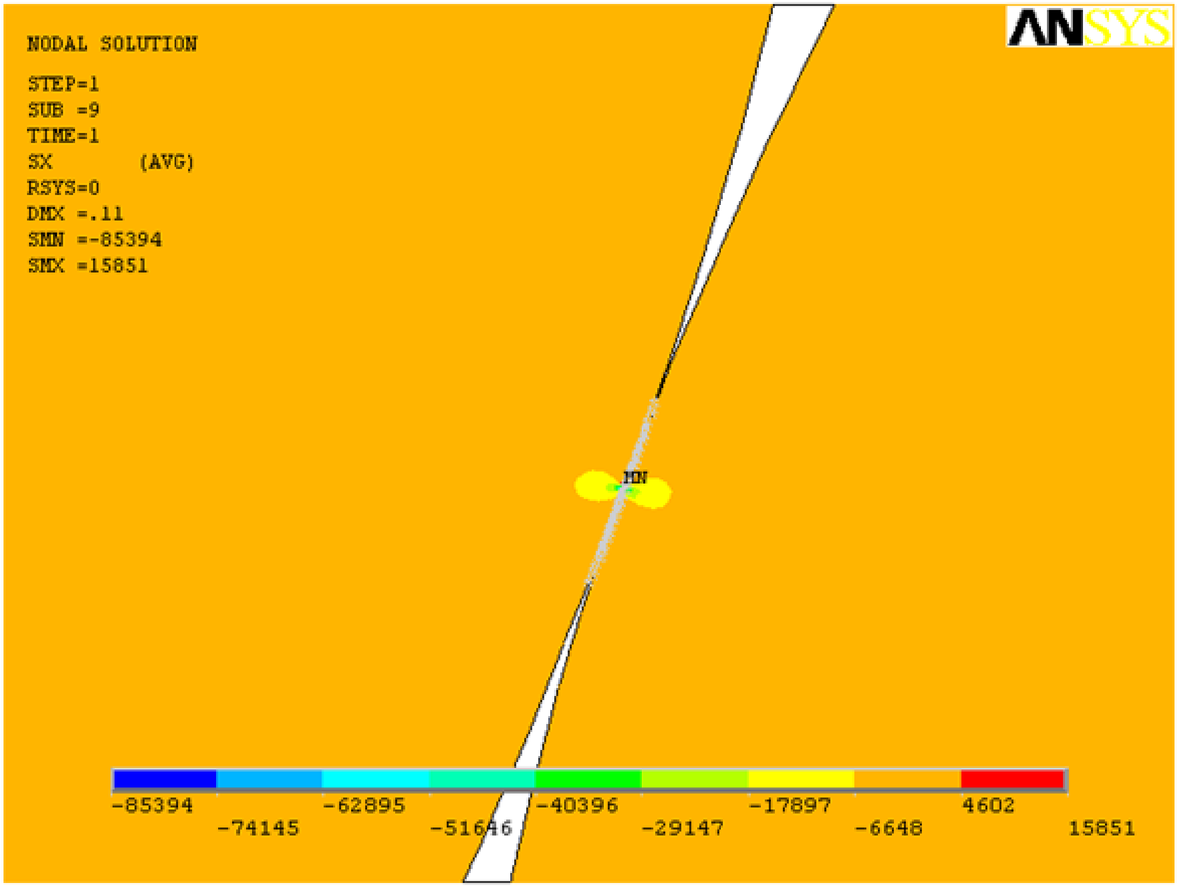


Figure 5.5 Contact stress along contact areas

Figure 5.5 shows the normal contact stress along the contact areas. The results are very similar to the results in the two cylinders in chapter 4. Different methods should show the close results of maximum contact stress if the same dimension of model and the same external loads are applied on the model. If there is a small difference it is likely because of the different mesh patterns and restricted conditions in the finite element analysis and the assumed distribution form of the contact stresses in the contact zone.

5.5 THE LEWIS FORMULA

There are several failure mechanisms for spur gears. Bending failure and pitting of the teeth are the two main failure modes in a transmission gearbox. Pitting of the teeth is usually called a surface failure. This was already discussed in the last section. The bending stresses in a spur gear are another interesting problem. When loads are too large, bending failure will occur. Bending failure in gears is predicted by comparing the calculated bending stress to experimentally-determined allowable fatigue values for the given material. This bending stress equation was derived from the Lewis formula. Wilfred Lewis (1892) was the first person to give the formula for bending stress in gear teeth using the bending of a cantilevered beam to simulate stresses acting on a gear tooth are Cross-section =$b * t$, length = l, load = F_t uniform across the face. For a rectangular section, the area moment of inertia is I= $bh^3/12$.

M = F_t* l and c = t/2, stress then is

$$\sigma = \frac{M}{I/c} = \frac{F_t\, l(\frac{t}{2})}{bt^3/_{12}} = \frac{6F_t\ l}{bt^2} \tag{5.6}$$

Where b= the face width of the gear. For a gear tooth, the maximum stress is expected at the point 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 found 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 start from that point only.

$$\tan \alpha = \frac{t/_2}{x} = \frac{l}{t/_2} \quad \text{where } l = \frac{t^2}{4x} \tag{5.7}$$

Substituting (5.7) into (5.6):

$$\sigma = \frac{6F_t \frac{t^2}{4x}}{bt^2} = 3F_t/2bx = 3F_t p_d/2bp_d x = F_t p_d/bY \tag{5.8}$$

Where p_d = diametral pitch

$$Y = 2xp_d/3 = \text{Lewis form factor} \tag{5.9}$$

Equation (5.10) in the next page is known as the Lewis equation, and *Y* is called the Lewis form factor. The Lewis equation considers 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 20^0 and a full – depth involute. The Lewis form factor is dimensionless, and is also independent of tooth size and only a function of shape. The above stress formula must be modified to account for the stress concentration K_c. The concentrated stress on the tooth fillet is taken into account by K_c and a geometry factor where Y_j= Y/K_c is introduced. Other modifications are recommended by the AGMA for practical design to account for the variety of conditions that can be encountered in service. The following design equation, developed by Mott (1992) is used,

$$\sigma_t = \frac{p_d * K_a * K_{s*}\, K_m * F_t}{b\, Y_j\, K_v} \tag{5.10}$$

where K_a = Application factor , K_s = Size factor ,
K_m = Load distribution factor, K_v = Dynamic factor,
F_t = Normal tangential load, Y_j = Geometry factor.

Each of these factors can be obtained from the books on machine design such as. This analysis considers only the component of the tangential force acting on the tooth, and does not consider effects of the radial force, which will cause a compressive stress over the cross section on the root of the tooth. Suppose that the greatest stress occurs when the force is exerted at top of tooth, which is the worst case. When the load is at top of the tooth, usually there are a least two tooth pairs in contact. In fact, the maximum stress at the root of tooth occurs when the contact point moves near the pitch circle because there is only one tooth pair in contact and this teeth pairs carries the entire torque. When the load is moving at the top of the tooth, two teeth pairs share the whole load if the ratio is larger than one and less than two. If one tooth pair was considered to carry the whole load and it acts on the top of the tooth this is adequate for gear bending stress fatigue.

5.6 FEM MODELS

5.6.1 The 2-Dimensional Model

Fatigue or yielding of a gear tooth due to excessive bending stresses are two important gear design considerations. In order to predict fatigue and yielding, the maximum stresses on the tensile and compressive sides of the tooth, respectively, are required. 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.

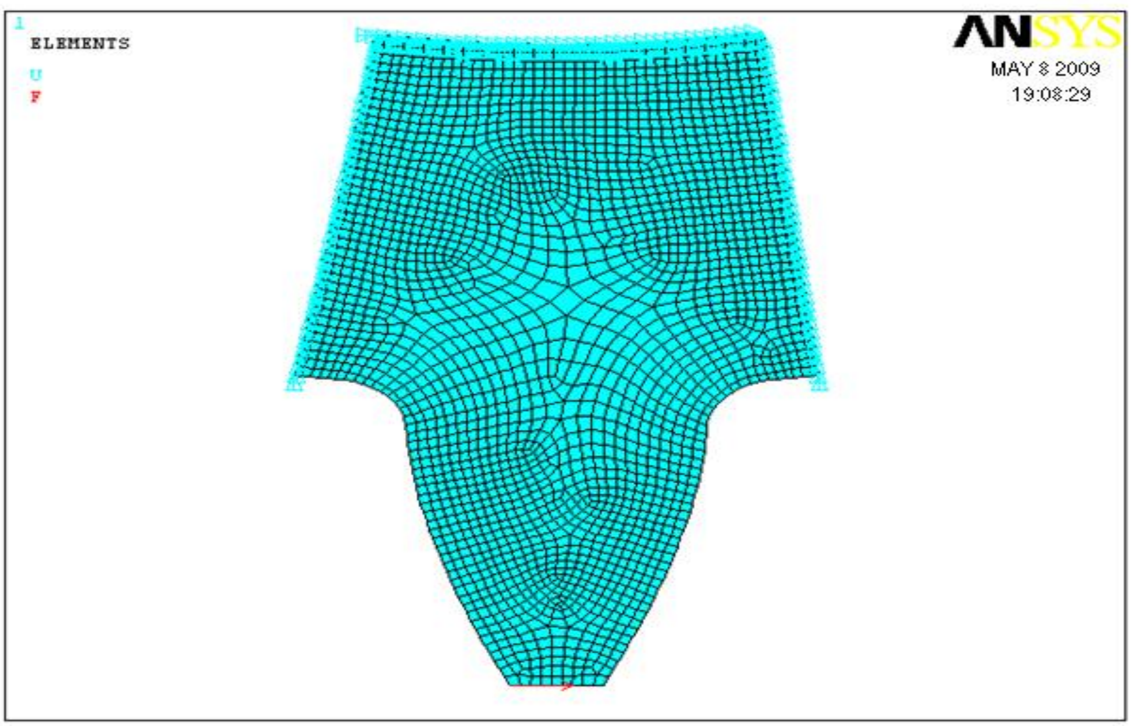


Figure 5.6 a two dimension tooth from a FEM model with 28 teeth

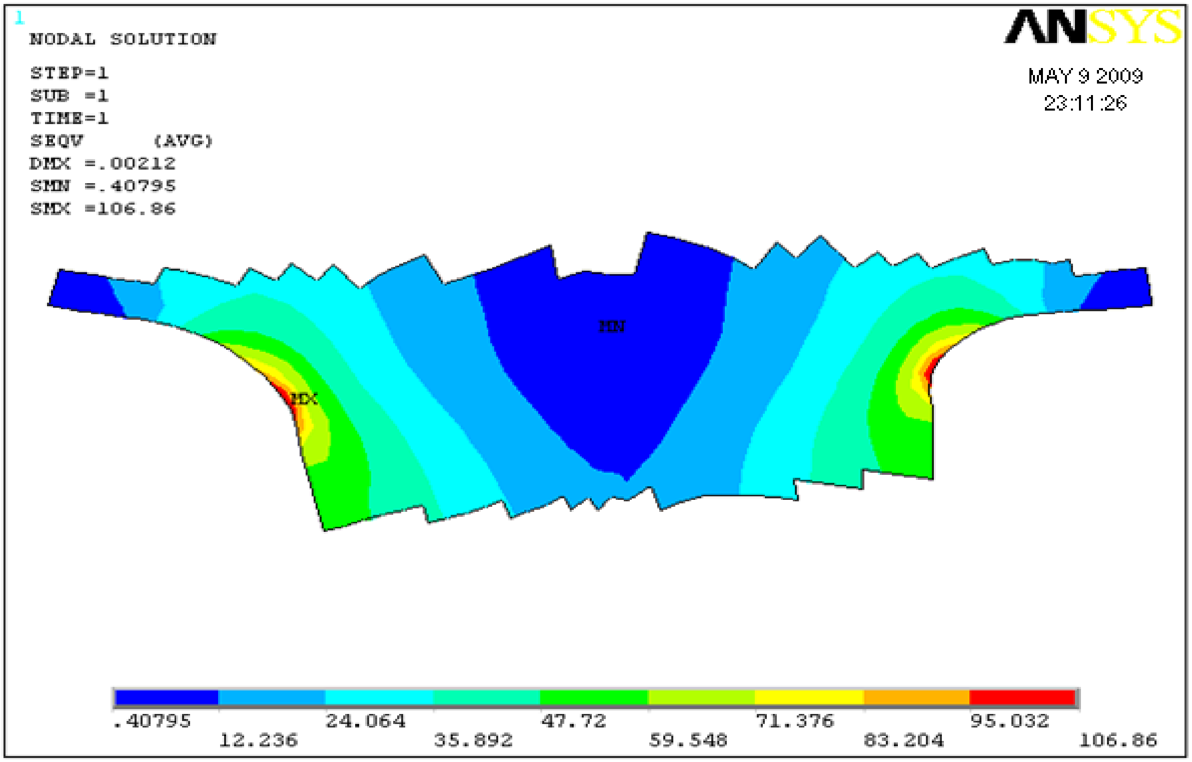


Figure 5.7 Von Mises stresses with 28 teeth on the root of tooth

In the procedure for generating a FEM model for bending stress analyses, the equations used to generate the gear tooth profile curve were the same as the ones in section 5.2. When meshing the teeth in ANSYS, if "SMART SIZE" is used the number of elements near the roots of the teeth are automatically much greater than in other places. Figure 5.6 shows that the maximum tensile stresses on the tensile side and maximum compressive stresses on other side of the tooth, respectively. It also indicates that only one tooth is enough for the bending stress analysis for the 3-D model or the 2-D model. Figure 5.6 shows one tooth FEM model and Figure 5.7 shows how much Von Mises stress is on the root of tooth when the number of teeth is 28 for the gear. There are more detailed results for different number of teeth in table 5.1 in section 5.7, which are compared with the results from the Lewis Formula.

5.6.2 The 3-Dimensional Model

In this section the tooth root stresses and the tooth deflection of one tooth of a spur gear is calculated using an ANSYS model. For the bending stresses, the numerical results are compared with the values given by the draft proposal of the standards of the AGMA in the next section.

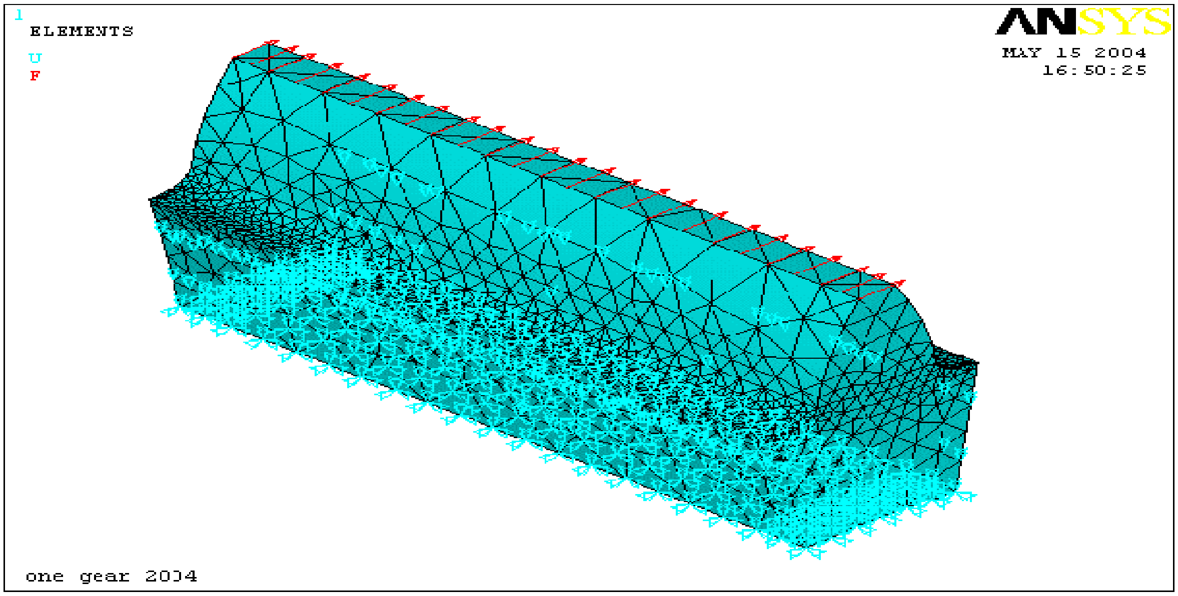


Figure 5.8 FEM 3D bending model with meshing

Figure 5.9 shows how to mesh the 3D model and how to apply the load on the model. The element type "SOLID TETRAHEDRAL 10 NODES 187" was chosen. Because "SMART SET" was chosen on the tool bar there are many more elements near the root of the tooth than in other places. There are middle side nodes on the each side of each element. So a large number of degrees of freedom in this 3D model take a longer time to finish running.

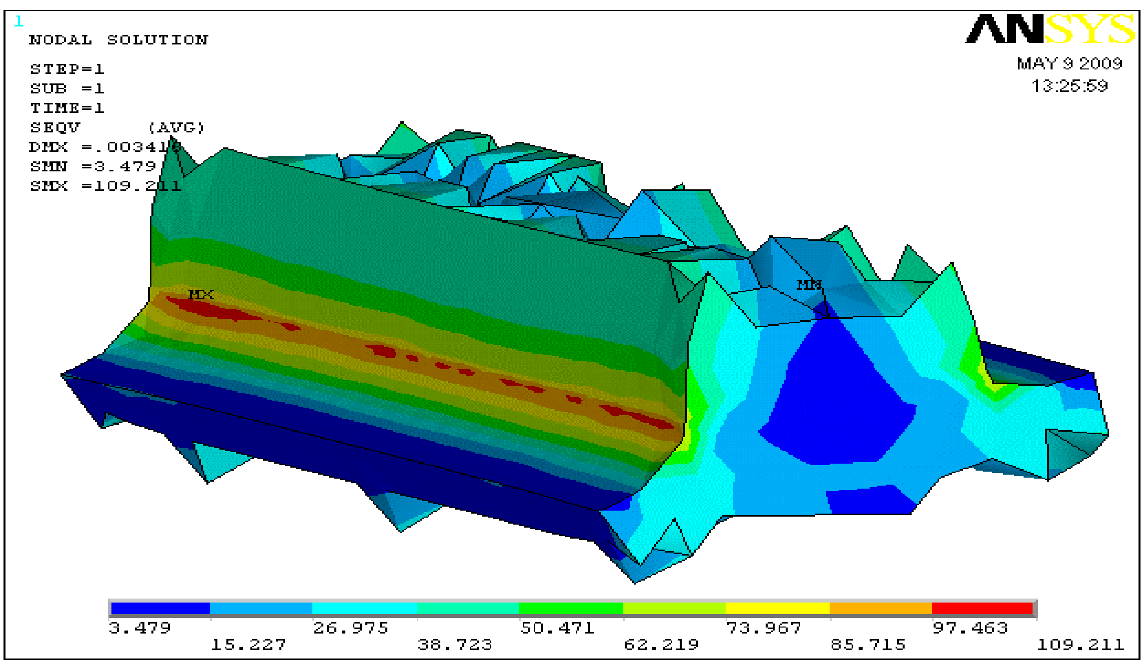


Figure 5.9 Von Mises stresses with 28 teeth on the root of tooth

From the stress distributions on the model, the large concentrated stresses are at the root of the tooth. Figure 5.9 shows large Von Mises stresses at the root of the tooth. They are equal to the tensile stresses. The tensile stresses are the main cause of crack failure, if they are large enough. That is why cracks usually start from the tensile side. From the Lewis equation if the diameters of the pinion and gear are always kept the same and the number of teeth was changed, the diametral pitch will be changed or the module of gear will be changed. That means that there are different bending strengths between the different teeth numbers. Different Maximum Von Mises with different numbers of teeth are shown in the table 5.1.

5.7 COMPARISON WITH RESULTS USING AGMA ANALYSES

In this section, a comparison of the tooth root stresses obtained in the three dimensional model and in the two dimensional model using ANSYS with the results given by the standards of the AGMA is carried out. Eq. (5.10) is recommended by the AGMA and the other coefficients, such as the dynamic factor, are set at 1.2. Here analysis of gears with different numbers of teeth is carried out. First, the number of gear teeth is 28. The meshing spur gear has pitch radii of 50 mm and a pressure angle of 20◦. The gear face width, b = 8 mm. The transmitted load is 2500 N.

$$p_d = N/d = 28/\ 50*2 = 0.28$$

$$\sigma_t = \frac{p_d * K_a * K_{s*}\ K_{m\ *} F_t}{b\ Y_{j\ K_v}} = 2500*0.28*1.2*1.2*1.15/38*0.37*0.8$$

$$= 103.05 \text{ MPa}$$

Detailed investigations, including the effects with the two different numbers of teeth on the tooth root stress were carried out. The number of teeth is changed from 28 to 23 and also from 28 to 25 with the other parameters being the same and the calculations are carried out and tabulated.

The above calculations of the Von Mises stresses on the root of tooth were carried out in order to know if they match the results from ANSYS. The results are shown in Table 5.1. In this table, the maximum values of the tooth root stress obtained by the ANSYS method were given. For the number of teeth of 28, the ANSYS results are about 97% (2D) of the values obtained by the AGMA. For the cases from 23 teeth to 37 teeth, the values range from 91% to 99% of the value obtained by the AGMA. From these results, it was found that for all cases give a close approximation of the value obtained by the methods of the AGMA in both 3D and 2D models. These differences are believed to be caused by factors such as the mesh pattern and the restricted conditions on the finite element analysis, and the assumed position of the critical section in the standards.

Here the gears are taken as a plane strain problem. 2D models are suggested to be use because much more time will be saved when running the 2D models in ANSYS. There are not great differences between the 3D and 2D model in Table 5.1.

Table 5.1 Von Mises Stress of 3-D and 2-D FEM bending model

No of Teeth	3D stresses(ANSYS)	2D stresses(ANSYS)	Lewis Stresses	Difference 2D	Difference 3D
23	86.418	85.05	84.65	0.47%	2.08%
25	95.802	91.12	92.00	0.41%	4.1.3%
28	109.21	106.86	103.05	3.6%	5.8%
34	132.06	128.46	125.14	2.6%	5.52%
37	143.90	141.97	136.18	4.2%	5.66%

5.8 CONCLUSION

In the present study, effective methods to estimate the tooth contact stress by the two-dimensional and the root bending stresses by the three-dimensional and two dimensional finite element method are proposed. To determine the accuracy of the present method for the bending stresses, both three dimensional and two dimensional models were built in this chapter. The results with the different numbers of teeth were used in the comparison. The errors in the Table 5.1 presented are much smaller than previous work done by other researchers for the each case. So those FEA models are good enough for stress analysis.

CHAPTER 6 CONCLUSIONS AND FUTURE WORK

6.1 CONCLUSIONS

The contribution of the thesis work presented here can be summarized as follows:

It was shown that an FEA model could be used to simulate contact between two bodies accurately by verification of contact stresses between two cylinders in contact and comparison with the Hertzian equations.

Effective methods to estimate the tooth contact stress using a 2D contact stress model and to estimate the root bending stresses using 2D and 3D FEA model are proposed. The analysis of gear contact stress and the investigation of 2D and 3D solid bending stresses are detailed in Chapter 5.

In Chapter 4 the development of a new numerical method for FEA modeling of the whole gear body which can rotate in mesh including the contact problem is presented.

The values obtained from finite element analysis of the gears also confirm with the above results.

6.2 FUTURE WORK

The following areas are worthy of further research as computer capabilities increase. Further numerical method investigations should be conducted on:

- Simulation of an oil film in contact zone.
- Three-dimensionally meshed simulations for both spur and helical gears.
- The transmission error for all types of gears for example: helical, spiral bevel and other gear tooth form.
- A whole gearbox with all elements in the system such as the bearing and the gear casing.

REFERENCES

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

[2] Umezawa, K., 1988, “Recent Trends in Gearing Technology”, JSME International Journal Series III Vol.31, No. 2, pp 357-362

[3] Smith, J. O. Liu, C. K., “Stresses Due to Tangential and Normal Loads on an Elastic Solid with Applications to Some Contact Stress Problems”, *Journal of Applied Mechanics*

[4] Norton, R. L., “Machine Design: An Integrated Approach”, New Jersey: Prentice-Hall Inc.

[5] Hamrock, B. J., Jacobson, S. R., “ Fundamentals of Machine Elements”.

[6] Buckingham, E., 1949, “Analytical Mechanics of Gears”, McGraw-Hill, New York.

[7] Wang, J., 2003, “Survey of Nonlinear Vibration of Gear Transmission Systems” *Appl Mech Rev* vol 56, No 3.

[8] Chong, T. H., Bar, I., 2001, “Multiobjective optimal Design of Cylindrical Gear Pairs for the Reduction of Gear Size and Meshing Vibration”, JSME International Journal, Vol. 44, No. 1, pp 291-292

[9] Coy, J. J., Chao, C. H. S., 1982, “A method of selecting grid size to account for Hertz deformation in finite element analysis of spur gears”, Trans. ASME, J. Mech. Design 104 759-766

[10] Gatcombe, E.K., Prowell, R.W., 1960, “Rocket motor gear tooth analysis”, (Hertzian contact stresses and times) Trans. ASMA, J. Engng Industry

[11] Tsay, C.B., 1988, “Helical gears with involute shaped teeth: Geometry, computer simulation, tooth contact analysis, and stress analysis”, Trans, J. Mechanisms, Transmissions, and Automation in Design.