

## Contact Stresses for Different Gear Design Parameter

Dr. Muhsin J. Jweeg<sup>1</sup> & Wisam Auday Hussain<sup>2\*</sup>

Received on: 16/2/2006

Accepted on: 5/7/2007

### Abstract

Gearing is one of the most critical components in mechanical power transmission systems. This work investigates the characteristics of an Involute gear system including contact stresses. Current methods of calculating spur gear (Non-linear Analysis), contact stress using Hertz's equation, which were originally derived for contact between two cylinders. To enable the investigation of contact problems with Finite Element Method (FEM), the stiffness relationship between the two contacts 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. A computer program was built up using (MATLAB 6.5). The results of the two dimensional FEM analyses from ANSYS are presented. These stresses are compared with the theoretical values (Hertz's equations). Both results agree very well. This indicates that the Finite Element Method (FEM) model is accurate. The results of contact stress analysis indicates that increasing the geometrical parameters (Pressure angle, number of teeth and module) lead to improve the tooth contact stress, with the contact position, because the increasing of the geometrical parameters will results in an increase of the tooth stiffness which leads to decrease the tooth contact stress.

**Keywords:** Contact Stress, Finite Element Method, Spur Gear, Hertz.

### الاجهادات التلامسية لتروس ذات متغيرات تصميمية مختلفة

#### الخلاصة

تستخدم التروس في معظم تطبيقات اجزاء ناقلات الحركة الميكانيكية. هذا العمل يهدف في بحث ميزات نظام جانبية السن و الاخذ بنظر الاعتبار اجهادات التلامس. لقد تم أيجاد خواص الاسنان العدلة بواسطة طريقة العناصر المحددة ودراسة تأثير اجهاد الانحناء و التلامس على نموذج ثنائي الابعاد.

الطرق الشائعة في الحسابات اللاخطية للسن العدل, اجهاد التلامس باستخدام معادلة لويس التي اصلا اشتقت من حالت التماس بين اسطوانتين ولكي تتمكن من اجراء التحليل بطريقة العناصر المحددة تم تمثيل العلاقة بين منطقتي التلامس للاسنان بواسطة نابض. تم بناء برنامج حاسوب باستخدام لغة MATLAB 6.5, النتائج من النموذج الثنائي الابعاد التي استخرجت من برنامج ANSYS, Ver.8.0, هذه النتائج تم مقارنتها مع النتائج النظرية من معادلة لويس و نتائج المقارنة كانت جيدة.

\* College of Engineering , Al-Nahrain University/Baghdad

\*\* Mechanical Engineering Department of ,University of Technology, Baghdad

تبين أن زيادة المتغيرات التصميمية للترس العدل (زاوية ضغط, عدد الاسنان, و التضمين) يؤدي الى تحسين اجهاد التلامس بالنسبة الى موقع التلامس, حيث ان زيادة المتغيرات التصميمية يؤدي الى زيادة جساءة السن وبدوره يؤدي الى تقليل اجهاد التلامس.

system including contact stresses, bending stresses, and the transmission

## 1. Introduction

In almost all mechanical and structural engineering system, interaction occurs between two mechanical components or two parts of a single component when they come into contact with each other. Contact problems are highly nonlinear and require significant computer resources to solve. Contact nonlinearities occur when two or more components come into or out of contact with each other. In engineering applications, most contact processes are dynamic in a restrictive sense. Many of them can, however, be regarded as static for simplicity. By nature, contact phenomena always involve friction phenomena. However, friction effect may be neglected for simplicity in situations where frictional forces are sufficiently small.

One may argue that the subject of contact mechanics started in 1882 with the publication of the classical work by Hertz [1]. The Hertz theory is however restricted to frictionless contact and perfectly elastic solids. Finite element analysis of meshing gear pairs will be subject to non-linear contact analysis. In this situation. JIANDE WANG (Sep.2003) [2] investigated numerical methods for modeling the contact stresses of involute spur gears in mesh, over the mesh cycle, which forms the major part of this work. Zeping Wei (Oct.2004) [3] investigated the characteristics of an involute gear

errors of gears in mesh. The characteristics of involute spur gears are analyzed by using the finite element method.

The contact is highly non-linear because one or both of the following are unknowns. [4]

- The actual regions of contact are unknown until the problem has been solved. Depending on the load, material, boundary conditions and other factors, and surfaces can come into and go out of contact with each other in a largely unpredictable and abrupt manner.
- Most contact problems need to account for friction. There are several friction laws and models to choose from, and all are non-linear. Frictional response can be chaotic, making solution convergence difficult.

## 2. Hertz Theory of Elastic Contact

The first satisfactory analysis of the stresses at the contact of two elastic solids is due to Hertz. He is studied Newton's optical interference fringes in the gap between two glass lenses and was concerned at the possible influence of elastic deformation of the surfaces of the lenses due to the contact pressure between them. His theory, worked out during the Christmas vacation 1880 at the age of twenty-three, aroused considerable interest when it was first published and has

stood the test of time. In addition to static loading he also investigated the quasi-static impacts of spheres. Hertz also attempted to use his theory to give a precise definition of hardness of a solid in terms of the contact pressure to initiate plastic yield in the solid by pressing a harder body into contact with it. This definition has proved unsatisfactory because of the difficulty of detecting the point of first yield under the action of contact stress. He introduced the simplification that, for the purpose of calculating the local deformations, each body can be regarded as an elastic half-space loaded over a small elliptical region of its plane surface. By this simplification, generally followed in contact stress theory, the highly concentrated contact stresses are treated separately from the general distribution of stress in the two bodies which arises from their shape and the way in which they are supported. In addition, the well developed methods for solving boundary value problems for the elastic half-space are available for the solution of contact problems. In order for this simplification to be justifiable two conditions must be satisfied: [4]

- (a) The significant dimensions of the contact area must be small compared with the dimensions of each body.
- (b) The significant dimensions of the contact area must be small compared with the relative radii of curvature of the surfaces.

The first condition is obviously necessary to ensure that the stress field calculated on the basis of a solid which

is infinite in extent is not seriously influenced by the proximity of its boundaries to the highly stressed region. The second condition is necessary to ensure firstly that the surface just outside the contact region approximate roughly to the plane surface of a half-space, and secondly that the strains in the contact region are sufficiently small to lie within the scope of the linear theory of elasticity.

Metallic solids loaded within their elastic limit inevitably comply with this latter restriction. However, caution must be used in applying the results of the theory to low modulus materials like rubber where it is easy to produce deformations which exceed the restriction to small strains.

### 2.1. Force transmitted at point of contact:

The resultant force transmitted from one surface to another through a point of contact is resolved into a normal force  $P$  acting along the common normal, which generally must be compressive, and a tangential force  $Q$  in the tangent plane sustained by friction. The magnitude of  $Q$  must be less than or, in the limit, equal to the force of limiting friction, i.e.

$$Q \leq \mu P$$

Where  $\mu$  is the coefficient of limiting friction.  $Q$  is resolved into components,  $Q_x$  and  $Q_y$  parallel to axes  $Ox$  and  $Oy$ . In a purely sliding contact the tangential force reaches its limiting value in a direction opposed to the sliding velocity,

$$Q_x = -\frac{\Delta n_c}{|\Delta n|} mP, \quad Q_y = -\frac{\Delta n_y}{|\Delta n|} mP$$

The force transmitted at a nominal point of contact has the effect of compressing deformable solids so that they make contact over an area of finite size. As a result it becomes possible for the contact to transmit a resultant moment in addition to a force Fig. (1). The components of this moment  $M_x$  and  $M_y$  are defined as rolling moments. They provide the resistance to a rolling motion commonly called "rolling friction" and in most practical problems are small enough to be ignored. The third component  $M_z$ , acting about the common normal, arises from friction within the contact area and is referred to as the spin moment. When spin accompanies rolling the energy dissipated by the spin moment is combined with that dissipated by the rolling moments to make up the overall rolling resistance. At this point it is appropriate to define free rolling ("inertia rolling" in the Russian literature). This term will be used to describe a rolling motion in which spin is absent and where the tangential force  $Q$  at the contact point is zero.

**2.2. Surface Traction:**

The forces and moments which we have just been discussing are transmitted across the contact interface by surface tractions at the interface. The normal traction (pressure) is denoted by  $p$  and the tangential traction (due to friction) by  $q$ , shown acting positively on the lower surface in Fig.(1).while nothing can be said at this stage about

the distribution of  $p$  and  $q$  over the area of contact  $S$ , for overall equilibrium:

$$P = \int_s p ds \quad Q_x = \int_s q_x ds \quad Q_y = \int_s q_y ds$$

With non-conforming contacts (including cylinders having parallel axes) the contact area lies approximately in the  $x$ - $y$  plane and slight warping is neglected,

$$M_x = \int_s p y ds, \quad M_y = -\int_s p x ds$$

$$M_z = \int_s (q_y x - q_x y) ds$$

Finally the surfaces are assumed to be frictionless so that only a normal pressure is transmitted between them as show in Fig. (1) Which shows the force and moment acting on contact area. Although physically the contact pressure must act perpendicular to the interface which will not necessarily be planar, or the linear theory of elasticity does not account for changes in the boundary forces arising from the deformation they produce (with certain special exceptions). Hence, in view of the idealization of each body as a half-space with a plane surface, normal tractions at the interface are taken to act parallel to the  $z$ -axis and tangential tractions to act in the  $x$  - $y$  plane.

The significant dimension of the contact area is denoting by  $a$ , the relative radius of curvature by  $R$ , the significant radii of each body by  $R1$ , and  $R2$  and the significant dimensions of the bodies both laterally and in depth by  $I$ . The assumptions made in the Hertz theory as follows: [4].

- (i) The surfaces are continuous and non-conforming:  $a \ll R$  ;
- (ii) The strains are small:  $a \ll R$  ;
- (iii) Each solid can be considered as an elastic half-space:  $a \ll R_{1,2}$  ,  $a \ll I$ ;
- (iv) The surfaces are frictionless:  $q_x = q_y = 0$ .

The development of mechanical science and the advances in engineering activities made it of interest to study increasingly complex contact problems. Local phenomena, like stress distributions on contacting surfaces, began to be observed. Among other, Hertz was a leading scientist in this area. His successful treatment of a static contact problem in elasticity in the 1880s, has been regarded as a milestone in the field. In his study, an example of the classical Hertzian contact problem is presented here for clarification.

Two elastic spheres in contact are shown in Fig (2-a). In the Hertzian contact theory, the displacements of the contacting boundaries are assumed to satisfy the following condition. [5]

$$u_3^1 + u_3^2 = g - (1/2R)(r^2) \quad \dots (1)$$

Where  $u_3^1$  and  $u_3^2$  are the displacements in the  $x_3$  direction of the contacting boundaries form the two spheres, respectively. The initial separation of the two contacting boundaries is given by  $g$ ,

and 
$$\frac{1}{R} = \frac{1}{R_1} + \frac{1}{R_2}$$

In which  $R_1$  and  $R_2$  are the radii of the two spheres, respectively, and  $r$  is the distance between a given contacting point and the centre of the contacting area, which a circle in this case.

On the basis of equation (1), the contact pressure distribution on the contacting area can be found.

$$q = q_0 \left[ 1 - \left( \frac{r}{r_0} \right)^2 \right]^{\frac{1}{2}} \quad \dots (2)$$

Where  $r_0$  is the radius of the contacting area and  $q_0$  is the maximum pressure at the center. The pressure distribution in equation (2) is plotted in Fig. (2-b).

In spite of the fact that the Hertz theory of contact has stood the test of time it been a landmark in applied mechanics for many decades, it suffers from major restrictions. These limitations are that the bodies have smooth continuous surfaces, the stress and displacements can be deduced from the small strain theory of elasticity applied to a linear elastic half-space, and that the surfaces are frictionless. Following Hertz's work, many researchers and scientists studied contact problems between elastic bodies of different shapes and under different circumstances with or without friction.

**2.3. Hertz Contact Stress Equations:**

Hertz theory assumes that the gear teeth as an equivalent contacting

cylinder as shown in Fig (3), and Fig (4) show the contact zone.

Where  $r_1, r_2$  is the radii of equivalent cylinders, and  $\vec{W}$  is the normal force

Contact Width

$$b = \sqrt{\frac{2F(1-n_1^2)/E_1 + (1-n_2^2)/E_2}{pL(1/d_1 + 1/d_2)}} \dots (3)$$

Maximum Contact Pressure

$$P_{max} = -\frac{2F}{pbL} \dots (4)$$

Contact Force

$$F = 2L \int_0^b p(y) dy \dots (5)$$

Eq. (3) can be written as shown below,

$$b = \sqrt{\frac{F}{L} \sqrt{\frac{1}{p} \left[ \frac{(1-n_1^2)}{E_1} + \frac{(1-n_2^2)}{E_2} \right]}} \sqrt{\frac{4}{\frac{1}{r_1} + \frac{1}{r_2}}} \dots (6)$$

where CP is the Elastic Coefficient,

$$C_p = \left[ \frac{1}{p \left( \frac{1-n_1^2}{E_1} + \frac{1-n_2^2}{E_2} \right)} \right]^{1/2} \dots (7)$$

The eq. (4) becomes,

$$P_{max} = -C_p \left[ \frac{F}{L} \left( \frac{1}{r_1} + \frac{1}{r_2} \right) \right]^{1/2} \dots (8)$$

$$F = W_n = W_t / \cos f \dots (9)$$

where  $f$  is the pressure angle.

$$P_{max} = S_c = -C_p \left[ \frac{W_t}{L \cos(f)} \left( \frac{1}{r_1} + \frac{1}{r_2} \right) \right]^{1/2} \dots (10)$$

where  $S_c$  is the contact stress.

We know that the  $r_1, r_2$  are the radii of equivalent cylinders and equal to,

$$r_{1,2} = \frac{d_{1,2} \sin f}{2} \dots (11)$$

where  $d_1, d_2$  are the pitch diameters, and  $d_1$  is for Pinion and  $d_2$  for gear.

So the gear ratio is,

$$m_g = \frac{d_g}{d_p} \text{ (For external gears)} \dots (12)$$

Then, the terms  $\left( \frac{1}{r_1} + \frac{1}{r_2} \right)$  can be written as shown below,

$$\frac{1}{r_1} + \frac{1}{r_2} = \frac{2}{\sin f} \left( \frac{1}{d_p} + \frac{1}{d_g} \right) \dots (13)$$

Substituting Eq. (4.21) into Eq. (4.22) gives,

$$\frac{1}{r_1} + \frac{1}{r_2} = \frac{2}{d_p \sin f} \left( \frac{m_g + 1}{m_g} \right) \dots (14)$$

Substituting the Eq. (14) into Eq. (8) give.

$$S_c = -C_p \left[ \frac{W_t}{d_p L I} \right]^{1/2} \quad \dots (15)$$

(Contact Stress Between to teeth gear)

where  $I = \frac{\cos f \cdot \sin f \cdot m_g + I}{2 \cdot m_g}$

### 3. The Finite Element Method of Contact Analysis: [6].

The finite element general equation can be describe by :

$$[M] \{A\} + [K] \{U\} = \{F\} \quad \dots (16)$$

Where [M] is the mass matrix, [K] is the stiffness; {U} is the displacement vector. {A} is the acceleration vector, and {F} is the external load vector as in the standard finite element procedure.

If contact does occur,  $q_i$  will be non-zero and will contribute to equation (16). In the finite element method,  $q_i$  can be obtained from its nodal values, denoted by  $q_{im}$  at node  $m$ , at discrete finite element nodes by interpolations. Then  $q_{im}$  can be taken as the primary unknowns to be solved for Denoting the contribution of contact forces to the load vector by {F<sub>c</sub>}, we can write,

$$[M] \{A\} + [K] \{U\} = \{F\} + \{F_c\} \quad \dots (17)$$

In equ. (17), {F<sub>c</sub>} is unknown and is to be calculated under the constraint

given in equ. (18). with the finite element discretization, the kinematics constraint on contacting nodes can be put into the form,

$$[Q] \{U\} + \{G\} = 0 \quad \dots (18)$$

Where {G} is calculated from initial gaps of contacting nodes and [Q] is a coefficient matrix resulting from the finite element discretization.

In order to solve equations (17, 18), first determine the total number of contacting nodes, which are unknown until the solution is found. Thus, trial contacting nodes need to be used and iterations need to be carried out to find the true contacting nodes, At the same time, the contact condition must be enforced to solve the unknown contact force, which necessitates a constraint method. If frictional effects are to be considered, a friction law governing the tangential contact force is required. Furthermore, both displacements and accelerations are unknowns in equations (17, 18). Therefore, a time integration method is also required for the solution. The above discussion on the finite element solution procedure for contact problems is neither general nor complete; rather, it can help to show some of the ingredients in solving contact problems with the finite element method. These basic ingredients may be classified as follows.

- Variational formulation, which provides a basis for the finite element discretisation.

- Element formulation, which calculates the element contribution to the coefficient matrix and the load vector.
- Material modeling which determines the stress-strain relationships and plays an important role in the element formulation.
- Friction law, which governs the frictional forces if frictional effects are to be included.
- Contact constraint method, which provides a means of calculating the unknown contact forces under contact constraints.
- Contact searching algorithm, which searches potential contacting nodes and determines the locations of contacting nodes accurately and efficiently.
- Time integration method, which integrates in the time domain.
- Linearisation and iterative procedure, which transforms a geometrical non-linear problem into a series of geometrical linear problems. In general contact problems with geometrical non-linearities, a linearisation procedure will be geometrical non-linearity.

With the use of advanced finite element techniques, no restriction is in principle necessary on the geometry, material properties, and deformation patterns contacting bodies. The versatility of the finite element method makes it possible to attack extremely complicated problems. Over the last few decades, great efforts have been devoted to the finite element study of contact problems. Remarkable progress

has been made in both theoretical studies and engineering applications.

#### A. 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),

- Rigid - to - flexible bodies in contact.
- Flexible - to - flexible bodies in contact.

In Rigid-to-flexible bodies in contact, 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.

#### B. Type of Contact Models:

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

- 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.



- 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 discretization or a compatible mesh. Point to surface contact was used in this work.
- Surface-to-surface contact is typically used to model surface-to-surface contact applications of the rigid-to-flexible classification.

### C. 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,

$$k_n = fEh$$

Where  $f$  is a factor that controls contact compatibility. This factor is usually being between 0.01 and 100. [6],

$E$  = smallest value of Young's Modulus of the contacting material

$h$  = 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. [3]

The contact problem is addressed using a special contact element (two and three dimensional, spring and damper combinations). For the

problem in hand, the element to be used is a two-dimensional, the three nodes, and point-to-surface contact element. In the input file, the CONTACT element from the ANSYS element library as the contact elements between the two contact bodies shown as Fig. (5), is chosen. It is applicable to 2-D geometry, plane strain, plane stress, or axisymmetry situations). The area of contact between two or more bodies is generally not known in advance. It may be applied to the contact of solid bodies for static or dynamic analyses, to problems with or without friction, and to flexible-to-flexible or rigid-to-flexible body contact.

#### D. Contact Element Kinematics:

The (FEM) treats contact problems by extending the variational formulation upon which the FE method is based. The stiffness matrix associated with contact elements and other element stiffness matrices of the body are formulated and assembled into the origin FE code. The solution is then obtained by solving the resulting set of nonlinear equations.

Fig. (6), shows a typical example of such contact element. This particular element, 2D contact element for point of surface contact problem, is adopted in several FE packages including (ANSYS® ver. (8)). It is applicable to 2-D geometry, plane strain, plane stress, or axisymmetry situations. The area of contact between two or more bodies is generally not known in advance. It may be applied to the contact of solid bodies or dynamic analyses, to problems with or without friction and to flexible-to-flexible or

rigid-to-flexible body contact. In this case, the element is based on two stiffness values. They are the combined normal contact stiffness  $KN_c$  and the combined tangential contact stiffness  $KT_c$ . The combined normal contact stiffness  $KN_c$  is used to penalize interpenetration between the two bodies, while the combined tangential contact stiffness  $KT_c$  is used to approximate the sudden jump in tangential force, as represented by Coulomb's friction law when sliding is detected between the two contacting nodes.[6]

In order to satisfy contact compatibility, forces are developed see fig. (6), where  $f_{n,IJK}$  and  $f_{s,IJK}$  are the normal and tangential forces at node I,J,K respectively, in a direction (n) normal to the target plane that will tend to reduce the penetration to an acceptable numerical level. In addition to compatibility forces, friction forces are developed in direction (s) tangent to the target plane. Two methods of satisfying contact compatibility are available:

- (i) A penalty method.
- (ii) A combined penalty plus Lagrange multiplier method.

#### E. Development of Finite Element Models

The development approach for finite element models is accomplished as follows [3]:-

Step 1:- Tooth surface equation of pinion and gear portions of corresponding rim are considered for determination of the designed bodies. Loss of accuracy introduced by CAD

computer programs for the generation is avoided.

Step 2:- The proposed approach does not require an assumption on the load distribution in the contact area. To get the contact stresses the contact algorithm of the general purpose (ANSYS) is used.

Step 3:- F.E. Models of one pair of teeth are applied and therefore the boundary conditions are far enough the loaded area of the teeth.

Step 4:- Setting of boundary conditions is accomplished automatically and are shown in Fig. (7) (a) and (b).

- Nodes on the two sides and bottom part of the portion of the gear rim are considered as fixed Fig. (7).
- Nodes on the two sides and bottom part of the portion of the pinion rim are considered as rigid body Fig.(7).
- Only one degree of freedom is defined at rotation pinion axis, while all other degree of freedom is fixed. Application torque (T) in rotational motion at pinion rim nodes Fig. (7-b).
- Input torque can be expressed as the sum of applied nodal forces at the radius of shaft ( $r_{sh}$ ), thus.

$$T = \sum_{i=1}^n F_i r_{sh}$$

Where T is the input torque load, (n) is the total number of constrained nodes and  $F_i$  is the tangential nodal force (usually  $F_i = F$ , F is constant value). The displacement of two sides and bottom part on the pinion tooth has a unique value because a coupled equation was used to enable these surfaces to be rigid in motion.

Step 5:- The contact algorithm of FEM computer program [11] requires definition of contacting surface. To define a contact pair completely, contact and target element have to be referred to same characteristic parameters. The contact element 172 and target 169 with three nodes are used as a contact point-to-surface in the present analysis.

Step 6:- The select contact algorithm was the Penalty Method. The penalty method uses a contact "spring" to establish a relationship between the two contact surfaces.

#### F. Contact Element Advantages, Disadvantages and their Convergence:

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 terms 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 is within the radius. If solution fails to converge, the start is 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 lead to a more accurate solution, but require 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. Hertz Contact Stress Calculation Program:

The basic steps of computer programs are preformed by equations which are description of the Hertz Contact Stress. The programs are built up using program (MATLAB Ver.6.5) to evaluate the contact stresses in spur gear. And Table. (1) Shows the codes used in the program to calculate Hertz Contact Stress.

#### 5. RESULTS AND DISCUSSIONS:

In this section, symmetrical spur gear teeth with different pressure angles, modules and different number of teeth without profile correction are investigated.

Also, this section contains comparisons of the numerical and theoretical results obtained from the available published results and with the

results of ANSYS package, Ver. (8.0), and this section investigates the characteristics of an involute gear system including contact stresses.

Case Studies The selection of suitable models to carry out the required analysis includes selection of important parameters such as: -

- a) Geometry: - In order to achieve the required objectives, six parameters of spur gear geometry are changed, which are: -
- Pressure Angle.
  - Module.
  - Number of Teeth.

In Table (2), the geometry of the selected models is summarized.

b) Material of Models: -The material used for models is steel with modulus of elasticity ( $E$ ), yield stress ( $S_y$ ), ultimate stress ( $S_u$ ), material density ( $\rho$ ) and Poisson's ratio ( $\nu$ ) as follows:-

$$E = 207 \times 10^9 \text{ (N/m}^2\text{)},$$

$$S_y = 650 \times 10^6 \text{ (N/m}^2\text{)},$$

$$S_u = 900 \times 10^6 \text{ (N/m}^2\text{)},$$

$$\rho = 7860 \text{ (kg/m}^3\text{)}$$

and  $\nu = 0.3$ .

c) Loading Condition: - For Contact Analysis the applied torque is (25 KN.m) per (mm). as shown in section (E).

#### 5.1. Contact Analysis:

In this analysis an external torque of amplitude (25 KN.m) was applied in all cases of table. (2), with five different positions in gear profile in

order to understand the behavior of contact stresses for complete mesh cycle.

### 5.2. Verification Test for Contact Stress Analysis:

The studied gear in Ref. [11] is a spur gear of module (24mm), pressure angle ( $20^\circ$ ), number of teeth for pinion (20), and for gear (96). The material of the model is steel. Table (3) shows the comparison between (FEM), ANSYS Ver. (8) and Ref. [12], this comparison shows that there is a good agreement between them, therefore the validity of the ANSYS Ver. (8) is satisfied.

### 5.3. Discussions of Contact Analysis Results:

The variation of the tooth maximum contact stress with the contact position for different gear design parameters is shown in Fig. (8). The effect of pressure angle on the tooth stress is shown in Fig. (8-a), while Figs (8-b) and (8-c) show the effect of number of teeth and module on the tooth stress respectively.

Fig (9) shows the variation of contact position, and Fig (10) show the comparison between the contact stresses resulting from (FEM) and the result from theoretical method, for all cases of table. (2).

The results of contact stress analysis indicates that increasing the geometrical parameters (Pressure angle, number of teeth and module) lead to improve the tooth contact

stress, with the contact position, because the increasing of the geometrical parameters will results in an increase of the tooth stiffness which leads to decrease the tooth contact stress.

One case of table. (2) has been accomplished in Fig. (11) to represent the sample of the nodal solution stress (Von Mises), with different contact position, were plotted using Ansys package, Ver (8.0), the von Mises criterion is best applied and best understood when used to predict the onset of yielding in a structure where the material behaves in a ductile fashion. Fig. (12) Shows the contact area change with increasing torque by force reaction.

### 6. Conclusions:

The main conclusions obtained from the present work can be summarized as follows: -

**1-** It is shown that a FEA model could be used to simulate contact between two bodies accurately by verification of contact stresses between two gears in contact and comparison with the Hertzian equations. Effective methods to estimate the tooth contact stress using a 2D contact stress model are proposed.

**2-** The module has the greater effect on the behavior of the tooth contact stresses. Decreasing the module leads to increase in the contact stress.

**3-** Increasing the spur gear design parameters (pressure angle, number of teeth and module) leads

to improvement in the tooth strength by increasing the thickness of the critical section which results in increasing the area of tooth critical section and makes it able to withstand higher loads.

### 7. References.

- [1] Vijayakar, S.M. and Houser, D.R., (Contact Analysis of Gear Using a Combined Finite Element and Surface Integral Method), *Journal of Gear Technology*, P- 26-33, (1993).
- [2] JIANDE WANG, (Numerical and Experimental Analysis of Spur Gears in Mesh), PhD Curtin University of Technology September 2003.
- [3] Zeping Wei, (Stresses and Deformations in Involute Spur Gears by Finite Element Method), M.Sc University of Saskatchewan, Canada, October, 2004.
- [4] (Contact Mechanics), K. L. Johnson, Cambridge University Press, Ninth prints 2003.
- [5] Spyrakos, Constantine and Raftoyiannis, John. (Linear and Nonlinear Finite Element Analysis in Engineering Practice), Algor Publishing Division, 1997.
- [6] Seney Sirichal, (Torsional Properties of Spur Gears in Mesh Using Nonlinear Finite Element Analysis), PhD, thesis Curtin University of Technology, June, 1999.
- [7] Intisar A. R. Hadi, (Torsional Mesh Stiffness, Transmission Error, Load Shearing Ratio for Pair and Double Pair of Spur Gear Teeth in Mesh Using Finite Element Method), PhD University of Technology, 2004.
- [8] Klensz, S. R., (Finite Element Analyses of A Spur Gear Set), M.Sc Thesis, Dept. of Mechanical Engineering, University of Saskatchewan, 1999.
- [9] Faydor L. Litvin, Alfonso Fuentes and Baxter R. Mullins, Ron Woods, (Computerized Design, Generation, Simulation of Meshing and Contact, and Stress Analysis of Formate Cut Spiral Bevel Gear Drives), NASA Center for Aerospace Information, June, 2003.
- [10] Faydor L. Litvin, Alfonso Fuentes, J. Matthew Hawkins and Robert F. handschuh, (Design, Generation and Tooth Contact Analysis (TCA) of Asymmetric Face Gear Drive With Modified Geometry), NASA Center for Aerospace Information, January, 2001.
- [11] ANSYS 8.0, (Structural Analysis Guide), Ansys Inc., 2003.
- [12] Vera Nikolić-Stanojević, Ivana (Atanasovska) Cvejić, (The Analysis of Contact Stress on Meshed Teeth's Flanks along the Path of Contact for a Tooth Pair), *Mechanics, Automatic Control and Robotics* Vol.3, No 15, pp. 1055 – 1066, 2003.

Table (1) Hertz Contact Stress Calculation Program.

```

function segma=hcs(ep,eg,vp,vg,np,ng,r1,m,l,t,th)
% univcrity of technology
% dept. of Maechanical Engineering-Applied Mechanic
% Wisam Auday
% this program computes gear tooth contact stress analysis
-----
% hcs: hertz contact stress equation. (N/m^2)
% ep: Young's Modulus of elasticity for pinion. (N/m^2)
% eg: Young's Modulus of elasticity for gear. (N/m^2)
% vp: poisson's ratio for pinion.
% vg: poisson's ratio for gear.
% np: number of teeth for pinion.
% ng: number of teeth for gear.
% r1: radius contact position. (m)
% m: modules. (m)
% l: face width. (m)
% t: torque. (N.m)
% th: pressure angle. (degree)
-----
% th:degree ---> rad.
th=th*pi/180
% rp: pinion pitch radius.
rp=0.5*np*m
% dp: pinion pitch diametere.
dp=np*m
% mg: gear ratio.
mg=ng/np
% beta: loading angle.
beta=tan(acos(rp/r1*cos(th)))-pi/(2*np)-(tan(th)-th)
% wt: tangential force.
wt=2*t*cos(beta)/(np*m)
% i: form factor.
i=(cos(th)*sin(th)/2)*(mg+1)/mg
% cp: elastic coefficient.
vpp=(1-vp^2)/ep;
vgg=(1-vg^2)/eg;
cp=sqrt(1/(pi*(vpp+vgg)))
% segma: contact stress.
segma=cp*sqrt(wt/(dp*i))
save hcs

```

**Table (2) Various geometrical properties of spur gear studied cases.**

Case no.	Gear Tooth Characteristics		
	Press. Angle $\phi$	No. of Teeth (N)	Module (mo) mm
1	14.5	20	12
2	20	20	12
3	25	20	12
4	14.5	52	12
5	14.5	42	12
6	14.5	32	12
7	20	20	6
8	20	20	8
9	20	20	10

**Table (3) Verification test for Contact Stress analysis.**

Contact Stress (MPa)		Percentage Error (%)
Ref. [29]	Present Work	
1540	1531.571	2.219



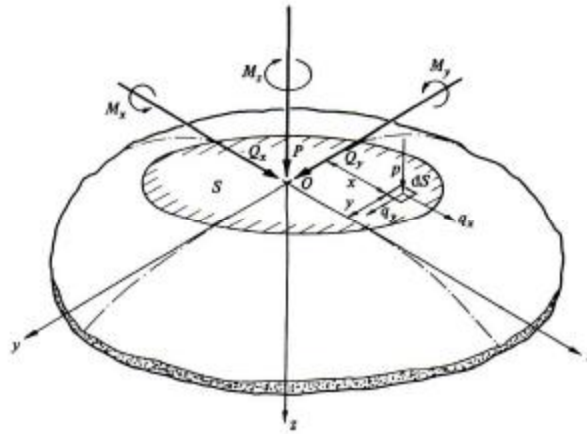


Figure (1) Force and Momentum Acting in Contact Area S.

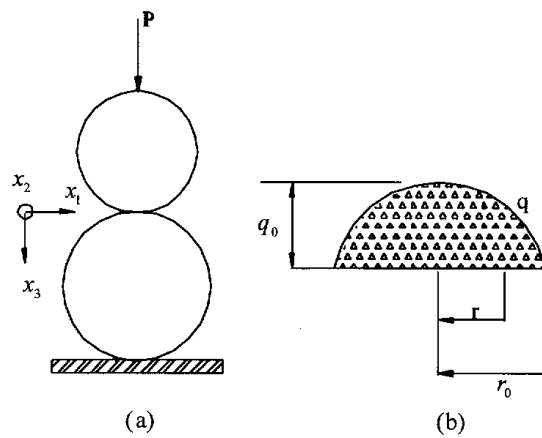


Figure (2) (a) Two Spheres in Contact (b) Contact Pressure Distribution between Two Spheres According to Hertzian Contact Theory.

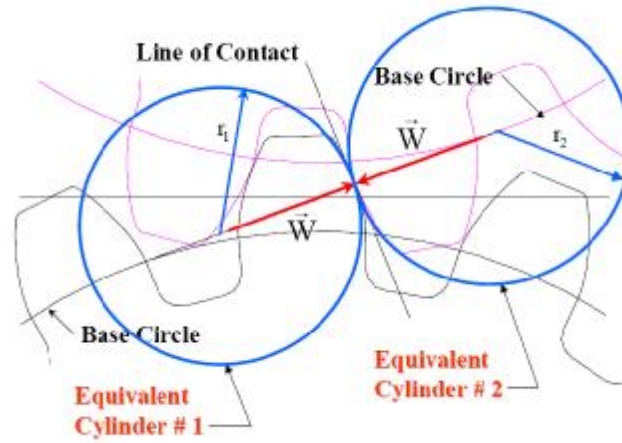


Figure (3) Equivalent Contacting Cylinders.

### Contact Zone

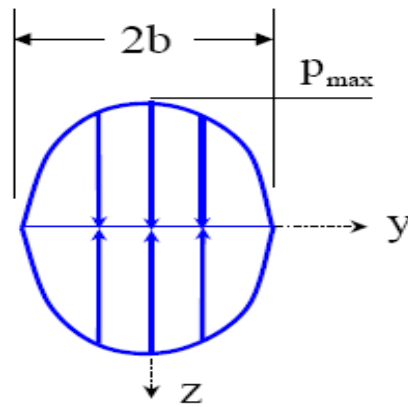


Figure (4) The Contact Zone in Gear Teeth.

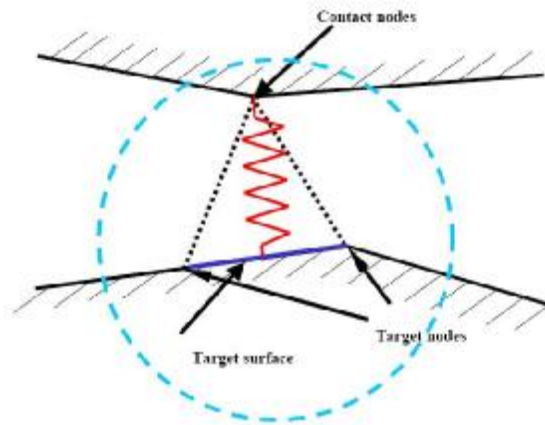


Figure (5) Point-to-Surface Contact Element.

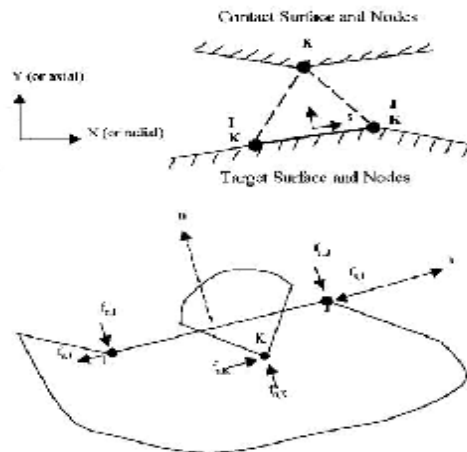


Figure (6) 2-D Point-to-Surface Contact Element and Nodal Contact Forces

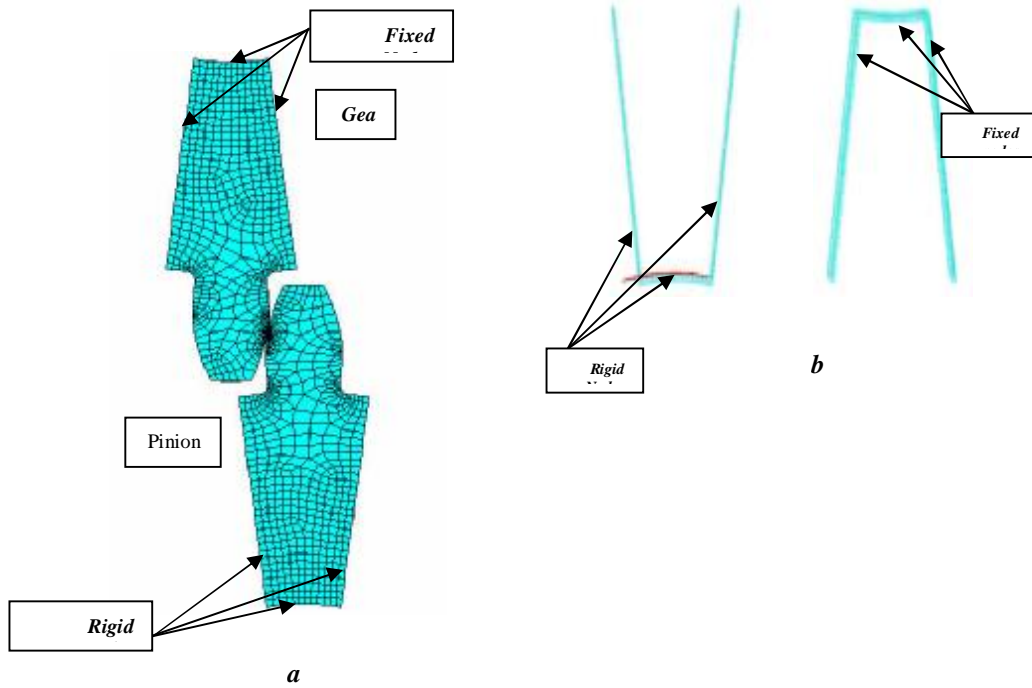
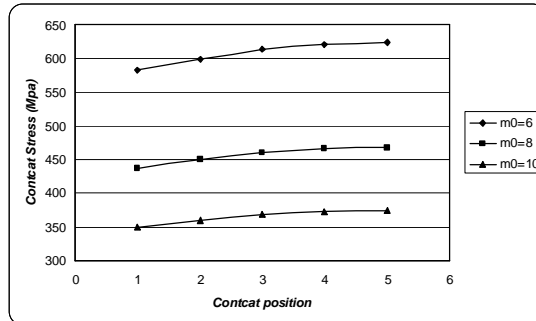
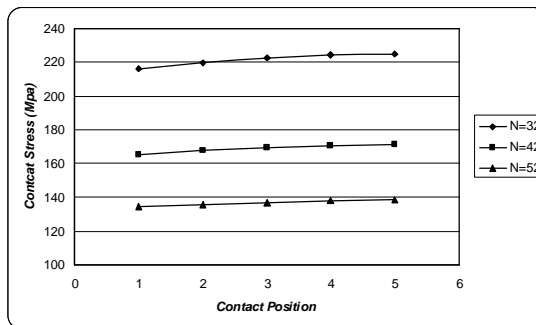


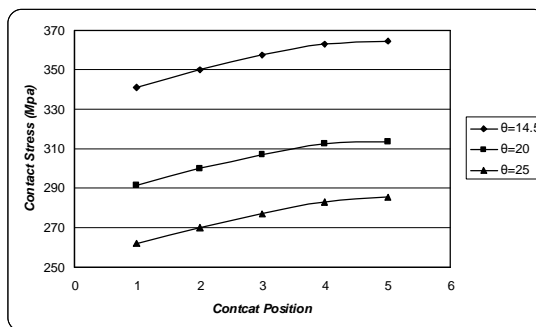
Figure (7) Boundary Conditions for Pinion and Gear



(a) Variation of contact stress with contact position for different values of pressure angle.



(b) Variation of contact stress with contact position for different values of number of teeth.



(c) Variation of contact stress with contact position for different values of module.

Figure (8) Variation of Contact Stress with Contact Position for Different Values of Gear Design Parameters.

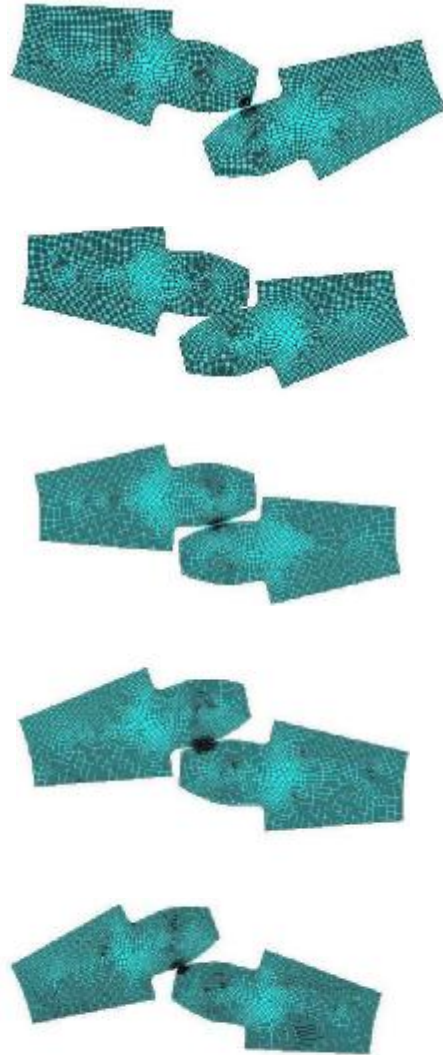
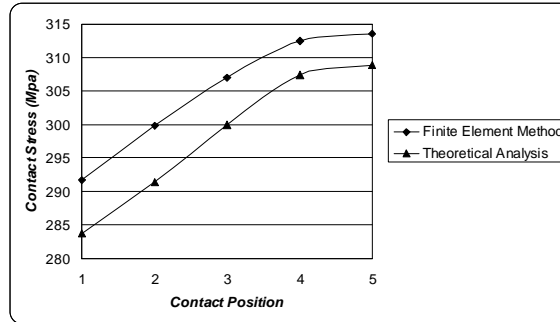
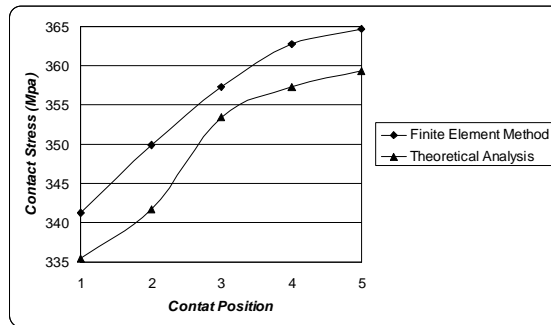


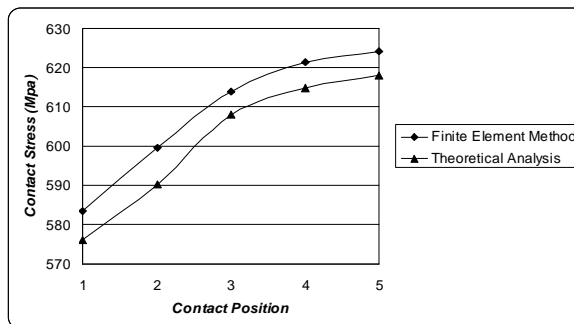
Figure (9) Contact position for complete mesh cycle



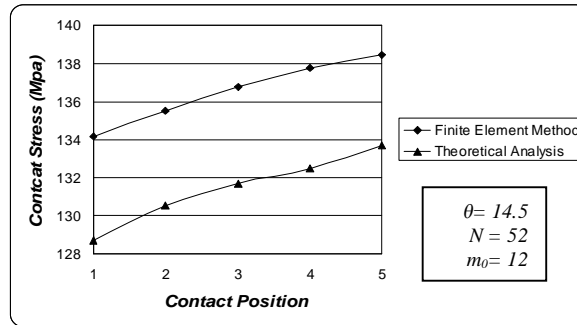
Case 1



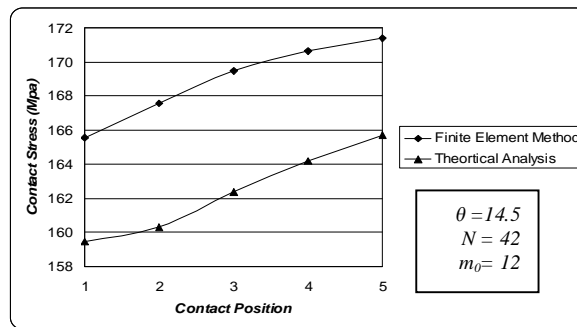
Case 2



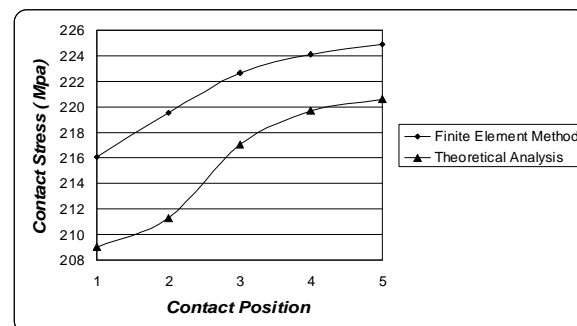
Case 3



Case 4

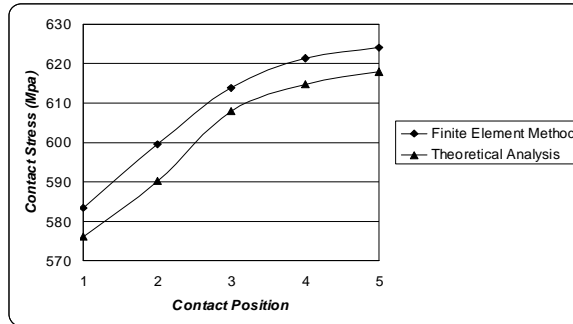


Case 5

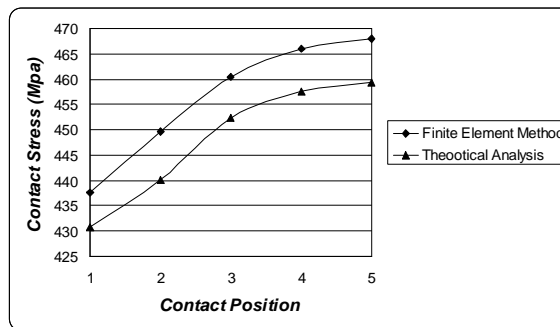


Case 6

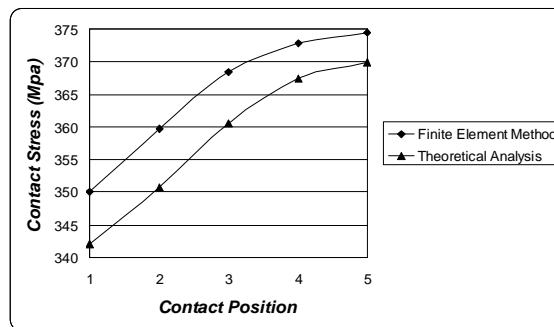




Case 7

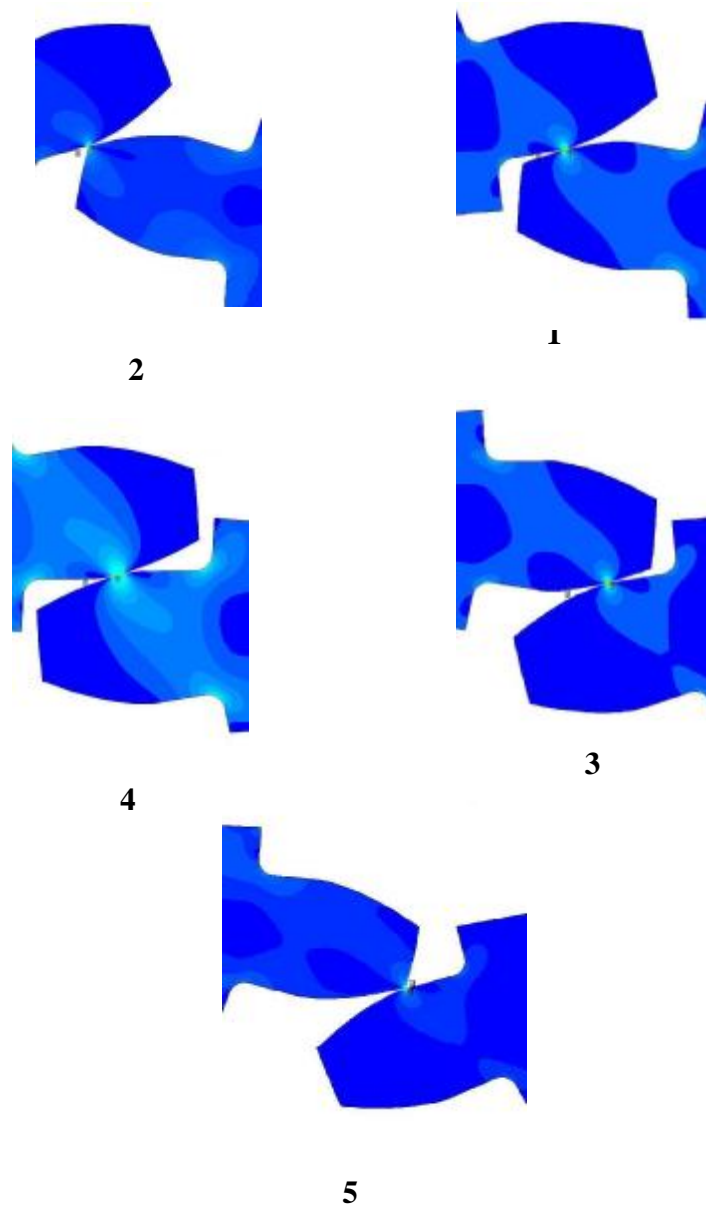


Case 7

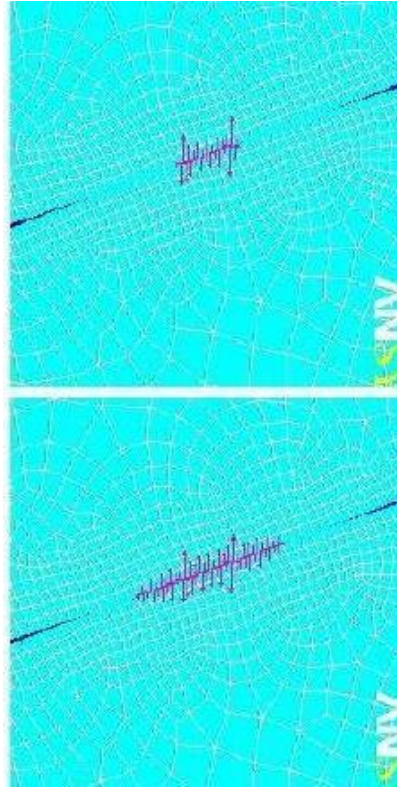


Case 7

Figure (10) The Change of Relative Contact Stress along the Line of Contact for Different Values of Gear Design Parameter.



**Figure (11) Contact Stress (Von Mises), with Different Contact Positions.**



**Figure (12) The Contact Area Shows by the Reaction Force.**