Volume : 3 | Issue : 6 | June 2014 • ISSN No 2277 - 8160



Research Paper | Engineering

Bending Stress and Deformation Analysis of Spur Gear by FEM

Sachindra Kumar	Asst. Prof. Cambridge Institute of Technology, Tatisilway, Ranchi, Jharkhand, India
Anjani Kumar Singh	Ph.D Scholar, Materials and metallurgical Engineering National Institute of Foundry and Forge Technology, Hatia, Ranchi, Jharkhand, India
Nitesh Kumar	M.Tech, Materials Science and Engineering, National Institute of Foundry and Forge Technology, Hatia, Ranchi, Jharkhand, India
Sushil Patel	M.Tech, Materials Science and Engineering, National Institute of Foundry and Forge Technology, Hatia, Ranchi, Jharkhand, India
Ajit Kumar	M.Tech, Materials Science and Engineering, National Institute of Foundry and Forge Technology, Hatia, Ranchi, Jharkhand, India

ABSTRACT

This research work on characteristics of an involute gear system including, bending stresses of gears in mesh. Gearing is one of the most critical components in a mechanical power transmission system, and in most industrial rotating machinery. 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. Transmission error is considered to be one of the main contributors to noise and vibration in a gear set. Transmission error measurement has become popular as an area of research on gears and is possible method for quality control. To estimate transmission error in a gear system, the characteristics of involute spur gears were analyzed by using the finite element method. 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.

KEYWORDS : Involute spur gear, FEM, Bending Stress, ANSYS.

Introduction

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. Analysis of the characteristics of involute spur gears in a gearbox was studied using nonlinear FEM [1, 2].

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.[3]

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.

Analytical Procedure

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. 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. 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 involute profile of the pinion and gear were calculated to plot an involute profile using a cylindrical system. The equations of an involute curve below were taken from Buckingham [4].

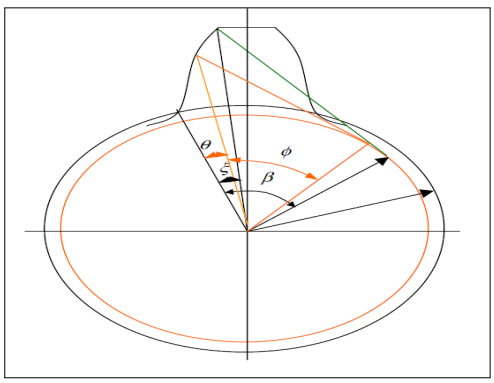
$$r = r_b * (1 + \beta^2)^{1/2}$$
$$\psi = \theta + \pi/2n_1 - \xi \qquad (1)$$
$$\theta = \tan\theta - \varphi = \text{inv}\,\varphi$$

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
$\varphi 1$= pressure angle at radius r

One spur tooth profile was created using equation (1), shown in Figure 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. 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 2.



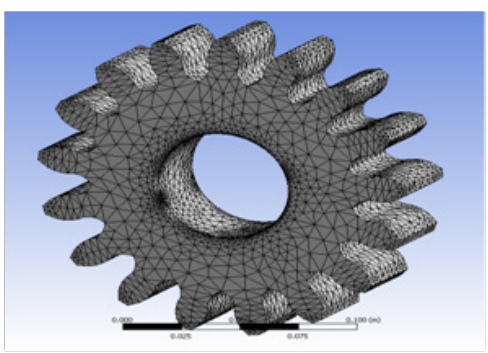


Figure 1 Involutometry of a Spur gearFigure 2 FEM Mesh Spur Gear

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.

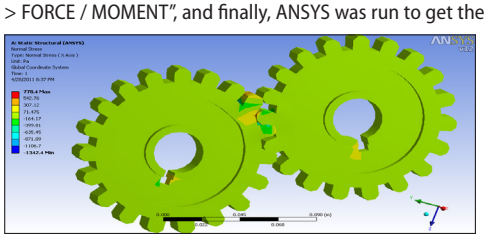


Figure 3 Gear contact stress model

Rotation Compatibility of the Gear Body

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. When the gears are put into mesh, the line tangent to both base circles is defined as the line of action for involute gears. In one complete tooth mesh circle, the contact starts at points a asshown in Figure 3 [5] where the outside diameter circle, the addendum circle of the gear intersects the line of action. The mesh cycle ends at point E, as shown in Figure 4 where the outside diameter of the pinion intersects the line of action.

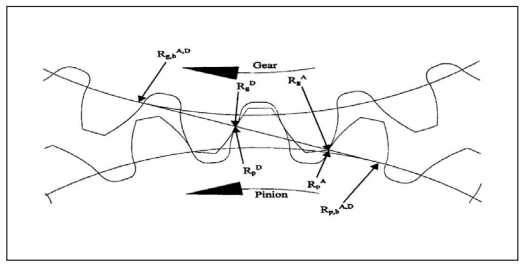


Figure 4 Illustration of one complete tooth meshing cycle

Consider two identical spur gears in mesh. When the first tooth pair is in contact at point A it is between the tooth tip of the output gear and the tooth root of the input gear (pinion). At the same time a second tooth pair is already in contact at point D in Figure 3. As the gear rotates, the point of contact will move along the line of action APE. When the first tooth pair reaches point B shown in Figure 4, the second tooth pair disengages at point E leaving only the first tooth pair in the single contact zone. After this time there is one pair of gear in contact until the third tooth pair achieves in contact at point A again. When this tooth pair rotates to point D, the tooth pair begins engagement at point A which starts another mesh cycle. After this time there are two pairs of gear in contact until the first tooth pair disengages at point E. Finally, one complete tooth meshing cycle is completed when this tooth pair rotates to point E. 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 [5].

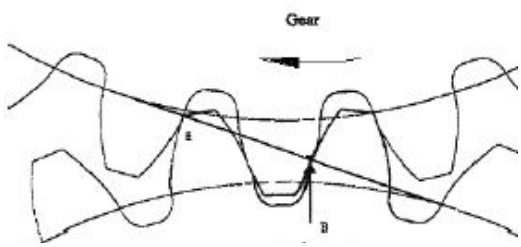


Figure 5 Different positions for one complete tooth meshing cycle

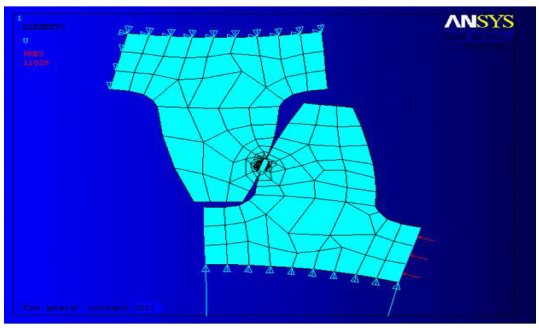
Analytical equations can also be developed for the rotation of the gear and pinion hubs, including the effects of tooth bending deflection and shearing displacement and contact deformation. In the pinion reference frame, it is assumed that the pinion hub remains stationary, while the gear rotates due to an applied torque. Considering the single pair contact zone at point B, the condition of angular rotation of the gear body will then be given by [7, 8].

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 ofteeth 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-determined 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 [11]. 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 2 and three points in Figure 5.

There are two ways to get the contact stress from ANSYS. Figure 5

shows the first one, which is the same to create the contact element COCNTA 48 and the rectangular shape fine mesh beneath the contact surfaces between the contact areas. Figure 6 shows the enlarged-area with a fine mesh which is composed of rectangular shapes.



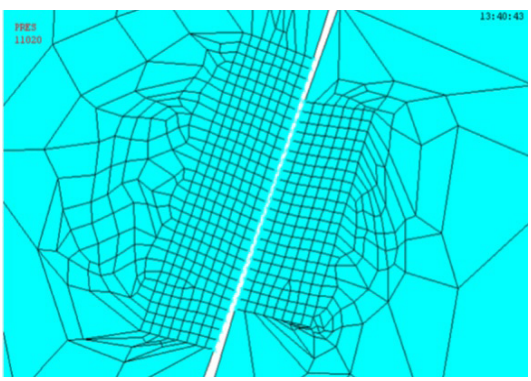
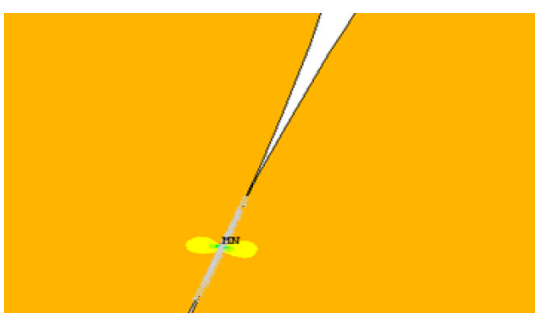


Fig.6 FEM Model of the gear tooth pair in contactFig 7 Fine meshing of contact areas



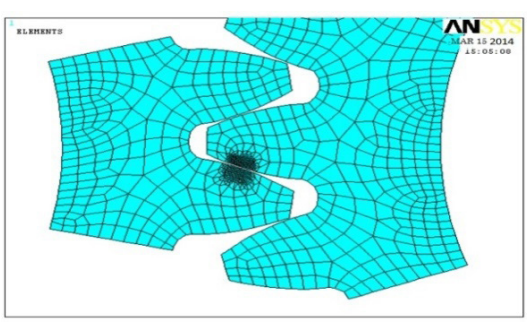


Figure 8 Contact stress along contactareasFigure 9 a fine mesh near contact areas

Figure 7 shows the normal contact stress along the contact areas. The results are very similar to the results in the two cylinders. Figure 8 presents how to mesh using a second method. 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.

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) [4] 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 = Ft, uniform across the face For a rectangular section, the area moment of inertia is $I = bh^3/12$

$M = Ft\ l$ and $c = t/2\pi$

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

Where b = the face width of the gear. For a gear tooth, the maximum stress is expected at point A, which is a tangential point where the parabola curve is tangent to the curve of the tooth root fillet called parabola tangential method. Two points can be 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 the point A [9].

For similar triangle $\tan\alpha = (t/2)/x = l/(t/2)$ where $l = t^2/4x$ (3)

Substituting (2) into (3):

$$\sigma = \frac{6F_t\frac{t^2}{4x}}{bt^2} = \frac{3F_t}{2bx} = \frac{3F_t P_d}{2bP_d x} = \frac{F_t P_d}{by} \qquad (4)$$

Where Pd = diametral pitch

$Y = \frac{2xP_d}{3}$ = Lewis form factor

Equation (4) 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° 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 Kc. The concentrated stress on the tooth fillet is taken into account by $Kc$and a geometry factor Yj, where $Yj = Y / Kc$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 [10].

$$\sigma = F_t P_d K_a K_s K_m / b Y_j K_v$$

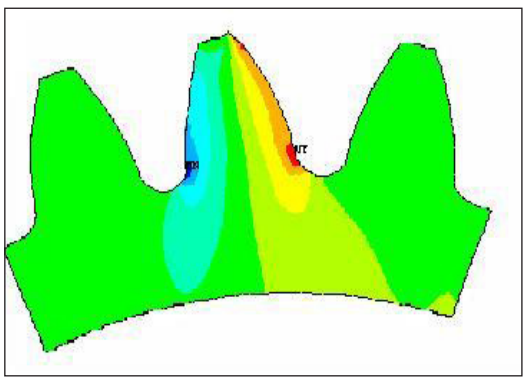
WhereKa= application factor,Ks= size factor,
Km= load distribution factor, Kv= dynamic factor,
Ft = normal tangential load, Yj= Geometry factor

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 [3].

FEM Models

The Two Dimensional Models

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.



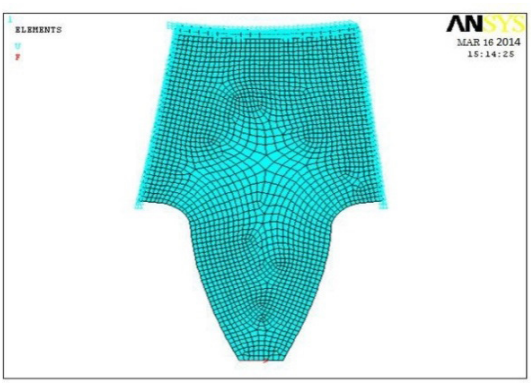


Fig. 10 FEM gear tooth bending model with 3 teethFig. 11 A two dimension tooth from a FEM model

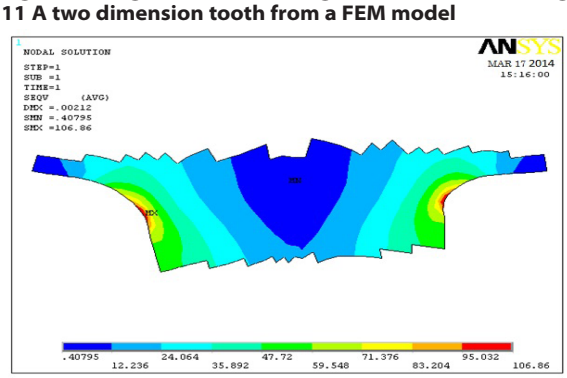


Figure 12 Von Mises stresses on the root of tooth

In the procedure for generating a FEM model for bending stress analyses, the equations used to generate the gear tooth profile curve. 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 10 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 11 shows one tooth FEM model and Figure 12 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 1, which are compared with the results from the Lewis Formula.

The 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 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 14 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.

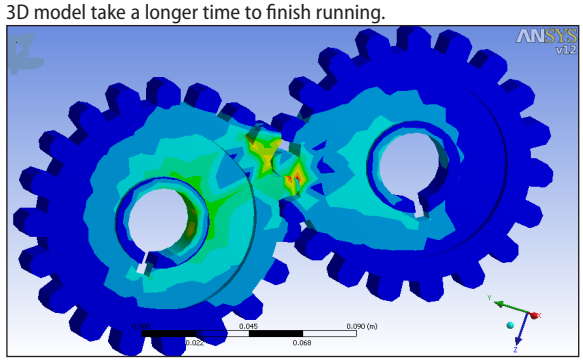


Figure 13 FEM bending model with meshing

From the stress distributions on the model, the large concentrated stresses are at the root of the tooth. Figure 14 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 1.

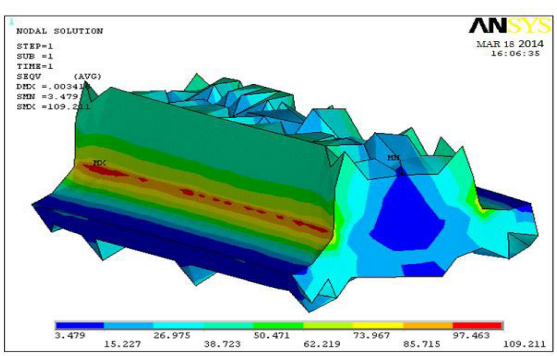


Figure 14 Von Mises stresses on the root of tooth Results using AGMA Analyses

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. (4) 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 number of teeth is carried out. First, the number of gear teeth is 28. The meshing spur gear has a pitch radii of 50 mm and a pressure angle of 200 .The gear face width, b = 38.1mm. The transmitted load is 2500 N.

Pd=N/d=28/50*2=0.2

$\sigma_t = F_t P_d K_a K_s K_m / b Y_j K_v = 2500*0.28*1.2*1.15/38.1*0.37*0.8=103.05$ Mpa

Detailed investigations, including the effects with the two different numbers of teeth on the tooth root stress were carried out. If the

number of teeth is changed from 28 to 23 and the other parameters were kept the same.

$\sigma_t = F_t P_d K_a K_s K_m / b Y_j K_v = 2500*0.23*1.2*1.2*1.15/38.1*0.37*0.8 = 84.43$ Mpa

If the number of teeth is changed from 28 to 25 and the other parameters were kept the same.

$\sigma_t = F_t P_d K_a K_s K_m / b Y_j K_v = 2500*0.25*1.2*1.2*1.15/38.1*0.37*0.8 = 91.77$ Mpa

If the number of teeth is changed from 28 to 34 and the other parameters were kept the same.

$\sigma_t = F_t P_d K_a K_s K_m / b Y_j K_v = 2500*0.34*1.2*1.2*1.15/38.1*0.37*0.8 = 124.81$ Mpa

If the number of teeth is changed to 37, with the other parameters kept the same.

$\sigma_t = F_t P_d K_a K_s K_m / b Y_j K_v = 2500*0.37*1.2*1.2*1.15/38.1*0.37*0.8 = 132.149$ Mpa

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 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 1.

Table 1 Von Mises Stress of 3-D and 2-D FEM bending model

Num. of teeth	Stress3D (2D) (ANSYS)		Stresses (AGMA)	% Difference 3D (2D)	
23	86.418	(85.050)	84.429	2.35	(0.74)
25	95.802	(91.129)	91.770	4.39	(0.69)
28	109.21	(106.86)	102.78	6.26	(3.97)
31	123.34	(116.86)	113.79	8.39	(2.69)
34	132.06	(128.46)	124.80	5.82	(2.93)
37	143.90	(141.97)	132.15	8.89	(7.43)

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 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.

REFERENCES

1. Sachindra Kumar, Anjani Kumar Singh,Nitesh Kumar, Sushil Patel, Ajit Kumar "Contact Stress Analysis of Involute Gear through FEM" International Journal Of Scientific Research And Education ||Volume||2||Issue|| 6||Pages 948-962 |||2014|| ISSN (e): 2321-7545 | | 2. Ishikawa, H., 1989, "An Effective Method for Three-Dimensional Finite Element Analysis of a Gear Pair under Partial Contact", JSME International Journal Series III Vol.32, No. 2, pp 246-252. | 3. Anon., 1965, "Surface Durability (Pitting) of Spur Teeth", AGMA Standard 210.02. | 4. Hamrock, B. J., Jacobson, S. R., "Fundamentals of Machine Elements". | 5. Sirichai S., 1999, "Torsional Properties of Spur Gear in Mesh using nonlinear Finite Element Analysis", Ph.D. Thesis, Curtin University of Technology. | | 6. Anon., 1965, "Gear Scoring Design Guide for Aerospace Spur and Helical Power Gear", AGMA Information Sheet 217.01. | 7. Kasuba, R., "An Analytical and experimental Study of dynamic Loads on Spur Gear Teeth", Ph.D., University of Illinolis. | 8. Aida, T., 1969, "Fundamental Research on Gear Noise and Vibration I", Transaction of the Japanese Society of Mechanical Engineering, 35, pp. 2113 2119. | 9. Tobe T., Kato, M., Inoue K., 1978, "Bending of Stub Cantilever Plate and some Application to Strength of Gear Teeth", Journal of Mechanical Design, Vol. 100, pp 374-381. | 10. 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. | 11. Vijayarangan S., Ganesan N. 1994, "Static Contact Stress Analysis of A Spur Gear Tooth Using the Finite Element Method Including Friction Effects", Computer &Struture Vol. 51 No. 6 pp767-770 | |