DEVELOPMENT OF MODELS OF GEAR TEETH FOR FINDING CONTACT STRESSES, USING ANSYS

Mayuresh Ghosalkar[1] | Dr. Pradeep Kumar Patil[2]
[1](Mechanical Engineering Department, RGPV SVCE, Indore, India)
[2](Mechanical Engineering Department, RGPV SVCE, Indore, India)

Abstract— *Proposed project is to modify design and FE analysis of S shaped agitator to perform pulp stirrer and feeder operations in pulping equipment. Generally in pulping industry pulp is mostly feed to different processes by screw feeder mechanism, here also screw feeder is already designed and standardized, in that standard screw mechanism upper hollow tank collecting the pulp which is very unmixed and not properly stirred after bleaching and other operation in chemical as well as in mechanical pulping. So, to make this purpose solve practically in existing setup, we need to design an agitator which will help to distribute the pulp in feeder opening of screw conveyor. Now while distribution pulp must be previously stirred for that on the top of agitator assembly stirrer is to be mounted preliminary it can be T shaped vertically mounted as shown. Stirrer will be designed in such a way that it must hold load of coming pulp from the top and must be rotate without any bending deformation occurred in the agitator.*

Keywords— FE Analysis; Pulp Stirrer and Feeder Operations; Screw Conveyor

1. INTRODUCTION

Gears are essential to the global economy and are used in nearly all application where the power transmission is required such as automobiles, industrial equipment airplanes helicopters and marine vessels. Frequency of product model changeover, also called time-based competition has become a character feature of global manufacturing of new product development in automotive aerospace and other industries. This forces gear manufacturer to respond with improved gear. The increasing demand for quite power transmission in machines, vehicles, elevators and generators, has created a growing demand for a more precise analysis of the characteristics of a gear system. In the automobile industry, for 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 and hence the transmission inaccuracy of gear. The reduction of noise through vibration control can only be achieved through research efforts by specialists in the field.

2. EASE OF USE

A. Objectives of the Research

Many research methods uses detailed finite element methods to predict transmission error, but without the prediction being fully integrated within advanced optimization procedures, as they require complete automatic FE solutions. 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, contact and bending stresses, torsional mesh stiffness and transmission errors. The objective of this work is to analyze gear using analytical method which required number of assumptions and simplifications which aims at getting the maximum stress values only but gear analyses are multidisciplinary including calculations related to the tooth stresses and to failures like wear. In this thesis, contact stress analysis of Spur gear is studied using finite element analysis and an attempt is made to analyze contact stress using Hertz theory. The main focus of the current research as developed here is:

- To develop and to determine appropriate three dimensional 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 calculate of gear bending stress using Lewis equation & hence check the feasibility of modified gear profile.
- To compare the static transmission errors of standard & modified profile of the gear teeth

B. Focus of the work

- 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, and prognosis, reliability and fatigue life.

3. PROCEDURE FOR FINITE ELEMENT ANALYSIS

1. Pre-processing: The user constructs a model of the part to be analyzed in which the geometry is divided into a number of discrete sub regions, or elements," connected at discrete points called nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads. These models can be extremely time consuming to prepare, and commercial codes vie with one another to have the most user-friendly graphical pre-processor" to assist in this rather tedious chore. Some of these pre-processors can overlay a mesh on a pre-existing CAD model, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process.

2. Solver: The dataset prepared by the pre-processor is used as input to the finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations

$K_{ij} \quad u_i = f_i$

where u and f are the displacements and externally applied forces at the nodal points. The formation of the K matrix is dependent on the type of problem being attacked, and this module will outline the approach for truss and linear elastic stress analyses. The static analysis of gear can be carried out exactly, and the equations of even complicated gear can be assembled in a matrix form amenable to numerical solution. This approach, sometimes called matrix analysis," provided the foundation of early FEA development. Matrix analysis of trusses operates by considering the stiffness of each truss element one at a time, and then using these to determine the forces that are set up in the truss elements by the displacements of the joints, usually called nodes" infinite element analysis. Then noting that the sum of the forces contributed by each element to a node must equal the force that is externally applied to that node, we can assemble a sequence of linear algebraic equations in which the nodal displacements are the unknowns and the applied nodal forces are known quantities. These equations are conveniently written in matrix form, which gives the method its name:

The Kij coefficient array is called the global stiffness matrix, with the ij component being physically the influence of the jth displacement on the ith force. The matrix equations can be abbreviated as

$K_{ij} u_j = f_i \qquad \text{or} \qquad K\,u = f$

using either subscripts or boldface to indicate vector and matrix quantities. Either the force externally applied or the displacement is known at the outset for each node, and it is impossible to specify simultaneously both an arbitrary displacement and a force on a given node.

$$\begin{bmatrix} K_{11} & K_{12} & \cdots & K_{1n} \\ K_{21} & K_{22} & \cdots & K_{2n} \\ \vdots & \vdots & \ddots & \vdots \\ K_{n1} & K_{n2} & \cdots & K_{nn} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ \vdots \\ u_n \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \\ \vdots \\ f_n \end{Bmatrix}$$

Using either subscripts or boldface to indicate vector and matrix quantities. Either the force externally applied or the displacement is known at the outset for each node, and it is impossible to specify simultaneously both an arbitrary displacement and a force on a given node. These prescribed nodal forces and displacements are the boundary conditions of the problem. It is the task of analysis to determine the forces that accompany the imposed displacements and the displacements at the nodes where known external forces are applied.

3. In the earlier days of finite element analysis, the user would pore through reams of numbers generated by the code, listing displacements and stresses at discrete positions within the model. It is easy to miss important trends and hot spots this way, and modern codes use graphical displays to assist in visualizing the results. A typical postprocessor display overlay colure contours representing stress levels on the model, showing a full field picture similar to that of photo elastic experimental results.

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 epicycle 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.
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.
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 modeling 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 significant 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. This loading system is decomposed into finite Fourier series with the number of harmonics being equal to the number of teeth.

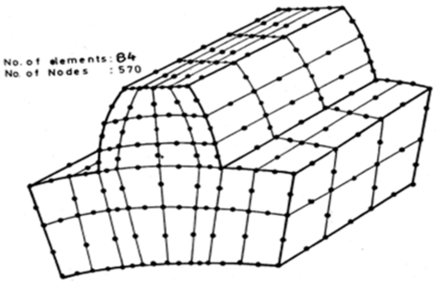


Figure 3.1 :-Discretisation of spur gear tooth (20 node element)

One substructure of spur gear teeth is discretized as shown in Fig 3.1. One substructure is treated as a three-dimensional stress problem with 570 nodes and 84 node elements (Fig. 3.2)

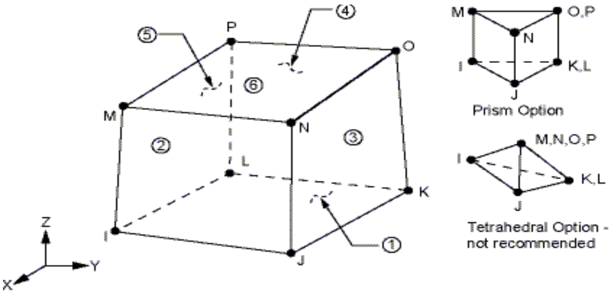


Figure 3.2:- Tetrahedral Mesh with 3D Brick 8 node Structural solid element

4. Contact and Bending Analysis of Gear tooth using Finite Element Method

4.1 Contact analysis

There are mainly two types of failure modes are there due to which the spur gear tooth may break they are,
1. Tooth Breakage - from excessive bending stress, and
2. Surface Pitting/Wear - from excessive contact stress.
Contact stress: - when two bodies having curved surfaces are compressed together, point or line contact changes to area contact & the stress developed in the two bodies are three dimensional. Contact –stress problem arise in the contact of a wheel & rail, in automobile valve cams tappets, in the meting teeth, & in the action of rolling bearings. Typical failures are seen as crack, pits or flanking in the surface material. In general, there are three basic types of contact modeling application as far as practical application 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 discrimination or a compatible mesh. Point to surface contact was used in this chapter.
Surface-to-surface contact is typically used to model surface-to-surface contact applications of the rigid-to-flexible classification. There are some difficult while dealing with contact problems 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). The most general case of contact stress occurs when each containing body has a double radius of curvature that is when the radius in the plane of the rolling

is different from the radius in a particular plane, both planes taken planes through the axis of the contacting force. The stress determine from these are also known as Hertzian stresses.

4.1.1 Contact Analysis of Two Cylinders

One of the main gear tooth failure is pitting which is a surface fatigue failure due to many repetition of high contact stresses occurring in the gear tooth surface while a pair of teeth is transmitting power. Contact failure in gears is currently predicted by comparing the calculated Hertz contact stress to experimentally determined allowable values for the given material. 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. The contact pressure is intensified near the pitch circle, where the contact is pure rolling with zero sliding velocity.

Consider two solid cylinder of diameter d1& d2 of length l, press together with force F. As shown in figure, the area of contact is a narrow rectangular of width 2b & length l, and the pressure distribution is elliptical. 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.

In machine design, contact problems frequently occurs when two members with curved surfaces are deformed when pressed against one another giving rise to an area of contact under compressive stresses. Of particular interest to the gear designer is the case where the curved surfaces are of cylindrical shape because they closely resemble gear tooth surfaces and the result obtained here is due to Hertz and so frequently known as Hertzian stresses.

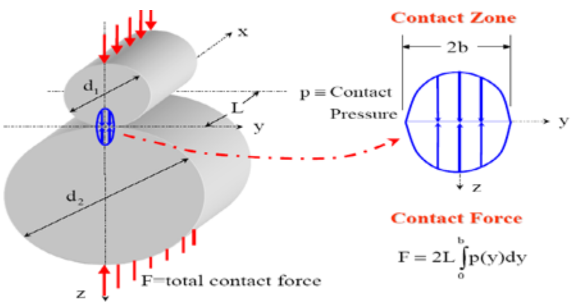


Fig. 4.1:-Cylinders in contact under compression

4.1.2 Hertz Contact Stress Equations

When two solid cylinders of diameters d1,d2 and length l are pressed together with a force F, the area of contact is a narrow rectangle of width 2b and length l, and the pressure distribution is elliptical [2].

The half width b is given by the equation,

$$b = \sqrt{\frac{\frac{2F}{\pi L}\left[(1-\nu_1^2)/E_1 + (1-\nu_2^2)/E_2\right]}{\frac{1}{d_1}+\frac{1}{d_2}}}$$

And the maximum pressure is given by,

$$p_{max} = \frac{4\times F}{\pi \times b \times l}$$

Above equations applied to cylinder and plane surface, such as rail by making d=∞ and also to the contact of cylinder and an internal cylindrical surface when d is negative . The stresses state on z-axis is given by the equations,

Along X axis

$$\sigma_X = -2v\,.\,p_{max}\left(\sqrt{1+\frac{Z^2}{b^2}} - \frac{Z}{b}\right)$$

Along Y axis

$$\sigma_y = -p_{max}\left[\left(2-\frac{1}{1+\frac{z^2}{b^2}}\right)\sqrt{1+\frac{z^2}{b^2}} - 2\frac{z}{b}\right]$$

Along Z axis

$$\sigma_z = \frac{-p_{max}}{\sqrt{1+\frac{z^2}{b^2}}}$$

F = Applied load

V_1 =Poisson’s ratio of materials of the cylinders 1

d_1 =diameter of cylinder 1

d_2 =diameter of cylinder 2

E_1= modulus of elasticity of material of cylinder 1.

All the above equations are essential in validating the results from a numerical analysis. Although a numerical analysis can greatly enhance the ability to analyze a structure, the results obtained must be accurate.

4.1.3 ANSYS model of two cylinders

Contact stress is generally the deciding factor for the determination of the requisite dimensions of gears. The objective of the contact stress analyses is 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 is modeled in figure. To reduce computer time, only half cylinders were meshed in the model as shown in Figure. The fine meshed rectangular shaped elements were generated near contact areas shown .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.

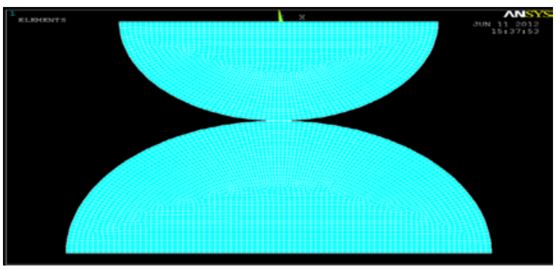


Figure 4.2:-Mesh of the two cylinders using ANSYS

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 parameters used in this analysis are defined as follows: Diameter d_1 =105 mm, d_2 = 195 mm,$V_{1,2}$= 0.30, $E_{1,2}$ = 200 GPa ,F =1*10^5 Newton. Plane 42 elements were used to mesh the geometry, which are four node quadrilaterals. Each node has two degrees of freedom; displacement in the horizontal and vertical directions. In the area away from the contact region, the average element size was 0.01 in. However, the mesh near the contact zone was much finer, with an average element size of 1.35x10-5 in. Point-to-line contact was modeled with Contact 175 and Target 169 elements. The Contact 175 elements were found to give the best results. Each element is composed of a single point. These were applied at the contact zone on the top cylinder. Target 169 elements represent a straight line between two nodes and were applied at the contact zone on the lower cylinder. Because the contact elements represent a spring between the two bodies, stiffness is applied to each contact pair. The default stiffness was adequate for this analysis, which was the modulus of elasticity of the upper cylinder.

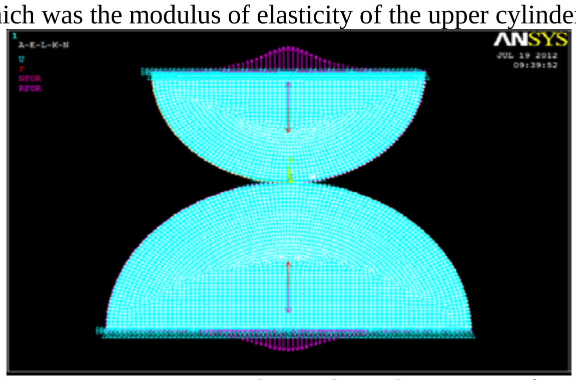
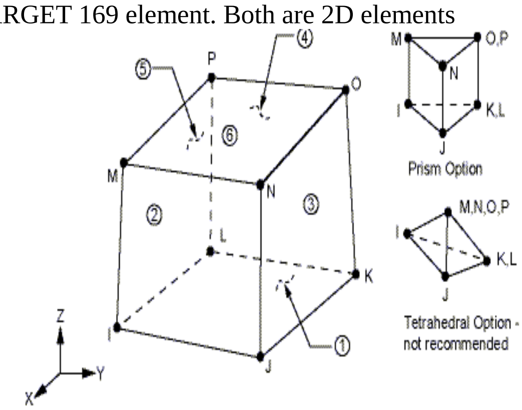


Figure 4.3:- Boundary conditions during contact of two cylinders

In the given Figure Source is pinion with CONTACT 175 (Point to surface contact) element and gear is meshed with TARGET 169 element. Both are 2D elements



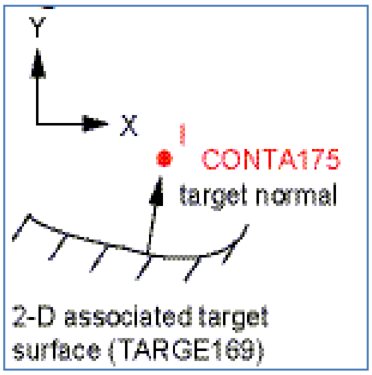


Figure 4.4:- Target element for gear teeth meshing

As ANSYS computes the solution, it is constantly testing the status of each contact pair and updating as necessary. This requires the non-linear solver to apply the load in small increments. After each load step is applied, ANSYS determines if the contact element has penetrated the target. If the load step is too big, the contact and target elements could move far enough away from each other that ANSYS does not recognize that contact occurred. This results in a solution, which does not converge or is unconstrained. To ensure convergence, this case utilized 24 equal load increments.

Now to check the Validation of Contact Pressure for Base case and also to check the accuracy of the present method first we have to calculate maximum contact pressure with the help of Hertz theory and compare the result with ANSYS.

Now first to Calculate Contact pressure with the help of Hertz theory, by using base specifications given by company, Consider two circular elastic discs under two-dimensional contact are analyzed, and the numerical solutions are compared with that of the Hertz theory.

The calculations are carried out under a plane strain condition with a Poisson's ratio of 0.3 using eight-node iso-parametric elements.

- Gear dimensions: No. Of teeth: 28 (m=3.75)
- Gear Material: Steel Alloy (Su=410 e6 N/m2)
- Tangential Force: 2500N (562.02Pounds)
- Normal force:- 1*10^6 N
- Application: Medium (loading)
- Dimensions: module: 3.75,
- Addendum: 1*module,
- Dedendum: 1.25*module,
- Pitch Circle Dia: 105 mm

By using Hertz contact stress theory for finding maximum contact pressure,

$$p_{max}=\sigma_H= 0.564\frac{\sqrt{F\left({}^{1}/{r_1}+\frac{1}{r_2}\right)}}{\sqrt{\frac{1-v_1^2}{E_1}+(1-v_2^2)/E_2}}$$

$$= 0.564\frac{\sqrt{1\times10^6\left(\frac{1}{52.5}+\frac{1}{97.5}\right)}}{\sqrt{\frac{1-v_1^2}{2*10^6}+(1-0.3^2)/2*10^6}}$$

$$= 325.735 \text{ MPa}$$

Where,

F is the load per unit width,

E_1 and E_2 are the modulus of elasticity of the gear material

Comparison of Hertzian contact stress value with the ANSYS value :-

The values which are compared to those obtained by Hertz theory, are half-width b, contact pressure σ_y (the stress normal to the contact surface), σ_x (stress orthogonal to the contact pressure) and the shear stress σ_{xy} Commonly, the results are displayed as the ratio of stress to the maximum stress, or maximum pressure P_{max}. Figure shows the results obtained by the finite element analysis and those calculated by Hertz theory using Equations.

Before you begin to format your paper, first write and save the content as a separate text file. Keep your text and graphic files separate until after the text has been formatted and styled. Do not use hard tabs, and limit use of hard returns to only one return at the end of a paragraph. Do not add any kind of pagination anywhere in the paper. Do not number text heads-the template will do that for you.

Finally, complete content and organizational editing before formatting. Please take note of the following items when proofreading spelling and grammar:

5. Result Table

Case	Module	Structural Analysis			Wear and Tear Analysis		
		Deflection	*Stress*	*Shear Stress*	*Deflection*	*Contact Stress*	*Von-mises*
Base	3.75	0.002	62.3e3	74.9 e3	0.95 e-3	37.5	145.9
1	4	0.3e -3	48.2 e3	56.6 e3	0.55 e-3	27	113.2
2	4.5	0.132 e-3	38.2 e3	45.4 e3	0.112 e-3	5.53	22

From above table it can be shown that, module 4.5 is the feasible module for the company for designing the gear tooth.

Graph between results of deflection & respective gear module.

Scale on X-axis: 1 Div = 1module,

Y-axis: 1 Div = $0.001 * 10^{-3}$ mm

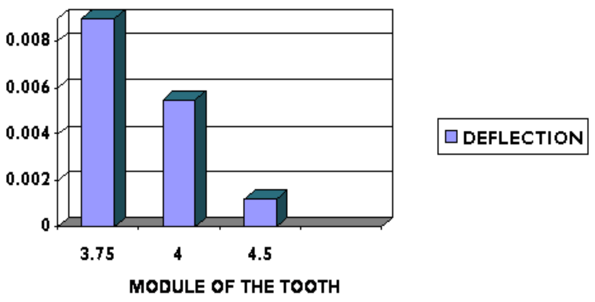


Figure 5.1Result of deflection of different tooth profiles

Form this graph we can see that value of deflection is minimum in concept 2 having gear teeth module 4.5 hence theses profiles can be taken optimize profile. As deflection is minimum it certainly reduce the transmission error, but we also have to consider the effect of stress due to this profile modification.

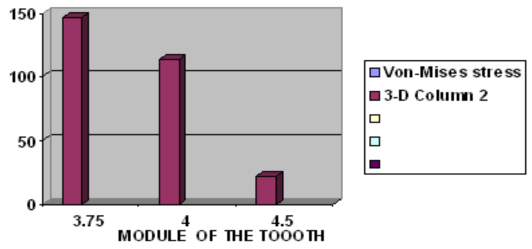


Figure 5.2 Result of maximum stress of different tooth profile

The following observations are given from the above table,

- The Su for the Alloy steel (given) 410MP=41e3N/cm2, For case 2, design stress is lower than the Su.
- For Case 2 combination, contact stress is very less than for the failed base case.
- The shear stress at tangential load and von -mises stress due to contact is quite lower than the base case values

From the above graph results the module is varied from 3.75 to 4.5 .It is seen that with decrease in module from 3.75 to 3,the stress always increases, which is not feasible for the given objective function. Gradually the module is increased from 3.75 to 4.5 as material weight is also considered; the stresses are gradually decreased with deflection. Hence it is seen that from the above observations, the module is important geometrical parameter during the design of gear. As it is expected, in this work the maximum contact stress decreases with increasing module and it will be higher at the pitch point. As a result, based on this finding if the contact stress minimization is the primary concern and if the large power is to be transmitted then a Spur gear with higher module is preferred.

References

[1] G. Eason, B. Noble, and I.N. Sneddon, “On certain integrals of Lipschitz-Hankel type involving products of Bessel functions,” Phil. Trans. Roy. Soc. London, vol. A247, pp. 529-551, April 1955. (*references*)

[2] J. Clerk Maxwell, A Treatise on Electricity and Magnetism, 3rd ed., vol. 2. Oxford: Clarendon, 1892, pp.68-73.

[3] I.S. Jacobs and C.P. Bean, “Fine particles, thin films and exchange anisotropy,” in Magnetism, vol. III, G.T. Rado and H. Suhl, Eds. New York: Academic, 1963, pp. 271-350.

[4] K. Elissa, “Title of paper if known,” unpublished.

[5] R. Nicole, “Title of paper with only first word capitalized,” J. Name Stand. Abbrev., in press.

[6] Y. Yorozu, M. Hirano, K. Oka, and Y. Tagawa, “Electron spectroscopy studies on magneto-optical media and plastic substrate interface,” IEEE Transl. J. Magn. Japan, vol. 2, pp. 740-741, August 1987 [Digests 9th Annual Conf. Magnetics Japan, p. 301, 1982].

[7] M. Young, The Technical Writer’s Handbook. Mill Valley, CA: University Science, 1989.