Finite Element Analysis of Large Spur Gear Tooth and Rim With and Without Web Effects-Part I

Ravichandra Patchigolla and Yesh P. Singh

Department of Mechanical Engineering & Biomechanics
The University of Texas at San Antonio

Abstract

A finite element modeling approach is developed for determining the effect of gear rim thickness on tooth bending stresses in large spur gears. These low addendum gears are used in cement plants, sugar mills, ball mills, coal mills, kilns, grinding mills, copper converters, and anode furnaces. A program is developed using ANSYS Parametric Design Language (APDL) to generate 1, 3, and 5 tooth segment finite element models of a large spur gear. A controlled meshing approach is used with free and mapped meshing capabilities of ANSYS to generate 2-D model of the gear tooth with 4-node (PLANE42) elements. As same configuration exists at all sections along the face width of the gear, the 2-D models are extruded to obtain 3-D models using 8-node (SOLID45) elements. The controlled meshing approach employed here has the following advantages: it prevents high stress at the point of application of load, avoids too many elements in the low stressed region, and generates a fine mesh in the high stressed fillet region. This paper describes details of meshing and modeling techniques employed. Part II of this paper emphasizes on results of the finite element analyses and effect of rim thickness on gear tooth bending stresses.

Introduction

A number of researchers have worked on gear tooth failure and used experimental, analytical and numerical techniques to determine the stresses in the gear tooth. Most commonly used experimental techniques include photoelastic and strain gages, and finite element method was the mostly used numerical technique.

Photoelastic technique was widely used for many years. Baud and Timoshenko[1] introduced the photoelastic technique to examine the stress concentration effect at the gear tooth fillets. Sopwith and Heywood[2] used photoelastic technique to develop a fillet stress formula that accounted for some pressure angle unbalance. Kelly and Pederson[3] improved this formula by employing more realistic tooth shapes in their photoelastic models. Drago and Luthans[4] conducted experiments using 2 and 3 dimensional photo elastic techniques to evaluate the combined effects of rim thickness and gear pitch diameter on tooth root and fillet stresses. They calculated stresses for the load applied at LPSTC, HPSTC and Pitch point along the tooth profile. The main drawback of this method is that the experimental investigation is time consuming and it is very difficult to construct and prepare the models for investigation.

Many investigators have used different finite element approaches in evaluation of the gear tooth stresses for a long time. Wilcox and Coleman[5] used analytical method of finite elements in analyzing the gear tooth stresses. Quadrilateral elements have generally been used in two dimensional models. In regions of anticipated high stress gradients they incorporated more elements and low densities in regions of low stress gradients. They developed a new stress formula based on the stresses obtained from finite element analysis, which takes tooth shape and loading condition into account to evaluate the tensile stress in the fillet region. Oda[6] et al. analyzed the root stresses on the fillet of gear teeth as a two dimensional elastic problem by means of the FEM with typical triangular elements. They also measured these stresses experimentally with strain-gage method by carrying out a static bending test. These stresses were analyzed for spur gears of different rim thickness. The effects of rim thickness on root stresses and on the critical section were studied. Their results obtained by FEM confirm with results measured by strain gage. Chong[7] et al. used finite element method to model the rack teeth as an example of thin-rimmed spur gear and to confirm the results of the approximate formula. They investigated the influences of radius of curvature of tooth fillet, pressure angle, and loading position on tooth flank on the tooth fillet and root stresses under a single and double tooth pair meshing. They proved that the formula was not valid when the load was applied extremely near to the tooth fillet. Chang[8] et al. used SAP IV finite Element technique to investigate the fillet and root section stresses for a variety of loading positions, mounting support, different fillet radii and rim thickness on a single tooth model. They also studied the surface stress distribution on the entire tooth profile for the tip and pitch point loading. Reddy[9] et al. used 6-node isoparametric plane stress triangular element to build the finite element model of a thin rim spur gear. They calculated the effect of variation of rim thickness of a 5-tooth segment model on the location and magnitude of maximum bending stress value. Filiz and Eyercioglu[10] evaluated the effects of module, contact ratio, fillet radius, pressure angle and teeth numbers of driving and driven gears on gear tooth stresses for three different loading conditions (i.e., point load, distribute load and simulated contact) using the finite element method. MSC/NASTRAN was used for finite element analysis. Based on their study, they modified Chabert and Tobe's formula in order to include the effects of the above variables and presented a new formula that gave closest results to their study. Vijayarangan and Ganesan[11, 12] employed the FE approach for stress analysis of a composite spur gear. They also used Lagrangian multiplier technique along with 2D FE method to evaluate tooth contact stresses. Triangular elements were used to discretize the spur gear tooth sector for the tooth contact stress analysis. Gordana[13] determined the actual state of stress in the spur gear tooth root fillet by use of 3D finite element approach. His model used parabolic triangle, quad brick and parabolic tetrahedron solid elements and also used both h and p-convergence approaches. He studied the effect on tangential stress component, equivalent von Mises stress and axial stress in the direction of gear axis in the fillet area of the number of teeth, addendum modification factor, rim thickness and the tooth face width. Shuting[14] developed a 3D finite element model of a thin-rimmed gear using 11-node solid element to perform the deformation and bending stress analysis in a standard involute spur gear. He analyzed the gears deformations and stresses at every part of a whole gear deformation model and also presented the effect of rim thickness on bending stresses both at the root and the joint of the rim and web. Shuting[15] also performed loaded tooth contact analysis of a 3 dimensional, thin-rimmed gear by combining a mathematical programming with 3D FEM.

The research work by most of the investigators implied that diametral pitch, shape of the tooth profile, highest location of full load on the tooth profile and fillet geometry of the gear tooth influenced the bending strength of the gear tooth. For the gears with thin rim, rim thickness is another significant factor due to rim deflections.

The gears considered by most of the authors are either small or they having less number of teeth. In this study a large spur gear with 192 teeth and diametral pitch of one teeth per inch is considered. These gears are used in the kiln, grinding mill drives, mining industry's etc. In such large and low addendum gears making a solid gear is not possible. Therefore rim thickness is optimized to achieve constant bending stress at the root.

In this paper finite element modeling of 1, 3 and 5 tooth segment of the large spur are discussed.

Finite Element Modeling

The gear on which rim thickness effect has to be examined is a large spur gear with 192 teeth and diametral pitch of 1. The addendum, dedundum, arc tooth and tooth space thickness are not standard. Table 1 summarizes the gear mesh geometry. A program is developed using ANSYS Parametric Design Language (APDL) to generate 1, 3 or 5 tooth segment finite element models of a large spur gear.

Generation of Spur Gear Tooth Profile

The gear tooth involute profile is modeled in ANSYS. In order to generate involute profile using ANSYS, 51 keypoints are created in the range of dedundum and addendum circle radii representing the involute profile. These points represent radii values in the range of dedundum and addendum circle. All these radii values are stored in an array '*r*' of size 51x1 using "*VFILL" command. The coordinates of each point on the involute profile can be obtained by evaluating involute and pressure angle values at each radii value. Involute (*inv*) and pressure angle (θ) values are evaluated using the following relations.

$$inv\theta = \tan(\theta) - \theta, \qquad (1)$$

Where $\theta = \cos^{-1}\left(\frac{r_G \times \cos\Phi}{r}\right)$, (2)

θ and Φ are in radians.

Knowing the involute and pressure angles at all the radii values from the above expressions, the arc tooth thickness at each radii value can be obtained from the following expression[17].

$$T_r = (2 \times r)\left[\frac{T}{2 \times r_G} + inv\Phi - inv\theta\right], \quad (3)$$

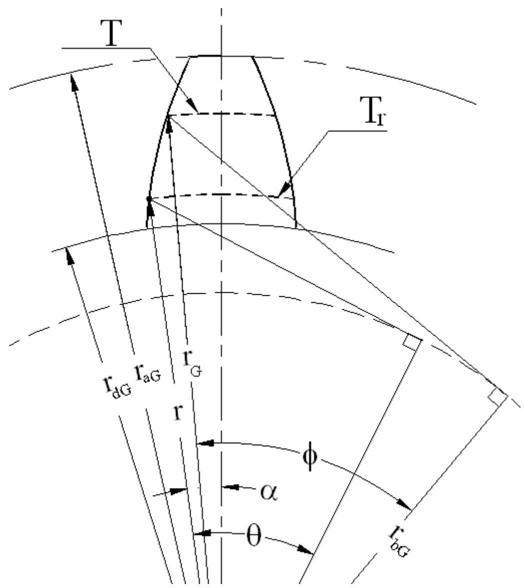

Figure 1 Geometry of Spur Gear Tooth

Obtaining the tooth thickness at each radii value (Figure 1), the angle subtended by the half tooth thickness values at the center of the gear is obtained from the following relation.

$\alpha = \frac{T_r}{2 \times r}$, Where α is in radians. (4)

The coordinates of each point on the involute profile in the gear tooth coordinate system with y-axis coinciding with the gear tooth centerline are obtained from the following expressions.

$$x = r \times \sin(\alpha)$$
$$y = r \times \cos(\alpha) \quad (5)$$

All the keypoints are plotted on GUI using the X and Y coordinate values and a B-spline is generated passing through all these keypoints.

The shape of the fillet has a direct effect on maximum bending stress developed at the root. Hence the root fillet geometry is vital. The generation process decides the shape of the root fillet. Form cutting operation is employed to manufacture large gears. The shape of the root fillet is a circular arc when form cutting is employed. This can be achieved by using line fillet option in ANSYS. The portion of the involute between the clearance and the root circle and circular arc in the tooth space are the entities selected for the line fillet option.

Generation of Finite Element Model

The analysis is performed on two dimensional and three dimensional finite element models of 1, 3 and 5 tooth segments. This work uses 4-node PLANE42 elements for the 2-D analysis. Four nodes having two degrees of freedom at each node define the element. Each node has translations in the nodal x and y directions. Plane stress with unit thickness option was used for the 2D analysis. While for the 3-D models, 8-node SOLID45 elements are used. Eight nodes having three degrees of freedom at each node define the element. Each node has translations in the nodal x, y, and z directions. The number of elements and nodes in two and three-dimensional models are tabulated in Table 2.

Table 2 Number of Elements in the Finite Element Model

MODELS	NUMBER OF ELEMENTS	NUMBER OF NODES	NUMBER OF ELEMENTS (REFINED)	NUMBER OF NODES (REFINED)
1-Tooth 2D	142	172	298	342
1-Tooth 3D	7384	9116	15496	18126
3-Teeth 2D	283	337	439	507
3-Teeth 3D	14716	17861	22828	26871
5-Teeth 2D	403	481	559	651
5-Teeth 3D	20956	25493	29224	34662

Free and mapped meshing capabilities of ANSYS are employed to generate the finite element mesh. Refinement in the fillet region is performed using free mesh option and for the other regions of the gear tooth mapped meshing is employed. The meshing (generation of finite elements) is initiated after the gear tooth sector is divided into regular four sided areas capable for mapped meshing. The area model and finite element model is shown in the Figure 2.

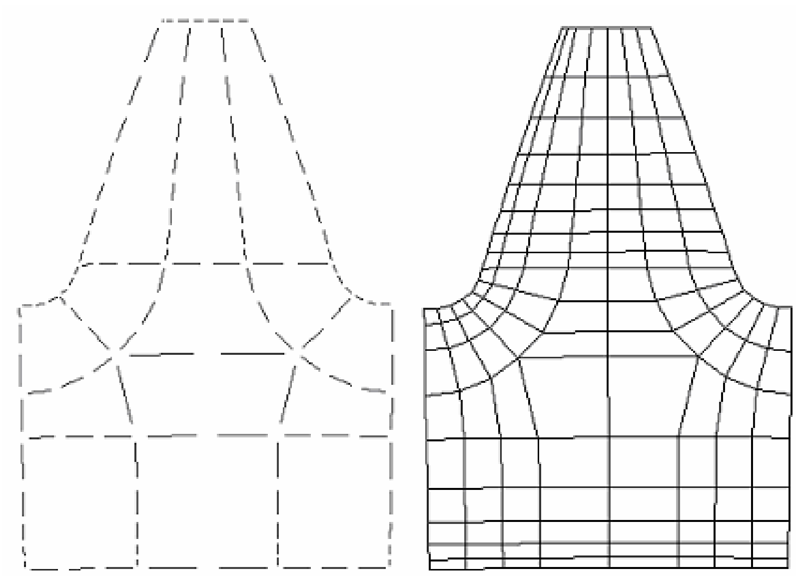

Figure 2 Area Model and Finite Element Model

The finite element mesh is generated in such a way to satisfy the following criteria so that it would reduce the overall element count in the model.

- The size of the element should decrease when approaching the fillet (maximum stress region) from the addendum circle in the radial direction.
- The size of the elements should increase when moving away from the fillet until middle of the rim portion and then decrease as approaching the rim surface in the radial direction.
- Number of elements should be less in low stress regions. The middle of the tooth is a low stress region.
- More number of small size elements in the fillet region.
- Connectivity with the other teeth should be maintained.

Modifications in the Finite Element Model

The finite element models are analyzed for the case of entire load acting at the HPSTC (Highest Point of Single Tooth Contact). To apply the load at a particular location on the involute profile there should be a nodal point at that location. If the mesh does not have a nodal point at this location the element that is intersected by the radius value of the HPSTC along the involute profile is determined. The edge length of the element along the profile is calculated and the arbitrary load is applied as a pressure on this edge (model has unit thickness). The Figure 3 shows the pressure applied on the element edge, the arc of radius HPSTC intersecting the element and constraints in the rim. The inside rim surface is left unconstrained. It can be seen that load is applied near the HPSTC but not exactly at that position. If the element size changes the position at which pressure is applied moves further away from HPSTC. The load can be applied exactly at HPSTC by providing a node at that location. The area model in the Figure 2 is modified by providing a partition in the top portion of the tooth as shown in the Figure 4. The partition is provided at a radius of HPSTC by creating an additional key point on both the involute profiles. B splines are divided at this key point. Key points are also generated at the intersection of HPSTC arc with the lines inside the tooth profile. The resulting model is subjected to mapped meshing. The modified area and meshed area models are shown in the Figure 4 with a node at HPSTC.

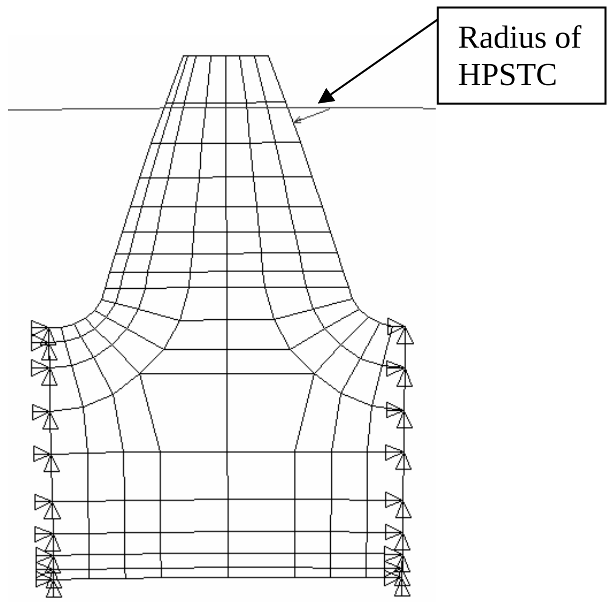

Figure 3 Pressure on the Element Edge, HPSTC Circle Arc, and Rim Constraints

FE model is analyzed by applying constraints and an arbitrary load (Figure 4) at the HPSTC. The results of the analysis showed high stress value at the point of application of load. High stress is produced at this location because load is applied along a single edge of the element. High stress at this location can be prevented by introducing a triangle element at the HPSTC, which enables distribution of the load along the slanted edges of the triangle. The apex of the triangle element coincides with the node at the HPSTC. The area model of the Figure 4 is modified in such a way to enable the generation of triangle element at this location. The modified area model is shown in the figure 5. The analysis of this model produced a stress value, which was more than 50% lower than the stress value produced by the model in Figure 4 at the node of load application. Based on the above discussion, meshed model of Figure 5 is chosen as the middle tooth for the actual analysis of 1, 3 and 5 tooth segment finite element models. The two dimensional finite element models are extruded to produce three dimensional finite element models.

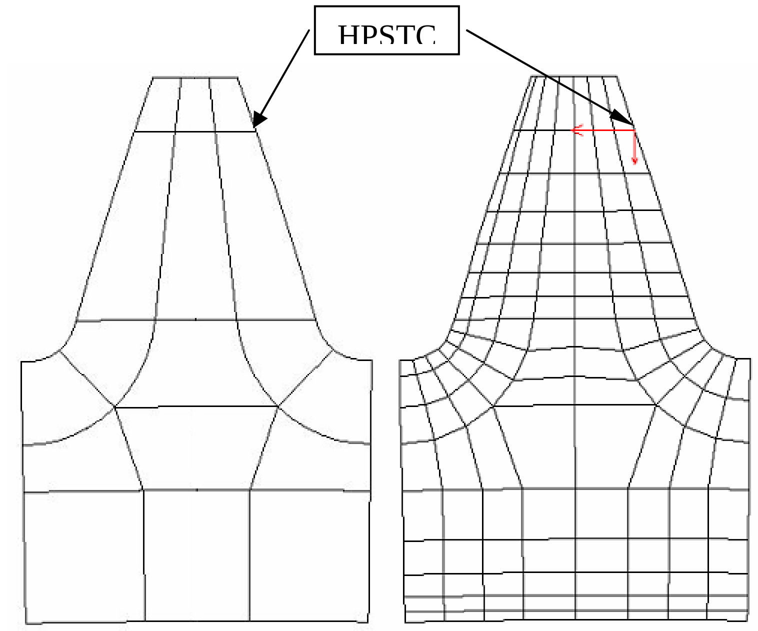

Figure 4 Modified Area Model and Finite Element Model

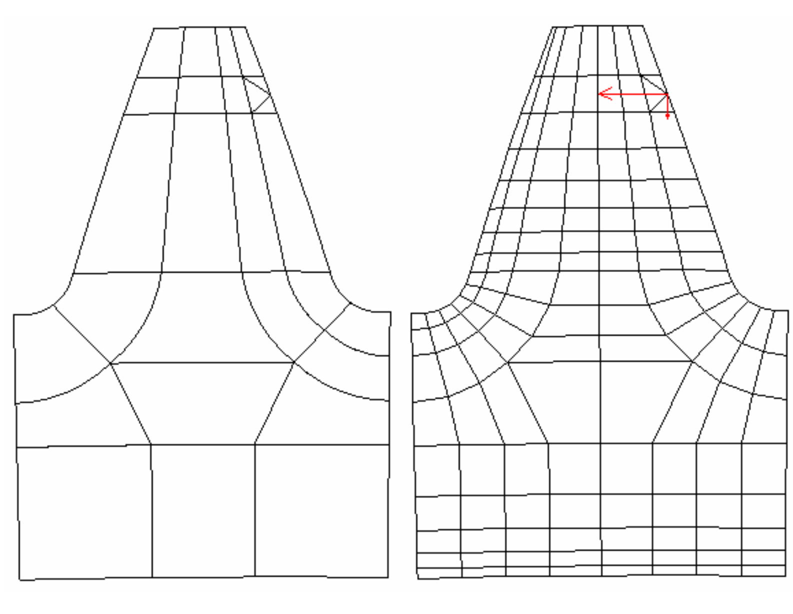

Figure 5 Modified Area Model and Finite Element Model of Figure 3

Refinement in the Fillet Region

Ansys has the capability of adding more elements at any specified node or element. The density of the elements is given in terms of levels from 1 to 5. Value of one being minimal refinement at the specified location and 5 provides maximum refinement. The elements are generated in a haphazard manner and even violate shape limits. The Figure 6b shows the mesh using Ansys refinement. Shaded elements violated element shape limits. In order to avoid element shape warnings, lines in the fillet region represented by 1, 2 and 3 in Figure 6c are divided based on number of elements required in this region. Using the Free mesh option the elements are forced to form as a web (Figure 6d) and thus following a regular pattern. The division of the line segments is increased in the fillet region (Figure 6c) until the stress values of consecutive analysis are converged. The compressive side fillet region converges for 14 divisions and the tensile side converges for 6 divisions.

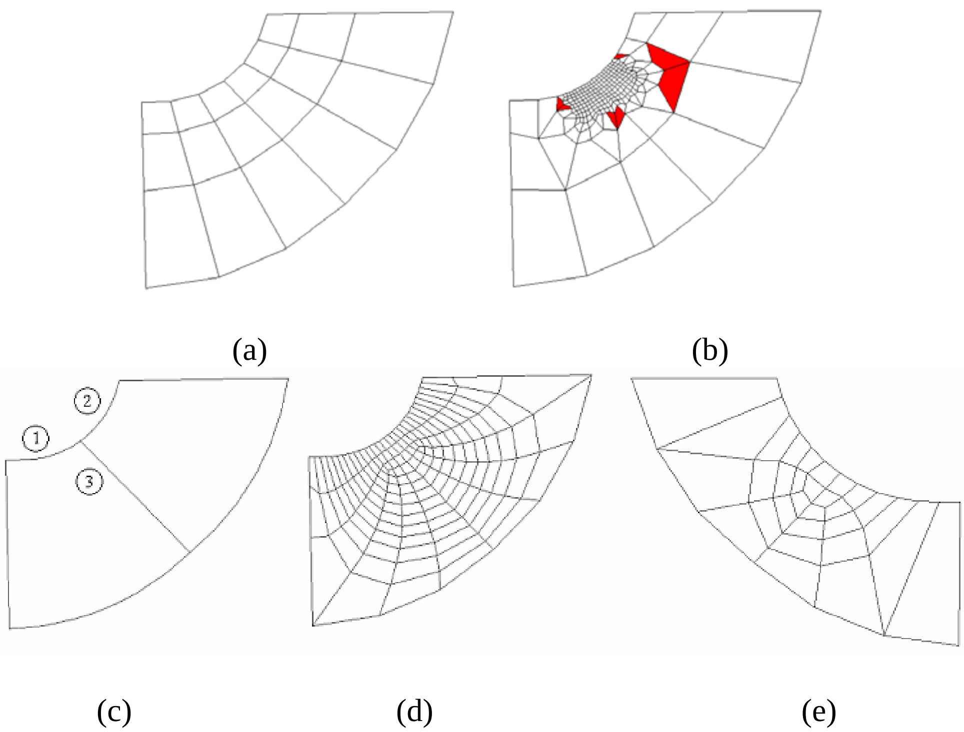

Figure 6 (a) Elements in the Fillet Area (b) ANSYS Refinement(c) Lines Attached to the Fillet (d) Refining by Division of Line Segments on Compression Side (e) Refining by Division of Line Segments on Tension Side.

Summary

Finite element based approach was used to investigate the effect of the gear rim thickness on the tooth bending stresses in large spur gears. A program was developed using ANSYS Parametric Design Language (APDL), which can generate two dimensional or three dimensional finite element model of 1, 3 or 5 teeth segment with user defined rim thickness value. Models were constrained on the radial sides in the rim portion and also on the nodes located circumferentially along the bottom surface at the rim-web interface. The models are studied for the case of full load acting at Highest Point of Single Tooth Contact (HPSTC). A different meshing approach was developed in the generation of finite element grid.

References

1. Timoshenko, S., and Baud, R. V., 1926, "Strength of Gear Teeth," Mech. Eng., Vol. 48, No.11, p.1108
2. Sopwith, D.G, and Heywood, R.B., 1948, "Loads and Stresses in Screw Threads and Projections," Applied Mechanics Division of the Institution of Mechanical Engineers, Vol.159.
3. Kelly, B.W., and Pedersen, R., 1957, "The Beam Strength of Modern Gear Tooth Design," SAE Earthmoving Industry Conference, Peoria, IL.
4. Drago, R. J., and Lutthans, R. V., 1983, " Combined Effects of Rim thickness and Pitch Diameter on Spur gear Tooth Stresses," Journal of the American Helicopter Society, Vol. 28, No.3, pp.13-19.
5. Wilcox, L., and Coleman, W., 1973, "Application of Finite Elements to the Analysis of Gear Tooth Stresses," ASME Journal of Enginering for Industry, Vol.95, No.4, pp.1139-1148.
6. Oda, S., Nagamura, K., and Aoki, K., 1981, " Stress Analysis of Thin Rim Spur Gears by Finite Element Method," Bulletin of Japanese Society of Mechanical Engineers, Vol.24, No.193, pp.1273-1280.
7. Chong, T.H., Suzuki, T., Aida, T., Kubo, A., and Fujio, H., 1983, "Tooth fillet Stresses of Gear with Thin Rim," Bulletin of Japanese Society of Mechanical Engineers, Vol.26, No.220, pp.1799-1806.
8. Chang, S.H., Houston, R.L., and Coy, J.J., 1983, "A Finite Element Stress Analysis of Spur Gears Including Fillet Radii and Rim Thickness Effects," ASME Journal of Mechanisms, Transmissions, and Automation in Design, Vol.105, No. 3, pp.327-330.
9. Reddy, S.K., Bibel, G.D., Savage, M., and Handschuh, R.F., 1994, "Effects of Rim Thickness on Spur Gear Bending Stress", Journal of Mechanical Design, Vol.116, No. , pp.1157-1162.
10. Huseyin, F.I., and Eyercioglu, O., 1995, " Evaluation of Gear tooth Stresses by Finite Element Method", ASME Journal of Engineering for Industry, Vol.117, No. , pp.232-239.
11. Vijayarangan, S., and Ganesan, N., 1993, "Stress Analysis of Composite Spur Gear Using The Finite Element Approach", Computers and Structures, Vol.46, No.5, pp.869-875.
12. Vijayarangan, S., and Ganesan, N., 1994, "Static Contact Stress Analysis of a Spur Gear Tooth Using the Finite Element Method, including Frictional Effects", Computers and Structures, Vol.51, No.6, pp.765-770.
13. Gordana, M., 2000, " 3D Approach for Spur Gear Stress Evaluation", Strojarstvo, Vol. 42, No. 5-6, pp.249-254.
14. Shuting, Li., 2002, " Deformation and Bending Stress Analysis of a Three-Dimensional, Thin-rimmed Gear", Journal of Mechanical Design, Vol.124, No. , pp.129-135.
15. Shuting, Li., 2002, " Gear Contact Model and Loaded Tooth Contact Analysis of a Three-Dimensional, Thin-Rimmed Gear", Journal of Mechanical Design, Vol.124, No. , pp.511-517.
16. Gitin M Maitra., 1994, "*Handbook of GEAR Design*", Tata McGraw Hill Publishing, NewDelhi, India.
17. Buckingham, E., 1963, *Analytical Mechanics of Gears*, Dower, New York.
18. ANSYS 10.0 Reference Manual, 2005

RAVICHANDRA PATCHIGOLLA

Mr. Ravichandra Patchigolla completed his Masters in Mechanical Engineering from University of Texas at San Antonio. He has also a Bachelors degree in Mechanical Engineering from JNT University in India. He has worked as a teaching assistant in the areas of machine element design and finite elements area. He also worked as a lab assistant for the ANSYS and SolidWorks software programs.

YESH P. SINGH

Dr. Singh currently serves as Professor of Mechanical Engineering and Biomechanics Department at the University of Texas at San Antonio. He has served as Chair of Mechanical Engineering (9/1993-12/1996), Chair of ME Graduate study Committee and ME Graduate advisor of Records (9/1998-8/2001), and Director of Engineering Machine Shop (1/1998-3/2002). His teaching and research interests are in Mechanical Design. Professor Singh is a registered professional engineer in the states of Texas and Wisconsin, and is an ASME Fellow.