DYNAMIC STRESS ANALYSIS OF SPUR-MESH GEAR TOOTH USING FEM BASED ALGORITHM

T.VADIVELU

[1](Department of Mechanical Engineering, JNTU KAKINADA, Kodad, India, vadimay28@gmail.com)

Abstract— Gears are the most common means for transmitting the power in the modern dynamic machines. Gears are used to change the speed and power ratio as well as direction between an input and output of the line shaft. The constant pressure to build less expensive, quieter running, light weight, and more powerful machinery has resulted in a steady improvement of gear designs. Despite having high cost, complicated manufacturing, need of precise alignment of shafts and lubrication, the gear drives are preferred over other power transmission drives. One of the important reasons of preference being that of efficiency which is very high in gear drives, even up to 99% in case of spur gears. Spur gears are the most simplest of the gear drives having teeth cut parallel to the axis of the shaft. In the present study a "spur gear is investigated using a two dimensional finite element model". The two dimensional model offers significant advantages for dynamic gear analysis. The gear teeth are analyzed for different operating speeds. Displacements, stress distributions and strains were investigated using finite element analysis software "ANSYS 12 Mechanical APDL". The emphasis of this study was to predict the displacements, strains and stress distributions for a spur gear tooth. The gear tooth model was generated in ANSYS 12 APDL package. Various mesh configurations were obtained in the meshing section of the package. The effect of rotational speed on dynamic stress was also studied and compared with the work of reference [4] for the same specifications. The obtained results show a good degree of agreement with the results of reference.

Keywords— Spur Gear,ANSYS 12 ,APDL, Bending Stress, Finite Element Modeling.

1. INTRODUCTION:

The minimization of dynamic loads is of paramount importance while designing gears due to the fact that dynamic loads give rise to bending stresses at the tooth root which result in tooth fracture due to fatigue. In case of spur gears the bending stresses are theoretically analyzed by Lewis equation by treating a spur gear tooth as a cantilever beam. Since spur gears have complicated geometry, a need arises for improved analysis using numerical methods which provide more accurate solutions than the theoretical methods. Finite element analysis is one such method which has been extensively used in analysis of components used in various mechanical systems. The Finite element analysis for analyzing bending stress in spur gears has been reported by some researchers in the recent past. Mrs. Shinde S.P. et al. [1] studied the static analysis of spur gear using Finite element analysis. In this paper, bending stress analysis was performed, while trying to design spur gears to resist bending failure of the teeth. The results of bending stresses as obtained from FEA Software ANSYS were compared with the results of Lewis equation and the results of both methods showed a good degree of agreement with each other. Sushil Kumar Tiwari et al. [2] analyzed the contact stress and bending stress of involute spur gear teeth by FEA and compared the same with Hertz equation, Lewis equation and AGMA/ANSI equations. T. Shoba Rani et al. [3] have performed finite element analysis on spur gear using different materials viz, nylon, cast iron and polycarbonate. They observed that in order to get good efficiency, life and less noise cast iron gears can be replaced by nylon gears because of the fact that the deflection of cast iron is more than that of nylon. Ali Raad Hassan et al. [4] studied the effects of natural frequency and rotational speed on dynamic stress in spur gear and concluded that with increase in rotational speed, the dynamic also increases. With increase in rotational speed of gear, the ratio of maximum dynamic bending root stress to maximum static bending root stress increases gradually for moving load.

In the present study, the displacements, stress distributions and strains for the dynamic response of spur gear were investigated using finite element analysis "software ANSYS 12 Mechanical APDL". The simulations were carried out for a load of 28000N, face width of 60mm, pitch diameter of 750mm and pressure angle of 20 degrees, for the same specifications as in [4].

2. SIMULATION:

The Simulation was carried out in Finite element analysis software ANSYS 12 Mechanical APDL. Finite element analysis is a computational tool for engineering analysis which makes use of mesh generation techniques for dividing a complex problem into small elements coupled with the use of a software program coded FEM algorithm.

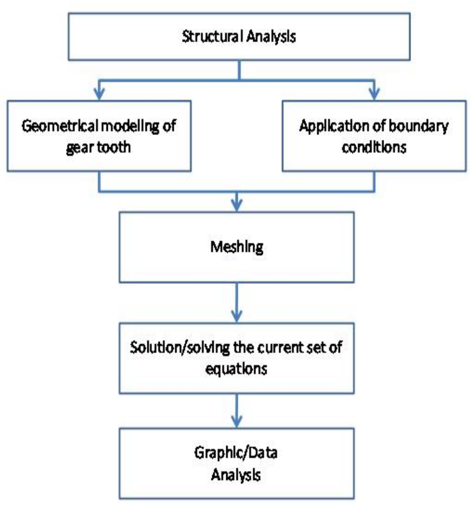

Fig 1: Simulation Flow chart

3. MATERIAL PROPERTIES:.

The material chosen for the study is Steel: Table-1 shows the properties of Steel as required for analysis.

Table-I: Properties of Steel

S.NO	PROPERTY	VALUE	UNIT
1.	Density	8.65e-8	Ns^2/mm^4
2.	Young's module	2.14e3	N/mm^2
3.	Poison's ratio	0.29	Nil

A. Model Generation:

A gear tooth model is generated in "ANSYS 12 APDL" with the following specifications

Table-II: Model Specifications

S. No.	Description	Value
1	No. of teeth	30
2	Pitch diameter	750mm
3	Module	25mm
4	Pressure angle	20degree
5	Face width	60mm
6	Addendum	1m = 25mm
7	Dedendum	1.25m = 31.25mm

The generated model comprises all the nodes, elements, material properties, real constants, boundary conditions and other features that are used to represent the physical system.

B. Meshing or Grid generation:

The gear tooth geometry created was transferred to the meshing section of the package. In the meshing section the method of meshing is selected to have various configurations. The unstructured and structured meshing with different mesh size options was created.

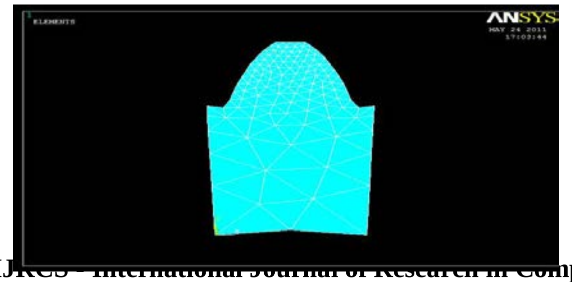

Fig 2: Unstructured or Triangular Meshing of Gear Tooth

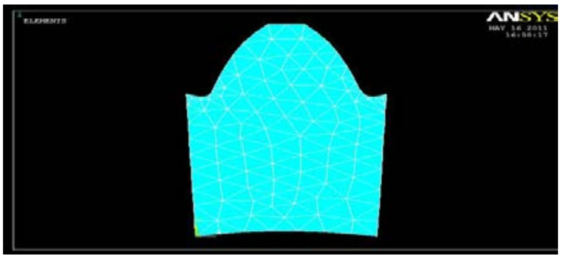

Fig 3: Gear Tooth with a Refined Unstructured Triangular Mesh

To specify the boundary conditions, all the nodes on the two radial lines defining the ends of the segment and the bottom rim were selected and given zero displacement in both directions.

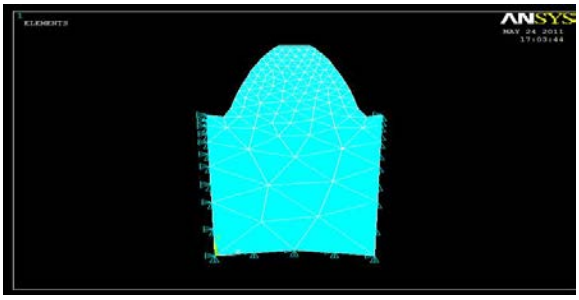

Fig 4: Unconstrained Tooth Surface and Inside Rim Surface

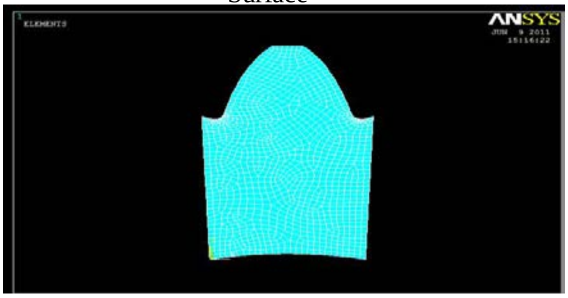

Fig 5: Refined Structured or Quadrilateral Mesh

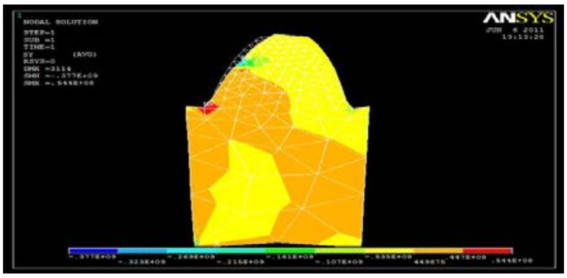

Fig 6: Stresses in Y- Direction

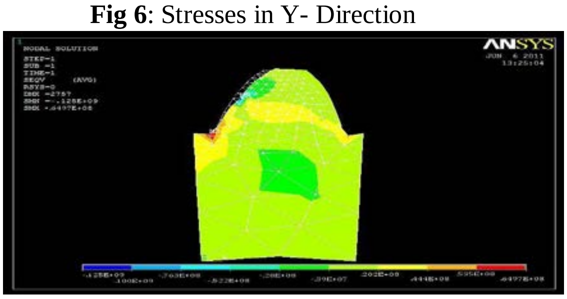

Fig 7: Stresses in Y- Direction

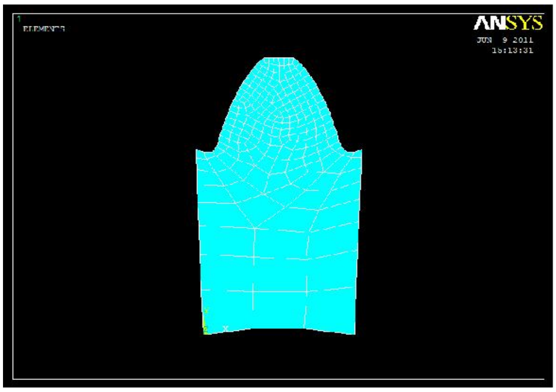

Fig 6: Quadrilateral or Structured Meshing of a Gear Tooth

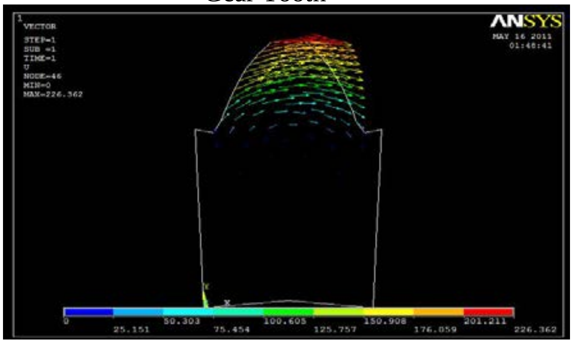

Fig 8: Displacement Vector Plot

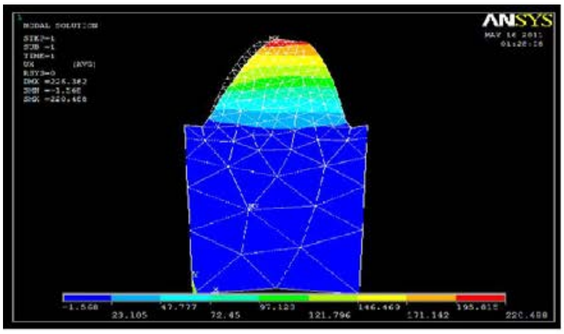

Fig 10: Displacement of the Model in X- Direction

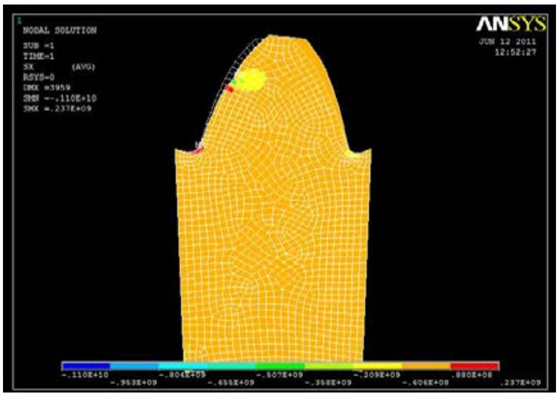

Fig 13: X-Component of Stress of the Model

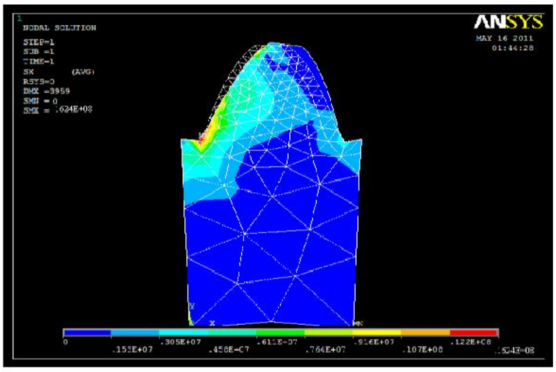

Fig 7: Stresses in X- Direction

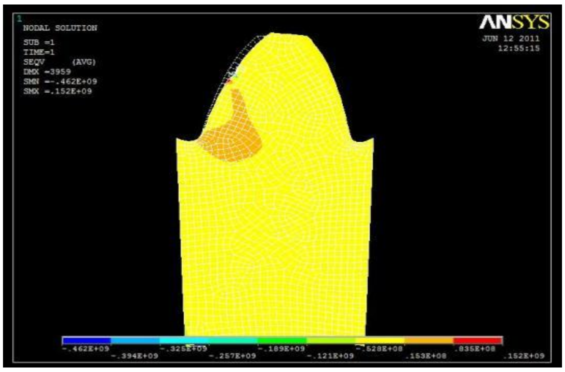

Fig 9: Von Mises Stress of the Model

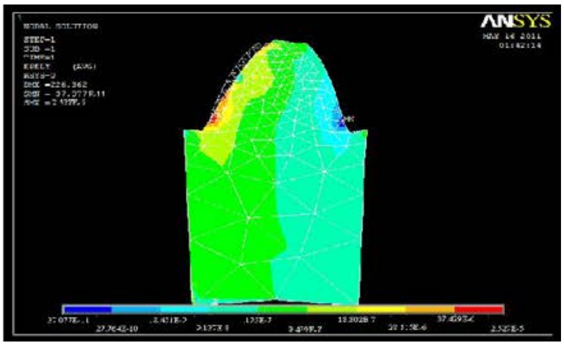

Fig 11: Y-Component of Elastic Strain of the Model

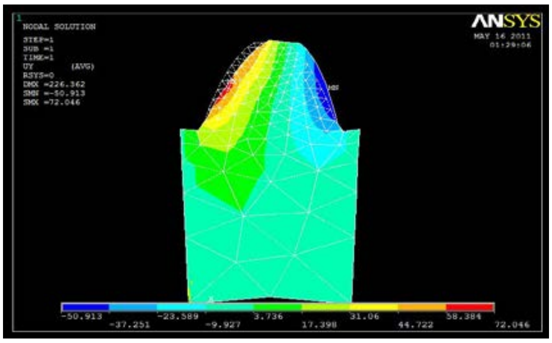

Fig 12: Displacement of the Model in Y-Direction

Figure 10 shows the displacement vector plot. **Figure 11** and **Figure 12** show the displacement of the model in X and Y-directions respectively. It is evident from the figures that the maximum displacement of the model is at the top of the gear tooth. This result holds good as the maximum displacement occurs at the free end of the cantilever beam. At the fixed end there is no displacement. Figure 13 shows the elastic strain of the model in X-direction and Figure 14 shows the elastic strain in the Y-direction. The range of the strains in X and Y directions are 68.918E-12 to 7.553E-5 and -37.007E-11 to 2.527E-5, respectively.

Figure 13 shows the development of high stresses when rotational speed is increased to 2500 rpm. The range of the stresses is from -0.11E+10 Pa to .237E+09 Pa. Figure

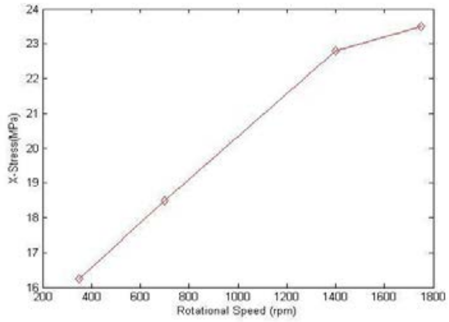

Fig 14: Curve Relation of the Speed and Maximum Stresses in X-Direction

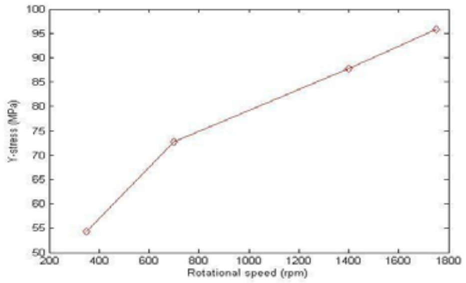

Fig 15: Curve Relation of the Speed and Maximum

Stresses in Y-Direction

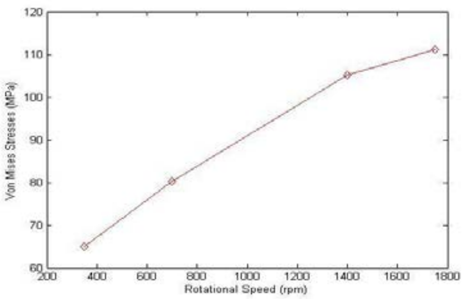

Fig 16: Curve Relation of the Speed and Von

Shows high Von Misses Stresses at 2500 rpm. The range of the stresses is from -.462E+9 to .152E+9.

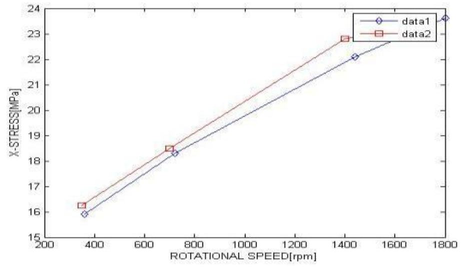

Fig 17: Comparison of Stresses in X-Direction with the Work of Reference

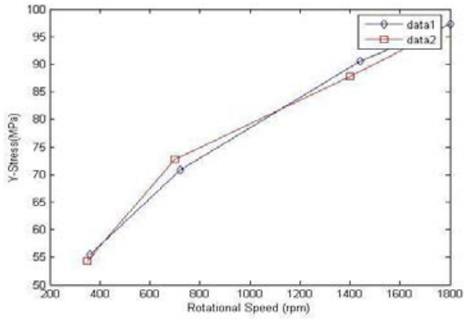

Fig 18: Comparison of Stresses in Y-Direction with the

Work of Reference

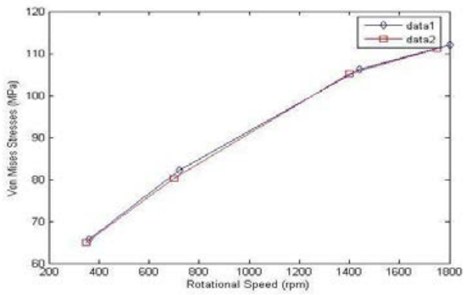

Fig 19: Comparison of Von Mises Stresses with the Work of Reference

Table III: Maximum Stresses for different Speeds of the Model

Speed (In rpm)	Mpa$_x$ signa$_x$	Mpa$_y$ sigma$_y$	**Von Mises Stresses (MPa)**
360	15.94	64.45	74.87

Mises Stresses.

Table 3 gives the various values of stresses in X and Y directions, at various speeds. These stresses are obtained

800	17.6	74.56	82.25
1450	21.7	79.44	106.14
1850	22.4	97.47	112.22

by using “ANSYS Finite Element Software”

It is observed from Table III that the maximum magnitudes of σ_x and σ_y at the tooth root for various speeds of the selected gear model, in the range of 350 rpm to 1750 rpm, are increasing gradually from 16.24 MPa to 23.5 MPa, and from 54.4 MPa to 95.87 MPa, respectively. The variation is shown in Figure 17 and Figure 18. Therefore, the stresses at the root of the high modules spur gear tooth are increasing with increase in the rotational speed of the gear

Figures 20, 21 and 22 show the stresses in X- direction, stresses in Y-direction and Von Mises Stresses of the model respectively. Data 1 is the work done by the author of reference [4] while data 2 are the results obtained in the present investigation. The obtained results show a good level of agreement with that of the reference [4].

4. CONCLUSION:

The displacement, strain and stress distribution were predicted along the gear tooth. The results obtained for stress distribution show that the maximum stress occurs at the root of the gear tooth. It is also found that with increase in rotational speed the bending stress increases. The obtained results of rotational speed compared with the results of reference [4] show a good level of agreement.

REFERENCES

[1] S. Shinde, A. Nikam and T. Mulla, “Static analysis of spur gear using finite element analysis,” Journal of Me-chanical and Civil Engineering, 2278-1684, pp. 26-31.

[2] S. Tiwari and U. Joshi, “Stress analysis of mating in-volute spur gear teeth,” International Journal of Engi-neering Research and Technology, vol. 1, no. 9, 2012.

[3] V. Bhandari, “Design of machine elements,” New Delhi: Tata McGraw-Hill, 2010.

[4] T. Rani and T. Khalandar, “Spur Gear,” International Journal of computational Engineering Research, vol. 3 no. 11, pp. 7-12, 2013.

[5] A. Hassan, G. Thanigaiyarasu, and V. Ramamurti, “Ef-fect of natural frequency and rotational speed on dy-namic stress in spur gear,” International Scholarly and Scientific Research & Innovation, vol. 2, no. 12, 2008

[6] L. Wilcox and W. Coleman, “Application of Finite Elements to the Analysis of Gear Tooth Stresses,” [7] Journal of Engineering for Industry, vol. 95, no. 4, [8] p. 1139, 1973

[7] V. Ramamurti, N. Vijayendra and C. Sujatha, “Static and dynamic analysis of spur and bevel gears using FEM,” Mechanism and Machine Theory, vol. 33, no. 8, pp. 1177-1193, 1998

[8] Z. Chen and Y. Shao, “Dynamic simulation of spur gear with tooth root crack propagating along tooth width and crack depth,” Engineering Failure Analysis, vol. 18, no. 8, pp. 2149-2164, 2011.

[9] V. Ramamurti, N. Vijayendra and C. Sujatha, “Static and dynamic analysis of spur and bevel gears using FEM,” Mechanism and Machine Theory, vol. 33, no. 8, pp. 1177-1193, 1998.