Comparison of Bending Stresses for Different Face Width of Helical Gear Obtained Using Matlab Simulink with AGMA and ANSYS

Pushpendra Kumar Mishra[1], Dr. M.S. Murthy[2]

[1]PG Scholar, Mechanical Engineering, Indore Institute of Science and technology, Indore, India

[2]Professor, Mechanical Engineering, Indore Institute of Science and technology, Indore, India

Abstract— **Helical gears are widely used in industry where the power transmission is required at heavy loads with smoother and noiseless operation. The study in this paper shows that the complex design problem of helical gear requires superior software skills for modelling and analysis. The above problem can be resolved using MATLAB Simulink environment which provides equivalent results to the AGMA and also with ANSYS. In this paper we modelled a helical gear on Pro engineer wildfire 5.0 and stress analysis part is done on ANSYS 12.0. A Simulink model is generated by using curve fitting. The results are compared with both AGMA and FEM procedures.**

Keywords— **Helical Gear, Modelling, AGMA, Curve Fitting, Simulink.**

I. INTRODUCTION

Designing lighter and smaller automotive power transmission systems which can be used for heavy loads with noise free operation necessitate the intense studies on helical gears. Helical gears have shown finer noise and smoother action reduction as compared to the spur gear. In addition, the load transmitted may be slightly higher and also the life of gear greater for the same loading, than with an equivalent pair of spur gear.

The best way of transmitting power between the shafts is gears. Gears are mostly used to transmit torque and angular velocity. The design of gear is a complex process. Generally it needs large number of iterations and data sets. In many cases gear design is traditional and specified by different types of standards [1], [2]. Involute shaped gears found to be almost everywhere because of the contact forces act along a straight line. Many researchers have studied the behaviour of helical gears under different conditions.

Yi-Cheng Chen et al. [3] in their study stress analysis of a helical gear set with localized bearing contact have investigated the contact and the bending stresses of helical gear set with localized bearing contact by using finite element analysis. S .Vijayaragan and N. Ganesan [4] carried out a static analysis of composite helical gears using three dimensional finite element methods to study the displacements and stresses at various points on a helical gear tooth. For determining the stresses at any stage during the design of gears helix angle and face width are important

features. Rao and Muthuveerappan [5] have explained the geometry of helical gears by simple mathematical equations. A parametric study was made by varying the face width and the helix angle to study their effect on the root stresses of helical gears. Zhang et. al [6] in their research on Analysis with varying mesh Stiffness, have proposed a new approach to analysis of the loading and stressing distribution for spur and helical gears. J. Lu et al. [7] introduced a new approach which was based on the application of computerized simulation of meshing and contact loaded gear drives by using finite element method.

S. T.Yusuf et.al [8] explained a simplified Matlab Simulink model of DC rotating machine at the place of complex mathematical model and discussed the simulation results. A. A. Ansari and D M Deshpande [9] have studied the steady state performance of induction motor and created a model in Matlab Simulink.

The Simulink model in this work is generated for two main reasons. First of all, the MATLAB Simulink models facilitate Ansys users to theoretically predict the response of the analysis Software. Instead of going through number of iterations and various typical design processes of the gear analysis the Simulink program allows the user to predict the response of the system without actually doing the Analysis on actual gear itself.

II. MODELLING OF GEAR

In total 15 numbers of gears are modelled in Pro/ENGINEER Wildfire [10], which are having the following parameters. The material for the gear is taken as structural steel.

TABLE I

Description	Specifications
No. of teeth(Z)	20
Module(m)	10 mm
Pitch Circle Diameter(D)	200 mm
Gear face width (b)	50.0622 mm
Pressure angle(α)	20
Helix angle (β)	15^0
Addendum (ha)	10 mm
Dedendum (hf)	1.157*m

Modulus of Elasticity	2E+05 MPa
Poisson's Ratio	0.3

The procedure to model the gear of 20 number of teeth with the combination of the all above mentioned parameters in the Pro/ENGINEER Wildfire, other set of gears are modelled in the similar way. Part parameters are the basic parameters defining the gear. These part parameters determine all the other parameters that define the gear tooth profile using the Tools/Relation menu. Figure 1 showing the part parameters.
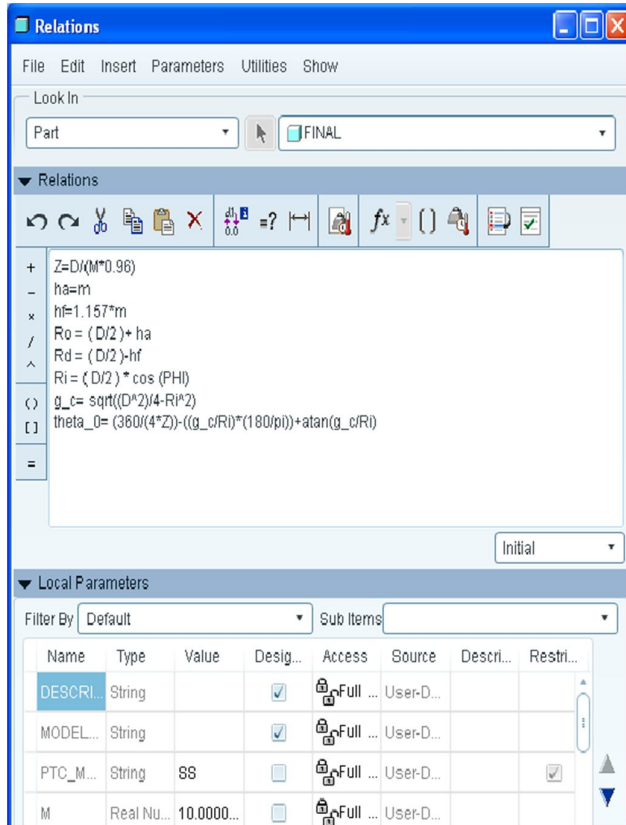

Fig. 1 Tools / Relation Menu

III. STRESS CALCULATION

All the calculations are carried out on the basis of eq. (1) recommended by AGMA [11].

A. *AGMA Stress calculation*

Bending stress (σ) $= \dfrac{F_t \times P_d \times K_s \times K_m \times K_a}{b \times y_j \times K_v}$ (1)

Where,

Diametral Pitch (P_d) $= \dfrac{Z}{D \times \cos \beta}$

$= \dfrac{20}{200 \times \cos(15°)}$

Application Factor (K_a) $=$ 1.2
Size Factor (K_s) $=$ 1

Load distribution Factor (K_m) $=$ 1.25
Dynamic Factor (K_v) $=$ 0.8
Normal tangential Load (F_t) $=$ 1424.75 N
Geometry Factor (y_j) $=$ 0.4

Tangential force calculation:
Power (P) = 15 HP=11.190 kW
Speed = 750 R.P.M

Torque (T_p) $= \dfrac{9549.3 \times P(Kw)}{speed(RPM)} = 142.47$ N-m

Tangential Force (F_t) $= \dfrac{2000 \times Tp}{Pitch\ diameter} = 1424.75$ N

Bending Stress Calculation (σ AGMA) :
for face width (b) = 50.0622 mm

σ AGMA $= \dfrac{1424.75 \times 1 \times 1.2 \times 1.25 \times P_d}{50.0622 \times 0.4 \times 0.8} = 13.81$ MPa

B. *Static structural Analysis*

The structural analysis of the helical gear tooth model is carried out using the finite element analysis in Ansys 12.0. The load applied at the tooth of the helical gear. The Mess is generated with tetrahedron nodes. Figure 2, is shows the generated mesh. Maximum element size of 3.5 mm is selected for the Mesh Control. By applying the analysis over the tooth which is facing the load we get the stress distribution in the numeric as well as in the form of colour scheme. By varying the face Width and keeping the other parameters constant various models of the helical gear are created. Various models created by varying face width are shown in Figure 3 and Figure 4 represents the finite element analysis.
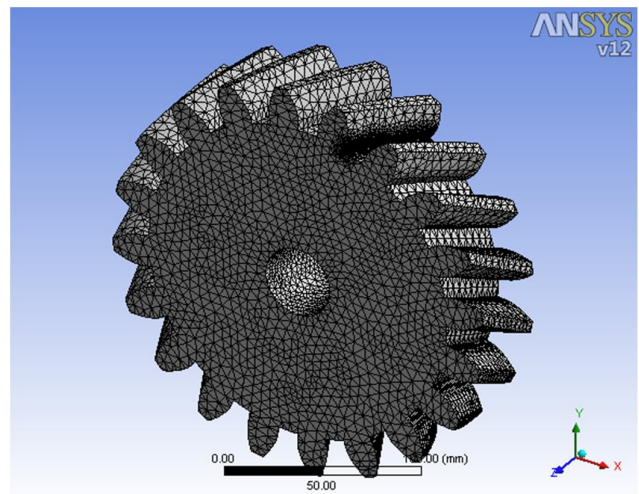

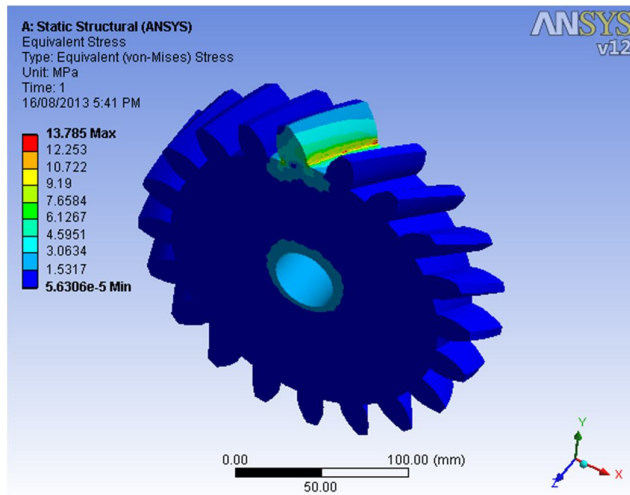Fig. 2 Meshed 3-D Model of Helical gear

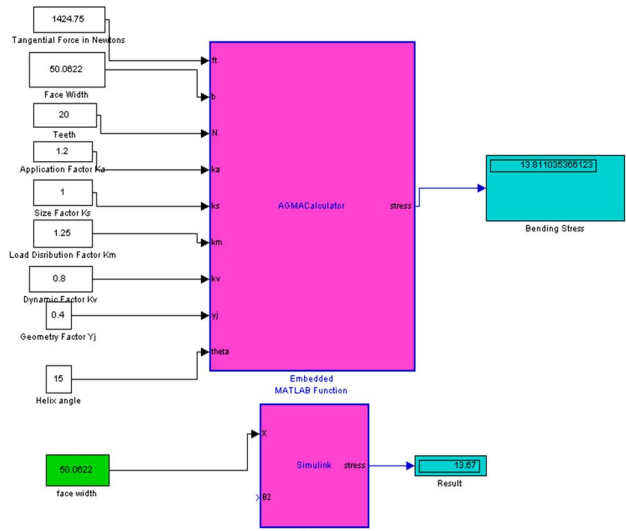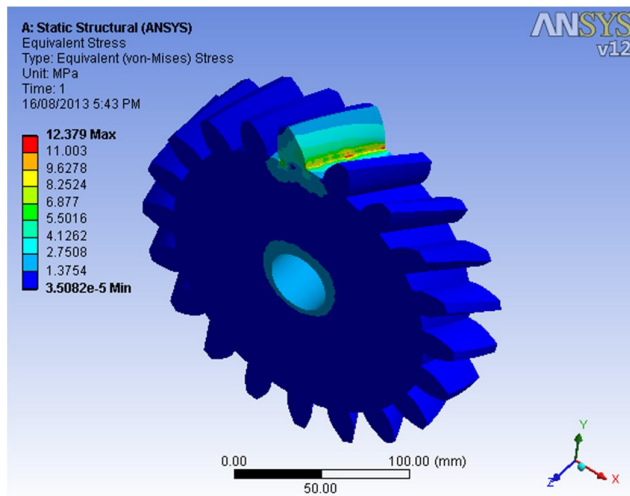Fig. 3 Static analysis of gear having 20 teeth (face width =50.0622 mm)

Fig. 4 Static analysis of gear having 20 teeth (face width=54.0591)

C. Simulink Stress calculation:

For building the Simulink model an equation is formed by curve fitting using the values obtained from Ansys 12.0 FEM package. The Simulink model is shown in the fig. 5. By providing necessary input parameters to this model we can calculate the desired output easily.

Fig.5 Simulink Model

IV. RESULT & DISCUSSION

For determining the stresses at any stage during the design of gears face width is an important parameter. To determine the stress variation with the face width, various models of helical gear are made by keeping other parameters i.e. Number of teeth, helix angle etc constant. Table II shows the results of bending stress with the variation in the face width of the helical gear tooth.

TABLE III
Bending Stress (MPa)

S. No	Face Width (mm)	AGMA (MPa)	Ansys (MPa)	Simulink (MPa)
1	40.0062	17.28	17.46	17.31
2	42.0055	16.46	15.64	16.34
3	44.0251	15.7	15.49	15.51
4	46.1012	14.99	14.81	14.78
5	48.0788	14.38	14.31	14.19
6	50.0622	13.81	13.71	13.67
7	52.0437	13.28	13.27	13.2
8	54.0591	12.79	12.37	12.76
9	56.0619	12.33	12.47	12.33
10	58.0212	11.91	11.76	11.89
11	60.0551	11.51	11.42	11.41

Table II clearly shows the results of variation in face width from 40.0062 mm to 60.0551 mm, there is continuous decrement in the value of stress at the tooth of the helical gear. For constant load and speed the greater face width is suitable.

The result table shows that the results of AGMA and Simulink are much closer than results of ANSYS.
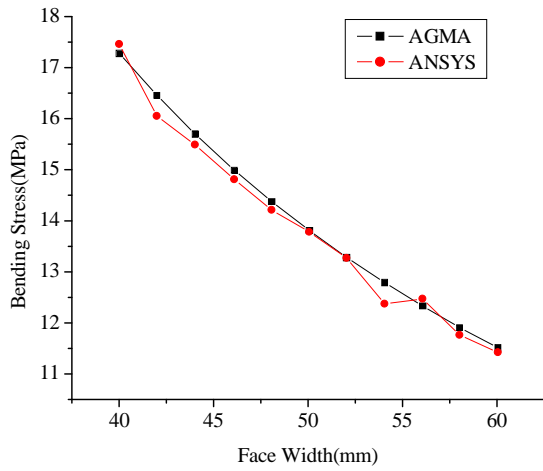

Fig. 6 Bending Stress (AGMA-Ansys)

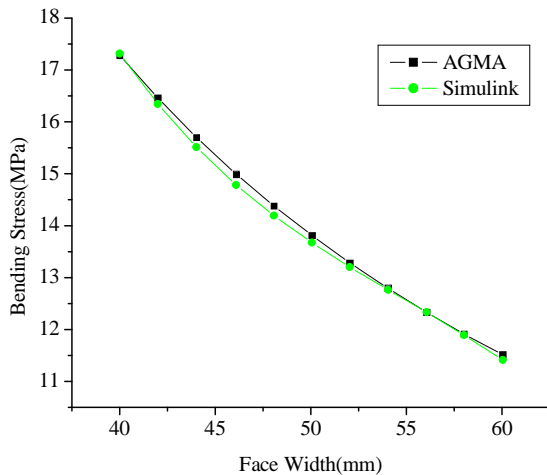Figure 6 shows the comparison of results obtained through AGMA and Ansys.

Fig. 7 Bending Stress (AGMA-Simulink)

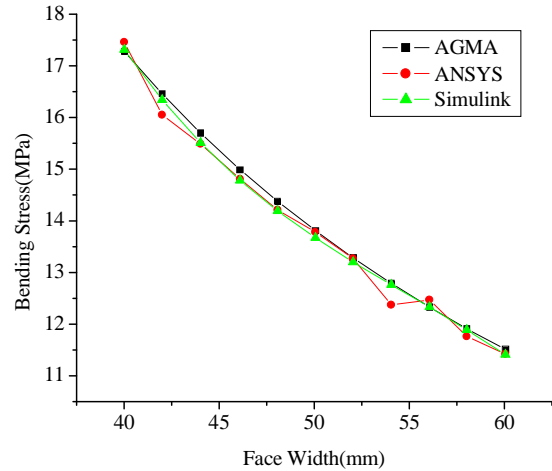Figure 7 shows the comparison of results obtained through AGMA and Simulink.

Fig.8 Bending stress (AGMA – Ansys-Simulink)

Fig.8 shows the comparison between the bending stress variation between AGMA, Ansys and Simulink.

V. CONCLUSIONS

The results obtained from both ANSYS and Simulink are closed to the results obtained from AGMA procedure. From the results it is justified that Simulink can also be used for predicting the values of bending stress at any required face width which is much easier to use to solve complex design problems.

REFERENCES

[1] A. Fernandez del Rincon, et.al," *A model for the study of meshing stiffness in spur gear transmissions*", *Mechanism and Machine Theory 61* (2013) 30–58.
[2] Shanmugasundaram Sankar & Muthusamy Nataraj, "*Optimization of asymmetric spur gear drives to improve the bending load capacity*", *Int J Adv Manuf Technol* (2011) 55:1–10.
[3] S Cheng, Y., and Tsay C.B., "*Stress analysis of Helical Gear set with Localized Bearing Contact*, *Finite Element in Analysis and Design*", *38, pp.707-723, 2002.*
[4] Vijayarangan, S., and Ganesan N., "*A Static Analysis of Composite Helical GearsUsing Three-dimensional Finite Element Method*", *Computers & Structures, 49, pp.253- 268,1993.*
[5] Rao, C.M., and Muthuveerappan G., Rao, C.M., and Muthuveerappan ,"Finite Element Modeling and Stress Analysis of Helical Gear Teeth", *Computers & structures, 49, pp.1095-1106, 1993.*
[6] Zhang, J.J., Esat, I.I., Shi, and Y.H., "*Load Analysis with Varying Mesh Stiffness*", *Computers and Structures, 70, pp.273-280, 1999.*
[7] Lu, J., Litivin, F., and Chen, J.S.," Load Share and Finite Element Stress Analysis for Double Circular-Arc Helical Gears", *Mathl.Comput.Modeling, 21,pp.13-30.1995.*
[8] T.Yusuf et.al, *"Simplified model of a DC rotating machine for mechanical energy generation using MATLAB/SIMULINK, International Journal of Engineering and Innovative Technology"* Vol. 2, Issue 6, Dec 2012**.**
[9] A A Ansari, D M Deshpande, "*Mathematical Model of Asynchronous Machine in MATLAB Simulink*, *International Journal of Engineering Science and Technology* ", Vol. 2(5), 2010, 1260-1267.
[10] Pro/ENGINEER Wildfire 5.0
[11] Dudley' Gear Handbook, second edition, pp. 6.15-6.22.