Finite Element Method for the Nonlinear Contact Analysis of Helical Gears

Jerin Sabu[1], Dr.Y.V.K.S.Rao[2], Alen John[3], Rajeev V.R[4]

[1] *PG Scholar, Dept.of Mechanical Engineering,Mar Baselios College of Engg &Technology, Trivandrum, India.*
[2] *Professor, Dept. of Mechanical Engineering, Mar Baselios College of Engg &Technology, Trivandrum, India.*
[3,4] *PG Scholars, Dept. of Mechanical Engineering,Mar Baselios College of Engg &Technology, Trivandrum, India.*
Email: jerinsabu11@gmail.com

The increasing demand for quiet power transmission in machines, vehicles, elevators has created a growing demand for a more precise analysis of the characteristics of gear systems. Helical gears are currently used increasingly as a power transmitting gear due to their relatively smooth and silent operation, large load carrying capacity and their operation at higher speeds. This paper proposes the development of the finite element model for the helical gear pair for monitoring the deformation and stress state of teeth flanks, teeth fillets and parts of helical gears during the tooth pair meshing period. For the investigation of contact problems with finite element method, the stiffness relationship between the two contact areas is usually established through a spring placed between the two contacting areas. This can be achieved by inserting a contact element placed in between the two areas where contact occurs. Initially helical gear pairs are modelled in SolidWorks and then import the IGS file to the ANSYS Workbench 11 environment. Thereafter in ANSYS Workbench, nonlinear contact analysis is done. The stresses generated on gear teeth flanks, teeth fillets and parts of helical gears during the tooth pair meshing period are obtained.

Index Terms- Helical gears; Teeth flank; Teeth fillet; Stress state.

1. INTRODUCTION

Gears are one of the most critical components in a mechanical power transmission system, and in most industrial rotating machinery. Gearing is one of the most effective methods of transmitting power and rotary motion from the source to its application with or without change of speed or direction. The increasing demand for quiet power transmission in machines, vehicles, elevators has created a growing demand for a more precise analysis of the characteristics of gear systems. Helical gears are currently being used increasingly as a power transmitting gear owing to their relatively smooth and silent operation, large load carrying capacity and higher operating speed. This paper describes the development of the finite element model for the helical gear pairs for simultaneously monitoring the deformation and stress state of teeth flanks, teeth fillets and parts of helical gears during the tooth pair meshing period [1]. To enable the investigation of contact problems with FEM, the stiffness relationship between the two contact areas is usually established through a spring placed between the two contacting areas. This can be achieved by inserting a contact element placed in between the two areas where contact occurs. Proper selection of the contact element between the contact portions of the meshed helical gears is important especially in the case of non-linear

analysis. Due to the progress of computer technology many researchers tended to use numerical methods to develop theoretical models to calculate the effect of whatever is studied. Numerical methods are capable of providing more truthful solution since they require very less restrictive assumptions. A proper computation of forces and deformations in the contact zone is crucial for determination of stress throughout the model, i.e. in teeth roots and bodies. Experimental determination of stress and strain state in helical gear teeth with real contact conditions is associated with many limitations. The Finite Element Method (FEM) is the most appropriate numerical method that can solve this nonlinear problem. The main characteristics of the gear pair are number of teeth on pinion is 16 and on the wheel is 24 facewidth is 25 mm a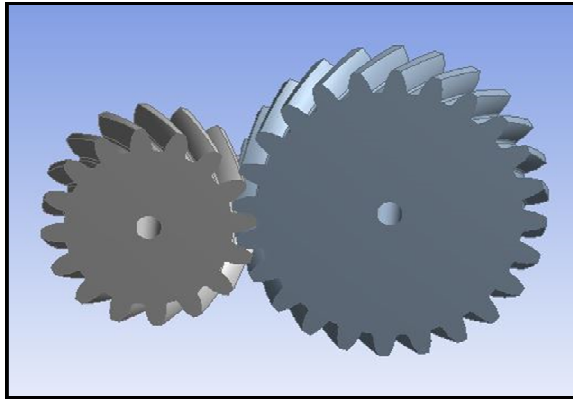nd module is 4.Pressure angle is 20° and Helix angle is 30°. The stresses generated on gear teeth flanks, teeth fillets and parts of helical gears during the tooth pair meshing period have to be analyzed.

2. MODELLING OF THE HELICAL GEARS

Modelling of the helical gear pair is done in SolidWorks and it can be directly generated by giving the dimensions of gear. From the SolidWorks design library, tool box is given and in the tool box consists of different standards of various mechanical parts such as bearings, bolts and screws, keys, nuts, pins, retaining rings and also the power transmission type is given. From the power transmission type chains and gears are given. Different types of gears are available and from that helical gear is selected. By entering the dimensions of helical gear such as number of teeth on pinion and wheel, facewidth, module, pressure angle,

helix angle etc…..we can generate the helical gear. The main characteristics of the gear pair are number of teeth on pinion is 16 and on the wheel is 24 facewidth is 25 mm and module is 4.Pressure angle is 20º and Helix angle is 30º.

Figure 2.1 Helical gear assembly

3. THE FEM NUMERICAL PROCEDURE

In this finite element analysis the continuum is divided into a finite numbers of elements, having finite dimensions and reducing the continuum having infinite degrees of freedom of unknowns. It is assumed that the elements are connected only at the nodal points. The accuracy of solution increases with the number of elements taken. Proper selection of the contact element between the contact portions of the meshed helical gears is important especially in the case of non-linear analysis. Finite element analysis (FEA) is done in ANSYS Workbench 11.Initially the modelled helical gear assembly is imported to ANSYS Workbench by means of an IGS file. Here the static structural analysis is done.

Material properties of the wheel and pinion are given as Young's modulus = 206GPa, Poisson's Ratio = 0.3. Main factor in the contact analysis is the definition of connection between the wheel and pinion. Here in the case of nonlinear contact type, contact element used is 'Rough'. For the teeth contact 'pure penalty' method is chosen. Penalty method uses a contact "spring" to establish a relationship between two contact surfaces. Another important one in the connection type is the 'Interface Treatment'. Interface Treatment indicates how the contact interface for the pair is treated. For nonlinear contact types (frictionless, rough, and frictional), Interface Treatment is displayed. In this paper 'Adjust to Touch' interface treatment is used. So any initial gaps are closed and any initial penetration is ignored creating an initial stress free state. Here the proper choice of the contact stiffness is critical. Consider softening the normal contact stiffness KN to a value of .1. The default value is 1 and may be changed by setting the Normal Stiffness.

Smaller KN multipliers will allow more contact penetration which may cause inaccuracies but may allow problems to converge that would not otherwise In this paper normal stiffness factor is taken as 0.1.
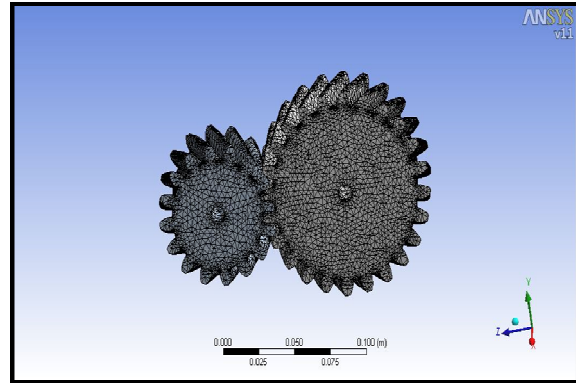

Figure 3.1 Meshing of the helical gear pair

Then fine meshing is done for the better convergence of the solution. Finer meshing generally leads to a more accurate solution, but requires more time and system resources. During the meshing process total number of elements used is 78971 and number of nodes used is 138275.Types of elements used are Solid187,Contact174,Target170,Surface154.Boundary conditions are defined and external loads are to be applied. Two cases for helical pair is done. First case in which gear with fixed support and pinion with frictionless support and in the second case gear with frictionless support and pinion with fixed support is given. Load is applied as a pressure on the two faces of gear contact at a particular instant and its value for the helical gear is 111MPa.
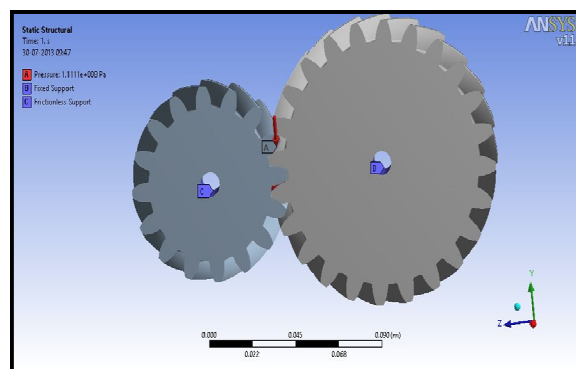

Figure 3.2 Application of boundary conditions and external load for the first case

Here the gear can be either fixed or pinion can be fixed. Both cases are studied and the results are obtained.
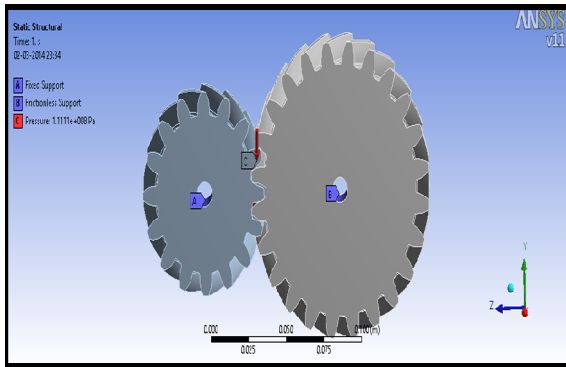
Figure 3.3 Application of boundary conditions and external load for the second case.
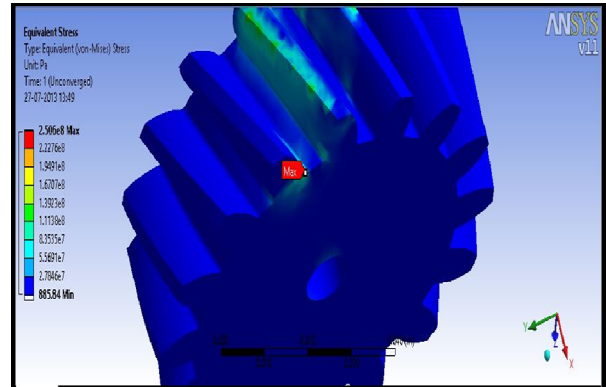
4. RESULTS AND DISCUSSION

The objective of the contact stress analyses was to gain an understanding of the modeling and solution difficulties in contact problems and to examine the contact stresses in the helical gear. Here in the first case the gear is with fixed support and pinion with frictionless support is given and the number of teeth on the pinion is 16 and the number of teeth on the gear is 24. The maximum stress obtained in the teeth fillet is 42 MPa.
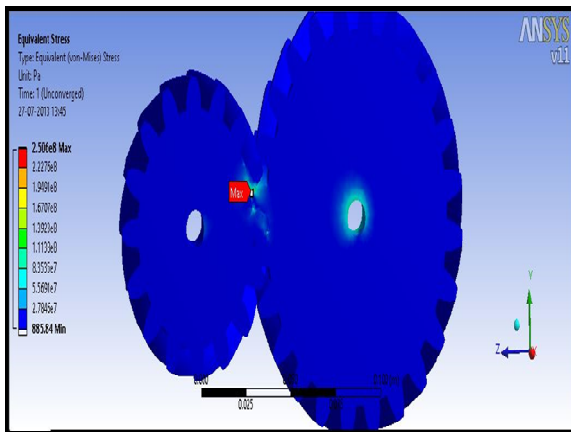

Figure 4.1 Equivalent stress for helical gear pair

For knowing the maximum stress produced portion in helical gear pair, the enlarged view of the pinion stress is shown below which will show the maximum stress produced in the teeth contact zone. Also the deformation on the teeth's are obtained and the maximum value is given by 0.02374mm.

Figure 4.2 Enlarged view of pinion stress for the helical gear pair

Figure 4.3 Total deformation of the helical gear pair

For the second case the same helical gear pair is used with the number of teeth on the pinion as 16 and the number of teeth on the gear as 24 but the boundary conditions are reversed i.e. gear with the frictionless support and pinion with fixed support is given The maximum stress obtained in the teeth fillet is 70 MPa. For the two cases explained stress generated is more in the case where pinion is given the fixed support. This is due to the fact that the pinion is the driving member and also pinion is small in size compared to that of gear and when pinion is fixed, stress generated will be always more compared to that in which the gear is fixed. For knowing the maximum stress produced portion in helical gear pair, the enlarged view of the pinion stress is shown below which will show the maximum stress produced in the teeth contact zone.
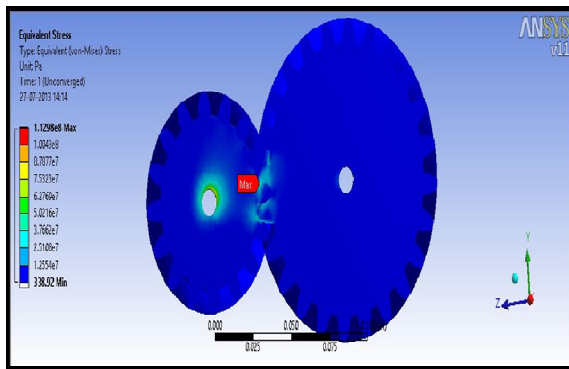
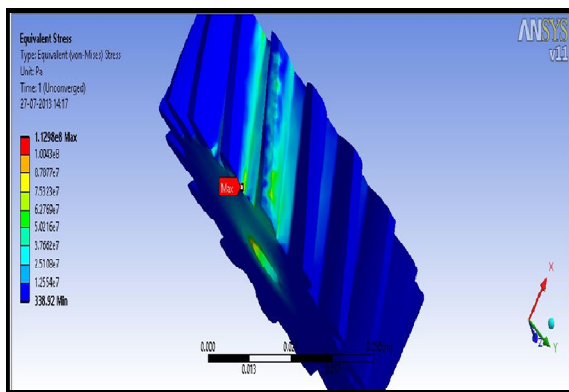Figure 4.4 Equivalent stress for helical gear pair (Second case)

Figure 4.5 Enlarged view of pinion stress for the helical gear pair (Second case)

5. CONCLUSIONS

Analysis of stresses on gears is very much necessary to minimize or to reduce the failures and for optimal design of gears. Analytical methods of gear analysis uses a number of assumptions and simplifications and it is intended to determine the maximum stress values. 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. Proper contact element in teeth contact region is very much necessary for the simulation of nonlinear contact problem .The 3-D diagrams obtained during the work showing the maximum stresses on the gear tooth is very helpful for the detailed analyses. Here maximum von-mises stress state of teeth flanks, teeth fillets and parts of helical gears during the teeth pair meshing period and deformation is obtained The total deformation of gear teeth was also found to be very less and which is acceptable. The future work includes the optimization of the gear tooth profile in which various parameters like helix angle, face width, number of teeth are to be changed so as to attain reduced stresses on the gear tooth during the power transmission time.

REFERENCES

[1] Ivana Atanasovska, Vera Nikolic.(2009):Finite element mode for stress analysis and nonlinear contact analysis of helical gears, Scientific Technical Review,Vol.LVIX,No.1.

[2] Jovanovic.M..(2004):3D spur gear fem model for the numerical calculation of face load factor, mechanical Engineering, Vol.2

[3] Pasta.A,Virzi'Mariotti.G.(2012): Dynamic model for the stres and strain state analysis of a spur gear transmission, Journal of mechanical engineering 58, 56-67

[4] Haval Kamal Asker.(2012):Three dimensional dynamic stress analyses for a gear teeth using finite element method, ARPN Journal of Engineering and Applied Sciences, vol.7, no.7.

[5] JongBoon O, Xin Wang.(2012): Modal and stress analysis of gear train design in portal axle using finite element modeling and simulation, Journal of Mechanical Science and Technology 26 (2), 575~589