FEA Analysis of An Electric Wiper Mechanism

Amit A. Nimbalkar¹ Dr. B.E.Narkhede²
¹M-Tech student ²Professor and Head
¹²Department of Production Engg. VJTI, Mumbai, India.

Abstract: In this study, a procedure for FEA analysis of an electric wiper mechanism is developed using flexible multi-body dynamic simulation and dynamic stress analysis. Both rigid and flexible body dynamic models are simulated by using ANSYS software. This was done in two parts - first rigid dynamics analysis was performed and secondly transient structural analysis was performed.

Key words: FEA, wiper mechanism, ANSYS.

I. INTRODUCTION

A windscreen wiper or windshield wiper is a device used to remove rain and debris from a windscreen or windshield. Almost all motor vehicles, including trains, aircraft and watercraft, are equipped with such wipers, which are usually a legal requirement. A wiper generally consists of an arm, pivoting at one end and with a long rubber blade attached to the other. The blade is swung back and forth over the glass, pushing water from its surface. The speed is normally adjustable, with several continuous speeds and often one or more "intermittent" settings. Most automobiles use two synchronized radial type arms, while many commercial vehicles use one or more pantograph arms. Wipers may be powered by a variety of means, although most in use today are powered by an electric motor through a series of mechanical components, typically two 4-bar linkages in series or parallel. Also, today it is a common practice to fit two wiper blades for the front windscreen and both blades driven from a single motor. As per the law the wiper on the driver’s side must operate effectively and efficiently. Hatchback cars often use a wiper for the rear window. Considerable driving force is required for a rubber wiper blade to move across a glass surface, especially when the blade has to sweep away a large volume of water or snow. The windscreens of modern vehicles have a double curvature, which requires long articulated wiper blades with the ability to flex to the contour of the glass. Wiper systems generally use two wipe speeds to suit the driving conditions. A car wiper motor on a modern vehicle should be a high powered quiet unit operating on a current of 2 – 4 A. In the past, shunt-wound motors were used but now-a-days the permanent-magnet motor is commonly used. The layout of a typical wiper system is shown in Fig. 1.

II. FINITE ELEMENT ANALYSIS

The finite element analysis is a numerical technique. In this method all the complexities of the problems, like varying shape, boundary conditions and loads are maintained as they are but the solutions obtained are approximate. Because of its diversity and flexibility as an analysis tool, it is receiving much attention in engineering.

The finite element method (FEM) (its practical application often known as finite element analysis (FEA)) is a numerical technique for finding approximate solutions of partial differential equations (PDE) as well as of integral equations. Finite Element Analysis is a simulation technique which evaluates the behaviour of components, equipment and structures for various loading conditions including applied forces, pressures and temperatures. Thus, a complex engineering problem with non-standard shape and geometry can be solved using finite element analysis where a closed form solution is not available. The finite element analysis methods result in the stress distribution, displacements and reaction loads at supports etc. for the model.

Fig.1: General Layout of Wiper System

The finite element method (FEM) (its practical application often known as finite element analysis (FEA)) is a numerical technique for finding approximate solutions of partial differential equations (PDE) as well as of integral equations. Finite Element Analysis is a simulation technique which evaluates the behaviour of components, equipment and structures for various loading conditions including applied forces, pressures and temperatures. Thus, a complex engineering problem with non-standard shape and geometry can be solved using finite element analysis where a closed form solution is not available. The finite element analysis methods result in the stress distribution, displacements and reaction loads at supports etc. for the model.

The finite element analysis originated as a method of stress analysis in the design of aircrafts. It started as an extension of matrix method of structural analysis. Today this method is used not only for the analysis in solid mechanics, but even in the analysis of fluid flow, heat transfer, electric and magnetic fields and many others. This method is used extensively for the analysis and design of ships, aircrafts, space crafts, motors and heat engines.

Design engineers must use their experience and the latest design tools to maintain reasonable safety levels while providing the most cost effective design. One tool being used on an ever increasing basis is Finite Element (FE) analysis software. The current capabilities of FE software on
desktop computers provide supports design engineers with the ability to employ FE analysis on a nearly routine basis.

III. LITERATURE REVIEW
Zexun Wu et. al. [1] studied durability analysis of an electric wiper linkage mechanism subjected to operation cycles. Authors in their study proposed that the structure fatigue life could be evaluated at the design stage and the structural revision could also be carried out to satisfy the target, which could accelerate product development cycle, reduce product cost, and raise product competitiveness.

K.Kalyan Kumar et. al. [2] studied crack propagation and stress distribution in a typical Ti-6Al-4V aerospace bracket by using ANSYS. Analysis is more economic and time saving phenomenon and can be used to monitor the cracks in various components of aerospace structures and components.

Priyanka D. Toliya et. al. [3] investigated the failure analysis of the connecting rod of the automotive engine. Static analysis is done to determine the von Misses stress, elastic strain, total deformation in the present design connecting rod for the given loading conditions using the FEM Software Ansys.

Bhumesh J. Bagde et. al. [4] studied Static Structural Analysis and fatigue analysis of crank shaft in ANSYS software for five different materials.

In present work, rigid dynamic analysis is performed and joint loads are taken for stress analysis of each component of wiper linkage mechanism in ANSYS.

IV. PROBLEM DEFINITION
Dynamic stress analysis of each component of wiper-linkage mechanism is to be performed to check whether design is correct or not. 3D model of wiper mechanism was taken from Catia V5 in step format. Model is as shown in Fig.2.

V. METHODOLOGY
Firstly rigid dynamic analysis was done in ANSYS. From this joint loads were taken for further analysis. Motor speed of 65RPM was applied.

After this transient structural stress analysis was performed for each component of wiper linkage mechanism. Loads for this were taken from rigid dynamic analysis.

Fig.3: Rigid Dynamics Analysis
After this transient structural stress analysis was performed for each component of wiper linkage mechanism. Loads for this were taken from rigid dynamic analysis.

Meshing is the next step in finite element analysis. In this step part or component is divided into number of small elements. The purpose of discretization is to perform the analysis on each small division separately.

Meshing of each component is shown in following figures.

Fig.4: Meshing of crank
Fig.5: Meshing of Link 1A
Fig.6: Meshing of Link 1B
After meshing, loading and boundary conditions are applied. Loading is as shown in figures.
After the application of boundary conditions and force, the next step is to perform the transient structural analysis of each component. In this transient structural analysis, we are mainly concerned with the stresses acting on each component (von-mises stresses).

Stresses on each component are as shown in figures.

VI. RESULTS & CONCLUSION

From stress results obtained, we can say that stress for each component is within limit. We can conclude that design made is correct. So we can say that FEA is very helpful tool in early design stage which reduces time & cost of after production.

REFERENCES


All rights reserved by www.ijsrd.com 278