Regional Conference on Computational Mechanics & Numerical Analysis (CMNA 2006)
12-14 May 2006, Banda Acheh, Indonesia.

MODELING OF GEAR MECHANISM USING TRANSIENT FINITE ELEMENT METHOD

F.R.M. Romlay
Faculty of Mechanical Engineering
Kolej Universiti Kejuruteraan & Teknologi Malaysia (KUKTEM),
P.O. Box 12, 25000 Kuantan Pahang, MALAYSIA.
Phone: 609-5492219, Fax: 609-5492244, e-mail: fadhlur@kuktem.edu.my

W. A. W. Yusoff
Faculty of Mechanical Engineering
Kolej Universiti Kejuruteraan & Teknologi Malaysia (KUKTEM),
P.O. Box 12, 25000 Kuantan Pahang, MALAYSIA.
Phone: 609-5492204, Fax: 609-5492244, e-mail: wazhar@kuktem.edu.my

Abstract

This paper presents a dynamic analysis of gear model using Finite Element Method (FEM) via transient characteristic. Traditionally, the stress analysis is done separately with dynamic properties due to limitation of complex equation and error control. A FEM analyst is carrying out to formulate and solve large systems of algebraic equations to obtain the relationship between the contact parameter and the kinematics function. The loading conditions are chosen due to affects of gear mechanism which can influence the gear transmission. Therefore, model analysis for gear mechanism is obtained to investigate the stress distribution on real time by transient FEM. From the transient FEM results, the analysis has to determine which contacts occur at each stage of the work cycle, to derive the resulting kinematics functions, and to identify qualitative kinematics variations due to contact changes in time-step domain. The results from the simulation are in good agreement comparing with the theory calculation.

Keywords: Gear mechanism, transient, Finite Elements Method

1. Introduction

Now days, there are so many mechanisms those involve with load and requirement to understand the stress in component is increased. The mechanisms and the stress always come together and they have a strong relation between each other.

In real application, displacement and stress are dependent with the dynamic criteria. The loads that apply to the model that got motion consider as transient load. This will involve kinetic and kinematic criteria of the model.

Presently, the model was analyzed by finite element method to identify the stress and displacement. The same model need to be design again or is possible, is exported to the dynamic analyzer for a purposed of dynamic study. This needs a long time to finish a simulation work. The CAD design also can be damage during the file transfer process. This factor can affect the accuracy of the result.

Using of finite element result for the dynamic analysis input is very difficult because of the involvement of a complex mathematical equation and high possibility of error to be occurred. That is why both of the analysis needs to be combined in a single analyzer.

2. Stress Analysis

Usually, stress analysis for the dynamic component is done by stand alone analyzer using finite element method. Using a traditional method, the stress of the dynamic mechanical part is estimated by separated time step using quasi-static stress analysis approach [1]. The equation is given by Eq. (1), (2) and (3).

$$\sigma_x(t)=\sum_{i=1}^{n}\sigma_{xi}F_i(t) \tag{1}$$

$$\sigma_y(t)=\sum_{i=1}^{n}\sigma_{yi}F_i(t) \tag{2}$$

$$\tau_{xy}(t)=\sum_{i=1}^{n}\tau_{xyi}F_i(t) \tag{3}$$

where n is the number of applied load histories and $\sigma_{xi}(t)$ $\sigma_{yi}(t)$ $\tau_{xyi}(t)$are the stress due to a unit load. The stress is applied at a specific nodal and a same direction with the load history $F_i(t)$.

In this condition, the analysis is chosen just for a certain moment from the whole mechanism. Usually engineer need to predict the critical time and result for the whole process cannot be performed.

3. Transient Modeling

Transient dynamic analysis is used to predict the motion behavior of the component. The dynamic modeling includes a path and trajectory. To be clear, the path only considers the coordinate and vector of the line, while the trajectory integrated a time domain of the path.

Modeling the mechanism of the part by transient analysis parallel with the finite element method will make simulation become easier. The stress and displacement results will come together with the kinetic and kinematic information. The model is verified by repeating the program in an iteration form which is defined by time-step function.

Static stress analysis exist because of the load applied and by the same load, the part got a motion. That means the equation of the dynamic mechanism is dependent with the static stress result. Consider this factor, the error occur from static stress analysis will dramatically increase the error of the dynamic modeling especially by the involvement of multiple function [2]. That is why the performance of error control is very important in combining static stress analysis and transient dynamic modeling.

4. Simulation of Transient Finite Element

Prior to this section, this research focus on understanding of stress distribution in gear mechanism. The model was developed as illustrates in Fig. 1 using brick elements.

First of all, the gear assembly was defined by finite element method. The CAD model was meshed and defined using brick element. The constraint was declared by a pin at the centre of the gear. A torque is applied at the pin to drive the gear mechanism and at the same time applied the load to the part.

With is consider the assembly of the main and pinion gear, the affect of the surface contact will result the stress around the gear part. To make it happen, surface contact need to be identify and it is driven by actuator element. It should be noted that, actuator elements are used to specify the relative motion of two points of a structure or mechanism.

Consider the dynamic affect direct to the model, the angular velocity is applied and the time-step needs to be defined. All these kinetic and kinematic parameters were contributed the mechanism of gear motion.

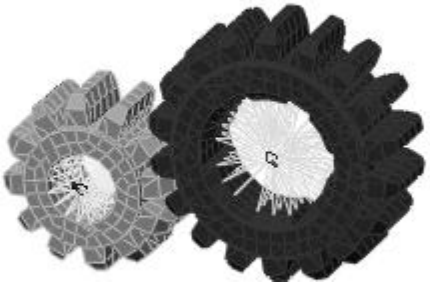

Fig. 1 Modeling of main and pinion gear with respect to the torque at a pin for each gear.

Fig. 2 illustrated the result of the stress distribution for the forth time-step. The Von-Mises stress result was represent by time-step within 0.0002 seconds for a single time-step. The maximum Von Mises stress is 234029 lbf/in^2. The maximum Von Mises stress is increase to 3596.89 lbf/in^2 at eighth time-step as shown in Fig. 3. However, the value decreases to 2893.84 lbf/in^2 at twelfth time-step as shown in Fig. 4. The value continues reduce to

1827.17 lbf/in^2 at sixteenth time-step as shown in Fig. 5.

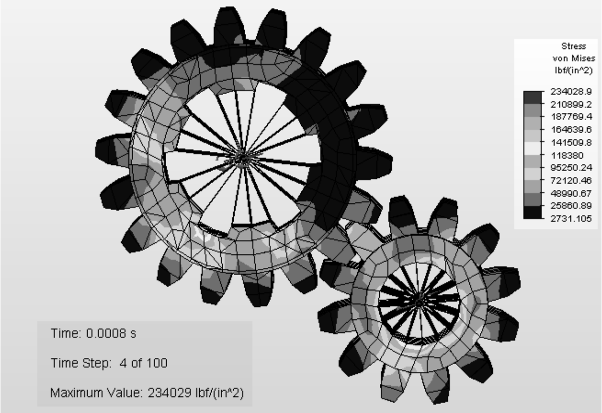

Fig. 2 Stress distribution for forth time-step

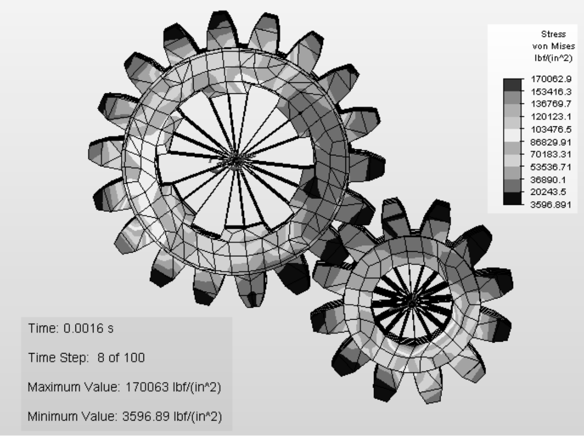

Fig. 3 Stress distribution for eighth time-step

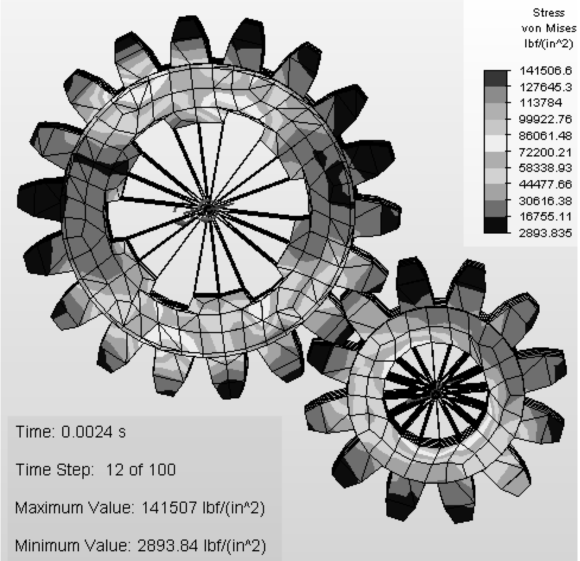

Fig. 4 Stress distribution for twelfth time-step

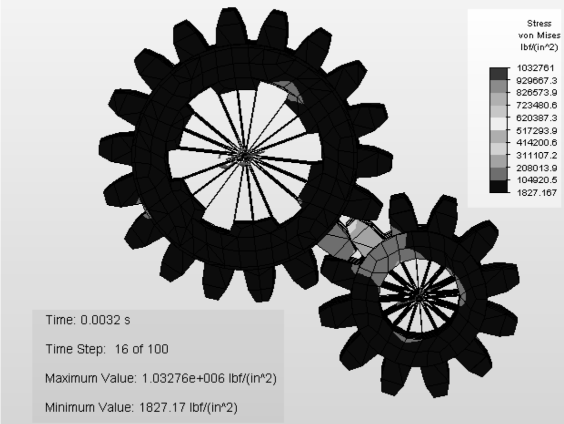

Fig. 5 Stress distribution for sixteenth time-step

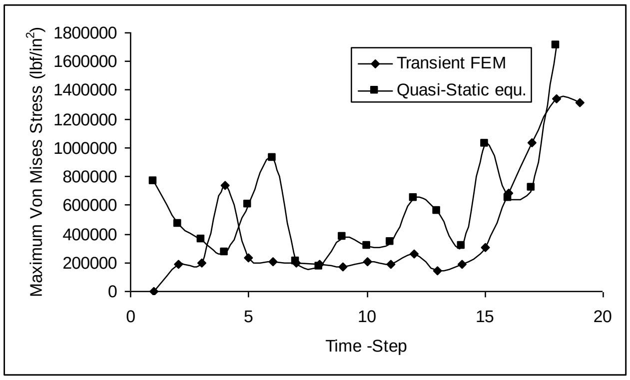

Fig. 6 Graph of maximum Von Mises Stress versus time-step.

Fig. 6 shown the result of transient finite element method validate with deterministic approach by quasi-static equation. Both the curve is not in uniform shape and the maximum Von Mises stress is variable along the time step of the event.

5. Conclusions

The result of the analysis has shown that the factor of dynamic properties will affect the stress distribution of the component by showing the changing of the stress value in time domain function. By the using transient FEM, the stage of the computational analysis will be short and this will create an efficient design work.

References

[1] M. Haiba, D. C. Barton, P. C. Brooks & M. C. Leveslay, *Review of life assessment techniques applied to dynamically load automotive components*, Computer and Structures, **80,** 2002, pp. 481-494.

[2] A. K. Ariffin & F. R. M. Romlay, *Deterministic and Probabilistic Approach in Modelling of Fatigue Crack Propagation*, International Conference on Computational Methods, Singapore, 15-17 Dec. 2004.

[3] M. M. Rahman, A. K. Ariffin, N. Jamaluddin and C. H. C. Haron, *Prediction of fatigue lifetime caused to vibrating cylinder block of a two-stroke free piston engine using transient dynamic approach*, Conference on Spatial and Computational Engineering, 2005.