CHAPTER 1

INTRODUCTION

1.1 BACKGROUND

Connecting rods are highly dynamically loaded components used for power transmission in combustion engines. They can be produced either by casting, powder metallurgy or forging. There exist only few publications on the simulation of the forging process for the production of connecting rods. (Grass et al., 2005). Connecting rod transfer force from the piston after the combustion in engine to the crankshaft. (Bell, 1997). The connecting rod was invented sometime between 1174 and 1200 where a muslim inventor, engineer and craftsman named al-Jazari build five machine to pump water for the king of the Turkish Artaqid dynasty. The connecting rod are most made of steel for production engines, but can be made of aluminum (for lightness and the ability to absorb high impact at the expense of durability) or titanium (for combination of strength and lightness at the expense of affordability) for high performance engines, or of cast iron.

Automotive industry requirements for quality, productivity and cost efficiency are at a level where study of the design and manufacturing of a product must occur in the earliest stages of conception. The time spent in trial and error analysis in the design process needs to be eliminated for a manufacturer to remain competitive in a global market. Therefore, computational method have been used in the early stage of the design. (Lauwagie et al., 2008)
Finite element method is applied during modal analysis of connecting rod. Modal analysis is the process of determining the inherent dynamic characteristics of a system in form of natural frequencies, damping factors and mode shapes, and using them to formulate a mathematical model for its dynamic behavior. Hence, mesh determination is too critical in order to ensure that the best mesh size is to be use in carry out the analysis for other parameter involves. As stability and convergence of various mesh processing applications depend on mesh quality, there is frequently a need to improve the quality of the mesh (Taubin, 1995). This improvement process is called mesh optimization (Hoppe et al., 1993).

For the improvement of the design, modal updating can be run in order to determine the effect of the material properties to the dynamic characteristic of the design. Furthermore, modal analysis can be used for the experimental modal analysis (EMA) for the experiment setup to locate the accelerometer and the result of the experimental modal analysis can be correlate with the result of the computational modal analysis.

1.2 PROBLEM STATEMENT

Any physical system can vibrate. The frequencies at which vibration naturally occurs, and the modal shapes which the vibrating system assumes are properties of the system can be determined analytically using modal analysis. Analysis of vibration modes is a critical component of a design, but is often overlooked. Inherent vibration modes in structural components can shorten connecting rod life, and causes premature or completely unanticipated failure, often resulting in hazardous situations. Detailed modal analysis determines the fundamental vibration modes shapes and corresponding frequencies. This can be relatively simple for basic components of a simple system, end extremely complicated when qualifying a complex mechanical device or a complicated structure exposed to periodic wind loading. These system required accurate determination of natural frequencies and mode shape using techniques such as Finite Element Analysis. Using Finite Element models to predict the dynamic properties of
structures becomes more and more important in modern mechanical industries, such as the automobile industries. Whenever there is new design or modification of an existing design, the structural dynamic properties of the product must be examined to fulfill some criteria proposed either by the industry itself and/or external agencies before the product can be launched on the market. FE model predictions are used more and more to take the place of practical dynamic test data (Chen, 2001).

1.3 MOTIVATION

Automotive industry requirements for quality, productivity and cost efficiency are at a level where optimization of the design and manufacturing of a product must occur in the earliest stages of conception. The time spent in trial and error analysis in the design process needs to be eliminated for a manufacture to remain competitive in a global market. The Finite Element optimization approach allows an efficient evaluation of designs using advance mathematical tools. Methods must be developed, however, to create fully parameterized models for optimization which can be experimentally verified under pertinent loading conditions (Lauwagie et al., 2008).

1.4 OBJECTIVES

- To develop a finite element model of structure
- To study the vibration of a connecting rod in order to determine its natural frequencies and mode shapes.
- To investigate the effect of the modal updating to the dynamic characteristic of the connecting rod

1.5 SCOPE OF STUDY

- Solid work – The model of connecting rod are developed using Solid Work software after taking dimension.
- Finite Element Method – using for modal analysis