UNIVERSITI TEKNIKAL MALAYSIA MELAKA

FINITE ELEMENT ANALYSIS
OF
CAR ALLOY WHEEL

Thesis submitted in accordance with the partial requirements of the
Universiti Teknikal Malaysia Melaka for the
Bachelor of Manufacturing Engineering (Manufacturing Design)

By

MOHD ZUL HAZMI BIN MHD FAUZY

Faculty of Manufacturing Engineering

April 2008

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

BORANG PENGESAHAN STATUS TESIS*

JUDUL: FINITE ELEMENT ANALYSIS OF CAR ALLOY WHEEL

SESI PENGAJIAN : 2007/2008

Saya **MOHD ZUL HAZMI BIN MHD FAUZY**

(HURUF BESAR)

mengaku membenarkan tesis (PSM/Sarjana/Doktor Falsafah) ini disimpan di Perpustakaan Universiti Teknikal Malaysia Melaka (UTeM) dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hak milik Universiti Teknikal Malaysia Melaka .
2. Perpustakaan Universiti Teknikal Malaysia Melaka dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (√)

☐	SULIT	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia yang termaktub di dalam AKTA RAHSIA RASMI 1972)
☐	TERHAD	(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)
√	TIDAK TERHAD	

Disahkan oleh:

(TANDATANGAN PENULIS) (TANDATANGAN PENYELIA)

Alamat Tetap:
654 JALAN AMAN 13,
TAMAN AMAN, ANAK BUKIT
06550 ALOR STAR
KEDAH.
Tarikh: 15/05/08

Cop Rasmi:
WAHYONO SAPTO WIDODO
Pensyarah
Fakulti Kejuruteraan Pembuatan
Universiti Teknikal Malaysia Melaka
Karung Berkunci 1200, Ayer Keroh
75450 Melaka

Tarikh: ____________

* Tesis dimaksudkan sebagai tesis bagi Ijazah Doktor Falsafah dan Sarjana secara penyelidikan, atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM).
** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali sebab dan tempoh tesis ini perlu dikelaskan sebagai SULIT atau TERHAD.

DECLARATION

I hereby, declare this thesis entitled "Finite Element Analysis of Car Alloy Wheel" is the result of my own research except as cited in the references.

Signature	:	....
Author's Name	:	Mohd Zul Hazmi Bin Mhd Fauzy
Date	:	25th March 2008

APPROVAL

This thesis submitted to the senate of UTeM and has been accepted as partial fulfillment of the requirements for the degree of Bachelor of Manufacturing Engineering (Manufacturing Design). The members of the supervisory committee are as follow:

WAHYONO SAPTO WIDODO
Pensyarah
Fakulti Kejuruteraan Pembuatan
Universiti Teknikal Malaysia Melaka
Karung Berkunci 1200, Ayer Keroh
75450 Melaka

....

Main Supervisor
(Mr Wahyono Sapto Widodo)

....

Co-Supervisor
(En. Ismail Bin Abu Shah)

ABSTRACT

Finite element analysis is one of the methods to analyze the product by doing a simulation. This method is quite long used in manufacturing industry and had earned many benefits to the company who used this kind of software. It is because this simulation can reduce the times, costs and can improving the quality of the product.

The tests on the product can be done by using the simulation without producing the real product to do the tests. The engineers just make the design and its specification of the product before ready to do the simulation. By this method, not only the time can be reduced but the cost for producing the testing product also can be reduced. The costs earning from the reducing can be invest to develop the upcoming products.

ABSTRAK

‘Finite Element Analysis’ merupakan salah satu cara untuk menganalisis sesuatu produk dengan menggunakan kaedah simulasi. Kaedah ini sudah agak lama digunakan di dalam industri pembuatan dan telah mendatangkan banyak keuntungan kepada syarikat yang mengendalikannya. Ini kerana kaedah simulasi ini dapat menjimatkan dari segi masa, kos, dan juga dapat meningkatkan kualiti sesuatu produk.

Ujian terhadap sesuatu produk dapat dijalankan dengan hanya menjalankan simulasi tanpa perlu untuk menghasilkan produk diuji. Jurutera hanya perlu menghasilkan rekabentuk dan juga spesifikasi untuk produk sebelum sedia untuk diuji. Dengan kaedah ini, bukan masa sahaja dapat dijimatkan malah kos untuk menghasilkan bahan ujikaji juga tidak diperlukan lagi. Kos yang telah dapat dijimatkan boleh digunakan untuk membangunkan produk yang baru.

DEDICATION

To my beloved father, mother and to all of my friends especially who had helped me for finish up this project.

ACKNOWLEDGEMENT

First of all, I want to thank to Allah SWT for giving me strength and wealthy to complete my Projek Sarjana Muda (PSM) titled "Finite Element Analysis of Car Alloy Wheel". Then, I also would like to thank my supervisor, Mr. Wahyono Sapto Widodo who had gives me more knowledge and guide to complete this project, all of my friends especially Mr. Rizal from class BMCD who had monitoring me in my analysis by using Nastran/Patran software and also who either directly or indirectly helped me in this task. Thanks you very much.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1

INTRODUCTION

1.1 Background

Finite Element Analysis (FEA) was first developed in 1943 by R. Courant, who utilized the Ritz method of numerical analysis and minimization of variational calculus to obtain approximate solutions to vibration systems. Shortly thereafter, a paper published in 1956 by M. J. Turner, R. W. Clough, H. C. Martin, and L. J. Topp established a broader definition of numerical analysis. The paper centered on the "stiffness and deflection of complex structures".

By the early 70's, FEA was limited to expensive mainframe computers generally owned by the aeronautics, automotive, defense, and nuclear industries. Since the rapid decline in the cost of computers and the phenomenal increase in computing power, FEA has been developed to an incredible precision. Present day supercomputers are now able to produce accurate results for all kinds of parameters.

FEA consists of a computer model of a material or design that is stressed and analyzed for specific results. It is used in new product design, and existing product refinement. A company is able to verify a proposed design will be able to perform to the client's specifications prior to manufacturing or construction. Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition.

There are generally two types of analysis that are used in industry: 2-D modeling, and 3-D modeling. While 2-D modeling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3-D modeling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively. Within each of these modeling schemes, the programmer can insert numerous algorithms (functions) which may make the system behave linearly or non-linearly. Linear systems are far less complex and generally do not take into account plastic deformation. Non-linear systems do account for plastic deformation, and many also are capable of testing a material all the way to fracture.

1.2 Problem statement

For this study, the main problems that will consider are to study and redesign the previous design. The story of a wheel begins with a design idea. Design specification documentation is drawn up outlining the basic definition, appearance and the technical structure of the wheel (one-section, two-section or multi-section). This idea is then brought to life using special CAD software to create the first 3D model for testing. The software allows the wheel to be turned three-dimensionally and for improvements to be made easily. Many specifications that must be consider before the alloy wheels are designed and ready to produce. It's about the ability of the alloy wheels to support the car whether in static or dynamic situation. The correct in selected material is also important to make sure the alloy wheels are good in quality and its performances.

The process to develop a new design of alloy wheels will cause a high costs. With the new technology, we try avoiding the test by using the actual product which is can help to reduce the previous costs. By using the new software today, we not only can design the alloy wheels but also can analysis the product in details. The defects from the design can be detected when doing a simulation on the product. By using FEA simulation, the more precise and accurate result will be get.

1.3 Objective

The main objectives that have to be in consideration for this project; stated as below:

a) To study about the alloy wheel design nowadays.
b) Try to design the alloy wheel by using CATIA software.
c) Do the test via simulation by using the CATIA or Nastran software.
d) Learn how to improve the design of the alloy wheels.
e) Analyzed the results and make future work recommendations.

1.4 Scope

The task will be started by studying about the alloy wheels before try to analysis the product. The alloy wheel then must be design by using CATIA software. The design is very important in getting a good result during the analysis. The analysis can be done by simulate the design using CATIA/Patran.

There are too many tests that must be considered before an alloy wheels are ready to enter the market. For this project, static test were choose to analyze on the design of the alloy wheels. The result will be analyzed and the recommendation to improve the design of the alloy wheels was list out based on the result obtained.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction to FEA

Computer-aided engineering (CAE) is the application of computer software in engineering to evaluate components and assemblies. It encompasses simulation, validation, and optimization of products and manufacturing tools. The primary application of CAE, used in civil, mechanical, aerospace, and electronic engineering, takes the form of FEA alongside computer-aided design (CAD).

In general, there are three phases in any computer-aided engineering task:

a) Pre-processing – defining the finite element model and environmental factors to be applied to it
b) Analysis solver – solution of finite element model
c) Post-processing of results using visualization tools

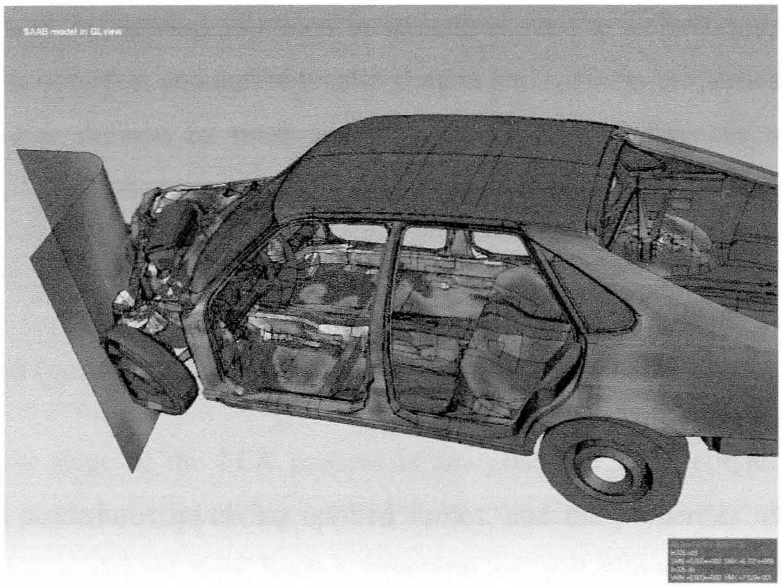

Figure 1. FEA analysis

2.1.1 Pre-processing

The first step in using FEA, pre-processing, is constructing a finite element model of the structure to be analyzed. The input of a topological description of the structure's geometric features is required in most FEA packages. This can be in either 1D, 2D, or 3D form, modeled by line, shape, or surface representation, respectively, although nowadays 3D models are predominantly used. The primary objective of the model is to realistically replicate the important parameters and features of the real model. The simplest mechanism to achieve modeling similarity in structural analysis is to utilize pre-existing digital blueprints, design files, CAD models, and/or data by importing that into an FEA environment. Once the finite element geometric model has been created, a meshing procedure is used to define and break up the model into small elements. In general, a finite element model is defined by a mesh network, which is made up of the geometric arrangement of elements and nodes. Nodes represent points at which features such as displacements are calculated. FEA packages use node numbers to serve as an identification tool in viewing solutions in structures such as deflections. Elements are bounded by sets of nodes, and define localized mass and stiffness properties of the model. Elements are also defined by mesh numbers, which allow references to be made to corresponding deflections or stresses at specific model locations.

2.1.2 Analysis (computation of solution)

The next stage of the FEA process is analysis. The FEM conducts a series of computational procedures involving applied forces, and the properties of the elements which produce a model solution. Such a structural analysis allows the determination of effects such as deformations, strains, and stresses which are caused by applied structural loads such as force, pressure and gravity.

2.1.3 Post-processing (visualization)

These results can then be studied using visualization tools within the FEA environment to view and to fully identify implications of the analysis. Numerical and graphical tools allow the precise location of data such as stresses and deflections to be identified.

A wide range of objective functions (variables within the system) are available for minimization or maximization:

a) Mass, volume, temperature
b) Strain energy, stress strain
c) Force, displacement, velocity, acceleration
d) Synthetic (User defined)

There are multiple loading conditions which may be applied to a system. Next to Figure 3, some examples are shown:

a) Point, pressure (Figure 3), thermal, gravity, and centrifugal static loads
b) Thermal loads from solution of heat transfer analysis
c) Enforced displacements
d) Heat flux and convection
e) Point, pressure and gravity dynamic loads

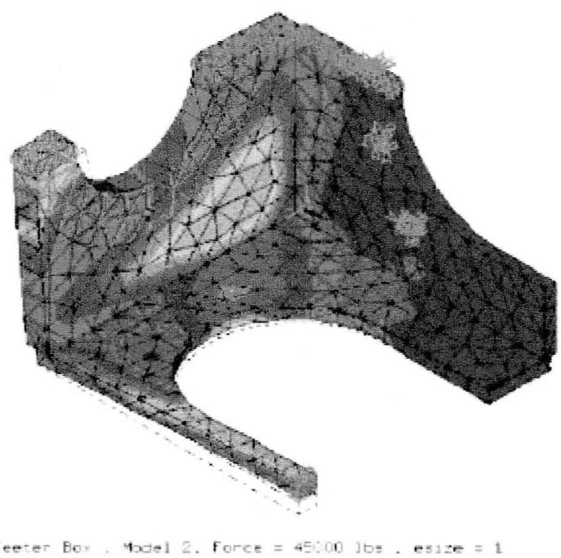

Figure 3. FEA using pressure

2.2 Finite Element Analysis Principles

FEA uses a complex system of points called nodes which make a grid called a mesh (Figure 2). This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. Regions which will receive large amounts of stress usually have a higher node density than those which experience little or no stress. Points of interest may consist of: fracture point of previously tested material, fillets, corners, complex detail, and high stress areas. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes. This web of vectors is what carries the material properties to the object, creating many elements.

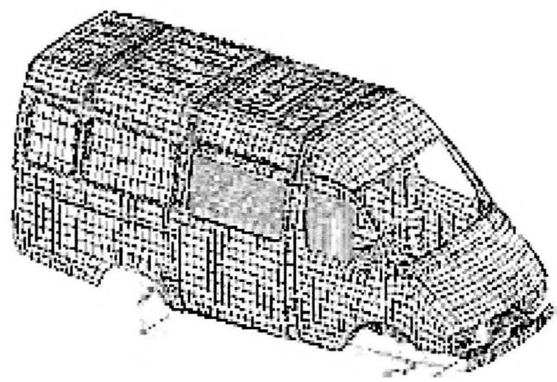

Figure 2. FEA using nodes

Each FEA program may come with an element library, or one is constructed over time. Some sample elements are:

a) Rod elements
b) Beam elements
c) Plate/Shell/Composite elements
d) Shear panel
e) Solid elements
f) Spring elements
g) Mass elements
h) Rigid elements
i) Viscous damping elements

Many FEA programs also are equipped with the capability to use multiple materials within the structure such as:

a) Isotropic, identical throughout
b) Orthotropic, identical at 90 degrees
c) General anisotropic, different throughout

2.3 Types of Engineering Analysis

Structural analysis consists of linear and non-linear models. Linear models use simple parameters and assume that the material is not plastically deformed. Non-linear models consist of stressing the material past its elastic capabilities. The stresses in the material then vary with the amount of deformation as in Figure 4.

Vibrational analysis is used to test a material against random vibrations, shock, and impact. Each of these incidences may act on the natural vibrational frequency of the material which, in turn, may cause resonance and subsequent failure.

Figure 4. amount of deformation

Fatigue analysis helps designers to predict the life of a material or structure by showing the effects of cyclic loading on the specimen. Such analysis can show the areas where crack propagation is most likely to occur. Failure due to fatigue may also show the damage tolerance of the material.

Heat Transfer analysis models the conductivity or thermal fluid dynamics of the material or structure. This may consist of a steady-state or transient transfer. Steady-state transfer refers to constant thermo properties in the material that yield linear heat diffusion.

Results of Finite Element Analysis

FEA has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material and allowing designers to see all of the theoretical stresses within. This method of product design and testing is far superior to the manufacturing costs which would accrue if each sample was actually built and tested.

2.4 Applications of FEA

2.4.1 Applications of FEA to the Mechanical Engineering Industry

A variety of specializations under the umbrella of the mechanical engineering discipline such as aeronautical, biomechanical, and automotive industries all commonly use integrated FEA in design and development of their products. Several modern FEA packages include specific components such as thermal, electromagnetic, fluid, and structural working environments. In a structural simulation FEA helps tremendously in producing stiffness and strength visualizations and also in minimizing weight, materials, and costs. FEA allows detailed visualization of where structures bend or twist, and indicates the distribution of stresses and displacements. FEA software provides a wide range of simulation options for controlling the complexity of both the modeling and the analysis of a system. Similarly, the desired level of accuracy required and the associated computational time requirements can be managed simultaneously to address most engineering applications. FEA allows entire designs to be constructed, refined, and optimized before the design is manufactured. This powerful design tool has significantly improved both the standard of engineering designs and the methodology of the design process in many industrial applications. The introduction of FEA has substantially decreased the time taken to take products from concept to the production line. It is primarily through improved initial prototype designs using FEA that testing and development have been accelerated. In summary, the benefits of FEA include increased accuracy, enhanced design and better insight into critical design parameters, virtual prototyping, fewer hardware prototypes, a faster and less expensive design cycle, increased productivity, and increased revenue.

2.4.2 Computer-aided Design and Finite Element Analysis in Industry

The ability to model a structural system in 3D can provide a powerful and accurate analysis of almost any structure. 3D models, in general, can be produced using a range of common computer-aided design packages. Models have the tendency to range largely in both complexity and in file format, depending on 3D model creation software and the complexity of the model's geometry. FEA is a growing industry in product design, analysis, and development in engineering. The trend of utilizing FEA as an engineering tool is growing rapidly. The advancement in computer processing power, FEA, and modeling software has allowed the continued integration of FEA in the engineering fields of product design and development. In the past, there have been many issues restricting the performance and ultimately the acceptance and utilization of FEA in conjunction with CAD in the product design and development stages. The gaps in compatibility between CAD file formats and FEA software limited the extent to which companies could easily design and test their products using the CAD and FEA combination, respectively. Typically, engineers would use specialist CAD and modeling software in the design of the product and then wish to export that design into a FEA package to test. However, those engineers who depended on data exchange through custom translators or exchange standards such as IGES or STEP cite occasional reliability problems causing unsuccessful exchange of geometry. Thus, the creation of many models external to FEA environments was considered to be problematic in the success of FEA.