DESIGN OPTIMIZATION FOR THE ACTIVE BUMPER SYSTEM TEST RIG

MUHAMMAD SYUKRI B. ISMAIL

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

"I admit that I have read this report and it has followed the scope and quality in partial fulfillment of requirement for the degree of Bachelor of Mechanical Engineering (Automotive)

Signature : ……..

First Supervisor : Mr. Mohd Zakaria b. Mohamad Nasir

Date : ……..

Signature : ……..

Second Supervisor : Dr. Yusmady

Date : ……..

DESIGN OPTIMIZATION THE ACTIVE BUMPER SYSTEM TEST RIG

MUHAMMAD SYUKRI B. ISMAIL

This report is written as a partial fulfillment of terms in achieving the awards for

Bachelor of Mechanical Engineering (Automotive)

Faculty of Mechanical Engineering

Universiti Teknikal Malaysia Melaka

APRIL 2010

"I admit that this report is from my own work and idea except for the summary and a few sections which were extracted from other resources as being mention".

Signature : …….....

Writer Name : Muhammad Syukri b. Ismail

Date : ……...

To my lovely parents, my brother and sister who give me encouragement to success in my studies and not to forget special thanks to all my lecturers and friends that give me guideline and support during my study in UTeM

ACKNOWLEDGEMENT

Firstly I would like to forward my utmost gratitude to my supervisor, En Mohd Zakaria b. Mohd Nasir because give me guideline and inducement to complete my PSM (Projek Sarjana Muda). Special thanks to En Alif Zulfakar b. Pokaad as a master student who also guide me and gave me many information to perform my project.

I would like to send my appreciation to all lab management, especially to the technicians who give corporation for me to complete my project.

Lastly I would like to thanks to all person who involve directly and indirectly during the project research. I hope that my report will be source for other student in the future.

ABSTRAK

Kertas Projek Sarjana Muda ini menunjukkan langkah yang diambil untuk meningkatkan keburukan yang ada pada alat ujian sistem aktif bumper yang sedia ada di dalam makmal autotronik. Alat ujian sistem aktif bumper dijalankan dengan melepaskan bandul dan bandul memukul model kereta pada ketinggian dan berat tertentu. Dua cara digunakan untuk memperbaiki kelemahan alat ujian ini iaitu melalui menganalisa bahagian alat ujian menggunakan perisian berbantu kejuteraan dan dengan menjalankan eksperimen. Bandul alat ujian sistem bumper aktif telah dianalisis menggunakan perisian CATIA analisis manakala pegangan rig telah dianalsis menggunakan FEA Abaqus perisian. Beberape modifikasi telah dilakukan pada alat ujian sistem aktif bumper iaitu penukaran bandul dan penambahan bahagian depan model kereta ditambah dengan spring untuk menyerap daya daripada bandul agar daya yang berkali-kali sepanjang ujikaji tidak merosakkan model kereta. Spring juga boleh diwakilkan sebagai bahagian remuk hadapan kereta ketika perlanggaran. Cara melakukan ujikaji untuk menguji keberkesanan sistem bumper aktif juga ditunjukkan dan keputusan ujikaji diterangkan seperti kelakuan MR damper dan keberkesanan sistem “Skyhook”.

ABSTACT

This Final year project (FYP) report shows the step taken to improve the current test rig for active bumper system experiment that located in Autotronic lab. There are 2 methods to improve to disadvantage of Active bumper crash test rig which is by using CAE/FEA software analysis and analysis active bumper system experiment result. Active bumper system test rig part is analyzed using CATIA analysis software and FEA Abaqus software. Some modification has been made to the crash test rig which is change of new pendulum and spring addition into the front of the car model to reduce the force and damage of the car model during repetitive experiment or force applied to the car model. This spring can represent as front crumple zone that absorb impact force. Step to perform experiment to study the effectiveness of active bumper system also shown in this report and result of the experiment also are explained in this report.

TABLE OF CONTENT

LIST OF FIGURE

LIST OF GRAPH

LIST OF TABLE

LIST OF SYMBOL

F	=	Force, N
a	=	Acceleration, m/s^2
v	=	Velocity, m/s
t	=	Time, s
m	=	Mass, kg
ρ	=	Density, kg/m^3
E_k	=	Kinetic Energy
E_p	=	Potential Energy
A	=	Area m^2
k	=	Specific material stiffness
$\ddot{S}v$	=	Deceleration of the vehicle as function of time f(t)
$\dot{S}v$	=	Velocity of the vehicle during the impact
Sv	=	Deformation of the vehicle during the impact
ΔV	=	Change in velocity
ζ	=	Damping Ratio
ω_n	=	Natural Frequency
e	=	Coefficient of restitutions
A	=	Ampere
V	=	Volt

LIST OF APPENDIX

CHAPTER 1

INTRODUCTION

1.1 Background

The purpose of this thesis is to improvise current test rig for active bumper system for passenger vehicle model in UTeM Autotronic Lab. There are two way to improve the test rig design. Firstly the test rig critical part is analyzed using CAE software. The result data for CAE analysis then will be used to optimize the test rig. The result of test rig experiment using sensor such as accelerometer, load cell, LVDT and others is used to check the sensitivity of the sensor whether the test is suitable for the sensor reading capability and the experiment data is used to improve the design of test.

1.2 Problem Statement

The development of Active Bumper System using Magnetorheological fluid will absorb some amount of energy during crash due to impact. This study have to find out how much energy and the impact on vehicle model. Therefore the test rig must sustain for repetitive impact or experiment.

1.3 Objective

To perform analysis and optimization of current test rig use in Autotronic lab for active bumper system experiment method.

1.4 Scope

1. Literature review on FEM, Crash method, Frontal crash, Active bumper system, and MR fluid.
2. Perform analysis on current test rig structure using CAE software (CATIA & ABAQUS)
3. Perform active front bumper experiment using current test rig.
4. Study and analysis disadvantages on current crash test rig in Autotronic lab.
5. Design optimization of current test rig crash.

1.5 Thesis outline

CHAPTER 1

Chapter 1 is introduction for this project. This chapter explains briefly about the project background, problem statement, objective and scope.

CHAPTER 2

Chapter 2 is literature review. This chapter explain briefly about Finite Element Method, Crash Method, Frontal Crash, Active bumper system, Magnetorheological fluid, and skyhook controller.

CHAPTER 3

Chapter 3 is methodology. This chapter consists of Flowchart, Explanation, Experiment setup and procedure, Equipment, Instrumentation, Instrument setting, and Technical specification.

CHAPTER 4

Result and discussion is in chapter 4. Chapter 4 shows the result of analysis of test rig part which is previous pendulum, new pendulum, and rig arm. The experiment result and improvement of test rig is also shown in this chapter.

CHAPTER 5

Conclusion for this project will be stated here and also recommendation to improve the result of this project in the future use.

CHAPTER 2

LITERATURE REVIEW

This chapter will discuss current of issue of crash statistic in Malaysia and current vehicle front bumper technology. Type of crash performed by international automotive society and the main test equipment which is Magnetorheological (MR) damper or MR Fluid also will be explains. Besides that, this chapter also explains the basic principle of Finite Element Method and Skyhook Controller.

2.1 Finite Element Analysis

There are several methods for solving engineering problems. New mechanical design need to be analyzed before the manufacturing process to reduce the cost and failure of the design. The finite element method is one of several methods for solving engineering problems. Figure 2.1 shows method of engineering analysis.

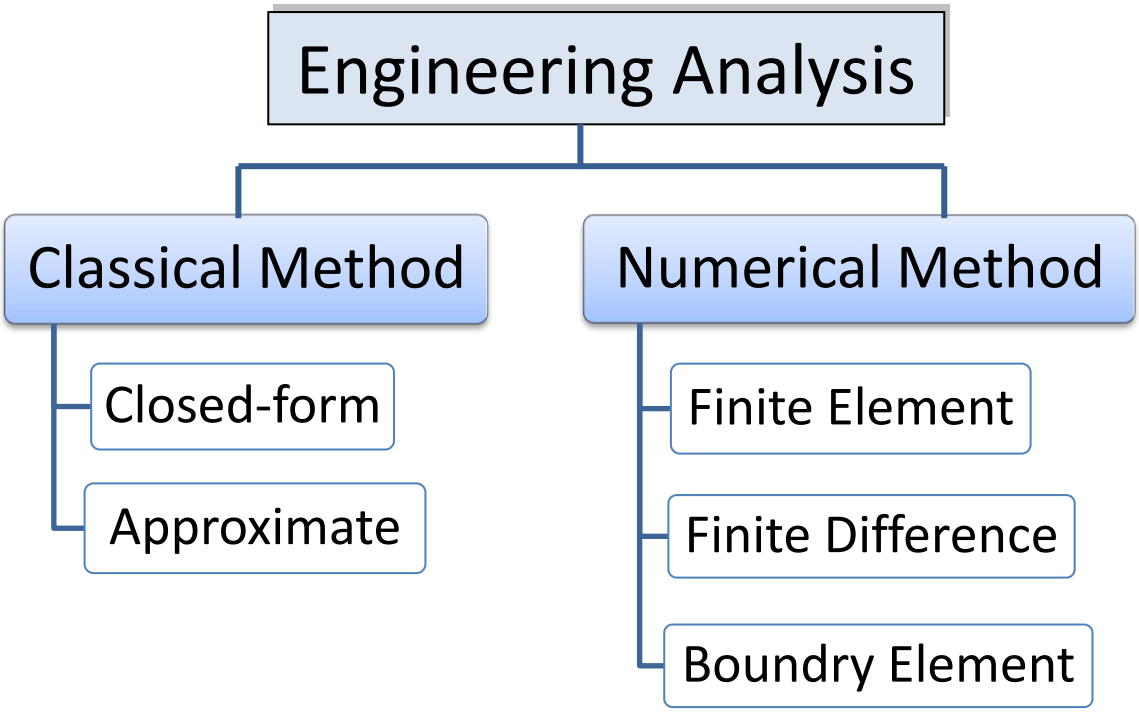


Figure 2.1: Type of engineering analysis

Classical Method:

- Closed-form solutions are available for simple problems such as bending of beams and torsion of prismatic bars
- Approximate methods using series solutions to governing differential equations are used to analyze more complex structures such as plates and shells
- The classical methods can only be used for structural problems with relatively simple geometry, loading, and boundary conditions.

Numerical Methods:

- Boundary Element Method
  - Solves the governing differential equation for the problem with integral equations over the boundary of the domain. Only the boundary surface is meshed with elements.
- Finite difference Method
  - Replaces governing differential equations and boundary conditions with corresponding algebraic finite difference equations.
- Finite Element Method (FEM)
  - Capable of solving large complex problems with general geometry, loading, and boundary conditions.
  - Increasingly becoming the primary analysis tool for designers and analysts.
  - The finite element method is also known as the Matrix method of structural analysis is the literature because it uses matrix algebra to solve the system of simultaneous equations.

Finite elements have shapes which are relatively easy to formulate and analyze. Figure 2.2 shows three basic types of finite elements which are beams, plates, and solids.

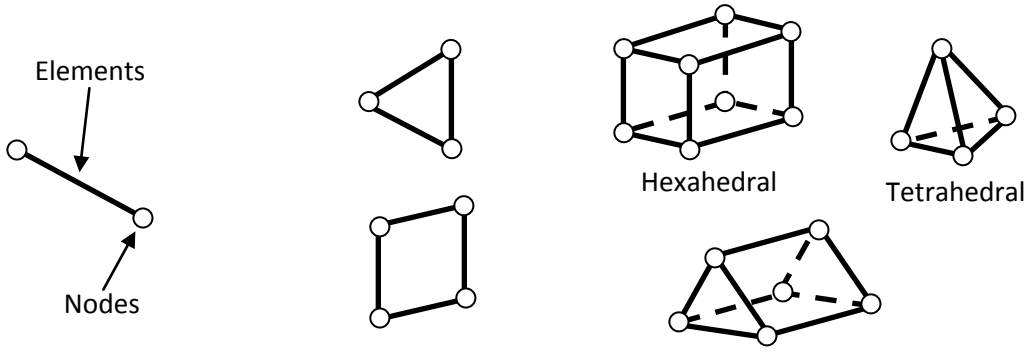


Figure 2.2: Type of FEM element shape

- The Finite element method (FEM) is a numerical approximation method. It is a method of investigating the behavior of complex structures by breaking them down into smaller, simpler pieces.
- These smaller pieces of structure are called elements. The elements are connected to each other at nodes.
- The assembly of elements and nodes is called a finite element model.

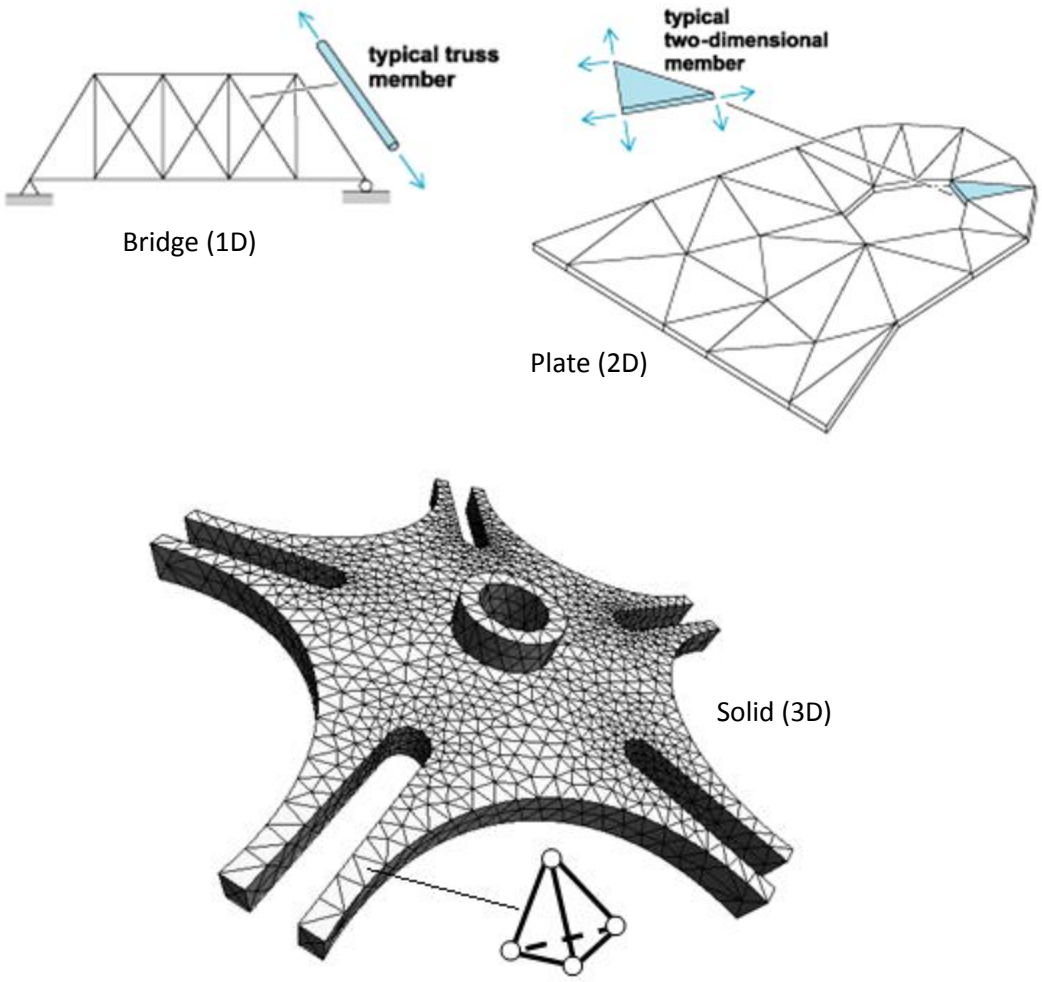


Figure 2.3: Three type of analysis dimensional
(Source: www.answers.com/topic/finite-element-method)

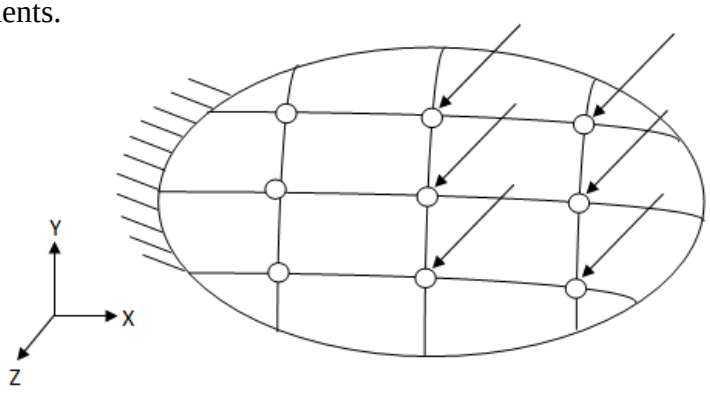
- One dimensional (1D) elements are used to model long, slender structural members, bridge and others.

- Two dimensional (2D) elements are used to model thin structural members such as thin plate, aircraft fuselage skin or car body and others.
- Three dimensional (3D) solid elements are used to model thick components such as piston head, cylinder block, and others.
- The finite element method approximates the behavior of continuous structure with a finite number of elements.
- As one increase the number of elements (and hence, decrease the size of the elements), the results become increasingly accurate, but the computing time also increase.

The operation of Finite Element Method

Basic approach

- A given problem is discretized by dividing the original domain into simply shaped elements.



- Elements are connected to each other by nodes.
- Each node is capable of moving in six independent directions: three translations and three rotations. These are called the degrees of freedom (DOF) at a node.