LINEAR STATIC STRESS ANALYSIS OF AN AUTOMOBILE FUEL TANK

MUHAMAD HAZRUL HAKIMIN B ABU SHAARI

Thesis submitted in fulfilment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this project and in my opinion, this project is adequate in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

Signature
Name of Supervisor: ASSOC. PROF. DR. MD MUSTAFIZUR RAHMAN
Position: LECTURER
Date: 17 NOVEMBER 2009

*Delete if unnecessary

STUDENT'S DECLARATION

I hereby declare that the work in this project is my own except for quotations and summaries which have been duly acknowledged. The project has not been accepted for any degree and is not concurently submitted for award of other degree.

Signature

Name: MUHAMAD HAZRUL HAKIMIN B ABU SHAARI

ID Number: MH06016

Date: 17 NOVEMBER 2009

*Delete if unnecessary

Dedicated to my parents

Mr. Abu Shaari Bin Hasan

Mrs. Zaiton Binti Omar

ACKNOWLEDGEMENTS

I am grateful and would like to express my sincere gratitude to my supervisor Assoc. Prof. Dr. Md Mustafizur Rahman for his germinal ideas, invaluable guidance, continuous encouragement and constant support in making this research possible. He has always impressed me with his outstanding professional conduct, his strong conviction for science, and his belief that a Degree program is only a start of a life-long learning experience. I appreciate his consistent support from the first day I applied to degree program to these concluding moments. I am truly grateful for his progressive vision about my training in science, his tolerance of my naïve mistakes, and his commitment to my future career. I also sincerely thanks for the time spent proofreading and correcting my many mistakes.

My sincere thanks go to all my labmates and members of the staff of the Mechanical Engineering Department, UMP, who helped me in many ways and made my stay at UMP pleasant and unforgettable. Many special thanks go to my classmates for their excellent co-operation, inspirations and supports during this study.

I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life. I acknowledge the sincerity of my brothers and sisters, who consistently encouraged me to carry on my higher studies. I am also grateful to my fellow colleuges for their sacrifice, patience, and understanding that were inevitable to make this work possible. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to attain my goals. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this study.

ABSTRACT

This thesis deals with linear static stress analysis of an automobile fuel tank. The objectives of this project are to develop a 3D model of an automobile fuel tank, finite element model, and linear static stress analysis of an automobile fuel tank. The thesis describes the finite element analysis techniques starting from the selection of best mesh types and size until the identification of the critical locations of an automobile fuel tank under static loading from. The material chosen for the automobile fuel tank was AISI S21900 Stainless Steel. The structural three-dimensional solid modelling of an automobile fuel tank was developed using the computer-aided drawing software. The strategy of the mesh optimization for the best mesh selection of finite element model was developed. The finite element analysis was then performed using MSC.NASTRAN code. The finite element model of the automobile fuel tank structure was analyzed using the linear elastic approach. The comparison of mesh configuration between tetrahedral with 4 nodes and tetrahedral with 10 nodes were made in many aspect of the results obtain including number of total nodes, displacements and stresses. From the results, it is observed that tetrahedral with 10 nodes give higher accuracy in many aspects than of tetrahedral with 4 nodes. The results obtain were then used to compare between three type of stress namely maximum principal stress, Von Mises stress and Tresca stress. The obtained results indicate that the stress not exceeding the tensile strength of the steel but rapid deformation will result leakage to the automobile fuel tank structure. The durability assessment results are significant to improve the automobile fuel tank venting system in the future. The results can also significantly help in the process of reducing the cost, and improve product reliability and customer confidence decesively.

ABSTRAK

Tesis ini berkisar tentang analisis tekanan statik berkadar terus bagi sebuah tangki bahan bakar sebuah kereta. Objektif daripada projek ini adalah memajukan model tiga dimensi bagi tangki bahan bakar sebuah kereta, model elemen terhingga, dan analisis tekanan statik berkadar terus bagi tangki bahan bakar sebuah kereta. Tesis menjelaskan teknik analisis elemen terhingga mulai dari pemilihan jenis jaringan terbaik dan saiznya sehingga pengenalpastian lokasi-lokasi penting yang kritikal bagi tangki bahan bakar sebuah kereta di bawah beban statik. Bahan yang dipilih untuk tangki bahan bakar ini adalah Keluli Tahan Karat AISI S21900. Struktur tiga dimensi model pepejal bagi tangki bahan bakar sebuah kereta ini dimajukan dengan menggunakan perisian lukisan kejuruteraan bantuan komputer. Strategi pengoptimuman pemilihan bagi jaringan terbaik untuk model elemen terhingga dibangunkan. Analisis elemen terhingga kemudian dilakukan dengan menggunakan kod 'MSC.NASTRAN'. Elemen terhingga model struktur tangki bahan bakar sebuah kereta dianalisa dengan menggunakan pendekatan elastik kerkadar terus. Perbandingan jaringan tatarajah antara tetrahedral dengan 4 titik dan tetrahedral dengan 10 titik dibuat dalam banyak aspek, termasuk memperoleh keputusan jumlah titik, deformasi dan tekanan-tekanan. Dari keputusan, didapati bahawa tetrahedral dengan 10 titik memberikan ketepatan yang lebih tinggi dalam banyak aspek daripada tetrahedral dengan 4 titik. Keputusan diperoleh itu kemudian digunakan untuk perbandingan antara tiga jenis tekanan iaitu tekanan prinsipal maksimum , tekanan 'Von Mises' dan tekanan 'Tresca'. Keputusan yang diperolehi menunjukkan bahawa nilai tekanan tidak melebihi had kekuatan tekanan, tetapi deformasi yang kerap akan mengakibatkan kebocoran pada struktur tangki bahan bakar sebuah kereta. Ujian ketahanan yang signifikan membantu untuk memperbaiki sistem pengudaraan tangki bahan bakar sebuah kereta di masa hadapan. Keputusan ini juga akan sangat membantu dalam proses mengurangkan kos, dan meningkatkan kebolehpercayaan produk dan kepercayaan pelanggan seterusnya.

TABLE OF CONTENTS

	Page
SUPERVISOR'S DECLARATION	ii
STUDENT'S DECLARATION	iii
DEDICATION	iv
ACKNOWLEDGEMENTS	v
ABSTRACT	vi
ABSTRAK	vii
TABLE OF CONTENTS	viii
LIST OF TABLES	ix
LIST OF FIGURES	xi
LIST OF SYMBOLS	xiii
LIST OF ABBREVIATIONS	xiv

CHAPTER 1 INTRODUCTION

1.1	Background of the Study	1
1.2	Problem Statement	2
1.3	Objectives	3
1.4	Scope of the Study	3
1.5	Overview of Report	3

CHAPTER 2 LITERATURE REVIEW

2.1	Introduction	4
2.2	Automobile Fuel Tank	5
2.3	Three Dimensional (3D) Structural Model	7
2.4	Finite Element Method (FEM)	7
2.5	Conclusion	10

CHAPTER 3 METHODOLOGY

3.1 Introduction 12
3.2 Project Flow Chart 12
3.3 3D Design 14
3.4 Analysis 16
3.4.1 Linear Static Stress Analysis 19
3.5 Conclusion 20

CHAPTER 4 RESULTS AND DISCUSSION

4.1 Introduction 21
4.2 Mesh Optimization 21
4.3 Displacement 23
4.4 Stress Analysis 26
4.5 Conclusion 35

CHAPTER 5 CONCLUSION AND RECOMMENDATIONS

5.1 Introduction 36
5.2 Conclusions 37
5.3 Recommendations for the Future Research 37

REFERENCES 38
APPENDIX (IF ANY) 40

LIST OF TABLES

Table No.	Title	Page
3.1	Tetrahedral mesh configuration	18
3.2	AISI Type S21900 Stainless Steel Properties	20
4.1	Nodes with variable global edge length	22
4.2	Displacement with variable global edge length	24
4.3	von Mises stress with variable global edge length	28
4.4	Maximum Principal Stress with variable global edge length	30
4.5	Tresca Stress with variable global edge length	32
4.6	Stresses with variable pressure	34

LIST OF FIGURES

Figure No.	Title	Page
1.1	Typical leakage of an automobile fuel tank	2
2.1	Basic configurations of an automobile fuel tank system	6
3.1	Project methodology flow chart	13
3.2	Typical automobile fuel tank	14
3.3	Fuel tank 3D model designed using SolidWorks	15
3.4	Technical drawing with dimensions	16
3.5	Basic finite element analysis flow chart	16
3.6	Tetrahedral with 10 nodes; Global Edge Length = 5 mm	18
3.7	Tetrahedral with 10 nodes; Global Edge Length = 10 mm	19
3.8	Loading and boundary conditions of the automobile fuel tank	20
4.1	Mesh for tetrahedral with 10 nodes with 5 mm global edge length	22
4.2	Mesh for tetrahedral with 10 nodes with 10 mm global edge length	23
4.3	Displacement for tetrahedral with 4 nodes with 4 mm global edge length	25
4.4	Displacement for tetrahedral with 10 nodes with 4 mm global edge length	25
4.5	Various displacement with different global edge length	26
4.6	Von Mises stress for tetrahedral with 4 nodes with 5 mm global edge length	27
4.7	Von Mises stress for tetrahedral with 10 nodes with 5 mm global edge length	27
4.8	Von Mises Stress with variable global edge length	28
4.9	Maximum Principal Stress for tetrahedral with 4 nodes with 5 mm global edge length	29

4.10	Maximum Principal Stress for tetrahedral with 10 nodes with 5 mm global edge length	29
4.11	Max Principal Stress with variable global edge length	30
4.12	Tresca Stress for tetrahedral with 4 nodes with 5 mm global edge length	31
4.13	Tresca Stress for tetrahedral with 10 nodes with 5 mm global edge length	31
4.14	Tresca Stress with variable global edge length	32
4.15	Different Stresses with variable global edge length	33
4.16	Different Stresses with variable pressure	34

LIST OF SYMBOLS

E	Modulus of elasticity
σ	Total Stress
ε	Total strain
ρ	Density
v	Poisson Ratio

LIST OF ABBREVIATIONS

CAD	Computer Aided Design
CAE	Computer-Aided Engineering
CPU	Computer Processing Unit
FEM	Finite Element Method
FEA	Finite Element Analysis
FTP	Fuel Tank Pressure
NIST	National Institute of Standards and Technology
3D	Three dimensional

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND OF THE STUDY

The amount of fuel vapor is ordinarily created in the fuel tank while the vehicle is being refueled, parked, or driven. Typical vehicle user surely not really familiar about the state of violent mixture of air and fuel can cause internal pressure increment during refueling of an automobile fuel tank. Under other circumstances especially like during the car crash the automobile fuel tank can cause explosion that is too risky for the passengers and the pedestrians. Companies in automotive sector like various car manufacturers, suppliers of tank systems, operators of filling stations, manufacturers of filling stations, and supplier of the fuel itself are involved, it is inevitable to conduct the technical discussion publicly. The real thing that happens during refueling process is the multiphase or multi-component flow enters the tank where the liquid falls to the bottom of the tank and the gasses are fulfilling the vapor space of the automobile fuel tank causing the additional pressure inside the fuel tank (Fackrell et al., 2003). Although the maximum internal pressure of the automobile fuel tank is permitted for 0.5 bars when pressurized, there is no guarantee that the pressure is not exceeding that level.

Finite element method (FEM) is applied while study the stress analysis of an automobile fuel tank. Hence, mesh determination is 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).

1.2 PROBLEM STATEMENT

A pressure strength test for the purpose of which is to check that, at a pressure with a defined safety margin in relation to the maximum allowable pressure, the fuel tank does not exhibit significant leaks or deformation exceeding a determined threshold. The test pressure must be determined on the basis of differences between the values of the geometrical and material characteristics measures under test conditions and values used for design purposes. Filling the fuel tank through fill pipe causes the level of fuel in the tank to rise, displacing and pressurizing air and fuel vapor contained in the tank or introduced during the filling process. When the engine is running and filler pipe is closed, surplus fuel heated by its proximity to the engine while in fuel pump is returned via surplus line. The return of this heated fuel into the fuel tank also increases the internal vapor pressure of the tank. Vent valve bleeds air and fuel vapor from the tank to reduce the internal tank pressure, thereby ensuring that the internal tank pressure does not reach an unsafe pressure point. However, when the level of the liquid fuel in the tank nears or reaches valve by tilt or slosh, the valve closes to prevent dangerous leaks of fuel from the tank to the carbon canister. Repeated fuel leaks from the automobile fuel could cause fuel spillage on the ground. The typical leakage of an automobile fuel tank is shown in Figure 1.1.

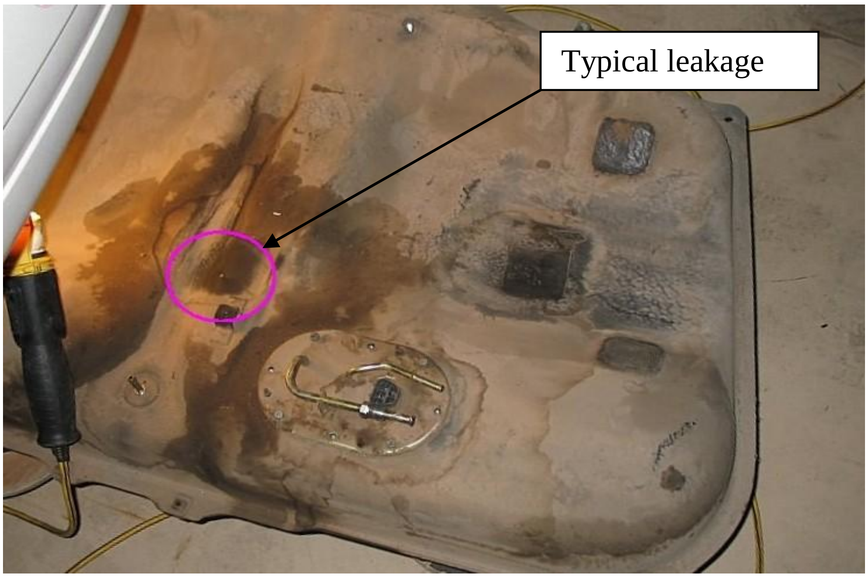


Figure1.1: Typical leakage of an automobile fuel tank

1.3 OBJECTIVES

There are three main objectives that were highlighted according to the field of study for the study on the mesh optimization for structural analysis of an automobile fuel tank:

- To design 3D model of fuel tank using SolidWorks software.
- To develop a finite element model.
- To carry out the linear static stress analysis of an automobile fuel tank.

1.4 SCOPE OF THE STUDY

In this area of study SolidWorks software package is used to design the 3D model of an automobile fuel tank. The term fuel tank means that the model consists of the tank, inlet pipe and the venting pipe. However the analysis to be carry out only consider the internal part of the fuel tank which comprising the internal bottom, left internal wall, right internal wall, back internal wall and the front internal wall. These are five face of a solid that is considered in the analysis. The 3D model design needs to consider those parts as they are critically containing the parameter to be studied through the FEA.

1.5 OVERVIEW OF REPORT

Introduction of this thesis covers the basic understanding of the overall picture of studied field of the linear static stress analysis of an automobile fuel tank. Meanwhile, literature review of this thesis covers the revisions on previous work that has been done by other researchers about the related studies field on the linear static stress analysis of an automobile fuel tank. Methodology of this thesis deals with the procedure that has been used from designing to analyzing the overall things of throughout the linear static stress analysis of an automobile fuel tank. Results and discussion will be presenting the findings from the study. Lastly, conclusion and recommendations will conclude the overall project and the recommendations for further study.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

A review of the literature was performed to identify studies relevant to the topic. The main source for the literature search was the current studies and references books that are correlated with the field of study of linear static stress analysis of an automobile fuel tank. A limited number of studies were found thus made the task for linear static stress analysis of an automobile fuel tank using FEM quite challenging and interesting. The urgent need for an elaboration for the linear static stress analysis governing for an automobile fuel tank using FEM has motivated a series of researches for the results.

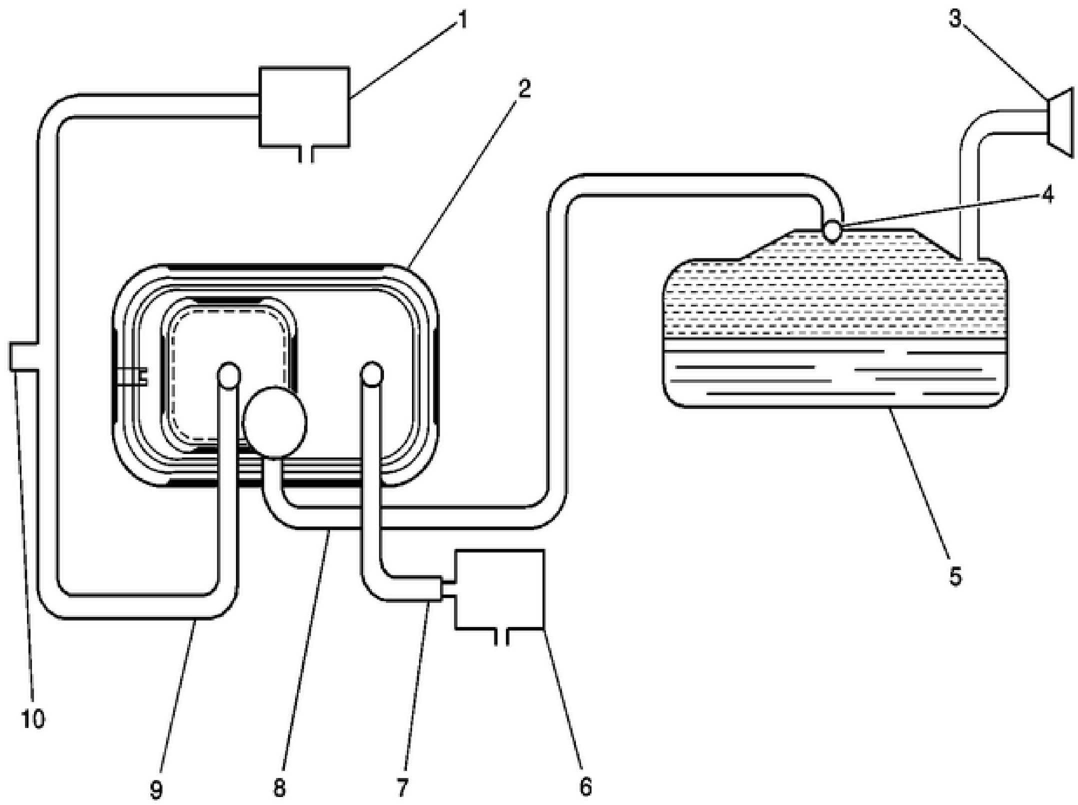
The linear static stress analysis for an automobile fuel tank is undoubtedly critical instead of the fast evolution of automotive technology. Although the evolution of electrical or hybrid car and natural gas vehicle is set to take place, another source of fuel such as bio fuel and hydro fuel ensure the relevancies to study the nature of an automobile fuel tank.

National Institute of Standards and Technology (NIST) notes that it is important to consider which parts of an automobile fuel tank are used to determine its capacity rating and what happens to these components when operating and fuelling a vehicle. The tank's rated capacity does not include the "vapour head space," the uppermost portion of the tank compartment, nor does it include the volume of the filler pipe where fuel enters the vehicle. Drivers, however, sometimes fill the tank beyond the pump's automatic shut-off point, resulting in fuel being drawn into the vehicle's vapour recovery system or filler pipe. Similarly, if the lanes that surround the service station pumps are not level, fuel can shift into the vapour space allowing more fuel to be delivered into the tank.

2.2 AUTOMOBILE FUEL TANK

Fuel is the liquid or gas that initiates and continues the combustion of internal combustion engines whereas the fuel tank is the storage device that store sufficient amount of fuel for the engine required capacity and mileage of the journey. Thus, the automobile fuel tank is one of the most important parts for the continuity of an internal combustion automobile engine. Fuel supply system is one of the major parts in an automobile body. Typically the system consisting four main components which are the filler tube, the fuel tank, the vent tube, and the rollover valve (Fackrell et al., 2003). It is clear that automobile fuel tank can’t be avoided when discussing about fuel supply system.

Commonly the shape of the automobile fuel tank is quadrilateral square. However the actual shape of the automobile fuel tank is base on the manufacturer itself. Rectangular tanks come in standard cost effective sizes and offer the largest capacity for any given space. The uppermost part of the automobile fuel tank containing pressure domes and venting pipe since presence automobile fuel tank is built as an open system. This is to encounter the internal pressure problem and this is to make sense the uppermost part of the automobile fuel tank can be neglected when studying the linear static stress of on the internal surface of an automobile fuel tank. The basic configuration of an automobile fuel tank system is shown in Figure 2.1.



1. Evaporator Canister Purge Solenoid Valve
2. Evaporator Canister
3. Fuel Fill Neck/Fill Cap
4. Rollover Valve/Fuel Tank Pressure (FTP) Sensor
5. Fuel Tank
6. Evaporator Canister Vent Solenoid Valve
7. Vent Hose/Pipe
8. Evaporator Vapor Pipe
9. Evaporator Purge Pipe
10. Evaporator Service Port

Figure 2.1: Basic configurations of an automobile fuel tank system

Source: Chevy & GMC Truck, SUV, Crossover and Van Forums 2009

2.3 THREE DIMENSIONAL STRUCTURAL MODEL

A structural model can be defined as an assembly of structural members or elements that is interconnected at the boundaries such as surfaces, lines and joints. Thus, a structural model consists of three basic components namely, structural members, joints (nodes, connecting edges or surfaces) and boundary conditions. In the extent of linear static stress analysis of an automobile fuel tank using FEM, a 3D structural model is undoubtedly the important things. Its shape and dimension are the most influential characteristic in determining the further meshing criteria for the analysis purposes.

The FEM is necessarily required the development of a 3D model of the structural model. This procedure is quite critical since it is affect the final analysis result of the 3D structural model. Structural model analysis is an activity of analyzing a structural system in order to predict the effect of the real structural model under the excitation of expected loading and external environment during the service life of the structural model. The purpose of a structural model analysis is to validate the adequacy of the design from the view point of safety and serviceability of the structural model (Kanok-Nukulchai, 2002).

2.4 FINITE ELEMENT METHOD

Recently vehicle components and systems development processes are highly depending on computer-aided engineering (CAE) analysis. Commonly, the CAE analysis provides quick and accurate assessment of newly designed vehicle components and systems in terms of their manufacturability and targeted performance. Also, the analysis helps to improve the design by virtually optimizing design parameters to achieve the most optimum design requirements. With the implementation of the CAE analysis, the amount of expensive and time-consuming physical tests is greatly reduced nowadays. However the extension of CAE still has its limitations and some undiscovered paths in analysis. Exemplary problems are events and processes involving severe deformation, material separation, fluid–solid interaction, phase changing and other complex physics (Wang, et. al., 2009).

In this project the linear static stress analysis is considered**.** This computational process is based on an approximation of domain where the problem is formulated. Thus, a first condition regarding the mesh quality is to make this approximation precise. In fact we want to solve the given problem in domain and not in approximate domain different from the real domain. This being established (that results in conditions about the boundary elements of the mesh leading to good boundary approximation) the mesh quality is related to the solution and thus the nature of the problem under investigation (Frey & George, 1999).

Mesh distortion catalyzed numerical instability is the most hated barrier in automotive sector finite element simulations. Some solutions were recommended such as wrapping elements with null shells and equivalence of meshes have been introduced but none of them seems practical enough to survive various scenarios. FEM have been developed over the past almost twenty years in view of their capabilities in dealing with large material deformation and separation, but have remained in academic research due to their unaffordable high computational cost in solving large-scale industrial applications. FEM allows engineers modeling severe deformation area with the FEM while keeping the remaining area that has been modeled (Wang et. al., 2009).

Linear static stress analysis is often enough for situations in which loads are distinctive and the time or location of maximum stress is evident. Engineers apply static loads (such as forces or pressures) or known "imposed" displacements to a finite element model in a linear static stress analysis. They then add elastic material data, constraints and other information such as the direction of gravity. Static forces are assumed to be constant for an infinite period of time while resulting strain, movement and deformation are small. Engineers assume that the material will not deform beyond its elastic limit and that any resulting dynamic effects from the loading are insignificant (i.e., inertial effects can be neglected), known as mechanical equilibrium. In the other hand, the FEA can become excessively stiff; this results in a severe under or over prediction of the displacements and other analysis objectives. This phenomenon is known as shear-locking. Significantly, it is due to the inability of shear deformable elements to accurately model bending within an element under a state of zero transverse shearing strain (Moleiro et al., 2008).

The FEM is a numerical method that is laterally influenced by some errors due to its particular formulation and implementation. Errors are introduced as the domain is divided into several small (but finite) elements, and polynomials or harmonic functions are used to represent the overall behavior of the calculated quantities. Other sources of errors come from the algorithms used to solve the system equations such as the tolerance used in the balance between internal and external forces, etc.

Elements are connected at points called nodes. Particularly the arrangement of elements is called a mesh. The field quantity is locally approximated over each element by an interpolation formula expressed in terms of the nodal values of the field quantity. The archipelago of elements represents a discrete analog of the original domain and the associated system of algebraic equations represents a numerical analog of the mathematical model of the problem being analyzed (Reddy, 1993). The final value for nodal quantities, when totaled with the assumed field in any given element, completely determines the convergence variation of the field in that element (Cook et al., 2002). These are the two most important concepts of FEA: discretization of the domain and approximation of the field quantity using its nodal values. However, the displacement-based fully compatible FEM has an inherent characteristic known as the overly-stiff phenomenon, especially when linear triangular elements are used (Cui et. al., 2009).

The accuracy of FEA is occasionally fumbled by errors and uncertainties, which may be related to the numerical tool itself such as discretization, element formulation, and equation solver or to the physics of the problem. Model uncertainty, discretization error, parameter uncertainty and rounding error are the common sources of uncertainties and errors in FEA (Cook et al., 2002; Bathe, 1996; Oberkampf et al., 2002; Muhanna and Mullen, 2004). The FEA is usually starts with the selection of a mathematical model to represent the physical system being analyzed. The actual problem is simplified and idealized, and is described by an accepted mathematical formulation such as the theory of elasticity, or thin-plate theory, or equations of heat conduction, and so on. The uncertainty about how well the mathematical model represents the true behavior of the real physical system is termed model uncertainty.

Typical model uncertainties in FEA are:

(i) The idealization of the boundary conditions

(ii) The use of plane model rather than three-dimensional model

(iii) The use of linear model rather than nonlinear model

(iv) The use of time-independent model rather than dynamic model

The established mathematical model is represented by a finite elements discretization. This involves selecting a mesh and elements. The computed solution of the FE model is in general only an approximation of the exact solution of the mathematical model, and the discrepancy is called discretization error. FEA solution is influenced by a variety of factors, such as the number of elements used, the nature of element shape functions, integration rules used and other formulation details of particular elements.

Parameter uncertainty occurs because the precise data needed for the analysis is not available. This type of uncertainty is sometimes called parametric uncertainty or data uncertainty. In FEA, the parameter uncertainty may exist in the geometrical, material or loading data. Parameter uncertainty may result from a lack of knowledge, an inherent variability in the parameters, or both.

Finite precision of the computer arithmetic is the limitation to the accuracy of FEA solution. During arithmetic execution are on for floating point numbers, the exact result will not, in general, be represent able as a floating point number. The exact result will be rounded to the nearest floating point number, and this loss of information is referred to as rounding error.

2.5 CONCLUSION

Conclusively the literature study in accordance with linear static stress analysis of an automobile has been done where several important findings already been found. The elaboration about automobile fuel tank, 3D structural model and finite element method has been able to give the brief understanding of the overall view of this project.