STRESS ANALYSIS ON TRUCK CHASSIS

MOHD HUSAINI BIN ABD WAHAB

A report submitted in fulfillment of the requirements
for the award of the Bachelor of
Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering

UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

ABSTRACT

Truck chassis forms the structural backbone of a commercial vehicle. The main function of the truck chassis is to support the components and load that mounted on it. The chassis is experienced a stress whether when it is moving or in a static condition. This paper presents an analysis of the static stress that acting on the upper surface of the truck chassis. These projects study the distributions of the stress that acting on the chassis. Critical parts that will lead to failure are also observed. The method used in this numerical analysis is finite element analysis (FEA). Finite element analysis helps in accelerating design and development process by minimizing number of physical tests, thereby reducing the cost and time for analysis. The commercial finite element package Algor was used for this simulation. 3-D model of the truck chassis was draw by using SolidWorks before analyzed into the simulation software. Numerical results showed that critical part was at the mounting bracket of the tire and also at the front part of the chassis. Some modifications are also suggested to reduce the stress and to improve the strength of the truck chassis.

ABSTRAK

Casis lori membentuk tulang belakang kepada kenderaan perusahaan ini. Fungsi utama casis lori ialah untuk menyokong komponen-komponen yang dipasangkan ke atasnya. Casis mengalami daya tekanan sama ada ketika kenderaan bergerak atau ketika kenderaan dalam keadaan statik Laporan ini mempersembahkan analisis ke atas casis di mana daya tekanan yang diaplikasikan adalah dalam keadaan statik. Bahagian-bahagian casis yang mengalami kesan daya kritikal juga diperhatikan. Jenis analisis yang akan digunakan di dalam projek ini adalah analisis elemen finit. Analisis elemen finit merupakan cara analisis yang membantu mempercepatkan analisis ke atas rekabentuk dan juga memudahkan analisis yang melbatkan eksperimen. Bagi tujuan analisis elemen finit ini, perisian komerial Algor telah digunakan sebagai medium simulasi. Rekabentuk casis lori tiga dimensi dihasilkan menggunakan perisian SolidWorks sebelum rekabentuk tersebut dianalisis menggunakan perisian simulasi. Daripada keputusan simulasi menunjukkan bahagian kritikal pada casis terletak pada pengikat braket dan juga bahagian hadapan casis. Beberapa modifikasi ke atas casis dicadangkan bagi mengurangkan daya tekanan dan juga bagi memperbaiki ketahanan casis lori

TABLE OF CONTENT

APPENDICES

LIST OF TABLE

LIST OF FIGURES

LIST OF SYMBOLS

Eq.	Equation
F	Force
A	Area
m	mass
a	gravity of acceleration
P	Pressure
δ	displacement
E	Young Modulus
I	Moment of inertia

LIST OF ABBREVIATIONS

2-D	Two Dimensional
3-D	Three Dimensional
CAD	Computational Aided Design
FEA	Finite Element Analysis
FEM	Finite Element Method

CHAPTER 1

INTRODUCTION

1.1 Project Background

Chassis forms the structural backbone of commercial vehicles. The main function of the chassis is to support the components and payload mounted upon it including engine, body, passengers and also luggage. Chassis function's also to maintain the desired relationship between the suspension and steering mechanism mounting points.

When the truck travels along the road, the chassis is subjected to stress, bending moment and vibrations induced by road roughness, weather and components that mounted on it. Stress that acting on chassis is varies with the displacement and each part on the car chassis. Because of the behavior of the chassis that always subjected to stress (moving or not), some of the critical part will collapse.

Computer based numerical stress analysis methods (e.g finite element analysis) have permitted the complex distributions of stress in engineering to be more calculated. These allow linear stress and non-linear stress analysis to be performed for static and dynamic loads. In finite element analysis, behavior of structure is obtained by analyzing the collective behavior of the elements. Algor is one of the software that usually used in engineering field to perform the static analysis. It provided a better solution to analyze impact of load on the chassis body including the critical part which experiences a high value of stress/load on it.

1.2 Project Objectives

There are several objectives regarding the title of stress analysis on chassis, which are:

- To analyze the static load that cause stress forces transmitted to the different part of chassis.
- To compare the finite element analysis result between different static load and different part of the truck chassis.

1.3 Project Scopes

Scopes oft this project are:

1. Draw the 3D model of truck car chassis using Solidwork software.
2. Using Algor software to run stress analysis based on simulation method (e.g finite element analysis).
3. Compare the result analysis by the different load value.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

This review covers the function and types of a chassis, stress analysis, truck chassis description and the analysis using Finite Element Analysis (FEA) method.

2.2 Automotive Chassis

The chassis is the main structural frame of an automobile. It connects all key vehicle components including suspension and drive train. There are five main functions of an automotive chassis; to provide an area for occupants and luggage, offer safety to the occupants and outside parties, provide points for mounting of suspension and drive train, provide a stiff framework linking all mounting .

The vehicle chassis provides an area within or above where occupants can be seated. This area can allow up to three occupants to sit adjacent to each other in up to four rows. Space for luggage is generally provided near the occupant area.

A vehicle chassis provides safety to occupants of the vehicle and outside parties. It provides a means of absorbing energy from frontal, side and rollover impacts. The greater the energy absorbed by the chassis on impact the lower the energy levels transmitted to a vehicles occupants and surroundings, lowering the chances of injury.

2.3 Types of Chassis

This section is a brief overview of the different types of chassis and their advantages and disadvantages.

2.3.1 Ladder frame

Early car chassis design began as ladder frame due to its simplicity, versatility, durability and low development costs. This is the earliest kind of chassis. From the earliest cars until the early 60s, nearly all cars in the world used it as standard. Even in today, most SUVs still employ it. Its construction, indicated by its name, looks like a ladder - two longitudinal rails interconnected by several lateral and cross braces. The longitude members are the main stress member. They deal with the load and also the longitudinal forces caused by acceleration and braking. The lateral and cross members provide resistance to lateral forces and further increase torsion rigidity.

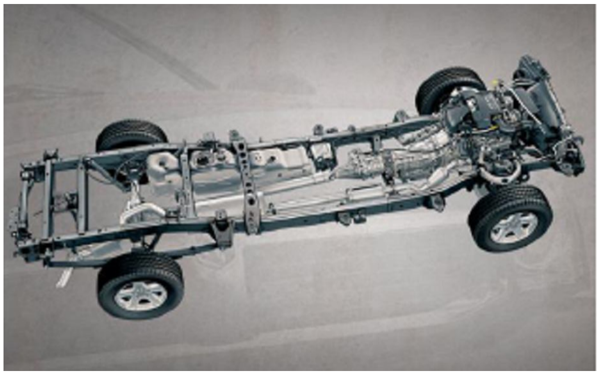

Figure 2.1: Dodge Ram 2500 ladder frame chassis (Chrysler, 2007)

An example of a Dodge ladder frame chassis can be seen in (Figure 2.1). The ladder frame chassis is versatile as it allows virtually any body shape to be placed at top the chassis. This is beneficial because it allows a vehicle manufacturer to produce many different types of vehicle with the same chassis and driveline, lowering development costs.

The ladder frame chassis weaknesses are: it is weak in torsion, it generally has a higher centre of mass (COM), it can be heavy, a crumple zone is not able to be integrated and they usually have higher production costs.

2.3.2 Backbone Chassis

Backbone chassis is an improvement to the ladder frame chassis. The backbone chassis design contains of a large, single, longitudinal structural beam placing at the down vehicle with lateral beams connecting the suspension system. The suspension and motor lateral beams are not mounted on the backbone. This chassis is stiffer than the ladder chassis but not stiff enough for high performance cars due to their higher torsion and performance. Another weakness of this chassis is it didn't have impact protection from the side for the occupants.

Figure 2.2: The Lotus Elan backbone chassis (Wan, 1998-2000)

2.3.3 Space Frame

Space frame is a chassis design that consists many tubes that joined together to create a complex, light and very stiff structure. The complex design leads to a very rigid frame. This design usually incorporates raised the door sills to increase the bending strength and stiffness. The space frame chassis is light due to the minimal amount of structural material. The amount of material that required is minimal because the triangulated shape which makes all beams under tension or compression, not torsion. When the beams not under torsion the cross sectional area can be reduced. This design of chassis also increases in stiffness compared to the ladder frame and backbone chassis causes by adding the three dimensional shape heights to the design.

2.3.4 Monocoque/ Uni-body

A monocoque or uni-body chassis is a chassis that is integrated with the body. The monocoque chassis is the chassis that have been used in modern vehicles nowadays which almost 99% of modern vehicles use monocoque chassis. The monocoque chassis is very complex but cheap to produce has large spaces and is very safe but has rigidity to weight ratio similar to a ladder frame chassis.

The monocoque chassis is very complex design. The complexity of the chassis is due to the integration with the body shell where the integration makes the costs large as development of the chassis requires considerable time and money. The monocoque, while being cost effective is also efficient at saving space due to the chassis being part of the body shell as there can then be large component free areas within the chassis. The monocoque chassis has a low rigidity to weight ratio due to the large amount of material used but the monocoque has high bending and torsional stiffness This is attributed to the chassis design placing more emphasis on space efficiency than strength. The large amount of material in a steel monocoque chassis is beneficial for crash protection as the use of large amounts of material is offset by the material absorbing more energy through crumple zones in the event of an impact.

Figure 2.3: Monocoque chassis (Jaguar, 2007)

2.4 Finite Element Analysis

The finite element method is a numerical procedure that can be applied to obtain approximate solutions to a variety of problems in engineering. Steady, transient, linear, or nonlinear problems in stress analysis, heat transfer, fluid flow, and electromagnetism problems may be analyzed with the finite element method.

2.4.1 General Stress Analysis

The element stiffness matrix could be formed exactly for truss elements, but this is not the case for general stress analysis situations. The relation between nodal forces and displacements are not known in advance for general two- or three-dimensional stress analysis problems, and an approximate relation must be used in order to write out an element stiffness matrix.

In the usual \displacement formulation" of the finite element method, the governing equations are combined so as to have only displacements appearing as unknowns; this can be done by using the Hookean constitutive equations to replace the stresses in the equilibrium equations by the strains, and then using the kinematics equations to replace the strains by the displacements. This gives Eq. (2.1)

$$LT\_ = LTD\_ = LTDLu = 0 \quad (2.1)$$

Of course, it is often impossible to solve these equations in closed form for the irregular boundary conditions encountered in practical problems. However, the equations are amenable to discretization and solution by numerical techniques such as finite differences or finite elements.

Finite element methods are one of several approximate numerical techniques available for the solution of engineering boundary value problems. Problems in the mechanics of materials often lead to equations of this type, and finite element methods have a number of advantages in handling them. The method is particularly well suited to problems with irregular geometries and boundary conditions, and it can be implemented in general computer codes that can be used for many different problems.

To obtain a numerical solution for the stress analysis problem, let us postulate a function ~u(x; y) as an approximation to u using Eq. (2.2):

$$u(x; y) \sim u(x; y) \qquad (2.2)$$

Many different forms might be adopted for the approximation u. The finite element method discretizes the solution domain into an assemblage of sub regions, or \elements," each of which has its own approximating functions. Specifically, the approximation for the displacement ~u(x; y) within an element is written as a combination of the (as yet unknown) displacements at the nodes belonging to that element as expressed in the Eq. (2.3) below:

$$u(x; y) = Nj(x; y)uj \qquad (2.3)$$

Here the index j ranges over the element's nodes, uj are the nodal displacements, and the Nj are interpolation functions." These interpolation functions are usually simple polynomials (generally linear, quadratic, or occasionally cubic polynomials) that are chosen to become unity at node j and zero at the other element nodes. The interpolation functions can be evaluated at any position within the element by means of standard subroutines, so the approximate displacement at any position within the element can be obtained in terms of the nodal displacements directly
from Eq. 2.3

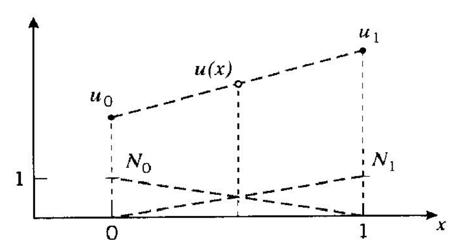

Figure 2.4: Interpolation in One Dimension (Roylance, 2001)

The interpolation concept can be illustrated by asking how we might guess the value of a function u(x) at an arbitrary point x located between two nodes at x = 0 and x = 1, assuming we know somehow the nodal values u(0) and u(1). We might assume that as a reasonable approximation u(x) simply varies linearly between these two values as shown in Fig. 2.4, and write

$$u(x) \_ \sim u(x) = u0\ (1 - x) + u1\ (x) \quad (2.4)$$

or

$$u(x) = u0N0(x) + u1N1(x) \qquad N0(x) = (1-x) \quad (2.5)$$
$$N1(x) = x$$

Here the N0 and N1 are the linear interpolation functions for this one-dimensional approximation. Finite element codes have subroutines that extend this interpolation concept to two and three dimensions.

An unsophisticated description of the FE method is that it involves cutting a structure into several elements (pieces of structure), describing the behavior of each element in a simple way, then reconnecting elements at nodes as if nodes were pins or drops of glue that hold elements together (Figure 2.5). This process results in a set of simultaneous algebraic equations. In stress analysis these equation are equilibrium equations of the nodes. There may be several hundred or several thousand such equations, which mean that computer implementation is mandatory.

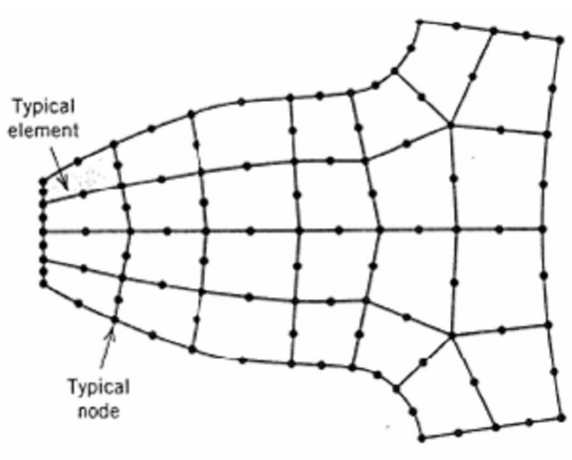

Figure 2.5: A course –mesh, a two-dimensional model of gear tooth. All nodes and elements lie in plane of the paper (Roylance, 2001)

The idea of representing a given domain as a collection of discrete parts is not unique to the finite element method. It was recorded that ancient mathematicians estimated the value of π by noting that the perimeter of a polygon inscribed in the circle approximates the circumference of the latter. They predicted the value of π to accuracies of almost 40 significant digits by representing the circle as a polygon of a finitely large number of sides. In modern times, the idea found a home in aircraft structural analysis, where, for example, wings and fuselages are treated as assemblages of stringers, skins, and shear panels.

In the early 70's, FEA was limited to expensive mainframe computers generally owned by the aeronautics, automotive, defence and nuclear industries. Due to the rapid decline in the cost of computers and the phenomenal increase in computing power, current FEA models have usually much greater number of elements and thus increased precision. Present day supercomputers are now able to produce more accurate results in much shorter amounts of time. At the same time, modern desktop computers can be used to analyze larger and more complex problems than could be done by the mainframes of yesteryear.

FEA systems now have powerful graphics capabilities, automated functionality, and advanced user interfaces that make the technology considerably faster and easier to use. These improvements notwithstanding, however, full-blown advanced FEA still requires considerable time and the expertise of a dedicated analyst with the knowledge necessary to apply proper mesh densities, element types, and boundary conditions. These expert analysts also must know how to go about translating CAD geometry into proper format for building the FEA model as well as correctly interpreting plots and other output information.

One of the driving forces in manufacturing companies is the continuing demand for reduction in product development time and cost to maintain profitability and competitiveness. Over the years, this requirement has prompted organization in a wide range of industries to find different ways of making product development more efficient. Advancement in the entire spectrum of CAD/CAM/CAE tools in particular have automated many design, engineering, and analysis tasks to shorten development cycles, mostly as a labor savings to minimize overhead costs.

The advantage of leveraging CAE is to drive the design to more mature levels by interactions far ahead of hardware or prototype's availability. This virtual development lays the foundation for shortening development time, optimizing the design and reducing cost and weight. Potential risks and failure modes are predicted and prevented much earlier in the development stage. The model elaboration with its respective elements is based in turning a real complex component into a concept mode, determining the geometric dimensioning parameters, stiffness and elasticity modulus. This way to a new development the methodology to be used at each phases will be explained as follows; verify it the new project has the same characteristics of the previous one such as weight of vehicle, distance between axles, classified considering the type of vehicle, whether compact, standard, sedan or van and kind of engine, whether longitudinal or transversal.

Progressive manufacturers are now investigating ways to further reduce design cycle-time by evaluating and changing the product development process itself. The goal here is not so much economic savings in the engineering department but rather a broad business advantage in getting product innovations to customers faster and thereby increasing a company’s market share. FEA allows designers and engineers to quickly iterate back and forth in performing basic conceptual “what-if” studies to evaluate the merit of different ideas, compare alternatives, and filter out design weaknesses before more detailed analysis, prototype testing, and production planning are conducted.

2.5 Stresses on Truck Chassis

As a truck travels along the road, the truck chassis is excited by dynamic forces induced by the road roughness, engine, payload, transmission and more. For the static condition, the truck chassis will only get a stress from its own weight, which is includes curb weight, trailer, and also the load that it will carrying. Under such various force, the truck chassis tends to cracks if exceeded its limit. The weight of the chassis will also cause high stress concentrations at certain locations, fatigue of the structure, loosening of mechanical joints, and creation of noise and vehicle discomfort. To solve these problems, study on the truck chassis static characteristics is thus essential.

The static characteristics of the truck chassis are also important for the mounting of components on the chassis. The existing truck chassis is normally emphasized on the strength of the structure to support the loading placed upon it. The dynamic aspects are normally not treated seriously. The mode shapes of the truck chassis at certain stress are very important to determine the mounting point of the components like engine, suspension, transmission and more.

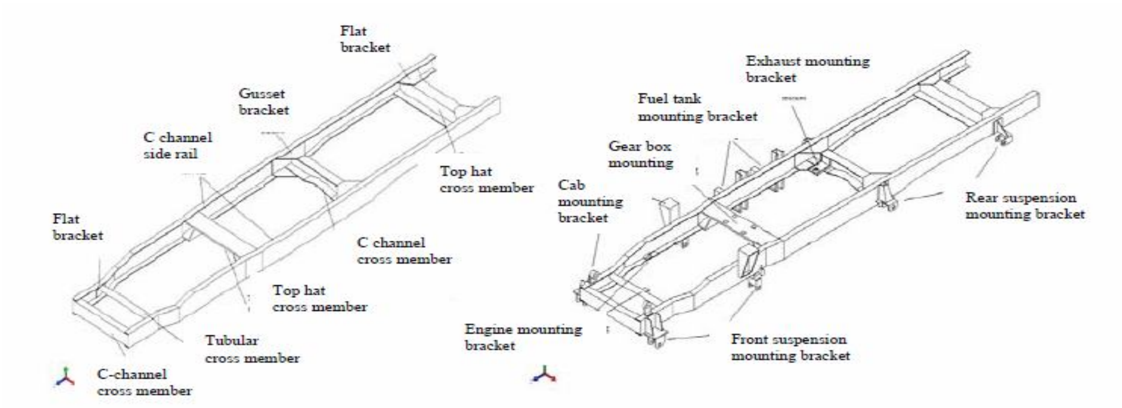

Figure 2.6: Truck chassis ladder frame (Roslan, 2007)

During the last decades, several methods have been developed for the solution of the dynamics of flexible multi-body systems. Very often, simple and merely academic examples have been studied in order to show the performance of the different formulations. However, the solution of complex and realistic mechanical systems should be addressed to achieve a more meaningful validation from the industrial point of view. A suitable way to such a validation may be the comparison between measured and calculated stresses in components of the multi-body system as the motion takes place. In fact, the knowledge of stresses becomes essential when dealing with mechanical design.

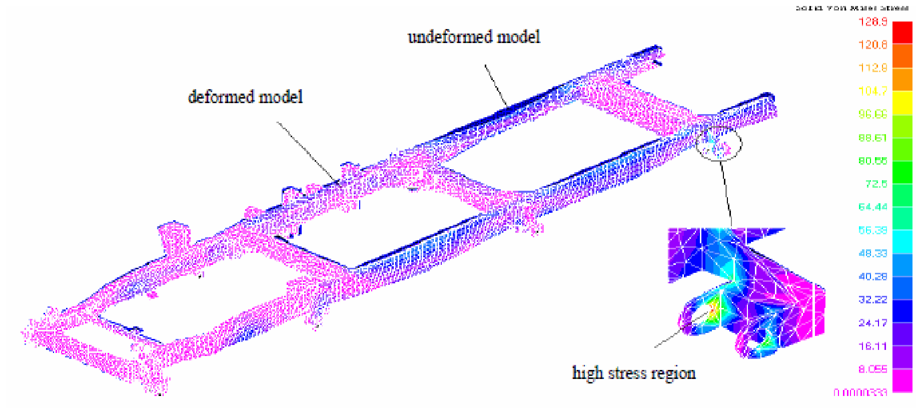

Figure 2.7: Stress Contour and Deformation Pattern of the Chassis under Truck Components Loading (Roslan, 2007)

The loads associated with fatigue occur many times during the life time. Fatigue damage caused by slow but gradual microstructure degradation may occur under cyclic loading, even if the maximum stress is lower than the material yield. Loads, geometry, material and manufacture process plus environmental conditions determine system/subsystem/component durability. The factors interact with each other and make the durability design a very complex and challenging task. In order to reduce the product development time, reduce prototypes, optimize weight and costs, the automotive makers have established their systematic durability design and verification process.

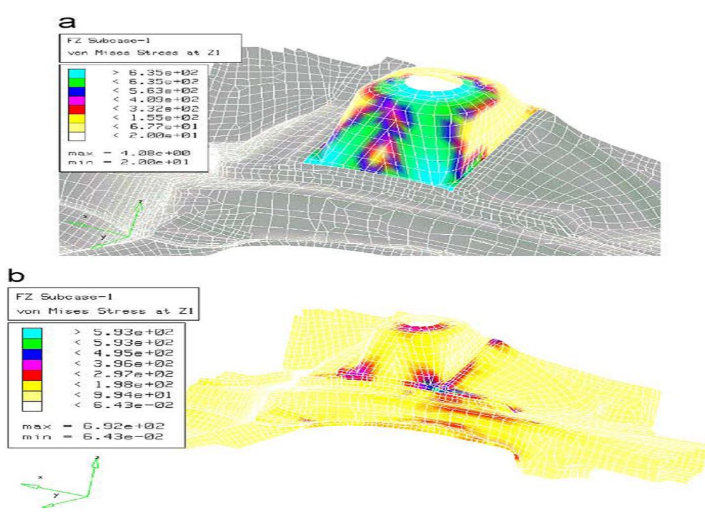

Figure 2.8: Stress Analysis for Chassis Part (Chang, 2008)