STRUCTURAL CHARACTERISTICS OF CHASSIS MODEL FOR A TECHNOLOGY DEMONSTRATOR

TUAN NOORDIYANA BINTI TUAN ISMAIL

Report submitted in fulfillment of the requirements
for the award of the degree of
Bachelor of Manufacturing Engineering

Faculty of Manufacturing Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2013

ABSTRACT

This study presents predictions of structural properties and deformation of vehicle chassis for Automated Guided Vehicle (AGV) that was designed to support at most 15kg work in progress. Finite element analysis involved static loading on vehicle chassis. The scope of this study include identifying location of high stress area and evaluate the torsional stiffness of the chassis. CAD design for the chassis is carried out using Solid works version 2012 and FEA prediction is performed using Comsol multiphysics version 4.2. The result show that the high concentration stress likely to occur at the joints and specific area. The magnitude the stresses depend on the material properties and size of the chassis. High yield strength material will produce less deformation and more rigid chassis structure.

ABSTRAK

Kajian ini bertujuan untuk menentukan sifat-sifat struktur dan perubahan bentuk yang berlaku pada kerangka kenderaan “Automated Guided Vehicle (AGV)” yang direka untuk menampung beban sekurang-kurangnya 15kg. Analisis unsur terhingga ini melibatkan pergerakan dalam keadaan statik pada kerangka kenderaan. Skop kajian ini adalah untuk mengenal pasti lokasi yang mempunyai tekanan tinggi dan juga menilai kekukuhan nilai kilasan kerangka tersebut. Reka bentuk CAD untuk kerangka ini menggunakan Solidworks versi 2012 dan analisis unsur tidak terhingga ini dilakukan menggunakan perisian *Comsol multiphysics versi 4.2*. Hasil dari analisis ini, ia menunjukkan bahawa tekanan yang tinggi tertumpu pada bahagian yang bersambung. Magnitud tekanan pula bergantung kepada sifat-sifat bahan dan saiz kerangka. Bahan yang mempunyai sifat kekuatan yang tinggi akan menghasilkan kurang perubahan dan struktur kerangka lebih tegar.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

1.1 Project Background

Chassis forms the structural backbone of a vehicles. For a passenger car, the main function of the chassis is to rigidly support the car 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 a vehicle 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), a weak of structurally designed part will collapse.

Computer based numerical stress analysis methods such as finite element analysis have permitted the complex distributions of stress in engineering to be more deliberate. 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. The FEA 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.

Structural analysis comprises a set of physical laws and mathematics predicts the behavior of structures and to evaluate structural ability of the structure to withstand the loads. The structural analysis provide the deformations, internal forces, and stresses in the structure. To perform an accurate analysis a structural engineer must determine such information as structural loads, geometry, support conditions, and materials properties. The results of such an analysis typically include support reactions, stresses and displacements. This information is then compared to a criteria that prescribed the conditions of failure.

1.2 Objective

The general objective of the study is to build an automated guided vehicle. The specific of objectives are :

- To compute the structural characteristics of finite element model for vehicle chassis.
- To evaluate the structural behavior of the chassis when subjected to static loadings.

1.3 Scope of Study

- Structural analysis for static loading up to 60kg for assuming linear elastic modulus for vehicle chassis.

1.4 Problem Statement

During the development of chassis model, the analysis of each chassis element need to be considered so that the magnitude of internal stress and displacement can be comprehend. The details of type of material used also must consider in order determining the strength of the chassis.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

This review covers the function and types of a chassis, truck chassis description, material selection and the Computer Aided Design approach to evaluate the structural characteristics of the vehicle chassis.

2.2 Definition of a Chassis

According to Jones, 1975, the chassis is the framework support a vehicle and its load. In a modern vehicle, the chassis provide mounting points for the suspensions, the steering mechanism, the engine and gearbox, the final drive, the fuel tank and the seating for the occupants. It also provide rigidity for accurate handling and protect the occupants against external impact.

In addition, the chassis should be light enough to reduce inertia and offer satisfactory performance. It should also be tough enough to resist fatigue loads that are produced due to interaction between the driver, engine, power transmission and road conditions.

2.3 Vehicle Chassis Research

Callister, W. D. (2006) developed a Multi-Body Dynamic Model of the Tractor-Semitrailer for ride quality predictions. The studies involved representing the distributed mass and elasticity of the vehicle structures the non-linear behavior of shock absorbers, reproduce the fundamental system dynamics that influence ride and provide output of the acceleration, velocity and displacement measures needed to compute ride quality. The study proposed a predictive tool, ADAMS multi-body dynamics model for evaluating the ride quality design. The model includes frame, cab and model generated from finite element component mode synthesis. Second, the construction and correlation of the model has been developed and followed a multi-step process in which each of the major sub-systems were developed and validated to test results prior to corporation in the full vehicle model. Finally, after a series of refinements to the model, the next steps were implemented to obtain an acceptable degree of correlation. The author had managed to evaluate the model's ability to predict ride quality by using accelerations measured in the component, which were then processed through an algorithm to compute an overall ride comfort rating.

Ibrahim, et.al. 2009, had conducted a study on the effect of frame flexibility on the ride vibration of trucks. The aim of the study was to analyze the vehicle dynamic responses to external factors. The driver acceleration response has been weighted according to the ISO ride comfort techniques was found that the excessive levels of vibration in commercial vehicles were due to excitation from the road irregularities which led to ride discomfort, ride safety problems, road holding problems and to cargo damage or destruction. Also, it has been found that the frame structure vibrations due to flexibility have a similar deleterious effect on the vehicle dynamic behavior.

One of case study conducted by Linton (n.d) from Cranfield University on the chassis supplied by Luego Sports Cars Ltd stated that, the initial torsion value was calculated about 1330 Nm/deg. from a mass of 120.1 kg. There were many types of adjustment and modification made to the chassis to improve the torsion stiffness values such as the additional cross bar structures, improvement of the structure material and many others. As a result, the overall torsion stiffness has been significantly increased to

337% of its initial value. Therefore, a significant changed on modification has resulted the overall chassis structure has tremendously improved and enhanced the chassis performance such as ride quality, vibration and etc.

The history of the ladder frame chassis dates back to the times of the horse drawn carriage. It was used for the construction of 'body on chassis' vehicles, which meant a separately constructed body was mounted on a rolling chassis. The chassis consisted of two parallel beams mounted down each side of the car where the front and rear axles were leaf sprung beam axles. The beams were mainly channeled sections with lateral cross members, hence the name. The main factor influencing the design was resistance to bending but there was no consideration of torsion stiffness.

A ladder frame acts as a grillage structure with the beams resisting the shear forces and bending loads. Ladder frames were used in car construction until the 1950's but in racing only until the mid 1930's. A typical ladder frame shown at figure

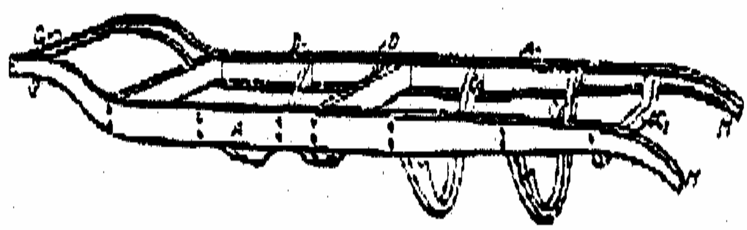

Figure 2.1 : Ladder frame chassis

To increase the torsion stiffness of the ladder chassis cruciform bracing was added in the 1930's. The torque in the chassis was retrained by placing the cruciform members in bending, although the connections between the beams and the cruciform must be rigid. . J. Vidosic 2007, recommends some value of safety factor for various condition of loading and material of structures. He recommends the value of 1.5 to 2 for well known materials under reasonably environmental condition, subjected to loads and stresses that can be readily.

2.4 Finite Element Analysis

The finite element analysis (FEA) used numerical method or often known as finite element method (FEM) that can be applied to approximate solution for an engineering problem. The approximate solution is obtained by idealized a product model by splitting it into as many small discrete pieces called finite elements or more commonly known as elements, which are connected by nodes. This dividing process is known as mesh generation. Each of the generated elements has exact equations that define how it reacts to certain load. Hence, accuracy of the solution can be increased by refining the mesh generation.

Ibrahim, et.al. 2009, had conducted a study on the frame flexibility, the author had came out with the truck frame modeled using the Finite Element Method (FEM) and its modal properties have been calculated. Numerical results were presented for the truck, including power spectral densities and root mean square values of the vehicle dynamic response variables. The results show that there was good agreement with the experimental analysis and that modeling technique was a very powerful and economical for the analysis of complex vehicle structures. From the comparison of the responses of the rigid and flexible body models it has been found that the frame flexibility strongly affects the accelerations of both driver and truck body. Therefore, the author suggested that the frame flexibility effects were taken into account in the design of primary, cab and engine suspension systems.

Another case study was presented by Romulo Rossi Pinto Filho, who analyzed on the Automotive Frame Optimization. The objective of his study was basically to obtain an optimized chassis design for an off-road vehicle with the appropriate dynamic and structural behavior. The studies were consisted of three main steps. Firstly, the modeling of the chassis used in a commercial off-road vehicle using commercial software based on the finite elements method (FEM). Secondly, a series of testing were conducted to obtain information for modeling and validation. Finally, the validated model allowed the optimization of the structure seeking for higher torsion stiffness and maintenance of the total structure mass.

Other than that, Zaman (n.d) has conducted a study on the application of dynamic correlation and model updating techniques. These techniques were used to develop a better refinement model of existing truck chassis with approximately 1 tone and also for verification of the FEA models of truck chassis. The dynamic characteristics of truck chassis such as natural frequency and mode shape were determined using finite element method. From the initial result, both analysis show that bending mode for 2the truck chassis experienced 1st torsion mode for 1st natural frequency, 1st bending mode for 2nd natural frequency, 2nd torsion mode for 3rd natural frequency and 2nd bending mode for 4th natural frequency. In addition to measuring the overall stiffness of a chassis, the author also had recommended that the fixture also could be used to measure the deflection distribution along the length of a chassis. Using several additional dial indicators located at key locations, the fixture could determine sections of the chassis that deflect more than others. Therefore, the chassis could be strengthened in those areas to increase the overall torsion stiffness.

Finally, the used of twist fixture also could be used to validate finite element models. Several models have been developed to predict the torsion stiffness of the chassis as well as the roll stiffness of the combined suspension and chassis system. The use of twist fixture on a chassis that has been measured for a finite element model will be very beneficial in ensuring that the models were accurate.

Krishna (n.d) has presented his study on the Chassis Cross-Member Design Using Shape Optimization. The problem with the original chassis was that the fundamental frequency was only marginally higher than the maximum operating frequency of the transmission and drive shaft, which were mounted on these cross-members. The aim of this testing was to raise the cross-member frequency as high as possible (up to 190-200 Hz) so that there was no resonance and resulting fatigue damaged. The Finite Element Model (FEM) of the frame with the #3 cross member was shown in figure. The frame was completely fixed at the four corners of the side rails as shown. A modal analysis performed on this model indicated that the first natural frequency was about 179 Hz. Firstly, a sizing optimization was attempted which indicated that the mass was a predominant factor. Four additional holes were added to the sides of the cross-member to reduce its mass. Another tests also have been conducted which the holes on the sides had to be expanded, the bottom holes to be reduced in size, the thickness of the attachment bracket to be increased etc. Based on those testing, the fundamental frequency of the cross-member was raised by about 4 Hz, resulting in a better design.

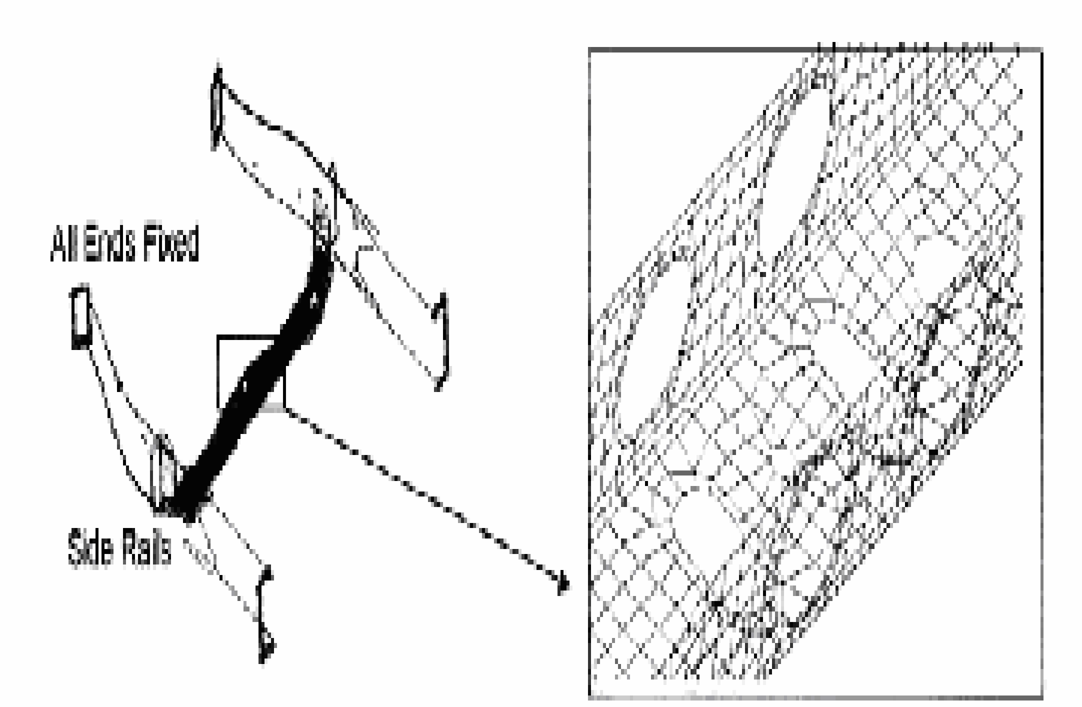

Figure 2.2 : Finite Element Model of #3 Cross-member

On the other hands, the author had analyzed the cross-member in order to obtain the global torsion value of truck chassis. This value was utilized as a reference for future study in order to improve ride ability and conformability during operation. From the literature study, it has been found both theories and practices found that the torsion affect was more severe load case rather than bending load.

The ladder frame chassis became obsolete in the mid 1930's with the advent of all-round independent suspension, pioneered by Mercedes Benz and Auto Union. The suspension was unable to operate effectively due to the lack of torsion stiffness. The ladder frame was modified to overcome these failings by making the side rails deeper and boxing them. A closed section has approximately one thousand times the torsion stiffness of an open section. Mercedes initially chose rectangular section, later switching to oval section, which has high torsion stiffness and high bending stiffness due to increased section depth, while Auto Union used tubular section. The original Mercedes design was further improved by mounting the cross members through the side rails and welding on both sides. The efficiency of twin tube chassis' is usually low due to the weight of the large tubes. They were still in use into the 1950's, the 1958 Lister-Jaguar being an example of this type. A typical twin-tube chassis is shown in following figure.

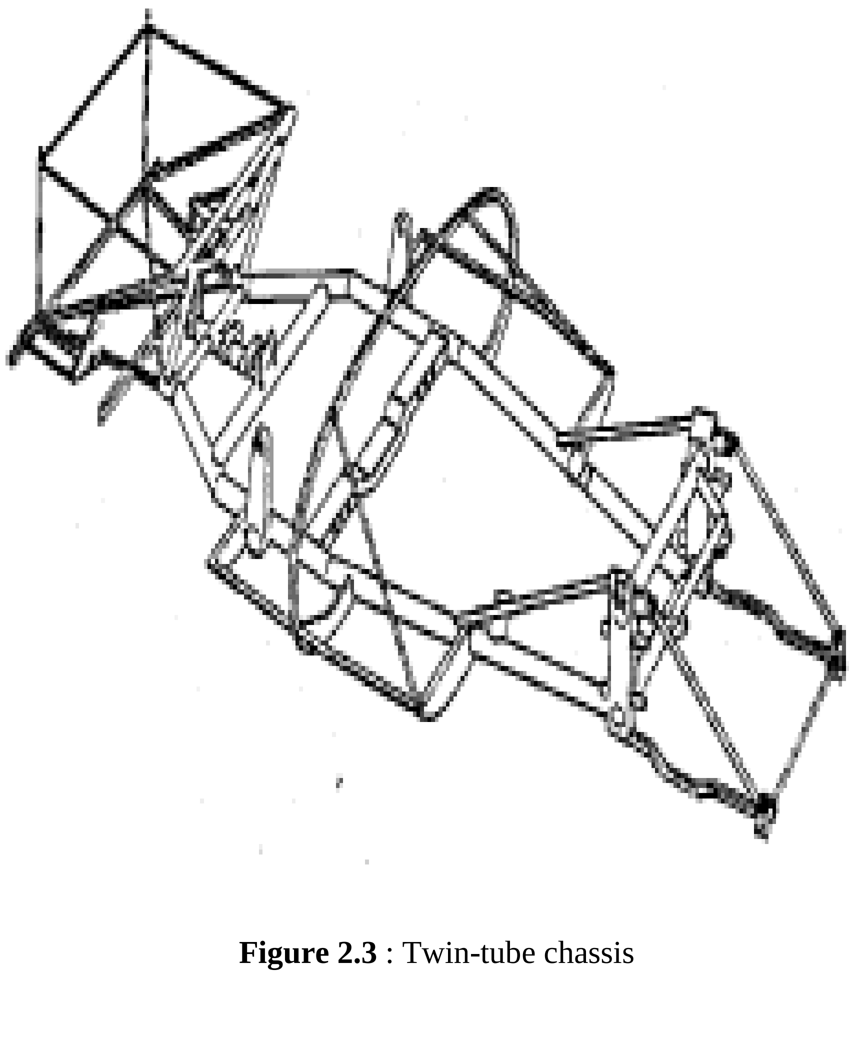

Figure 2.3 : Twin-tube chassis

2.5 Aluminum

Aluminum is normally used for robotic approach for its resistance to corrosion and its light weight. Aluminum is a soft, lightweight metal with normally a dull silvery appearance caused by a thin layer of oxidation that forms quickly when the metal is exposed to air. Aluminum oxide has a higher melting point than pure aluminum. Aluminum is nontoxic (as the metal), nonmagnetic, and non-sparking. It has a tensile strength of about 49 MPa in a pure state and 400 MPa as an alloy. Aluminum it is malleable, ductile, and easily machined and cast. It has excellent corrosion resistance and durability because of the protective oxide layer.

This metal is used in many industries to manufacture a large variety of products and is very important to the world economy. Structural components made from aluminum and its alloys are vital to the aerospace industry and very important in other areas of transportation and building. Aluminum has a density around one third that of steel and is used advantageously in applications where high strength and low weight are required. This includes vehicles where low mass results in greater load capacity and reduced fuel contraption. When the surface of aluminum metal is exposed to air, it will produce a protective oxide coating forms almost instantaneously. This oxide layer is corrosion resistant and can be further enhanced with surface treatments such as anodizing.

Aluminum is not only non-toxic but also does not release any odors or taint products with which it is in contact. This makes aluminum suitable for use in packaging for sensitive products such as food or pharmaceuticals where aluminum foil is used.

CHAPTER 3

RESEARCH METHODOLOGY

3.1 Overall Methodology

Methodology gives the brief idea to what the method that has been adopted throughout the project. The research methodology flowchart for this project was shown in figure 3.1. The literature review of the vehicle chassis was carried out to obtain basic understanding of the project. The Computer Aided Design (CAD) modeling was performed on the chassis model by using Catia software. Then, the modeled chassis was completely transformed into the finite element software for engineering analysis. The finite element torsional analysis were also performed. The objective of these tests were to find the torsion stiffness of the structure and the response of the applied load at different loading condition.

3.2 Process Flow Chart

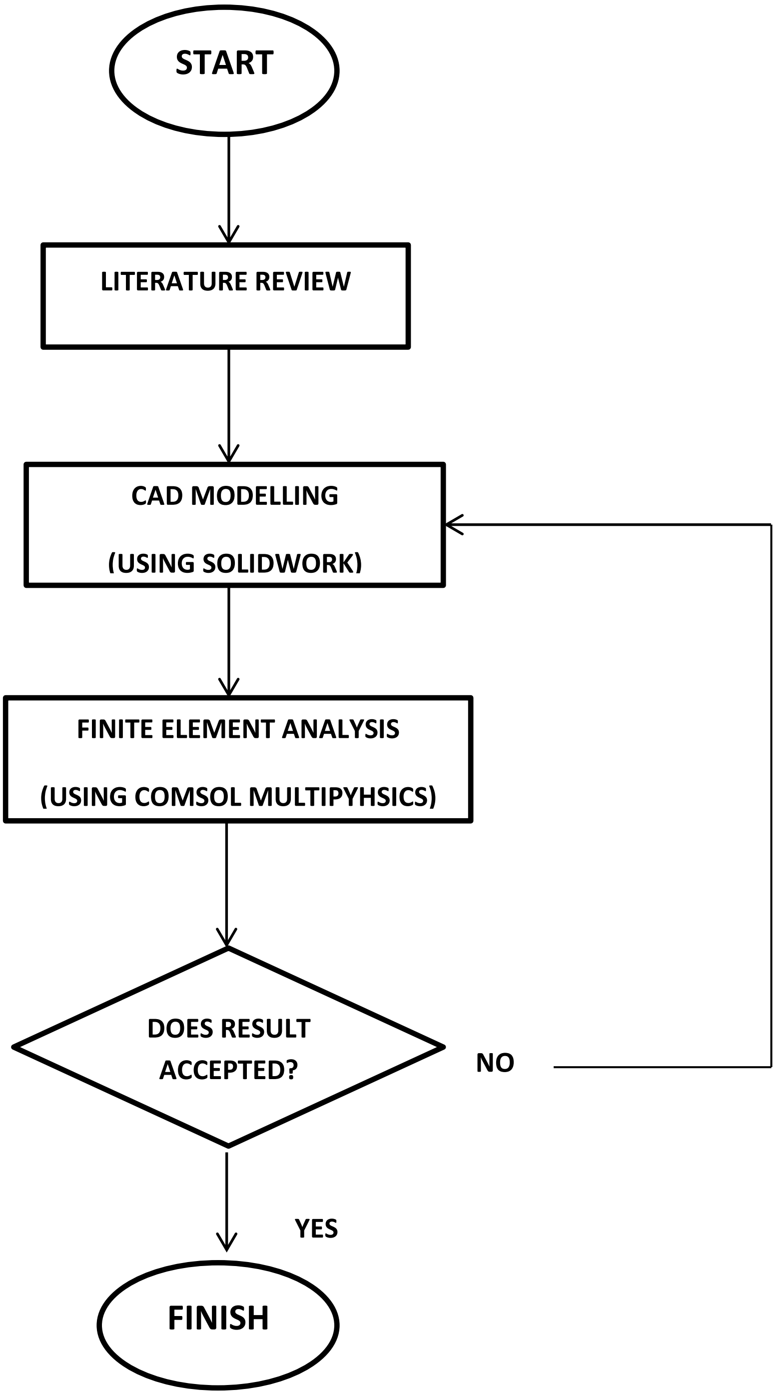

Figure 3.1

In this project, Finite Element Analysis was used to determine the characteristics of the chassis. For the purpose of this study, the chassis model was modeled using

Catia software. The model was then imported into a commercial Finite Element software (Comsol multipyhsics).

The chassis model was modeled using CAD software. The CAD model may need to be remodel if the results of FEA do not meet with the predetermined requirements. In this phase, all the required detailed engineering drawings for manufacturing prototype of the chassis model are prepared. The FEA will compromise structural analysis such as torsional stiffness and bending deflection simulations where the behaviors of the designed chassis model under different conditions are reviewed. The severity of any undesirable results will be assessed and any necessary modification on the design will be made accordingly.

3.3 Pre Processing Stage

The Solid Mechanics interface, through its equations, describes the motion and deformation of solid objects in a 2- or 3-dimensional spatial frame and positions in the frame are identified by lowercase spatial coordinate variables x, y, and z.

By default, the solid mechanics interface uses the calculated displacement and equation below to define the difference between spatial coordinates **x** and material coordinates **X**. This means the material coordinates relate to the original geometry, while the spatial coordinates are solution dependent. For example coordinate of element due to deformation.

$$x = x(X,t) = X + u(X,t)$$

Figure 3.2 : Equation 1.1

3.4 Type of Finite Element

As mentioned in the previous part, finite elements are often just called elements. Basically dimensional (1D), two-dimensional (2D) and three-dimensional (3D) elements are the three most common elements, where the typical general idealization geometry for each type of elements is illustrated in figure below.

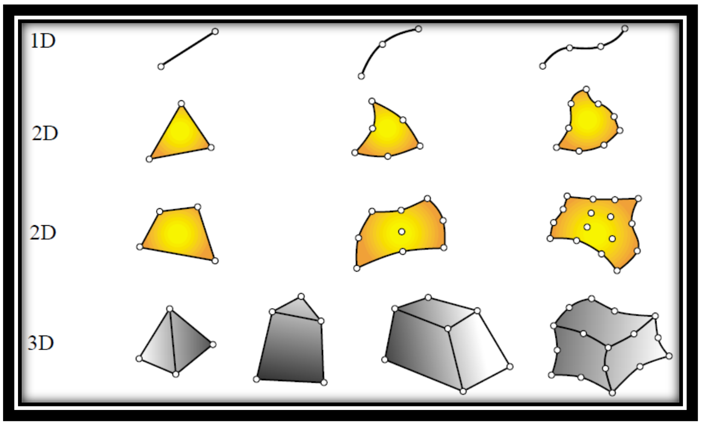

Figure 3.3 :Typical finite element geometries in one through three dimensions.(Felippa, 2010)

3.5 Governing Equation

The formulation used for structural analysis in COMSOL Multiphysics for both small and finite deformations is totally Lagrangian. This means that the computed stress and deformation state is always referred to the material configuration, rather than to current position in space. Material properties are always given for material particles and with tensor components referring to a coordinate system based on the material frame. This has the obvious advantage that spatially varying material properties can be evaluated just once for the initial material configuration and do not change as the solid