CHAPTER 2

REVIEW OF LITERATURE

2.1 INTRODUCTION

Primarily driven by the development of powerful and expensive and inexpensive computers, the field of computer aided design and computer aided engineering emerged. It provides predictive tools as well as insights into complex engineering processes. Hence, engineers working in many different application areas demand numerical solution tools for their investigations which some years were only accessible through experiments. In many cases modeling of engineering problems leads in many cases to ordinary and partial differential equations which often are of nonlinear nature. A powerful tool to solve these differential equations is the finite element method is a powerful tool for to solve these, which was developed over the last 50 years.

2.2 RESEARCH ON AUTOMOTIVE CHASSIS

There are two main objectives, with respect to the development of truck chassis. Firstly, the appropriate static characteristics and fatigue life of the existing chassis have to be determined. Secondly, there are many factors involve and must take into account, which can affect on the vehicle rolling, handling, ride stability and fatigue life etc.

Today, there are many research and development programs available in the market especially by the international truck manufacturers, which are very much related to this research work. Therefore, there are
several technical papers from the 'Society of Automotive Engineering' (SAE) and some other sources which have been reviewed.

2.2.1 Overall Discussion on Research

Many of the truck structures found in the research are subject to internal and external loads, which affected the ride quality of vehicle. These loads and behavior can be determined through a series of processes namely, Modal Analysis, Finite Element bending and Torsion Analysis. Besides that, the correlation and modal updating technique are also important to create a good model for further analysis. From the global torsion analysis, it has been found that the torsion load is more severe than bending load. In order to overcome this problem, a cross bar and material selection are important to be considered during design stage. Furthermore, the overall achievement is mainly to reduce the vibration level, so that the life of the chassis structure and performance can be maximized.

The idea of a chassis carries several different connotations, depending upon the referenced source. For the purpose of this work, the chassis will be thought of in its racing context as a structure which carries and connects all of the major components including the body, cabin, power, and other vehicle systems. The chassis structure must safely support the weight of the vehicle components and transmit loads that result from longitudinal, lateral, and vertical accelerations that are experienced in a racing environment without failure. There are many aspects to consider when designing a chassis, including component packaging (including the individual systems), material selection, strength, stiffness and weight.

The primary objective of the chassis is to provide a structure that connects the front and rear suspension without excessive deflection. When considering a automotive chassis, a frame that is easily twisted will result in
significant stability problems related to stability. Suspension setup is based upon the assumption that all four corners of the vehicle are connected by an infinitely rigid body. If the chassis is not sufficiently stiff, the structure is merely another variable in the system as it acts as a torsional spring and damper. This variability within the chassis structure would very difficult to dial in a suspension setup that will generate the necessary levels of lateral grip to be competitive. Generally speaking, a chassis which is able to resist torsional loads resulting from inertial accelerations of components experienced during cornering or from applied loads acting on one or opposite corners (ie. Torsion) of the vehicle will almost always be sufficiently strong.

2.2.2 Definition and Types of Chassis

The chassis is the framework to which is everything in a vehicle is attached. In a modern vehicle, it is expected to fulfill the following functions 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, provide rigidity for accurate handling, protect the occupants against external impact. For a chassis to fulfill these functions, it should be light enough to reduce inertia and offer satisfactory performance. It also should also be tough enough to resist fatigue loads produced due to interaction between the driver, engine, power transmission and road conditions.

2.2.2.1 Ladder frame

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
The beams were mainly channeled sections with lateral cross members, hence the name. The main factor influencing the design was resistance to bending. However, there of torsion stiffness was not considered.

A ladder frame acts as a grillage structure with the beams resisting the shear forces and bending loads. To increase the torsion stiffness of the ladder chassis, cruciform bracing was added in the 1930’s. The torque in the chassis was restrained by placing the cruciform members in bending, although the connections between the beams and the cruciform are rigid. Ladder frames were used in car construction until the 1950’s but in racing only until the mid 1930’s.

### 2.2.2.2 Twin tube

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 rectangular section and later switching to oval section, which has high torsion stiffness and high bending stiffness due to increased section depth, while Auto union. 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.

### 2.2.2.3 Space frame

Although the space frame demonstrated a logical development of the four-tube chassis, the space frame differs in several key areas and offers
enormous advantages over its predecessors. A space frame is one in which many straight tubes are arranged so that the loads experienced all act in either tension or compression. This is a major advantage, since none of the tubes are subject to a bending load. Since space frames are inherently stiff in torsion, very little material is needed so they can be lightweight. The growing realization of the need for improves chassis torsion stiffness of a chassis. During the II World War led to the space frame, or a variation of it. It becomes universal among European road in 1952.

2.3 FINITE ELEMENT METHOD

Finite Element Analysis (FEA), also known as the finite element method (FEM) is based on the concept that a structure can be simulated by the mechanical behavior of a spring, in which the applied force is proportional to the displacement of the spring and the relationship \( F = ku \) is satisfied. In FEA, structures are modeled by a CAD program and represented by nodes and elements. The mechanical behavior of each of these elements is similar to a mechanical spring, obeying the equation, \( F = ku \). Generally, a structure is divided into several hundred elements, generating a very large number of equations which can only be solved with the help of a computer. The term ‘finite element’ stems from the procedure in which a structure is divided into small but finite size elements (as opposed to an infinite size, generally used in mathematical integration). The endpoints or corner points of the element are called nodes. Each element possesses its own geometric and elastic properties. Spring, Truss, and Beams elements, called line elements, are usually divided into small sections with nodes at each end. The cross-section shape doesn’t affect the behavior of a line element; only the cross-sectional constants are relevant and used in calculations. Thus, a square or a circular cross-section of a truss member will yield exactly the same results as long as the cross-sectional area is the same. Plane and solid elements require more than two nodes and can have over 8 nodes for a 3 dimensional element. A line element
has an exact theoretical solution, e.g., truss and beam elements are governed by their respective theories of deflection and the equations of deflection can be found in an engineering text or handbook. However, engineering structures that have stress concentration points e.g., structures with holes and other discontinuities do not have a theoretical solution, and the exact stress distribution can only be found by an experimental method. However, the finite element method can provide a more acceptable efficient solution more efficiently. Problems of this type call for use of elements other than the line elements mentioned earlier, and the real power of the finite element is manifested. In order to develop an understanding of the FEA procedure, we will first deal with the spring element. In this chapter explained how spring structures are used as building blocks for developing an understanding of the finite element analysis procedure. Both spring and truss elements give an easier modeling overview of the finite element analysis procedure; this is because of that each spring and truss element, regardless of length, is an ideally sized element and does not need any further division.

Finite element analysis, also called the finite element method, is a method for numerical solution of field problems. A field problem requires the determination of spatial distribution of one or more dependent variables. Mathematically, a field problem is described by differential equations or by an integral expression. Either description may be used to formulate finite elements which can be visualized as small pieces of a structure. The elements are connected at points called 'Nodes'. The assemblage of elements is called finite element model or structure. The particular arrangement of elements is called a mesh. Numerically, a finite element mesh is represented by a system of algebraic equations to be solved for unknowns at nodes. Finite element modeling is the process of preparing a FE model. It decides about the significant features of the actual problem that can be incorporated in the model. Simulation is the prediction of the intended output results of the computational FE model.
2.3.1 Basic Concept of FEM

The finite element method (FEM) is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and specific conditions on the boundary of the domain.

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

2.3.2 FEM Steps

There are certain common steps in formulating a finite element analysis of a physical problem, whether structural, fluid flow, heat transfer and some others problem. These steps are usually embodied in commercial finite element software packages. There are three main steps, namely: preprocessing, solution and post processing. The preprocessing (model definition) step is critical. A perfectly computed finite element solution is of absolutely no value if it corresponds to the wrong problem. This step includes: define the geometric domain of the problem, the element type(s) to be used, the material properties of the elements, the geometric properties of the elements (length, area, and the like), the element connectivity (mesh the model), the physical constraints (boundary conditions) and the loadings.
The next step is solution, in this step the governing algebraic equations in matrix form and computes the unknown values of the primary field variable(s) are assembled. The computed results are then used by back substitution to determine additional, derived variables, such as reaction forces, element stresses and heat flow. Actually the features in this step such as matrix manipulation, numerical integration and equation solving are carried out automatically through commercial software.

The final step is post processing during which the analysis and evaluation of the result is done. Examples of operations that can be accomplished include sort element stresses in order of magnitude, check equilibrium, calculate factors of safety, plot deformed structural shape, animate dynamic model behavior and produce color-coded temperature plots. The large software has a preprocessor and postprocessor to accompany the analysis portion and the both processor can communicate with the other large programs. Specific procedures of pre and post are depends upon the program.

### 2.3.3 FE Equations of Chassis

There are several different formulations of FEM involving variational calculus and weighted residuals. The variational formulation is based on Green’s Identity.

$$u^e(x) = N_1^e u_1^e + N_2^e u_2^e = N_1^e N_2^e \begin{bmatrix} u_1^e \\ u_2^e \end{bmatrix} = N^e u^e$$

where

- \( N \) - Shape Function
- \( U(x) \) - Interpolation Function
- \( u \) - Displacement function
2.4 FINITE ELEMENT ANALYSIS OF TRUCK CHASSIS

An important aspect of chassis design and analysis is the stress distribution and fatigue life of prediction process. Since, the chassis design and analysis involves design calculations and modeling, it is meshed with solid and link elements. The finite element analysis was performed using hyper mesh. The chassis rail structure and bumper are meshed with solid elements. The suspension mounting points are meshed with link elements. Thus, the chassis is meshed to have totally around 81679 nodes and 230104 elements. The stress distribution of the chassis under the bending and torsion loads. Chassis analysis mainly consists of static analysis to predict stress distribution and subsequently, the fatigue simulation to predict the life of the chassis.

2.4.1 Computation of Chassis Design

The main aim of the chassis design is capture the static stress and fatigue life of the chassis. As is needed for any finite element analysis, the Governing differential Equation (2.1) provided by equation of equilibrium

\[
[K] U = F \tag{2.1}
\]

where, \( [K] \) is the square matrix,

\( U \) is the vector of unknown value;

\( F \) is the vector of known concept.

Equation (2.1) is the basic finite element equation. The finite element method is used for the solution of static equilibrium of chassis. In order to formulate the problem, the structural deformation and stresses of a chassis can be calculated matrix method. The initial and boundary conditions
need to be specified. The appropriate initial condition would be displacement in the chassis applications.

The some of the essential elements of the design and analysis of chassis

![Diagram of 1-D, 2-D, and 3-D elements]

**Figure 2.1 Elements**

- **Tetrahedron:**
  - Linear (4 nodes)
  - Quadratic (10 nodes)

- **Hexahedron (brick):**
  - Linear (8 nodes)
  - Quadratic (20 nodes)

- **Penta:**
  - Linear (6 nodes)
  - Quadratic (15 nodes)

**Figure 2.2 Three dimensional elements**
Many of the early research works in chassis design and analysis were limited to the computation of stress distributions and fatigue life in the chassis with many assumptions.

Miner (1945) explained fatigue damage during the crack initiation phase. Damage during the initiation phase can be related to dislocations, slip bands, micro cracks, etc. Since these phenomena can only be measured in a highly controlled laboratory environment, most damage summation approaches for the initiation phase are empirical in nature. These methods relate damage to the expended life for a small laboratory specimen. For this purpose, life is defined as the separation of a specimen, which is equivalent to the formation of a small crack in a large component or structure.

Gurney (1976) studied about the analyses carried out in this work were restricted to results which had been obtained for K butt joints under axial loading and transverse non-load-carrying fillet welds under both axial and bending loads. However, by far the greatest amount of data examined was that relating to as-welded transverse fillet welds under axial loading. In all cases the thickness range considered did not extend beyond 10-26mm.

Tanaka et al (1981) studied about the stress analysis of a truck chassis with riveted joints was performed by using FEM. The commercial finite element package ANSYS version 5.3 was used for the solution of the problem. Determination of the stresses of a truck chassis before manufacturing is important due to the design improvement. In order to reduce the magnitude of stress near the riveted joint of the chassis frame, side member thickness, connection plate thickness and connection plate length were varied. Numerical results showed that stresses on the side member can be reduced by increasing the side member thickness locally. If the thickness change is not possible, increasing the connection plate length could be a good alternative.
Beermann et al (1984) described vertical as well as horizontal and torsion static and dynamic loads that act on chassis frames. The torsion behavior of most commercial vehicle frames is dominated by warping torsions, because warping is inhibited in the joints where the cross-members are attached to the side-members. This paper presents a hybrid method of analysis, which combines finite element idealization of the joint areas with analytically derived beam elements for the cross-member and side-member sections. The beam element includes warping torsion force displacement relationships. The flexibility of the joints is included together with the compatibility of their displacements. The method gives close agreement with experimental results.

McCullough et al. (1986) represented spatial dynamics of high mobility track vehicle suspension systems is derived. Using results from a companion paper [1], the equations of motion for a suspension system with an arbitrary number of road wheels are systematically derived. Track is represented as a complex internal force element that acts between ground, wheels, and the chassis of the vehicle. Track tension is computed from a relaxed catenaries relationship and track bridging effects are modeled. Numerical results for driver acceleration and absorbed power, as well as track tension have been presented. A factor of 90 decreases in compute time is achieved over a comparable multi body model of the same vehicle.

Koyanagi et al (1989) discussed the first tilting train in regular public service was the electric multiple unit train operated by Japanese National Railways this technology was not fully implemented world-wide as the marginally increased curve speeds did not justify the extra expense and technology in many cases. Active tilting is the mechanism most widely used today.
Conle et al (1991) described an analytical study of the fatigue life of automobile chassis components using automotive proving ground load history results combined with recent computational advances. This work advances knowledge in two ways: a vehicle dynamics model is used to generate the history of the load vectors acting on the components and the element stress equivalency procedure used until now is improved. It can be concluded that the combination of vehicle dynamics modeling, finite-element analysis and fatigue analysis is a viable technique for the design of automotive components. However, before our durability process can be suitable for applied engineering work a number of improvements are required, which has been outlined.

2.4.2 Existing Research Work on Vehicle Chassis Finite Element Analysis

Thompson and Vissert (1991) presents an overview of structural design methods for mine haulage roads. Through an analysis and quantification of the structural performance of existing pavements they recommend the mechanistic design procedure, together with a revised structural design and associated limiting design criteria. It can be concluded that the application of the mechanistic method to a design project can reduce the costs of haulage roads construction and accrue additional benefit in terms of reduced operating and maintenance costs. Load of 2086 kN In comparison, public road authorities permit a legal maximum dual-wheel axle loading of 80 kN, which is similar in magnitude to that associated with a 25 t truck with tandem rear axles. Large mine haulage trucks impose axle loads ranging from 110 to 170 t, which are applied to haulage roads that have been, at best, designed empirically on the premise of satisfactory.

Wannenburg (1993) described an overall philosophy to generate intelligence from automotive warranty data and provides an approach supported by examples of situations, where analyzing warranty data led to additional insight in predicting organizational risk and driving earlier action.
The method has proven successful in detecting an early transition to a field durability issue, the prediction of warranty risk from a diluted “quality spill.” This process has been employed for some time and has been further enhanced by considering the dual nature of automotive warranty risk related to time and usage, involving a conditional probability analysis.

Fung and Smart et al (1994) investigated joints using the finite element method. Initially a single lap joint has been modelled as a ‘stepped plate’ and the results for the stress concentration factor are found to be in reasonable agreement with published data. However, the stress concentration for this joint occurred at a point away from the point of failure of a riveted joint.

Ibrahim et al (1994) 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 spectral analysis technique was used. From the author’s point of view, the excessive levels of vibration in commercial vehicles were due to excitation from the road irregularities, which led to ride and comfort problems. In order to study the frame flexibility, the author had came out with the truck frame modeled using (FEM) and its modal properties have been calculated. The results there were found to be good agreement with the experimental analysis and the modeling technique proved to be a very powerful one.

John Crawford (1994) discussed racecar may be achieved by tailoring chassis stiffness so that roll stiffness between sprung and unsprung masses are due almost entirely to the suspension. In this work, the effects of overall chassis flexibility on roll stiffness and wheel camber response were determined using a finite element model. To validate the model, the change in wheel loads due to an applied jacking force that rolls the chassis agreed closely with measured data. The roll stiffness predicted from finite element models of the front and rear suspension compare closely to those calculated using a rigid body.
2.4.2.1 Formulation of vehicle chassis design

Keiner and Henning (1995), understood the influence of the various structural members on the torsional stiffness of a NASCAR Winston Cup race car chassis. In this work, we identify the sensitivity of individual structural members on the torsional stiffness of a baseline chassis. A high sensitivity value indicates a strong influence on the torsional stiffness and of the overall chassis. Results from the sensitivity analysis are used as a guide to modify the baseline chassis with the goal of increased torsional stiffness and with a minimum increase in weight and low center-of gravity placement. The torsional stiffness of the chassis with various combinations of added members in the front clip area, engine bay, roof area, front window and the area behind the roll cage were predicted using finite element analysis. Twist angle and the rate of change in twist angle under torsion is calculated at several loc.

Elbeheiry and Karnopp (1996) described the design of active vehicle suspensions with integral constraints to control the response of a vehicle traversing a road is considered. The problem was initially formulated in the linear quadratic regulatory framework with full-state feedback. Alternate formulations based on optimal output feedback and the minimum norm criterion approaches in the absence of complete state information are then presented. A general expression for the required optimal value of the control force based on easily measurable feedback quantities was developed.

Zehsaz et al (1996) explained that automotive chassis is an important part of an automobile. The chassis serves as a framework for supporting the body and different parts of the automobile. Also, it should be rigid enough to withstand the shock, twist, vibration and other stresses. Along with strength, an important consideration in chassis design is to have adequate bending stiffness for better handling characteristics. Hence, strength and stiffness are two important criteria for the design of the chassis. Structural
systems like the chassis can be easily analyzed using the finite element techniques.

Ryu et al (1997) studied an automotive industry to employ multi-parameter strain-life methods. Combination with dynamic finite element analysis. Based on very extensive measurement exercises. Analytical fatigue life assessments, which is then verified through intensive durability testing. The expense and complexity of this approach makes its application impractical for low volume “special” vehicles, a fatigue equivalent static load methodology for the numerical durability assessment of heavy vehicle structures is presented, where fatigue load requirements are derived from measurements as quasi-static g-loads, the responses to which are considered as stress ranges applied a said number of times during the lifetime of the structure.

Raju and Srikanth (1998) discussed about the design and Analysis of a Winston Cup Stock Car Chassis for Torsional Stiffness using the Finite Element method. Roll stiffness between sprung and unsprung masses. The application of the method is demonstrated using two case studies, namely a road tanker and a load haul dumper. In both cases, it was possible to obtain adequately accurate fatigue life prediction results, using simplified loading, static finite element analyses and a stress-life approach to fatigue damage calculations, with material properties available in design codes.

Lonny L. Thomson et al (1998) presented his paper on the twist fixture which can measure the torsion stiffness of the truck chassis. The fixture was relatively lightweight and portable with an ability to be transported and set-up by one person. The extensive testing was carried out to check on the accuracy of the fixture and was found to be within 6% accuracy. Using the twist fixture design, the author managed to test the several chassis of different manufacturers. These tests were performed to compare the
stiffness values of the different chassis. The results leveled that the uncertainty and standard error were below 5%. Due to this uncertainty in the measured data, small changes in stiffness be measured reliably with the fixture. In addition to measuring the overall stiffness of a chassis, the author has also recommended that the fixture also could be used to measure the deflection distribution along the length of a chassis. By 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 should be strengthened in those areas to increase the overall torsion stiffness.

Mothiram K. Patil and Palanichamy (1998) described about the tractor-occupant system is modelled as a lumped parameter system. The composite model is analyzed by computer simulation for vertical vibration responses for a new type of seat suspension. It is was demonstrate that the new tractor seat suspension system (by proper selection of parameters) drastically improves the tolerance to high-intensity vibrations, in the range 0.5–11-Hz range, experienced by tractor occupants, by reducing the maximum (i) amplitude ratios and relative displacements of the body parts to 0.029 and 0.19 mm, respectively, and (ii) body parts “acceleration levels’ to much below the ISO specified 7-h “exposure limit” curve.

Heyes (1998) revealed that the connecting rod is one of the most important parts of an automotive engine. The connecting rod is subjected to a complex state of loading. High compressive and tensile loads are due to the combustion and connecting rod’s mass of inertia respectively. It is the connecting rod should be able to withstand tremendous load and transmit a great deal of power smoothly. The objective of this paper is to investigate the failure analysis of the connecting rod of the automotive engine. The materials including carbon steel, mild steel, bass and aluminum are considered in this study. The linear static analysis was carried out utilizing the finite element
analysis codes. The numerical results were verified with the experimental results. It can be seen from the acquired results that the carbon steel gives good results in terms of hardness and endurance limit were compared to other materials.

Murali M.R. Krishna (1998) explained on the Chassis Cross-Member Design Using Shape Optimization. The problem with the original chassis was that the 17 fundamental frequencies 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. 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.

Lampert and Jon (1998) studied the torsional stiffness of a vehicle's chassis significantly affects its handling characteristics and is therefore an important parameter to measure. In this work a new twist fixture apparatus designed to measure the torsional stiffness of a Winston Cup series race car chassis is described. The twist fixture is relatively light weight, adjustable, and easily transportable by one person for quick set-up on different chassis. Measured values of torsional stiffness are reported for several different chassis. The fixture applies vertical displacements (using linear, jack-screw actuators) at the front spring perches of the chassis while holding the rear perches fixed. Conventional race car scales located under the front assembly
measure the resulting reaction forces due to the displacements. Dial indicators are placed at selected locations along the chassis to measure deflections. Using the dial indicator readings, the measured reaction forces and the chassis geometry, the torsional stiffness of the chassis can be calculated. Ball-joint connections between the twist fixture and chassis have been carefully designed to minimize unwanted rotational restraints.

### 2.4.2.2 Behaviour of fatigue life of chassis

Sumi et al (1998) studied about the fitness for serviceability of structural members of marine structures in which fatigue cracks might be found during in-service inspection is investigated in order to prevent instantaneous failures of ships, as well as a loss of serviceability such as the oil- and/or watertightness of critical compartments. The essential features of fatigue crack propagation and the remaining life assessment are discussed in the first part of the paper, where the effects of weldment, complicated stress distributions including stress biaxialities at three-dimensional structural joints, structural redundancy, and crack curving are found to be of primary importance.

Conle and Chu (1998), reviewed literature on fatigue analysis of welded joints mainly consisting papers and books published during the past 10–15 years. After a short introduction, the different approaches for fatigue analyses are covered, i.e. the nominal stress approach, the structural or hot-spot stress approach, the notch stress and notch intensity approach, the notch strain approach. The crack propagation approach. Only seam-welded joints are considered, and not the behavior of spot-welds, which is a very special.eld. Due to the vast amount of relevant literature, some specific areas are left for other reviews or only touched, i.e. fatigue testing and evaluation, fatigue loading and variable amplitude effects, environmental effects and fatigue reliability.
Karaoglu and Kuralay (2000) explained mechanical behavior of a semi-heavy truck chassis. The ANSYS Code and the stress distributions in the chassis have been obtained and examined. For this purpose, the FE model of the truck vehicle was using 3D shell elements. To validate the FE model of the chassis; firstly, experimental modal analysis has been used. Secondly, the modes of chassis-vibration, natural frequencies and modal shapes have been obtained from the FE analysis and were compared with the results of experimental modal analysis. The dynamic forces due to the unevenness of the road have been calculated using a simple 3D dynamic model of the truck body. Then, stress analysis for the truck chassis have been carried out under static and dynamic loads. Different types of joints and their thickness in the chassis of truck vehicles are one of the important parameters which have significant effect on their strength. To study the effect of the connecting plates on the strength of the chassis, the strength of the welded and also the combined welded-riveted joints have been analyzed with three different plate thicknesses: 5, 8, 12 mm. The results show that the amount of stresses in chassis and connection plates decrease significantly with an increase in the thickness of connection plates. Also, it has been shown that the use of combined welded-riveted joints reduces the stress level of the chassis. The results prove the precision of the FE modeling and also show that the numerical modeling is accurate Therefore, the stresses which have been obtained are reliable and can be used to design the chassis.

Srivatsan and Anand (2000) studied the cyclic strain resistance and fracture behavior of an oxide dispersion strengthened copper alloy. The alloy was cyclically deformed over a range of strain amplitudes giving lives of less than $10^4$ cycles to failure. The specimens were cycled using tension-compression loading under total strain control. The alloy displayed combinations of cyclic hardening and softening to failure. The observed cyclic hardening and softening behavior is a mechanical effect and can be
attributed to concurrent and competing influences of dislocation accumulation and dislocation-microstructure interactions. The alloy followed the Coffin-Manson relationship and exhibited a single slope for the variation of cyclic plastic strain amplitude with reversals to failure. Heat treatment was observed to improve cyclic ductility. The low-cycle fatigue properties and fracture behavior of the alloy are discussed in terms of competing and synergistic influences of cyclic plastic strain amplitude, response stress, intrinsic microstructural effects and macroscopic aspects of fracture.

Dave Anderson and Greg Schade (2001) developed a Multi-Body Dynamic Model of the Tractor-Semi trailer for ride quality prediction. The studies involved representing the distributed mass distribution of vehicle structures. There were three main factors were contributed in the study. Firstly, the model includes frame and other sub systems. Second, the construction and correlation of the model has been developed by FE steps and implemented to obtain an acceptable degree of correlation.

Deweer et al (2001) explained how numerical models as basis to assemble or modify all kind of new structures is increasing over the last year. It has the benefit reduces the number of expensive, physical prototypes. These numerical models however must be verified and validated against measured data. Updating is generally needed to guarantee accurate correspondence with reality.

Cicek Karao et al (2001) investigated and developed stress analysis of a truck chassis with riveted joints was performed by using FEM. The commercial finite element package ANSYS version 11.0 was used for the solution of the problem. Determining the stresses of a truck chassis before manufacturing is important due to improvement in design. In order to achieve a reduction in the magnitude of stress near the riveted joint of the chassis frame, side member thickness, connection plate thickness and connection
plate length were varied. Numerical results showed that stresses on the side member can be reduced by increasing the side member thickness locally. If the thickness change is not possible, increasing the connection plate length may be a good alternative.

Fermer and Svensson (2001) studied the mechanical finite element analyses of welded structures performed on a daily basis in the automotive industry. One objective is to estimate the fatigue strength, which is given mainly by the strength of the joints. This article give some insight into the dimensioning process, with special focus on fatigue analysis of spot welds and seam welds in thin-walled car body structures made of steel. Experiences from daily use at Volvo Car Corporation, limitations of methods, future and on-going work are discussed.

Dong et al (2001) discussed the structural stress approach, which considers the stress increase due to the structural configuration, allows the fatigue strength assessment of welded ship structures with various geometries on the basis of an S–N curve depending only on the type of weld. However, a unique definition and the numerical calculation of the structural stresses are problematic, which has resulted in the development of different variants of the approach. These are discussed and compared with each other work.

Berger et al (2002) explained a vehicle steering knuckle undergoes time varying loadings its service life and fatigue behavior performance evaluation. Static and cycling loading behavior were calculated and analyzed. The non-linear force-strain behavior close to the weld toe was not expected and caused a large amount of work with the FE-model. Since also the manufacturing tolerances were shown to influence the measurements, the verification of the FE-model became quite hard to accomplish. The influence of different parameters such as weld geometry, defects, misalignments, pretension forces and installation accuracy of the gauges should be studied further.
Wolfgang Fricke (2002) reviewed about the literature on fatigue analysis of welded joints is reviewed, considering mainly papers and books published during the past 10–15 years. After a short introduction, the different approaches for fatigue analyses are covered, i.e. the nominal stress approach, the structural or hot-spot stress approach, the notch stress and notch intensity approach, the notch strain approach and finally the crack propagation approach. Only seam-welded joints are considered, and not the behavior of spot-welds, which is a very special field. Due to the vast amount of relevant literature, some specific areas are left for other reviews or only touched, i.e. fatigue testing and evaluation, fatigue loading and variable amplitude effects, environmental effects and fatigue reliability.

Hiaba et al (2002) discussed the structural optimization based on fatigue life of dynamically loaded a structure of realistic complexity is rarely attempted due to computational costs. Very efficient stress analysis and fatigue life assessment techniques are needed if this is to become routine. For the first time, this paper compares several approaches to fatigue life prediction using a real automotive engineering case study; taking into account that optimization based on fatigue life requires accurate relative distribution rather than exact values. The paper concludes that although both the quasi-static and frequency domain approaches are potentially more efficient than transient dynamic analysis, parameter sensitivity of the frequency domain approach may preclude its eventual use.

Pettersson (2002) explained regarding the accuracy of life predictions based on finite element analyses and four different fatigue design methods. Different fatigue design codes, such as nominal stress, geometrical stress, notch stress and linear elastic fracture mechanics are compared regarding work effort and analysis accuracy. The paper also describes the process of developing and verifying a finite element (FE) model as well as
other practical work such as load data acquisition, static strain measurements and full-scale fatigue tests under spectrum loading. The component that was used was a stay from an articulated hauler. The results from the fatigue tests showed that all cracks started from relatively large root defects and that a large scatter of the weld dimensions can be expected when the welds are manually performed. The comparison between the different fatigue design codes showed a large scatter in the estimated fatigue life. The verification of the FE-model showed some interesting and in some cases unexpected results.

Ferry et al (2002) describes vehicle on a computer with a multi-body dynamics simulation software package and to merge that work with physical proving ground and laboratory tests in order to shorten vehicle development time. The intention is to mirror as closely as possible the behavior of a physical vehicle in order to assist in determining its durability characteristics under varying road conditions. This modeling work is important because, if done with sufficient fidelity, it can be used in order to assess vehicle responses by using different suspension components or payloads. Also, potential issues associated with vehicle structure, suspension components or payload positioning can be observed on a computer prior to performing physical tests. The process has the potential to reduce vehicle development cost and time. The virtual dynamic vehicle model has been created by using Automatic dynamic analysis of mechanical systems (ADAMS) software package. The calculated outputs from the model are being compared to force and displacement data collected from actual vehicle on-road testing or a servo-hydraulic road test simulator (RTS). The virtual model can be adjusted until the calculated responses are in close agreement with those of the physical vehicle, thus linking the virtual and real-world results.
Conle et al (2003) discussed the ‘strain-life’ fatigue analysis philosophy has achieved the status of industry standard in the automotive, truck and earth moving industries in North America. Although the fundamental concepts of the method are quite simple, the recent massive computerization of the techniques, and their incorporation into large dynamically loaded body and chassis durability calculation models, has resulted in new capabilities and insights for engineers, but also has meant new challenges for the creators of fatigue simulation software.

Romulo Rossi Pinto Filho (2003) explained the chassis by using steel with closed rectangular profile longitudinal rails and tubular section cross-member. The geometry of the chassis was measured directly in the reference vehicle real structure. Then, a modal analysis procedure was accomplished on the real chassis and a finite element model structure was formed in order to establish the real structure to the chassis structures. The natural frequencies were extracted. For the frame optimization, the author tried to use groups of numerical and programming techniques to search for the optimum value of mathematical functions. In other words, the purpose of the optimization is to facilitate in finding results that best fills out the needs. Based on the result, the analysis and experimental procedure applied had significantly improved the overall structural stiffness by 75% by maintaining the center of gravity and the total weight was increased by 6%.

Masahiro et al (2004) discussed dump truck in mining. Receive various dynamic loads during travel. The strength of main frames of dump truck has been predicted based on static stress analysis where dynamic loads obtained by measurement with actual machines are replaced with static loads. However, when a developed model employs a new structure, it is difficult to precisely determine the conditions that are critical for main frames on development stage. For the articulated dump truck that was developed
recently, it was required to precisely predict, on design stage, the stresses that act on the frames during travel. Therefore, we introduced elastic characteristic of main frame into kinematical analysis models by using kinematical analysis software ADAMS and finite element method software NASTRAN and developed the method to calculate frame stress that occurs during travel, which was applied to the rear frame of the articulated dump truck. As a result, it was confirmed that frame stress during travel can be calculated correctly with this method.

Sonsino et al (2004) described about the cumulative damage were carried out on two types of welded joints (butt welds and transverse stiffeners) of four structural steels, S355M, S690Q and S960Q in the thickness of $t = 10$ and 25 mm. Further investigations were performed with laser beam welded hollow hat specimens of an automotive steel DC04 and an aluminium alloy AA 5083 with sheet thicknesses of 2 and 3 mm. The random spectra used were of Gaussian type.

Rubinstein and Hitron (2004) reviewed the currently available models for dynamic simulation of tracked vehicles usually includes super-elements to describe the tracks and the suspension systems. In these models, the dynamics of the track, the interaction between each track link and the ground, and their effect on the vehicle dynamics cannot be considered properly. The rapid increase in computing speed enables the utilization of more complex models, including numerous bodies and force elements. A three-dimensional multi-body simulation model for simulating the dynamic behavior of tracked off-road vehicles was developed using the LMS-DADS simulation program.

Chiandussi et al (2004) discussed the results obtained by using a topology optimization code to solve a three-dimensional problem concerning a real automotive component. The implemented optimization method is based
on the maximization of the total potential energy with a volume constraint by optimality criteria. The volume of the optimal solution depends on the imposed static (displacement, stress, stiffness) and dynamic (natural frequency) constraints and has not to be specified a priori. The optimization process converges toward a quite well defined structure made of the base material with a very little percentage of elements characterized by intermediate material properties.

Eryurek et al (2005) explained the failure of the rear suspension spring is analyzed in detail. The rear axle suspension system of the truck and fractured flat spring is investigated. Fracture surface, mechanical and chemical properties and microstructure of the spring material is analyzed. Forces acting on the spring is determined and strength calculations are carried out. Later, failure behaviour and cause of fracture is revealed after carefully analysis of microstructure and results of calculations. At the end precautions to be taken to prevent a similar failure is recommended.

Deprez et al (2005) discussed off-road vehicles demonstrate that a lot of effort still has to be put into the design of effective seat and cabin suspensions. Owing to the nonlinear nature of the suspensions and the use of in situ measurements for the optimization, classical local optimization techniques are prone to getting stuck in local minima. Therefore this paper develops a method for optimizing nonlinear suspension systems based on in situ measurements, using the global optimization technique DIRECT to avoid local minima. Evaluation of the comfort improvement of the suspension was carried out using the objective comfort parameters used in standards. As a test case, the optimization of a hydro pneumatic element that can serve as part of a cabin suspension for off-road machinery was performed.

using spectral methods. Loading of Gaussian distribution and narrow- and broad-band frequency spectra were assumed. Various characteristic states of multiaxial loading were considered. The equivalent stress history was determined with use of the failure criteria of multiaxial fatigue based on the critical plane. For determination of the critical plane position, the method of variance was applied. During simulation, the authors compared the results obtained by a spectral method in the frequency domain with those from the rain-flow algorithm in the time domain. The paper also contains the results of fatigue tests for 18G2A structural steel subjected to bending and combined bending with torsion. The tests were performed in order to verify the proposed algorithms for determination of fatigue life. It has been shown that under multiaxial random loading results of fatigue life calculated according to the considered algorithms in frequency and time domains are well correlated with the results of experiments.

2.4.3 Finite Element Analysis of Vehicle Chassis Design

Izzudin B. Zaman (2006) 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 ton 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.

Zhongzhe and Ping (2006) discussed fatigue life analysis and improvements of the autobody in a sports utility vehicle (SUV) were performed. The stress distribution under unit displacement excitation was obtained by the finite element (FE) method. A bilateral track model was adopted to obtain load spectra in accordance with the vehicle reliability test. The total life of the autobody was evaluated by the nominal stress method with the assumption of a uniaxial stress state, and thus the critical regions were determined. The life of components with critical regions was further
investigated on the basis of multiaxial fatigue theory. The results show that some components near to the suspension are easy to damage because they are directly subjected to impact loading from the road. It is also indicated that the result from multiaxial fatigue analysis is more reasonable than that from the nominal stress method, which was verified by experimental results. Finally, topological optimization of the spot weld location in the critical region was carried out by the homogenization method to improve its fatigue life.

Hoffmeyer et al (2006) discussed some issues in multi axial fatigue and life estimation is presented. While not intended to be comprehensive, these are a relatively broad range of issues which are commonly encountered when dealing with multi axial fatigue. They include damage mechanisms, non-proportional hardening and constitutive behavior, damage parameters and life estimation, variable amplitude loading, cycle counting, damage accumulation, and mixed-mode crack growth. Some simple approximations in capturing some of these effects in multi axial life estimations are also presented.

Jung - Seok Kim (2006) discussed about the Fatigue assessment of tilting bogie frame for Korean tilting train: Analysis and static tests and complete explanation about the Dynamic and fatigue response is one such issue requiring careful evaluation. This paper studies the behavior of a truss bridge, where an FRP deck replaced an old deteriorated concrete deck, using experimentally validated finite element (FE) models. FE models were employed to conduct dynamic time-history analyses with a moving AASHTO fatigue truck over the bridge. The results were used to evaluate the effects of the rehabilitation process on the remaining fatigue life of the structure. Numerical results show that the fatigue life of the bridge after rehabilitation would be doubled compared to a pre-rehabilitated reinforced concrete deck system. Based on the estimated truck traffic that the bridge carries, stress ranges of the FRP deck system lie in an infinite fatigue life regime, which
implies that no fatigue failure of trusses and floor system would be expected anytime during its service life.

Smith et al (2007) explained an integrated motor and gearbox package drives a 3.9 kN flocculator train at a water filtration plant. The 4–20 rpm output shaft turns the flocculators at 1–5 rpm. Mounted on this shaft is a torque limiting coupling, utilizing a SAE 1040 steel shear pin. Analogous to a fuse in an electrical circuit, this shear pin fails if the flocculator paddles jam, thus protecting the equipment from overload. With the current design, a shear pin only last about one week. Analysis suggests that cyclic fatigue, caused by misalignment between driving and driven coupling elements, causes premature pin failure. A redesigned shear pin made from PH 13-8 Mo stainless steel, processed to improve fatigue resistance, is described.

Chua-bin Chen et al (2007) explained the unfitness of weld and gap variation are salient using traditional TIG procedure for the five-port connector flange. Insufficient reliability and the instability of the weld are catastrophe, which cannot accommodate to new generation type production’s requirement. In this study, a finite element model of the five-port connector was built and the distribution of temperature field and deformation were studied. The result shows that welding thermal cycle is greatly different from the ordinary butt weld during welding of the flange and spherical shell. The welding deformation is complex from the direction of UX and UZ. Especially maximum deformation from the direction of UZ is about 4.96mm. According to the result, the optimum welding fixture is designed on the view of the smallest deformation, to critically control the deformation of the both sides of the welding line, finally to accomplish the robotic flexible welding.

Bayarakceken et al (2007) discussed power transmission system of vehicles consist several components which sometimes encounter unfortunate failures. Some common reasons for the failures may be manufacturing and
design faults, maintenance faults raw material faults, material processing faults as well as the user originated faults. In this study, fracture analysis of a universal joint yoke and a drive shaft of an automobile power transmission system are carried out. Spectroscopic analyses, metallographic analyses and hardness measurements are carried out for each part. For the determination of stress conditions at the failed section, stress analyses are also carried out by the finite element method.

Chiewanichakorn et al (2007) explained lighter fiber reinforced polymer (FRP) decks are gaining popularity among bridge owners as an alternative to replace old deteriorated heavy concrete bridge decks to increase the live load capacity of old steel superstructures requiring minimal repairs. This is more attractive in case of old truss bridges, which are relatively in good condition, but are not designed to current live loads and cannot be easily rehabilitated to improve their capacities. Replacing the heavy concrete decks on these bridges will extend their service life by reducing the dead load and thus increasing the live load capacity. When an FRP deck is used in such cases, a system level approach should be used to evaluate the bridge condition for all possible failure mechanisms.

Prawoto et al (2007) discussion of an automotive suspension coil springs, their fundamental stress distribution, materials characteristic, manufacturing and common failures. An in depth discussion on the parameters influencing the quality of coil springs is also presented. Following the trend of the auto industry to continuously achieve weight reduction, coil springs are not exempt.

Jiang et al (2007) explained the crack growth based criterion were evaluated using the combined axial-torsion fatigue testing results obtained from extensive experiments on thin-walled tubular specimens made from S460N. The Fatemi–Socie criterion combines the maximum shear strain
amplitude with a consideration of the normal stress on the critical plane. The Jiang criterion makes use of the plastic strain energy on a material plane as the major contributor to the fatigue damage. By assuming an initial crack length, the short crack model attributes the fatigue life to the action of a crack driving force, namely the effective cyclic $J$-integral. The results show that all the three criteria correlated well with the experimental observations in terms of fatigue life predictions. A great discrepancy was found between the predicted cracking directions and the observed cracking orientations.

Ye and Moan (2007) discussed static and fatigue behavior of three types of aluminum box-stiffener/web connections are investigated in this study. The main purposes are to provide a connection solution that can reduce the fabrication costs by changing the cutting shapes on the web frame and correspondingly the weld process and meanwhile sufficient fatigue strength can be achieved. Finite element analyses (FEA) show the influence of local geometry and weld parameters on the stress gradient near the fatigue cracking area. The influence of the weld parameters on the structural stress concentration factors is also studied. Twelve specimens of every type were tested and the test data are compared both to a nominal stress based design SN curve Eurocode9/31 and a structural stress based design SN curve Eurocode9/44.

Roslan Abd Rahman et al (2008) discussed stress analysis of heavy duty truck chassis. The stress analysis is important in fatigue study and life prediction of components to determine the critical point which has the highest stress. The analysis was done for a truck model by utilizing a commercial finite element packaged ABAQUS. The model has a length of 12.35 m and width of 2.45 m. The material of chassis is ASTM Low Alloy Steel A 710 C (Class 3) with 552 MPa of yield strength and 620 MPa of tensile strength. The result showed that the critical point of stress occurred at the opening of
chassis which is in contact with the bolt. The stress magnitude of critical point is 386.9 MPa. This critical point is an initial to probable failure since fatigue failure started from the highest stress point.

Miyake et al (2008) discussed influencing variables and causes of hot forging die failures for automotive components are summarized. Characteristics of hot forging die failures are exemplified. An important role of microfractography is stressed in engineering failure analysis for hot forging die failures. Then failure analyzed examples for hot forging die failures and their countermeasures are presented with influencing variables such as die materials, die design, die manufacturing and forging operations. Finally a couple of recommended works for engineering failure analysis for hot forging die failures for automotive components are touched on briefly.

Rahman Rosli et al (2008) discussed the technique of the fatigue analysis of spot-weld joints to predict the lifetime and location of the weakest spot-welds due to the imposed loading conditions. A simple model was used to illustrate the technique of spot-weld fatigue analysis. Finite element model and analysis were carried out utilizing the finite element analysis commercial codes. Linear elastic finite element analysis was carried out to predict the stress state along the weld direction. It can be seen from the results that the predicted life greatly influence the sheet thickness, spot diameter and loading conditions of the model. Acquired results were shown the predicted life for the nugget and the two sheets around the circumference of the spot-weld at which angle the worst damage occurs. It is also observed that the sheet-2 appeared the maximum stress range among the model. The spot-welding fatigue analysis techniques are awfully essential for automotive structure design.

Okurdi et al (2008) discussed about the one of the most important steps in development of a new truck chassis is the prediction of fatigue life
span and durability loading of the chassis frame. The age of many truck chassis in Malaysia are of more than 20 years and there is always a question arising whether the chassis is still safe to use. Thus, fatigue study and life prediction on the chassis is necessary in order to verify the safety of this chassis during its operation. Stress analysis using Finite Element Method (FEM) can be used to locate the critical point which has the highest stress. This critical point is one of the factors that may cause the fatigue failure. The magnitude of the stress can used to predict the life span of the truck chassis. In this study, the stress analysis is accomplished by the commercial finite element packaged ABAQUS.

Chinnaraj et al (2008) explained current trend in automotive design is to optimize components for weight. To achieve this, automotive designers need to have complete understanding of various stresses prevalent in different areas of the component. The chassis frame assembly of a heavy truck used for long distance goods hauling application is chosen for this investigation and dynamic stress-strain response of the component due to braking and cornering maneuvers are experimentally measured and reported. A quasi-static approach that approximates the dynamic maneuvers into number of small processes having static equilibriums is followed to carry out the numerical simulation, approximating the dynamic behavior of frame rail assembly during cornering and braking. With the help of commercial finite element package ANSYS, the quasi-static numerical simulations are carried out and compared with experimental results. This study helps in understanding prevailing stresses in truck frame rails especially during cornering and braking maneuvers and brings out all geometric locations that may be potential failure initiation locations. This study makes a case for further investigation on the effects of residual and assembly stresses on frame rails.
Awoto et al (2008) discussed about automotive suspension coil springs, their fundamental stress distribution, materials characteristic, manufacturing and common failures. An in depth discussion on the parameters influencing the quality of coil springs is also presented. Following the trend of the auto industry to continuously achieve weight reduction, coil springs are not exempt. A consequence of the weight reduction effort is the need to employ spring materials with significantly larger stresses compared to similar designs decades ago. Utilizing a higher strength of steel possesses both advantages and disadvantages. The advantages include the freedom to design coil springs at higher levels of stress and more complex stresses. Disadvantages of employing materials with higher levels of stress come from the stresses themselves. A coil’s failure to perform its function properly can be more catastrophic than if the coil springs are used in lower stress. As the stress level is increased, material and manufacturing quality becomes more critical. Material cleanliness that was not a major issue decades ago now becomes significant. Decarburization that was not a major issue in the past now becomes essential.

Hobbacher et al (2009) explained the stress intensity factors of welded joints have been calculated for a variety of joint types and dimensional parameters. The following have been considered: non-load carrying transverse and longitudinal attachments, cruciform joints with K-butt and with fillet welds, and lap joints with fillet welds. The correction function of the stress intensity factor $Y_t$ has been split into a function for the general configuration and a function, which covers the local stress concentration field of the weld.

Veloso Magalhães et al (2009) discussed the failure investigation and stress analysis of a longitudinal stringer of an automobile chassis Fiat Automóveis, Rod. Fernão Dias, km 429, Betim, MG, Brazil Pontifical
Catholic University of Minas Gerais (PUC Minas), Mechanical Engineering, Belo Horizonte, MG, Brazil. A prototype vehicle was submitted to durability test, on road at a proving ground test track. Failures of posterior longitudinal stringers were observed during this test. Cracks were nucleated on these stringers during durability test, before the designed life of these components is reached. These cracks were observed at nearly the bumpers fixation points of the vehicle suspension. Loads are transmitted by wheels to the body of the vehicle through the suspension components. Thus, the longitudinal stringers are subjected to these localized cyclic stresses.

Topac et al. (2009) explained the premature failure that occurs prior to the expected load cycles during the vertical fatigue tests of a rear axle housing prototype is studied. In these tests, crack mainly originated from the same region on test samples. To determine the reason of the failure, a detailed CAD model of the housing was developed. Mechanical properties of the housing material were determined via tensile tests. Using these data, stress and fatigue analyses were performed by finite element method. Fatigue crack initiation locations and minimum number of load cycles before failure initiation were determined. Results provided from tests were compared with the analyses. Design enhancement solutions were proposed to increase the fatigue life of the housing.

Palma et al. (2009) investigated to analyze the fatigue behavior of an automobile body part, according to the standards of performance. The methodology is based on experiments performed on a rear trailer tow hook pin of a passenger automobile vehicle. Experiments were performed simulating the actual conditions in the customer environment. Stress and strain were experimentally measured by using strain gages, bonded on assembly critical points. Besides, stress analysis was also performed using a
finite element program. Fatigue analysis is used to access and to compare the fatigue damage imposed during laboratory experiments.

Kim et al (2009) explained an automotive electronics industry, demand for low-cost, high strength-to in-service performance for electronic components continues to drive the development of vehicles’ door Wiring Harness (W/H) system for new applications. The problem of the fatigue strength estimation of materials or components containing natural defects, inclusions or in homogeneities is of great importance from both a scientific and industrial point of view. This article gives some insight into the dimensioning process, with special focusing on fatigue analysis of W/H in a vehicle’s door structures. An en-durance life prediction of door W/H was calculated using finite element analyses. Endurance test data for slim test specimens were compared with the predicted fatigue life for verification. The final life expectancy of the component combines the effects of these micro structural features with the complex stress state arising from the combined service loading and residual stresses.

Johann Wannenburg et al (2009) carried out common practice in the automotive industry to employ multi-parameter strain-life methods, in combination with dynamic finite element analyses, based on very extensive measurement exercises, to conduct analytical fatigue life assessments, which is then verified through intensive durability testing. The expense and complexity of this approach makes its application impractical for low volume “special” vehicles. In this paper, a fatigue equivalent static load (FESL) methodology for the numerical durability assessment of heavy vehicle structures is presented, where fatigue load requirements are derived from measurements as quasi-static g-loads, the responses to which are considered as stress ranges applied a said number of times during the lifetime of the structure. The application of the method is demonstrated using two case
studies, namely a road tanker and a load haul dumper. In both cases, it was possible to obtain adequately accurate fatigue life prediction results, using simplified loading, static finite element analyses and a stress-life approach to fatigue damage calculations, with material properties available in design codes. The effect of connection plat thickness on stress of truck chassis with riveted and welded joint under dynamic loads is carried out by Zehsaz et al (2009).

Xiaofeng Wanga and Xiaoge Zhang (2010) discussed the practical and comprehensive method for simulating the dynamic cornering fatigue test of the automotive wheels. The test of a steel passenger car wheel is simulated by combined use of the linear transient dynamic finite element analysis and the local strain approach. A rotating force of constant magnitude is applied to the moment arm tip to simulate the rotating bending effect on the wheel, with the wheel stationary. It is found that only a radial component of the rotating force is needed to obtain the sufficiently accurate radial normal strain histories of the elements located along the radial direction. The strain history of the element whose local stress–strain characteristic keeps linear and closest to the critical element is applied to predict the fatigue life of the critical element with Neuber’s rule and local strain approach, which is quite close to the test results.

Yongjie Lu et al (2010) investigated numerical simulation and field test are used to investigate tire dynamic load. Based on multi-body dynamics theory, a nonlinear virtual prototype model of heavy duty vehicle (DFL1250A9) is modeled. The geometric structural parameters of the vehicle system, the nonlinear characteristics of shock absorber and leaf springs are precisely described. The dynamic model is validated by testing the data, including vertical acceleration of driver seat, front wheel, intermediate wheel and rear wheel axle head. The agreement between the response of the virtual
vehicle model and the measurements on the test vehicle is satisfactory. Using the reliable model, the effects of vehicle speed, load, road surface roughness and tire stiffness on tire dynamic load and dynamic load coefficient (DLC) are discussed. The results demonstrate that the proposed model can offer efficient and realistic simulation for stochastic dynamic loads, so as to investigate vehicle road-friendliness.

Mergen H Ghayesh and Sara Balar (2010) explained Non-linear parametric vibration and stability of an axially moving Timoshenko beam are considered for two dynamic models; the first one, with considering only the transverse displacement and the second one, with considering both longitudinal and transverse displacements. The set of non-linear partial-differential equations of both models are derived using an energy approach. The method of multiple scales is applied directly to both models, and using the equation order one, the mode shape equations and natural frequencies are obtained. Then, for the equation order epsilon, the solvability conditions are considered for the resonance case and the stability boundaries are formulated analytically via Routh–Hurwitz criterion. Eventually, some numerical examples are provided to show the differences in the behavior of the above-mentioned non-linear models.

Daowu Zhou et al (2010) explained fatigue module in OPTWELD takes into account welding residual stresses and distortion in the structure. This is done by carrying out a thermal-metallurgical-elasto-plastic finite element analysis to predict the residual stress. This is followed by elastic analyses under unit loads on the distorted structure. A spectrum of applied loads is then used to calculate the applied stress range spectrum which is also modified to account for the residual stresses. The fatigue damage factor is finally evaluated using a Miner's summation rule.
Savaidis et al (2010) discussed numerically investigated analytical elastic–plastic model is presented describing the fatigue life of arbitrary engineering components with elliptical notches under constant amplitude loading. A DJeff-based crack growth law is integrated from a starting crack size of micro-structural dimension up to the total fracture of the component. Plasticity-induced crack opening and closure effects are explicitly taken into account. Calculated opening load levels for cracks growing in notch affected areas have been found to be in good agreement with corresponding experimental values determined from notched specimens made of two different metallic materials. Comparison of experimentally determined and calculated crack growth curves for specimens with central notches confirm the satisfactory accuracy of model. The experimental verification of the model’s calculation.

Pedersen et al (2010) investigated experimental fatigue data for welded joints have been collected and subjected to re-analysis using the notch stress approach according to IIW recommendations. This leads to an overview regarding the reliability of the approach, based on a large number of results (767 specimens). Evidently, there are some limitations in the approach regarding mild notch joints, such as butt joints, which can be assessed non-conservatively. In order to alleviate this problem, an increased minimum notch factor of KwP2.0 is suggested instead of the current recommendation of KwP1.6. The data for most fillet-welded joints agree quite well with the FAT 225 curve; however a reduction to FAT 200 is suggested in order to achieve approximately the same safety as observed in the nominal stress approach.

Kutay Yilmazcoban and Yasar Kahraman (2011) studied heavy duty vehicles like trucks, has wide range of structural and conceptual design. In Turkey ground transportations and heavy duty works like mine and rock transportation are made with Long Vehicles and especially with middle
tonnage trucks (10 – 40 tones). Many truck chassis manufacturers use thick profiles for the chassis for reliability of the frame. Using more than enough material causes expensive product manufacturing conditions. To reduce the expenses of the chassis of the trucks, the frame’s structure design should be changed or the thickness should be decreased. In this study, for manufacturing reliable and inexpensive middle tonnage truck chassis, used chassis structures by some manufacturers are optimized by the thicknesses of the profiles. This study showed that the thinner chassis profiles can be reliably used in the truck chassis sections with the help of structural finite element analysis.

Wiebesiek et al (2011) reviewed Laserbeam-welded overlapped tubular joints made from structural steel St35 (S235 G2T), artificially hardened aluminium alloy AlSi1MgMn T6 (EN AW 6082 T6) and self-hardening aluminium alloy AlMg3.5Mn (EN AW 5042) were investigated under combined proportional and non-proportional axial and torsional loading with constant amplitudes in the range of $2 \times 10^4 – 1 \times 10^7$ cycles. The multiaxial fatigue behaviour of these joints was assessed by the notch stress concept with the reference radius $r_{ref} = 0.05$ mm. The equivalent stresses, especially considering the fatigue life reducing influence of non-proportional loading in comparison to proportional loading, were calculated by a recently developed hypothesis, which is called the Stress Space Curve Hypothesis (SSCH). This hypothesis is based on the time evolution of the stress state during one load cycle.

Witold Pawlus et al (2011) explained the present application of regressive models to simulation of car-to-pole impacts. Three models were investigated: RARMAX, ARMAX and AR. Their suitability to estimate physical system parameters as well as to reproduce car kinematics was examined. It was found out that they not only estimate the one quantity which was used for their creation (car acceleration) but also describe the car’s
acceleration, velocity and crush. A virtual experiment was performed to obtain another set of data for use in further research. An AR model to predict the behavior of a low-speed car impacting a rigid barrier was created and verified.

Marco Cavazzuti et al (2011) explained weight reduction is a major issue for carmaker companies due to the need to comply with the emission regulations without reducing the vehicle safety. A classic trial-and-error approach to design in the automotive industry is becoming inadequate and new means are needed to enhance the design process. A major improvement on the end product can be achieved by adopting suitable optimization techniques from the early design stage. In the present paper the problem of automotive chassis design in view of weight reduction is tackled by means of topology optimization. The design methodology proposed is applied twice: at first addressing a chassis for spider vehicles, then for couple vehicles.

Nan Zhou et al (2012) described the prediction of fatigue life of metal welded joints plays an important role at lower manufacturing costs and reduces accidents for engineering materials, the response of metal welded joints to fatigue properties has highly non-linear, so it is difficult to establish an accurate theoretical model using traditional method to predict its fatigue life. It is appropriate to consider modeling methods developed in other fields in order to provide adequate models for metal welded joints behavior on fatigue properties. Accordingly, a new system predict method, based on a hybrid genetic algorithm (GA) with the Back-propagation neural network (BPNN), for the simultaneous establishment of a predict model structure of fatigue life of metal welded joints and the related parameters is proposed. Based on the self-learning ability and approximation of non-linear mapping capability of the BPNN, by taking the advantages of the powerful ability of global optimization, implicit parallelism and high stability of the GA, the
optimal parameters have been automatically determined, we establish a parameter adaptive optimization of GANN model to fit and predict the fatigue life of metal welded joints. GANN establishes the mapping relationship between the fatigue properties of metal welded joints and a variety of influencing factors, having greatly increased the computational efficiency for the fatigue properties of metal welded joints, also had a higher predict accuracy. The superiority of GANN had been tested by the prediction of the fatigue life of welded joints in different process parameters.

Patel Vijay Kumar and Patel (2012) investigated automotive chassis is an important part of an automobile. The chassis serves as a frame work for supporting the body and different parts of the automobile. Also, it should be rigid enough to withstand the shock, twist, vibration and other stresses. Along with strength, an important consideration in chassis design is to have adequate bending stiffness for better handling characteristics. So, strength and stiffness are two important criteria for the design of the chassis. This report is the work performed towards the static structural analysis of the truck chassis. Structural systems like the chassis can be easily analyzed using the finite element techniques. So a proper finite element model of the chassis is to be developed. The chassis is modeled in PRO-E. FEA is done on the modeled chassis using the ANSYS Workbench.

Washington (2012) described an automotive chassis is an important part of an automobile. The maximum stress, maximum equilateral stress and deflection are important criteria for the design of the chassis. This report is the work performed towards the optimization of the automotive chassis with constraints of maximum shear stress, equivalent stress and deflection of chassis under maximum load.

Shaohua Li et al (2012) reviewed three dimensional heavy vehicle-pavement-foundation coupled system is modelled as a seven DOF vehicle
moving along a simply supported double-layer rectangular thin plate on a linear visco elastic foundation. Using the Galerkin method and a quick direct integral method, the dynamic responses of the coupled system with stochastic road surface roughness are obtained numerically. The effects of vehicle and road parameters on vehicle riding comfort and road damage are also investigated and some interesting conclusions are found which are beneficial to parameter design. In addition, the validity of simulation results is verified by a field test.

Hengji et al (2012) explained about the fatigue life for frame of the 220t mining dump truck, a fatigue life analysis method is presented, integrating multi body dynamic analysis and finite element method. The forces of main joints at frame are measured from the multi body dynamic model, whose road is restructured based on ISO/TC108/SC2N67. According to GB/T27025-2008, the dynamic stress test of the whole truck is implemented to obtain the peak stress of the mainly forced area, which is compared with the simulated stress. It is found out that the error is allowable so that the accuracy of the finite element model is definitely ensured. The quasi-static stress analysis method is employed to acquire stress influence coefficient under unit load, which is associated with load histories of the frame to get the dangerous stress area. The fatigue life of the frame is calculated on the basis of Palmgren–Miner damage theory. It is turned out that the minimum life area of the frame is located at the frame joints of suspension, which matches the practice.

Awlad Hossain et al (2012) explained about the cantilever beams for characterization of rheological properties of viscous materials is demonstrated. The dynamic response of a mini cantilever beam partially submerged in air and water is measured experimentally using a duel channel PolyTec scanning vibrometer. The changes in dynamic response of the beam
such as resonant frequency, and frequency amplitude are compared as functions of the rheological properties (density and viscosity) of fluid media. Next, finite element analysis (FEA) method is adopted to predict the dynamic response of the same cantilever beam. The numerical prediction is then compared with experimental results already performed to validate the FEA modeling scheme. Once the model is validated, further numerical analysis was conducted to investigate the variation in vibration response with changing fluid properties. Results obtained from this parametric study can be used to measure the rheological properties of any unknown viscous fluid.

Haval Kamal Asker et al (2012) discussed the frame of the standard dump truck supports all types of complicated loads coming from the road and freight being loaded. So, the intensity and the strength of the frame play a big role in the truck's design. A frame of 6 wheels, standard dump truck has been studied and analyzed using ANSYS package software. The static intensity of the frame has been analyzed when exposed to pure bending and torsion stress, within two cases. First case is when the rear wheels zigzag gets over block (only one side of the chassis steps the block), and the second case is when both wheels gets over the block. Finite element model of a stress analysis of the vehicle chassis has been built using three dimension hyper elastic elements for the modeling. The results show important differences between the two case studies, especially in the torsion and deformations results obtained from the chassis model. Also, vibration modes have been analyzed during the loading conditions. The more damping ratio used, the more stabilizing of the stresses with respect to time.

2.5 COMMERCIAL FINITE ELEMENT SOFTWARES

Commercial finite element software’s play an important role in solving many engineering analysis problems. In early research works, finite element codes were written to compute static and dynamic using popular
programming languages such as ‘C’, ‘C++’, ‘FORTARN’ etc. While using these programming languages for writing finite element codes, certain difficulties are experienced by the researchers especially in the description of complex shapes and in post processing the results. With the advent of powerful CAD and CAE packages and meshing algorithms, popular general purpose standard finite element software such as HYPER MESH, ANSYS, ADAMS & FE – FATIGUE etc have been introduced and they are being increasingly used by the chassis design researchers for the past few decades.

HYPER MESH is pre processing software which is mainly used for meshing of the system. Altair Hyper Mesh is a high-performance finite element pre- and post-processor for major finite element solvers, which allows engineers to analyze design conditions in a highly interactive and visual environment. Hyper Mesh’s user-interface is easy to learn and supports the direct use of CAD geometry and existing finite element models, providing robust interoperability and efficiency. Advanced automation tools within Hyper Mesh allow users to optimize meshes from a set of quality criteria, change existing meshes through morphing, and generate mid-surfaces from models of varying thickness.

‘ANSYS’ is the shortened term of ‘System Analysis’. It contains many bench mark tests drawn from a variety of resources such as NAFEMS (National Agency for Finite Element Methods and Standards), based in the United Kingdom, to validate the performance of elements under distorted or irregular shapes, different meshing schemes, different loading conditions, various solutions algorithms, energy norms etc.

FEM software programmed is organized into different processors such as described below:

i) Pre-processor

ii) Solution processor

iii) Post-processor
The first step define the geometric domain of the problem, the element type to be used, the material properties of the elements, the geometric properties of the elements, the element connectivity (mesh), the physical constraints (boundary conditions) and the loadings.

The second step is processing ie solution, in this step the governing algebraic equations in matrix form and computes the unknown values of the primary field variables are assembled. The computed results are then used by back substitution to determine additional, derived variables, such as reaction forces and element stresses.

The third step is post processing, the analysis and evaluation of the result is conducted in this step. Examples of operations that can be accomplished include total deformation, principal calculate factors of safety, animate the output results and produce color-coded stress plots.

2.6 CONCLUSION

A detailed literature survey has been made on the following areas in order to determine a static stress distribution and fatigue life of the chassis using finite element simulation.

1. First, deals with the literature in respect of simulation of static and dynamic analysis histories involved in the chassis design and analysis of bending, torsion and fatigue life analysis.

2. Secondly, it covers the some of the research work in the development of static and dynamic simulation of chassis frame analysis for off - highway application with emphasis of on the prediction of fatigue life and stress distribution of the critical area.

3. It also covers some of the earlier works of modeling, design and analysis of an automotive chassis.