A SYSTEMATIC METHODOLOGY FOR FATIGUE ANALYSIS OF MACHINE ELEMENTS WITH CHARACTERIZED DYNAMIC LOADS

by

Rahul Mula

A Thesis

Submitted to the Faculty of Purdue University

In Partial Fulfillment of the Requirements for the degree of

Master of Science in Engineering

Department of Civil and Mechanical Engineering

Fort Wayne, Indiana

May 2019

THE PURDUE UNIVERSITY GRADUATE SCHOOL
STATEMENT OF COMMITTEE APPROVAL

Dr. Zhuming Bi, Chair

Department of Civil and Mechanical Engineering

Dr. Nashwan Younis

Department of Civil and Mechanical Engineering

Dr. Hosni Abu-Mulaweh

Department of Civil and Mechanical Engineering

Approved by:

Dr. Carol Sternberger

Head of the Graduate Program

ACKNOWLEDGMENTS

This study was completed under the supervision and guidance of my advisor Dr. Zhuming Bi. I am very much thankful for his support, encouragement and most notably patience during the past two years of my studies. I would like to extend my sincere gratitude to Dr. Hosni Abu-Mulaweh for supporting while pursuing this degree. I would express my sincere appreciation to my graduate committee member Dr. Nashwan Younis for his great support. The entire faculty of Civil and Mechanical Engineering department is thanked for preparing me to this point in the curriculum.

Finally, my deep and sincere gratitude to my family and friends for their unconditional trust, timely encouragement, and endless patience.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

ABSTRACT

Author: Mula, Rahul. MSE
Institution: Purdue University
Degree Received: May 2019
Title: A Systematic Methodology for Fatigue Analysis of Machine Elements with Characterized Dynamic Loads
Committee Chair: Dr. Zhuming Bi

Fatigue analysis is essential for the optimization of products subjected to dynamic loads. However, a number of fatigue analysis theories have been developed, how to apply an established method in real-world product designs is not a trivial task. Most of small or medium sized enterprises (SMEs) still rely heavily on the experiments to evaluate the fatigue lives of products. Among existing fatigue design methods (i.e., experiments, analytical methods, and simulations), the simulation-based methods have the advantages of low cost, low risk environment and enable a designer to determine the accuracy and performance of a product design without building physical prototypes.

Regarding the methodologies for fatigue analysis, some identified challenges are (1) the characterization of dynamic loads, (2) the formulation of finite element models which can be aligned with applications or testing scenarios, and (3) the verification and validation of simulations. To make a simulation-based fatigue analysis more practical for real-world product designs, the solutions to the aforementioned problems must be found. This thesis aims to establish a systematic methodology to perform the fatigue analysis for product design with any material, carbon steel material is used for the present case study to illustrate and verify the proposed methodology for fatigue analysis.

Major tasks involved in this thesis study are:

1. The method for the characterization of dynamic loads. It is a numerical method to simulate the kinematic and dynamic behaviors subjected to the given motion, and it is expected to extract interacting dynamic forces of components to be analyzed.
2. The systematic method and procedure to formulate the problem of fatigue analysis as a finite element analysis model and find the solution of fatigue life of product.
3. The procedure and approaches are developed to verify and validate fatigue analysis models and procedure used for the present case study.
4. The parametric studies with a set of design variables to show the feasibility and flexibility of using simulation methods to evaluate the influence of multiple design variables on wheel products.

Key Words: Wheel products; Fatigue analysis; Static analysis; SolidWorks; Finite Element Analysis (FEA); Characterization of dynamic loads; Motion simulation; Verification and validation.

1 INTRODUCTION

In general, wheel products are used in many industries such as automotive, railway, agriculture, and medical applications. In every application, wheel components can be critical since they make direct contacts between products and environment. Therefore, a major concern of a wheel is its durability subjected to the given operating conditions. In their design processes, wheel products are mostly tested to estimate the durability of wheels by testing on rolling drums and the machines for dropping. It is well expected that virtual analysis can be applied to partially replace physical tests in predicting fatigue lives of wheels.

The load on a tested wheel is fluctuated. The wheel products are mostly tested experimentally, and no such a procedure is available to analyze wheel designs virtually to replace physical tests. As far as numerical methods are concerned, finite element methods become prevalent as a default virtual analysis tool. However, most of the relevant works on wheel designs were performed with the assumptions of static loads, few works were reported on the fatigue studies of wheels under dynamic loads.

1.1 Purpose of Thesis Study

From the existing literatures of FE analysis, wheel products are mostly analyzed subjected to static loadings; those works are insufficient to estimate the failure life of a wheel under dynamic loading conditions. In practice, the durability of wheels must be tested physically using rolling drum tests and weight drop tests. While fatigue analysis by physical experiments is time-consuming, which could take a few hundreds of hours for only one-wheel design subjected to one specified set of loading conditions. Therefore, two main objectives of this study are:

(1) identify the dynamic response of wheel assembly when undergoes dynamic loading that happens over time as opposed to static loading condition, which particularly carries into account the response of a system at step by step. This requires inertial and damping effects to be included in defining the loads. This type of loading is featured as variable amplitude cyclic loading;

(2) develop a systematic approach to simulate the behaviors of wheel products under dynamic loading conditions and predict a fatigue life of products a virtual environment. The developed numerical approach is expected to fully or partially replace physical tests.

Moreover, the thesis work is driven by a client company. The company is seeking a computer aided engineering (CAE) solution for their product design and testing. The company is especially interested in developing finite element models using the SolidWorks Simulation for the scenarios of applications similar to their current rolling-drum tests. With such a model, engineers will be able to predict the fatigue lives of wheel products with the minimized need of physical tests.

1.2 Technical Challenges

A fatigue analysis by the finite element method involves three following steps, and this thesis aims to develop the solutions to perform the activities in these three steps:

1. Characterization of dynamic loads - A new method is needed to simulate the behaviors of products to obtain the interacting forces on wheels equivalent to the scenarios in rolling tests.
2. Static stress analysis – A template of FEA model is needed to determine the stress distribution over products subjected to normalized static loading conditions.

3. Fatigue analysis – A template of FEA model is needed for fatigue analyses of products subjected to characterized dynamic loads.

In step one, dynamic loads will be characterized by a Motion Study in the SolidWorks to obtain the information of dynamic loading conditions in testing conditions. For the same system constraints, the boundary and loading conditions will be defined accordingly in the template of FEA models. The setup for a Motion Study allows the changes of some design parameters such as spring constant value and geometry of wheels to match the testing conditions. Besides the challenges in finite element modelling, this study also needs to address the following issues:

1. For verification and validation, there should be an appropriate physical test results. Therefore, it is very important to have some physical studies to verify and validate the simulation model and compare the procedures, the behaviors of the components when the loads are applied.
2. For virtual analysis, better computing resources are the prerequisites for the extensive investigation on different design parameters as well as some critical settings of a finite element model such as the options of element types, mesh sizes, solving methods to overcome convergence issues, and the definitions of contact conditions.
3. A parametric study is expected to examine the impact of various design variables. However, it needs many iterations and takes a long time if a large number of combinations of design variables and levels are evaluated. In general, design variables should be carefully selected for some main factors on fatigue lives.

4. There are very few previous studies relevant to the proposed study where the fatigue analysis will be conducted by virtual analysis on castor wheels. A few of the articles on fatigue analysis of wheels were based on experimental dynamic loads. This poses a challenge for the verification and validation of the proposed study since the data to be used will not be the same to the targeted testing environment.
5. It is always better to have the sufficient material data, in particular, the Strength versus Number of cycles (S-N) data for the fatigue study. However, such data is scarce for plastics or composites.

1.3 Scope of Thesis

The objective of the research is to develop the solutions to address the aforementioned issues in three critical steps. Those solutions are fundamental to fatigue analysis on wheel products with any selective materials and designs. The thesis includes (1) the characterization of dynamic loadings applied to roller bump test, and (2) the development of new virtual model or system representing the physical setup and run the model with relevant loading and boundary conditions. A systematic approach refers to the procedure of developing an FE model, which can be used as a general template for static and fatigue analysis of any specialized castor wheels. The inputs of the test setup for static and fatigue analysis are made manually; these inputs include material data, S-N data for fatigue study, loading and boundary conditions. For the present simulation, the verification and validation must be conducted by comparing with existing case studies in the relevant literatures.

1.4 Organization of Thesis

This thesis includes seven chapters. The first chapter gives an overview, the purposes and the scope of this study and the rest of the chapters are organized as follows.

In Chapter 2, the literatures related to the proposed research field are surveyed and discussed, and the covered topics include failure modes of products, static and fatigue analyses, mathematic methods on statistical analysis, and the preparation for virtual analysis of product designs.

In Chapter 3, the procedure for the characterization of dynamic loads is presented; it is achieved by the Motion Study in the SolidWorks, and it is used to evaluate the influence of dynamic factors for fatigue analysis.

In Chapter 4, the systematic approach is developed to define FEA models for static and fatigue analyses of wheel products. All the activities in defining an appropriate FEA for wheel products, including product geometries, boundary and loading conditions, meshing, and terminating conditions, are discussed in detail.

In Chapter 5, the verification and validation of simulations are performed to illustrate the effectiveness of the proposed methods.

In Chapter 6, the feasibility of using the proposed simulation approach is elaborated, the capability of developing parametric study for the comparison of a variety of design options is developed.

In Chapter 7, the presented work is summarized, the main contributions are discussed with the recommendation of future research upon the developed approach.

2 LITERATURE REVIEW

In this chapter, the literature review is presented, and the focuses are on (1) static and fatigue analysis of products and (2) the characterization of dynamic loads. In Section 2.1, existing methodologies for static and fatigue analysis of products are reviewed. In Section 2.2, the characterization of dynamic loads for fatigue analysis is examined thoroughly. In Section 2.3, the relevant works of using numerical simulations for static and fatigue analysis are summarized. In Section 2.4, the specific area of performing fatigue tests using rolling drum testers for wheels is discussed. In Section 2.5, existing works on the simulation-based fatigue analysis of wheels are covered. Finally, in Section 2.6, the limitations of existing works on fatigue analysis of wheels are identified to clarify the direction of research in this thesis.

2.1 Static and Fatigue Design Theory

Fatigue analysis, as it known today has come a long way. According to Wöhler (1855), in the 19th century, fatigue was considered as 'mysterious' failure; since a fatigue fracture did not show a visible plastic deformation. Later, Jean-Victor Poncelet, a designer of cast iron axles for mill wheels, formally coined the word "fatigue" for the first time in a text on mechanics (Poncelet 1939). Systematic fatigue fractures tests were done in laboratories, notable by August Wohler (1855). Since then, fatigue analysis was considered as a major engineering problem. Fatigue refers to the formation of crack initiation as well as the crack growth from a pre-existing defect, Fatigue damage is accumulated until a critical stage is reached (Wang et al. 2011). Fatigue analysis was conducted to describe a relationship among applied loads and strength (Albert 1838).

Budynas et al. (2011) divided the progress of fatigue damage in-to three different stages. The first stage was the initiation of micro cracks due to the cyclic plastic strain. The corresponding size of cracks at the first stage was often invisible to naked eyes. At the second stage, micro level cracks were progressed to macro level cracks, with a fluctuated loading condition, the cracks on the surface were repeatedly opened and closed which causes a continuous growth of crack at the third stage. The leftover material could no longer carry the employed loads and the cracks growth uncontrollably to cause a sudden fatigue fracture.

In using a finite element method for a fatigue study, the dynamic loads have to be identified. Fatigue strength can be analyzed in different ways such as analytical methods, experimental methods and numerical simulations. The approaches to evaluate the fatigue life of solids have been presented widely by Hamrock et al. (1999), Bundynas et al. (2014) and Nickerson et al. (2017). The three major procedures used in design and analysis are stress life methods, strain life methods and linear elastic fracture mechanics methods.

1. Stress life method – In this method, the fatigue is quantified as the number of cycles the object can survive without a failure. Fatigue is the phenomenon of weakening caused by cyclic loads (Ramachandran et al. 2005). Fatigue is continuous and confined structural damage under a dynamic load. Depending on the load range, fatigue is classified as Low cycle and High cycle fatigue (Coffin 1954). The component behavior not only depends on material properties, but also relates to characteristics of loads such as static or dynamic, temperature, environment, and surface conditions (Nanninga 2008). The stress life method (S-N method) is based only on stresses in the model, and it is mostly suited to the prediction of a high cycle fatigue. It is the

most traditional method and easy to implement in large range of applications. A S-N curve is plotted to provide the correlation among stress range and cycles to the failure (Basquin 1910). The stress life method was widely used by most of the researchers (Richart 1948).

2. Strain life method (ε-N method)- This method needs more detail analysis, since it takes in to account plastic strains in case of material fails due to fatigue (Laird 1963). The stresses occurring to stress concentration locations exceeds the material's elastic limit then causes plastic strains. A strain life method is more suitable for low cycle fatigue analysis.

3. Linear elastic fracture mechanics method (LEFM) was discussed by Anderson (1995) and Irwin (1957); it applies when the crack already exists in object. Damage can be analyzed by estimating the increase of the crack subjected to the given stress cycles, and the crack length is associated with the stress intensity. This approach is particularly helpful when the fatigue crack has previously been recognized (Griffith 1920, Westergaard 1939). Mott (1948) proposed a dynamic fracture mechanics, which considered of the Griffith's kinetic energy. Dynamic stress-intensity factors were then also proposed together with the dynamic generalization of Irwin's correlation (Freund 1990).

Out of three different methods, the stress life method is mostly used to predict fatigue lives of products by most researchers (Basquin 1910). The stress-life method has its advantages of requiring the minimum amount of material properties on fatigue, i.e., Strength-Number of cycles (S-N) curves of materials. It is mostly used to predict the fatigue life of a part at a low-level stress in comparison to its yield strength. Dowling et al. (2009) stated that for an application with higher elastic strains, the stress life method was equivalent to a strain life method. One essential piece of the information is to determine the endurance limit of the material. An endurance limit refers to the critical stress value for an infinite fatigue life. In practice, an infinite fatigue life corresponds to an acceptable large number of cycles, more especially greater than 10^6 cycles. To simplify a fatigue analysis, a dynamic load is usually simplified as a periodical sinusoid to represent the dynamic characteristics of loads. However, for the majority of components involved in machines, external loads are very complicated. Simplified periodical sinusoids have their limitations in characterizing dynamic properties appropriately since frequencies and amplitudes change continuously with respect to time.

In describing a dynamic load for a fatigue analysis, the waveform or shape is not so critical; however, peak values and frequencies of loads are critical. To this end, the maximized stress σ_{max} and the minimized stress σ_{min} in each loading cycle should be determined to characterize the dynamic change of stress in the cycle. In regard to the impact of static stress level on the fatigue life, a mean stress in a cycle should also be determined. Ranging above and below the mean stress can be modelled to characterize the changing pattern of stress.

A mean stress and alternating stress correlate to the maximized and minimized stresses. Assume that the maximized and minimized stresses are denoted as σ_{max} and σ_{min}, respectively, the mean stress σ_m and alternating stress σ_a can be determined as:

$$\sigma_m = \frac{\sigma_{max}+\sigma_{min}}{2} \tag{2.1}$$

$$\sigma_a = \frac{\sigma_{max}-\sigma_{min}}{2} \tag{2.2}$$

Where σ_m is the mean component of stress and σ_a is the alternating component of stress.

As a further note, the mean and alternating stresses will be used to analyze the fatigue cycle of objects in the specified failure criterion. Figure 2.1 shows some commonly used failure criteria (Budynas et al. 2011, Nickerson 2017) to take into the impact of mean and alternative stresses on fatigue life simultaneously. These methods are widely used to predict whether or not a fatigue failure will occur subjected to given dynamic loads. To evaluate the fatigue damage with respect to number of cycles (Aid et al. 2011, Manson et al. 1981), the accumulation of fatigue damage must be determined, the Miner's rule by Miner (1945) can be applied to take into consideration of all the effects due to frequency and amplitude changes of stresses (Yang 1998).

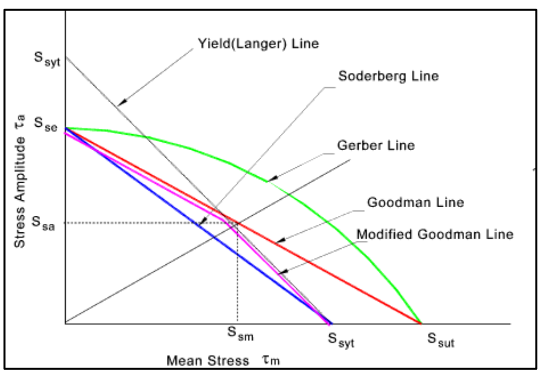


Figure 2.1 Fatigue Failure Criteria (Budynas et al. 2011)

If there are k levels of stress S_i over the give number of cycles, each stress level has a repetitive cycle of n_i, and the number of cycles at each stress level is given as N_i, then, the accumulated damage can be determined as (Miller et al. 1977, Ye 1996).

$$\sum \left(\frac{n_i}{N_i}\right) = D \tag{2.3}$$

where n_i is the cycle number of stress level accumulated in the loop count matrix, N_i is the failure corresponding to S_i, and D is the accumulated damage over the given number of cycles. Note that D is normalized and dimensionless.

When the damage fraction is equivalent to one, then a fatigue damage is expected to happen. The number of the remaindered cycles leading to a fatigue damage can be defined as the inverse of the damage fraction. In the Minor's rule, the stress level S_i at each sub-period is actually an alternating stress. If there is a combination of σ_a and σ_m in the period, the σ_a must be firstly modified by taking into consideration of σ_m to find an equivalent stress level (S_i) at a zero-mean stress (Aid et al. 2012). This is necessary since the Minor's Rule assumed the fatigue strength in S-N curve is fully alternative one. Therefore, a stress state with a non-zero mean stress has to be adjusted to use the Minor's rule. The defined S_i is also used to calculate N_i based on the S-N curve.

Numerous works have been reported on fatigue analysis of parts (Lee et al. 2005, Chaboche et al. 1988, Lagoda et al. 2005, Shang et al. 1998). For example, Fatemi et al. (1998) developed an improved model of cumulative damage and made a comparative study with other models. They conducted a comprehensive literature review on cumulative damage theories from 1945 to 1992. The presented approaches included linear damage rules, nonlinear damage curve and two-stage linearization methods, the life curve modification techniques, crack growth procedures, the methods are based on continuum failure mechanics models, and energy-based theories.

In a cumulative damage model, the numbers of cycles under different levels of stresses must be defined. Lee et al. (2005) discussed the method to calculate the number of cycles from a specified dynamic load with respect to time. Many researchers adopted the rain-flow method to calculate the cycles (Dietz et al. 1998, Stichel et al. 1998, Luo et al. 1998, Han et al. 2013, and Li et al. 2015). In a rain-flow method, the number of cycles is defined by following the procedure of counting the number of fatigue cycles present in a load-time history and used to extract their respective range and mean (Endo et al. 1968).

When a numerical simulation such as finite element analysis (FEA) is applied to determine dynamic loads, it turns into a challenge to define appropriate boundary conditions in a FEA model (Dietz et al. 1998). It is true when relative motions occur at contacts subjected to the dynamic loads. Moreover, the uncertainties involve in contact surface increase the complexity of boundary conditions. Liao (2011) proposed an alternative method called the inertia relief analysis technique to determine boundary conditions. Unfortunately, this technique was applicable only to the unconstrained structures where the external forces and moments were determined at the equilibrium states of objects. The dynamic load related to the motion can be calculated by the multi-body simulation (MBS), an is MBS allowed to generate the realistic loads for static and fatigue analyses of components.

Normally, a cumulative failure model is developed within the time domain. However, the history of dynamic stress can be analyzed in a frequency domain (Fatemi et al. 1998, Cui 2002). Younesian et al. (2009) compared different methods to predict fatigue life in the time and frequency domains, respectively. In evaluating fatigue life in a frequency domain, the power spectral density (PSD) function of stress was obtained using the FE software, and the fatigue life was estimated by applying the Rayleigh technique on PSD. It was found that the cumulative damage model in the time domain was more conservative than that in the frequency domain; but the frequency domain simplified the fatigue analysis for a dynamic load with random vibrations. Since the dynamic loads of objects were determined though numerical simulation, the more conservative method in the time domain was preferred.

2.2 Characterization of Dynamic Loads

A dynamic load refers to a load that is subjected to change over the time. When an object is in motion, a dynamic load counts for all types of forces including external, inertial, friction, damping forces. In contrast, a static load refers to a load on a structure whose amplitude and direction remain constant over the time. An external load with slow change can be simplified as a static load (Kurowski 2005). The dynamic load must be determined to evaluate the variables for fatigue analysis. These variables include mean and alternating stresses, as well as their dependences on time factors. In characterization of a dynamic load, the time is viewed as a reference variable, and the damage growth depends on both of time factor and stress level (Dattakumar et al. 2017).

Many researchers used the experiments to identify dynamic loads of structures or products. For examples, Redfield et al. (2012) established a procedure to evaluate dynamic loads and they developed a dynamic model for the testing of wheel drums. Marques et al. (2014) evaluated the dynamic loads and performed the fatigue failure on a railway bridge; they used the seismographs that were fixed with tri-axle accelerometers for measurements. An FEA model with 3D beam elements was built to analyze the dynamic behaviors of bridge. In addition, the mean stress effect caused by the weight was considered, and equivalent alternating stress was determined using the Goodman criterion. To characterize a dynamic load, Pozuelo et al. (2014) proposed to use a vertically mounted accelerometer to track the engaged force when the vehicle passed over the bumps. The accelerometer was placed at the center of gravity approximately.

Quantifying the effect of a dynamic load on a component is not a trivial task. Note that understanding a dynamic load with continuous impact forces is essential to analyze fatigue life of a part. One option to obtain the dynamic load is through the motion simulation, which can be performed using commercially available software tools such as the Motion Study in SolidWorks (Kurowski 2005). These tools allow engineers to investigate the dynamics of moving parts, so that the forces involved in mechanical components can be determined. Early computer aided tools for motion analysis can be traced back to the computer codes such as SEURBNUK, REXCO and ICECO (Kendall et al. 1980). Motion Study will help to calculate the effect of transient loads on the structure. It is desirable that an FEA package can be integrated with a motion-analysis package so that the loading conditions of an FEA model can be directly retrieved for finite element analysis. In addition, when these two types of numerical simulation tools are integrated, the impact of design variables in a product can be accessed using a parametric study to optimize product design (Kurowski, 2005). For example, ADAMS was applied to investigate the combination of most critical loading conditions in an FEA model (Bianchi et al. 2000). As a summary, the integration of motion simulation and FEA helps to build and examine virtual models of mechanical systems in a shortened time. One limitation of an FEA is that the model cannot evaluate the forces of a rigid body subjected to motion. Therefore, the motion simulation is widely used to characterize of dynamic loads. For example, Jianfeng et al. (2013) and Sha et al. (2014) used ADAMS to define dynamic loads for the fatigue analysis of a connecting rod in piston assembly. Chen et al. (2015) had studied the prediction of the fatigue life of an all-terrain vehicle when it was maneuvered on rough and uneven surfaces. Bo et al. (2011) indicated that most of the vehicle manufacturers use the dynamic simulation to evaluate loads of the components in a vehicle. In the flexible multi-body model developed by Cuadrado et al. (2004), the structural deformations were considered, which was considered as an improvement of dynamic simulation of rigid bodies. Yang et al. (2012) investigated the fatigue life of wheels whose deflections subjected to the loads were taken into consideration. Rathod et al. (2009) presented an analytical model of a multi-body railroad vehicle; it dealt with the dynamic coupling of 3D contacts with rail track, both the cases of a flexible rail track and a rigid rail track were taken into consideration.

The method of motion study was widely applied in some scenarios such as the case where a wheel experiences dynamic loads due to track joint locations and track turn out locations. In these locations, the load at contacts will be fluctuated. Besides, the issue of wheel flatting has been widely studied over the years, a flatted wheel in motion was another good example of dynamic loads over the structure. Han et al. (2017) and Vyas et al. (1992) developed the numerical method to characterize the dynamic loads of wheel-rail interaction; this was for the scenario of flatted tires in the high-speed motion. Dukkapati et al. (1999), Nielsen et al. (1995), and Baeza et al. (2006) proposed a theoretical model to predict the rail interaction of the flat induced wheel. An extended state-space vector approach was used together with the complex modal superposition for the track; the relative displacement excitation was determined using the Kalker's methods. Pieringer et al. (2014), Cai (1988), Thambiratnam et al. (2008), Chang et al. (2010), and Bian et al. (2013) used the finite element analysis structure to analyze the simply supported beams on an elastic foundation of arbitrary length. Unfortunately, the accuracy of the simulation output of the dynamic contact model was unsatisfactory in contrast to the real values in the application. Xu et al. (2016) investigated the dynamic loads of the wheel at rail turnout locations, and they obtained the dynamic loads using multi body dynamics SIMPACK. Furthermore, ANSYS were used to evaluate the stress distribution at the contact; the obtained dynamic loads were then used as external forces in an FEA model. Wu et al. (2000) investigated the dynamic behavior of railway and load under the higher-velocity motion. The dynamic load of the structure and load due to action of multi-roller carriage was identified using an FEA model. Han et al. (2017) considered the wheel with the real-time dynamics of high-speed motion. The wheel rail was involved in the dynamic interaction, and this depended on many factors such as design parameters of wheel and rail, material properties, and contact nonlinearities. They found that the impacting force at the wheel-track contact was greater than static axial load on wheel. Wen et al. (2005) used the FEA simulation to characterize dynamic behaviors of wheel at rail joint regions, and they found that the axial load of structure affected the contact forces of wheel and rail significantly in contrast to the case under a constant speed. Cai et al. (2007) developed an ANSYS implicit and LS-DYNA explicit code to solve the model, and they concluded that train speed affected contact force more than the axial force of wheel.

2.3 Numerical Simulation for Testing

The literature review for the numerical simulation is further divided in to the activities in 1) finite element analysis for numerical simulation, 2) experimental testing for validation, and 3) numerical simulation as validation.

2.3.1 Finite Element Analysis for Numerical Simulation

The FEA technique is a mathematical solution applied to engineering system; it uses a computing system to find solutions (Cook et al. 1989). To illustrate its value in engineering application, the results from FEA must be acceptable to the original problems, and they have to be obtained at a reasonable cost. With the current FEA technique, running an FEA model for a simple and well-defined problem is straightforward. However, it may face some technical challenges when the design problem become complexity.

It is easy to analyze a simple structure but to analyze a complex structure; the structure must be made (hypothetically) into the elements; these are small enough to obtain correct displacement and stress distributions. The finite element method can be dated in early 1940s, when the mathematician Courant wrote a report in the Bulletin of the American Mathematical Society. However, even though that Clough was known as the one to coin the concept of finite element analysis, it is hard to tell who invented the finite element technique. Numerous mathematicians and engineers contributed to the discretization methods in the 1940s and 1950s (Strang et al. 1975). The papers by Turner et al. (1956) and Argyris et al. (1955) were regarded as one of the important contributions to FEA. In the automotive field, the FE method was used to evaluate many structures from steering columns (Gotoh et al. 1992), car body joints (Sharman et al. 1987) to wheel products (Morita et al. 1987).

The failure of metals due to a reformed load was firstly reported by Albert (1829). There by, an important care is taken on the deformation behavior of metals in reversed loading conditions. The fatigue has been examined from many various aspects and some of the notable methods were discussed by Lee et al. (2012).

2.3.2 Experimental Testing for Validation

The experiments provide a high level of confidence of testing results. The basic settings of an application can be recreated with less time. However, if a large number of design options are evaluated and/or the operating conditions are broadly changed, the setups and operations of experiments take a long time and the cost is high (Pace 2004).

In addition, experimental results are profoundly subjective because of the likelihood of human errors since a physical testing needs to specify the varying level of each design variable (Groesser 2012). There is a high risk where a human error occurs in setting up the experiments. Design of experiments (DOE) is not a trivial task; it might lead to unrealistic situations such as unrealistic loading and boundary conditions (Balci et al. 1994). In general, an experimental method is very time-consuming process and it is very expensive in contrast to numerical simulation. A fatigue test could take several days, months, and years to complete the process. The experiments also have the limitations in providing sufficient data and guides to match testing conditions with real-world operation conditions.

2.3.3 Numerical Simulation as Verification

For products subjected to dynamic loads, FEA tools can be used together with the motion study tools. Integrating FEA with the motion study might minimize the need of costly physical prototypes and tests. Engineers can estimate and handle the complicated interactions occurring to motion, structures, actuation, and controls, and the results can be utilized to optimize product designs for review, safety, and comfort. Many software tools are commercially available for motion studies. ADAMS and SolidWorks are two of prevalent software tools for motion studies. For example, ADAMS is optimized for the dynamic simulation of large-scale multi-body systems. The methods and tools for motion study have discussed by many researchers (Nikravesh 1988, Roberson et al.1988). For example, Haug (1989) reviewed different computational methods for the motion analysis and multi body systems.

The procedures include the use of multi body simulation packages by Jalon (1995) and Shabana et al. (1989) to determine the dynamic loads, use the dynamic loads finite element model, and identify stress concentrations of components on which the dynamic forces are acting. Gang et al. (2013) and Bo et al. (2010) have studied and developed a model of connecting rod using multi body dynamic followed by FEA to solve the fatigue life of the component. Kłodowski et al. (2015) had created the human motion simulation using multi body dynamics, by which he could be able to define the dynamic knee cartilage loading, allows evaluations of stresses and strains for a problem across the whole motion cycle. Chen et al. (2013) have modeled a terrain vehicle using multi body simulation to predict the fatigue life.

After building the finite element model, the simulation can be carried to predict the stress and strain of the structure. The fatigue loading time history can be established according to the actual situation (Cheng et al. 1998, Dowling 1977, Lemaitre et al. 1979, and Mesmacque et al. 2005). In the case of the high cycles and low stress level, the stress life method is adequate; in the case of the low cycles with a high stress level, the strain life method is adequate (Lemaitre et al. 1979). In the case of multiple amplitudes of stresses, the Palmgren-Miner linear damage rule is used to calculate the damage accumulation (Kocabicak et al. 2001).

2.4 Existing Works of Fatigue Analysis

Fatigue life simulation is the most common type of numerical analysis, which is carried out on wheels to predict the life and investigate the location of stress concentration. In general, stresses in wheels are caused by dead loads by its weight and dynamic loads due to ground unevenness. There are various types of wheels varying with designs and materials. For examples, plastic wheels, composite wheels are mostly used for agriculture and medical equipment, and the wheels made of steel and aluminum material are used mostly in automobile and railway industries.

Castor wheels are majorly used for medical and agricultural equipment. Castor wheels are made of plastic, nylon and polyurethane materials. The wheel manufacturers evaluate the fatigue lives of these wheels by performing an experimental test called rolling drum test and obstacle test. Fig 2.2 shows the setup of such experiments (Mhatre et al. 2017).

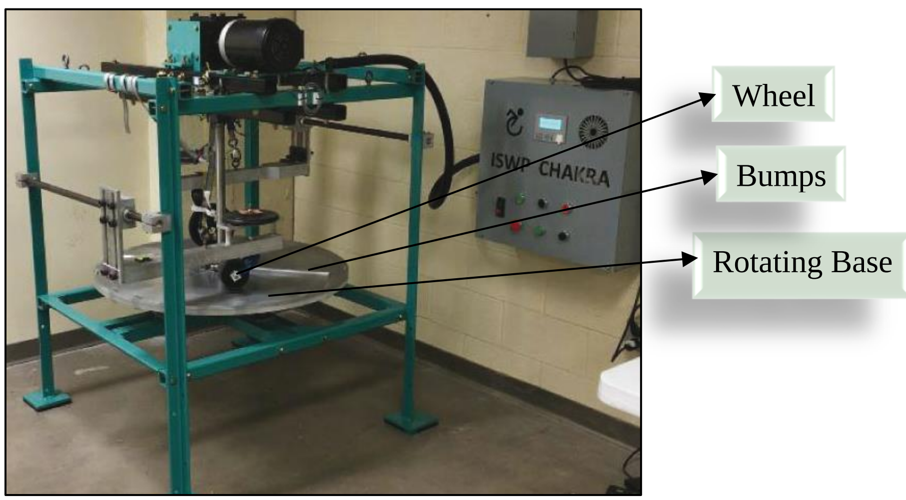


Figure 2.2 Setup of Drum Test (Mhatre et al. 2017)

The existing testing procedures of wheels used in other industries such as railways, the researchers had developed in direction of high speed and over load of locomotives (Bullet trains) by China (Xiong et al. 2015). Locomotive wheel undergoes in various types of loads such as static loads, dynamic loads, lateral loads and loads caused by low level vibrations; these dynamic loads cause overall fatigue damage to products. Yang et al. (2012) had developed the procedure of estimating the durability of a locomotive wheels subjected to random or variable amplitudes of dynamic loads, and it was combined with the multi-body dynamic simulation with finite element analysis. Liu et al. (2006) had studied the fatigue crack initiation and life prediction of locomotive wheels, finite element model was developed using Ansys software to predict fatigue damage using the stress histories of one complete wheel rotation. Patel et al. (2013) studied the fatigue occurring to wheel rail contact and focused on the initiation of damage and crack. Palaniselvam et al. (2014) studied locomotive wheel with rolling contact fatigue. Various failure modes are discussed, and simulation results were compared with the Hertz solution of stress calculation. Fatemi et al. (1988) developed a multi axial durability model for rolling contact fatigue analysis. Sraml et al. (2003), Kim (2005), and Akeel et al. (2011) used the Hertz theory to evaluate the stress distribution and treated a multi axial fatigue problem. Miyashita al (2003) explained the wheel rolling contact fatigue using the 3D wheel track interaction model. The Hertz theory was used to estimate the pressure distribution. The simulation results were compared to criteria for rolling contact fatigue life estimation by Dan Van et al. (1989). Katheriya et al. (2014) studied the design variables of axial load and the vehicle speed at rail joint. They used a 3D finite element model by Ansys and applied the dynamic loads on the structure to predict the stress at sections, and they concluded that the effect of axel load increased more von-Mises stress and maximum shear in components than the speed of train. Akeel et al. (2011) had studied the RCF damage in wheel rail contact by considering the interaction between the both wheels at the transition, straight and curved areas.

In these days, most of the investigations on the durability of wheel structures aimed at the automotive industry. Wang et al. (2009) studied the reliability of wheel durability under constant and variable amplitude of loading spectrum and obtained distance-related curves of reliability under varying road situations. Zhou et al. (2010) and Zheng et al. (2009) had solved for stress change of FE model under varying working requirements by using FE analysis and solved for crack initiation in spokes for multi axial loading conditions. Firat et al. (2010) introduced a computational procedure for the failure evaluation of metallic automotive parts. Vijayan et al. (2008) explained an FE model of a 3 -piece mining wheel. Taheri (2014) and Wei et al. (2014) had studied the durability of the wheel subjected to dynamic loads when wheels roll over obstacles and path holes. Chang et al. (2008) and Cerit et al. (2010) had performed a FE model subjected to dynamic force to representing the physical test, it led to the relation of function of time with maximum wheel displacement.

Fatigue analysis by finite element method involves in different types of loadings of wheels. The loading condition can match the real situations well; the better results can be obtained by numerical simulation. Ganesh et al. (2014) analyzed the wheels with various types of materials. Satyanarayana et al. (2012) explained the durability of wheels under radial loading conditions. Moreover, both of static and fatigue studies of aluminum alloy wheel (A 356.20) were conducted using the finite element analysis. In addition, the static stresses were identified on the specific zones with stress concentrations. Wang et al. (2013) modeled the wheel rim section using SolidWorks, to examine stresses and deformations under the conjoint impact of radial load and inflation pressure. Pan et al. (2014) and Thakare (2017) conducted the FE simulations of dynamic cornering fatigue test of the wheel with steel material. Furthermore, conducted fatigue analysis to estimate the durability of automobile wheels. Kumar et al. (2013) performed the topology optimization based on the impact test performed on the aluminum wheel. Das et al. (2014) optimized aluminum wheel for automotive usage from the perspective of weight reduction. Yadav et al. (2013) investigated the impact of camber angle on stress patterns and durability of the wheel rim subjected to the fluctuating radial loading condition.

2.5 Limitations of Existing Works

Due to the importance, numerous works have been reported on the fatigue analyses of products such as wheels, while most of relevant publications are generic, which were based on fatigue tests of rolling contacts; very few of the works are tired to certain products. Very few of research papers have discussed the dynamic loads on wheels (Yang et al. 2012), and their loading data was either from experiments or assumed artificially. It is encouraging that commercially available FEA software tools are very powerful now, and they have been widely applied to analyze motions. Therefore, we are highly motivated to integrate the motion-simulation software with the FEA package so that the need of dynamic load in fatigue analysis can be directly obtained from motion simulation. Only few articles are found in using similar strategy to conduct fatigue study with combination of motion study followed by finite element analysis. One of the articles was reported by (Yang et al. 2012): the multi body dynamics ADAMS was applied to obtain a random loading with respect to time, and the fatigue analysis was done in ANSYS.

As a summary, the following limitations have been identified from existing research works:

1. There is some limited works on fatigue analysis of machine elements incorporating their motions, in particular on polymer or metal wheels under dynamic load conditions. Few relevant numerical studies were found on caster wheels.
2. Very limited work was found on the characterization of dynamic loads thought motion simulation. Most of dynamic loads on fatigue analyses were obtained by experiments. Redfield et al. (2012) identified the disadvantages of experimental approaches such as it is more time consuming and costlier than a virtual setup.
3. No systematic approach or procedure has been developed to integrate the characterization of dynamic loads with fatigue analysis. There are limited studies

integrating the experimental way of characterizing dynamic loads with fatigue analysis but not exactly with numerical methods.

4. Limited work of using fatigue analysis by simulation have been found as an effective way to fully or partially replace experiments of fatigues tests. The limitations of fatigue tests are the previous study are related to automotive wheels and loading conditions are not relevant.

3 NUMERICAL SIMULATION

Abstract

This chapter focuses on the preparation of creating a numerical simulation for fatigue analysis. To utilize the capabilities of embedded algorithms in commercially available software tools, designers must prepare all the required data as the inputs of numerical simulation models. A numerical simulation model relies on various inputs such as part geometries and dimensions, external loads, characteristics, and materials properties. The accuracy of simulation depends greatly on the appropriateness of inputs, additional efforts are needed to collect background information about part and its application in working environment. The fatigue analysis of part relates to the dynamics of external loads, and different methods have been used to identify dynamic loads of parts. In this chapter, a systematic method of using SolidWorks Motion Simulation is proposed to evaluate the dynamic loads of parts, and the required dynamic loads are then characterized to determine mean and alternative stresses.

Keywords: motion simulation, fatigue design, design procedure.

3.1 Fatigue Design Procedure

In fatigue analysis, repeated loads are applied on a component to evaluate the lifespan of product. The fatigue analysis needs all of inputs such as materials properties, boundary conditions, external loads and their characteristics. A general procedure of fatigue analysis for a given part is shown in Fig 3.1.

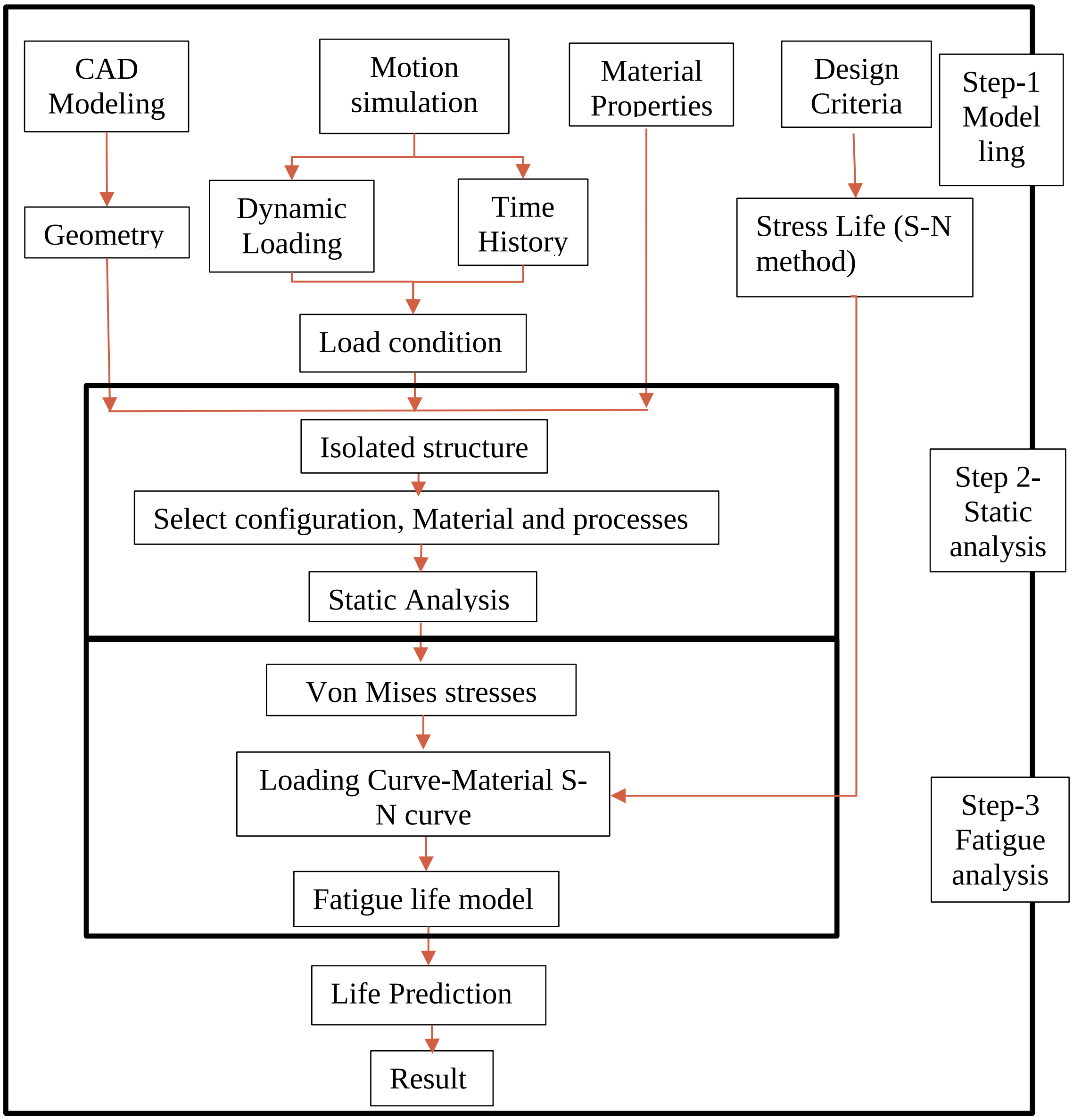


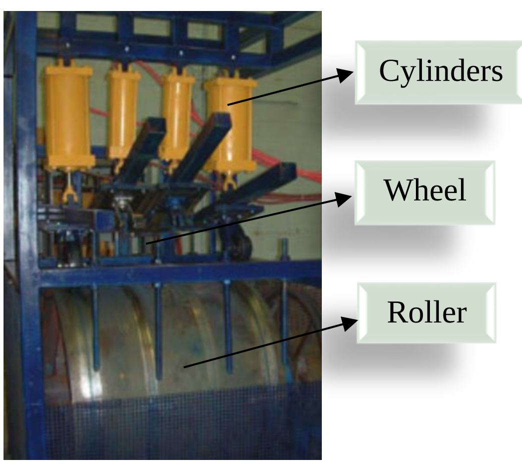
Figure 3.1. Flow Chart for Fatigue Analysis

In the procedure of finite element analysis, step one begins with the preparation of inputs for parts and its application conditions such as part geometry, load conditions, and material properties. The part models are created and assembled in the assembly interface by applying constraints. The dynamic loads are obtained from a separated motion analysis where the dynamic loads characterized using SolidWorks motion study. The motion analysis helps in kinematic and dynamic analysis of the mechanism to study the reaction forces and stress variations in wheel throughout the specified time of simulation. During dynamic analysis, special care has to be given to the higher stresses due to the inertial loads. Finally creates time history of the structure as output to create a variable loading event in fatigue analysis.

Material properties of a component characterize the response behavior of the part to external loads. The materials which are ductile in nature commonly considered as cycles $>10^6$ for operations such as steel material. S- N is the widely used method to predict the fatigue life of a component. In step two, the peak force data is extracted to evaluate the stresses in the wheel to verify the product can withstand from the applied peak loads. Static loads are applied in the global direction on the assembly and find the Von-Mises stresses, elastic strain and total deformations. From the results of static study, one can ensure that the structure can withstand applied loads. The next step is to validate the durability of part versus time. The estimated elastic stress range is utilized with S-N curve to determine the failure as per the stress level. The structure is loaded to small repeated loads that can produce cumulative failure upon the time.

In a fatigue analysis, the characterization of external loads is critical since the fatigue life for the materials at a given position depends to two main components, i.e., the mean component which is static associated with ultimate tensile strength, and the fully reverse alternating component which is dynamic associated with fatigue strength. In this thesis, wheel products are mainly discussed and used as case studies to illustrate the proposed methods for load characterization and fatigue analysis of products.

In its application, a wheel experiences dynamic force. For simplicity, it is assumed that the dynamic loads on a wheel can be obtained in a testing environment. As shown in Fig 3.2, a wheel under testing is mounted on a plate which is attached to the cylinder and due to cylinder pressure, the wheel is forced to sustain contact with drum. The drum is provided with four bumps/obstacles equally spaced and every time the wheel passes over a bump, the reaction force from the drum to the wheel is fluctuated. A dynamic load in the complicated application scenario requires an advanced evaluation technique to quantify the stress response. For the dynamic forces on wheels, there is always a non-zero mean force and the forces in case of wheel rolling over a rough surface can be purely tensile.



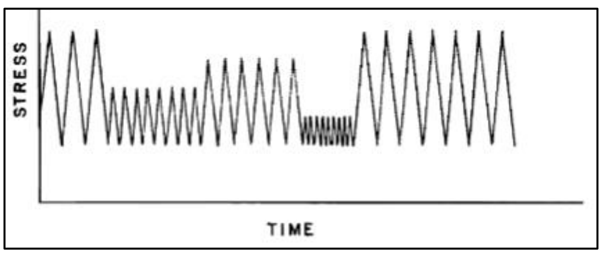
(a) Experimental Setup

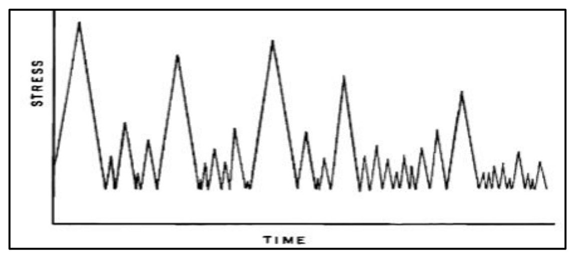
(b) Obstacle Plates Attached on Roller

Figure 3.2 Rolling Drum Test Setup (RWM Casters)

3.2 Load Characterization - Mean and Fluctuated Loads

Mean stress is an average of the maximum and minimum stresses in a motion cycle. In case of a completely reversed cycle, the fatigue strength in a fatigue test is usually obtained at the zero-mean stress. The stress ratio R is defined as the minimum stress over the max stress. R= -1 represents the constant reversed cycle. In case of variable stresses, the mean stress is not equal to zero. The most commonly used loading patterns are called block loading (Alder 1964, Fisher 1983) and random discrete loadings patterns (Albrecht et al. 1979) as shown in Fig 3.3. According to the Miner's theory, it is limited by two factors that the minor cycles are neglected for patterns when a constant minimum or mean stress is considered in the calculation.





(a) Type 1: Block Loading Pattern *(b) Type 2: Random Discrete Loading Pattern*

Figure 3.3 Types of Fatigue Loading Patterns (Kurt 1984)

Wheel rolling over a rough surface or over a roller with bumps produces the contact stress patterns as shown in Figure 3.3 (b). The mean stress effect is captured by using the rain flow counting method and the hypothesis of summation of fatigue damage by the Palmgren-Miners rule. The rain flow method helps to account for the minor cycles in the loading curve.

3.3 Motion Simulation for Characterization

A multi-body dynamic system is a system which consists of a set of solid components and links which are connected by joints. Comparing with flexible structure studied using FEA, these mechanisms are represented as the assemblies of rigid components have a few degrees of freedom. Finite element analysis cannot analyze such a mechanism because the motion analysis can move the parts without experiencing deformations. Most of the motion simulation software's have the graphical animation capability and also have the CAD/CAM integration such as SOLIDWORKS, ADAMS and Pro/Mechanica.

Motion study is performed utilizing a time-based method for rigid body kinematic and dynamic issues. The simulation begins by interpreting assembly mating circumstances into identical kinematic combinations such as universal joints, pivot joints, and sliders. And it also calculates and gives the complete quantitative information about velocity, accelerations and displacements, and the dynamics of this analysis gives joint reactions, inertial forces with in a needed range of moving mechanism. After the calculation, every component can be exported independently under the motion-induced loads. The exported component can be loaded in the FEA package as a deformable structure. In short, the motion study gives the inputs to finite element analysis with the necessary information to convert from a mechanism part to a structure. The followings are some of the properties used to simulate the assembly to extract the output force data from motion study.

Motor. A motor tool is used to define motors to move components in assembly. Rotary types of motors are used in wheel designs. The motor tool has its option to specify the constant or random revolutions per minutes (RPM).

Force. The force tool is to apply forces on an object; a force can be applied on any of the surface or edge of objects constantly thorough out the simulation time.

Spring. The spring tool defines the elastic force acting between two parts over a distance, the definition involves in selecting two different surfaces to employ a spring action between them. The definition of a spring element can replicate functions like hydraulics or suspensions.

Dampers. The damper tool specifies the force acting between components by some distance. A damper can be specified between two surfaces or components. The damping force is calculated based on the relative velocity in motion.

Gravity. The gravity tool defines the gravity force over the structure. all of the components account for the same gravitational force regardless of components mass. The gravity option can be specified in a direction by selecting a plane or a surface and can set the numeric gravity value other than the standard gravity value. Motion due to motors can be replaced by motion due to gravity.

Contact. The contact tool specifies the contact types of two surfaces. When the assembly have the components that come in contact during simulation. The parts can be grouped to prevent the penetration with each other. A frictional force can be assigned to the components, one can specify static or dynamic friction and elastic properties explicitly.

3.4 Free Body Diagrams (FBD)

When the rolling drum testing system is subjected to dynamic loads, the stress distribution on the wheel assembly is affected by all types of forces including the inertial forces and the reaction forces from other elements. It will be helpful from a free-body diagram (FBD) of the assembly, to identify the most critical loading factors related to the characterization of loading. The forces acting on the wheel are shown in Fig 3.4.

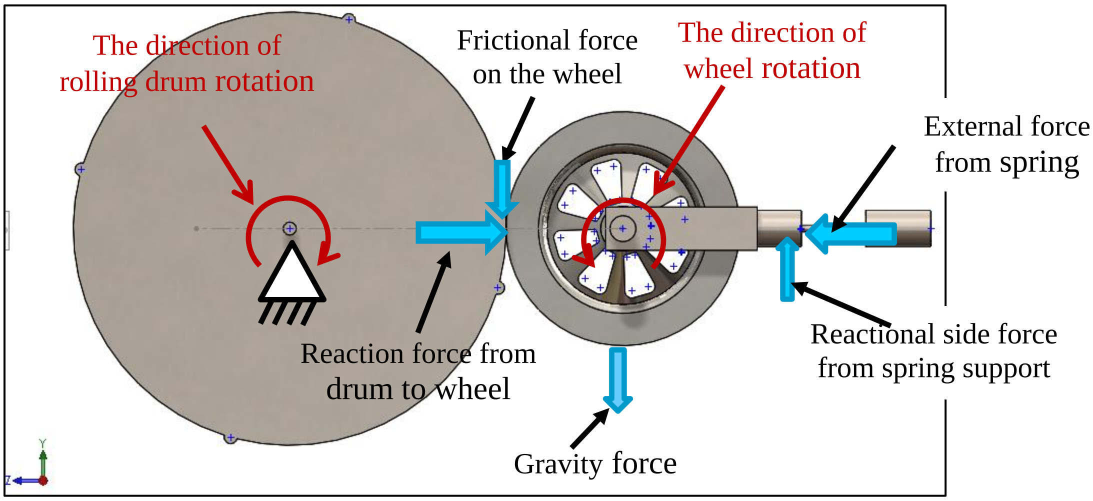


Figure 3.4 Free Body Diagram (FBD) of Wheel Assembly

It is subjected to five loads, i.e., the normal reaction force and friction force by driving rolling drum, the gravity force and external force, also the reaction side force from the spring support on the right side. Applied external force and gravity forces are constant throughout the study, and other force components will be varying when the rolling drum is in motion. Especially, wheel experiences a peak force when it rolls over a bump on the drum. A motion study will be used to obtain the historical data of dynamic loads with respect to time.

3.5 Kinematics and Dynamics of Machines

A machine is a combination of rigid bodies to transfer a motion and mechanical energy. A machine has relative motions involved in its constitutive components and a *kinematic chain* represents the assembling relations of rigid bodies by joints. A *mechanism* with a given kinematic chain is a representation of a machine without a fixed rigid body as the ground. To analyze a machine, *kinematics* concerns the relations of the motions of machine elements. *A motion* is represented by the functions of displacement, velocity, and acceleration with respect to time. While *the motions* of a machine can be described in either of a *joint space* and *task space*, *kinematics* can be classified as *forward kinematics* and *inverse kinematics*. Similar to kinematics, dynamics can be classified into forward and inverse dynamics, forward dynamics is to determine the displacement, velocity, and acceleration of the motion in task space when all of the driving forces and torques at joints are given and the inverse dynamics is to determine the required driving forces and torques at joints when the motion in the task space is pre-defined. The *motion study* of a machine is to model the kinematic and dynamic behaviors of the machine under the given application.

To characterize the dynamic loads on wheels, the rolling drum tester will be represented as a mechanical system; it will be modelled in the SolidWorks for a motion study to determine the dynamic loads under testing conditions. From the assembly interface, it allows to define any mating relations of two components as an appropriate joint. To make simulation results comparable to those from standardized fatigue tests, the virtual model of the mechanical system is based on actual testing setup, such a virtual model is shown in Fig 3.5.

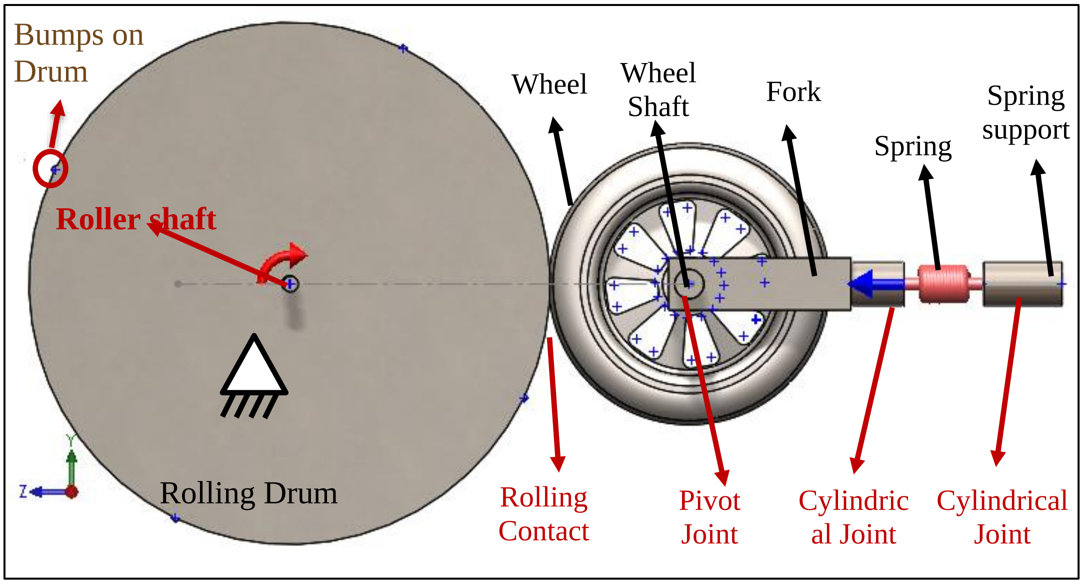


Figure 3.5 Virtual Model of Rolling Drum Tester

The roller drum is driven by a motor (an option from SolidWorks motion study) with 200 RPM (the number of revolutions per minute). An external load is applied on the surface of cylindrical joint in the direction of wheel. Note that the testing system is set horizontally, so the direction of the gravity is vertically downside. The spring is placed between the wheel assembly and the cylindrical support at the other end to ensure that the wheel always stays in contact with the rolling drum. It represents a hydraulically driven system whose force can be adjusted based on testing needs. A *contact* is applied between wheel and roller to ensure that there is no penetration between two components. To sustain a constant contact of the wheel assembly and the rolling drum, the preload of the spring force must be larger than inertial forces at any time. In determining the values of these attributes, the accelerations of components must be obtained, since these relate to inertial forces and thus to the stiffness requirements of spring. Axis of the roller shaft is constrained to stay in a same plane. After the mechanical system is modelled, a Motion Study can be created for kinematic and dynamic analysis to determine the loads on the wheel. In order to obtain the force data of one complete revolution of roller, the start and end time had to be adjusted accordingly.

From the results of the motion study, a curve of the reaction force is extracted as shown in Fig 3.6. The curve shows the force with respect to time. In the current scenario the wheel experiences a compressive force. The maximum force in the curve is around 15-KN and minimum is zero. The peak forces in the curve are due to the fact that the wheel contact with bumps on the roller. The fluctuations in the curve are due to bumps, every time the wheel rolling over a bump.

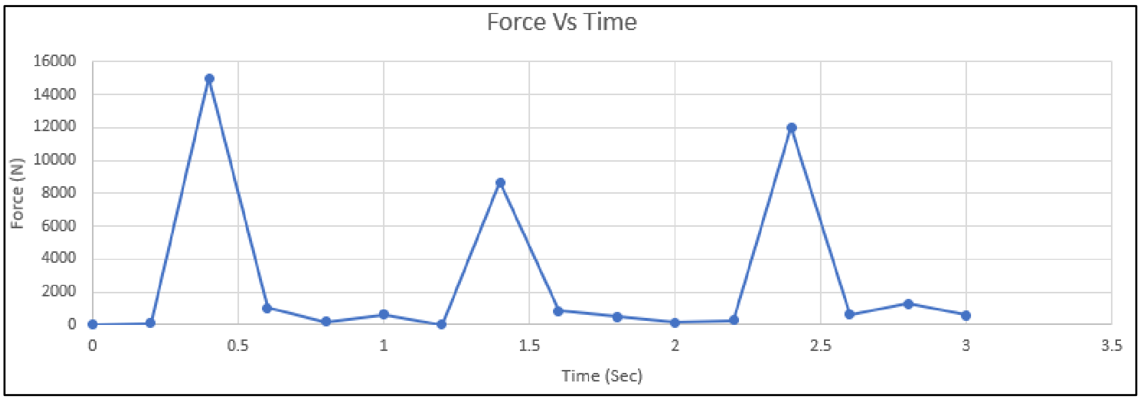


Figure 3.6 Magnitude of Reaction Force Curve

3.6 Impact of Dynamic Loads on Fatigue Damage

In case of variable amplitude of loading, the software uses the theory of collective fatigue failure study which plays a major role in estimating life of structures. The load history is then processed with the rain flow-counting algorithm (Endo et al. 1968) which helps in the simulation of fatigue data in order to reduce a spectrum of varying stress into a set of simple stress reversal (Downing et al. 1982). The software implements the method as follows,

1. Obtain the range of stress changes from the loading history.
2. Add initial and final data point similar to the data in middle if it is needed.
3. Identify the higher range and rearrange the loading pattern in such a way that higher peaks come in the first and last points.

4. From the first four peaks of 1 to 4, the rain flow cycle method is applied if the next peak is vertically smaller than first and third sections as shown in Fig 3.7.

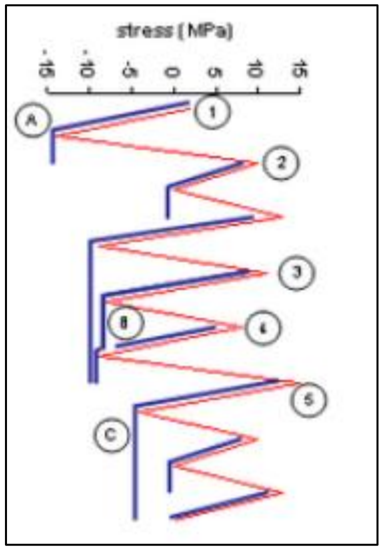


Figure 3.7 Rain Flow Counting Method (Matsuishi et al. 1968)

5. If a cycle is computed, the software begins from the initial stage of the record by avoiding peaks which are counted previously. If no peak is computed, the software inspects for the following collection of peaks (peaks 2, 3, 4, and 5) further procedure continues. At the end, every peak and every trough correspond to a Rain-flow cycle.
6. Distribute the stress amplitudes and mean stresses in several bins mention in study properties. The user can see the results from the rain flow matrix chart.

A 3D graphical representation of the rain flow matrix is shown in Fig 3.8. These graphs can be obtained from the simulation results directly if the given loading condition is variable. This method divides the mean and alternating stresses (range) in separate bins which represents the formation of the load history. The rain flow chart is a 3D histogram, in which the X and Y represents the mean load and load range. The values of vertical axis for bins represent for the number of counted cycles counted from rain flow charts and the limited or partial damage occurred based on the damage matrix chart.

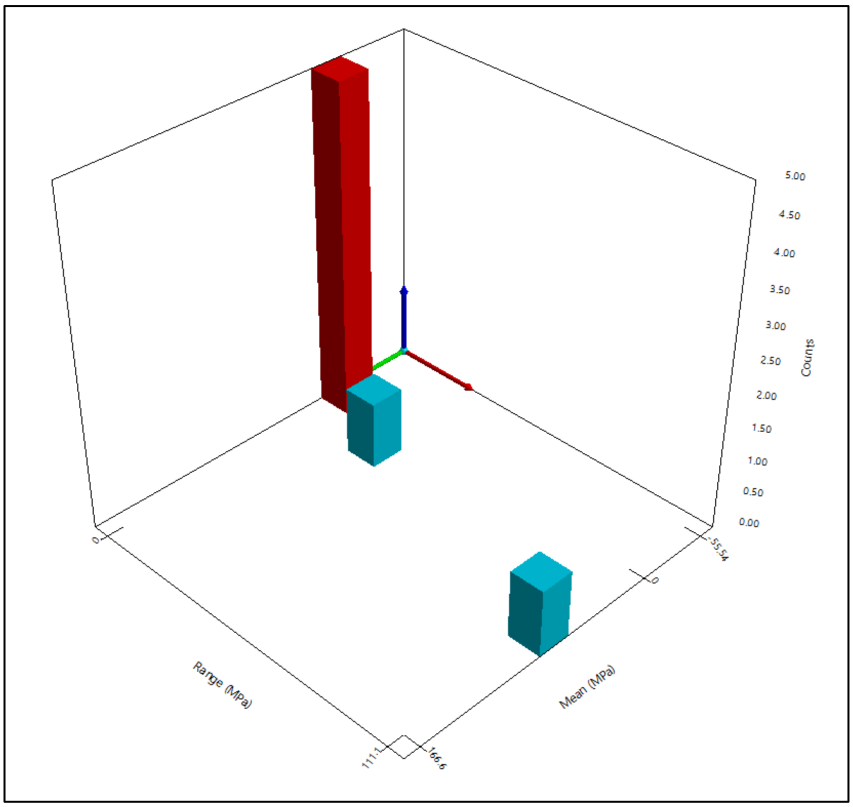


Figure 3.8 3D Rain Flow Matrix

4 NUMERICAL SIMULATION FOR STATIC AND FATIGUE ANALYSIS

Abstract

This chapter focuses on the numerical simulation approach to predict fatigue life of a part under a dynamic load and static analysis of a component under a static load. Any finite element software can be adopted as a numerical simulation tool to illustrate the procedures of static and fatigue analysis. A fatigue study requires a static analysis to determine the stress distribution for a nominal load, so that a fatigue study can be continued sequentially. In developing a fatigue analysis model, three main inputs are material properties of objects, boundary conditions, and loading condition. The software is equipped with a material library with most of commonly used engineering materials, it also includes S-N curves for popular materials such as alloy steels or carbon steels. The software provides the flexibility for users to define boundary conditions and loads via graphic user interface (GUI). In the post-processing, the software allows to export the curves in form of excel. The modelling procedure of static and fatigue analysis has been developed in detail in this chapter.

Key words: Fatigue analysis, Numerical simulation, Static analysis, S-N curve.

4.1 Case Study of Fatigue Analysis

To illustrate the procedure of using numerical simulation to predict fatigue life of wheels, the experimental setup as shown in Fig 3.2 is used as base in the modeling process. A motion study is conducted to determine the loads under testing conditions. As a result, from the motion study a reaction force curve is obtained as shown in Fig 3.6 and peak forces from this curve are noted.

A static simulation setup consists of the wheel placed on a roller vertically by representing the physical test scenario. The boundary conditions are applied as similar to the physical test setup, and the load applied from the center of the wheel hub against the roller. Wheel CAD geometry part for this simulation is modeled in SolidWorks as shown in Fig 4.1. As the wheel design is symmetrical, partial structure of the wheel design is used for the simplification of the simulation.

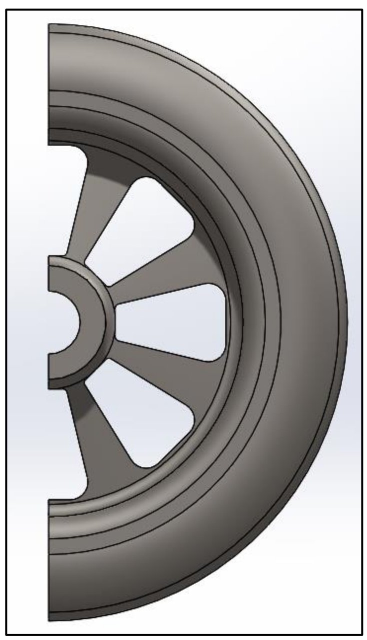


Figure 4.1 Wheel Model used for Static Analysis

Conducting a fatigue analysis starts with a static analysis subjected to nominal loads. In a static analysis, a set of nominal loads applied on the wheel against the roller, where the nominal loads can be determined as the averages of the dynamic loads over the period from motion simulation, which has been introduced in Chapter 3. According to the generic procedure of finite element analysis (FEA), the simulation model for static analysis needs to follow the steps below.

1) Import a CAD model,
2) Assign material properties,
3) Define boundary conditions and loads,
4) Create meshes,
5) Solve the boundary valued model, and
6) Post-process the results.

The fatigue analysis is further performed to predict total numbers of cycles that an object can survive without a fatigue failure. If the nominal loads are set as the maximized load in the application and the calculated maximized stress is below the yield strength of materials, one can only tell the material will not experience a static failure. However, any materials will lead to fatigue failure eventually. In design practice, if the material can survive sufficiently longer than the expected life of product one can refer such material or product has an *infinite fatigue life*. Fatigue analysis calculates the number of cycles the material at certain position can last based on the given characteristics of dynamic loads. Software provides options to compute the alternating stress for the fatigue study such as 1) Stress intensity 2) Von Mises Stress 3) Max Principal Stress, the selection depends on type of analysis. Goodman criteria is used for the mean stress correction; it defines a failure criterion by including the variable mean stresses. It is usually made by plotting mean stress versus alternating stress and it does not help to predict the life, but it shows if there is a potential for fatigue failure. Miner's rule is one of the most commonly used cumulative damage method for failures caused by fatigue. Step by step procedure for the fatigue life estimation using the S-N method is shown below:

1) Analysis of external forces acting on the structure.
2) Evaluating the stress distribution in the component upon applying peak static loads.
3) Selection S-N curve (Stress Versus No. of Cycles) from Material Library or construction of S-N curve adequate for given material.
4) Identification of the stress parameter used for the determination of the S-N curve (nominal/reference stress).
5) Identification of appropriate stress history.
6) Extraction of stress cycles (rain-flow counting) from the stress history.
7) Calculation of fatigue damage for each stress amplitude.
8) Fatigue damage summation (Miner-Palmgren hypothesis).
9) Determination of fatigue life in terms of number of stress history repetitions, the number of cycles to failure.
10) The procedure must be repeated several times if multiple stress concentrations or critical locations are found in a component or structure.

4.2 Static Analysis

The goal of this static study is to evaluate the stress distribution in the wheel for applied boundary and loading conditions. A fatigue study refers to linear or non-linear structural or to a particular-solution step from nonlinear or modal time history dynamic studies. It estimates the response of a design under the specified restraints and loads. If the analysis assumptions are observed and the computed stresses are within material strength for infinite life, it determines that the design can sustain in this environment despite how many times the load is applied. In static studies, the stress values such as Von Mises stress or Stress Intensity or Max Principle stress are used to evaluate the strength of material. The number of the cycles required for a fatigue failure to occur at a location depends on the material and the stress fluctuations. This information for a certain material, is provided by a curve called the S-N curve (Alternating Stress Versus Number of cycles).

In this case study the wheel component is analyzed for the static loads taken from the variable amplitude loading curve. The result of the static study is used as stress amplitude to perform fatigue calculations. The static load is applied on a component is to estimate how many impulsive loadings this component will sustain for the applied load. Minimum value of each factor of safety is greater than 1, therefore the part will not fail under applied load. Consequently, it is sensible to analyze this part under cyclically variable loads. Generally, in case of non-constant amplitude of loading the static stress is evaluated for 1(units depending on the time history) of load. Because the software multiplies the magnitude of maximum stress to the given time history curve.

4.3 Material Properties

Software provides a library of engineering materials whose properties are pre-defined. The majority of commonly used engineering materials, such as carbon steels and aluminum alloys, are covered in the library. In addition, the software tool provides the interface to create a custom material; the created new materials can be stored in the material library permanently. Without losing the generality, the engineering materials available from the material library are selected as materials of components in simulation.

Each solid object in a Static Analysis can be assigned with different materials. The simulation model for fatigue analysis consists of the wheel and roller. Since a stress level of a product with a long fatigue life is very low, the response of the product subjected to a low level of external load must be in the elastic range of deformation. *Table 4.1* shows that the selected material for the roller and wheel, which is linear isotropic material whose properties are same in all of the directions.

Table 4.1 Steel Material Properties (Ansys Material Library)

Material: Carbon Steel		
Properties	Value	Units
Elastic Modulus	205000	N/mm^2
Poisson’s Ratio	0.29	N/A
Mass Density	7858	Kg/m^3
Ultimate Strength	625	N/mm^2
Yield Strength	282	N/mm^2
Endurance Strength	87	N/mm^2

In fatigue study, the material properties of objects must be the same to the ones in static analysis. However, the fatigue strengths of materials must be defined which refers to the S-N curves. The performance of a material under repetitive loading is characterized by this curve. An *S-N curve* is the relation of *a fully reversed alternating stress* with respect to *number of cycles* as show in Fig 4.2. By plotting a result of structure on this log-log graph with the stress ranges on the vertical axis and the number of cycles on a horizontal axis would result a large scatter. If the alternating stress exceeds the endurance limit of the material, therefore the component can survive for finite number of cycles. Finite life has two different regions called low cycle fatigue and high cycle fatigue. Low cycle fatigue are high stress levels with low number cycles and the material behavior in the net section will be predominantly plastic. In high cycle region the stress levels are relatively low, and the material behavior is in elastic range. If the stress levels are below the endurance limit of a material, then the structure experiences a large number of load cycles without damage.

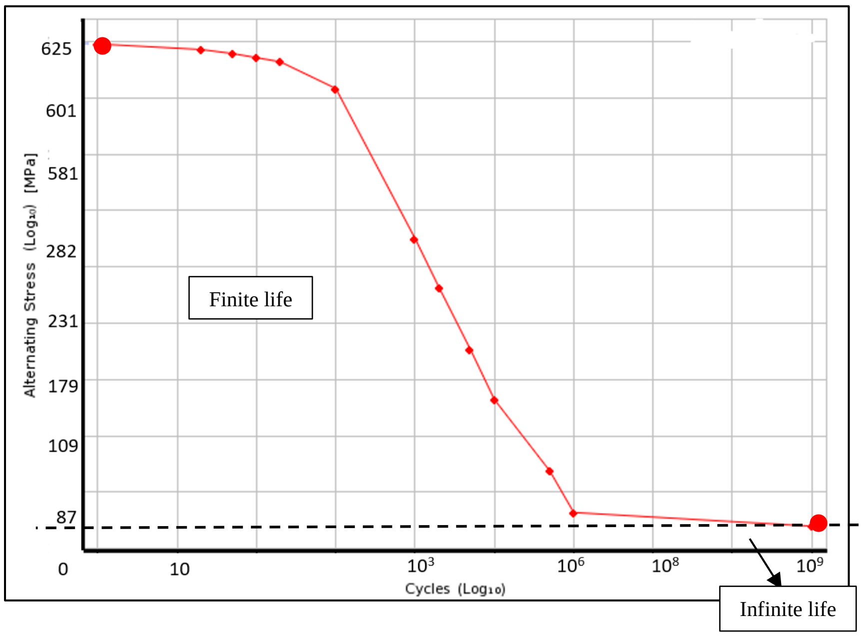


Figure 4.2 S-N Curve of the Material

4.4 Boundary Conditions and Meshing

To match the simulation model with the experimental setup up, the CAD model for the static analysis is developed. It is an assembly model consisting of the roller and the wheel. Two parts are assembled in the vertical position, i.e., the wheel is located on the top of the roller which represents the experimental scenario. The boundary conditions of the structure are defined based on the free body diagram (FBD) illustrated in Section 3.4. Note the model is simplified as shown in Fig 4.3 for effective analysis stress distribution of wheels subjected to external loads.

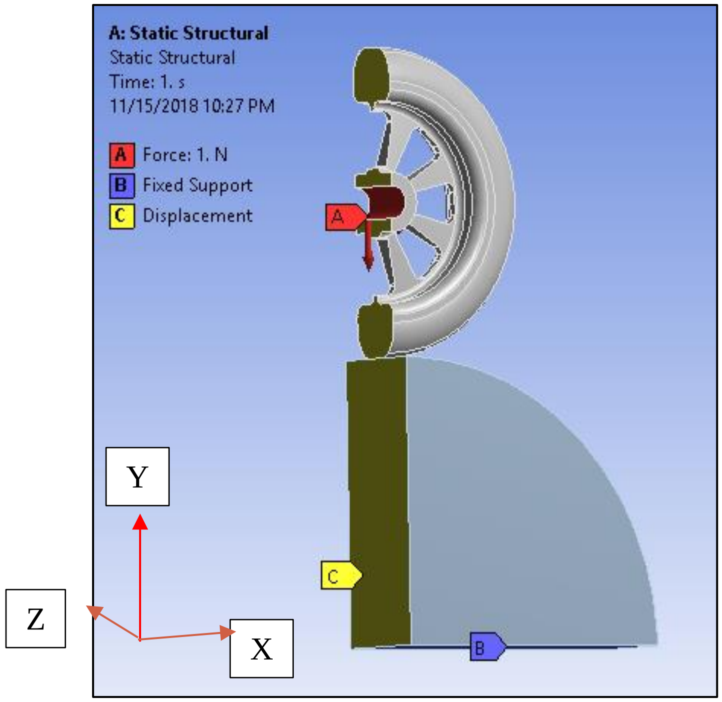


Figure 4.3 Static Analysis Setup

In defining the boundary conditions, the base of the roller is set as '**fixed'** with no mobility. A '**roller/slider**' condition is used to specify that the geometric element can move freely on the specified planar surface; its motion perpendicular to the plane is completely restricted. In this model, the wheel is fixed along X and Z translation. In defining the loading condition, the external force is applied at the center of the wheel, and the direction of the force is with the negative ***Y*** direction against roller which reflects the real-world scenario of experimental setup in Fig. 3.2. The magnitude of the load can be any value in the range of dynamic load over time. In this case study, the nominal load is set as the peak load of from the plot of the dynamic force obtained in the SolidWorks Motion Simulation.

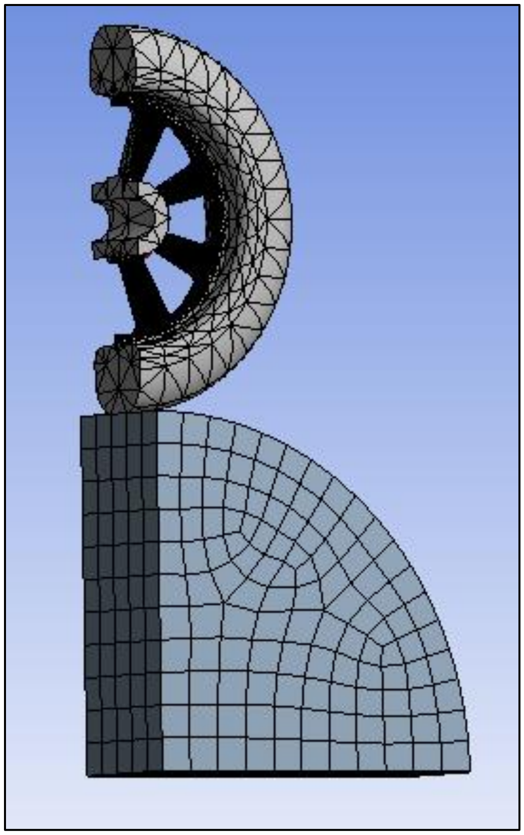


Figure 4.4 Mesh Model

A meshing process discritizes a contiual domain into discrete elements and nodes. The behavior within an element is represented by the behaviors on its associated nodes. *The meshing process* is crucial in the FE simulation in sense of *accuracy* and *computation efficiency*. In every pre processing tool there is an '**auto**' option for meshing; once it is activated, the software automatically creats the mesh with preferrable element types such as solid (3D), shell (2D) and beam (1D) elements. Moreover, the software determines the types of elements in a mesh automatically based on the geometric features of solids; from the above shown Fig 4.4 the model is automatically meshed with the commbination of tetra and hexa elements. There are three common types of solid elements , i.e., tetra, penta and hexa element. Among these 3D elements, tetrahedral elements are mostly used to mesh solid objects since tetrahedral elements are most capable to deal with the complexity of geometries.

The roller and the wheel makes one mutal contact in the assembly model. The goal of the static study is to determine the stress distribution in the wheel; while the stress distribution in the roller is secondary. As far as the geometry of a wheel is concerned, a wheel is usually not a one piece solid or solid with simple structure, it has an optimised design of weight distribution and restrengthed components, for an example, a structure with eight spokes in the wheel. The possiblity of the maximimized stress often occur to at the weakest portion of the geometry. As far as the discussed wheel with eight spokes, it is at the roots of spokes. The accuracy of the simulation results depends on the quality of the mesh. The quality of mesh rely on a number of circumstances such as '**mesh control'**, appropraiteness of '**contact condition'**, and the '**global element size'** and '**tolerence'**. In particular, the mesh control allows to select surfaces for a refined element size.

4.5 Contact Conditions

For an assembly model with multiple objects, the spatial relations of two objects must be taken into consideration. Fig. 4.5 shows that a '**bonded**' contact is assigned between the contact surfaces of the roller and wheel. As shown in the figure, the software allows one to specify two sets of contact surfaces, so that a bonded relation can be applied on the parts. With a bonded contact, the mesh can be specified as '**compatible**' or '**no compatible**'. If a compatible mesh is specified, the software merges the nodes along the interface. In addition, one side of contact surface should be specified as '**master**' surface and the other is treated as '**slave**'. In this case study, the wheel surface is specified as a 'master' and the roller surface is specified as a 'slave' surface. An incompatible bonding is dealt with in two ways. The first one is to use the simplified bonding, in this the source body is represented using its nodes, while the target is represented through the element faces. But the only elements that participate in the bonding are the ones upon which a node directly lies. The second one is to use more accurate bonding; with this algorithm the source entities use the full description of the geometry not just the nodes. This means that the edges of the source not just its nodes participate in the bonding leading to complete and accurate description of both the source and target.

Scope	
Scoping Method	Geometry Selection
Contact	3 Faces
Target	1 Face
Contact Bodies	Part 1
Target Bodies	Part 2
Definition	
Type	Bonded
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	1.4627 mm
Suppressed	No

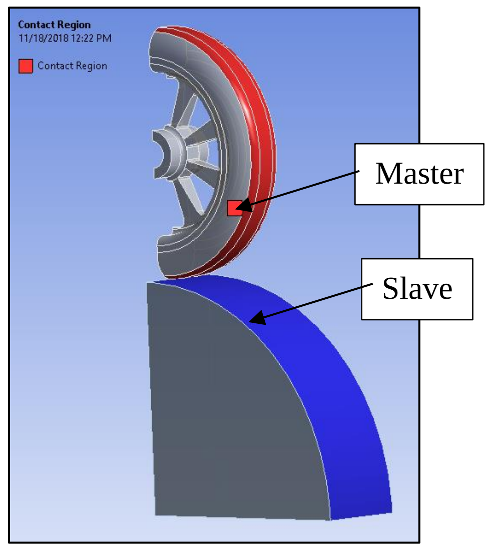


Figure 4.5 Settings to Create Contact

4.6 Mesh Independent Solution

In the solving process, one of critical issues is to determine the termination conditions where a satisfactory solution is reached. It is also referred as a *convergence* problem, i.e., how many iterations should be performed to obtain a solution with a specified tolerance of accuracy. In the majority of cases, the accuracy is mainly affected by element sizes. Therefore, the convergence can be quantified in sense of the influence of the simulation results affected by the change of average element size of model. The most common method to investigate the convergence of solving process is to generate the relations of average *von Mises stress* with respect to the mesh refinement. To plot a graph, the model at least needs two or more iterations for mesh refinements. In FEA model, there are two ways for mesh refinements are the '**h-adaptive'** method and '**p-adaptive'** method shown in table 4.2. The p-adaptive method uses high polynomial order elements in the regions where stresses are changed significantly in iteration. The h-adaptive method increases the solution accuracy just by reducing element sizes in the locations where the changes of calculated strain energy are significant in iteration; while mesh sizes at some regions where the change of the strain energy is insignificant can be increased. Either mesh refinement method has to run the solving process in multiple loops. Taking an example of using the p-adaptive method, *firstly*, calculate the strain energy of element at the initial mesh (or the mesh at previous step) and use it as a reference; *secondly*, do the same calculation for the refined mesh and compare the discrepancies of results; *thirdly*, determine it the discrepancies are within the acceptable tolerance; if they are, the solving process can be terminated. If not, one must check the number of iterations, if the number of iterations is below the maximum iterations, return to the first step for next iteration; if it reaches the maximum iterations, the solving process fails, and one must change the initial mesh for a retry. The table shown below explains the options given in motion study to set as accordingly.

Table 4.2 Adaptive Method for Convergence

Type of Setting	Explanation
Adaptive Method	H-adaptive method: Applies a mesh refinement at critical areas. P-adaptive method: Applies the mesh with higher order elements.
Target Accuracy	Allows to set the percentage of accuracy.
Maximum No. Loops	Allows to set the max number of iterations to achieve the accuracy percentage of results.

In general, finer mesh can generate a better and accurate results; in particular, it is desirable to increase mesh density at the stress concentrated areas. The software is equipped with the capabilities of using the discussed methods to refine the mesh automatically, note that it is not mandaory to use '**mesh control**' to create a fine mesh at critical locations manually. If the convergence method is employed in the simulation, the software automatically refines with the fine mesh in critical areas while the other areas remain with a global mesh size which is given initially. Fig 4.6 shows a mesh model with a combination of coarse and fine mesh.

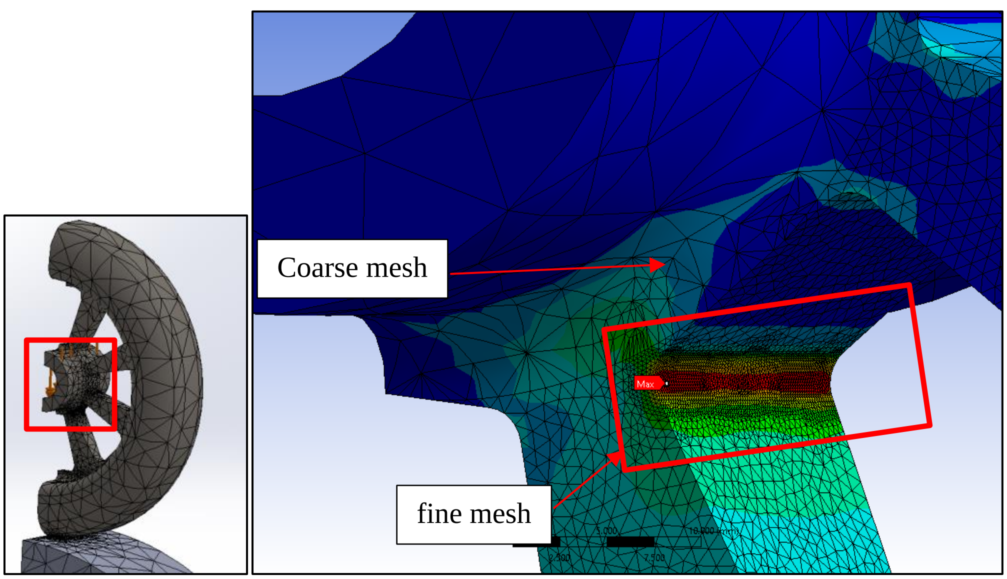


Figure 4.6 Combination of Coarse and Fine Mesh

From the results of static analysis a mesh convergence curve is generated as shown in Fig 4.7, *Y*-axis represents the Maximum stress of each iteration and X-axis represents the number of iterations. The element sizes must be continously refined until the solution has been converged with an accaptable accuracy.

- Max von Mises stress – The points in the graph represents the von Mises stress values for each iteration. If the model achieves two consucutive stress results by a acceptable difference, with in specified iterations then the model is converged.

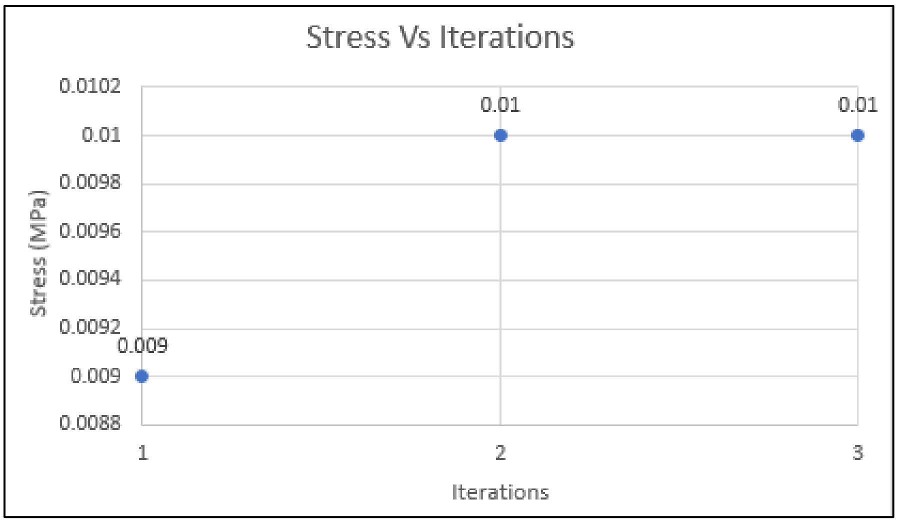


Figure 4.7 Convergence Graph for Case Study

For each iteration the mesh is refined untill the last two consucutive solution are same. From the above graph it shows that from first iteration to the second iteration the Maximum stress in the component raised 0.01MPa and from second to third iterartion the solution is not varied. At this point the results are independent of mesh, because even after refining the mesh size the results remain same.

4.7 Goodman Criteria

Mean stress is equal to the average of the maximum and minimum stress during a fatigue load cycle. Much of fatigue data is generated assuming a zero-mean stress, which means that the load cycle is completely reversed. The stress ratio, also referred to as R or R-ratio, is defined as the minimum stress over the maximum stress and is used to quantify the mean stress. A stress ratio of -1 represents a fully reversed loading. If the loading is other than fully reversed, a mean stress exists and may be accounted by using a Mean Stress Correction. There are multiple approaches to include the mean stress effect in the fatigue study. Goodman approach is one of the method to calculate a mean stress correction.

A Goodman method defines a failure criterion for a varying mean stress and it helps to include the varying mean stress to predict the fatigue life of a component. It is constructed by plotting a mean stress on x-axis and alternating stress on the y-axis. The first point of Goodman line can be plotted at a mean stress of zero and an alternating stress equal to the endurance limit. The second point is an alternating stress of zero and a mean stress equal to the ultimate tensile stress, connecting these two points with a line defines the Goodman line as shown in Fig 4.8. Any point that falls under the Goodman line will not fail in fatigue which is called as safe zone; any point that falls outside the Goodman line will eventually fail in fatigue. Goodman approach to calculate a mean stress correction to obtain an effective alternating stress that can be used with an S-N curve of stress ratio of -1.

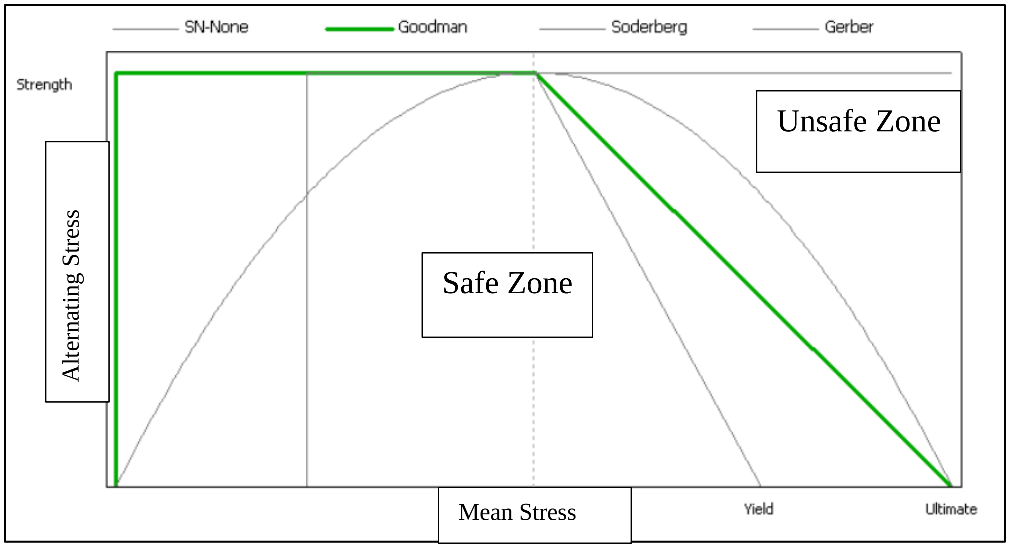


Figure 4.8 Goodman Criteria

4.8 Solving Process and Postprocessing

The complexity of a model depends on many factors such as types of elements, numbers of nodes and elements, non-linearity of material properties, and boundary conditions such as contacts. The time required by the solving process by a computer depends on the complexity of the model. It is common that a simulation needs several iterations for the mesh refinement to obtain a reasonable solution. Due to the settings of numerous parameters, one might leave these parameters as defaults at the beginning of the solving process, where the parameters for mesh refinement are set as default ones. For the mesh generation, we highly recommend using automatic where software decides the appropriate method for the simulation. After all of inputs are defined appropriately, the fatigue analysis can be performed.

- Loading Type: If the structure is experiencing a dynamic load, the fatigue loading type must be variable to predict the accurate results. For which a history data is given as input to the study.

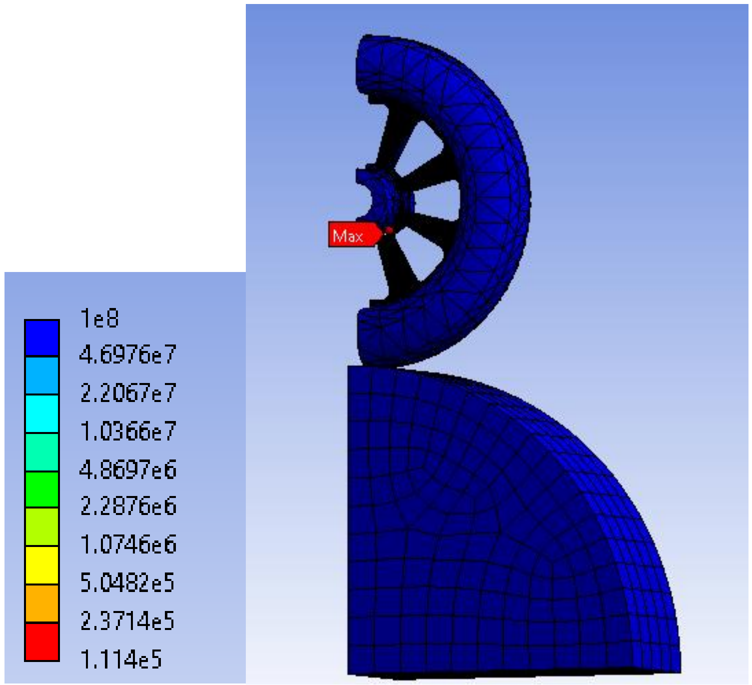
- Scale Factor: Loading Scale Factor that will scale all stresses, both alternating and mean by the specified value.
- Analysis Method: Stress Life is based on S-N curves (Stress – Cycle curves). Stress Life traditionally deals with relatively high numbers of cycles and therefore addresses High Cycle Fatigue (HCF), greater than 10^6 cycles inclusive of infinite life.
- Computing alternating stress: Provides options to compute the alternating stress for the fatigue study such as 1) Stress intensity 2) Von Mises Stress 3) Max Principal Stress, the selection depends on type of analysis.
- Infinite Life: By defining the infinite life of a model, it helps to calculate the damage by dividing the design life by available life. By setting up a higher value helps to control small alternating stresses beyond the limit of the SN curve.

Once the solution is obtained, the stress distribution and displacements in the model can be visualized. For the results of structural analysis model, three important field variables worth to examine are *von-Mises stress*, *displacement* and *strain*. In software tool, these results are stored and linked to model tree usually located on the left side of layout. One can access those results just by double click on each tab.

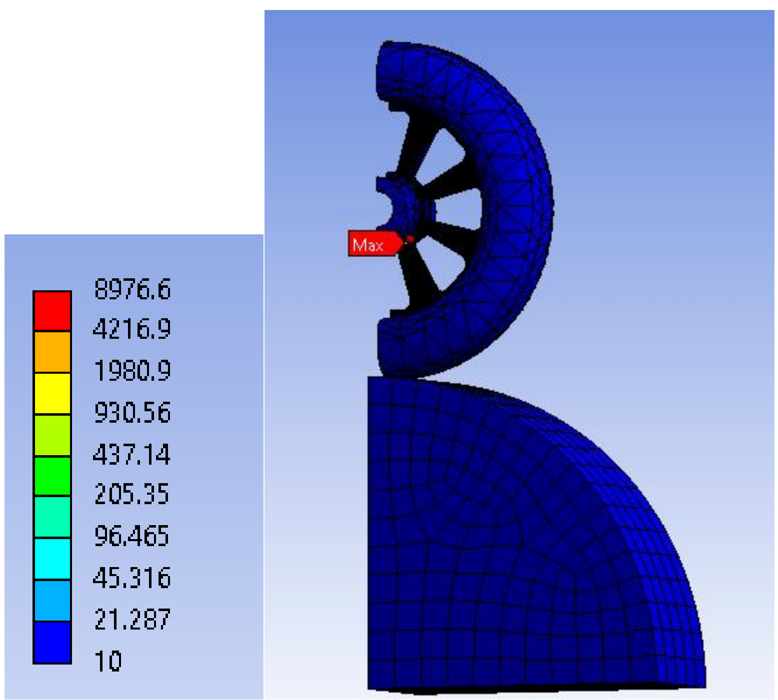
4.9 Summary

The finite element (FE) model of the wheel and roller structure was consisting of two isotropic, homogeneous and linear–elastic materials joined together along interfaces. Because of symmetry, only one half of the structure was modeled to predict the fatigue life. It has been found that the maximum von-Mises stress occurs at the expected location, which is below the yield strength of material. From the evaluated static analysis, it shows that the component can sustain for a given load.

The obtained results from the fatigue study are the number of cycles to the failure and fatigue damages over objects. Fig. 4.9 (a) shows the distributed contours of life over the part. Fig 4.9 (b) shows that the maximum damage occurs to the position with the maximized stress. And it is expected that the wheel component can survive about 1.1×10^5 cycles before failure. From the damage plot it is shown that there is a 100% damage in the model before the design life is reached. Note that the presented procedure is generic which is applicable to analyze wheel with any design and material.



(a) Life Plot



(b) Damage plot

Figure 4.9 Fatigue Simulation Results

In this chapter, a case study of wheel fatigue analysis is presented, and it is used to illustrate the procedure of fatigue analysis. Fatigue analysis is developed to estimate the durability of the materials. The details of the fatigue analysis have been discussed in Section 3.1. When using finite element method, static and fatigue analysis must be performed in a sequence. In the case study, the result of static analysis tells that the obtained maximum stress is around the wheel spoke location which is way below the yield strength of a material; which means the part will not experience static failure. However, we will provide the verification and validation to support these arguments in Chapter 5. As long as all of inputs can be defined correctly, a fatigue analysis is relatively straightforward, and the results from the fatigue analysis shows fatigue damage and fatigue life of parts. However, the obtained results will be validated, which will be discussed detail in Chapter 5.

4.10 Hertz Contact Stress

In the context of wheel design, not all industries such as agricultural, medical and manufacturing firms, use the wheels with spokes. To sustain heavy loads on wheels and to strengthen the structural integrity of the wheel, these products are made with one-piece solid body.

When any two solid and curved bodies of any radii come in to contact, the initial contact will be in form of a line or a point. The size of the contact area depends on the load and the materials of the components. With the application of the smallest load, the elastic deformation enlarges these into contact areas across which the loads are distributed as pressures. This concept was presented by Heinrich Hertz in 1881 based on some assumptions: 1) the bodies are isotropic and elastic 2) the surfaces are friction less and 3) the area of contact is small compared to bodies. The process toward the solution of this contact stresses is determining the size and shape of the contact area and the normal pressure distribution resulting in contact stresses and deformations. A schematic diagram is shown in Fig 4.10 explaining the scenario of the case study. When it is compared to the physical test setup the circular wheel is rolling against the drum, the possibility of the high stresses can be observed at the contact location which can be solved using the Hertz contact theory.

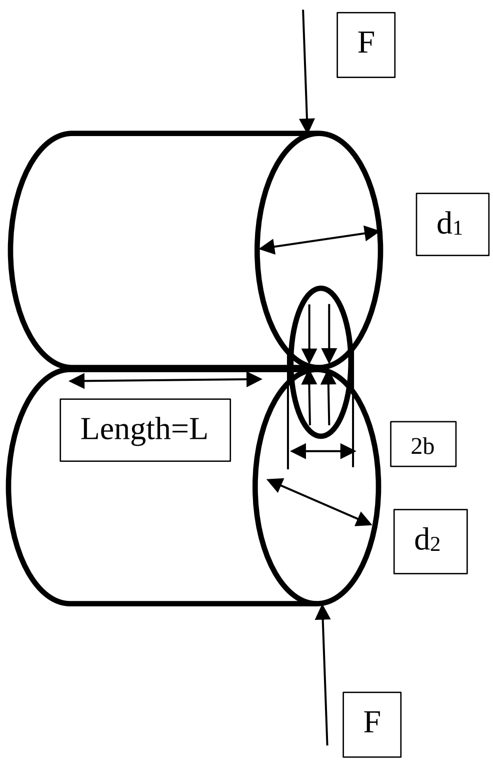


Figure 4.10 Schematic Diagram of Hertz Contact Stress Test Setup

The parameters used for the analytical model are:

Diameter (d_1) = 25 mm
Diameter (d_2) = 25 mm
Length of Contact (L) = 5 mm
Force = 10 N
2b = Width of the contact

Aluminum material properties are used for the analytical calculations (matweb.com)

E = 70000 MPa
Poisson’s ratio = 0.35

The analytical equations of the Hertz contact as follows:

$$b = \sqrt{\left(\frac{4*F\frac{(1-\mu1^2)}{E1}+\frac{(1-\mu2^2)}{E2}}{(\pi*l)\left(\frac{1}{R1}+\frac{1}{R2}\right)}\right)} \qquad (4.10)$$

$$b = \sqrt{\left(\frac{4*10\left(\frac{1-(0.35)^2}{70000}+\frac{1-(0.35)^2}{70000}\right)}{(\pi*5)\left(\frac{1}{12.5}+\frac{1}{12.5}\right)}\right)} \qquad (4.11)$$

$$2b = 0.04$$

$$P_{max} = \frac{2F}{\pi*b*L} \qquad (4.12)$$

$$P_{max} = 63.7\ MPa$$

Numerical simulation for the Hertz Contact stress is carried out using an Abaqus solver. Two cylinders are modeled with the thickness of 5 mm assembled on the same axis as to apply normal load on the cylinder. To reduce the run time of the analysis, each of the cylinder is divided in to two parts and meshed with different element sizes. A small element size is maintained at the location of contact to generate accurate result. Load is applied on a single node on top of the cylinder shown in Fig 4.11. The moment is fixed in side ways to avoid slip and the model is allowed to move only in the direction of applied force. The large portion of the bottom cylinder (at center nodes) is fixed in all 6 degrees of freedom. A bonded contact is established between two cylinders at the contact location to avoid penetration. Remaining parameters such as diameter, force, length of contact and material are used same as in analytical model.

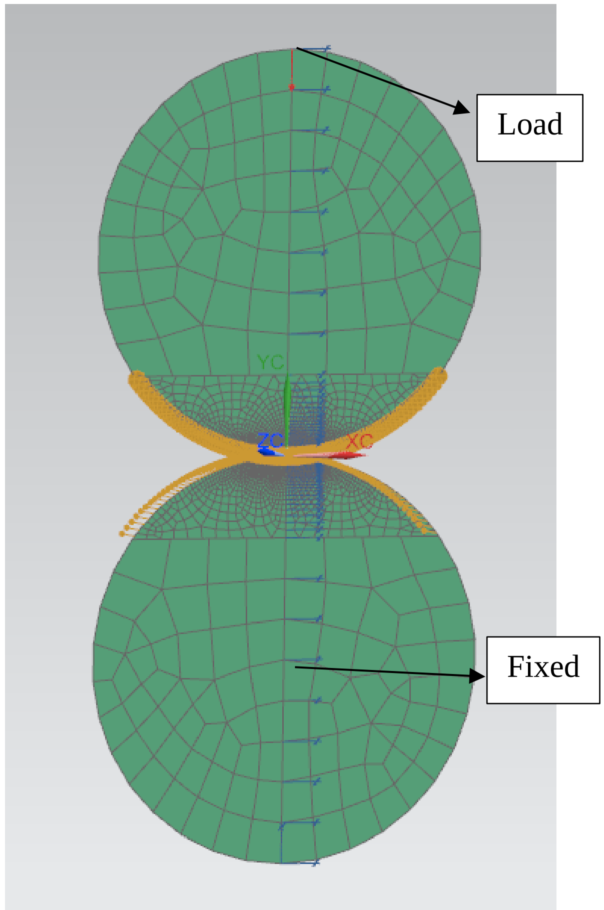


Figure 4.11 Simulation Model Setup in Abaqus

Fig 4.12 shown the counter plot for the location of maximum contact pressure. The numerical analysis, the contact pressure is 67.9 MPa. The percentage error is calculated by using equation 4.10, the difference is about 6.2% comparing the numerical with analytical solution. To ensure the acceptable solution from finite element analysis, one must be very careful in reviewing the difference in geometry, boundary conditions, magnitude of load and direction, type and size of elements. Further fatigue study can be conducted to estimate the life cycles for the obtained equivalent stress.

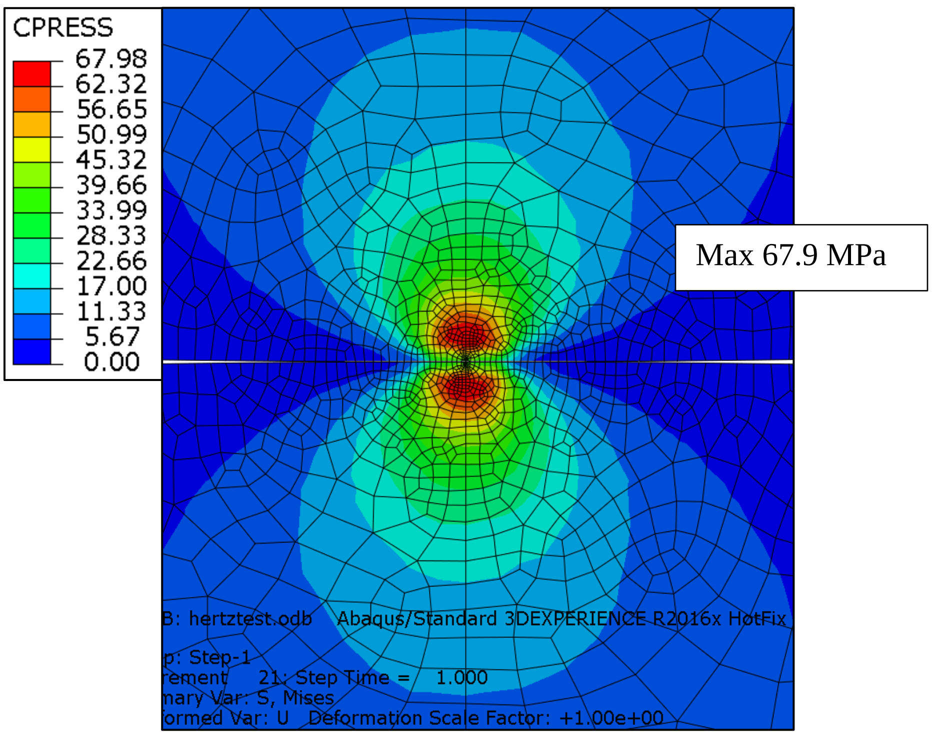


Figure 4.12 Simulation Results of Hertz Contact Analysis

5 VERIFICATION AND VALIDATION OF SIMULATION

Abstract

Numerical simulation using finite element analysis (FEM) has proven its great potentials in accelerating the development of new products, reducing design cost, and exploring more design options without prototyping and testing. However, one must ensure if a physical system is modelled adequately and satisfactory solutions can be found from the solving process. In this chapter, the V&V for the proposed simulation method is discussed. There are many different techniques for verification and validation. In general, a simulation model can be verified and validated by the comparisons with experimental results. When no physical test data is available, other alternatives can be explored. Since the client company will not be able to prepare for the testing data for V&V, some case studies validated by others from the literatures are used for our V&V purpose, this approach is said to be model to model method. The procedure for V&V of the proposed simulation is introduced, V&V is performed, and the accountability of the developed simulation method is proven.

Key Words: Verification and Validation, Model to Model correlation, Static analysis, Fatigue analysis, Numerical Simulations.

5.1 Purpose of Verification and Validation (V&V)

In general, the *verification and validation* (V&V) are involved in determining whether a model and its results can be used to obtain acceptable results for a purpose. Verification and validation are relevant but different. *Verification* is the task of determining if the implementation of a model has been done correctly, and *validation* is a task of determining if the model constructed accurately interpret the underlying real system being modeled; in other words, does the method lead to the right solution to a real-world problem? These simulation models represent the real-world system, but they are based on a number of assumptions, due to which model should be verified and validated to the extent required for the models' proposed purpose or application. If a simulation model has not been verified and validated, it is doubtful that even if the simulation plan will succeed to yield results, the results might not be the right answers to the defined problems. Verification and Validation of engineering designs are of primary importance as they directly influence production performance and ultimately define product functionality. In this chapter, verification and validation for the developed procedure of fatigue simulation will be performed, the benchmark problems from the literatures are used for V&V purpose, and the models for static analysis and fatigue analysis were verified, respectively.

5.2 Implementation of V&V

As we have discussed earlier, there are several other methods to verify and validate the model. In this thesis, an alternative method for V&V is used which refers to *model to model* method also known as a *back to back testing or docking* method. This system compares various results of the simulation model being validated to the results of other valid models. This means that output comparison between two different models with similar boundary conditions is conducted using this technique.

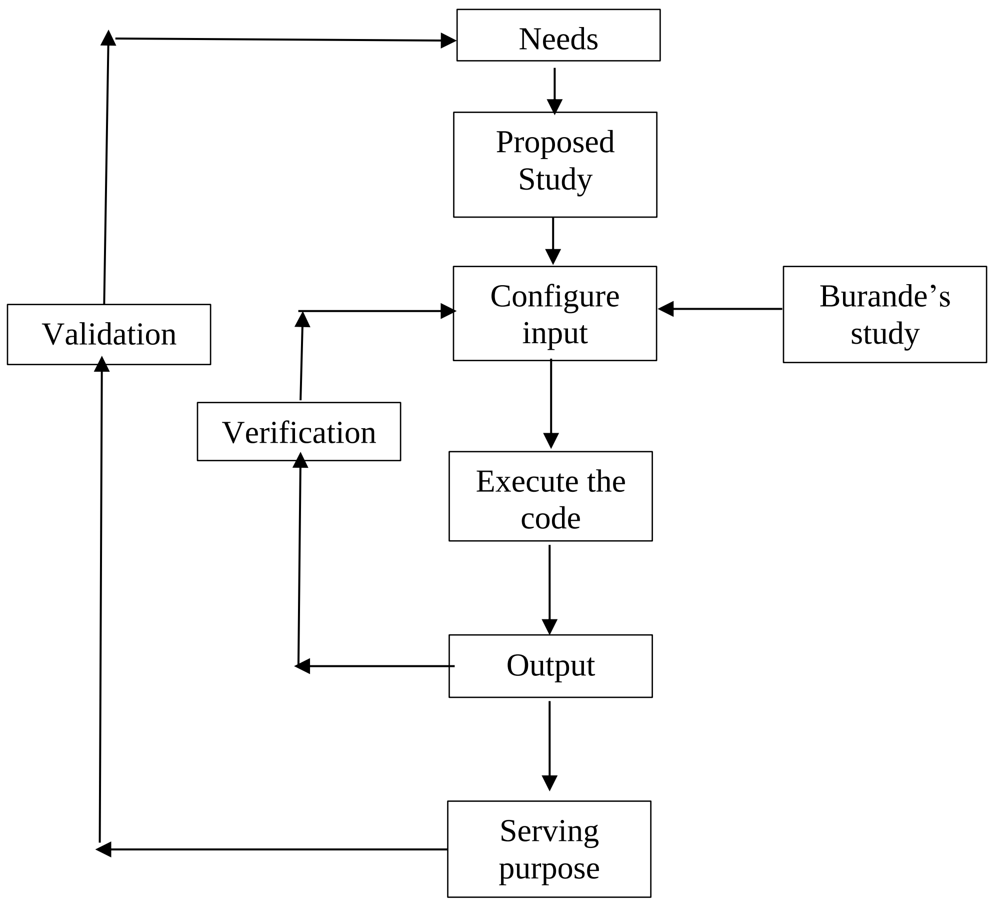


Figure 5.1 Flow Chart Representation for Verification and Validation

The contents or activities of each functional block in the flow chart are given in Table 5.1.

Tasks in V&V Implementation are-

Table 5.1 Explanation of Tasks in V&V

Tasks	**Meanings/Explanation**
Identification of Needs	The needs of simulation are to estimate the life of a structure for given boundary and loading conditions.
Proposed Study	Proposed study Procedure and results are verified with the reference model.
Input Configuration	Inputs of the current study needs to be configured, in order to correlate with the reference model.
Solving Simulation	The simulation is submitted to solve for the given boundary and loading conditions.
Output Comparison	The results are compared in order to verify the model.
Purpose Achieved	When results achieved serve the purpose, the simulation model is said to be verified.

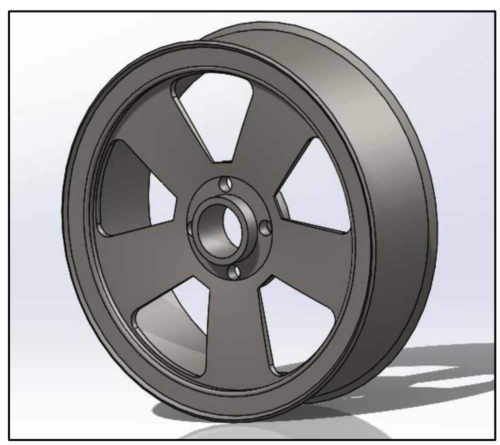
A previous study with a relevant validated simulation model is used for the purpose of verification and validation. In the process of verification, the proposed model used same inputs and boundary conditions with the existing simulation model in both cases of static and fatigue studies, respectively. The model building process is distinguished by determining its main components and their interaction. The results may widely differ depending on the configuration and input data. When the compared model is valid, the agreement within these two models can infer the validity of the model.

The method used for the V&V requires some additional simulations to be conducted in order to match with the results of the reference study. To serve for the purpose it needs a study which is closely related to actual scenario. The more complex point of how fine a distribution which is used to represents an original data scenario is part of the validation method. The major differences when comparing the simulation models are Geometric features and Material.

1) Geometrical Features

To verify the model by the selected method; a previously verified study in literatures is chosen. It should be solved by a similar procedure to obtain the results. Geometric features are one of the reasons to cause difference in output results, where the dimensions of the benchmark case are unknown. The wheel of the selected study belongs to automotive, and the wheel of the current study is used in medical and agriculture industry. The differences in shape are shown in Fig 5.2 below. The boundary and loading conditions are configured to match with the previous study, and the wheel geometry is modified by diameter, outer surface area and wheel spoke count to closely match with the Burande's model. Both models are constrained from the center hub location of the wheel and the wheel spokes in the reference model are placed on one side edge of the outer rim connecting to the center hub of the wheel, which makes a greater change in stress distribution in the wheel.

(a) CAD Model of Burande's Study



(b) CAD Model of Present Study

Figure 5.2 Comparison of Geometrical Models

2) Materials

Material can play a major role in obtaining a difference in output results; as the properties of the materials vary by many factors. The material used in the reference simulation model is aluminum

Al 356.2 and the current study is carbon steel. The wheel used in the current study belongs to castor industry, the most common materials used are steel, plastic and composite materials. As discussed previously, most of the manufacturers relay on experimental studies to test the durability of the wheels, so it is a difficult to obtain a similar study to verify the current simulation model. For the purpose of the verification, the material of the present model is changed Al 356.2 to match with the previous study.

5.3 Verification and Validation of Benchmark Case

The selected benchmark case for the verification and validation is by Burande et al (2016) fatigue simulation of Aluminum Wheel for Passenger Car under Radial Load. This study has been verified by experimental test. As per radial fatigue test standards, setup consists of a driven rotatable drum as shown in Fig 5.3. The axis of the drum and test wheel are parallel to each other, which gives a smooth surface broader than the section width of the loaded test wheel section width. The loading is provided by the test wheel normal to the drum surface in line radially with the center of drum and wheel. The wheel is secured to the center hub using nuts and bolts with an appropriate torque mentioned by the manufacturer. The vehicle weight is supported with a vertical reaction force from the base surface into the wheel. This load continuously squeezes the wheel in the radial direction while the vehicle is moving, the radial load shifts to a cyclic load with the rotation of the wheel. As compared with the current study, which is similar in most of the aspects; where the load is applied on the wheel in the horizontal axis in the current study.

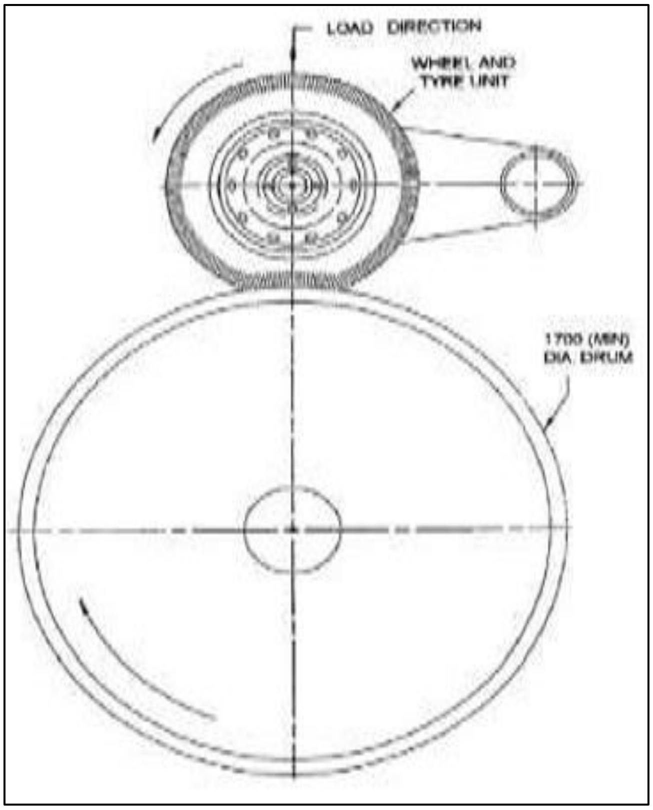


Figure 5.3 Schematic Diagram of a Test Setup (Burande et al. 2016)

The results comparison with numerical and experimental test results of Burande's model are shown in Table 5.2. In comparison of total fatigue life of a wheel, the results from the physical test shows 8.15×10^5 and the results from the numerical simulation shows 7.99×10^5. The difference in results is very small, as it is known fact that numerical analysis is always an apporaximation but not accurate.

Table 5.2 Comparison of Experimental and FE Results (Burande et al. 2016)

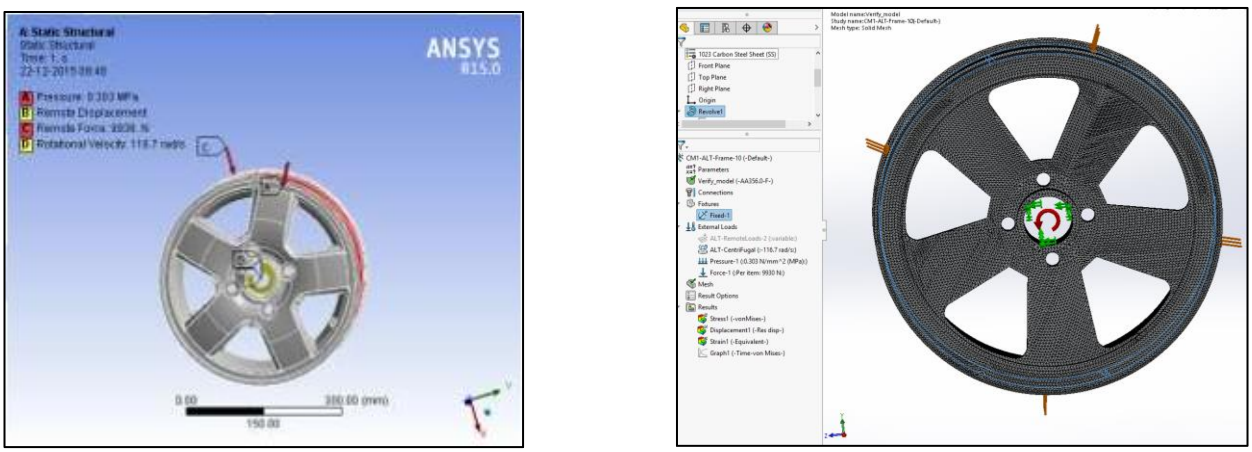
Material A 356.2	Fatigue Life Cycles
Simulation Results	Radial Fatigue Life- 7.99×10^5 Cycles
Experimental Results	Experimental Found- 8.15×10^5 Cycles

From the numerical analysis of the benchmark case, alloy wheels are intended to use in passenger cars stipulated to two types of loading, which are dynamic cornering and radial fatigue test. In this particular case study, the wheel is evaluated for the radial fatigue test. FE method is used to evaluate the performance of wheels over their life. The reference paper is aimed to perform and compare the numerical results of a wheel between two different materials which are Al 7075-T6 and Al 356.2. SolidWorks is used to model the wheel geometry in 3D and further FE analysis is conducted. As compared to the numerical procedure of the present study to estimate the maximum stress in the wheel and procedure to predict the fatigue life are similar. At first the static analysis is conducted to evaluate maximum stress in wheel, and the loading conditions are determined by the force and pressure due to vehicle weight on wheel against the ground. The procedure followed in the present study is quite similar, static study is conducted from the peak loads obtained from the motion study under dynamic loads and fatigue analysis is performed to estimate the life of a wheel.

5.3.1 Static Analysis

In this model, SolidWorks is used to build the assembly and carry out the finite element analysis. In comparison with the geometrical features, the spokes in the wheel model are modified. The wheel model is isolated from the assembly and all the loading and boundary conditions are configured to match with the benchmark case. The loading is changed as the radial load, which is applied on the outer surface of the wheel. Center hub location of the wheel is fully constrained with zero degrees of freedom. Angular velocity is applied at the center of wheel hub, as they are being aimed for the similar purpose. Ansys and SolidWorks both use tetrahedral elements as their primary element type. H-adaptive method is implemented for the analysis; it refines the mesh at concentrated locations as per the given number loops. The model is fixed at center hub location of the wheel. The comparison of the models with boundary conditions are shown in Fig 5.4.

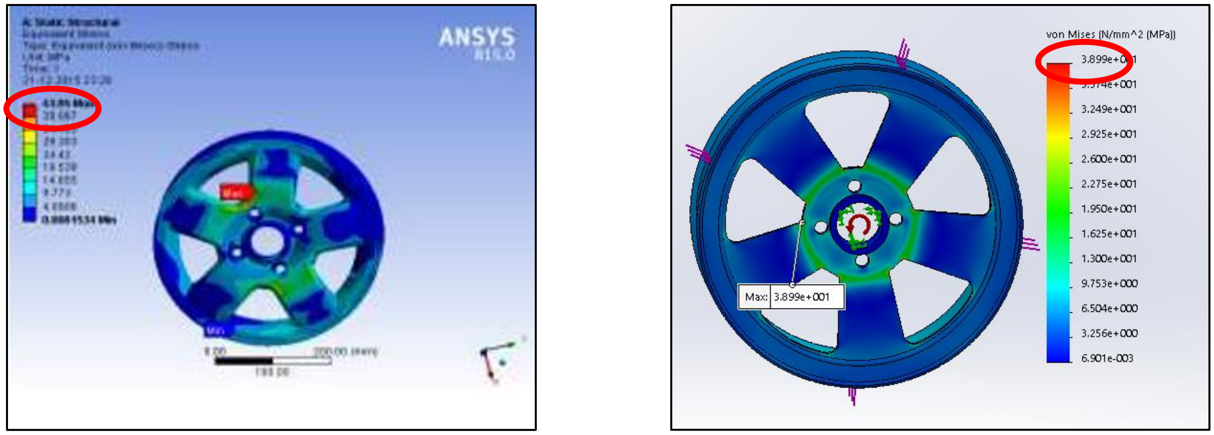
The details of boundary conditions and loading conditions are-
Pressure = 0.303 MPa,
Angular velocity = 116.7 rad/sec and
Remote force = 9.93 KN.



(a) Burande's Model *(b) Present Model*

Figure 5.4 Boundary and Loading Conditions

From the results of reference model with material Al 356.2 shown that 43.9 MPa of the maximum stress at the spoke location shown in Fig 5.5(a). From the results of static analysis, for the present study, the maximum stress obtained is 38.9 MPa at the spoke location as shown in Fig 5.5(b).



(a) *Burande's Model* (b) *Present Model*

Figure 5.5 Von Mises Stress Comparison

The results from the static analysis show the difference in von Mises stress compared with Burande's model and present models. There are various factors that affect the change in the results such as the change in geometric features and the volume of the component. However, the stress distribution or pattern in both models looks alike and the maximum stress is observed in the wheel spokes; which are connected to the wheel center hub. In comparison, the difference in results of Burande's model and present model is about 11.3% for static analysis. This percentage of difference is acceptable, and the model is said to be verified as per the static analysis.

5.3.2 Fatigue Analysis

The Ansys workbench is user-friendly software with well suitable options for fatigue studies. The fatigue analysis of the reference model is based on S-N method, the curves for the model are shown in Fig 5.6. The boundary conditions in the model are used similar to the static study. Equivalent von Mises stress obtained from the static study are used to compute the alternating stress for the fatigue study.

In the present model, modeling of CAD geometry and FE analysis are carried out in SolidWorks software. The study deals with a constant amplitude loading based on S-N method. The wheel is isolated from the assembly to conduct static and fatigue analysis, the boundary and loading conditions such as magnitude and direction of loads and location of constraints in the model are similar to the reference model. A solid tetrahedral element is used to mesh; same number of elements and nodes used in static analysis are also used in fatigue study, it means there is no need to generate a new mesh. Similar material properties, S-N curve data and maximum number of cycles are used in the present study to correlate with Burande's model. An S-N curve (Stress Versus Number of cycles) is plot of the magnitude of an alternating stress to the number of cycles to failure of a material. These curves are different for each material and are created from the experimental testing. In this plot if the stress levels are below endurance limit of a material, infinite no. of cycles can be applied without a failure. And if the alternating stresses are below the yield strength of a material which comes under the elastic region are said to be high cycle fatigue and in case of stresses above yield are likely to come under low cycle fatigue.

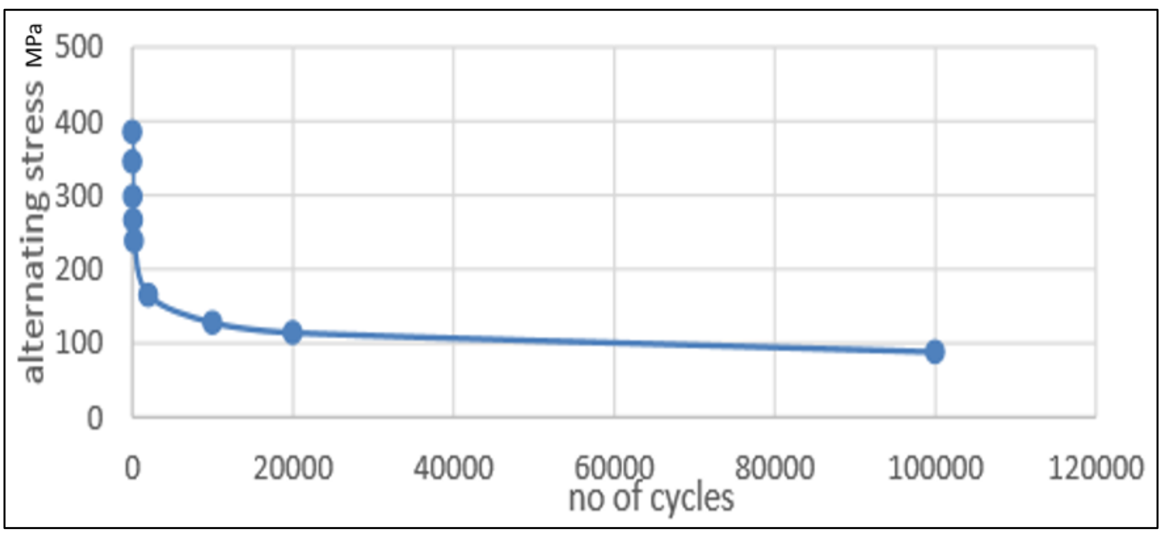
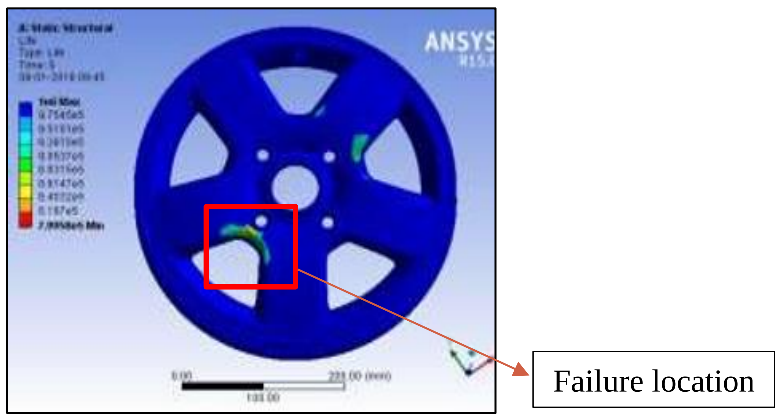
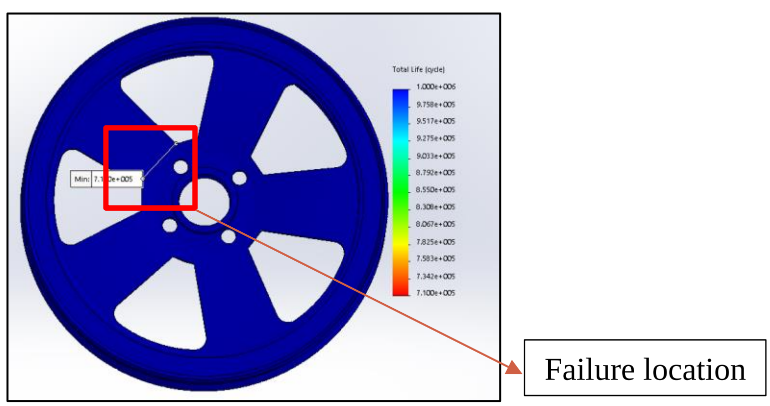


Figure 5.6 S-N Curve for the Material (Burande et al. 2016)

From the obtained results of the present study, comparison of simulation model is shown in Fig 5.7 and 5.8, which represents total life plot and damage plot respectively. In correlation with the results of both models, the total life of Burande's model is shown as 7.9×10^{5} and present model results shows 7.1×10^{5} life cycles before failure. The initiation of failures in both models are from the wheel spokes location. In comparison of both models the error percentage is about 11.1%, there is a negligible difference in total cycles to the failure. However, the stress distribution in the wheel is similar.



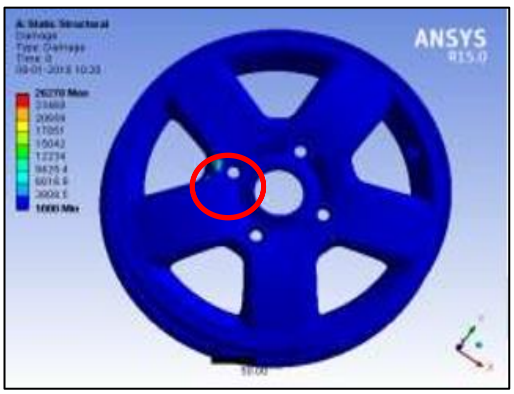
(a) Burande's Model

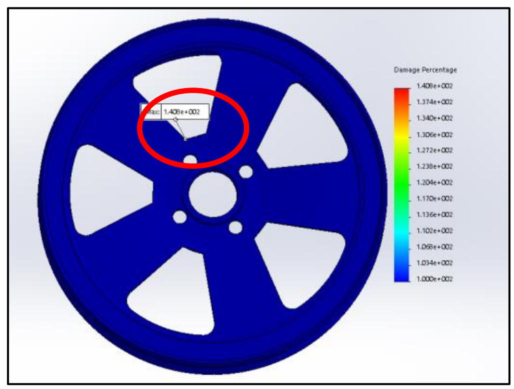


(b) Present Model

Figure 5.7 Fatigue Life Comparison

A comparison of damage in both previous and present model are shown in Fig 5.8. Maximum precentage of damage in the model are at wheel spoke locaction as it is expected due to weaker location of the component. The Burande's model is verified and validated as per the experimental procedure as shown in Table 5.2; and the numerical results of present study are being compared to already verified and validated model. After the comparison of present model behaviour with Burande's model, the results obtained are very similar including the stress distribution in component.





(a) Burande's *Model* *(b) Present Model*

Figure 5.8 Fatigue Damage Comparison

5.4 Summary

Multiple number of simulations are conducted in order to verify and validate the present study. From the obtained results it can be said that the model is verified because the von Mises stress results from the static analysis of a Burande's study and the present study are 43.9 MPa and 38.9 MPa respectively. The small difference in results can be neglected as it is a comparative study from model to model which has a difference in the geometric features, though the boundary and loading conditions are modified to match with Burande's simulation model. The results obtained from the fatigue study are quite alike, in comparison of both results the wheel model show failure initiated from the similar location of the component. As we have discussed earlier verification of model is mandatory to determine the procedure followed for the present study is accurate and the obtained results are significant. The error percentage is summarized in the following Table 5.3.

Table 5.3 Summary of Results

	Burande’s Model	Present Model	Error %
Static Analysis	43 MPa	38 MPa	11.3
Fatigue Analysis	7.9×10^5 Cycles	7.1×10^5 Cycles	11.1

In this process, the static analysis is conducted by configuring the inputs and the obtained results are compared with the reference model. Observed a difference in output results, which is due to several factors as discussed. It can be said that the model is verified although the obtained results are not closely matching but the procedure for solving the problem is identical. However, we observe the stress distribution in both wheels is similar. There are no experimental data or results available for the present study, the information given by the collaborated company (Customs Engineered Wheels); the rolling bump test results data is not available for any of the latest wheel models. For that reason, model to model method is used for the purpose is verification and validation. Selected reference model is verified and validated by the experimental methods.

In general, verification deals with the right way of solving the equations. The process FEA is always approximate and it is not robust, small errors in modeling or simulation set up leads to very large errors in results. As the present verification is conducted by model to model method the process includes few difficulties such as comparison of geometry, the volume of the components, material properties, element types, loading conditions, and boundary conditions. In verification of FE analysis, one of the important issues is to show the convergence with respect to mesh density;

it states that the simulation results are independent of an increase in mesh density using output variables. This means that the result is independent of mesh size and numerically correct.

6 PARAMETRIC STUDY

Abstract

In this chapter, the feasibility of using the simulation for analysis of a variety of products is concerned, and the procedure for creating and running parametric studies are discussed. The impact of design parameters and dimensional variations on performances of wheel products is investigated. The case studies of parametric studies are developed to understand the impact of specified design variables of a structure under certain loads and illustrate how a structure can be optimized using the numerical simulation-based approach. In case studies, design criteria are made on the highest stress concentration in a wheel product, and the geometric features of a product have compared each other with the maximized von-Mises stress. Materials of product are selected as the second design variable.

Key Words: Computer Aided Modeling, Parametric study, SolidWorks, FEA.

6.1 Parametric Study

A ***Parametric Study*** is used to optimize the design of product based on a number of numerical simulations using a virtual model. For example, a typical *design objective* is to minimize the weight of a part or assembly with some design criteria such as specifying allowable stresses and displacements in a computational domain. Incorporating a parametric study tool with an automated algorithm of design space definition allows the software tool to evaluate and optimize design variables. In *defining a parametric study,* one or more design parameters can be selected in the attempt for design optimization. Each design parameter in a parametric study is defined by a nominal parameter and a number or range of alternatives. In addition, a set of analysis goals such as maximized stress and strain should be defined. In this chapter, the feasibility of using a parametric study is investigated for wheel products from the perspective of material selection and geometric variants. In *executing a parametric study*, every combination of the specified parameter values of design variables is then analyzed to evaluate the properties of solids and the distribution of stresses and displacements. All the attributes in a model such as geometric dimensions, material properties, loads, and boundary conditions can be selected as design variables. A parametric study is performed using geometric features and material of the wheel component.

The study conducted using modeling and simulation for product design has many advantages such as,

- Modeling and simulation have an advantage of low cost and low-risk environment.
- Modeling and simulation enable the designer to determine the accuracy and performance of a design before the system is actually created. Therefore, one can investigate the benefits of alternative designs without making a real system.

- By analyzing the impact of particular design decisions while in the modeling stage rather than building stage, the overall value of developing the system decreases significantly.
- It allows the engineers to study the difficulties and problems at different stages.

The parametric study in this chapter emphasizes on the feasibility of using simulation for design of castor wheels for client company, i.e., Customs Engineered Wheels. This is one of the major manufacturers in castor wheel products. These wheels are used as components in many different products in industrial and commercial sectors, medical applications, mobility, garden and lawn cares, wastes and recycling, and light weight vehicles for material handling. The materials and geometric features of wheels can be varied widely for different designs. Most of the manufacturers customize wheels and make them by injection molding, Fig. 6.1 shows a few of examples of customized products at the client company. By using the proposed method of simulation, the wheels with any type of materials can be evaluated systematically to evaluate the fatigue life of a wheel. In addition, a parametric study is defined to take into consideration of material options. Note that this study was performed in SolidWorks and the design goal was set as the min von-Mises stress.

(a) *(b)* *(c)*

Figure 6.1 Examples of Customized Wheels from Injection Modelling

As discussed above, these wheels used in various industries, the loads on these wheels depend on the purpose it is used. In designing a wheel, one critical criterion is how long the part will survive without a failure. The design of the wheel should emphasize the fatigue resistance, so that a robotic system can sustain both of static and dynamic loads for high amplitude loads. In particular, the durability of the castor wheels is more important as these wheels experience a dynamic load. There is a scope of conducting various parametric studies, based on the design and material of wheels.

6.2 Procedure of Parametric Study

A parametric study allows to select a set of the parameters so that the impact of the selected parameter can be evaluated. Therefore, the first step is to determine one or a set of design parameters and varying ranges to be investigated. In addition, a finite element analysis tool is normally provided with the capabilities for simulation-based optimization. Actually, these practices initialize the trial-and-error methods of improving design variables and estimating the influence variable designs. Most commercial CAD software like (Inventor, Creo Parametric (ProE), SolidWorks, Solid Edge, NX etc.) support parametric studies. In this chapter, the parametric study is designed using SolidWorks, since the case study simulations are performed using the same code, no extra effort is needed to export files and also makes it easy to compare the results with actual results. Fig. 6.2, the procedure of defining and running a parametric study is explained in the flowchart.

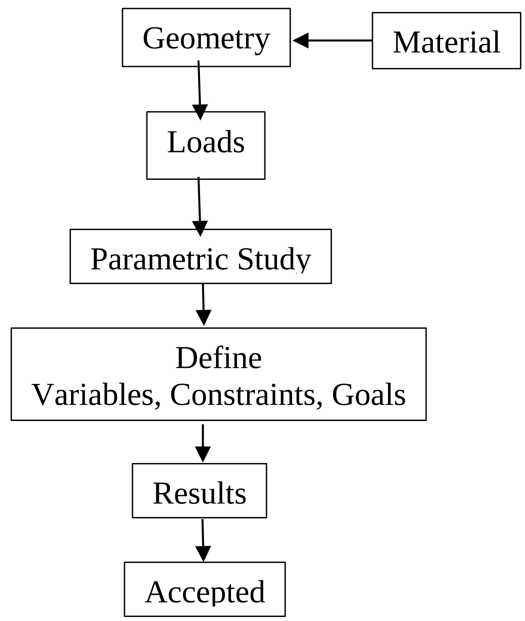


Figure 6.2 Procedure of Defining and Running a Parametric Study

In general, an optimization problem can be defined as follows. A design problem should be defined with the following elements:

- *Variables* are some attributes of a product model such as geometric dimensions, materials and some parameters related to a simulation model.
- *Constraints* are the conditions where all of the design variables and corresponding model must satisfy such constraints.
- *Goals* need to be defined when the simulation model is used for a parametric study. For example, minimizing the weight of a part can be a common goal for cost saving.

In SolidWorks, a parametric study is created by selecting a '*New Design Study*'. In specifying a parametric study, a split screen allows to select items from the list of the graphics window interactively, and the variables involved in the optimization as shown in a separated window. A user can specify design variables by selecting the names associated with geometric dimensions in the graphics window where all the design variables in the model are populated. Further, the interface allows one to specify a range for each design variable, and the range is defined by a minimum and maximum value, and the number of levels for each variable depends on with an increment for the variable. The following step of defining a parametric study is to define design constraints. In designing wheels, we are especially interested in stresses such as Von Mises stresses. Setting the design goals is optional for a parametric study. As the final step, the button '*Run*' is clicked to solve the system model for a number of combinations of design variables in sequence. And finally, the automated results report can be exported to the local drive by a single click from the same interface.

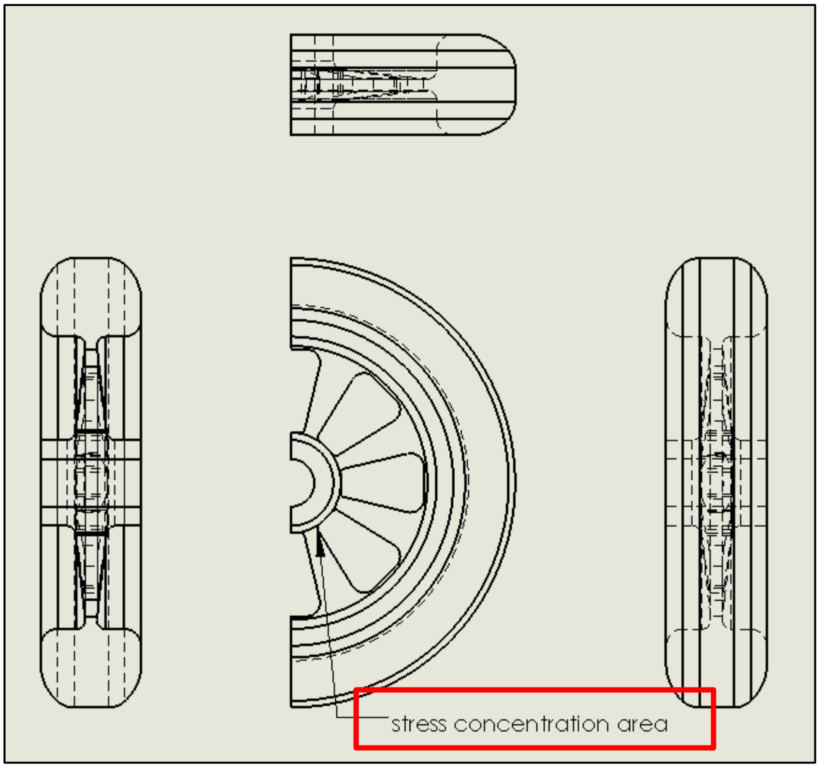
6.3 Case Studies

Two studies are conducted to illustrate the procedure of creating and running parametric studies. Two exemplifying design variables are (1) a variable for a geometric dimension, (2) the materials of a wheel design, and (3) spoke design. The design constraint for a mentioned parameter is selected as von-Mises stresses, there is no specified range mentioned but to monitor the values to supervise the impact of the selected parameters.

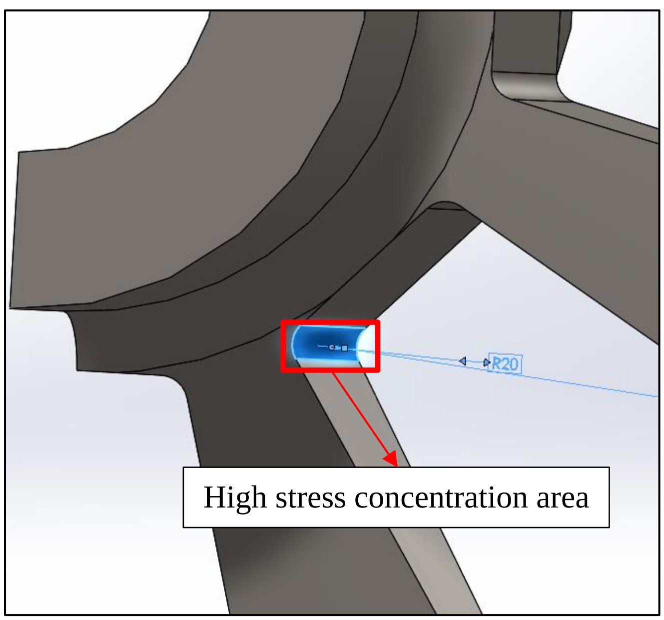
6.3.1 Parametric Study on Geometric Feature

In the proposed simulation model, one parametric study is created by considering the size of a geometric feature in wheel design, i.e., fillet size. As shown in Fig. 6.3(a), it has been found from a static analysis that the highest stress concentration occurs at the joint of sprocket and hub. Therefore, the variable for fillet size is taken as a design variable to see how it affects the stress level at this critical position.

The presented model of wheel has its weak structure on wheel spokes it is expected that max stress can occur at around the spoke. A fillet is added to reduce the failure as shown in figure 6.3 (b). The radius of the fillet also contributes the difference in magnitude of the von Mises stress distribution. The range of fillet radius is carefully adjusted to minimize the stress. In this parametric study, the radius of fillet is selected as a design variable, and its varying range is specified as 3 mm to 9 mm. The goal of this parametric study is to optimize the fillet size for a minimized stress concentration. The results show clearly that the fillet size affects the stress distribution at this critical location significantly. The maximum stress may be transferred to other locations. Note that it may be hard to identify some critical features; if a wheel design is very complex structure. But one can follow the same procedure and try a number of design variables together to narrow down critical locations.



(a) Wheel Drawing



(b) Showing Location of Fillet (Parameter)

Figure 6.3 Geometric Feature at Critical Location

From the results of the parametric study, it can be concluded that the radius of the fillet has impact on the stress distribution over the wheel. The model is scaled to observe the impact of fillet in the model, the fillet ranging from 3 mm to 9 mm with step size of 2 mm there are total of 4 scenarios. The maximum stress for each iteration is gradually decreased as the radius of the fillet is increased. It is obvious that stress reduces as the surface gets smoother. Fig 6.4 showing animation of von Mises stress plot for one of the iterations and Table 6.1 showing all the four scenarios with corresponding max von Mises stress values. By this parameter it is shown that there is a scope of reducing the failure by varying the geometric features in the model.

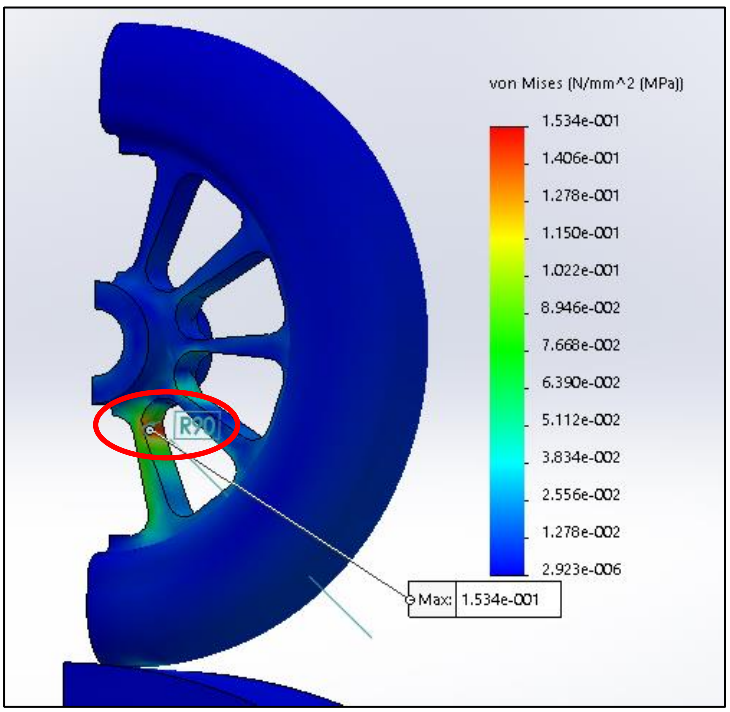


Figure 6.4 Stress Change under the Increment of Fillet Radius.

Table 6.1 Showing Scenario with Stress Values

Scenario 1	Scenario 2	Scenario 3	Scenario 4
3mm	5mm	7mm	9mm
0.20698 N/mm^2	0.18512 N/mm^2	0.16511 N/mm^2	0.15335 N/mm^2

6.3.2 Parametric Study on Materials

The second example of a parametric study is about the selection of materials, the design goal is to find a material with a minimized ratio of maximized von Mises stress and yield strength so that the fatigue life is maximized. In Section 6.2 discussed the types of wheels commonly used in castor wheel industries. Depending on the specifications of an application, the material must be selected to satisfy the given specifications adequately. From the perspective of the loading conditions, for example, the wheels used in a wheelchair must carry at least the weight of a human body as well as the self-weight of wheelchair. Some commonly used materials for wheels include Nylon, Copper, Aluminum, ABS, Steel and Titanium. In today's engineering practice, manufacturers are making their best efforts in optimizing products by minimizing the weight of components, extending the durability, and improving the performance of product. It makes sense to evaluate the impact of material selection on these aspects.

A parametric study is developed to compare the performance of product subjected to different selections of materials. The model for this study assumed that the boundary and loading conditions remain the same for all types of material selection. Moreover, the h-adaptive method is activated to ensure the convergence of the solving process.

Running the simulation for all the scenarios of the parametric study, corresponding results are shown in comparison with yield strength of a material in table 6.2. A stress distribution is compared with various strengths of the different materials, where both values are random. In general Stress-strength modeling is used to estimate the reliability of the model. Reliability is determined as the possibility that the system is strong enough to overcome the stress. In the simplistic form of the stress-strength design, a failure happens when the strength (or resistance) of the system falls under the stress. A detail comparison is shown in Table 6.2.

Table 6.2 Comparison of Stress and Yield Strength of Material

Material	Nylon	Copper	Aluminum	ABS	Steel	Titanium
Yield Strength (MPa)	60	258.6	27.5	40	206.8	744.6
Stress (MPa)	24.6	23.5	23.4	23.1	23.9	24.3

From the parametric study conducted, it is concluded that materials behavior depends on their properties and the method of simulation can be applied to a wheel with any material property. From the obtained results, shows that max stresses are below the yield strength which are in elastic state, out of all ABS shows the lowest stress. In Fig 6.5 the graph showing stress/strength of all the parameters used in the study. Titanium is shows more reliable material out of all materials. Aluminum shows a less reliable due to its max stress is very close to the yield strength of a material.

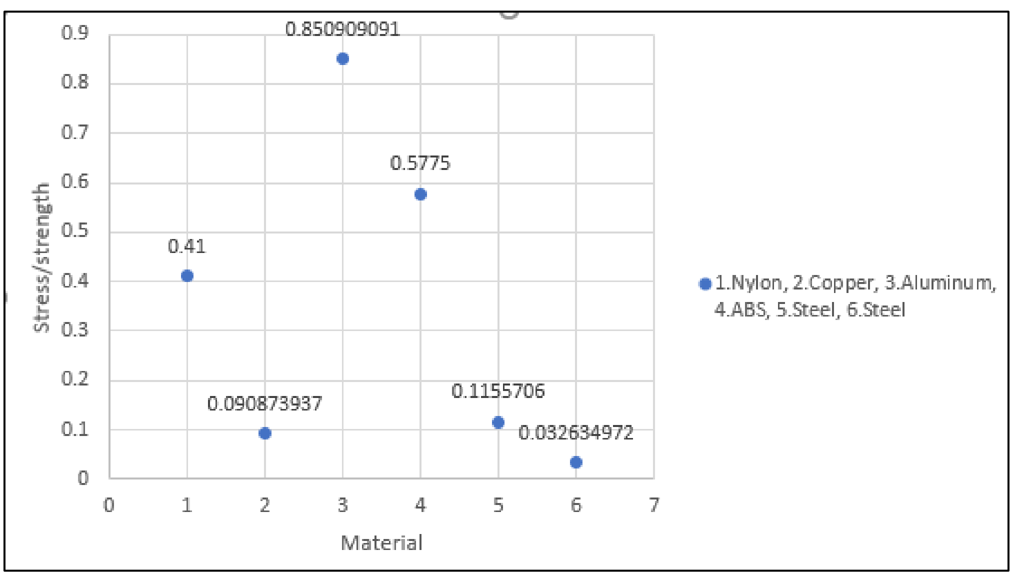
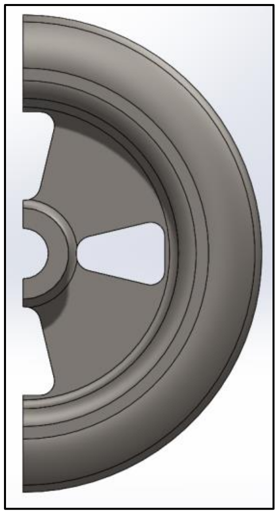


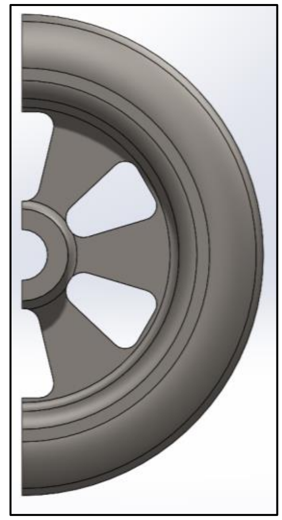
Figure 6.5 Stress Variations with Respect to Material Change

6.3.3 Parametric Study on Number of Spokes

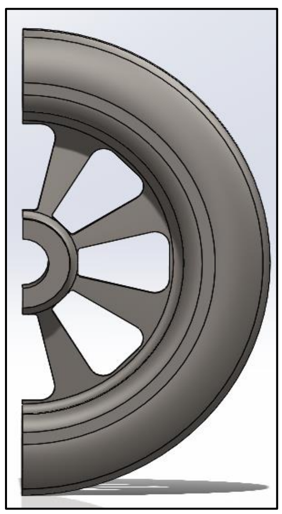
Spokes are used to strengthen the wheels with minimum amount of materials. Spokes in a wheel play a major role in carrying loads. In most of the cases, the load from shaft is applied through the center hub of wheel, while the reaction forces from the ground causes compressive loads on the spokes of wheel. The number of the spokes in wheel has a significant effect on stress distribution. Spokes can be designed differently. For example, the total number of Spokes can be odd or even number but arranged by even or uneven spacing. The third parametric study is performed to examine the impact of spokes on stress distribution. Again, the loading and boundary conditions are assumed to be fixed. The h-adaptive method is employed to ensure the convergence of the solving process. This parametric study has one design variable that is number of spokes in the wheel, which varies from 4 to 10 spokes. Ultimately this parametric study helps to optimize the number of spokes, because as the number of spokes in the wheel increases more the material from the component is reduced. From this study it can be determined that the number of spokes required for model to withstand the externally applied load. The modeling of wheel with different number of spokes are shown in Fig 6.6. It can be seen that the thickness of the spokes is reduced while the spokes count is increased. With 4 spokes in the wheel the component can with stand more loads with less failure and with 10 spokes in wheel can fail earlier with same load.



(a) 4 Spokes



(b) 6 Spokes



(c) 8 Spokes

(d) 10 Spokes

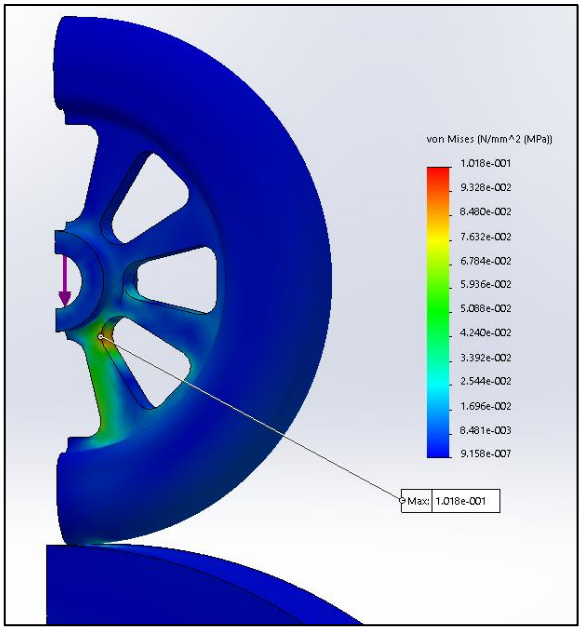
Figure 6.6 Number of Spokes in Parametric Study

As discussed above from the wheel geometry, as the number of spokes increased the thickness is reduces as shown in Table 6.3. As the spokes are in tapered shape average thickness is consider for comparison.

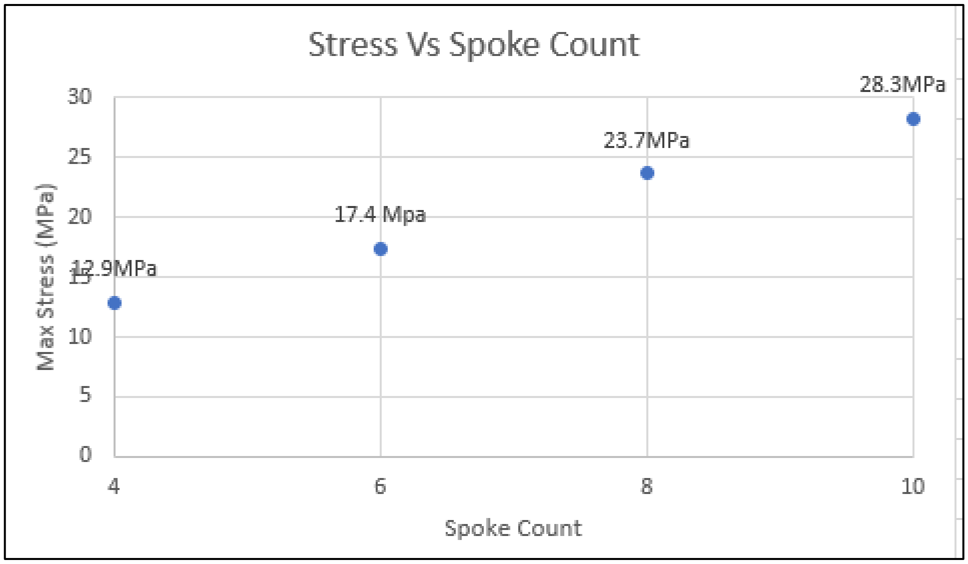
Table 6.3 Number of Wheel Spokes Corresponding Thickness

Spoke count	4	6	8	10
Average thickness (mm)	564	299	158	77
Stress (MPa)	12.9	17.4	23.7	28.3

From the obtained results there is a change in stress value for every iteration. A stress plot is shown in Fig 6.7 (a) and a graph is plotted to show the relation of the maximized von-Mises stress and the total number of spokes in wheel design is shown in Fig 6.7(b). The location of max stress occurs in wheel spoke with least thickness. It can be understood that as the spoke thickness is reduced the failure in model is increased. Ultimately this parametric study optimizes the design and helps to determine the required number of spokes for the wheel while staying with in allowable stress criteria.



(a) Stress Plot for 8 Spoke Wheel



(b) Graph with Stress Versus Spokes Count

Figure 6.7 Stress Variation with Respect to Wheel Spoke Count

6.4 Summary

A parametric study is used to optimize one or more design variables when the design constraints and goals are given. A parametric study, which allows to generate, execute and gather the results of multiple scenarios, each scenario corresponds to the set of design variables with specified values. The developed simulation models in Chapter four is flexible enough to calculate and compare the behavior of products subjected to various values of design variables. The flexibility of conducting parametric studies for product design is discussed in this chapter, and the procedure of a parametric study is discussed. Several options of the wheel design are discussed and utilized to illustrate the feasibility of the developed model for the parametric study of product design.

For the present case study, three design parameters are selected for the parametric study. The first design proposal was related to geometric feature of the wheel component. The dimensions of the one of the fillets is varied as discussed in section 6.3.1, to obtain the ideal or lowest stress in the model with varying feature dimension. The second variable for the parametric study is material of the wheel, it is selected to verify the feasibility of the simulation. The results from this study are shown in terms is stress/yield strength of the material, out of all titanium shows most reliable results for the present simulation. The third parameter for the study is selected as number of spokes in the wheel; which plays a major role in any wheel, in current parametric study the spokes count is varied from 4 to 10. As discussed in section 6.2.3 thickness of the spokes is decreased with increase in spokes count. From the obtained results for this study has variation in max stress. The max stress in the model is gradually increased with increase in number of spokes count. The results from three parametric studies are explained in Table 6.4 as below.

Table 6.4 Parameters versus Performance

Parameters	Max von-Mises Stress					
Fillet (3 mm to 9 mm) with 2 mm step	0.206 MPa	0.185 MPa	0.165 MPa	0.153 MPa	-	-
Materials (Stress/Strength) Ratio	2.4 (Nylon)	11 (Copper)	1.0 (Al)	1.7 (ABS)	8.6 (Steel)	30.6 (Ti)
Wheel spokes	4 Spokes 12.9 MPa	6 Spokes 17.4 MPa	8 Spokes 23.7 MPa	10 Spokes 28.3 MPa	-	-

7 CONCLUSION AND FUTURE WORKS

7.1 Summary of Thesis Work

This thesis aims to develop a systematic approach of using numerical simulation to (1) characterize dynamic loads, and (2) predict the fatigue lives of wheel products subjected to various dynamic loading conditions. It was motivated by the request for research and development from a client company who produces various castor wheels. In many applications of castor wheels, users need to know the strength and durability of given castor wheels. Current practice at the client company were to run numerous experiments to determine fatigue lives of wheel products. To conduct a fatigue test of a wheel product physically, it involves in a long testing time, a high cost but the limited capabilities of investigating the response of the wheel subjected to various operating conditions. Virtual analysis has its great potentials in substituting physical fatigue tests for lower costs, shortened time of analysis, higher flexibility to deal with various loading conditions, and virtual analysis allows evaluating a large number of design options for an optimized one without prototyping physically.

The goal of the thesis is achieved through the accomplishment of the following tasks.

1) **Background Study and Literature Survey**. The purpose of this task is to clarify the research needs and gain the understanding of existing works in using numerical simulation for fatigue analysis of products. The literature survey has discussed over 100 relevant works; while most of them were on the simulations of rail wheels under static loads; a few of papers discussed the procedure of characterizing dynamic loads for castor wheels, but to our knowledge, no work has been found on using numerical simulation to estimate the

durability of wheels under the dynamic loads, which is the ultimate goal of the proposed study.

2) **Systematic Approach for Fatigue Analysis**. This is the central part of the presented study, and the developed approach must be flexible and capable of predicting the fatigue life of any given wheel subjected to different dynamic loads; in particular, it is desirable to characterize dynamic loads under benchmarking experimental conditions, so that the obtained simulation results are comparable based on the same reference. The proposed approach consists of three steps with the details as follow,

- Characterization of dynamic loads on the wheel
    - This is the first step of the numerical analysis; SolidWorks motion analysis is used to characterize the dynamic loads. There is no previous work found on FE analysis of castor wheels under dynamic loads.
- FE modeling of static analysis
    - In the second step of numerical analysis, static study is used to evaluate the stress distribution in the wheel by applying loading and boundary conditions correlated to the experimental situation.
- Fatigue analysis of a wheel
    - This the final step as part of the numerical analysis, fatigue study is used to predict the life of a caster wheel by considering the results obtained from static study as mean and alternating stresses.

3) **Implementation of Numerical Simulation using commercial FEA Tool** The purpose of this task is to illustrate the procedure of FEA and the feasibility of using FEA for fatigue analysis of wheel products. Without losing the generality, the commercially available Ansys Simulation was selected as the software tool for implementation. As discussed in above section the FE analysis procedure is followed to estimate the fatigue life of a component. From the results of static analysis obtained 228.8 MPa of max stress in wheel component for a peak stress amplitude from the loading curve and 1.1×10^5 of total cycles before failure.

4) **Verification and Validation** (V&V). The purpose of V&V was to determine whether a model and its results can be used to obtain acceptable results for a particular purpose. V&V are performed by using a 'model to model' method, in this method the system compares various results of the simulation model being validated to the results of other valid models. To perform verification, we have configured the inputs of the present model to match with reference model and the model is said to be verified when the results are matched.

5) **Parametric Study**. To illustrate the flexibility of the proposed numerical approach to deal with various wheel designs subjected to a wide scope of dynamic loads. The procedure and tools to create a parametric study were developed. The study is performed using SolidWorks tool which can be initiated by 'Design study' within SolidWorks. From this study we can evaluate the influence, that varying some parameters can have on the design. The parameters can involve dimensional and material etc. Parametric analysis enables you to specify parameters for evaluation, determine the parameter limit, specify the design

constraints, and interpret the outcomes of each parameter modification. Software will organize the different parameters in combination and provides results with different scenarios. For the present study, three different design parameters are selected to explain the flexibility of the proposed numerical study.

7.2 Conclusion

The works reported in this thesis led to the conclusion that the proposed numerical simulation is capable of predicting fatigue life of a wheel product effectively and efficiently for various wheel products subjected to different loading conditions. More specifically,

1) Numerical simulation was utilized to characterize the dynamic loads under experimental conditions. The characterization of dynamic loads on the wheel when rolling over an uneven surface, the model setup for numerical analysis is same as experimental setup. The numerical simulation setup is built in SolidWorks part and assembly modeling; further model is developed to run the kinematic simulation using SolidWorks motion analysis. The reaction forces extracted from the motion study are used to determine mean and alternative loads with respect to time.

2) The fatigue analysis relied on a static analysis to determine stress distribution under normalized loads. The FE analysis is carried in Ansys Workbench. The preferable mesh type is tetra, which is default element type in Ansys FE analysis for the version we have used. H-adaptive method is implemented to ensure the convergence of the model because it provides high quality results without any investment during the solving process. Software obtains the target accuracy by estimating the change in strain energy and ends iterations when two

consecutive values vary by less than the given value. From the results of present case study, Max von Mises stress is used as alternating stress to compute the fatigue life of a wheel part.

3) The fatigue analysis was built upon static analysis. The final step in the numerical analysis is the fatigue study of the component by using the results retrieved from the static study. The alternating stress used to compute the life of a component in fatigue study and is extracted from the static results. This study conducted by S-N method (Alternating Stress Versus Number of cycles), with input of random loading curve obtained from motion study. In present case study the results from the fatigue study shows damage and life of a wheel component.

4) The procedure for a parametric study is developed and it covered the design variables of the real test situations such as geometric change in wheel design to avoid the damage, change in material type etc. The results obtained from the present case study are reasonable. Also, parametric study helps fast and cost-effective way to create robust product designs.

5) Verification and validation are conducted by comparing with already verified previous study. The model is verified by matching procedure and by achieving the similar results. It is validated for achieving the purpose of the simulation.

7.3 Future Works

The presented work has proven the feasibility of using numerical simulation for fatigue analysis of wheel products. However, some future works are demanded to make the presented method practically,

1) The simulation model can be refined by taking into consideration of other design variables, such as temperature and the nonlinearity of materials, in FEA models of numerical simulation. Durability of the components is more important when temperature is taken in to consideration.

2) A design library can be developed to include all design features of castor wheels, the selections of these features can be treated as parameters directly in FEA modelling procedure. By this design variable are easily accessible to evaluate the impact of multiple set of variables in shorter time.

3) The secondary development can be performed over the proposed simulation model, so that the inputs of new designs and the settings of an FEA model can be defined more interactively for parameterized fatigue analysis of wheel product families.

4) Some simplified tools can be developed for the prediction of fatigue life, based on the results of a number of simulations subjected to a variety of design options and loading conditions.

REFERENCES

1) Aid, A., Amrouche, A., Bachir, Bouiadjra B. (2011) Fatigue life prediction under variable loading based on a new damage model. Materials & Selection 32(1): 183–191.

2) Akeel, N. A., sajuri, Z., Ariffin, A. K. (2011). Analysis of rolling contact fatigue damage initiation at the wheel-rail interface, Australia journal of basic and applied science 5(12), 2011, pp. 937-945.

3) Albert, W. A. J. (1838) "Über Treibseile am Harz" Archiv für Mineralogie Geognosie Bergbau und Hüttenkunde, vol. 10, pp. 215-234.

4) Anderson, T. L. (1995). *Fracture Mechanics: Fundamentals and Applications*. CRC Press. ISBN 978-0849316562.

5) Argyris, J. H., Kelsey, S (1955). 'Energy Theorems and structural analysis, Aircraft Engineering, Vols. 26 & 27.

6) Baeza, L., Roda, A., Carballeira, J., Giner, E. (2006). Railway train-track dynamics for wheel flats with improved contact models. Nonlinear Dyn 45:385–397.

7) Balci. O. (1994). Validation, verification, and testing techniques throughout the life cycle of a simulation study. Annals of Operations Research, 53:121–173, 1994. doi:10.1109/WSC. 1994.717129.

8) Ballo, F., Frizzi, R., Gobbi, M., Mastinu, G., Previati, G., & Sorlini, C. (2017). Numerical and Experimental Study of Radial Impact Test of an Aluminum Wheel: Towards Industry 4.0 Virtual Process Assessment. *Volume 3: 19th International Conference on Advanced Vehicle Technologies; 14th International Conference on Design Education; 10th Frontiers in Biomedical Devices*. doi:10.1115/detc2017-67703.

9) Barhorst, A. A. (2008). Modeling Loose Joints in Elastic Structures-Variable Structure Motion Model Development. *Journal of Vibration and Control, 14*(11), 1767-1797. doi:10.1177/1077546307082917.

10) Basquin, O. H. (1910). "The exponential law of endurance test". *Proceedings of the American Society for Testing and Materials*. 10: 625–630.

11) Behroozinia, P., Khaleghian, S., Taheri, S., & Mirzaeifar, R. (2018). An investigation towards intelligent tires using finite element analysis. *International Journal of Pavement Engineering*, 1-11. doi:10.1080/10298436.2018.1475664.

12) Bertocci, G., Steireiro, J., Cooper, R., Young, T., Thomas, C. (1999). VA Research and Development. Testing and Evaluation of Wheelchair Caster Assemblies Subjected to Dynamic Crash Loading, 36(1), 32-41.

13) Bertocci, G. E. Hobson, D. A., Digges, K. H. (1996). Development of transportable wheelchair design criteria using computer crash simulation. IEEE Trans Rehabil Eng 1996;4:171-81.

14) Bian, J., Gu, Y. T., Murray, M. H. (2013). A dynamic wheel–rail impact analysis of railway track under wheel flat by finite element analysis. Vehicle System Dyn 51:784–797.

15) Bi, Z. (2018). Finite element analysis applications: A systematic and practical approach. London, San Diego: Academic Press. doi:https://doi.org/10.1016/B978-0-12-809952-0.18001-X, pp61-280.

16) Bo, W., Yue, H. Dingyun, H., Jiansong, T. (2010). Numerical simulation study on fatigue test of the connecting rod in diesel engine. Small Int. Combustion Eng. Motorcycle, 39: 47-50.

17) Budynas, R. G., & Nisbett, J. K. (2015). Shigleys mechanical engineering design. New York: McGraw-Hill. pp. 265-344.

18) Burande, D., Kazi, T. (2016). Fatigue Analysis of Alloy Wheel for Passenger Car under Radial Load. *International Journal of Engineering Research and General Science, 4*(2). Retrieved from www.ijergs.org. ISSN 2091-2730.

19) Cai, C. W., Cheung, Y.K., Chan, H. C. (1988). "Dynamic Response of Infinite Continuous Beams Subjected to a Moving Force", Journal of Sound and Vibration, Vol. 123, 461 - 472.

20) Cai, W., Wen, Z., Jin, X., Zhai, W. (2007). Dynamic stress analysis of rail joint with height difference defect using finite element method. Engineering Failure Analysis, 14(8), 1488-1499. doi: 10.1016/j.engfailanal.2007.01.007.

21) Casters and Material Handling Equipment (2018). Douglas Equipment. Retrieved April 12, 2018, from http://www.douglasequipment.com/.

22) Cera, A., Mancini, G., Leonardi, V., Bertini, L. (2008). Analysis of methodologies for fatigue calculation for railway bogie frames. 8th World Congress on Railway Research. https://www.researchgate.net/publication/267547154_Analysis_of_methodologies_for_fatigue_calculation_for_railway_bogie_frames

23) Cerit, M. (2010). Numerical simulation of dynamic side impact test for an aluminum alloy wheel. *Scientific Research and Essays, 5*, 2694-2701. Retrieved from http://www.academicjournals.org/SRE.

24) Chang, C., Wang, C., Chen B, Li L (2010). A study of a numerical analysis method for the wheel–rail wear of a heavy haul train. Proc Inst Mech Eng F J Rail Rapid Transit 224:473–482.

25) Chang, C., Yang, S. (2009). Simulation of wheel impact test using finite element method. Engineering Failure Analysis, 16(5), 1711-1719. doi: 10.1016/j.engfailanal.2008.12.010

26) Chang, K. (2015). Introduction to e-Design. https://doi.org/10.1016/C2009-0-63076-2, pp1175-1204. ISBN- 978-0-12-382038-9.

27) Chaboche, J. (1987), “Continuum damage mechanics and its application to structural lifetime prediction”, Rech. Aerosp, 4, pp37-53.

28) Cheng, G. X.Plumtree, A. (1998), a fatigue damage accumulation model based on continuum damage mechanics and ductility exhaustion. International Journal of Fatigue 20(7): 495–501.

29) Chen, J., Hwang, H. (2013). All-Terrain Vehicle Flexible Multi body Dynamic Simulation for Fatigue Prediction. Advances in Mechanical Engineering, 5, 725315, doi:10.1155/2013/725315.

30) Cook, R. D., Malleus, D. S. and Plesha, M. E. (1989). 'Concepts and Applications of Finite Element Analysis', John Wiley & Sons, Inc., New York.

31) Coffin, L. F. (1954). A study of the effects of cyclic thermal stresses on a ductile metal. Trans. ASME, 76, 931-950.

32) Correa, N., Vadillo, E. G., Santamaria, J., Blanco-Lorenzo, J. (2018). On the non-proportionality between wheel/rail contact forces and speed during wheel set passage over specific welds. Journal of Sound and Vibration, 413, 79-100. doi:10.1016/j.jsv.2017.10.031.

33) Cuadrado, J., Gutiérrez, R., Naya, M., González, M. (2004). Experimental Validation of a Flexible MBS Dynamic Formulation through Comparison between Measured and Calculated Stresses on a Prototype Car. Multi body System Dynamics, 11(2), 147-166. doi:10.1023/b: mubo.0000025413.13130.2b.

34) Cui, W. C. (2002). A state-of-the-art review on fatigue life prediction methods for metal structures. Journal of Marine Science and Technology 7(1): 43–56.

35) Dan Van, K., Griveau, B., Message, O., (1989). “On a new multi axial fatigue limit criterion: Theory and Application, in Biaxial and Multi axial Fatigue,” European Group on Fracture, EGF Publication 3, Mech. Eng. Publ., London, pp. 479-496.

36) Dattakumar, S., Ganeshan, V. (2017). Converting dynamic impact events to equivalent static loads in vehicle chassis (master's thesis). Chalmers University of Technology, Gothenburg, Sweden. ISSN number: 1652-8557.

37) Dietz, S., Netter, H. and Sachau, D. (1998). Fatigue Life Prediction of a Railway Bogie under Dynamic Loads through Simulation. Vehicle System Dynamics: International Journal of Vehicle Mechanics and Mobility, vol. 29, no. 6, pp. 385–402. ISSN 0042-3114.

38) Ding, Y., Yan, Y., Mcilhagger, R. (1995). Effect of impact and fatigue loads on the strength of plain weave carbon-epoxy composites. Journal of Materials Processing Technology, 55(2), 58-62. doi:10.1016/0924-0136(94)01785-9.

39) Dowling, N. E.; Calhoun, C.A.; Arcari, A. Mean stress effects in stress-life fatigue and the walker equation. Fatigue Fract. Eng. Mater. Struct. 2009, 32, 163–179.

40) Dowling, N.E., Brose, W.R. and Wilson, W.K., 1977, "Notched Member Fatigue Life Predictions by the Local Strain Approach", in Fatigue under Complex Loading: Analysis and Experiments, AE-6, SAE.

41) Dukkipati, R. V., Dong, R. (1999). Impact Loads due to Wheel Flats and Shells. Vehicle System Dynamics, 31(1), 1-22. doi:10.1076/vesd.31.1.1.2097.

42) Dvorak, P. (2013). A closer look at motion analysis. Retrieved April 12, 2018, from http://www.machinedesign.com/archive/closer-look-motion-analysis.

43) Eizmendi, G., Azkoitia, J., Craddock, G. (2007). Challenges for Assistive Technology. 'Development of the vibration absorption castor for wheel chair -The effect of vibration absorption and endurance'. pp. 58-62.

44) Endo and Matsuishi, M., T. (1968), "Fatigue of metals subjected to varying stress ", Japan society of Mechanical Engineers, Fukuoka, Japan.

45) Erdemir, A., Guess, T. M., Halloran, J., Tadepalli, S. C., & Morrison, T. M. (2012). Considerations for reporting finite element analysis studies in biomechanics. *Journal of Biomechanics, 45*(4), 625-633. doi:10.1016/j.jbiomech.2011.11.038.

46) Fatemi, A. and Socie, D. F. (1988). A critical plane approach to multi-axial fatigue damage including out-of-phase loading, Fatigue and Fracture of Engineering Materials and Structures, Vol. 11, pp. 149.

47) Fatemi, A., Yang, L. (1998), "Cumulative fatigue damage and life prediction theories: a survey of the state of the art for homogenous materials", Int. J. Fatigue, Vol.20, No.1, pp9-34.

48) Firat, M., Kozan, M., Ozsoy, R., Hamdi, Mete O., (2010). "Numerical modeling and simulation of wheel radial fatigue tests," International Journal of Heavy Vehicle Systems, vol. 17, no. 2, pp. 159–178.

49) Freund, L. B. (1998). *Dynamic fracture mechanics*. Cambridge: Cambridge University Press. doi: https://doi.org/10.1017/CBO9780511546761, pp. 55-103.

50) Ganesh, S., Periyasamy, P., (2014). Design and Analysis of Spiral Wheel Rim for Four Wheele, The International Journal of Engineering and Science (IJES), Volume 3, Issue 4, PP. 29-37, ISSN (e): 2319 – 1813 ISSN (p): 2319 – 1805.

51) García-Pozuelo, D., Diaz, V., Boada, M. J. (2014). Improving Vehicle Safety: A New Methodology for Vehicle Steering System Inspection by Means of Forces Measure. *Advances in Mechanical Engineering*, *6*, 870349. doi:10.1155/2014/870349.

52) Glodová, I., Lipták, T., ocko, J. (2014). Usage of Finite Element Method for Motion and Thermal Analysis of a Specific Object in SolidWorks Environment. Procedia Engineering, 96, 131-135. doi: 10.1016/j.proeng.2014.12.131.

53) Goodman, J. (1919). Mechanics Applied to Engineering Longmans, Green and Co., London (1919), pp. 631-636.

54) Gotoh, E., Ohsawa, S., Satoh, Y., Yasuda, S. (1992). "The Investigation of Steering Column Collapse Behavior using Finite Element Analysis", SAE, International Congress and Exposition, Detroit, Michigan, February 24-28.

55) Griffith, A. A. (1921). "The phenomena of rupture and flow in solids" *(*PDF), Philosophical Transactions of the Royal Society of London, A, 221: 163-198, *doi*:*10.1098/rsta.1921.0006*, archived from *the original* (PDF)on 2006-10-16.

56) Groesser, S, Schwaninger, M. (2012). Contributions to model validation: hierarchy, process, and cessation. System Dynamics Review, 28(2):157–181, 2012.doi:10.1002/sdr.

57) Hamrock, B. J., Jacobson, B. O., Schmid, S. R. (2014). Fundamentals of machine elements: SI version. Boca Raton: CRC Press, Taylor & Francis Group. pp. B92-B93.

58) Han, L., Jing, L., Liu, K. (2017). A dynamic simulation of the wheel–rail impact caused by a wheel flat using a 3-D rolling contact model. Journal of Modern Transportation, 25(2), 124-131. doi: 10.1007/s40534-017-0131-0.

59) Han J. W, Kim, J. D., Song, S. Y. (2013) “Fatigue strength evaluation of a bogie frame for urban maglev train with fatigue test on full-scale test rig”, Engineering Failure Analysis, vol.31, pp.412-420.

60) Haug, E. J. (1989). Computer Aided Kinematics and Dynamics of Mechanical Systems: Vol.I: Basic Methods. Allyn and Bacon.

61) Heavy Duty & Custom Casters. (2018). Retrieved April 12, 2018, from http://www.casterconcepts.com/.

62) Holt, W. L., Wormeley, P. L. (1925). Power losses in automobile tires, Tech. Pap. BS 16, 451 (1922): Dynamometer tests of automobile tires, ibid. 17, 559 (1923): Effect of tires resistance on fuel consumption, ibid. 19, 213 (1925).

63) ISWP Home. (2018). Retrieved April 12, 2018, from http://www.wheelchairnet.org/.

64) Irwin, G. R., (1957) “Analysis of Stresses and Strains Near the End of a Crack Traversing a Plate”, Trans. ASME, J. Appl. Mech. E24, pp. 361-364.

65) Jalon, J. G. D., Bayo, E., Bahar, L.Y. (1995). Kinematic and dynamic simulation of multi body systems: The real time challenge. Applied Mechanics Reviews Including Supplement, 1995. 48(1): B2.

66) Jean-Victor, Poncelet (1911) ". Encyclopedia Britannica. 22. Encyclopedia Britannica, Inc. 1911. pp. 59.

67) Jianfeng, L., Gang, C., Jie, S., Yi, L. (2013). Finite Element Analysis of the Fatigue Life for the Connecting Rod Remanufacturing. Journal of Applied Sciences, 13(9), 1436-1442. doi:10.3923/jas.2013.1436.1442.

68) Katheriya, Kumar, Choudhary, Nareliya. (2014). An Investigation of Effects of Axle Load And Train Speed at Rail Joint Using Finite Element Method. *International Journal of Research in Engineering and Technology, 03*(08), 41-47. doi:10.15623/ijret.2014.0308008.

69) Kauzlarich, J. (1989). The Palmgren-Miner rule derived. Tribological Design of Machine Elements, Proceedings of the 15th Leeds-Lyon Symposium on Tribology Held at Bodington Hall, The University of Leeds Tribology Series,175-179. doi:10.1016/s0167-8922(08)70192-5.

70) Kaleemulla, K. M., Siddeswarappa, B. (2009). Influence of Fiber Orientation on the In-plane Mechanical Properties of Laminated Hybrid Polymer Composites. *Journal of Reinforced Plastics and Composites, 29*(12), 1900-1914. doi:10.1177/0731684409340806.

71) Kerdanyan, G., Minkel, J., Maneekobkunwong, S., Waters, R., Landsberger, L. (2005). Measurement of Force and Acceleration Experienced in a Manual Wheelchair. RESNA 28th Annual Conference, https://www.resna.org/sites/default/files/legacy/conference/proceedings/

2005/Research/OUT/Kerdanyan.html.

72) Kendall, K., Benuzzi, A. (1980). The COVA program—Validation of the fast reactor containment code seurbnuk. Nuclear Engineering and Design, 57(1). doi: https://doi.org/10.1016/0029-5493(80)90225-3.

73) Khude, N. N. (2015) efficient simulation of flexible body systems with frictional contact/impact (Unpublished doctoral dissertation). University of Wisconsin Madison.

74) Kłodowski, A., Mononen, M. E., Kulmala, J. P., Valkeapää, A., Korhonen, R. K., Avela, J., Mikkola, A. (2015). "Merge of motion analysis, multi body dynamics and finite element method for the subject-specific analysis of cartilage loading patterns during gait: Differences between rotation and moment-driven models of human knee joint". Multi body System Dynamics, 37(3), 271-290. doi:10.1007/s11044-015-9470-y.

75) Kocabicak, U., Firat, M. (2001). Numerical analysis of wheel cornering fatigue tests, Engineering Failure Analysis, vol. 8, pp. 339-354.

76) Kumar, Ravi, P. V., Meher, R. Satya (2013). International Journal of Modern Engineering Research (IJMER), Vol. 3, Issue. 3, pp. 1548-1553.

77) Kurt. D. Swenson, Karl. H. Frank. (1984). the Application of Cumulative Damage Fatigue Theory to Highway Bridge Fatigue Design. (Research Report), pp. 59-87.

78) Kyung Su Kim, Dong in Cho, Jae Wook Ahn, Seung Bok Choi, (2005). An Experimental Study on Fatigue Crack Propagation under Cyclic Loading with Multiple Overloads. Key Engineering Materials, 297-300, pp.: 2495-2500.

79) Lagoda T, Macha E and Nieslony A (2005) Fatigue life calculation by means of the cycle counting and spectral methods under multi axial random loading. Fatigue & Fracture of Engineering Materials & Structures 28(4):409–420.

80) Laird, C. and Smith, G.C. (1963). Initial stages of damage in high stress fatigue in some pure metals, Phil. Mag., Vol. 8, pp. 1945-1963.

81) Lee, Y., Barkey, M. E., & Kang, H. (2012). Metal fatigue analysis handbook: Practical problem-solving techniques for computer-aided engineering. Amsterdam: Amsterdam.

82) Lee Y. L., Pan, J., Hathaway, R., (2005) Fatigue Testing and Analysis Theory and Practice. Elsevier: Butterworth–Heinemann Press, pp. 44–66.

83) Lemaitre, J., Plumtree, A. (1979) Application of damage concept to predict creep-fatigue failures. Journal of Engineering Materials and Technology, Transactions of the ASME 101: 284–292.

84) Liu, Yongming, Man, Brant Atrat, Mahadevan, Sankaran, Fatigue crack initiation life prediction of railroad wheels, international journal of fatigue 28 ,2006, pp.747-756.

85) Liu, Y. (2006). Stochastic modeling of multi axial fatigue and fracture (Unpublished doctoral dissertation). Vanderbilt University, May.

86) Li, J., Wang, J., Li, X., Yang, J., Wang, H. (2015). The Experimental Study for Fatigue Strength of Bogie Frame of Beijing Subway Vehicle under Overload Situation Bottom View. The Open Mechanical Engineering Journal, vol. 9, no. 1, pp. 260–265.

87) Liao, L. (2011). A study of inertia relief analysis. In: 52nd AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics and Materials Conference. Denver, Colorado.

88) Lu, S. B., Li, Y. N., Dong, Y. L. (2011). Dynamic Modeling and Virtual Test of a New All Terrain Off-Road Vehicle. Applied Mechanics and Materials, 86, 125-128. doi:10.4028/www.scientific.net/amm.86.125.

89) Luo, R. K., Gabbitas, B.L., Brickle, B.V., Wu, W.X. (1998). Fatigue damage evaluation for a railway vehicle bogie using appropriate sampling frequencies. Vehicle System Dynamics: International Journal of Vehicle Mechanics and Mobility, vol. 29, no. sup1, pp. 404–415. ISSN 0042-3114.

90) Marques, F., Moutinho, C., Magalhães, F., Caetano, E., & Cunha, Á. (2014). Analysis of dynamic and fatigue effects in an old metallic riveted bridge. Journal of Constructional Steel Research, 99, 85-101. doi:10.1016/j.jcsr.2014.04.010.

91) Mane, J., Shinde, I., Bhatia, E., More, K., Jadhav, V., Lele, G. (2016). Caster Wheel Endurance Testing Rig. IOSR Journal of Mechanical and Civil Engineering, 13(3), 29-35.

92) Manson, R., Hertzberg W., Bretz, P. E. (1981), Normalization of fatigue crack propagation behavior in polymers. In: ‘’Advances in Fracture Research,”. D. Francois, Pergamon Press, Oxford, UK, pp. 443-448.

93) Mesmacque, G., Garcia S, Amrouche, A., (2005) Sequential law in multi axial fatigue, a new damage indicator. International Journal of Fatigue 27(4): 461–467.

94) Mhatre, A., Ott, J., Pearlman, J. (2017). Development of wheelchair caster testing equipment and preliminary testing of caster models. African Journal of Disability, 6(0). doi:10.4102/ajod. v6i0.358.

95) Mhatre, A., Martin, D., Mc Cambridge, M., Reese, N., Sullivan, M., Schoendorfer, D. (2017), ‘Developing product quality standards for wheelchairs used in less-resourced environments’, *African Journal of Disability* 6(0), a288.

96) Miner, M. A. (1945). Cumulative damage in fatigue, Journal of Applied Mechanics, Vol. 67, pp. A159-A164.

97) Miller, K. J., Zachariah, K. P. (1977). Cumulative damage laws for fatigue crack initiation and stage i propagation. *The Journal of Strain Analysis for Engineering Design, 12*(4), 262-270. doi:10.1243/03093247v124262.

98) Miyashita, Y., Yoshimura, Y., Xu, J.-Q., Horikoshi, M., and Mutoh, Y., (2003),“Subsurface Crack Propagation in Rolling Contact Fatigue of Sintered Alloy,”JSME Int. J., Ser. A,463, pp. 341–347.

99) Morita, Y., Kawashima, H., Ishihara, K. (1987). "Finite Element Stress Analysis of Automotive Steel Road Wheel", Sumitomo Metals, V 39, p 245-263.

100) Mott, N.F., (1948). “Brittle fracture in mild steel plates”, Engine. 165, pp. 16-18.

101) MSC Software Corporation. Retrieved April 12, 2018, from http://www.mscsoftware.com/.

102) Nallusamy, S., Prabu, N. M., Balakannan, K., & Majumdar, G. (2015). Analysis of Static Stress in an Alloy Wheel of the Passengercar. *International Journal of Engineering Research in Africa, 16*, pp. 17-25. doi:10.4028/www.scientific.net/jera.16.17.

103) Nanninga, N. E. (2008). *High cycle fatigue of AA6082 and AA6063 aluminum extrusions*.

104) Nejad, R. M., Farhangdoost, K., Shariati, M. (2016). Three-Dimensional Simulation of Rolling Contact Fatigue Crack Growth in UIC60 Rails. *Tribology Transactions, 59*(6), 1059-1069. doi:10.1080/10402004.2015.1134738.

105) Nielsen, J. C. O., Igeland, A. (1995). Vertical dynamic interaction between train and track influence of wheel and track imperfections. J Sound Vib 187:825–839.

106) Nikravesh, P. E. (1984). Some Methods for Dynamic Analysis of Constrained Mechanical Systems: A Survey. *Computer Aided Analysis and Optimization of Mechanical System Dynamics*, 351-368. doi:10.1007/978-3-642-52465-3_14.

107) Nickerson, B. (2017). Development of an integrated numerical method for the fatigue analysis of railway bogies (master's thesis). Stellenbosch University https://scholar.sun.ac.za.

108) Otorabad, H. A., Tehrani, P. H., & Younesian, D. (2018). 3D transient elasto-plastic finite element analysis of a flatted railway wheel in rolling contact. *Mechanics Based Design of Structures and Machines*, 1-16. doi:10.1080/15397734.2018.1457446.

109) Pace. D. K. (2004). Modeling and simulation verification and validation challenges. Johns Hopkins APL Technical Digest (Applied Physics Laboratory), 25(2):163–172.

110) Patel, Sumant, Gohil, A. V. (2013). Wheel-rail contact fatigue, international journal of engineering research and application, Vol.3, No. 2, March-April, 2013, pp. 980-983.

111) Pan, Shu-Qin (2014). Numerical simulation of steel wheel dynamic cornering fatigue test, Engineering Failure Analysis, Vol. 39, pp. 124–134.

112) Pieringer, A., Kropp, W., Nielsen, J. C. O. (2014), The influence of contact modelling on simulated wheel/rail interaction due to wheel flats. Wear 314:273–281.

113) Pereira, A. M., Reis, P. N., & Ferreira, J. A. (2015). Effect of the mean stress on the fatigue behaviour of single lap joints. *The Journal of Adhesion, 93*(6), 504-513. doi:10.1080/00218464.2015.1110699.

114) Pozuelo, D., Gauchia, A., Olmeda, E., Diaz, V. (2014). Bump Modeling and Vehicle Vertical Dynamics Prediction. Advances in Mechanical Engineering, 6, 736576. doi:10.1155/2014/736576.

115) Pun, C. L., Kan, Q., Mutton, P. J., Kang, G., & Yan, W. (2014). A single parameter to evaluate stress state in rail head for rolling contact fatigue analysis. *Fatigue & Fracture of Engineering Materials & Structures, 37*(8), 909-919. doi:10.1111/ffe.12157.

116) Raju, P. R., Satyanarayana, B., Ramji, K., Babu, K. S. (2007). Evaluation of fatigue life of aluminum alloy wheels under radial loads. *Engineering Failure Analysis, 14*(5), 791-800. doi:10.1016/j.engfailanal.2006.11.028.

117) Ramachandran, V. (2005). *Failure analysis of engineering structures: Methodology and case histories*. Materials Park, OH: ASM International. pp. 10-30.

118) Rathod, C., Chamorro, R., Escalona, J. L., El-Sibaie, M., Shabana, A. A. (2009). Validation of three-dimensional multi-body system approach for modelling track flexibility. Proceedings of the Institution of Mechanical Engineers, Part K: Journal of Multi-body Dynamics, 223(4), 269-282. Doi: 10.1243/14644193jmbd212.

119) Ravi Kumar P.V, R. Satya Meher, International Journal of Modern Engineering Research (IJMER), Vol. 3, Issue.3, pp. 1548-1553, 2013.

120) Redfield, R. C., Sutela, C. (2012). Mountain bike wheel endurance testing and modeling. Procedia Engineering, 34, 658-663. doi: 10.1016/j.proeng.2012.04.112.

121) Recuero, A. M., & Escalona, J. L. (2013). Modeling Infinitely Long Flexible Railroad Tracks Using Moving Modes and Krylov Subspaces Techniques. *Volume 7A: 9th International Conference on Multi body Systems, Nonlinear Dynamics, and Control*. doi:10.1115/detc2013-12828.

122) Requejo, P. S. (2008). Effect of rear suspension and speed on seat forces and head accelerations experienced by manual wheelchair riders with spinal cord injury. *The Journal of Rehabilitation Research and Development, 45*(7), 985-996. doi:10.1682/jrrd.2008.01.0006.

123) Richart, F. E., Newmark, N.M. (1948). A hypothesis for the determination of cumulative damage in fatigue, Proceedings of ASTM, Vol. 48, pp. 767-800.

124) Roberson, R. E., Schwertassek, R. (1988). Multi body Systems. *Dynamics of Multi body Systems*, 3-37. doi:10.1007/978-3-642-86464-3_1.

125) RWM ‘Drum Testing’ White paper (2018). Retrieved April 12, 2018, from https://www.rwmcasters.com/wp-content/uploads/2017/07/DrumTesting_WhitePaper.pdf.

126) Satyanarayana, N., Sambaiah, Ch. (2012). Fatigue Analysis of Aluminum Alloy Wheel Under Radial Load, International Journal of Mechanical and Industrial Engineering (IJMIE), ISSN No. 2231 – 6477, Vol-2, Issue-1, pp. 1-6.

127) Sha, L. R., Shi, Y. C. (2014). Fatigue Reliability Analysis on Connecting Rod of Automobile Engine Using Artificial Neural Network Method. Applied Mechanics and Materials, 501-504, 1092-1095. doi:10.4028/www.scientific.net/amm.501-504.1092.

128) Shabana, A. A. (1989). Dynamics of Multi body Systems. New York: Wiley.

129) Shang, D. G., Yao, W. X. (1998). Study on non-linear continuous damage cumulative model for uniaxial fatigue. Acta Aeronautical ET Astronautic Sinica 19(6): 647–656.

130) Sharman, P. W., Al-Hammoud, A. (1987). "The Effect of Local Details on the Stiffness of Car Body Joints", Int J of Vehicle Design, Vol 8, nos. 4/5/6, pp. 526-537.

131) Sourav, Das (2014). Design and Weight Optimization of Aluminum Alloy Wheel, Int. Journal of Scientific and Research Publications, Vol. 4, Issue 6, pp. 1-7.

132) Sraml, M., Flasker, J., Potrc, I., 2003. Numerical procedure for predicting the rolling contact fatigue crack initiation. Int J Fatigue, 25(7): 585-95.

133) Srinivasan, K., M. B., S, Nesarikar, A. (2011). Simulation and Test Correlation of Wheel Impact Test. SAE Technical Paper Series. doi:10.4271/2011-28-0129.

134) Strang, G., Fix, G. J. (1975). An Analysis of the Finite Element Method, Prentice-Hall, Inc., Englewood cliffs, N. J. ISBN 0130329460.

135) Stichel, S., Knothe, K. (1998). Fatigue Life Prediction for an S-Train Bogie. Vehicle System Dynamics: International Journal of Vehicle Mechanics and Mobility, vol. 29, no. sup1, pp. 390–403. ISSN 0042-3114.

136) Taek, Young Kim, Ho, Kyungkim (2014). Three-dimensional elastic plastic finite element analysis for wheel-rail rolling contact fatigue, seoul national university of science and technology, vol.6, number 3, Jun July 2014. P. 139-722.

137) Taheri, S. (2014). Finite Element Modeling of Tire - Terrain Dynamic Interaction for Full Vehicle Simulation Applications (Unpublished master's thesis). Virginia Polytechnic Institute and State University, May 1st, 2014.

138) Thakare, R. (2017). Stress Analysis in Wheel Rim by using Dynamic Cornering Fatigue Test under Different Conditions. 3(2) DOI: 10.13140/RG.2.2.12512.17926.

139) Thambiratnam, D. P., Zhuge, Y. (2008). Finite element analysis of track structures. Computer Aided Civil Infrastructure Eng 8: 467–47.

140) Turner, M. J, Clough, R. W., Martin, H. C., Topp, L. J. (1965). Stiffness and Deflection Analysis of Complex Structures', (1956), Journal of Aeronautical Science, Vol 23, pp. 805-832.

141) Vijayan, V., Altenhof, W., Banting, R. (2008). "A finite element approach for prediction of fatigue life for a three-piece mining vehicle wheel," Engineering Failure Analysis, vol. 16, no. 8, pp. 1533–1541, 2008.

142) Vyas, N. S., Gupta, A. K., Modeling Rail Wheel-Flat Dynamics. Engineering Asset Management, pp. 222-1231. doi:10.1007/978-1-84628-814-2_135.

143) Palaniselvam, V., Sabareesan, S., Karthik, S. (2014). Design and analysis of train wheel during rolling action, international journal of software and hardware research in engineering, vol.2, issue 5, May 2014 pp.252-257.

144) Wang, Hongyu (2013). Geometric Parameters Optimal Design of Variable Cross-Section Rim, SAE China and FISITA (eds.), pp. 1003-1008.

145) Wang, L., Chen, Y., Wang, C., WA , Q. (2011). Fatigue Life Analysis of Aluminum Wheels by Simulation of Rotary Fatigue Test.– Journal of Mechanical Engineering, 57(1), 31-39. doi:10.5545/sv-jme.2009.046.

146) Wang, X., Zhang, Y., Wang, B. (2009). "Research on reliability-based dynamic fatigue of locomotive wheels," Journal of Engineering Design, vol. 16, no. 4, pp. 247–251.

147) Wei, C., & Olatunbosun, O. A. (2014). Transient dynamic behavior of finite element tire traversing obstacles with different heights. Journal of Terra mechanics, 56, 1-16. doi: 10.1016/j.jterra.2014.07.001.

148) Wen, Z., Jin, X., Zhang, W. (2005). Contact-impact stress analysis of rail joint region using the dynamic finite element method. Wear, 258(7-8), 1301-1309. doi: 10.1016/j.wear.2004.03.040.

149) Westergaard, H. M., (1939), “Bearing Pressures and Cracks, J. Appl. Mech., 6, pp. 49-53

150) Williams, J. (2001). Introduction to linear elastic fracture mechanics. Fracture Mechanics Testing Methods for Polymers, Adhesives and Composites European Structural Integrity Society, 3-10. doi:10.1016/s1566-1369(01)80024-8.

151) Wöhler, A. (1855) Theorie rechteckiger eiserner Brückenbalken mit Gitterwänden und mit Blechwänden, Zeitschrift für Bauwesen vol. 5 pp121-166.

152) Wu, Jong-shyong, Shih, Po-Yun (2000), "Dynamic Response of Railway and Carriage Under the High-speed Moving loads," Journal of Sound and Vibration, Vol. 236, 61 – 87.

153) Xiong, Q. Y., Yu, S. T., Ju, J. S. (2015). Fatigue Analysis on Wheel considering Contact Effect Using FEM Method. Mathematical Problems in Engineering, 2015, 1-8. doi:10.1155/2015/314634.

154) Xu, J., Wang, P., Wang, L., Chen, R. (2016). Effects of profile wear on wheel–rail contact conditions and dynamic interaction of vehicle and turnout. Advances in Mechanical Engineering, 8(1), 168781401562369. doi:10.1177/1687814015623696.

155) Yadav, S. N., Hanamapure, N. S. (2013). Analyze the Effect of Camber Angle on Fatigue Life of Wheel Rim of Passenger Car by Using Radial Fatigue Testing, International Journal of Engineering Science and Innovative Technology (IJESIT), Vol.2, Issue 5, PP. 231-239, 2013.

156) Yang, J. W., Shi, Q. L., Zhang, G. Y., Zhang, J. (2012). The Fatigue Life Simulation of the Wheel of CHR3 EMU in Random Loading. Advanced Materials Research, 430-432, 1424-1427. doi:10.4028/www.scientific.net/amr.430-432.1424.

157) Ye, D. Y., Wang, D. J., P. (1996). Characteristics of the change in the surface micro hardness during high cycle fatigue damage. Materials chemistry and physics, 44(2), 179-181.

158) Younesian, D., Solhmirzaei, A., Gachloo, A. (2009). Fatigue life estimation of MD36 and MD523 bogies based on damage accumulation and random fatigue theory. Journal of Mechanical Science and Technology, vol. 23, no. 8, pp. 2149– 2156. ISSN 1738-494X.

159) Zhao, Y., Deng, Y., Lin, F., Zhu, M., Xiao, Z. (2018). Transient Dynamic Characteristics of a Non-Pneumatic Mechanical Elastic Wheel Rolling Over a Ditch. *International Journal of Automotive Technology, 19*(3), 499-508. doi:10.1007/s12239-018-0048-6.

160) Zheng, H., Xie, J., Zhou, S., Feng, B., “Multi-axial fatigue strength analysis of wheel based on FEA,” Journal of Beijing Jiaotong University, vol. 33, no. 4, pp. 54–59, 2009.

161) Zhou, S.-X, Nan, X., Jilong, X., “Research on crack propagation characteristic of railway wheel with web plat under multi axial loading,” Journal of Engineering Mechanics, vol. 27, no. 1, pp. 41–46, 2010.

162) Zuo, F., Huang, H., Zhu, S., Lv, Z., Gao, H. (2014). Fatigue life prediction under variable amplitude loading using a non-linear damage accumulation model. *International Journal of Damage Mechanics*, *24*(5), 767-784. doi:10.1177/1056789514553042.