Workflow development of Flex body Dynamics for Utility Vehicle Suspension System

MTech. Thesis

by

Sowmya Kovvuri
(2102103026)

Discipline of Mechanical Engineering

Indian Institute of Technology Indore

May 2023

Workflow development of Flex body Dynamics for Utility Vehicle Suspension System

A Thesis

Submitted in partial fulfillment of the requirements for the award of the degree

of

Master of Technology

in

Mechanical Engineering

With specialization in

Mechanical System Design

by

Sowmya Kovvuri

Discipline of Mechanical Engineering

Indian Institute of Technology Indore

May 2023

Indian Institute of Technology Indore

Candidate's Declaration

I hereby certify that the work which is being presented in the thesis entitled "**Workflow development of Flex body Dynamics for Utility Vehicle Suspension System**" in the partial fulfillment of the requirements for the award of the degree of **Master of Technology** in **Mechanical Engineering** with specialization in **Mechanical System Design** and submitted in the **Discipline of Mechanical Engineering, Indian Institute of Technology Indore**, is an authentic record of my own work carried out during the time period from **May 2022 to May 2023** under the supervision of **Dr. Indrasen Singh** of Discipline of Mechanical Engineering.

The matter presented in this thesis has not been submitted by me for the award of any degree from any other institute.

Sowmya Kovvuri

This is to certify that the above statement made by the candidate is correct to the best of our knowledge.

(Dr. Indrasen Singh) 05/06/2023

Sowmya Kovvuri has successfully completed his M. Tech Oral Examination held on 25th May 2023

Teradale

Mr. Amol Teradale
Signature of M. Tech Thesis Supervisors
Date: 01/06/2023

Satyanarayan
Signature of Convener, DPGC
Date: 05/06/2023

Name and signature of PSPC member 1 with date: **Dr. Girish Verma** Girish 05/06/23

M. Jayaprakash 5.6.2023

Name and signature of PSPC member 2 with date: **Dr. Jayaprakash Murugesan**

ACKNOWLEDGEMENTS

The author would like to thank Dr. Indrasen Singh from Indian Institute of Technology, Mr. Chirag Sheth and Mr. Amol Teradale from John Deere Enterprise Technology and Engineering Center in India, Pune for providing opportunity and supporting throughout on thesis.

Sowmya Kovvuri

DEDICATION

This research is dedicated to my parents for all the inspiration along with my brothers, sisters, and friends for constant supportthroughout the process

Abstract

Suspension systems are essential components of any vehicle, providing support, stability, and comfort while adapting to various road conditions and loads. In utility vehicles, which are often used in off-road and heavy-duty applications, the suspension system plays an even more critical role in ensuring the safety and reliability of the vehicle and its occupants. Therefore, the design and optimization of utility vehicle suspension systems require advanced modeling and simulation tools that can capture the complex dynamics and interactions of the system components. The Suspension system is mounted on the frame through control arms. These control arms connects the vehicle frame and the tires. Suspension primary function is to maximize the performance of the vehicle and helps to absorb bumps in the road and provide a safe and comfortable ride.

This paper focus on workflow development of flex body dynamics for utility vehicle suspension system and to study the suspension durability and fatigue life cycle of suspension using flex body dynamics independent suspension system. Full car model representing the considered utility vehicle has been developed and calibrated with respect to test data available. This model is validated using simulation software. In addition, this paper also includes the different road conditions to perform the fatigue and durability of critical suspension components. The objective of the current project is to develop a workflow of flex body dynamics for suspension system of utility vehicle. To study the suspension durability and fatigue life cycle of the suspension using flex body dynamics workflow.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

ACRONYMS (if any)

FEA – Finite Element Analysis

MBD – Multi Body Dynamics

K & C – Kinematics & Compliance

Chapter 1

Introduction

Suspension systems are essential components of any vehicle, providing support, stability, and comfort while adapting to various road conditions and loads. In utility vehicles, which are often used in off-road and heavy-duty applications, the suspension system plays an even more critical role in ensuring the safety and reliability of the vehicle and its occupants. Therefore, the design and optimization of utility vehicle suspension systems require advanced modeling and simulation tools that can capture the complex dynamics and interactions of the system components.

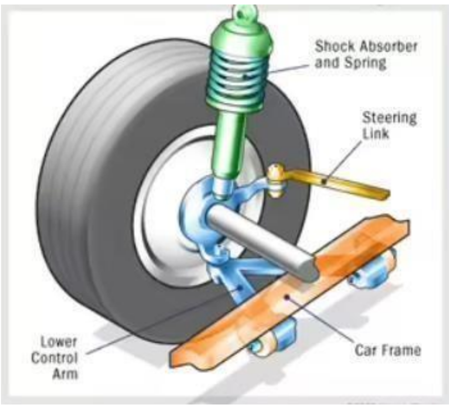

Fig 1. Independent Suspension Systems

However, the existing suspension system models and simulations often have limitations in their ability to account for the flexibility and deformation of the suspension components, which can affect the overall performance and durability of the system. Traditional rigid-body models may not be able to accurately capture the complex motions and forces that occur in utility vehicle suspension systems. Therefore, there is a need for more advanced approaches that incorporate the flexbody dynamics of the suspension components.

This thesis aims to develop a workflow for incorporating flexbody dynamics capability into a utility vehicle suspension system model, using commercially available software tools. The research will focus on a specific type of suspension system commonly used in utility vehicles, which consists of several components such as the leaf spring, shock absorber, and axle.

The flexbody dynamics of these components will be modeled using finite element analysis (FEA) software, and the results will be integrated into a multi-body dynamics (MBD) simulation software to model the overall suspension system.

Utility vehicle is off highway vehicle, and it performs various heavy-duty operations. It has independent suspension like automobile cars. It is important to ensure adequate strength and durability of suspension system. Field testing can be used to assess these parameters; however, it is often time consuming and expensive. Virtual simulation (computer aided simulation also called as CAE) is great way to assess strength and durability of suspension system in cost effective manner. It also provided engineers an ability to iterate design faster without building physical prototypes. However due to complexity involved and nonlinear response of suspension system, it’s always challenging to define proper virtual simulation process workflow.

FEA is well known method for assessing strength and durability of suspension parts/components. FEA was off to understand it, below field test data was collected Wheel force transducers at each wheel – it measures forces imparted due to ground interaction on the wheels. Forces passing through shock absorber is also measured. Final force transmitted to vehicle frame is also measured detail study was conducted on how ground forces being transferred to vehicle frame and what % of wheel forces transmitted to vehicle frame (let’s call it as force transmissibility)

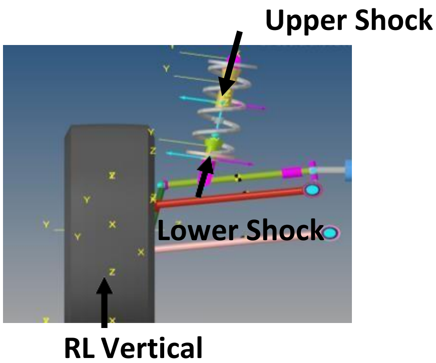

Fig 2. Suspension Geometry

$$\text{Wheel force transmissibility (WFT)} = \frac{\text{Shock Force lower}}{\text{Vertical Wheel load}}$$

From the test data we have received the force transmissibility ratio in between 0.93 to 1.13. From the static FEA we have received the force transmissibility ratio of 1.4. Due to this in static FEA, there are higher stress produces on the frame and the control arms which are not producing during the actual condition.

It’s important to accurately capture physics and non-linear behavior of shock which comprises of coil spring and damper. The damping forces are directly proportional to velocity at which wheel

is moving. This mean how fast or slow the wheels are moving (depending on ground track) would

results different damping forces.

Also, there is different response or resistance from shock during compression vs rebound action (non-linearity). It’s not possible to capture this behavior in FEA. Hence FEA predictions are not matching quite well with test data. Hence Multi body dynamics and Flex body dynamics makes more sense than traditional FEA methods.

The research objectives are as follows:

- Develop a comprehensive and accurate model of the utility vehicle suspension system that accounts for the flexbody dynamics of the components.
- Validate the model using experimental data and sensitivity analysis to ensure its accuracy and reliability.
- Analyze the effect of the flexbody dynamics on the overall performance and durability of the suspension system.
- Demonstrate the applicability and benefits of the proposed approach for utility vehicle design and optimization.

The proposed workflow will involve several steps, including the selection and preparation of the software tools, the modeling and simulation of the suspension system components, the integration of the FEA and MBD models, and the analysis and interpretation of the results. The contributions of the research will include a more advanced and realistic approach to suspension system modeling and simulation, as well as insights into the design and optimization of utility vehicle suspension systems.

The rest of this thesis report is organized as follows. In Chapter II, we provide a review of the literature on suspension system modeling and simulation, as well as the use of flexbody dynamics in vehicle applications. Chapter III describes the mathematical modelling the flexbody dynamics. Chapter IV presents in the utility vehicle suspension system model, including the software tools, the modeling assumptions, and the validation and sensitivity analysis procedures the results of the simulation, including the comparison of the flexbody and rigid-body models, the analysis of the effect of the flexbody dynamics on the suspension system stress - strain analysis. Chapter V summarizes the main findings and conclusions of the research, as well as the limitations and future research directions. Finally, Chapter VI provides a list of references cited in this thesis report.

Chapter 2

Literature Review

In this paper [1] they had considered the damper of a car Dacia logan and they fixed the damper into the automatic testing machine spider 8. Performed testing and performance practice on the suspension dampers using the strain gauge, transducers spider 8 device and by providing loads on the vehicle through vibrant platform. They measured the frequency response of the system. They also developed the MATLAB Simulink model of the suspension damper and ran the analysis recorded the frequency response of the system. Then compared the experimental results with the simulation results which are developed under MATLAB/Simulink. There is a good correlation between the experimental results and the simulation results from which they concluded that the damping system model can also be accepted by performing the computer simulation [1].

In this paper [2] they had ignored the 1 DOF freedom and modeled the entire car with 2 DOF. They had developed the Lagrange's differential equations and ran a modal analysis on it. Calculated the eigen vectors, natural frequencies and mode shapes all the plotting is done in MATLAB.

In this paper [3] they had considered double wishbone air suspension system. They had developed the double wishbone air suspension model in ADAM/View and customized all the parameters in FORTAN Subroutines. They had imported the meshed models from the hyper mesh to ADAMS and Considered suspension stiffness nonlinearity, joint clearances and suspension supporting components as flexible and performed the K&C analysis in ADAMS software. Performed K&C test using noncontact binocular vision system using 2 cameras as optical targets and by test conditions as input into the computer system and performed the K&C test. They had compared all the 3 results considering non linearities and joint clearances and flexible components with the traditional ones and with the test data where they found that the system which they considered all the nonlinear parameters is more correlating with the test results [3].

This paper [4] is more relevant to the current project which I am working on using Motion View, they had developed model from the existing library, and updated all the parameters with respect to the model they are going to perform the analysis and calibrated the wheel rate curves and all other data with the help of test data which is already available. They had considered trailing arm

and lower control arm of suspension as flex bodies and performed the quasi static & transient analysis. In quasi static they had correlated the load values Then correlated the simulation results with the test results which has 10 to 20% variation with the test data [4]. Modeled the entire SUV model in modelling software and represented the model to ADAMS software using a transfer function CAT/ADAMS [5]. They had performed the testing using servo hydraulic or road test simulator or on road and collected the output obtained from the tests. They considered the results obtained from the road test simulator or on road tests as an input to the ADAMS model in terms of ASCII codes and performed the analysis in ADAMS. After performing the analysis in ADAMS, the frequency domain of both physical and virtual model the amplitude accelerations of body are within 20% to each other [5].

Conducted the constant radius test of a vehicle in ADAMS vehicle is driven along a curve with a constant radius at various speeds [6]. They have evaluated the effects of camber and toe on car during cornering. Larger the value of camber lesser will be the slip and steering angle for various speeds camber cannot be increased more than the certain limit tire wear. Toe out conditions used to assist steering properties during cornering and toe in for straight line stability. These values also cannot be increased beyond the limit which may lead to tire scuffing and problems related to handling.

Chapter 3

Mathematical modeling

The temperatures, stresses, fluxes, or other desired unknown parameters in the finite element model are obtained through finite element analysis by minimizing an energy functional.

Each energy related to a certain finite element model is represented by an energy functional. The finite element's energy functional must be equal to zero in accordance with the rule of conservation of energy.

By minimizing the energy functional, the finite element technique finds the right answer for any finite element model. By setting the derivative of the functional with respect to the unidentified grid point potential to zero, the minimum of the functional is discovered.

Thus, the basic equation for finite element analysis is

$$\frac{\partial \mathbf{f}}{\partial \mathbf{p}} = 0$$

where F is the energy functional, and p is the unknown grid point potential to be calculated. This is based on the virtual work concept, which asserts that if a particle is in equilibrium with a system of forces, the virtual work for any displacement is zero. Every finite element will have a distinct energy function of its own.

In stress analysis, the governing equations for a continuous rigid body can be obtained by minimizing the total potential energy of the system.

The total potential energy P can be expressed as:

$$\prod = \mathbf{1/2} \int_{\Omega} \boldsymbol{\sigma^T} \varepsilon dV - \int_{\Omega} \boldsymbol{d^T} b dV - \int_{\Gamma} \boldsymbol{d^T} q dS$$

where σ and ε are the vectors of the stress and strain components at any point, respectively, d is

the vector of displacement at any point, b is the vector of body force components per unit volume, and q is the vector of applied surface traction components at any surface point. The volume and surface integrals are defined over the entire region of the structure **Ω** and that part of its boundary subject to load Γ. The first term on the right-hand side of this equation represents the internal strain energy and the second and third terms are, respectively, the potential energy contributions of the body force loads and distributed-surface loads.

The variation inside the element is expressed in terms of the nodal values by means of interpolation functions since the displacement is assumed to have unknown values only at the nodal points in the finite element displacement method. Thus, within any one element, **d** = **Nu** where **N** is the matrix of interpolation functions termed shape functions and u is the vector of unknown nodal displacements. (**u** is equivalent to **p** in the basic equation for finite element analysis.) The strains within the element can be expressed in terms of the element nodal displacements as **ε** = **Bu** where **B** is the strain displacement matrix. Finally, the stresses may be related to the strains by use of an elasticity matrix (e.g., young's modulus) as σ = **E ε**.

The total potential energy of the discretized structure will be the sum of the energy contributions of each individual element.

Thus $\prod = \sum_{e} \prod_{e}$ Where $\prod_{e}$ represents the total potential energy of the individual element.

$\prod_{e} = 1/2\int_{\Omega_e} \boldsymbol{u^T} (\boldsymbol{B^T}\mathbf{EB})^{\boldsymbol{T}}\mathrm{u}\boldsymbol{dv} - \int_{\Omega_e} \boldsymbol{u^T}\,\boldsymbol{N^T}\,\mathrm{p}\boldsymbol{dv} - \int_{\Gamma} \boldsymbol{u^T}\,\boldsymbol{N^T}\,\mathrm{q}\boldsymbol{dS} = 0$

Taking the derivative of $\frac{\partial \prod_e}{\partial \boldsymbol{u}} = 1/2\int_{\Omega_e} (\boldsymbol{B^T}\mathbf{EB})^{\boldsymbol{T}}\mathrm{u}\boldsymbol{dv} - \int_{\Omega_e} \boldsymbol{N^T}\,\mathrm{u}\boldsymbol{dv} - \int_{\Gamma} \boldsymbol{N^T}\,\mathrm{q}\boldsymbol{dS} = 0$

element equilibrium equation **k u – f = 0**

Where $\mathrm{f} = \int_{\Omega_e} \boldsymbol{N^T}\,\mathrm{u}\boldsymbol{dv} + \int_{\Gamma} \boldsymbol{N^T}\,\mathrm{q}\boldsymbol{dS}$

$\mathrm{k} = \int_{\Omega_e}(\boldsymbol{B^T}\mathbf{EB})^{\boldsymbol{T}}\mathrm{u}\boldsymbol{dv}$ and k is known as element stiffness matrix.

The physical significance of the vectors u and f varies according to the application being modeled.

Mathematical modeling of Flex Body:

In conventional multibody dynamic (MBD) assessments, rigid body systems are simulated while forces and/or motions are applied.

In the real world, any continuous media will deform when force is applied to it. Such deformations are not included in rigid body simulations, which can produce unreliable findings. Flexibility is accounted for in MBD simulations via flexible bodies.

What is a flexible body?

The number of degrees of freedom in finite element models is extremely high. MBD solvers have a difficult time dealing with these.

A modal representation of a finite element model is a flexible body. Very few modal degrees of freedom remain in the finite element model.

The nodal displacement is represented as a linear combination of a few modal coordinates in physical coordinates.

$$U = \phi Q$$

U is nodal displacements,

Φ is a modal matrix,

Q is a matrix of modal participation factors or modal coordinates that the MBD will determine.

Motion view uses the component mode synthesis process to reduce a finite element model to set of orthogonal mode shapes

Type of CMS methods

Craig-Bampton

One set of mode shapes from constrained interface static modes applying a unit displacement at each interface node in each degree of freedom individually

One set of mode shapes from fixed interface eigen modes with all interface nodes constrained.

Craig Chang

One set of mode shapes from an inertial relief analysis with applied unit forces
One set of mode shapes from a free-free eigen modes (unconstrained normal modes analysis

Flex bodies can:

Capture body deformation and Increase accuracy in load predictions

Determine the stress and/or strain distribution in a body

Help generate loads needed for fatigue analysis

Flex bodies provide an excellent balance between accuracy and simulation

Mathematical modelling of Fatigue Analysis:

Stress-Life Method: This method is also known as the S-N method, where S refers to the stress amplitude and N refers to the number of cycles to failure. This method is commonly used for metal components subjected to high-cycle fatigue. The method involves creating a stress-life curve for the material under consideration and comparing the calculated stress amplitude to the endurance limit of the material to predict its fatigue life. Log-log plot with the actual S-N line representing the mean of the data. - Endurance or fatigue limit (Se) is a stress level below which the material has an infinite life. - For engineering, this infinite life is usually considered to be 10^6 cycles (Fig.3). - Most nonferrous alloys have no endurance limit, and the S-N line has a continuous slope (Fig.4.). - A pseudo-endurance limit or fatigue strength for these materials is taken as the stress value corresponding to a life of 5×10^8cycles.

$\Delta\sigma = \sigma_{max} - \sigma_{min}$ = Stress range

$\sigma_a = \frac{\sigma max - \sigma min}{2}$ = Stress amplitude

$\sigma_m = \frac{\sigma max + \sigma min}{2}$ = Stress amplitude

$R = \frac{\sigma max}{\sigma_{min}}$ = Stress ratio

$R = \frac{\sigma a}{\sigma_m}$ = amplitude ratio

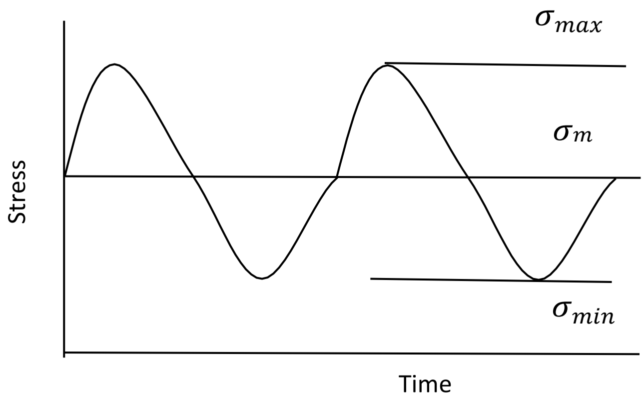

Fig 3. Stress time graph

Stress ratio and amplitude ratio values with respect to several common loadings

Fully reversed R= -1 A = ∞

Zero to max R=0 A=1

Zero to min R= ∞ A= -1

Haigh Diagram

The results of a fatigue test using a nonzero mean stress are plotted on a Haigh diagram by lines of constant life drawn through the data points.

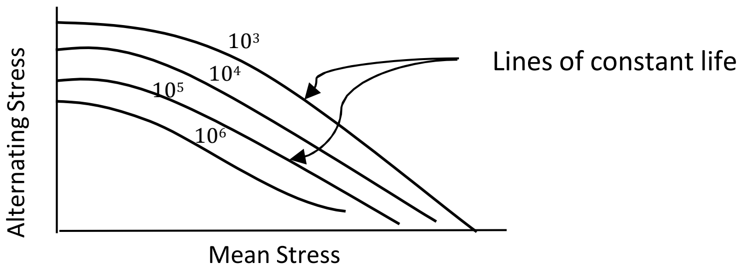

Fig.4 Haigh Diagram

Empirical relations:

Several empirical connections have been devised to construct the line defining the infinite-life design region because the testing needed to produce a Haigh diagram can be costly. - These techniques connect the yield strength, ultimate strength, and real fracture stress to the endurance limit on the alternating stress axis using various curves.

Soderberg - $\frac{\sigma a}{S_e} + \frac{\sigma m}{S_y} = 1$

Goodman - $\frac{\sigma a}{S_e} + \frac{\sigma m}{S_u} = 1$

Gerber - $\frac{\sigma a}{S_e} + \left(\frac{\sigma_m}{S_u}\right)^2 = 1$

Morrow - $\frac{\sigma a}{S_e} + \frac{\sigma m}{\sigma f} = 1$

a – Soderberg line b – Goodman line

c – Gerber line d – Morrow line

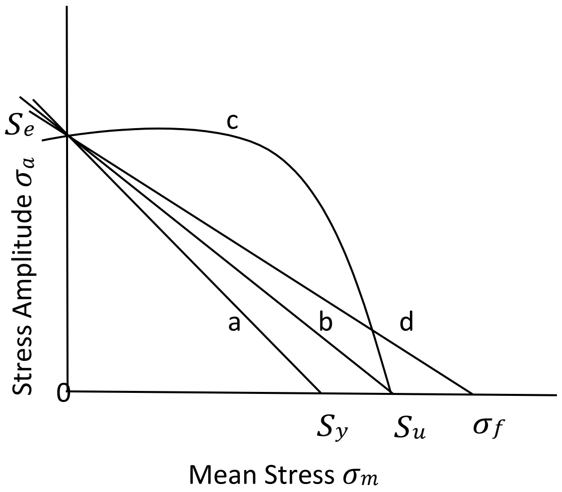

Fig.5 comparison of mean stress

Strain-Life Method: This method is also known as the ε-N method, where ε refers to the strain amplitude. This method is similar to the stress-life method, but it focuses on the material's strain instead of its stress. This method is commonly used for elastomers, plastics, and other materials that do not have a well-defined endurance limit.

Strain-life equations - Basquin (1910)'s equation: stress-life data could be plotted linearly on a log-log scale

$$\frac{\Delta\sigma}{2} = \sigma_f' \; (2N_f)^b$$

$\frac{\Delta\sigma}{2}$ = true stress amplitude

$2N_f$ = Reversals to failure (1 rev = $\frac{1}{2}$ cycle)

σ'_f = Fatigue strength coefficient

B = fatigue strength exponent

Coffin, Manson (1955)'s equation: plastic strain-life data could be linearized on log-log coordinates

$$\frac{\Delta\varepsilon_p}{2} = \varepsilon'_f \, (2N_f)^c$$

$\frac{\Delta\varepsilon_p}{2}$ = True strain amplitude

$2N_f$ = Reversals to failure (1 rev = $\frac{1}{2}$ cycle)

ε'_f = Fatigue ductility coefficient

C = fatigue ductility exponent

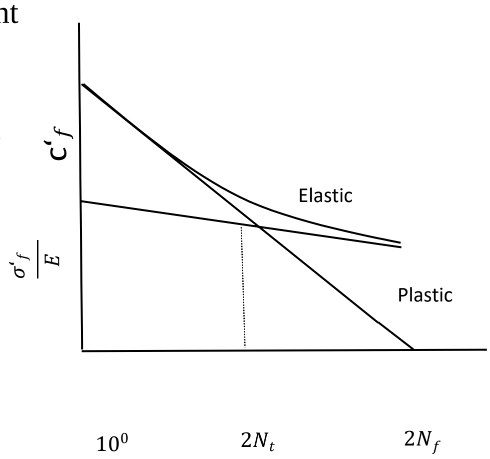

Fig.6 Strain Life curve

Total strain is the sum of the elastic and plastic strains

$$\frac{\Delta\varepsilon}{2} = \frac{\Delta\varepsilon_e}{2} + \frac{\Delta\varepsilon_p}{2}$$

The elastic term can be written as

$$\frac{\Delta\varepsilon}{2} = \frac{\Delta\sigma}{2E} = \frac{\sigma'_f}{E}(2N_f)^b$$

The total strain can now be rewritten as

$$\frac{\Delta\varepsilon}{2} = \frac{\sigma'_f}{E}(2N_f)^b + \varepsilon'_f \, (2N_f)^c$$

Modifications to the strain-life equation by Morrow

$$\frac{\Delta\varepsilon}{2} = \left(\frac{\sigma'_f - \sigma_o}{E}\right)(2N_f)^b + \varepsilon'_f (2N_f)^c$$

Modification to the strain-life equation by Manson and Halford

$$\frac{\Delta\varepsilon}{2} = \left(\frac{\sigma'_f - \sigma_o}{E}\right)(2N_f)^b + \varepsilon'_f \left(\frac{\sigma'_f - \sigma_o}{E}\right)^{c/b}(2N_f)^c$$

Modification to the strain-life equation by Smith, Watson, and Topper (SWT)

$$\sigma_{max}\frac{\Delta\varepsilon}{2} = \frac{(\sigma'_f)^2}{E}(2N_f)^{2b} + \sigma'_f \varepsilon'_f (2N_f)^{b+c}$$

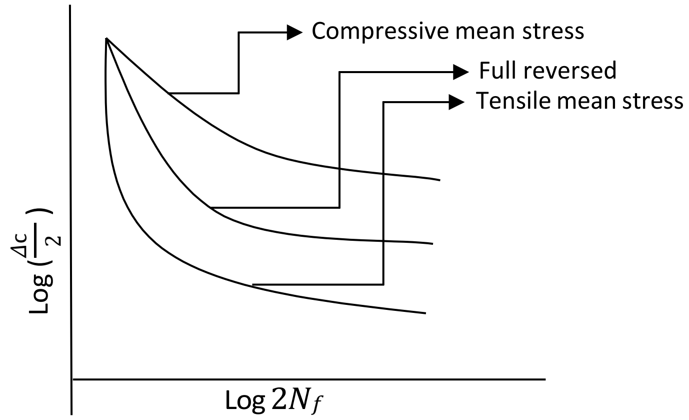

Fig.7 Effect of mean stress on strain life curve

The Smith-Watson-Topper life equation always accounts for mean stress and is typically used for low cycle fatigue.

$$\left[\frac{\frac{\Delta\sigma_1}{2} + \sigma_m}{\sigma'_f\,(2N_f)^b}\right]\frac{\Delta c_1}{2} = \frac{\sigma'_f}{E}(2N_f)^b + c'_f(2N_f)^c$$

$\frac{\Delta\sigma_1}{2}$ is maximum principal stress amplitude

$\frac{\Delta c_1}{2}$ is maximum principal strain amplitude

$2N_f$ is Reversals to failure

c'_f is Fatigue ductility coefficient

C is fatigue ductility exponent

σ'_f fatigue strength exponent

σ_m is mean stress along principal axis

Chapter 4

Simulation modeling of Flexbody Dynamics System

The utility vehicle simulation model was developed using the application Motion View 2022. The Altair default model was initially imported and changed to meet the needs of our car by calibrating the Altair default model using the results of the available test data and creating the necessary model. The process for creating the complete car model is described here.

Development of front half car model:

Using the pre-existing libraries, we created the Front half automobile model in the Motion view software. created all the characteristics specifically for our utility

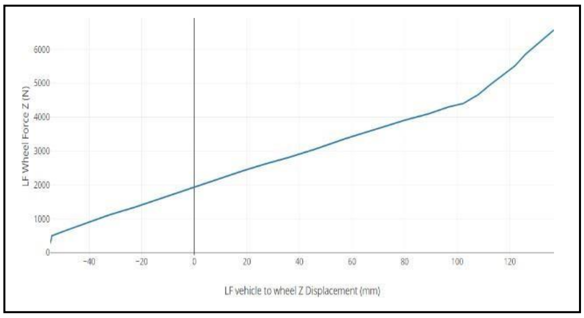

Fig 8 Wheel rate simulation Correlation for front half car model

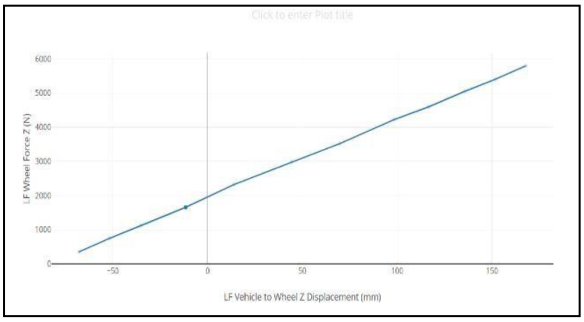

Fig 9 Wheel rate test data for front half car model

vehicle model. Compared to our utility vehicle model, we calibrated all the spring stiffness, damping, suspension inputs, moment of inertia information, and stabilizer bar data.

As seen in Figs. 3 and 4, the curves indicate the wheel rate data from simulation correlation and test data, where we calibrated the wheel rates using the test data that was available and the simulation model that was created. With test data, where the wheel rate is 27 N/mm, we were able to calibrate the model to get the wheel rate of 25.3 N/mm. In similar way we had calibrated other parameters like ride rates, suspension stiffness characteristics and wheel center displacements.

Development of Rear half car model:

Similar to the front half car model we had also the pre-existing libraries, we created the Fr Motion view software. created all the charac vehicle model. Compared to our utility vehicle stiffness, damping, suspension inputs, mome

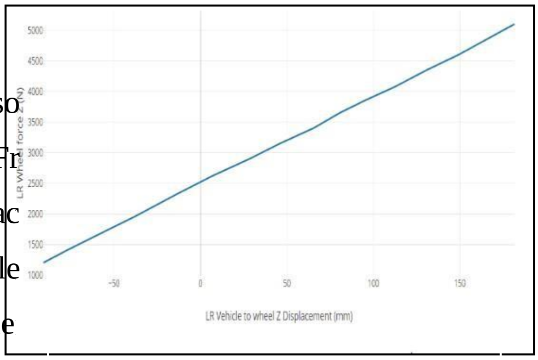

Fig 11. Wheel rate test data for front half car model

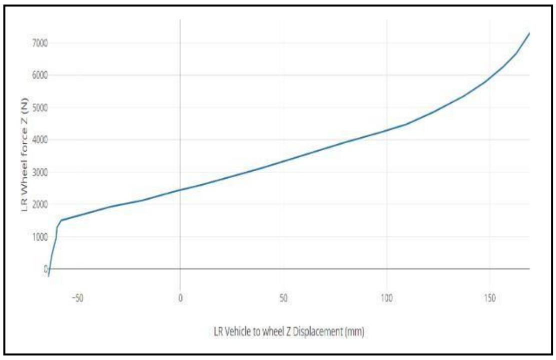

Fig 10. Wheel rate simulation Correlation for front half car model

stabilizer bar data.

As Seen in Figs. 10 and 11, the curves indicate the wheel rate data from simulation correlation and test data, where we calibrated the wheel rates using the test data that was available and the simulation model that was created. With test data, where the wheel rate is 17.5 N/mm, we were able to calibrate the model to get the wheel rate of 16.9 N/mm. In similar way we had also calibrated other parameters like ride rates, suspension stiffness characteristics and wheel center displacements.

Integration of Full car model:

After developing the front and rear half car models separately we had modeled the full car model using the Altair default library then updated the default model with the developed front and rear half car libraries. Then we had updated the model database with our details such as moment of inertia, vehicle mass, engine properties, CG of vehicle etc.

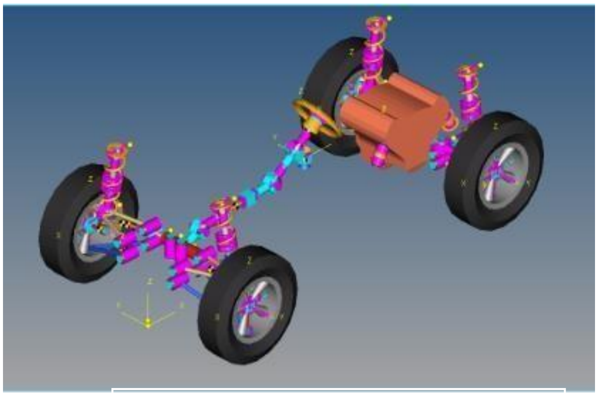

Fig 12. Full Car Model

Development & incorporating the flex bodies:

We had considered control arms which are critical components of suspension system as flexbody We had meshed the flex bodies in the separate FEA tool and imported them as flex bodies in the motion solve tool using flex prep option.

Flexbody dynamics modeling is a critical component of the methodology for developing flexbody dynamics capability in a utility vehicle suspension system. The flexbody dynamics model is a mathematical representation of the suspension system that accounts for the deformation of the suspension components. This is important because the deformation of the components affects the behavior of the suspension system, such as the ride comfort, handling, and stability.

The flexbody dynamics model is typically developed using finite element analysis (FEA) and multi-body dynamics (MBD) software. FEA is used to model the individual components of the suspension system, such as the shock absorbers, springs, and control arms, as well as the vehicle chassis and the tires. MBD software is used to model the overall behavior of the suspension system, including the interactions between the individual components.

The flexbody dynamics model incorporates both the rigid body dynamics and the flexible body dynamics of the suspension system. The rigid body dynamics represent the motion of the suspension system as a whole, while the flexible body dynamics represent the deformation of the suspension components.

The flexbody dynamics model includes the following steps:

Geometry Modeling:

The first step is to create the geometry of the suspension components, the vehicle chassis, and the tires using CAD software. The geometry is then imported into the FEA and MBD software.

Material Modeling:

The material properties of the suspension components are defined, including the elastic modulus, Poisson's ratio, and density. The tire characteristics are also modeled, including the tire stiffness, damping, and contact patch characteristics.

Mesh Generation:

The geometry of the suspension components is divided into small elements called meshes. The mesh size is chosen to balance accuracy and computational efficiency. We had meshed the components which we are going to consider as flex bodies in separate FEA software and imported into motion view using flex prep option.

Boundary Conditions:

The boundary conditions are defined, including the forces and moments applied to the suspension system, such as the weight of the vehicle, the road input, and the driver inputs.

Solver:

The FEA and MBD software solve the equations of motion to determine the motion and deformation of the suspension components. The solver accounts for the interactions between the individual components, including the nonlinear behavior of the suspension components and the tire-road interaction.

Results Analysis:

The results of the flexbody dynamics model are analyzed to determine the behavior of the suspension system under different loading and operating conditions. The analysis includes the ride comfort, handling, and stability of the suspension system.

Flexbody dynamics modeling is a critical component of the methodology because it allows for the accurate prediction of the behavior of the suspension system. This is important for optimizing the design of the suspension system and for identifying potential issues before the prototype is built.

Kinematics and Compliance Analysis

In order to ascertain the suspension geometry and stiffness properties, kinematics and compliance testing (K&C testing), a sort of quasi-static suspension testing, applies realistic loads and displacements to a vehicle's chassis and tires. There are sequence of tests which includes Kinematics, compliance test and Simulation performed in the K&C analysis as shown in the table. A sequence of tests are included in the kinematic test sequence in order to determine the vehicle's kinematic properties. Typically, this series involves a number of bounce, roll, and steer tests. To assess the frictional contribution of the bar in bounce and the stiffness contribution of the bar in roll, tests are commonly conducted both with and without anti-roll bars. When one end of the car should drive further than the other, a bounce-pitch test may occasionally be performed.

Since the other end of the car restricts the possible travel in pure bounce, this is helpful to describe bump stop loads or travel restrictions

Goal	Test Type	Test Limits
Kinematics tests	Bounce (W/ARB)	200mm compression 94mm rebound
	Roll, Fixed Axis(W/ARB)	+/- 7deg
	Roll, Natural Axis(W/ARB)	+/- 5 deg
	Bounce (No ARB)	200mm compression 94mm rebound
	Roll, Fixed Axis (No ARB)	+/- 7 deg
	Roll, Natural Axis (No ARB)	+/- 6 deg
	Steering Kinematics	+/- 375 deg at hand wheel
Compliance tests	Lateral, Opposed	+/- 75 G
	Lateral. Parallel	+/- 75 G
	Aligning Torque, Opposed	+/- 50N*m front +/- 70N*m rear
	Aligning Torque, Parallel	+/- 50N*m front +/- 70N*m rear
	Longitudinal, Braking	+/-1000N front, +/-500N rear
Simulation tests	Cornering Simulation	+/- 75G
	Cornering Simulation – Kinematics	Chassis motion from cornering situation, without lateral load
	Braking Simulation	-0.3G
	Braking Simulation – Kinematics	Chassis motion from braking situation, without lateral load

Table 1. K&C Analysis Conditions

Compliance test:

A set of tests are included in the compliance test sequence to determine how elastically the vehicle and its components respond to different loading circumstances. In this order, aligning torque tests, lateral and longitudinal load tests are frequently performed. Traction and braking tests are part of longitudinal compliance tests. There are two types of lateral compliance tests: lateral parallel and lateral opposed. There are parallel and oppositional alignment torque tests. Load circumstances are frequently coupled, as in a lateral parallel test with a 30mm trail, for example. This creates the desired load route by combining lateral and aligned torque loading conditions.

Simulation tests:

The simulation test sequence consists of a number of tests that simulate different circumstances. Typically, this sequence consists of computer simulations of cornering, braking, and traction. In every simulation test, a moment is applied to the vehicle as a function of the mass of the vehicle and the acceleration, measured in G, applied at the center of gravity. In response to the acceleration input, ground plane forces are simultaneously delivered at the tire contact patches. Depending on the normal load or a predetermined ratio, ground plane forces are distributed among the four tires. This test is followed by another test that analyses the kinematic effects separately from the compliance effects but does not use contact patch loads. The stresses on the control arms vary from 0 to 350Mpa in this simulation.

N-Post Simulation:

With Motion Solve, you can replicate an n-Post shaker test setup for any number of axles of a vehicle. The Post Shaker test is useful for determining component loads for fatigue analysis and for analyzing ride comfort. The authenticity of the virtual model can be determined by contrasting the outcomes of measurement and simulation using the road profiles from a testing facility as inputs to the simulation and specifying the outputs for loads or accelerations.

Due to the four axles on our car, we thought about performing a four-post analysis. To do this, we provided the n-post shaker with inputs in CSV format, including the vehicle's wheelbase, speed, the length of the course, the ride height, and other information. The figure displays the input signals for N-post analysis. The conditions for this simulation are vehicle is running at 5mph and passing through a bump the stress that are getting generated on the control arms vary from 1 to 135Mpa.

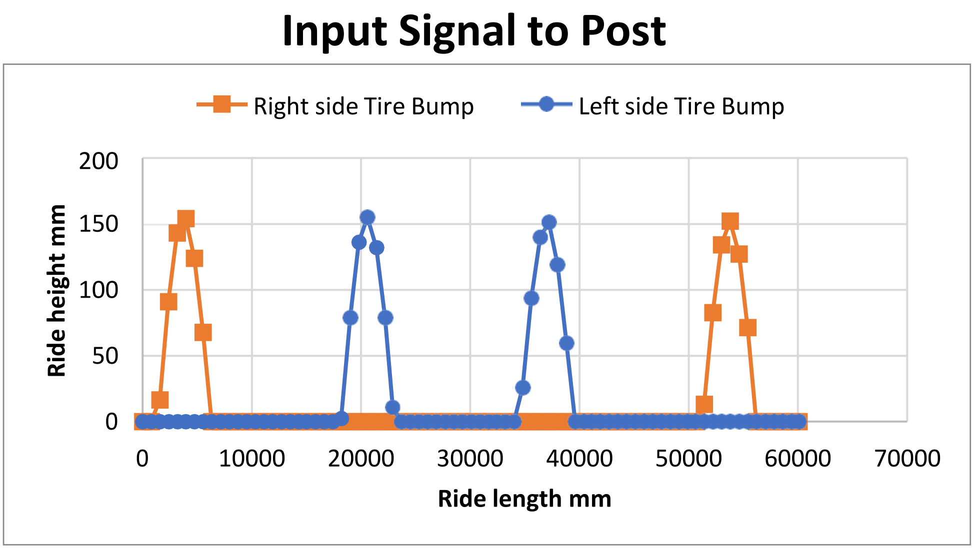

Fig13. Input Signal to Post

Road Profile

In this scenario, we will subject the utility vehicle to a challenging road profile. Through the use of a graphic file, the road profile will be entered, and an analysis will be performed to ascertain the amount of strain being applied to the vital suspension system components.

It is also possible to import various distinct road conditions that are used to evaluate challenging terrain profiles in order to conduct an analysis and determine the pressures on the suspension system's essential components.

Chapter 5

Fatigue Analysis

Particularly in the creation of mechanical systems like automobile suspension systems, fatigue analysis is a crucial part of engineering design. A mechanical system's strength and durability under cyclic loading circumstances are assessed through the process of fatigue analysis. The repeated application of loading and unloading cycles, which can result in the development and propagation of cracks in the material, can cause fatigue failure in mechanical systems.

The severe and dynamic operating conditions that the system encounters make fatigue studies particularly crucial in the context of vehicle suspension systems. The suspension system is subjected to a variety of loading circumstances, including torsional loads from braking and acceleration as well as vertical loads from the weight of the vehicle and lateral loads from cornering. The suspension components may experience cyclic stress and strain as a result of these loading circumstances, which may eventually cause fatigue failure.

Finite element analysis (FEA), for example, is frequently used in fatigue analysis to model the effects of cyclic loads and assess the performance of the suspension system. To assess how well the suspension components operate under various loading scenarios, the analysis may use fracture mechanics, stress-life, and strain-life methodologies.

In the context of vehicle suspension systems, the goal of fatigue analysis is to pinpoint probable points of failure due to fatigue and to improve the suspension system's design for increased sturdiness and longevity. Engineers can modify the suspension system's design and material selection to increase the performance and dependability by detecting probable fatigue failure sources. In order to ensure the safety and effectiveness of the system, fatigue analysis is a crucial step in the design process for vehicle suspension systems.

We had considered critical components of suspension system such as control arms as flexible components and performed further fatigue analysis on the control arms. The material specifications of the parts are mentioned in the below table2. As the load conditions are varying, we had considered the modal superposition type of loading and performed fatigue analysis for all the conditions which we had performed.

Material	Steel
Density	0.3
Poisons Ratio	7850kg/m.^{3}
Youngs Modulus	200Gpa
Yield Strength	350Mpa
Fatigue strength exponent	-1
Fatigue Ductility exponent	-0.57
Fatigue Ductility Coefficient	0.72
Fatigue Strength Coefficient	1023
Cyclic Strain Hardening Exponent	0.18

Tab2. Material Properties

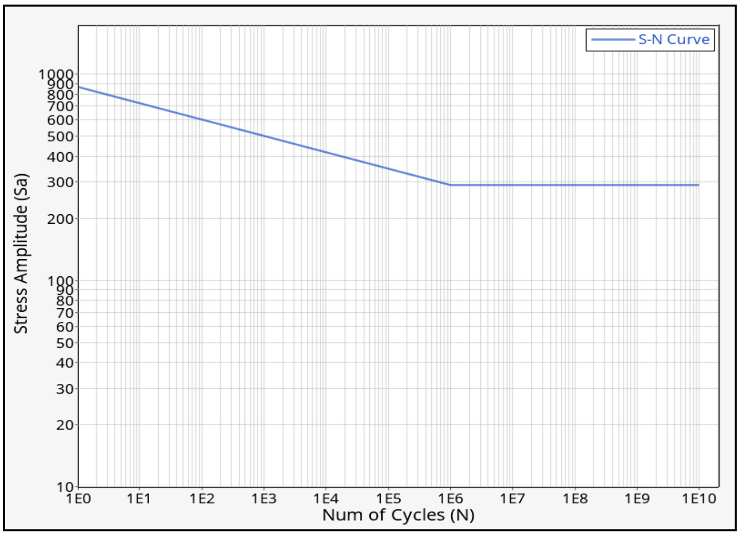

Fig 14. S-N Graph

Load Case - 1 K & C Analysis:

We had considered the material of the control arms shown in Tab2 and we had performed K&C simulation considering all the conditions mentioned in Tab1 and we had imported the load map as per the simulation we will be performing the fatigue analysis Fig 17 demonstrates the results of the fatigue analysis or control arm fatigue life cycle for the K&C simulation, where the control arm has an infinite life cycle and no damage is recorded except in certain locations, such as welding zones and other areas where holes are present, demonstrating the inappropriate life cycle of the control

Fig 15. K&C Analysis actualTest

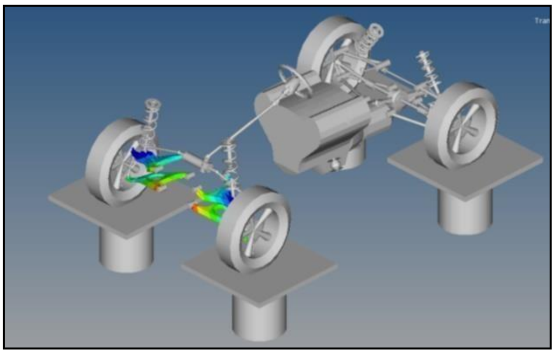

Fig 16 K&C Analysis virtual Test

arm's fatigue and damage that has been disregarded during the calculation of the fatigue life cycle. Fig 15 represents the actual K&C simulation and Fig 16 represents the K&C simulation virtually.

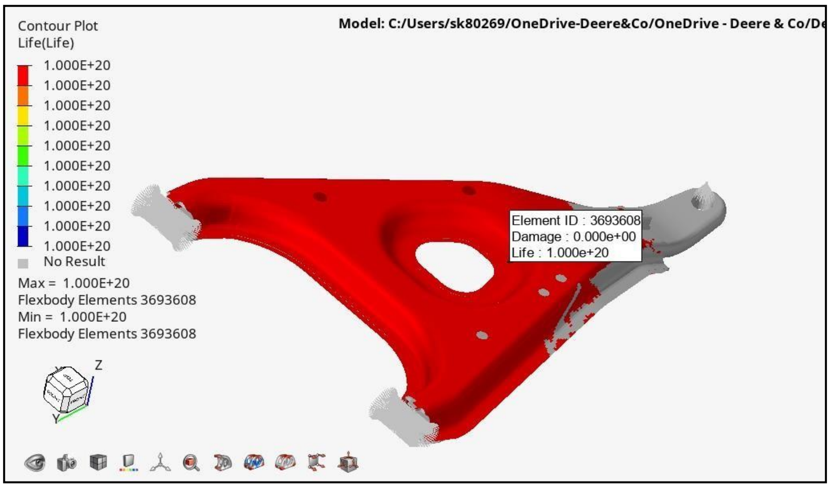

Fig 17. K&C Analysis Fatigue test results

Load Case - 2 N post Wheel displacement :

In the second case, Npost simulation for wheel displacement in which the vehicle was moving at 5 mph and passing over a bump.The road profile for the paticular case is shown in fig 18. Where fig 19 represents the virtual wheel displacement simulation..Fig 20 shows the stress plot in whichthe maximum stresses developed on the control arm is 135Mpa which is under the yeild limit.For the fatigue analysis we had considered the control arm and the same material properties from the tab2 and the EN graph for the material is shown in fig 14.

Considering all the parameters we had peroformed the fatigie life cycle where fig 21shows the fatigue life cycle and the damage of the control arms with for Npost simulation. Where the control arms have infinite life cycle with zero damage because of which there is an opportunity for design optimization.

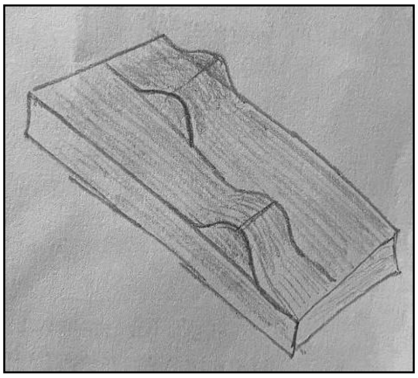

Fig 18. Sketch of road profile

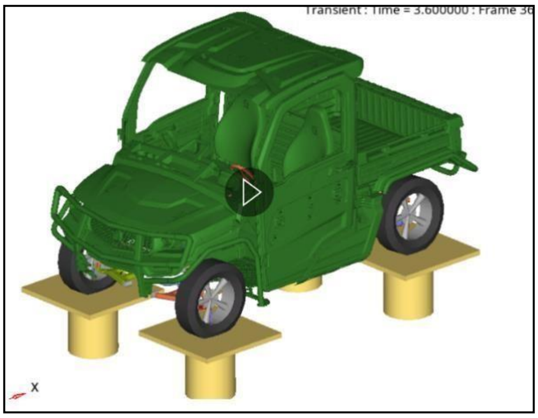

Fig 19. Virtual N Post Wheel displacement

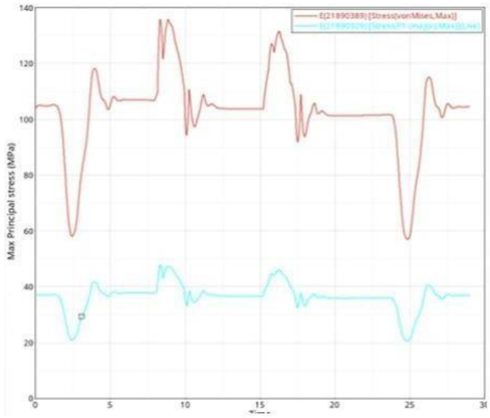

Fig 20. Stress plot for N Post wheel displacement load case

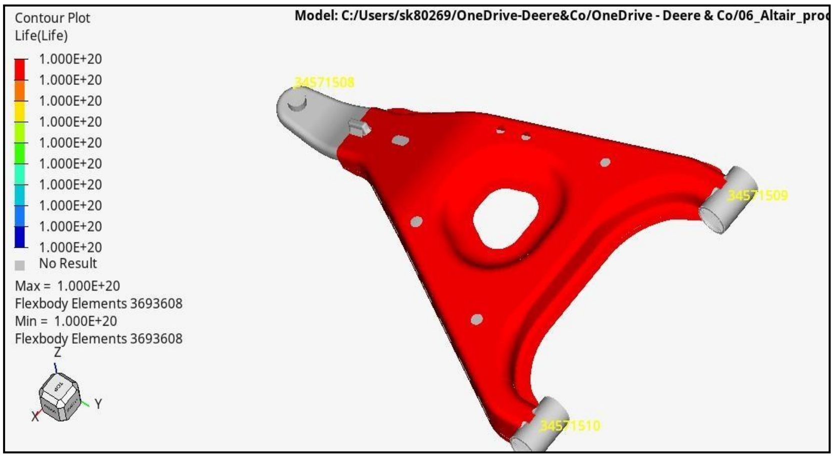

Fig 21. Npost fatigue analysis results for wheel displacement

Load Case - 3 N post Wheel force:

In the third case, Npost simulation for wheel force in which the vehicle was moving at passing over bumps on a tillage field.This road profile is very critical where higher stresses have been generated in this particular condition.The road profile for the paticular case is shown in fig 22. Where fig 23 represents the virtual wheel displacement simulation..Fig 24 shows the stress plot in whichthe maximum stresses developed on the control arm is 450Mpa which is more than the yeild limit.For the fatigue analysis we had considered the control arm and the same material properties from the tab2 and the EN graph for the material is shown in fig 14.Considering all the parameters we had peroformed the fatigie life cycle where fig 24 shows the fatigue life cycle and the damage of the control arms with for Npost simulation. Where the control arms have 2E+03 cycle which is under the acceptance criteria as it is a critical load condition.

Fig 22. Sketch of road profile

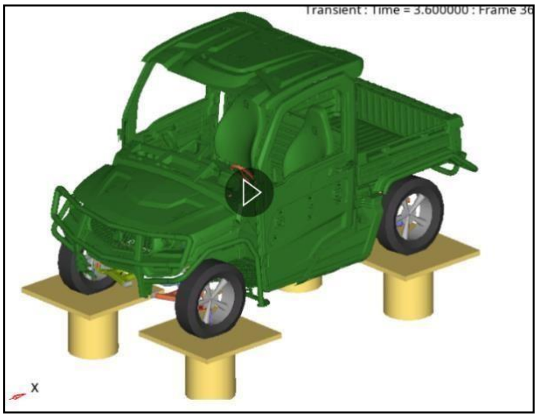

Fig 23. Virtual N Post Wheel force

c

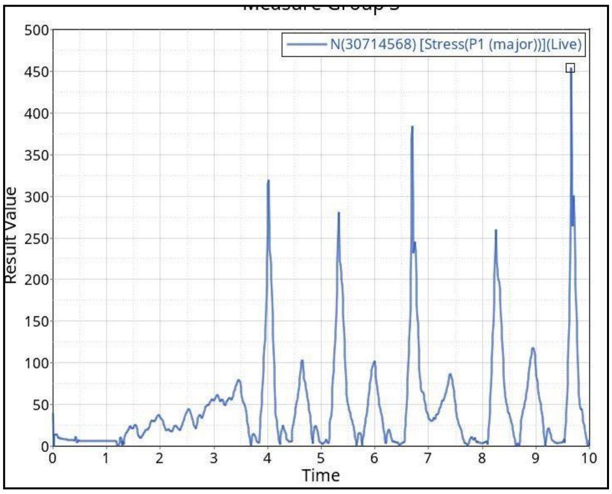

Fig 24. Stress plot for N Post wheel Force load case

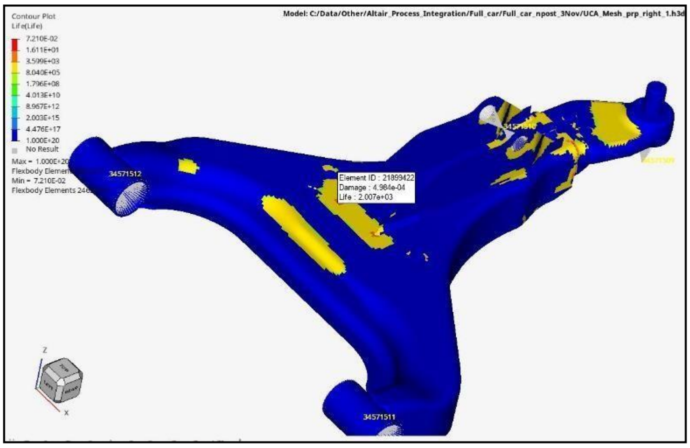

Fig 25. Npost fatigue analysis results for Wheel force

Load Case - 4 Vehicle moving on Road:

In the fourth case, vehicle moving on a road where we have considered a road profile on which the vehicle will be moving. The road profile for the paticular case is shown in fig 26 which also represents the virtual simulation of vehicle moving on road. For the fatigue analysis we had considered the control arm and the same material properties from the tab2 and the EN graph for the material is shown in fig 14.Considering all the parameters we had peroformed the fatigie life cycle where fig 27 shows the fatigue life cycle and the damage of the control arms with for vehicle moving on road. Where the control arms have 1E+20 cycle which means control arms have infinite life cycle with no damage.

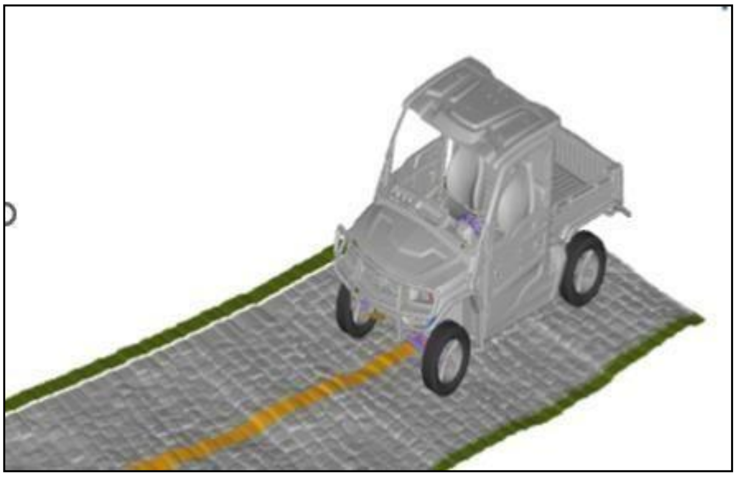

Fig 26. Virtual simulation vehicle moving on road

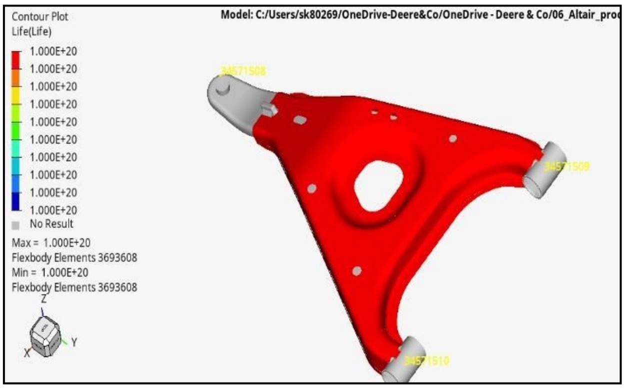

Fig 27. Npost fatigue analysis results for vehicle moving on road

Chapter 6

Results and Discussion

Modeling Results:

The developed flexbody dynamics model for the utility vehicle suspension system consists of several components, including the control arms, spring and damper assemblies, and the tire. The model was developed using solid modeling techniques and finite element analysis (FEA) software. The results of the modeling process include the material properties of the components, as well as the forces and moments acting on the components during operation.

Simulation Results:

The developed workflow was used to simulate the dynamic behavior of the utility vehicle suspension system under various loading conditions. The simulation results were obtained using multi-body dynamics (MBD) software and included the displacement, velocity, acceleration, and force profiles of the components. The simulation results were used to assess the performance of the suspension system in terms of ride comfort, stability, and handling.

The simulation results showed that the developed suspension system provided a good balance between ride comfort and stability. The system was able to absorb the shocks and vibrations generated by the road surface while maintaining the stability of the vehicle. The simulation results also showed that the system was able to maintain good handling characteristics during cornering and maneuvering.

Fatigue Analysis Results:

The developed workflow was used to perform fatigue analysis on the utility vehicle suspension system. The fatigue analysis was conducted using finite element analysis (FEA) software and included the estimation of the fatigue life of the components under different loading conditions. The fatigue analysis results were used to assess the durability of the system and identify any potential failure modes.

The fatigue analysis results showed that the developed suspension system was able to withstand the expected loading conditions over its service life without experiencing any significant fatigue damage. The results also identified some critical locations in the suspension system where fatigue

damage could potentially occur and suggested design modifications to improve the durability of these components.

Load cases	Max Stress Reported	Fatigue life	Acceptance Criteria	Comment
Load case 1 K&C Test	250 MPa	Na		Pass
Load case 2 Wheel displacement load case	135 MPa	1E10	Fatigue life >5475 Cycles	Pass
Load case 3 Wheel Force inputs for Tilling field	450 MPa	2007	>1000 (LCF phenomenon)	Pass
Load case 4 Vehicle motion on typical road		1E20 (2050 hrs.)	>1500 hrs.	Pass

Tab3. Results of all the load cases

Overall, the results of the workflow development for flexbody dynamics capability in utility vehicle suspension system showed that the developed workflow was effective in modeling, simulating, and analyzing the performance of the suspension system. The results also demonstrated the potential of the workflow to be used for the design and optimization of the suspension system for improved performance and durability.

From the results obtained we can see that there is an opportunity of design optimization by reducing the thickness of material or by changing the material grade which results for cost reduction.

Chapter 7

Conclusion

In conclusion, the workflow for the utility vehicle suspension system's flexbody dynamics capabilities was established, and it was effectively used to model, simulate, and analyze the suspension system's performance. The suspension system's components and their dynamic behavior were accurately represented by the created flexbody dynamics model. According to the simulation results, the suspension system that was created offered an excellent mix between ride comfort and stability, while yet keeping good handling qualities during turns and maneuvering.

The fatigue analysis results indicated that the suspension system was able to withstand the expected loading conditions over its service life without experiencing significant fatigue damage. However, some critical locations in the suspension system were identified where fatigue damage could potentially occur, suggesting design modifications to improve the durability of these components.

The design and optimization of suspension systems for utility vehicles can utilize the created workflow in a number of ways. It allows for the choice of the most effective design for the particular application by evaluating the performance and longevity of several suspension system designs. The workflow can also be used to evaluate how various operating circumstances affect the suspension system's performance and durability, enabling the design and performance of the system to be optimized for various operating circumstances.

In summary, the developed workflow for flexbody dynamics capability in utility vehicle suspension system has the potential to significantly improve the design and optimization of suspension systems for utility vehicles, leading to improved ride comfort, stability, and durability.

Future Scope

The workflow development of flexbody dynamics capability in utility vehicle suspension system opens up several opportunities for future work. Some of the potential areas of future work include:

Experimental Validation: The developed workflow can be experimentally validated by conducting physical testing of the utility vehicle suspension system. In this paper we had only correlated the K&C analysis report. We can also run the simulation in different road profiles and the experimental data can be compared with the simulation results obtained from the developed workflow to assess the accuracy of the model and identify any areas of improvement.

Optimization: The developed workflow can be used to optimize the design of the utility vehicle suspension system. By altering the design parameters, such as the dimensions and material properties of the components, the performance of the system can be improved in terms of ride comfort, stability, and durability.

REFERENCES

1. Marius-Constantin O.S. Popescu, Nikos E. Mastorakis “Testing and Simulation of a Motor Vehicle Suspension” https://www.researchgate.net/publication/237495565_Testing_and_Simulation_of_a_Motor_Vehicle_Suspension
2. M Umais Shabbir “Design and Analysis of Vehicle Suspension System” https://www.researchgate.net/publication/356812814_Design_and_Analysis_of_Vehicle_Suspension_System
3. Tianqi Lv, Yunqing Zhang , Yupeng Duan , James Yang ” Kinematics & compliance analysis of double wishbone air suspension with frictions and joint clearances” https://www.elsevier.com/locate/mechmachtheory
4. Tania Berber “Utilization of Motion View for Automotive Strength and Durability Analysis Application and Correlation “ https://www.sae.org/publications/technical-papers/content/2009-01-1196/
5. W.B. Frey, P.R. Frise,G.T.Andrews,and M.A.Malik” Combining virtual simulation and physical vehicle test data to optimize the durability testing” https://onlinelibrary.wiley.com/doi/epdf/10.1046/j.1460-2695.2002.00605.x
6. T. Vijayakumar, V. Ganesh” Simulated Study on the Effect of Camber and Toe on the Handling Characteristics of a Car during Cornering “ https://www.researchgate.net/publication/297414861_Simulated_study_on_the_effect_of_camber_and_toe_on_the_handling_characteristics_of_a_car_during_cornering?enrichId=rgreq-1a19279fda73ac1e9891b84de743ee5d-XXX&enrichSource=Y292ZXJQYWdlOzI5NzQxNDg2MTtBUzo0MTY2ODk5MTQxNzEzOTVAMTQ3NjM1ODAyNTU3OQ%3D%3D&el=1_x_2&_esc=publicationCoverPdf
7. “Altair Motion body solutions” https://2022.help.altair.com/2022.1/hwdesktop/altair_help/topics/release_notes/rn_2022_1_motionsolve_r.htm
8. Richard G Bhudyanas , J.Keith Nisbett “ Shigeley’s Mechanical design” page 208 – 340 https://ia903102.us.archive.org/33/items/MechanicalEngineeringDesign9th/Mechanical%20Engineering%20Design%209th.pdf

9. Seoul national university "Fatigue Analysis and design theory" https://ocw.snu.ac.kr/sites/default/files/NOTE/10989.pdf