Rigid Body and Modal Analysis of Independent Suspension Link

Vijaykumar V Solanki[1], Prof. N.S.Patel[2]

[1]PG Student, [2]Professor

[1,2]Department of Mechanical Engineering, Government Engineering Collage, Dahod, Gujarat, India

Abstract: **The durability of automotive components is a critical aspect in the product development. Failure in component will lead to a serious damage not only the design of the component itself, but the entire system of vehicle. In this paper, a forces aced on a critical component was determined from multi-body simulation system for double wishbone suspension system under braking and cornering load cases. Using conventional method we cannot derive the forces on each part as it will denote the forces of whole vehicle as we consider overall weight and geometry of vehicle. That's why ADAMS/view used as it gave output of forces on individual component of suspension. And complete FEM analysis of a suspension link for bending vibrations, braking, pitching, bouncing and combined mode dynamic analysis for deformation and stresses. For this a 3-D solid parametric model of a suspension link is generated in Pro-E Wildfire. Rigid body mode shape and participation factor effects by meshing in Hyper mesh by Shell 181, Solid 285 tetrahedral, MPC184 link/Beam, CP rig elements. It is analyzed for natural frequency and deflection. The ANSYS is used as solver. From the Modal Analysis normally it seems the lower arm is very stiff in breaking mode, i.e. higher the frequency higher the stiffness, the frequency is directly proportional to K and inversely proportional to Mass. The natural frequency for breaking mode is 391.5 Hz compared to natural frequencies respectively of rolling and pitching 132.5 HZ & 168.9 HZ. So, we can improve dynamic characteristic by increasing strength and frequency in rolling and pitching model.**

Keywords: Durability, FEM Analysis, Parametric Model, Mode Shape, rolling mode, pitching mode, breaking mode, ADAMS/View, Quasi-Static.

I. INTRODUCTION

Multi-body systems programs can often be used to determine the loads acting on suspension components and the body pickup points as inputs to finite element models of the components or vehicle structure. These simulations are aimed to match the series of tests that a vehicle manufacturer would perform on the proving ground to test the durability of the vehicle and chassis components. Different manufacturers will implement their own procedures but typically these will involve establishing road loads resulting.

The software can link or interface with CAD systems, finite element programs, software used for advanced visualization or additional software modules such as those used for tire modeling. The combined use of these systems can lead to the development of what may be referred to as virtual prototypes that is computer models that can simulate the tests and conditions that a real prototype would be subject to during the development of a new engineering product.

We can use modal analysis to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. It also can be a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic response analysis, or a spectrum analysis.

Objectives: The objective of this thesis is to analyze in terms of component load and stress study, a quarter vehicle of front wheel drive for typical double wishbone suspension system using multi-body simulation system (MBS) together with computer aided design and analysis environment (CAD & CAE).

The scopes of research are as follows:

1. Defining and Building a 3-D Solid parametric model of lower arm, upper arm, wheel, tie rod, and damper component into multi-body system analysis ADAMS/view.
2. Giving correct orientation to each component of suspension assembly.
3. Giving forces input to the wheel at contact patches.
4. Simulate the suspension geometry considering different load cases.
5. Carry out quasi-static analysis for typical load cases (braking and cornering) to predict the load acting on each components
6. Plot table of forces generated on lower arm suspension link.

II. DURABILITY TESTING METHOD

II.1 Durability testing

The loads that are applied to the suspension may be considered to act at the tire contact patch or at the wheel centre depending on the type of load case. Tie loads shown in Table are typical of those that might be used for a static analysis on a vehicle of the type for which data is provided in this textbook [1]. In this example, the loads are defined in the x, y and z directions for a co-ordinate system located at the centre of the tire contact patch as shown in Figure.
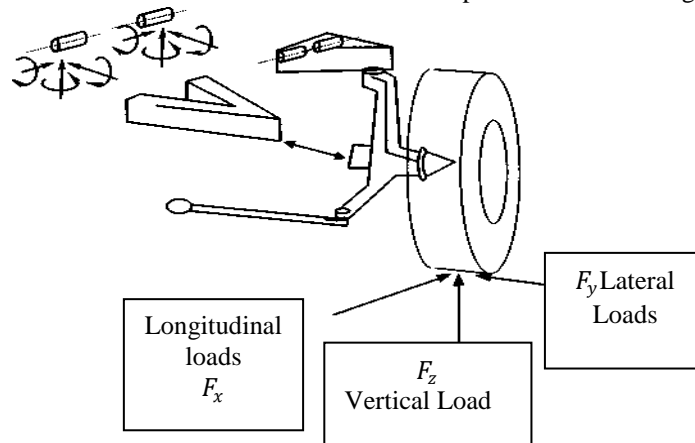

Fig.1   Application of road load at tire contact patch   .

To find out forces acting on front suspension we have to apply below forces in rigid body simulation of suspension in MSC ADAMS from wheel in vertical load direction considering 3g, 2g, 1g test method and same way other load cases are applied in rigid body dynamic analysis from which we get forces acting on lower arm, bump stop, load due to spring on lower arm, and load generated at wheel centre are achieved.

TABLE.I. Loads applied on wheel in respective direction

Load case	$Fx(N)$	$Fy(N)$	$Fz(N)$
3g Bump			11180
2g Rebound			-7460
0.75g Cornering (outer wheel)		4290	5880
0.75g Cornering (inner wheel)		-1180	1620
1g Breaking	5530		5530
0.35 Reverse Breaking	-2150		3330
Kerb Impact		9270	4120
Pothole breaking	15900		12360

For the loads shown in table it is possible to calculate values for cases such as cornering and braking using traditional vehicle dynamics and the principles of weight transfer. For cases involving impacts with kerbs and lumps it may be necessary to obtain instrumented road load measurements m the proving ground. A particularly severe case involves braking while driving through a pothole. To simulate this sort of case the input loads at the contact patch may be set to produce forces, say acting along a tie rod, that are consistent with measured strains on the actual component during the proving ground tests.

II.2 Durability Testing Methods

a) 2g Twist

This is full frame twist load case. This quasi-static load case applies load of 2g at the spindle in vertically upward direction for one front wheel & 2g load at the spindle in vertically upward direction for the diagonally opposite rear wheel. This load case simulates load ditch event in which the two diagonally opposite wheels are momentarily suspended in air.
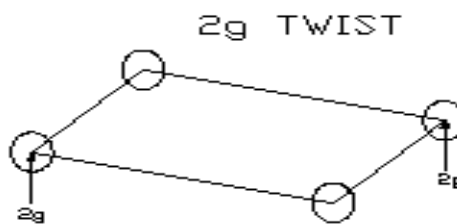

Fig.2   2g Twist

b) 3g Rear

This quasi-static load case applies load of 3g at the spindle in vertically upward direction for both rear wheels. A 1g vertical quasistatic load be applied to the front wheel spindles simultaneously also.
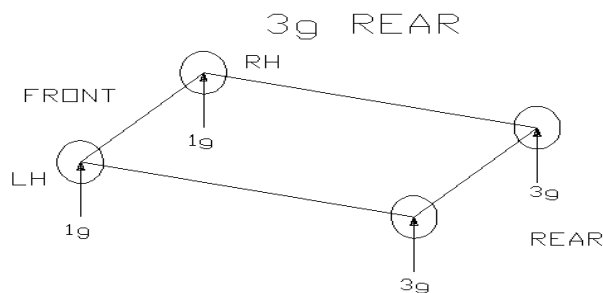

Fig. 3   3g Rear

c) 3g Front

This quasi-static load case applies load of 3g at the spindle in vertically upward direction for both the front wheels. A1g quasi-static load is applied to the rear wheel spindle also.
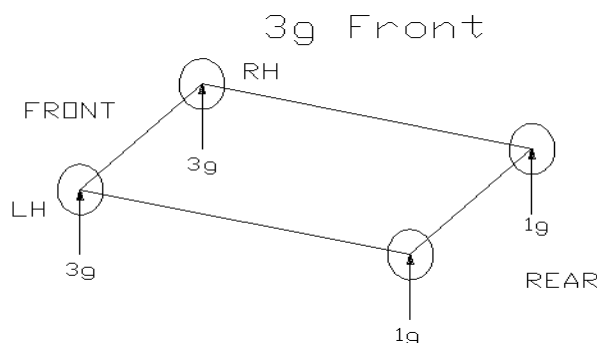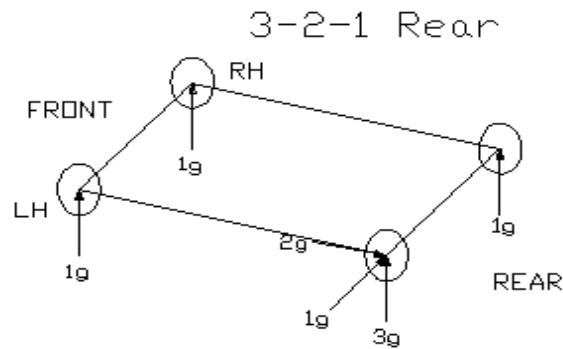

Fig.4   3g Front

d) 3-2-1 Rear

This quasi-static load case applies load of 3g at the spindle in vertically upward direction, 2g load at the spindle in longitudinally rearward direction, and 1g at the tire patch in lateral inboard direction at one of the rear wheels. A 1g vertical quasistatic load would be applied to the outer three wheel spindles simultaneously.
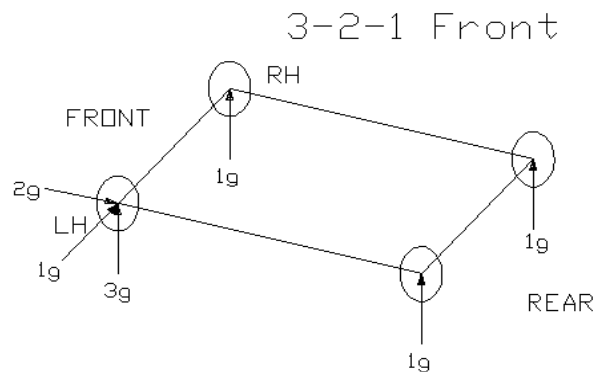

Fig.5 3-2-1 Rear

e) 3-2-1 Front

This quasi-static load case applies load of 3g at the spindle in vertically upward direction, 2g load at the spindle in longitudinally rearward direction, and 1g at the tire patch in lateral inboard direction at one of the front wheels. A 1g vertical quasistatic load would be applied to the other three wheel spindles simultaneously. [8]

Fig.6 3-2-1 Front.

III. ADAMS/View INTERFACE AND JOINT MOTION

A multibody systems analysis program will often provide a number of elements with the capability to model flexibility of bodies and elastic connections between parts. These may include features for modeling beam elements, rubber bushings or mounts, plus a general stiffness and damping field element. At various positions in a model rigid parts can be elastically connected together in preference to using a rigid constraint element such as a joint or joint primitive. Vehicle suspension bushes can be represented by a set of six action–reaction forces, which will hold the two parts together. The equations of force are linear and uncoupled. The user is only required to provide the six diagonal coefficients of stiffness and damping. For more complicated cases a general-purpose statement can be used to provide a linear or non-linear representation of a flexible body or connection.

Adams/View is a powerful modeling and simulating environment. Users can use Adams/View to build and simulate a model of any mechanical system that has moving parts. You can test your model in the same environments your actual product will experience.

You can also validate your model against test data, and refine your model with flexible parts, realistic geometry, and controls. Finally, you can find the important parameters that affect your product design and find the right combination of design values to arrive at the best product design. Best of all, you can do all this over and over again, saving time and money.

This release we are introducing an updated Graphical User Interface (GUI). This includes changes to the Start Screen where you are now provided with the option to create new model or import an existing model.

The Fig.7 shows the Adams/View interface in which there are different tools available which can be used for the dynamic simulation application.
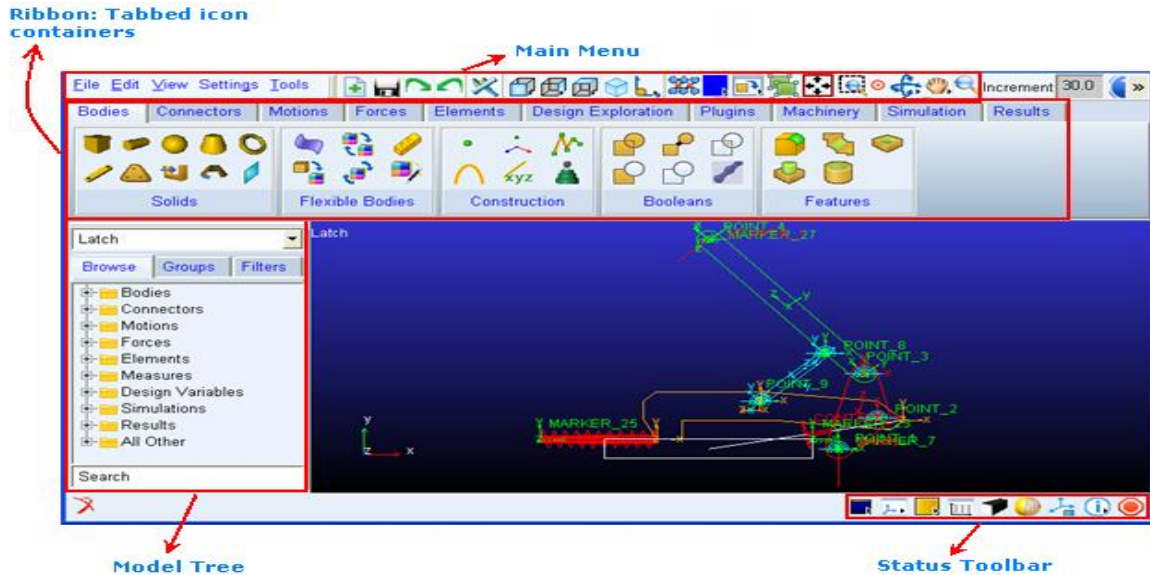

Figure7. Adams/View interface

III.1 Overview of Motion

A motion generator dictates the movement of a part as a function of time. It supplies whatever force is required to make the part satisfy the motion. For example, a translational joint motion prescribes that a joint on a part move at 10 mm/second in the z direction. You can apply the motion to either idealized joints or between a pair of parts.

III.1.1 Types of Motion

Adams/View provides you with the following types of motion:

- *Joint Motion* - Prescribes translational or rotational motion on a translational, revolute, or cylindrical joint. Each joint motion removes one degree of freedom (DOF) from your model. Joint motions are very easy to create, but they limit you to motions that are applied to the above listed joints and movements in only one direction or rotation.

- *Point Motion* - Prescribes the movement between two parts. When you create a point motion, you specify the direction along which the motion occurs. You can impose a point motion on any type of idealized joint, such as a spherical or cylindrical. Point motions enable you to build complex movements into your model without having to add joints or invisible parts. For example, you can represent the movement along an arc, of a ship in the ocean, or a robot's arm.
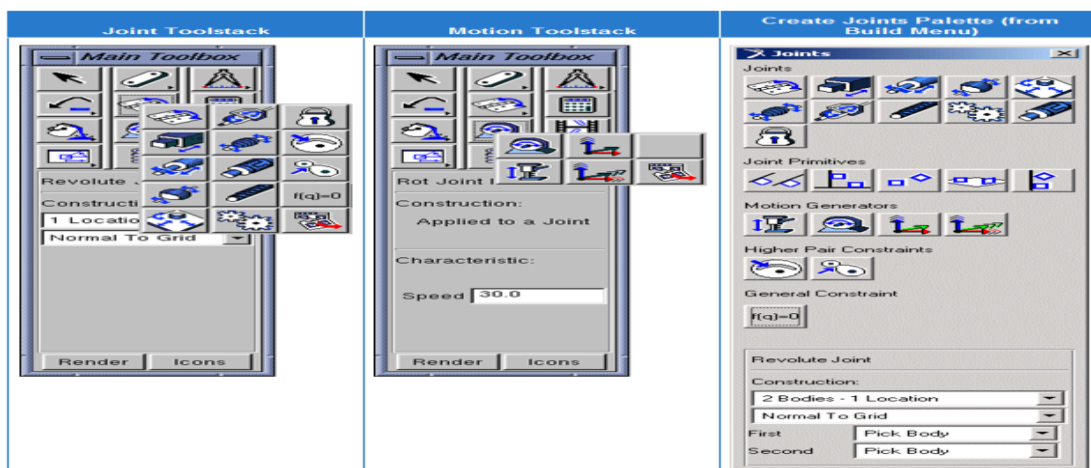

Figure8. Joint Palette and Joint and Motion Tool Stacks.

III.1.2   *Defining the Motion Magnitude*

You can define motion as acceleration, displacement, or velocity over time. By default, Adams/View creates a motion that moves at a constant velocity over time. When you create a motion, you can define its magnitude by entering one of the following:

- *Translational or rotational speed***:**
  As you create a motion, you can specify the translational or rotational speed of the motion. By default, you enter the rotational speed in number of degrees per second and the translational speed in length units per time unit (for example, number of inches per second).
  When Adams/View creates the motion, it uses the value you enter as the motion function. It also converts the rotational motion speed to radians. When you modify the motion, you can change the value or enter a function expression or a user-written subroutine as explained next.

- *Function expression:*
  You can use Adams/View function expressions to specify the exact movement applied to a joint as a function of time. For example, using function expressions you can define a motion function that holds the joint in a fixed position, as well as one that moves the joint with the required force to produce a constant velocity.

- *Parameters to be passed to a user-written subroutine:*
  You can create a much more complex motion by creating a MOTSUB User-written subroutine and entering the values to be passed to the subroutine to determine the motion.

IV.   ADAMS/View MODELLING OF SUSPENSION LINK

ADAMS/view has ability to realistically view the behavior of complicated mechanical systems. It is important the imported geometry is currently open and displayed in Adams/View.

IV.1   *Steps adopted for Simulation*

The steps mentioned as under

- Import geometry from Pro-E
- Define the Material
- Define the joints and motion to individual components
- Define the flexible Connectors to measure dynamic characteristic
- Post processing in ADAMS

The geometry of suspension link refines in to PRO-E (solid modeling software). The unnecessary parts are suppressed or removed from the assembly after refine. The refined assembly is exported into parasolid file format.

The first step in any simulation is to prepare a data set, which will define the system being modeled. This will include a description of the rigid parts, connecting joints, motion generators, forces and compliances. With most codes the data set is user friendly in that the data statements are easily understood with few restrictions on format. For each rigid body in the system it is necessary to define the mass, centre of mass location, and mass moments of inertia. Each body will possess a set of co-ordinates, which can be defined in global or local co-ordinate systems and are considered to move with the part during the simulation. These points are used to define centre of mass locations, joint locations and orientations, force locations and directions. The relative motion between different parts in the system can usually be constrained using joints, joint primitives, couplers, gears and user-defined constraints.

Then Import the geometry in the ADAMS View software and again check the orientation & position of the assembled components. Once it is imported without error the material property is applied as an input.

The below figure shows the suspension model in Adams/View and it is includes the different types of joints and motion, the motion will be applied by using the User-written subroutine and entering the values to be passed to the subroutine to determine the motion. And below figure indicate solid geometric of suspension model.

The next step in building the model would typically be the definition of external forces and internal force elements. External forces can be constant, time histories or functionally dependent on any state variable. These forces can also be defined to be translational or rotational. They can act in the global system or can act in the local system of the body so that they effectively 'follow' the part during the simulation.

Users can also set up internal force elements acting between two parts to represent springs, dampers, cables or rubber mounts. Internal force elements will always act along the line of sight between the points the force element connects on the two parts. These force elements are often referred to as action–reaction forces as they always produce equal and opposite forces on the two parts connected by the force element. The elements can also be defined to act in only tension or compression and may be linear or non-linear.
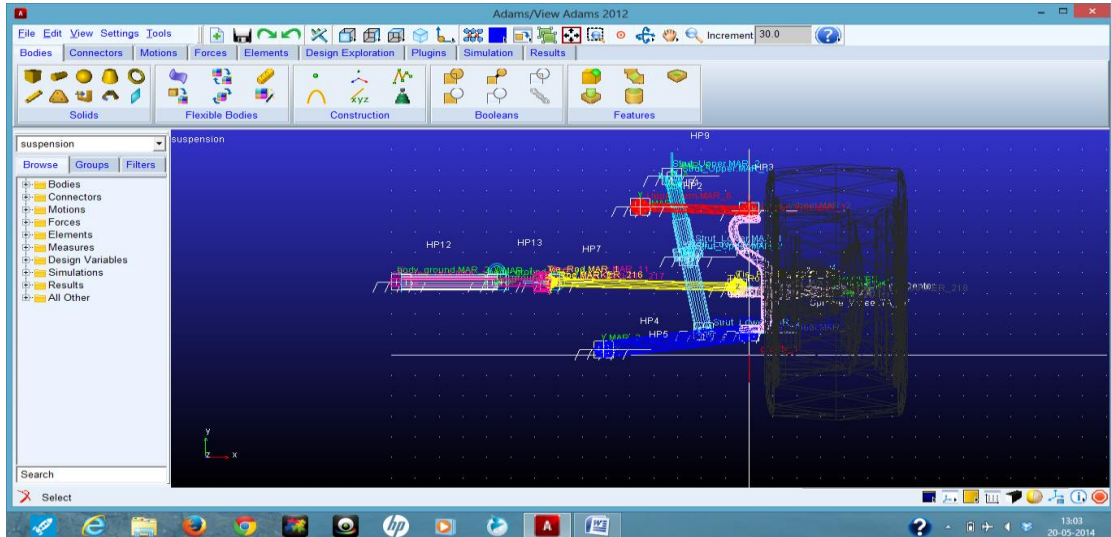

Figure 9. Suspension Model in ADAMS/View

Below Figure indicate the simulation script of suspension model in which forces cases are applied to the wheel from which it generate the forces that are developed on each part.
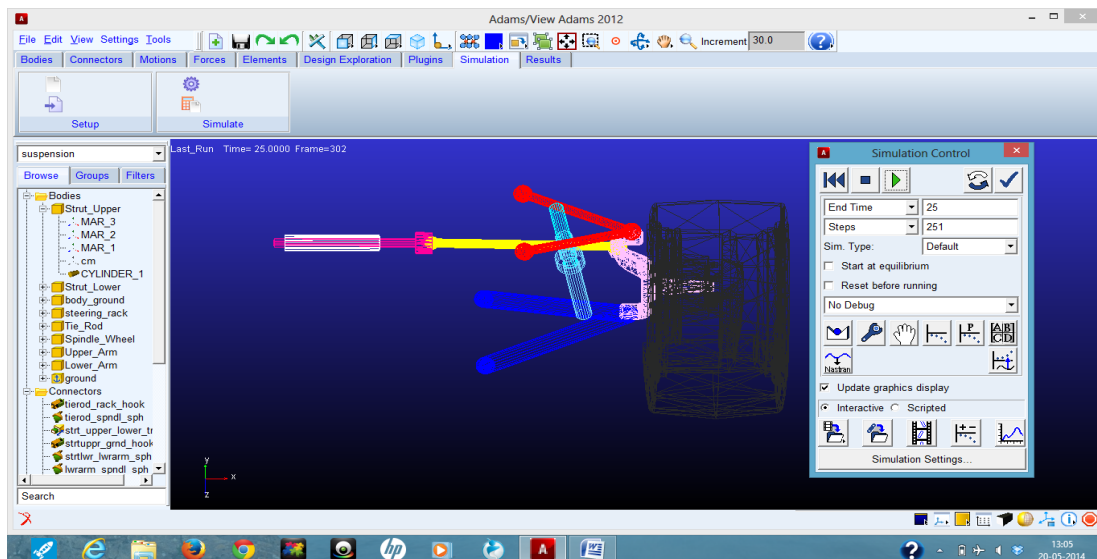

Figure10. Simulation script of suspension assembly in ADAMS/view.

V. OUTPUT OF RIGID BODY SIMULATION FOR DIFFERENT LOAD CASES

V.1 SUV Two Wheel Drive (SC 2WD) parameter considered:

Gross Vehicle Weight (GVW) (kg):–   3390 (fully loaded)
Front Axle Wheel (FAW) (Kg):-   1255
Rear Axle Wheel (RAW) (Kg):-   2135

The load generated on lower arm, bum stop, wheel centre and load due to spring are tabulated as result of rigid body dynamic simulation considering different load cases. The generated model of suspension is simulated with different load cases and it generates forces on component of suspension. And forces generated on lower arm are used for modal analysis of lower arm. After which this forces are used for optimization of component.

TABLE.II Loads on lower arm of suspension in different loading case in respective direction.

		Front			Rear		
		X	Y	Z	X	Y	Z
1	2g twist	553.171	-5022.88	1915.34	669.3489	-9233.69	-14338.9
2	3 2 1 front	-3078.79	-12773.5	5172	-2914.84	-35679.5	-19216.8
3	3 2 1 rear	348.582	-2532.29	436.11	399.629	-3808.48	-6558.01
4	3g front	661.79	-7483.3	3416.89	839.91	-14903.6	-22050.5
5	3g rear	**354.26**	**-2552.97**	**438.92**	**405.13**	**-3801.32**	**-6530.58**

Lower control arm (LCA) L.H.

TABLE.III Loads produced on lower arm due to spring

		X	Y	Z
1	2g twist	-305.484	180.407	4758.91
2	3 2 1 front	-140.907	240.857	4924.57
3	3 2 1 rear	-280.134	199.007	4520.13
4	3g front	-318.3	178.16	4848.2
5	3g rear	**-280.2**	**199.34**	**4517.25**

Forces of Spring

TABLE.IV Loads at bump stop of lower arm.

		X	Y	Z	X	Y	Z
1	2g twist	-400.694	7586.66	22571.1	285.618	-2.211	1345.93
2	3 2 1 front	-2595.32	12557.8	39536.6	302.556	-0.019	1438.71
3	3 2 1 rear	-197.614	2868.48	8898.6	832.137	-6.327	3400.08
4	3g front	-508.92	12374.5	36324.3	301.91	0	1436.13
5	3g rear	**-195.96**	**2853.36**	**8856.36**	2692.67	0	9551.84

Forces produced at Bump stop L.H.

TABLE. V Loads Produced at wheel centre

|		Forces produced at wheel Center			
		Front		Rear	
		LH	RT	LH	RT
1	2g twist	(+z)12311			(+z)12311
2	3g front	(z+)18467	(z+)18467	(z+) 10472	(z+) 10472
3	3g rear	(z+) 6155	(z+) 6155	(z+) 31416	(z+) 31416
4	3 2 1 front	(z+) 18467	(z+) 6155	(z+) 10472	(z+) 10472
	longitudinal	(x+)12311			
	Inboard	(y+) 6155			
7	3 2 1 rear	(z+) 6155	(z+) 6155	(z+) 10472	(z+) 10472
			Longitudinal	(x-)-20944	
			Inboard	(y-)-10472	

For modal analysis of suspension link the colored loads in tables are used for analysis in ANSYS.

VI. MODAL ANALYSIS

Modal analysis determines the natural frequencies and mode shapes of a structure. The natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions. They are also required if we want to do a spectrum analysis or a mode superposition harmonic or transient analysis.

We can do modal analysis on a prestressed structure, such as a spinning turbine blade. Another useful feature is modal cyclic symmetry, which allows you to review the mode shapes of a cyclically symmetric structure by modeling just a sector of it.

Modal analysis in the ANSYS family of products is a linear analysis. Any nonlinearity, such as plasticity and contact (gap) elements, are ignored even if they are defined. We can choose from several mode-extraction methods: Block Lanczos (default), subspace, Power-Dynamics, reduced, unsymmetric, damped, and QR damped. The damped and QR damped methods allow you to include damping in the structure. The QR Damped method also allows for unsymmetrical damping and stiffness matrices.
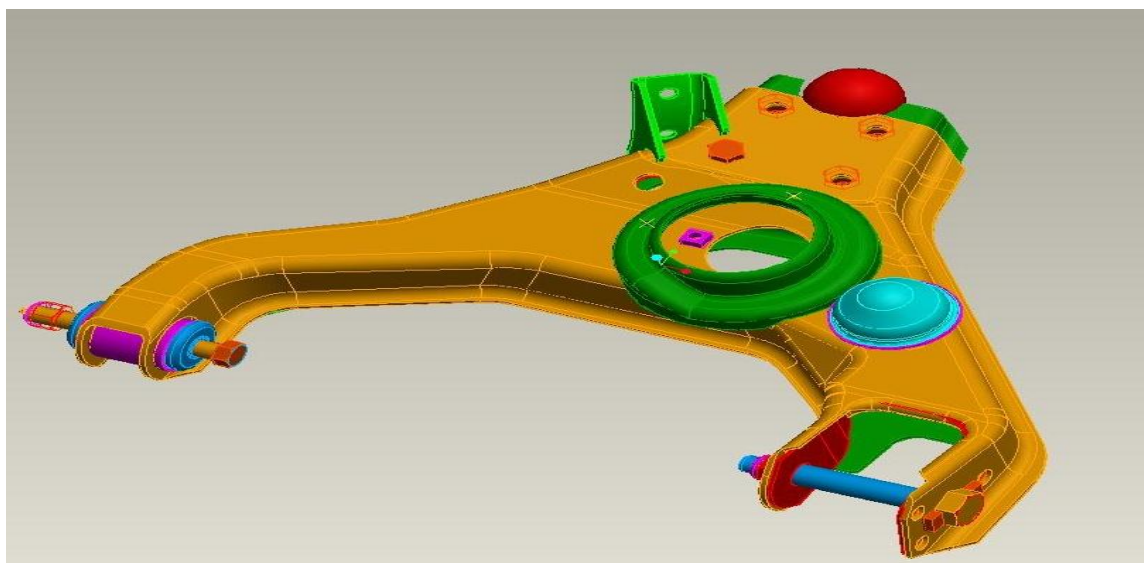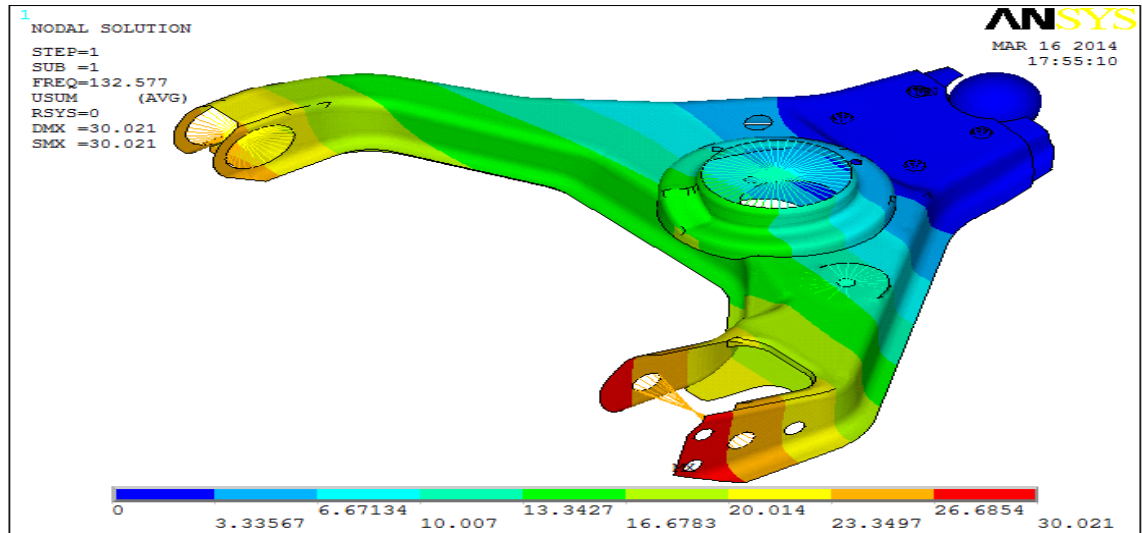

Fig. 11 Solid Modeling of Suspension link
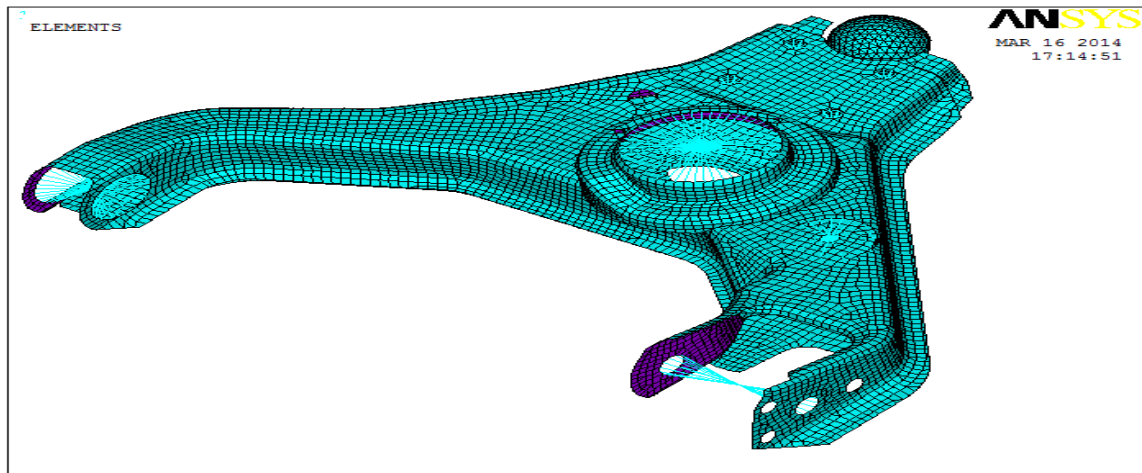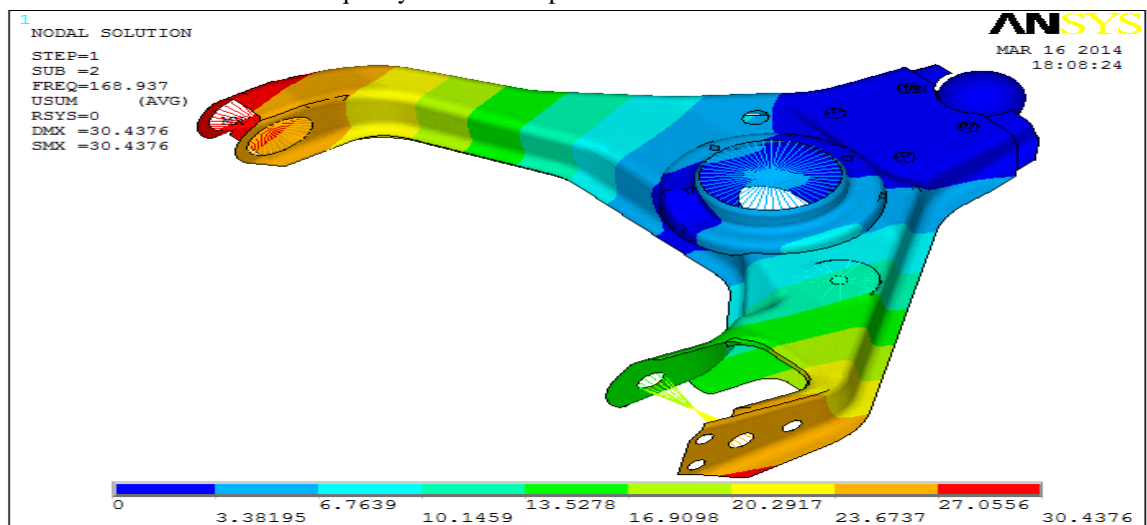
Fig.12 meshed model of suspension link

A) Deformation Plot For 1st natural frequency Mode of Suspension link

Fig.13 Mode shape and natural frequency for the rolling vibration mode

From above plot we found the maximum deflection is 30.021 mm and minimum deflection is 3.33567 mm.
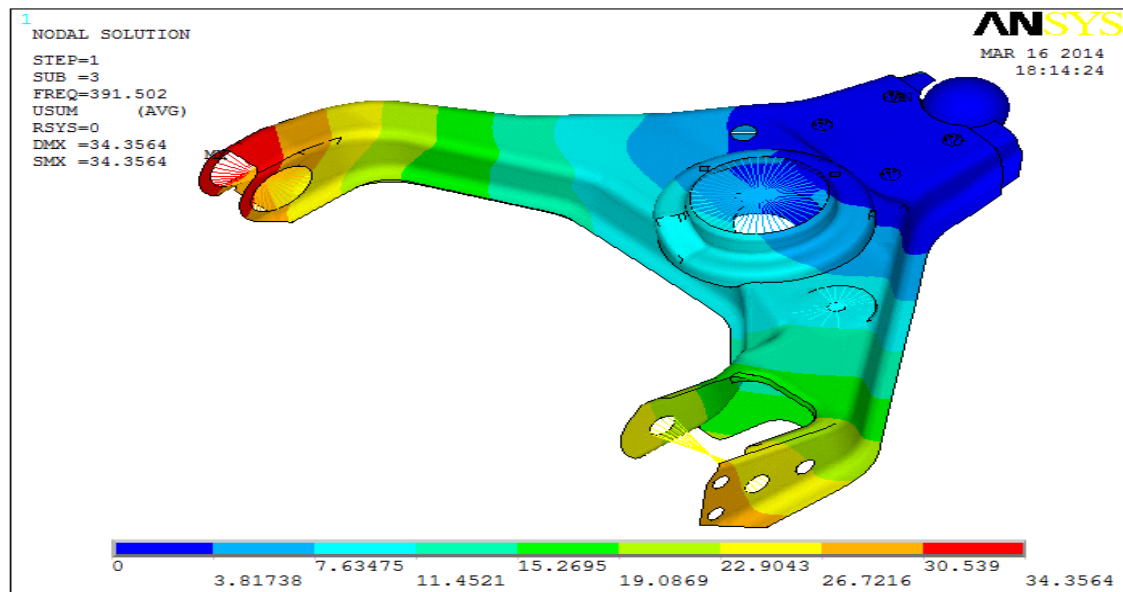
B) Deformation Plot For 2nd natural frequency Mode of Suspension link

Fig.14 Mode shape and natural frequency for the pitching vibration mode

From above plot we found the maximum deflection is 30.4376 mm and minimum deflection is 3.38195 mm.

C) Deformation Plot For 3rd natural frequency Mode of Suspension link

Fig.15 Mode shape and natural frequency for the breaking vibration mode

From above plot we found the maximum deflection is 34.3564mm and minimum deflection is 3.81738 mm.

VII. DICUSSION AND CONCLUSION

1. We have just done the calculation based on different load case to find out the natural frequencies and different modes which can be useful to study the dynamic behaviors of the lower suspension arm.
2. Result of the natural frequencies at different modes which can be useful to study the dynamic behaviors of the lower suspension obtained as follows.
   - Natural frequency in rolling mode = 168.9Hz
   - Natural frequency in pitching mode = 132.5Hz
   - Natural frequency in breaking mode = 391.5Hz
3. From the frequency calculation normally it seems the lower arm is very stiff in breaking mode, i.e. higher the frequency higher the stiffness, the frequency is directly proportional to K and inversely proportional to Mass. The natural frequency for breaking mode is 391.5 Hz compared to natural frequencies respectively of rolling and pitching 132.5 HZ & 168.9 HZ.
4. And rolling and pitching mode frequency is lower so component captures higher stress during dynamic operating condition and leads to failure.

VIII. FUTURE SCOPE

1. Static Structural analysis and dynamic analysis using ANSYS for various force and moment that exert on lower left arm.
2. The von-misses stress analysis evaluation and if the stress is higher than the safe limit then need to optimization either by area by providing an extra material condition or design change.
3. If we want to increase dynamic strength of the component need to increase the natural frequency to as higher as possible so natural frequency for rolling and pitching will be increased above 132.5 HZ & 168.9 HZ.
4. Generate parametric model of lower wishbone arm for its geometric Optimization will be presented in next research paper.

REFERENCES

Books:

[1] "Multibody Systems Approach to Vehicle Dynamics", Michael Blundell, Damian Harty, SAE International.

[2] Ansys Inc., "Ansys 15 Documentation, Structural Analysis Guide", Swanson Analysis System, United States.

[3] "CAE Structural Approach", Lotus, Ultra Light Steel Auto Suspension 2004.

Journals paper:

[1] Edmund, F. G., Salinas, A. R., 2003, "*Introduction to Formula SAE Suspension and Frame Design", SAE, Paper No. 971584.*

[2] Gadhiya Utsav D and Prof. Sumant P.Patel; *Quarter Model Analysis of Wagon-R car's Rear Suspension using ADAMS (IJERT)ISSN: 2278-0181, Vol. 1 Issue 5*, July – 2012

[3] Jihui Liang and Lili Xin;*simulation analysis and optimization design of front suspension based on ISSN 1392-1207. MECHANIKA. 2012 volume 18(3):* 337-340.

[4] S.Pthmasharma, J.K. suresh, P.Vishwanathan and R. Subramanian;*Analysis of passenger car suspension system using ADAMS.ISSN: 2278-7798. International journal of science, engineering and technology research, volume 2, ssue 5, may 2013*

[5] Xiaobin ning, cuiling zhao and jisheng shen;*Dynamic analysis of car suspension using ADANS/CAR for development of a software interface for optimization. Procedia engineering 16 (2011) 333-341.*

[6] Yu hai-bo, Li you-de, MEN yu-zhuo;*Kinematic characterization and optimization of vehicle front suspension design based on ADAMS .Journal of Chongqing university(English edition) ISSN 1671-8224 volume 7 no.* 1 , march 2008. Article id: 1671-8224(2008)01-0035-06.

[7] Hemin M M, Rahman M M and Omar R M ;*Dynamic Analysis of Vehicle Arm Based on Finite Element Approach Article info: journal of advanced science and engineering research 1(2011)124-136.*

[8] Hemin M. Mohyaldeen, M. M. Rahman, M. M. Noor and K. Kadirgama; *Suspension arm design based on stochastic optimization approach.*University Malaysia Pahang, 26300 UMP, Kuantan, and Pahang, Malaysia.

[9] Z. Husin, M.M. Rahman, K. Kadirgama, M.M. Noor and Rosli A.Bakar; *Prediction of fatigue life of lower arm subjected to variable amplitude loading*. Faculty of Mechanical Engineering, Malaysia Pahang.

[10] *Vehicle Dynamic Analysis Using Virtual Proving Ground Approach* by Han Ki Min *527-810*, Korea .Gi Seob Choi CAE Department, *506-302*, Jong An Jung, *502-742*,In Young Yang, *375* Seosukdong, Donggu, Kwangju *501-795*, Korea.