Cleat Impact Analysis of a Multi-body Quarter Vehicle System with Finite Element Models

Myeong Jae Han, Chul Hyung Lee and Tae Won Park
Department of Mechanical Engineering, Ajou University, Suwon, Republic of Korea
Email: hanmj2, okttg91, park@ajou.ac.kr

Young Sik Kim and Kyoung Duck Shin
R&D, Hands Corporation, Incheon, Republic of Korea
Email: yskim, kdshin@handscorp.co.kr

Kyung Seok Sim
Premium Vehicle R&D Center, Korea Automotive Technology Institute, Yeongam, Republic of Korea
Email: kssim@katech.re.kr

***Abstract*—The main purposes of the present work represent a modeling of a multi-body quarter vehicle system and a methodology of a cleat impact simulation. The multi-body quarter vehicle is modeled as a finite element. The non-linear tire model is a simple 3D solid type. The parts of the quarter vehicle were connected by a joint element. The spring-dashpot joint element was used to consider the effect of the suspension. A transient dynamic analysis was performed to confirm the vibrational result of the cleat impact. The analysis step consists of four parts: tire inflation, stabilization, tire-road interaction, and cleat impact.**

***Index Terms*—cleat impact analysis, finite element model, multi-body dynamic, non-linear tire, quarter vehicle**

I. Introduction

The cleat impact test is performed mainly to confirm the vibration and noise characteristics transmitted to the wheel-tire and suspension by impact excitation [1], [2] and [3]. The impact bar, called cleat, is mounted on the bottom of the drum dynamometer that the tire is in contact with. The cleats are used in different sizes depending on the magnitude of the impact excitation.

The quarter vehicle is the basis for analyzing the dynamic characteristics of the full vehicle system. The general quarter vehicle analysis is a rigid-body dynamics model consisting of sprung mass, un-sprung mass, spring, and damper [4] and [5]. However, rigid-body dynamics model can only confirm vibration displacement and acceleration results. Also, since the rigid-body dynamic model has no degree of freedom to consider local deformation of the actual shape, it is only possible to analyze the vibration in the low frequency band.

In this study, the finite element analysis model of quarter vehicle was developed to perform the cleat impact analysis. It is possible to perform not only the basic dynamic analysis but also the stress analysis by the finite degree of freedom of the nodal points of each part. Also, the non-linear tire was used for large deformation by frictional contact with roads. The kinematic motion of the multi-body quarter vehicle was confirmed.

II. Problem Formulation

A. Finite Element Model

The quarter vehicle analytical model consists of a McPherson strut suspension, lower control arm, hub and knuckle. The wheel-tire in contact with the road was also constructed. The 3D CAD model was reconstructed by acquiring the shape information of the actual vehicle part by the reverse engineering method. The reversely designed CAD model has the advantage of minimizing the number of mesh nodes for finite element analysis by simplifying complex non-linear shape such as lines, surfaces, and elements. The reduction of the number of mesh nodes is essential in the range of guaranteeing the error of the analysis result because the memory storage problem and the calculation time can be effectively reduced as the analysis scale becomes larger in the finite element analysis method. The finite element and material property information of the quarter vehicle parts is shown in Table I. The total number of nodes and elements in the analysis model is 88,846 and 187,549, respectively.

B. Joint Model

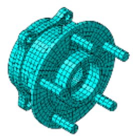
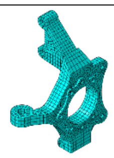
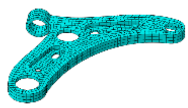
Each body of a multi-body system is connected to a different joint depending on the degree of freedom of kinematic motion. Four joints were used in the quarter vehicle analysis model. First, the fixed joint constrains both translational and rotational degree of freedom. The upper end of the suspension spring was fixed to the reference point. Second, the hinge joint has only one-axis rotational degree of freedom. The center node of the wheel-tire is connected by the hinge joint. Third, the translational joint has only one-axis translational degree of freedom. The hub and knuckle are connected by a

Manuscript received December 5, 2018; revised June 28, 2019.

doi: 10.18178/ijmerr.8.6.880-884

translational joint to allow a slight clearance movement. Fourth, the bushing joint has both translational and rotational degree of freedom. The bushing joint with damping coefficient is used for the mount and knuckle connection position of the lower control arm. Unlike the joint function for basic kinematic degree of freedom, the spring-dashpot joint element is used which can take both stiffness and damping effects together. The spring-dashpot joint element used as a vehicle's suspension is a major factor in controlling ride comfort. The spring stiffness and dashpot damping coefficients are 2,000 N/m and 350 N-s/m, respectively.

TABLE I. FINITE ELEMENT MODEL INFORMATION

Part	Finite element model	Material property
Wheel		Young's modulus, E: 70 (GPa) Poisson's ratio, v: 0.3 Density, ρ: 2,680 (kg/m^3) No. of nodes: 38,741 No. of elements: 155,329
Tire		Hyper-elastic material Density, ρ: 1,400 (kg/m^3) No. of nodes: 30,590 No. of elements: 20,140
Hub		Young's modulus, E: 210 (GPa) Poisson's ratio, v: 0.27 Density, ρ: 7,400 (kg/m^3) No. of nodes: 5,759 No. of elements: 3,357
Knuckle		Young's modulus, E: 75 (GPa) Poisson's ratio, v: 0.25 Density, ρ: 2,750 (kg/m^3) No. of nodes: 5,220 No. of elements: 3,215
Strut		Young's modulus, E: 170 (GPa) Poisson's ratio, v: 0.25 Density, ρ: 7,200 (kg/m^3) No. of nodes: 5,842 No. of elements: 4,273
Lower control arm		Young's modulus, E: 110 (GPa) Poisson's ratio, v: 0.3 Density, ρ: 5,820 (kg/m^3) No. of nodes: 2,694 No. of elements: 1,235

C. *Non-linear Tire*

A rubber is a hyper-elastic material that can be stretched about 300 % in the stress-strain relationship curve. To consider the hyper-elastic effect, the material property is defined as a strain energy density function rather than a stress-strain relationship [5] and [6].

There are various strain energy density functions expressing the hyper-elastic effect of the rubber material [7] and [8]. The Yeoh model is a strain energy density function in the form of third-order polynomial based on the first invariant I_1 [9]. The Yeoh model has good fitting results over a large strain range and follows various deformation modes for limited experimental data [10].

The form of the Yeoh model with compressible terms is shown in Eq. (1). If incompressible, the second term is omitted.

$$U = \sum_{i=1}^{3} C_{i0}(\bar{I}_1 - 3)^i + \sum_{i=1}^{3} \frac{1}{D_i}(J^{el} - 1)^{2i} \qquad (1)$$

where U is the strain energy per unit of reference volume, C_{i0} and D_i are temperature-dependent material coefficients, $\bar{I}_1$ is the first deviatoric strain invariant defined in Eq. (2).

$$\bar{I}_1 = \sum_{i=1}^{3} \bar{\lambda}_i^2 \qquad (2)$$

where the deviatoric stretches $\bar{\lambda}_i = J^{-\frac{1}{3}}\lambda_i$, J is the total volume ratio, J^{el} is the elastic volume ratio as defined in thermal expansion, and λ_i are the principal stretches. The initial shear modulus and bulk modulus are given in Eq. (3) and (4).

$$\mu_0 = 2C_{10} \qquad (3)$$

$$K_0 = 2/D_1 \qquad (4)$$

In commercial software Abaqus® Ver. 6.13, the strain energy density function is defined from uniaxial, biaxial, planar and volumetric test data for isotropic materials. In some cases, good fitting results can be obtained with only uniaxial test data. The coefficients of the strain energy density function can be obtained by evaluating test data using the Abaqus user interface. If a specific rubber material is used, numerical convergence problems arise in a large strain range. Therefore, the strain verification for the hyper-elastic material is essential using the Abaqus user interface. The stress-strain relationship curve for hyper-elastic material obtained from the calculated Yeoh model coefficients (C_{10}: 0.7325, C_{20}: -0.008972, C_{30}: 0.0001749) is shown in Fig. 1.

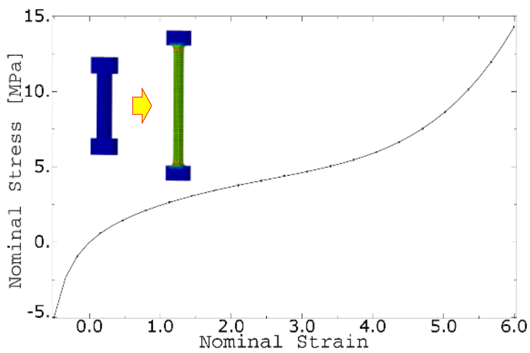


Figure 1. Nominal stress-strain relationship curve for the hyper-elastic materials from Yeoh model coefficients.

The finite element model of a tire is usually designed as a 3D axisymmetric model. However, it is difficult to obtain the exact specification of the structure and material of the tire that only the tire manufacturer knows. In this study, actual tires are modeled as simple tires with four elements for the efficiency of analysis. Simplified tires were modeled by assigning the properties of each element to the finite element tire model of 3D solid type without directly modeling each tire element. Therefore, the simplified tire is composed of one part, so there is no need to consider the contact and constraint between each element. The main characteristics of the tire components are considered through the four components. First, the tread is a thick layer of rubber that is in contact with the road surface. The tread protects the internal carcass against road impact. Second, the carcass is the structure frame of the tire. The carcass must be strong enough to withstand air pressure and flexible to absorb load changes and shocks. Third, the sidewall is the side of the tire. The sidewall resists repeated shrinkage and expansion during vehicle driving and protect carcass from moisture and friction. Fourth, the bead is the part that joins the tire and the rim. The bead fixes the tire to the rim to prevent air leakage. The finite element model of the simplified tire is shown Fig. 2. The hyper-elastic material was applied to the tread and sidewall where the deformation occurred largely, and the material of the elastic region was applied to the carcass and the bead.

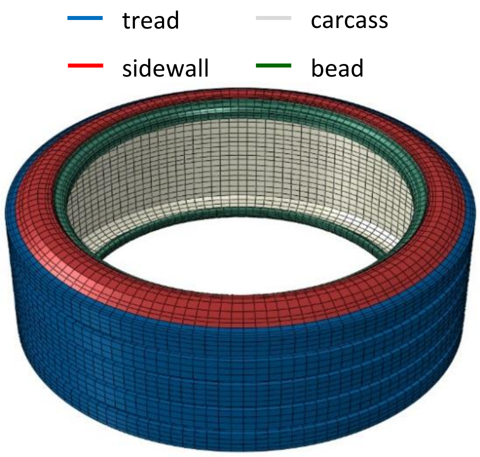


Figure 2. Finite element model of simplified tires.

III. Cleat Impact Analysis

A. Boundary Conditions

The cleat impact analysis is a method for evaluating vibration transmitted to a vehicle suspension while the tire passes through the cleat attached to the road. The actual indoor cleat impact test is performed by the drum dynamometer. However, the purpose of this analysis is to model the kinematic motion of the quarter vehicle and the deformation of the non-linear tire. Therefore, the drum model was replaced by a simple planar road model. The basic specifications for performing the cleat impact analysis are in Table II.

TABLE II. Analysis Specification

Vehicle speed	60 kph
Tire air pressure	32 psi
Cleat size	10 mm × 10 mm
Road size	400 mm × 5340 mm

The cleat impact analysis consists of four steps. First, the tire inflation step is the process of inflating the tire by applying a constant pressure to the inner surface of the tire. Second, the stabilization step applies gravity to the analysis model. The displacement of the quarter vehicle is stabilized by the suspension. Third, the tire-road interaction step is the contact process between the tire and the road. Fourth, the cleat impact step is the process by which the tire passes through the cleat. The cleat impact causes vibration in the quarter vehicle. All the analysis steps were performed by transient dynamic. The boundary conditions of the cleat impact analysis are shown in Fig. 3.

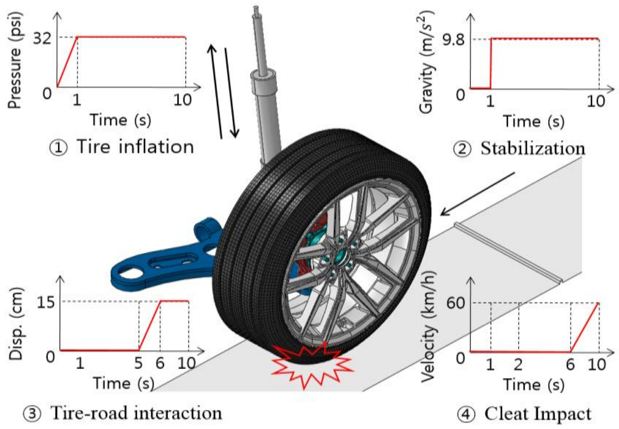


Figure 3. Boundary conditions of the cleat impact analysis.

B. Results

The advantage of the finite element analysis of the quarter vehicle is the result of the stress analysis by the local deformation mode which cannot be confirmed by the rigid-body dynamics analysis. The large deformation of hyper-elastic material can be confirmed by finite element analysis. Unlike other parts, tires undergo large deformation when applying boundary conditions. Therefore, analytical verification is required to confirm the strain and contact pressure distribution due to tire deformation. Fig. 4 shows the result of the strain analysis of the tire in contact with the road surface. The tread in contact with the road surface and the sidewall inflated by air pressure have a relatively large strain.

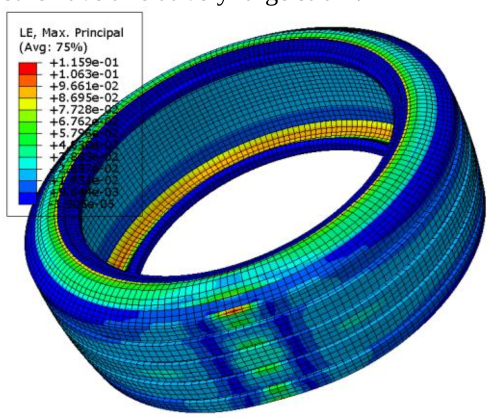


Figure 4. The strain of the tire.

The contact pressure distribution of the tire generated in contact with the road surface is formed to be similar to the actual tire’s footprint. Therefore, the contact pressure distribution is an important result to confirm the rolling friction distribution of the tire tread. The result of the contact pressure distribution of the tire is shown in Fig. 5. The contact pressure is concentrated in the tread pattern around the groove. Also, a relatively large contact pressure is concentrated at both ends of the frictional contact surface by the sidewalls and the carcass structure.

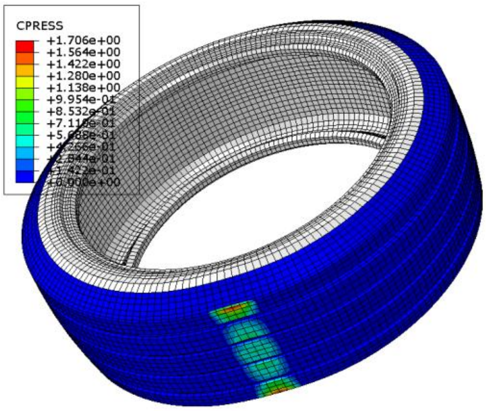


Figure 5. The contact pressure distribution of the tire.

The stress contour result of the finite element quarter vehicle analysis model is shown in Fig. 6. The quarter vehicle analysis model is assembled with each part components. Stress occurs in the wheel because it is directly connected to the low-stiffness tire, so that the reaction force against the road surface is concentrated on the high-stiffness wheel. Among them, the stress concentration mainly occurs in the part where the tire and the bead are connected by face-to-face tie constraint. Linear velocity boundary conditions were applied to the road model for friction rolling. Controlling the road model with a relatively low degree of freedom can improve the analysis efficiency.

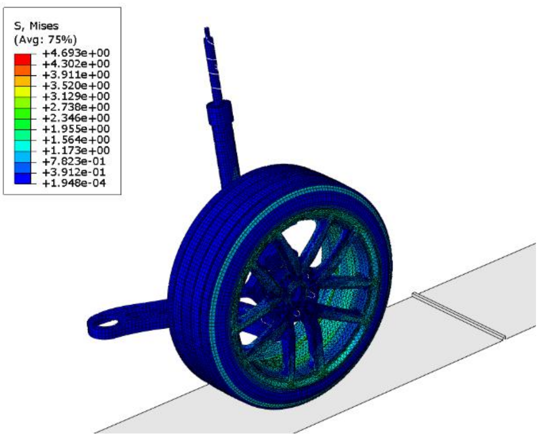


Figure 6. The stress result of the quarter vehicle.

The displacement and acceleration result of the quarter vehicle for the entire analysis step is shown in Fig. 7 and Fig. 8. In the tire inflation step (t = 0 ~ 1s), the tire air pressure acts on the analysis model, but the directions are opposite to each other, so they are canceled out and the external force is zero. This step is the static analysis. In the stabilization step (t = 1 ~ 5s), the position of the quarter vehicle converges to about -24 mm due to the effects of gravity and suspension. The magnitude of the vibration can be controlled by the stiffness and damping coefficient of the suspension. In the tire-road interaction step (t = 5 ~ 6s), the displacement of the quarter vehicle increases to the initial displacement value by giving the displacement boundary condition in the upward direction on the road. The contact vibration occurs between the road and the tire. Because of the relatively low impact contact conditions, the contact vibration is absorbed and dissipated by the strain energy of the tire itself without the suspension effect. In the cleat impact step (t = 6 ~ 10s), the impact response due to the cleat occurs in a very short time. A displacement of about 10 mm occurs, which is approximately equal to the cleat size. After passing through the cleat, the residual vibration lasts for about 1 second and then disappears. Also, the cleat impact has two large peak vibrations and residual vibrations due to the rebound motion after passing the cleat.

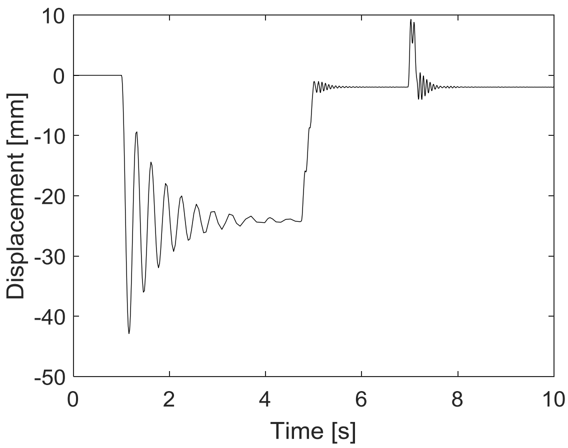


Figure 7. The displacement result of the quarter vehicle.

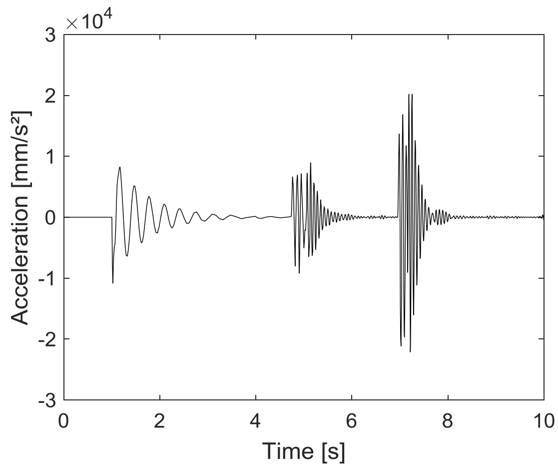


Figure 8. The acceleration result of the quarter vehicle.

IV. Conclusion

The finite element analysis model was developed to perform the cleat impact analysis of the multi-body quarter vehicle. Each part was connected by joint considering kinematic degrees of freedom. Suspension was modeled using spring-dashpot joint elements with stiffness and damping coefficients. The experimental coefficients of the strain energy density function to define the hyper-elastic material of rubber were calculated. The simplified non-linear tire model with hyper-elastic material was developed. The non-linear tire was verified by confirming the strain and the contact pressure distribution. Finally, the cleat impact analysis of the quarter vehicle model was carried out to confirm the stress, displacement and acceleration. The developed quarter vehicle analysis model can be used for design parameter study of tire and suspension for ride comfort analysis. Also, if the dynamic result of the road impact is used as the input data of the acoustic analysis simply through the frequency conversion, the acoustic analysis by the road impact will be possible.

ACKNOWLEDGMENT

This work has been financially supported by the Ministry of Trade, Industry and Energy (MTIE), “Development of low-priced sound absorbing wheel for improving NVH performance (N063600013)."

REFERENCES

[1] P. Kindt, F. De Coninck, P. Sas, and W. Desmet, “Analysis of tire/road noise caused by road impact excitations,” *SAE Technical Paper., 2007-01-2248, 2007*.

[2] C. S. Cristian, M. G, Catalin, D. Nicolae, and O. O. Victoria, “Finite element quarter vehicle suspension model under periodic bump and sinusoidal road excitation,” *Applied Mechanics and Materials. Achievements and Solutions in Mechanical Engineering.*, vol. 880, pp. 163-170, Mar. 2018.

[3] R. R. V. Neves, G. B. Micheli, and M. Alves, “An experimental and numerical investigation on tyre impact,” *International Journal of Impact Engineering.*, vol. 37, no. 6, pp. 685-693, Mar. 2010.

[4] C. Sandu, E. R. Andersen, and S. Southward, “Multibody dynamics modelling and system identification of a quarter-car test rig with McPherson strut suspension,” *IJVMM Trans. Vehicle System Dynamics.*, vol. 49, no. 1-2, pp. 153-179, Jan.-Feb. 2011.

[5] M. C. Levesley, S. A. Kember, D. C. Barton, P. C. Brooks, and O. M. Querin, “Dynamic simulation of vehicle suspension systems for durability analysis,” *Materials Science Forum. Modern practice in Stress and Vibration Analysis.*, vol. 440-441, pp. 103-110, Nov. 2003.

[6] G. L. Bradley, P. C. Chang, and G. B. Mckenna, “Rubber modeling using uniaxial test data,” *Journal of Applied Polymer.*, vol. 81, no. 4, pp. 837-848, July 2001.

[7] N. Yan, “Rubber hyperelastic constitutive model and its application in finite element analysis of radial tire,” *Journal of Advances in Vehicle Engineering.*, vol. 2, no. 4, pp. 191-200, Dec 2016.

[8] M. Shahzad, A. Kamran, M. Z. Siddiqui, and M. Farhan, “Mechanical characterization and FE modelling of a hyperelastic material,” *Materials Research.*, vol. 18, no. 5, pp. 918-924, Oct. 2015.

[9] M. Berdychowski, and K. J. Walus, “Verification of the simulation model with actual research vertical stiffness passenger car tire,” *Machine Dynamics Research.*, vol. 37, no. 2, pp. 5-14, Jan. 2013.

[10] S. P. Jung, T. W. Park, and W. S. Chung, “Dynamic analysis of rubber-like material using absolute nodal coordinate formulation based on the non-linear constitutive law,” *Nonlinear Dynamics.*, vol. 63, no. 1-2, pp. 149-157, Jan. 2011.

Myeong Jae Han received his B. S. degree in Mechanical Engineering from Ajou University in South Korea in 2016. He is currently a combined master’s and doctoral candidate at the Applied Mechanics Lab. at Ajou University. Mr. Han’s research interests are in the areas of flexible multi-body dynamics and computer aided engineering.

Chul Hyung Lee received his B. S. degree in Mechanical Engineering from Ajou University in South Korea in 2016. He is currently a combined master’s and doctoral candidate at the Applied Mechanics Lab. at Ajou University. Mr. Lee’s research interests are in the areas of acoustics and computer aided engineering.

Tae Won Park received his B.S. degree in Mechanical Engineering from Seoul National University. He then went on to receive his M.S. and Ph.D. degrees from the University of Iowa. Dr. Park is currently a Professor at the Department of Mechanical Engineering at Ajou University in Suwon, South Korea.