Application of Predictive Engineering Tool (ABAQUS) to Determine Optimize Rubber Door Harness Grommet Design

Praveen Mishra, Dayananda Gowda

Mercedes Benz R & D India, Bangalore, Karnataka, India

Abstract*: A cable grommet is a tube ring through which an electric cable passes. They are usually made of rubber or metal. The cable grommet is used to protect, improve friction or seal cables passing through it, from a possible mechanical or chemical attack and for aesthetic look. It becomes very important to determine the optimum length of the grommet used for particular application. Bigger length will have results into folding of grommet and lesser length will yield into stretching of rubber grommet. Finite element simulation technique presented in this paper can be used to determine the optimum geometry of the grommet. Component used for this study is an automotive door harness rubber grommet.*

Using Abaqus/CAE hyper elastic material models are evaluated by creating response curve using selected energy potentials. Based on the stability for that particular application hyper elastic model is finalized. Finite element simulation using Abaqus/Explicit helps the designer to optimize the length or material of grommet for that particular application. This gives engineering direction to designer to do necessary changes in the beginning of the design phase freeze which leads into save of time and money.

Keywords: *Grommet, Hyper elasticity, Optimization, Abaqus CAE, Abaqus /Explicit*.

1. Introduction

In Automotive cars, it requires a lot of wiring harness system to distribute power from one system to other. Any wiring harness should have sufficient strength to withstand any abrupt situations without affecting the performance of the total system. An opening and closing member is attached to a vehicle by a hinge enabling easy opening and closing of different moving member. Such members include side door, rear door, trunk lid etc. A guide member called Rubber grommet made of elastomer has an accurate portion which can be extend and contract while twisting with

A cable grommet is a tube ring through which an electric cable passes. They are usually made of rubber or metal. The cable grommet is used to protect, improve friction or seal cables passing through it, from a possible mechanical or chemical attack and for aesthetic look. It becomes very important to determine the optimum length of the grommet used for particular application. Bigger grommet length will have results into folding of grommet and lesser length will yield into stretching of rubber grommet

The objective of this paper is to suggest a methodology to model the behavior of rubber grommet under specified dynamic loading conditions. For this purpose rubber grommet has been modeled with hyperelastic material properties and simulation is conducted with a non-linear commercially available FE code Abaqus. First a stable hyperelastic material model is selected using Abaqus /CAE material evaluation option and later an explicit simulation is performed using Abaqus/explicit.

1.1 Defining Hyper Elastic Behavior using Test data

Rubber grommet is modelled with three dimensional solid elements. Fig 1 shows the rubber grommet in folded conditions. Main challenge for this component is to have the stable material law which can withstand for the given boundary and loading conditions. To have stable model a stress –strain data is obtained for

- Uniaxial tension data
- Uniaxial compression data
- Biaxial data
- Plane shear

1.2 Defining hyperelastic behavior using test data

The convenient way to defining a hyperelastic material is to give experimental data to Abaqus. Abaqus then calculate the constants based on the least squares method. Abaqus can fit the data for the following experimental tests.

- Uniaxial tension and compression
- Equibiaxial tension and compression
- Planar tension and compression (pure shear)
- Volumetric tension and compression

Unlike plasticity data, test data for hyperelastic material must be given as nominal stress and nominal strain values.For time-independent nonlinear elasticity, the fitting procedure may be carried out for polynomial representations of incompressible materials. The significance of (non-equivalent) multiple tests for material modeling cannot be overemphasized. In general, a combination of uniaxial tension/compression and simple shear is required in the very least. Data from biaxial tension or planar shear may also be needed depending on the deformation modes of the structure. Volumetric data must be included for materials undergoing large compressible deformations.

The quality of the results from a simulation using hyperelastic materials strongly depends on the material test data provided to Abaqus to generate the material coefficient. It is always recommended to have more than one deformation state to form accurate and stable material model. For our grommet simulation uniaxial tension and compression data is enough to capture the physics of the grommet behavior. It is common for the material model determined from the test data to be unstable at certain strain magnitude. Abaqus performs a stability check to determine the strain magnitude where unstable behavior will occur. So it is possible for the material to be unstable at the strain level indicated in the deformation is more complex and it is also possible for the material to become unstable at lower strain levels if the deformation is more complex.

Following Figure 1 & 2 shows the compression and tension data for the elastomer used for rubber grommet simulation.

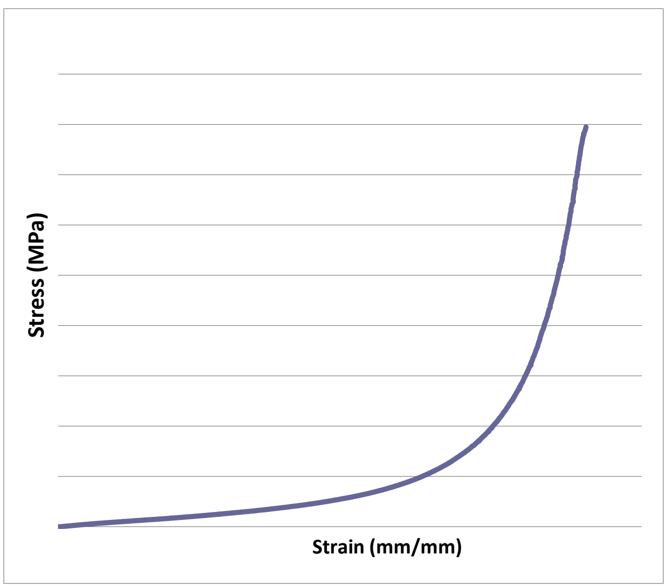

Figure 1. Uniaxial compression material data at 23^{0}C.

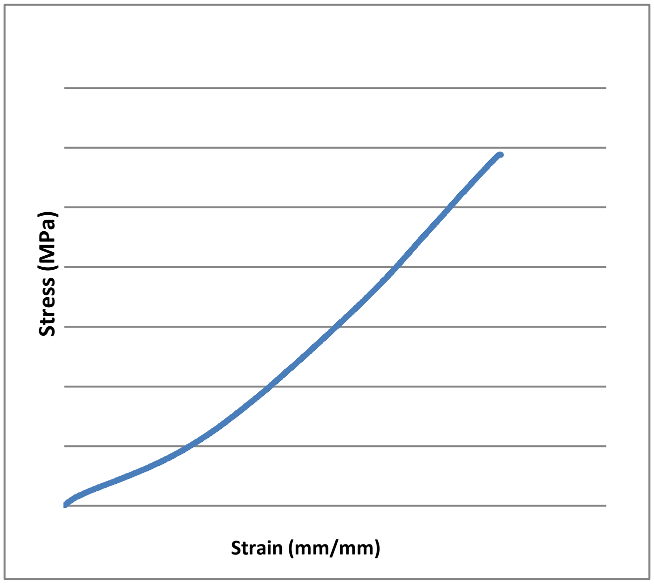

Figure 2. Uniaxial compression material data at 23^{0}C.

Both tension and compression data are allowed, with compressive stresses and strains entered as negative values. During dynamic simulation some part of the grommet will be under compression and some part will be under tension hence the both uniaxial tension as well as compression data is used to generate the hyperelastic material coefficients.

Using Abaqus /CAE material evaluate option various hyper elastic models are evaluated. Fig 3 shows the correlation of the various models with the test data. For this simulation second-order, polynomial strain energy function is used to model the rubber material. In input file these choices are invoked by using the N=2 and POLYNOMIAL parameters on the *HYPERELASTIC keyword option. TEST DATA INPUT parameter is used to indicate that Abaqus should find the material constants from the test data given in the input file. The test data are given on options that immediately follow the *HYPERELASTIC option. The data is entered as nominal stress and the corresponding nominal strain, with negative values indicating compression. Second order polynomial strain energy function shows a better fit with the test data and material is also stable at higher strain values. Hence for grommet simulation polynomial N=2 material model is used.

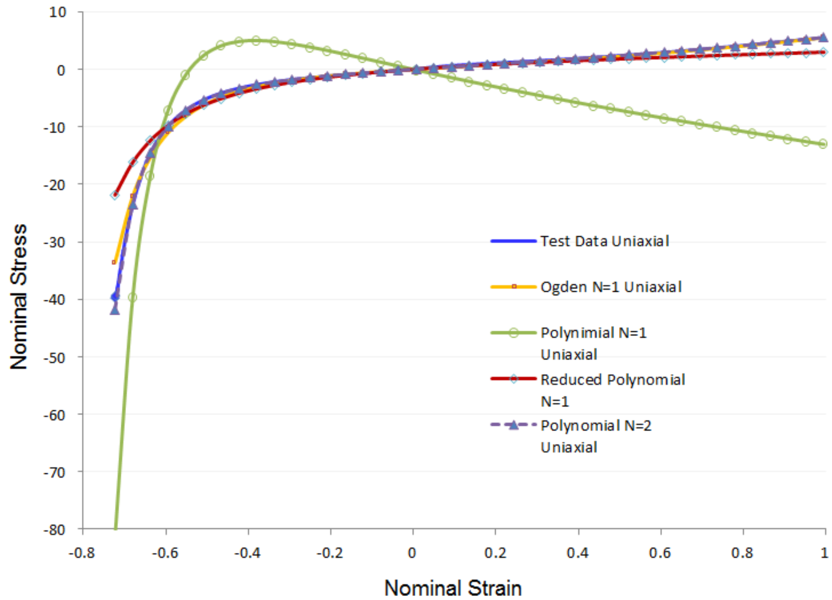

Figure 3. Least square fit for the various Hyperelastic models

2. FE Simulation

Rubber grommet has been modeled with very fine solid second order tetrahedral element (Figure 4). One end of the grommet is connected to the plastic connector and then attached to the door and other end of the rubber grommet is connected to the door. Node of the rubber are rigidly ties to the plastic connector .To simplify the problem and reduce the computational time door is model as a rigid.

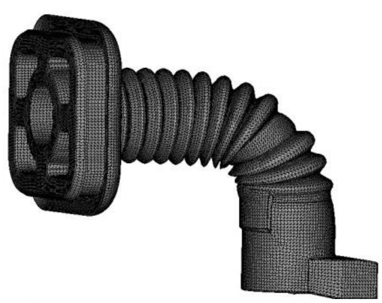

Figure 4. Grommet finite element model

Hinge location of the door is fixed in all degrees of freedom. A local coordinate system is defined (whose one axis is made parallel to the hinge axis) and the door is rotated 70 degrees along the hinge axis. Explicit simulation is performed and automatic general contact definition is used to define the contact.

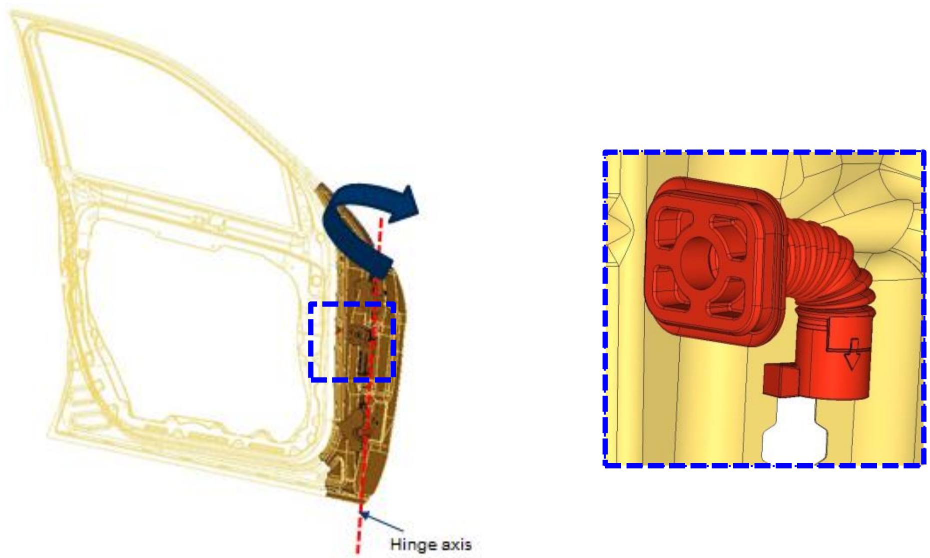

Figure 5. Door cut model with hinge rotation axis

3. Simulation Results

Figure 6 shows the stress and strain in the grommet which gives us an idea of the compression and tension occurring in the grommet during closing and opening of the door. For this simulation 70 % nominal strain is observed. Various other profiles are created based on the simulation results. By performing the simulation with different designs with varying convolutes of the grommet it becomes easy to find out the optimum length and design of the grommet.

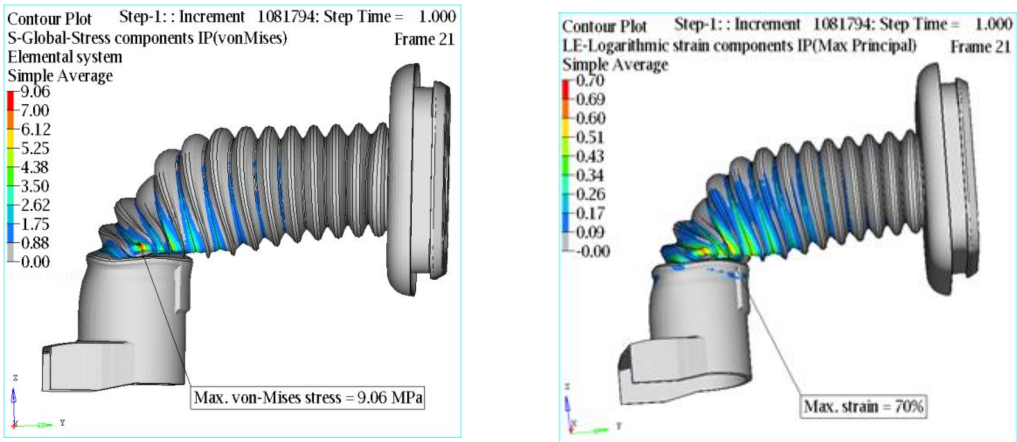

Figure 6. Stress & Strain contour in grommet finite element model

4. Conclusion

Rubber grommet profile is not uniform. With the accurate hyperelastic materials model it become easy to have the precise results. With the CAE simulation it is easy for designer to design an optimized shape and size of the rubber grommet. In the development phase of the automobile vehicle this simulation will help to avoid any last moment surprise at the end of the vehicle development cycle. This gives engineering direction to designer to do necessary changes in the beginning of the design phase freeze which leads into save of time and money.

5. Acknowledgement

I would like to thank Mercedes Benz research & development India management for giving me an opportunity to work on this topic and extending their support whenever required.

6. References

1. Byeong-Sam Kim, Kyoungwoo Park, Young-Woo Kim, “Endurance Analysis of Automotive Vehicle’s Door W/H System Using Finite Element Analysis,2009”
2. Abaqus Users Manual, Version 6.13-1, Dassault System Simulia Corp., Providence, RI.
3. Getting Started with Abaqus–Keyword Edition Version 6.13-1, Dassault System Simulia Corp., Providence, RI.