A finite element study on Plastic deformation of Suspension Bushing

A.K.Serbest [a], S.Kayaci [b]

Las-Par Rubber Components & Development Ltd. Co. Akcalar Industrial Zone Kale St. No:10 16225 Nilufer/Bursa/Turkey

Abstract

Force-Displacement data of elastic and plastic regions were collected with tensile test of flat metal alloy specimen. After simplification of collected data, engineering and true stress-strain values of specified materials were obtained. The data, obtained in the tensile of flat specimens, were applied to the finite element model of specimen to perform a finite element analysis (FEA). Results, which were collected in tensile test and calculated by FEA, were compared to evaluate the accuracy of obtained stress-strain values and FEA application. After data and application verification, obtained data applied to a FEA model of suspension bushing, whose serial production is in progress. The actual results obtained from the serial parts and the results obtained in FEA were compared to introduce the consistency of computational results. At the end of this study, a material library for a steel alloy has been created for FEA calculations, by using the collected and simplified stress-strain data for the steel alloy mentioned.

1. Introduction

During product design phases, production materials are to be determined according to its mechanical properties and working conditions. Determination of material properties is the most important point for this kind of design studies. Tensile test is an indispensable examination method of determining the mechanical properties of a material. It supplies mechanical properties of materials; which are needed for static and quasi static loading of machine and systems. Indeed it is applied to determine the maximum amount of force per unit cross-sectional area, elastic and plastic behavior of materials under an increasing load and strain change behavior. [1]. Tensile tests are comparable because they are carried out according to national and international standards. Today, tensile tests can be applied more precisely and accurately because of the developments in test methods and testing devices than past.

In today's engineering problems, conceptual designs can be solved with high accuracy by the benefit of FEM and other similar numerical methods without physical model production. FEM is a numerical method, which is used to solve complex and difficult engineering problems with a negligible error. The reason of widely usage of FEA, its easy algorithm for computer applications and its application areas like static analysis, fluid mechanics, heat transfer, electromagnetic analysis and acoustics. With the spread of computer use and techniques, computer softwares written are diversified for the FEA. The variations like material models and element types of FEA, has been developed for getting closer results to real results. Success of FEA results depends on correct and usable material data.

Today, metal alloys have a widely usage in every area. In this study, mechanical data collected from tensile test specimen which made of metal alloy. These collected data were edited

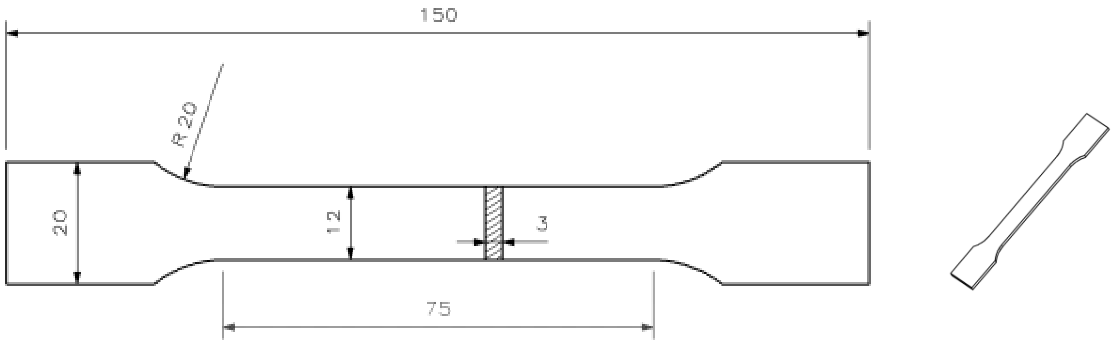
[a] Corresponding Author "kamil.serbest@laspargroup.com"

and converted to a form to use in FEA and therefore to create a material library. By the help of obtained data, elastic and plastic behaviors of metal alloys were defined. Defined material model is used in FEA of a real suspension bushing. FEA calculations and serial production results were compared to determine the accuracy of study.

2. Experimental Arrangements and Calculations

Figure 1 shows the test specimen, which was prepared according to DIN EN ISO 6892-1 standart. The specimens were prepared with laser cut according to the dimensions of the specified standard with an offset then CNC milling machine is used to remove the additional material exposed to heat in laser cut process also to give the final dimensions stated in the standard.

(a)



(b)

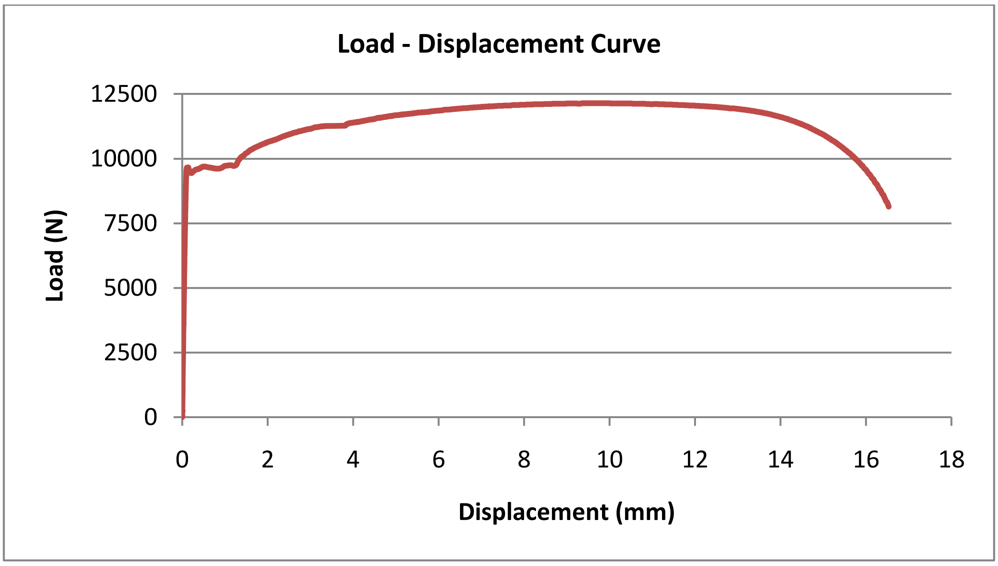
Figure 1 - Tensile Test Specimen (a), (b)

UTEST universal test machine is used for all tensile tests. Test machine has a 250 kN load capacity. Physical extensometer is used for measurement of elongation.

As a result of tensile tests, load-displacement data of specimens were collected.

2.1. Tensile Tests and Collection of Load-Displacement data.

The specimens prepared according to DIN EN ISO 6892-1 were tested until failure in the test machine and corresponding load displacement data were collected (Graph 1). Data is used to determine the true stress-strain data of material to define the material model in FEA [2]. Engineering stress-strain values are calculated with an assumption that the cross-section remains constant during tensile test. True stress-strain values are calculated, based on the fact that the volume remains constant, using the instantaneous cross-sectional area Formula 1, 2 and 3 are used to obtain true stress-strain data.



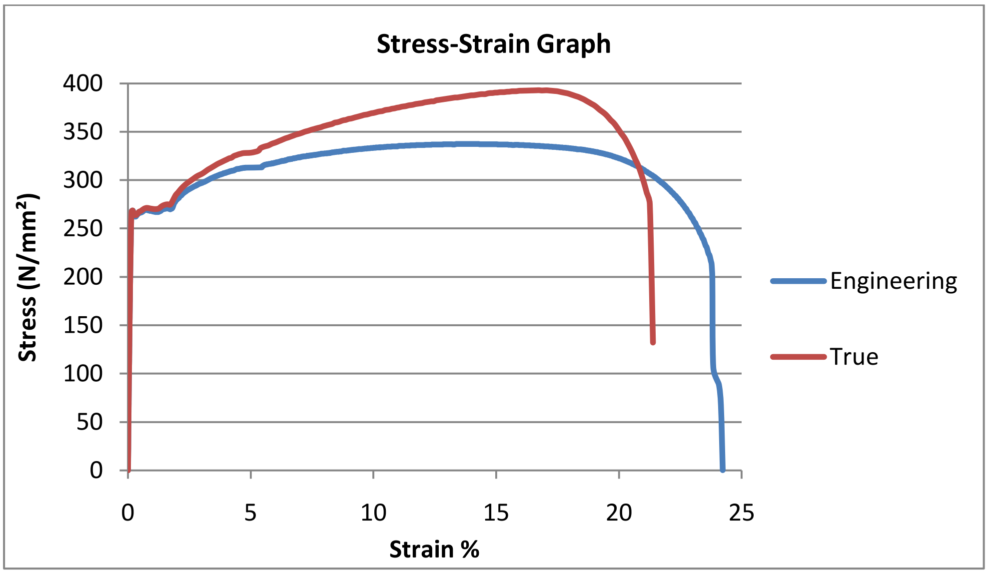
Graph 1 - Load-Displacement Curve obtained from Test Specimen

$$Ai = \frac{A_0 \, x \, L_0}{L_0 + \Delta l} \qquad (1)$$

$$\sigma_{t=} \frac{F_i}{A_i} \qquad (2)$$

$$\varepsilon_t = \ln \frac{L_0 + \Delta l}{L_0} \qquad (3)$$

Where, A_i is instantaneous cross-sectional area, A_0 is initial cross-sectional area, L_0 is initial parallel length, F_i is instantaneous force, σ_t is real stress and ε_t is real strain.



Graph 2 - Obtaioned Stress-Strain Graphs

With the help of obtained values, tensile strength, ultimate strength was determined and elasticity modulus calculated with formula 4. (Hookes Law)

$$E = \frac{\sigma_y}{\varepsilon_y} \quad (4)$$

Where E is elasticity modulus, σy is stress at yield and εy is strain at yield. Modulus of elasticity was calculated as an average of three different specimens to be 206 GPa.

2.2. Usage of obtained Tensile Test values in Finite Element Model

MSC.Patran 2008 interface was used to prepare the finite element model of test specimen. 2712 Hex8 element are used in FEM. The aim of fine mesh usage in the middle of the specimen is to get more precise results at large deformation expected areas. (Figure 2)

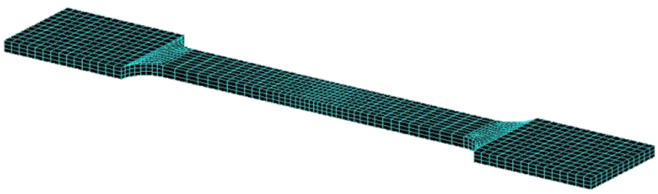
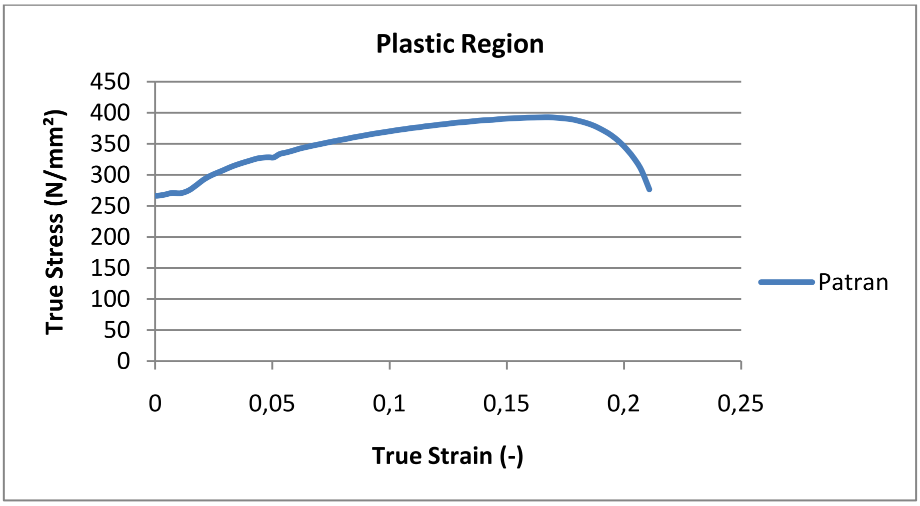


Figure 2 - FEM of Test Specimen

True stress-strain values, which obtained from load-displacement data of tensile tests, were used for material data in FEA. MSC.Patran 2008 is used to define elastic and plastic behaviors of material, for MSC.Marc 2008 r1 solver.

Elasticity modulus: 206GPa and Poisson Ratio: 0,30 were used for elastic behavior of material.

Plastic region of true stress-strain curve, which is obtained from test specimen, was used to define the plastic behavior of material. (Graph 3)



Graph 3 - Plastic Section Curve

Plastic section data inputs was done in MSC.Patran 2008 interface with "Fields-Material Property" tab as "Strain=e". The important point of material data input is that strain (e) value must begin with value "0". [2] Then, plastic range behavior assigned to material at tab "Materials-Input Properties-Plastic". "Piecewise Linear" option used for "Strain Rate Method". Corresponding attributes of this option is explained deeply in reference [2]. Basically, it is associated with work hardening. After completion of material property inputs, it is expected that, the behavior of material must be according to Hookes Law if part subjected by a force in only elastic region and must follow Plastic curve if part is subjected by a force in plastic region.

2.3. Comparison of Tensile Test Results and FEA Results

According to comparison of tensile test results and FEA results, it is easy to see that, data collection and application processes are successfully performed.

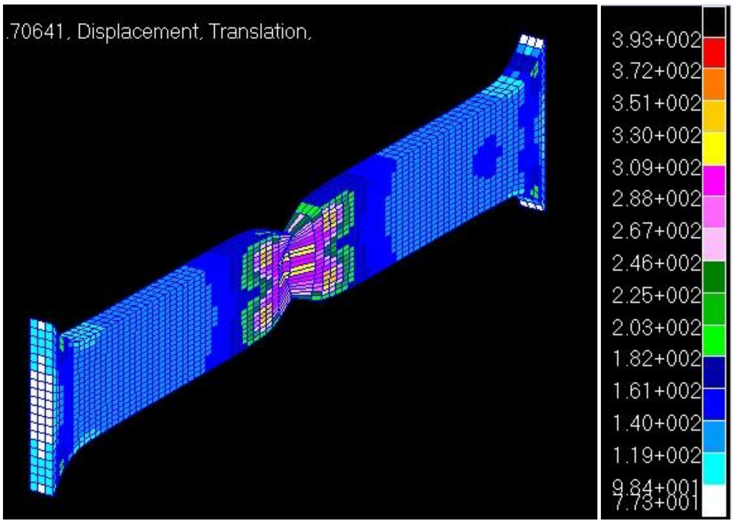
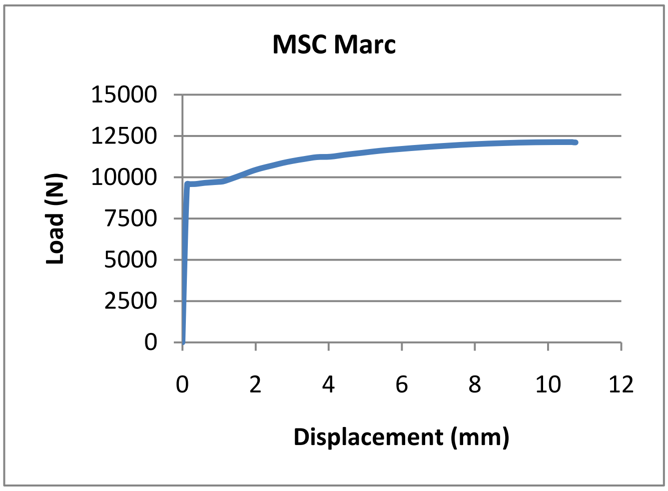
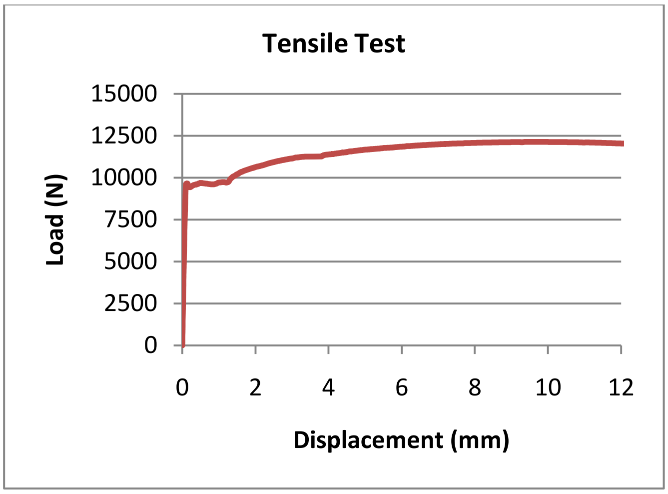


Figure 3 - Tensile Test Results with FEA

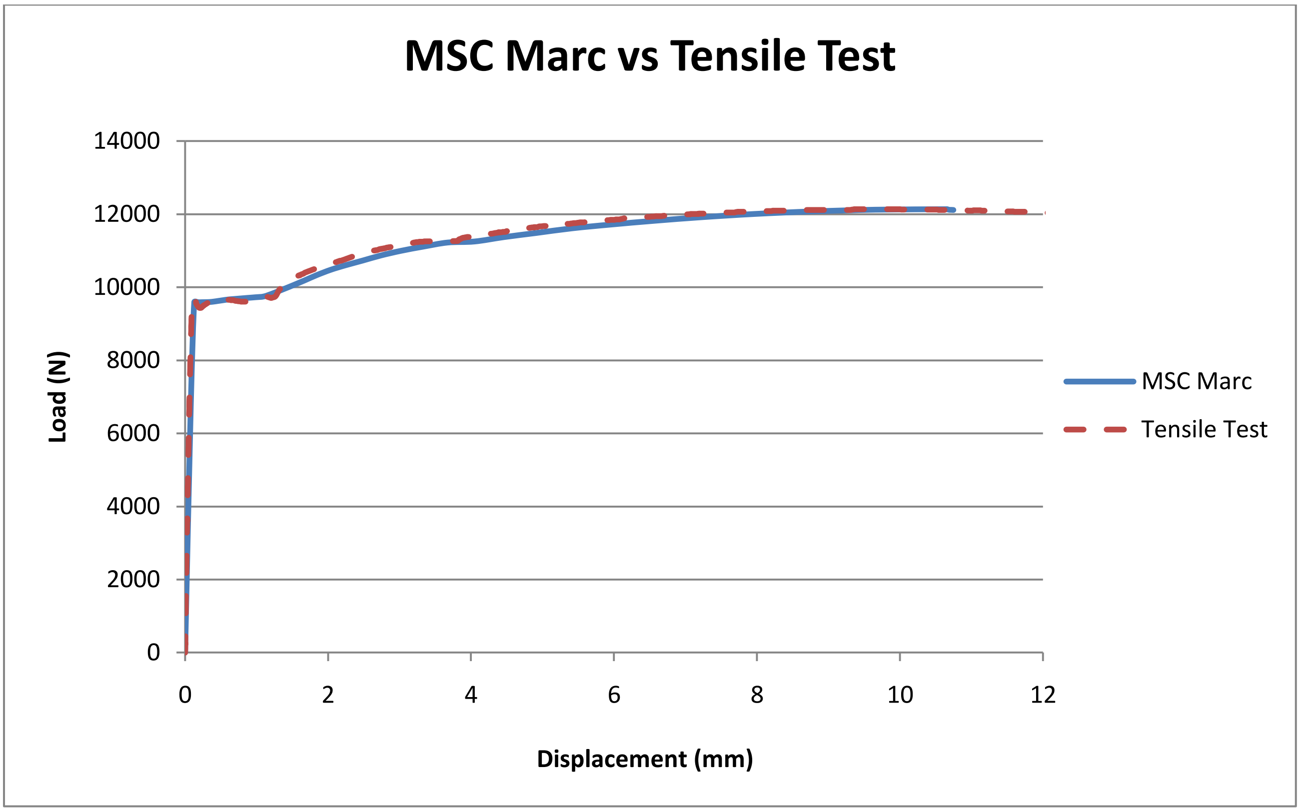


(a)



(b)

Graph 4 - MSC Marc (a) and Tensile Test (b) Load-Displacement Graphs



Graph 5 - Comparison Graphs for Tensile Test and FEA Results

	Tensile Test		FEA		% Error
	Displacement (mm)	Load (N)	Displacement (mm)	Load (N)	Load
1	0,1279984	9630	0,12799048	9591,9063	-%0,40
2	2,038002	10660	2,0385141	10480,898	-%1,70
3	4,057999	11400	4,0490518	11254,372	-%1,29
4	5,968002	11850	5,9656715	11720,451	-%1,11
5	7,968002	12080	7,9670215	12008,695	-%0,60
6	10,662	12120	10,6596	12136,8260	+%0,14

Table 1 - Comparison table for Tensile Test and FEA Results

2.4. FEA application on a real Suspension Bushing with obtained material values

A FEA is performed on a real suspension bushing to verify created material values. Suspension bushing has 4 sub-components which are listed below. (Table 2)

	Part Name	Material	Note
1	Inner Tube	DIN 2393 ST37.2 BK	1 Part, 60 mm length pre-shaped
2	Inter leaf	EN 10130 / BS 1449 CR1	2 Part, 1,2mm thickness pre-shaped
3	Outer Tube	DIN 2393 St37.2 BK	1 Part, 41mm length
4	Rubber	70ShA Natural Rubber	

Table 2 - Parts of Suspension Bushing

After production of suspension bushing in vulcanization presses, some plastic deformation processes are applied on the part to get the finished product.

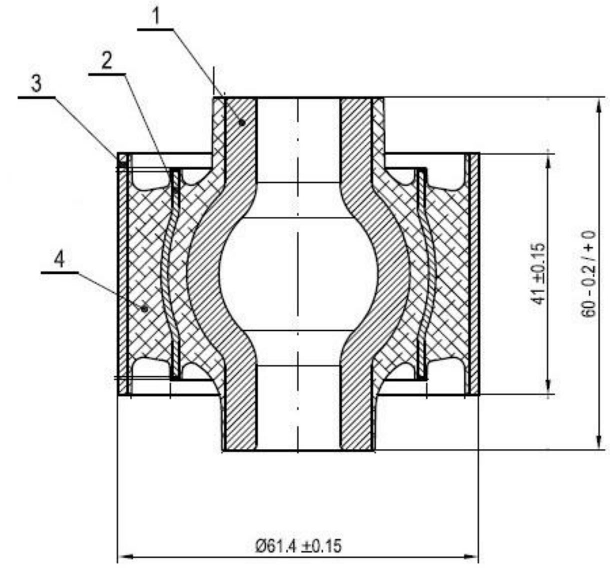
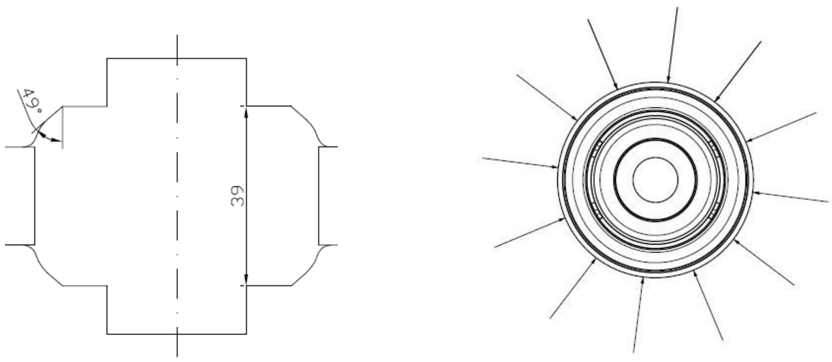


Figure 4 - Suspension Bushing after Vulcanization

Aim of plastic deformation processes after vulcanization is to increase the durability of rubber section of the complete part. Firstly end-forming process is applied to both ends of the outer tube. (Figure 6a) With this process, rubber material is partially trapped in outer tube. Then outer tube is swaged inwards in radial direction. (Figure 6b) This process applies a pre-compression on part. Metal alloys make a spring-back movement while plastic deformation if the applied force removed. So metals need are to be subjected to smaller or larger geometrical tools than required to get proper final dimensions.



(a) (b)

Figure 5 - End-Forming Tool (a), Swaging (b)

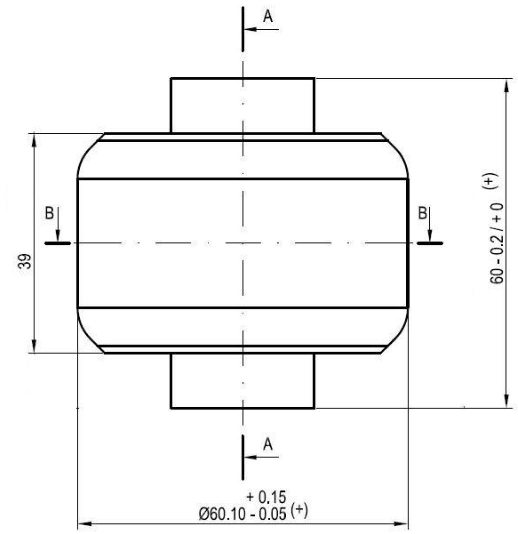


Figure 6 - Final Suspension Bushing

At final part, end-forming angel is 45°, outer tube length is 39mm. After vulcanization outer dimension of outer tube is 61,1mm and after swaging operation this dimension is 60,1mm. In end-forming process ends of outer tube subjected with angle 49° and in swaging process outer tube is swaged to 59,9 mm diameter. After both processes, part makes a spring-back move and gets its final dimensions. (Figure 6) The material model discussed in the previous section is used to define the material behavior of outer tube, the material model of rubber is taken from a previous study made by S.Kayaci *et al*. [3] In this study, mechanical properties of rubber materials were defined by a two-coefficients Mooney-Rivlin formula.

All plastic deformation processes, which are using in serial production, are applied in FEA. FEM model is shown in Figure 7. 1/8 of model has prepared for analysis by using symmetry boundary conditions. 47480 Hex 8 elements are used for FEM. During analysis, first rigid body 1 performs end-forming process and turns back to its initial place to release the load. Then rigid body 2 performs swaging process and it also turns back. Analysis is completed by these two phases.

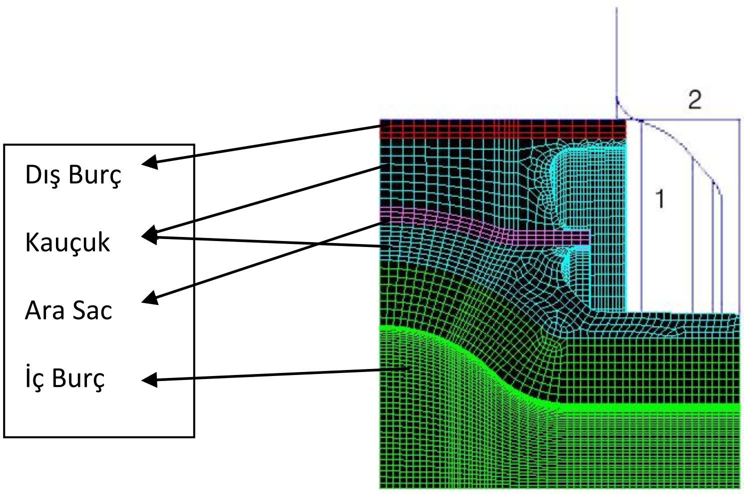


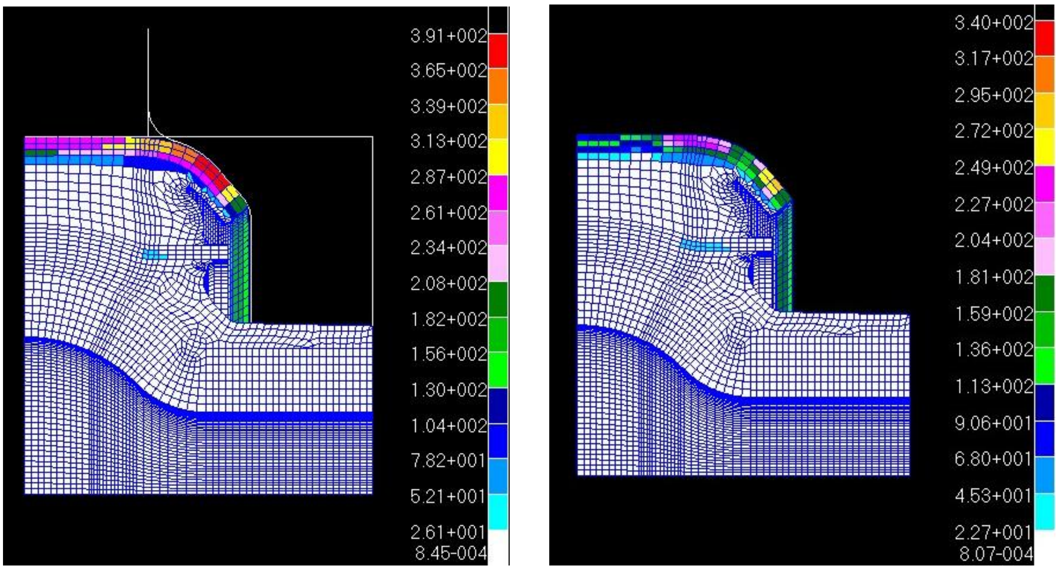
Figure 7 - FEM of Suspension Bushing

3. Results and Discussions

Results of this study shown in Table 3, Fig. 8, 9 and 10. Two dimensions are used to compare results. Displacement results are multiplied by 2 due to the symmetrical boundaries to find the diameter and length results. Table 3 shows a comparison between real results and FEA results. Results have an acceptable deviation. According to these results, obtained material data can be used for a material library creation.

Dimension	After Vulcanization (mm)	Required (mm)	FEA Result (mm)	Deviation %	Image
Outer Diameter	61,4	$60{,}1^{+0{,}15}_{-0{,}05}$	60,182	+0,137	Figure 11 – (a)
Outer Tube Length	41	39	39,776	+1,980	Figure 11 – (b)

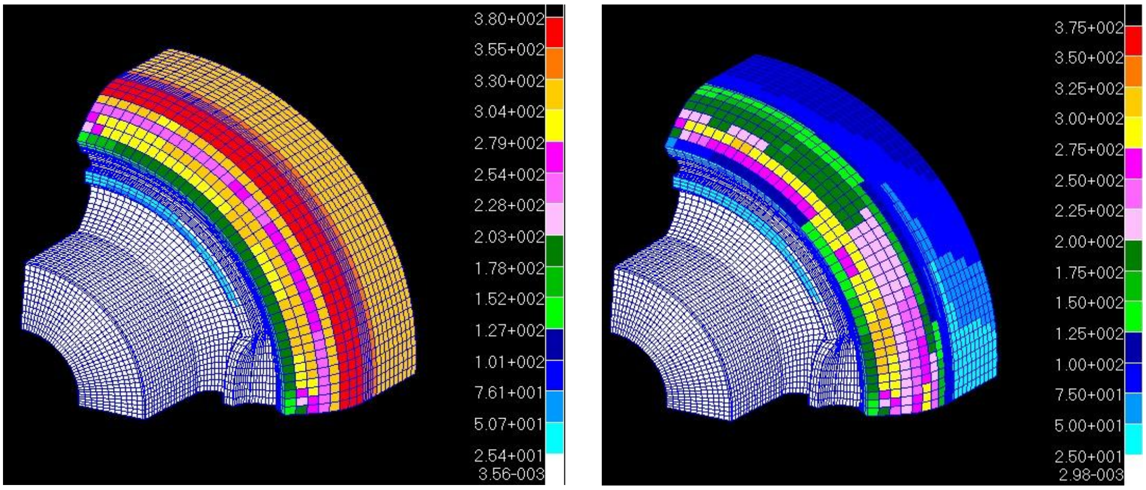
Table 3 - Dimensional Comparison Table



(a) (b)

Figure 8 - End-Forming Von misses Stress, (a) Full Load, (b) Load Removed

Figure 8 shows von misses stress distribution on the part. Maximum stress value at full load case (Figure 8a) is around 391MPa. This value is closed the maximum point of entered Plastic stress curve. If load is released, the maximum residual stress can be read around 340 MPa.



(a) (b)

Figure 9 - After Swaging Von misses Stress, (a) Full Load, (b) Load Removed

Von misses stress distribution is shown in Figure 9 for swaging operation. Maximum von misses stress value is around 380 MPa under full load. Residual stress is around 375 MPa when load is removed. Both residual stress values are placed under ultimate point FEA result geometries are nearly similar to real part. (Figure 10)

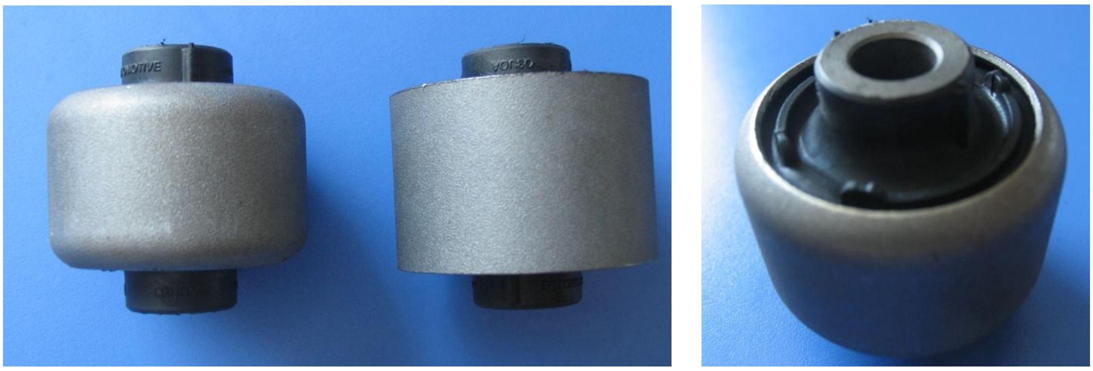
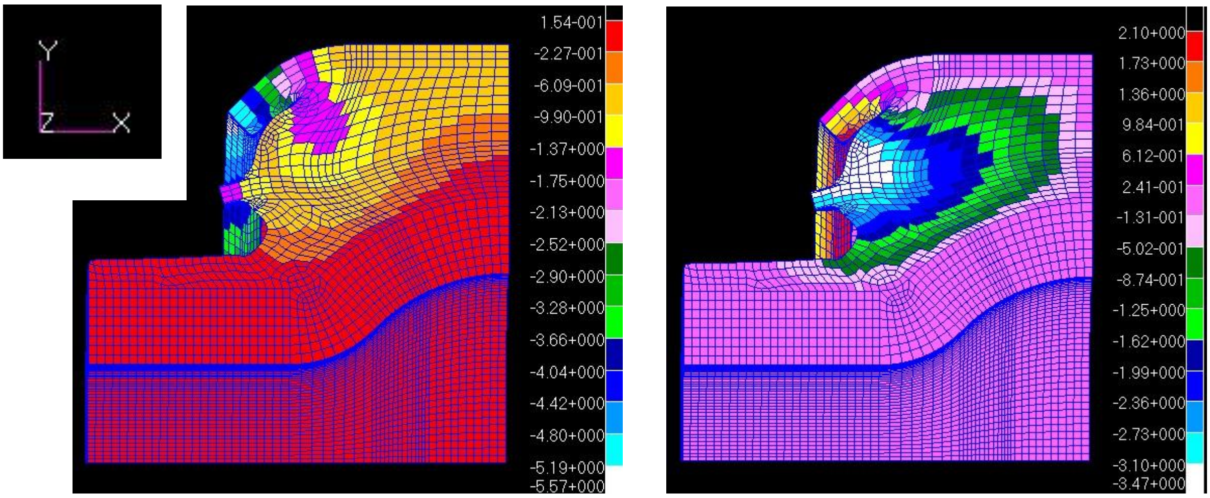


Figure 10 - Final Part and After Vulcanization Part



(a) (b)

Figure 11 - Final Part Displacement Results, (a) Y Direction, (b) X Direction

4. Conclusion

For manufacturers who manufacture their own raw materials, want to determine the mechanical properties of own raw materials, is inevitable. Raw material suppliers generally serve only generalized material properties with a wide range. In design and product development process, it is hard to get proper design requirements with these generalized material properties. The material library which was created in this study can be used for FEA to get more precise and proper results. Reference [3] for rubber materials and this study for metal alloys give us road maps for creating material libraries. The methodology of these studies can be used for other possible engineering materials' characterization.

Figure – Table – Graph Index

References

1. Demirci, A.Halim: Malzeme Bilgisi ve Malzeme Muayenesi. Alfa Basım Dağıtım, 2004
2. MSC Software Corporation : Marc 2008 r1 Volume A : Theory and User Information
3. Kayaci, S., Eşiyok Y.E., Cevher, M., Las-Par Rubber Components & Development Ltd. Co.: Experimental investigation on the dynamic modulus change of Natural Rubber under principal loading modes and different dynamic conditions. Bursa-2011