January 2009 Volume 5 Issue 1

Materials Science

Trade Science Inc.

An Indian Journal

Full Paper

MSAIJ, 5(1), 2009 [68-72]

Numerical and experimental analysis of the damped energy of polymer material by viscoelastic behaivior

Mojtaba Soleimanipour, Hasan Khademizadeh*

Mechanical Engineering Department, Islamic Azad University of Khomeinishahr, (IRAN)

Tel : +989133008827; Fax: +983113800138

E-mail : soleim09@yahoo.com

Received: 14th September, 2008 ; Accepted: 16th September, 2008

ABSTRACT

The intention of this paper is to give a comparative review of material polymer such asRubber with special focus on impact analysis. For quasi-static problems,elastomers can be considered as incompressible ruber-like materials.In this research first in order to simulate the mentioned material in impact condition,the coefficients related to viscoelastic equations and rubber constitutive equatoins, the properties of viscoelastic material as the terms of Prony series will be determined,then the hyperelastic model was obtained by using the results of an expriment .then by simulating rubber in the finite elemet code,(ABAQUS/Explicit) the expected analysis carried out. The result of simulating was compared with similar amount of practical,available result and by this the numerical method was conffirmed by experimental and theoritical results .at the end by simulating of impact expected out puts such as damped energy will be obtained.

KEYWORDS

Rubber;
Impact;
Prony series;
Finite element code;
ABAQUS/Explicit;
Damped enegy.

INTRODUCTION

No doubt, the best method to solve any physical problem that is obtained based on deferential equation,is the analitical solution of these equations. But in many situation because of the complexities related to geometry or material model, these solution can not be obtained for these equations. In these situations using numerical methods is unavoidable.these method are faster, more accurated and cheaper than previous methods like analytical and experimental in their workability and the range of their uses is much vaster. One of these methods,has been the finite element methods in away that in resent years powerful softwares to model and also to analyze different problems based on this method have been submitted to users. The problems of contacts and also analyzing the impact because of too many complexitres needs to use this mothod.in this method first by using conducted tests by Mr.Yorgen Bergstrom on rubber NR-55pphCB[1] and viscoelastic model, viscoelastic properties NR-55pphCB and hyperelastic model will be determined. In the next section by using the above coefficients the considered rubber will go under impact loading and expected outputs such will be btained.

2.RESULTS AND DISCUSSION

1. Viscoelastic properties.2

By testing a viscoelastic material like NR and by using viscoelastic models the viscoelastic properties of this material can be determined.First by conducted practical tests including uniaxial test, biaxial test,and planar,

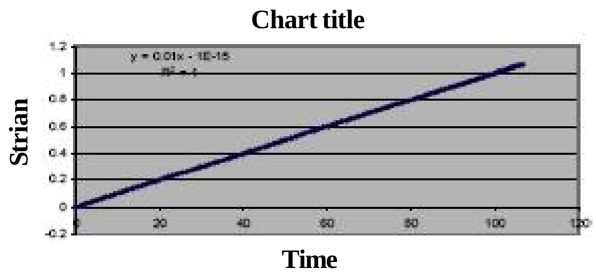


Figure 1: Strain-time curve for NR-55pphCB

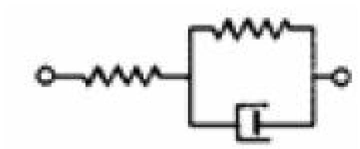


Figure 2: Parameter solid model

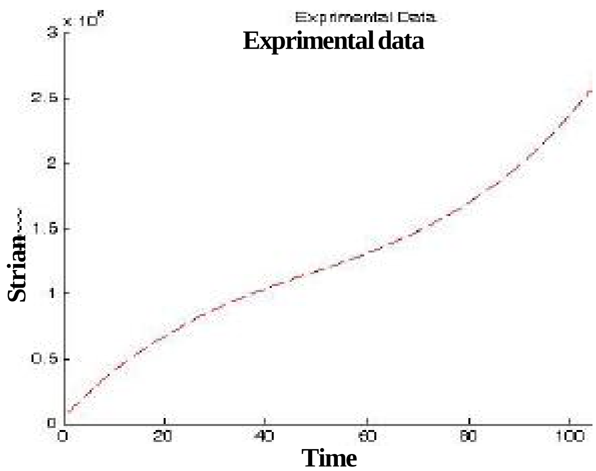


Figure 3: Stress-time curve of experimental data for NR-55pphCB

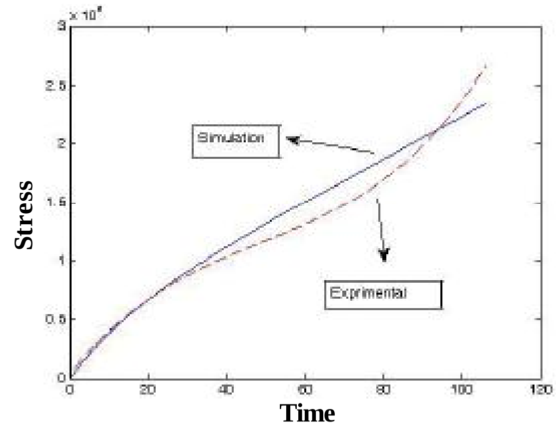


Figure 4: Comparision of experimental and simulation if p1-p1q0>0

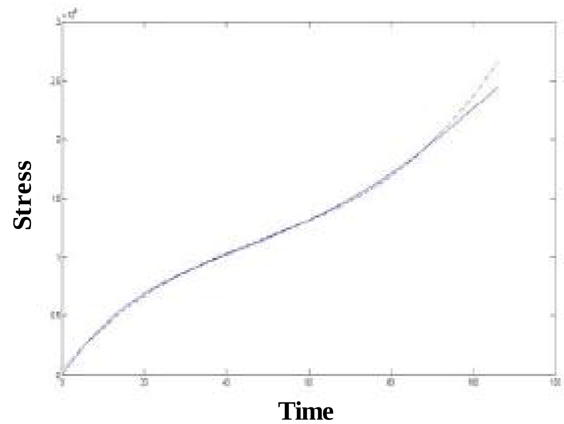


Figure 5: Comparision of experimental and simulation p1-p1q0>0

the viscoelastic coefficients of this material will be obtained .The standard of conducted test is ASTM standard[2]. The standard dimentions of uniaxial test is: thickness :13mm; diameter :28mm

In the strain-time diagram the rate of strain that is the same as the slope of the line can be found.

$$\dot{\varepsilon} = .01$$

Now from among the viscoelastic models, the three parameter solid that is one of the most standard viscoelastic model will be chosen[3].

Whose differential equation is as the following:

$$\sigma + p_1\dot{\sigma} = q_0\varepsilon + q_1\dot{\varepsilon} \qquad q_1 > q_0 p_1 \tag{1}$$

By trial and error method the coefficients of p1,q0,q1 are obtained. The terial and error method is preformed in away that in the written program of Matlab, the coefficients of p1,q0,q1 are entered and the obtained curve will be compared with curve obtained from practical tests(Figure 4) and this terial and error continious until the above mentioned curve get close to each other.

Now by using these numbers (q1,q0,p0), the necessary coefficients of ABAQUS software will be calculated.

2.2. Determining the viscoelastic coefficients NR-55pphCB

The first method in determining the prony series, ABAQUS software in order to define viscoelastic properties of material uses perony series[4].

The differential equation coefficients NR is as the following:

$$\begin{cases} q_0 = 1.2e8 \\ q_1 = 4.8e9 \\ p_1 = 16 \end{cases} \tag{2}$$

The Creep compliance is:

$$\begin{cases} \frac{p_1}{q_1} e^{-\lambda t} + \frac{1}{q_0}\left(1 - e^{\lambda t}\right) \\ \lambda = \frac{q_0}{q_1} \end{cases} \qquad (3)$$

And the Relaxation module is:

$$\frac{q_1}{p_1} e^{\frac{-t}{p_1}} + q_0(1 - e^{\frac{-t}{p_1}}) \qquad (4)$$

With convert the Relaxation module to prony series we can determine a term of prony series

$$\frac{q_1}{p_1}\left[1 - (1 - \frac{q_0 p_1}{q_1})\left(1 - e^{\frac{-t}{p_1}}\right)\right] \qquad (1 - \frac{q_0 p_1}{q_1}) < 1 \qquad (5)$$

The coefficients of prony series is as the following:

$$g_i = (1 - \frac{q_0 p_1}{q_1}) \Longrightarrow \text{g-iprony}=0.52$$

$$\text{k-iprony}=0 \Longrightarrow \text{compressibility factor} \qquad (6)$$

$$\tau = p_1 \Longrightarrow \text{tau-iprony}=16$$

2.2.1. The first method for verification of prony series coefficients

Using the above mentioned method to calculate the prony cofficients can also be used in the frequency domain[5]. In order to verify the above visco elastic coeffients,the material with the above mentioned charactritics in ABAQUS will go under strain rate of experimental test(Figure 6) and the result will be compared with the curve fitted on experimental in MATLAB(Figure 8).

2.2.2.The second method for verification of peony series coefficients

In this method unlike the pervious ones in wich prony series coefficients were calculated by direct solving of equations and then compared with experimental results,first directly in viscoelastic properties determining section of material the prony series coefficients enter the ABAQUS in a limited way and the necessary out put namely tense curve will becalculated from the software based on stress-time and along with compared experimental results and given coefficients to the software in order to make the best diagram correspondence will be used as the practice criteria.

In practice the obtained results in the second method because of better fitting of the curve of results

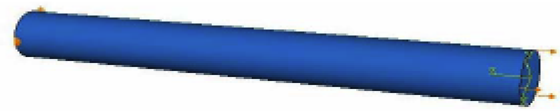


Figure 6 : Simulation of NR-55pphCB in ABAQUS

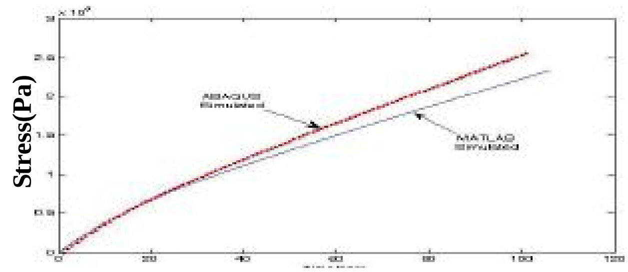


Figure 7: Stress-time curves of ABAQUS and MATLAB

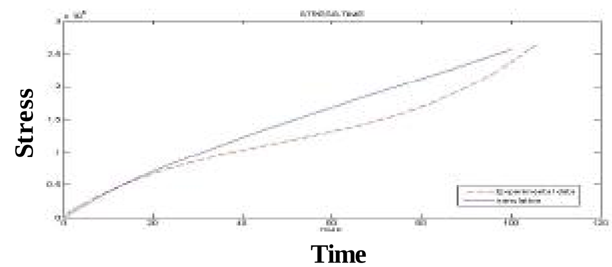


Figure 8: Stress-time curves of ABAQUS and MATLAB (first method)

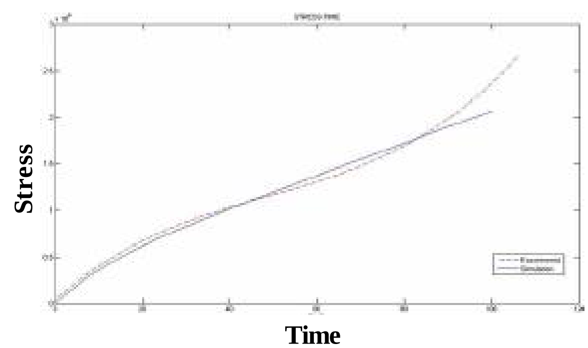


Figure 9 : Stress-time curves of ABAQUS and MATLAB (second method) g-iprony=0.61; k-iprony=0; tau-iprony=11

will be used as the criteria in the next stage.

3. Determining the hyperelastic properties NR-55pphCB

In finite elements software like Ansys and ABAQUS in order to model make the rubbers and like rubber materials, a choice has been made called hyperelastic in which different models have been presented including Yeoh,Neo-Hookean, Moony-Rivlin[6]. Noticing the kind of used rubber and also shape changes

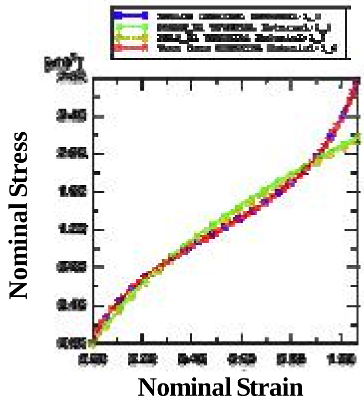


Figure 10: Comparition of three models strain energy with uniaxial test

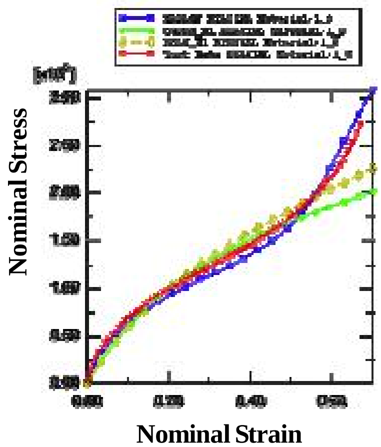


Figure 11: Comparition of three models strain energy with biaxial test

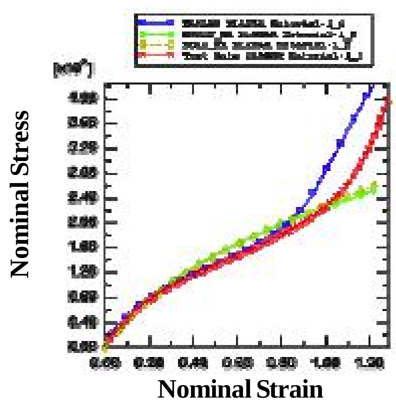


Figure 12: Comparition of three models strain energy with planer test

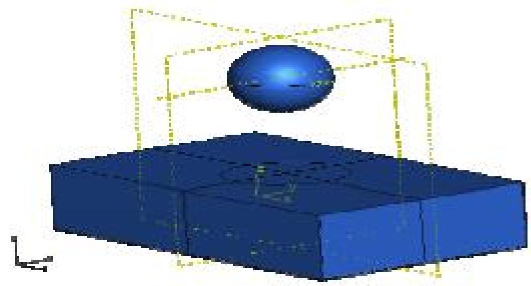
exerted on the material, these models can be validate as it is observed in the domain of strains under %50, all three models have good correspondence with the results of experimental tests but the Marlow model has better correspondence in comparison with other models. The coefficients calculated for each of the models

TABLE 1: Coefficients of equations hyperelastic in ABAQUS

Coefficients of equations hyperelastic (Mpa)				Material model
	D1= 2.46	C10=0.564	C01=0.045	Moony-Rivlin
I	MU_I	ALPHA_I	D_I	OGDEN
1	1.2473	0	0	
0	D1	C10	C01	NEO-HOOKEAN
	0	0.6144	0	
Stability limit information				Marlow
E=5 ν-0.499				Elasticity lin model

are presented in (Figure 9). Also elasticity model of rubber based on primary slope of elastic domain of tensile test was obtained.

4. Simulating the impact of a sphere on NR-5pphcb



(rubber length: 80 cm, rubber width:50 cm, rubber tickness: 20 cm, sphere mass 50 kg, sphere radius 10 cm)

Figure 13: Sample geometry for a 3-D simulation

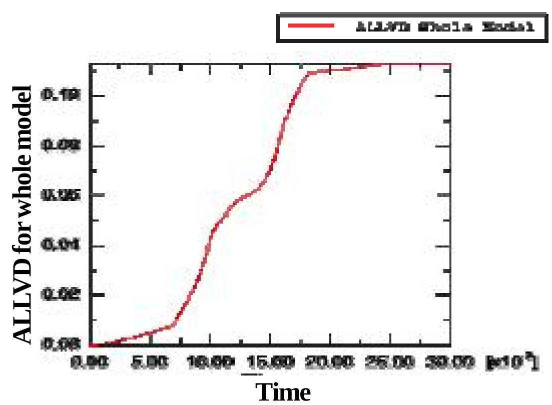


Figure 14: Damped energy-time curve for NR-5pphcb

After calculating viscoelastic coefficients and choosing the hyperelastic model of NR-5pphcb,the impact will be simulated by finite element ABAQUS/Explicit, Geometrical specifications is: (Figure 13)

As the diagram shows,damped energy in this model approximately equals to 0.122 J

As the above diagram shows the maximum of impact loading on rubber is 12923 N.

As the above diagram shows the maximum of displacement is 13 mm.

Full Paper

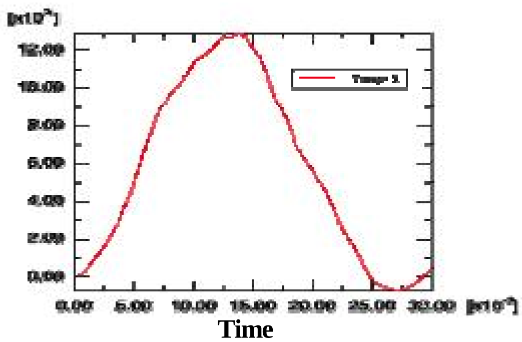


Figure 15: Force-time curve for NR-5pphcb

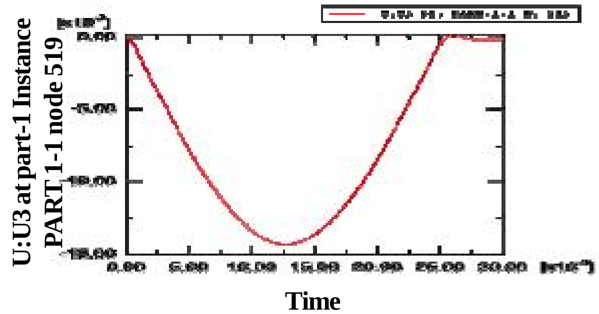


Figure 16: Displacement-time curve for NR-5pphcb

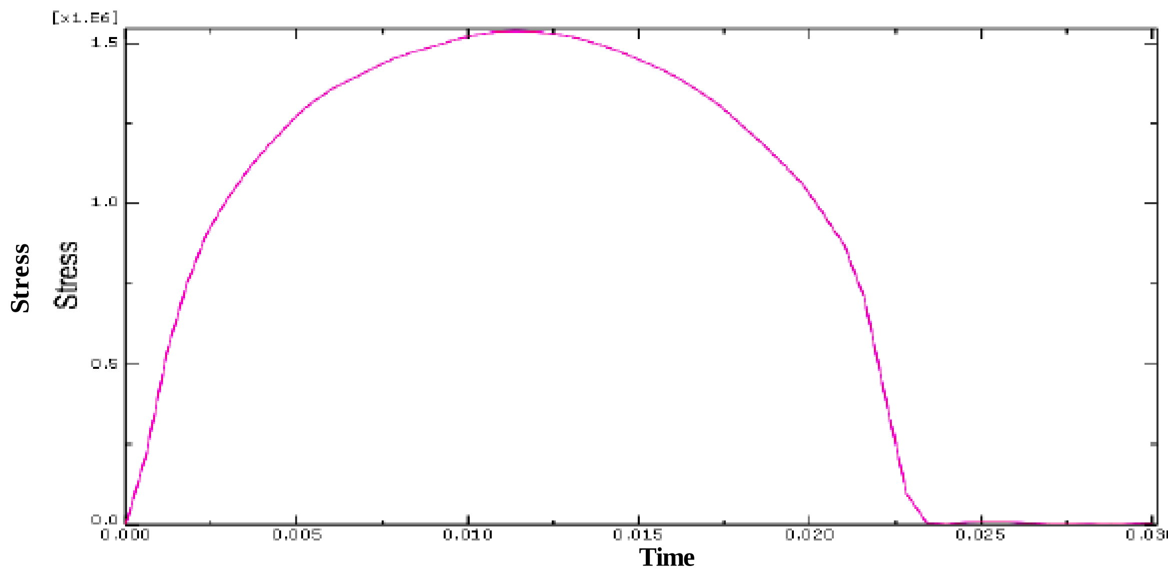


Figure 17: Mises tension for NR-5pphcb

As the above diagram shows the maximum of mises tension is 1.53 Mpa which in elastic area.

5. CONCLUSIONS

In this paper, first for simulation of the material in impact condition, the coefficient

of viscoelastic and constitutive equation are extracted by means of experimental test. After that by the simulation of the rubber in finite element code, comprehensive Analyses are performed. The results are compared with the available experimental tests which show that the numerical results are in a good agreement with theorical solution and experimental test. The above method can also be used to calculated prony coefficients in the frequency domain. In the next stage the hyperelastic coefficients rubber is determined, and finally by using the above coefficient and Simulating the impact in finite element code, extracted the out puts that we need by noticing the maximum amount of produced strain on rubber under impact loading That is a very low number compared with hyperelastic behaviour, different unliner models and also elastic model can be used to simulate the impact.

REFERENCES

[1] www.Polymer.com

[2] Annual book of ASTM Standards, American Society for testing and materials, Philadelphia, Published Annually, Vols.09.01 and 09.02,

[3] E.Riande; Polymer viscoelasticity. Stress and Strain in Practice, Marcel Dekker Press, 401 **(2000)**.

[4] P.A Du Bois, S.Kolling, M. Koesters, T. Frank; International Journal of Impact Engineering, **32**, 725-740 **(2006)**.

[5] E.Riande; 'Polymer Viscoelasticity', Stress and Strain in Practice, Marcel Dekker Press, 419, **(2000)**.

[6] ABAQUS user manual.