The Numerical Simulation of the Rubber Diaphragm Behavior

JAKUB JAVORIK, MICHAL STANEK
Department of Production Engineering, Faculty of Technology
Tomas Bata University in Zlin
nam. T.G. Masaryka 5555, 760 01 Zlin
CZECH REPUBLIC
javorik@ft.utb.cz   http://web.ft.utb.cz

Abstract: There was a need to analyze the behavior of the rubber diaphragm inside a pneumatic valve. The hyperelastic material constants [1] of the diaphragm material were determined by the testing of the material. Numerical model of the diaphragm was created and analyzed in the FEM (fine element method) system [2]. Based on the obtained results the optimized shape of the diaphragm was designed and its behavior was simulated. The new more suitable shape of the diaphragm is the result of this work.

Key-Words: diaphragm, elastomer, FEM (finite element method), hyperelasticity, numerical simulation, rubber

1 Introduction

Using finite element method (FEM) analysis leads to significant reduction of cost of design and moreover we can analyze deformation and stress of FEM model in more details then deformation of prototype.

We need to know how the rubber diaphragm will behave in real conditions in valve. If we would made and test a number of prototypes during design procedure it would be very time and money consuming. The monitoring of the diaphragm behaviour in real valve would be next problem and in a many industrial applications it is absolutely impossible to study behavior of the material in real situations [3, 4].

The diaphragm is made from silicone rubber. The large elastic strain is characteristic for behaviour of rubber. The stress-strain relation of elastomers is strongly nonlinear. Such materials are called hyperelastic and we can use a number of hyperelastic material models to simulate this nonlinear behaviour today [1].

A big progress in application of the Finite element method for simulation of physical problems was made in last years. All common hyperelastic models are incorporated in FEM systems today and the full apllication of nonlinear models of elastomers is allowed thanks development of the FEM systems.

We have to measure properties of every particular material for FEM analysis in laboratory tests. The elastomer for diaphragm was tested in two modes - uniaxial and equibiaxial tension [5, 6, 7].

The diaphragm is used as a component of pneumatic regulatory valve and it will be loaded by compressed air. The movable rod is fixed in the centre of the diaphragm. Different pressure on inner and outer side of the diaphragm moves the rod to its upper or lower position and thus some parameters can be regulated by the rod position.

2 Problem Formulation

2.1 Testing of the Diaphragm Material

For the exact evaluation of hyperelastic material constants, the test data obtained from the uniaxial and equibiaxial tension tests are suitable. The uniaxial tension test was performed on standard testing machine in accordance with ISO 37 standard.

There are not currently ISO standard methods for equibiaxial tension test and such tests are rarely performed in industrial laboratories. Thus the bubble inflation technique was used for equibiaxial characterization of diaphragm [6, 7]. In this method a uniform circular specimen of elastomer is clamped at the rim and inflated using compressed air to one side. The specimen is deformed to the shape of bubble. The inflation of the specimen results in an equibiaxial stretching near the pole of the bubble and in the planar tension near the rim. The inflation of the specimen and current value of pressure is recorded in short time intervals using a high resolution digital camera.

Obtained stress-strain relations for uniaxial and equibiaxial tension of diaphragm material are shown in Fig. 1.

2.2 The Material Model

The James-Green-Simpson (by another name 3rd order deformation) hyperelastic model was able to fit experimental data most closely and was chosen for analysis (Fig. 1). The strain energy density function W for this model is in the following form:

$$W = c_{10}(J_1 - 3) + c_{01}(J_2 - 3) + c_{11}(J_1 - 3)(J_2 - 3) + c_{20}(J_1 - 3)^2 + c_{30}(J_1 - 3)^3 \quad (1)$$

Measured coefficients (in Pa) for this model are: c_{10} = 510140; c_{01} = 70468.1; c_{11} = - 946.852; c_{20} = 36418.3; c_{30} = - 234.302, error of model is 1.822.

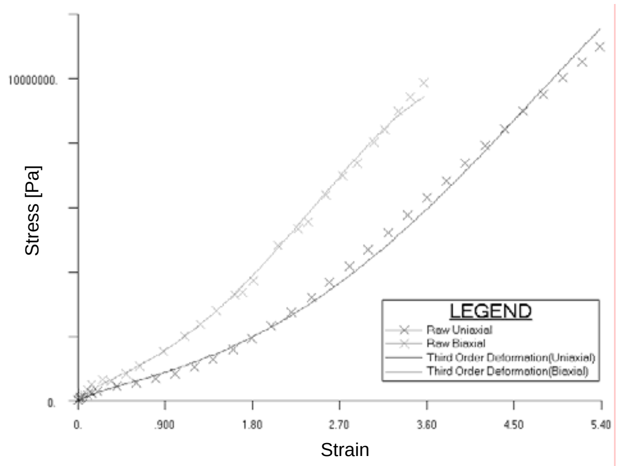

Fig. 1 Stress-strain diagram of experimental data and hyperelastic material model

2.3 Shape and Function of the Diaphragm

Initial shape of the diaphragm is shown in Fig. 2. Radius of the diaphragm is 35 mm, height is 12 mm and diaphragm thickness is 0.3 mm. Maximum stress value in the diaphragm must not exceed 2.5 MPa.

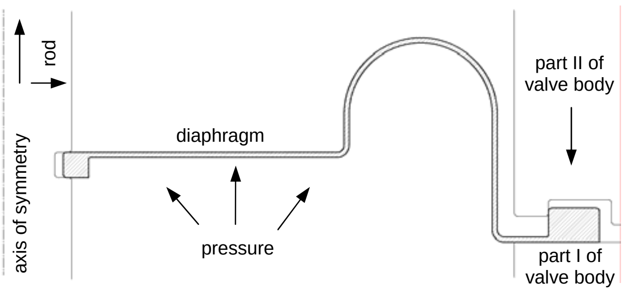

Fig. 2 Initial shape of the diaphragm, its boundary conditions and loading

2.4 Analysis

Due to the fact that the diaphragm has axisymmetric shape it is very useful to create 2D axisymmetric FEM model. The diaphragm is mounted on the rod in centre of valve and the outer rim of the diaphragm is clamped between two parts of valve body (Fig. 2).

Both parts of the valve body and the rod are modelled as absolutely rigid. The diaphragm is modelled as hyperelastic using material model (1) described above. Analysis consists of four steps (Fig. 2). The rod moves right in first step. In other words the diaphragm is mounted on the rod. Part II of valve body moves down in second step. The diaphragm is fixed in basic position in valve now. The rod is moved to its upper position in third step and pressure is applied in last step because maximum of pressure can occur only when the rod is up.

3 Problem Solution

Two criteria were used for results evaluation: maximum stress in the diaphragm (2,5 MPa) and functionality of the diaphragm. It means that the diaphragm must stay fixed in all other parts (the rod and the body of valve) during maximal loading and it must remain hermetic.

First critical point of initial design is located at the rim of diaphragm and it is shown in Fig. 3. This situation occurs at the end of second step. Next two steps were not carried out because very high stress (12 MPa) arose here. It is evident from Fig. 3 that there is not enough space in the groove of valve body for deformation of the diaphragm rim. Thus the enlargement of this groove was the first modification of design.

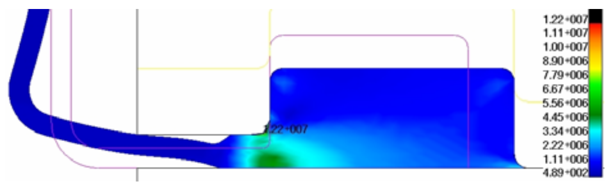

Fig. 3 Results of initial design of diaphragm (after 2nd step) – Von Mises stress

Results of the second version of design are shown in Fig. 4. Problem at the outer rim was

ISBN: 978-1-61804-004-6

solved and the maximum stress in this area is 1.5 MPa.

The most critical point in the second version is near the central rim of the diaphragm. Its position is pointed as A in Fig. 4 and stress maximum (4.5 MPa) is located here. We can also see that the rim of the diaphragm is almost pulled out from the rod groove. But we can find others points where stress values are locally much higher than stress in their vicinity (points B and C in Fig. 4). Although stress does not exceed 2.5 MPa in these points we can reduce stress values by next modification of shape of the diaphragm. The two corners of the initial shape are reason of stress concentration in these areas.

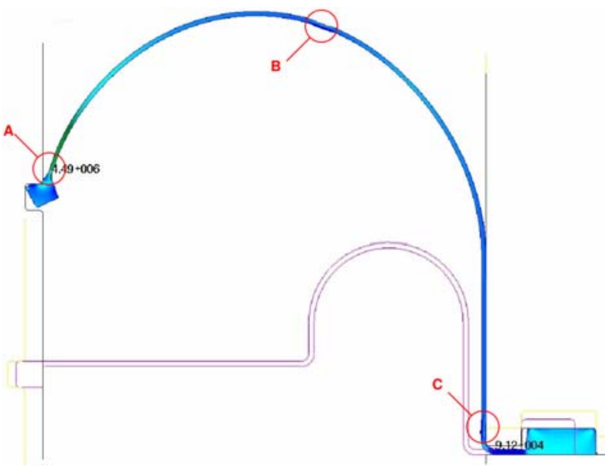

Fig. 4 Results of second version of design (after 4th step)

With consideration of these results the final shape modifications were done: for point A – widening diaphragm thickness close to central rim and increasing the depth of rod groove; for point B – elimination of corner; and for point C – increasing corner radius (Fig. 5).

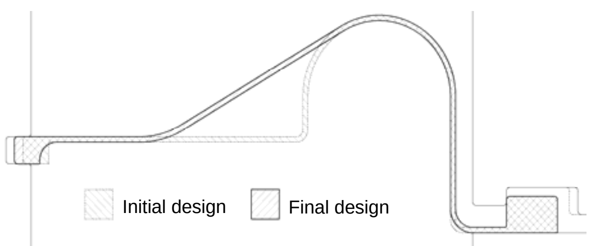

Fig. 5 Final shape of the diaphragm

The shape shown in Fig. 5 was taken as the final design because its analysis did not reveal any critical points. The stress maximum of this shape is located in the same point as in previous version, but its value was significantly reduced to 2.3 MPa. Also the central rim is much steadier in the rod groove. Reduction of stress in points B and C was reached too. Nevertheless it is not as significant as in case of the central rim (point A).

4 Conclusion

All criteria for the diaphragm were fulfilled and the final design of the diaphragm was created. Three versions of the diaphragm were analysed – initial, second and final version. The enlargement of groove in valve body was necessary after analysis of the initial version. The next model (second version) with this modification was created and analysed. With consideration of results of second version the rod groove was deepened and the shape of diaphragm was changed (final version – Fig. 5). The analysis of this modified shape proved that there are not exceeded stress/strain limits in the final version of diaphragm. Deformation does not exceed limit even in the initial model. But high stress was the reason of shape modification for the final version. The point where stress maximum is reached in diaphragm is near to the central rim (Fig. 4A).

Acknowledgement

This article is financially supported by the Ministry of Education, Youth and Sports of the Czech Republic under the Research Plan No. MSM 7088352102 and by the European Regional Development Fund under the project CEBIA-Tech No. CZ.1.05/2.1.00/03.0089.

References:

[1] Ogden, R.W., *Non-linear Elastic Deformations*, Dover Publications, 1997.

[2] Bower, A.F., *Applied Mechanics of Solids*, CRC Press, 2009.

[3] Manas, D., Manas, M., Stanek, M., Zaludek, M., Sanda, S., Javorik, J., Pata, V., Wear of Multipurpose Tire Treads, *Chemické listy*. Vol.103, No.13, 2009, pp.72-76.

[4] Manas, D., Stanek, M., Manas, M., Pata, V., Javorik, J., Influence of Mechanical Properties on Wear of Heavily Stressed Rubber Parts, *KGK – Kautschuk Gummi Kunststoffe*, Vol.62, No.5, 2009, pp. 240-245.

[5] Smith L.P., *The language of Rubber*, Butterworth-Heinemann, 1993.

[6] Reuge N., Schmidt F.M., Le Maoult Y., Rachik M., Abbé F., Elastomer Biaxial Characterization Using Bubble Inflation

technique. I: Experimental Investigations, *Polymer Engineering and Science*, Vol.41, No.3, 2001, pp. 522-531.

[7] Javorik J., Dvorak Z., Equibiaxial test of elastomers. *Kautschuk Gummi Kunststoffe*, Vol.60, No.11, 2007, pp. 608-610.

ISBN: 978-1-61804-004-6