ON STRESS ANLYSIS
FOR A HYPERELASTIC MATERIAL

Xiao-Yan Gong and Riyad Moe

Sulzer Carbomedics Inc.
Austin, Texas

ABSTRACT

Performing a finite element analysis (FEA) on a hyperelastic material is difficult due to nonlinearity, large deformation, and material instability. This paper provides a brief review of the hyperelastic theory and discusses several important issues that should be addressed when using ANSYS. Analysis on a fatigue specimen is used as an example of one of our product development challenges upon which these issues were illuminated. We believe that a stable material model in combination with a good understanding of structural instability for traditional materials is the key to success in simulating hyperelastic materials using ANSYS.

1. INTRODUCTION

The human heart contains four heart valves, which open and close during heartbeats to produce the one-way blood flow to supply the human body. One or more heart valves may malfunction or fail for many reasons including congenital defect, aging, or disease. In any event, surgery can be done to replace the failed or malfunctioned heart valve with a prosthetic heart valve. Flexible materials are very attractive to the prosthetic heart valve industry for their outstanding hemodynamic performance. Heart valves using these kinds of materials typically have a trileaflet design (Figure 1). The leaflets sometimes are fixed to a frame called a stent for structural support. Currently such flexible valves are commercially available with leaflets made of treated animal tissues. The drawback of such a tissue valve is its service life. Generally speaking, the average lifetime of a tissue valve is about a decade. The main reason for a tissue valve to malfunction is the tearing of the treated animal tissues due to aging and calcification.

To improve upon the durability of current flexible valves, our company, Sulzer Carbomedics Inc., is working on solving the design challenges in order to produce a polymer prosthetic heart valve. At a normal heart rate of 70 beats per minute, a heart valve is subjected to four hundred million cycles in ten years. It is obvious that for a polymer prosthetic heart valve to outlast the tissue valves, it has to be subjected to even more cycles. Such an application discourages the old-fashioned design cycles, *i.e.*, design-test and redesign-retest processes. Therefore we turn to the most up-to-date computer technology for virtual design and prototyping. The core of virtual design is precise stress analysis. With the precise stress analysis and stress-life (S-N) curve, the analyst can estimate the device service life. We picked ANSYS for its built-in hyperelastic material model and geometric modeling capability.

Figure 1. An example of a tissue valve (Sulzer Carbomedics, Mitroflow Synergy PC)

In using ANSYS for stress analyses on hyperelastic materials, we experienced many convergence difficulties mainly because a hyperelastic material can undergo several orders of magnitude higher strain than a traditional material. In finite element simulation, large deformation can both cause the excessive distortion of originally well-shaped elements, resulting in element failure, as well as increase the chance of structural instability such as buckling and wrinkling. However, mesh distortion and structural instability also occur in traditional materials, and are, therefore, nothing new to analysts. There are special techniques to handle them [1][2]. Material instability, on the other hand, may be new to the analyst who is not familiar with hyperelasticity theory. It adds additional confusion and difficulties in the finite element analyses. This is mainly because, when mixed with structural instability and mesh distortion, the cause of a divergent solution is not so clear. Most of the time what the analyst sees is an error message indicating a divergent solution after long nonlinear ANSYS runs. The code cannot distinguish the cause of a divergent solution. The analyst will find old helpful techniques are not useful when facing an unstable material model. Fortunately, material instability can be anticipated by experts in the field of hyperelasticity, in particular, the violation of Drucker's stability postulation, volumetric locking in element formulation, and failure of linear elasticity theory. Thus the most effective way to resolve the divergence issues in FEA on a hyperelastic material is to obtain a stable material model before doing the analysis. With a stable material model, the analyst can then apply techniques for traditional materials to solve the divergence problems due to element distortion and structural instability.

We used ANSYS 5.4 and 5.5 for our analyses and tested ANSYS 5.6 beta. In all these versions, for a nearly incompressible hyperelastic material, ANSYS supports a nine-term up to third order polynomial strain energy density function model (called Mooney-Rivlin in ANSYS) from which the stress-strain relation of a hyperelastic material is derived. The nine coefficients, called hyperelastic constants, are obtained by curve-fitting the three recommended material characterization experiments [3]. ANSYS allows the analyst to input either the hyperelastic constants or the test data. Test data can be either typed into the GUI directly or imported from a file. When test data are used as input, ANSYS calculates the hyperelastic constants and provides a quick graphical comparison of the material model response and the experimental data [2].

We noticed that there are several improvements in ANSYS 5.6, e.g., addition of another material model (Boyce model), additions of hyperelastic element HYPER84 and HYPER86 and SOLID156, SOLID157 and SOLID158, and, above all, the addition of the stability check on uniaxial and biaxial loading cases [1][4].

However, none of these completely resolve the issues presented in this paper. First, certain warning messages are displayed before a solution procedure is actually performed in ANSYS on hyperelastic materials. Second, the appropriate quantity of Poisson's ratio in ANSYS is a mystery. Third, even though ANSYS may fit the testing data nicely, there is no guarantee that this fit leads to a stable material model.

2. BASIC ISSUES IN HYPERELASTICITY

2.1. General Review

To address these issues, it is useful to briefly review implications of polymers and hyperelastic theory. Polymers are capable of large deformations and are subject to tensile and compressive stress-strain curves that are not centrally symmetric through the origin. The simplest yet relatively precise description for this type of material is isotropic hyperelasticity. Malvern showed that for isotropic hyperelastic material, the recoverable strain energy density with respect to the initial, unstressed configuration could be written as a function of the principal invariants of the Cauchy-Green deformation tensor [5]. Furthermore, the whole mathematical process is standardized and nicely summarized by Kaliske and Rothert [6]. The key idea is to define a stress energy density function from which the stress can be derived by taking the partial derivatives conjugate to the strain. In theory, the selection of such a strain energy density function is unlimited and arbitrary. However, ANSYS (by release 5.5) only supports one of the most generic strain energy density functions, *i.e*., a third order polynomial form up to nine terms. We choose to use a complete second order polynomial form because it appears that five terms are sufficient for our application. One advantage of the polynomial material model, in our case, is that it approximates all the testing data nicely. One drawback is that there is a material model stability issue, *i.e*., the material model may violate Drucker's stability postulation (a monotonic increase of strain energy density with increase in strain). Violation of Drucker's stability postulation can lead to a numerical failure and a divergent solution. Such a material stability issue is unique to hyperelasticity because there are flow rules in metal plasticity to assure Drucker's stability postulation is satisfied. Additionally, for theoretical simplicity, hyperelastic materials are assumed to be incompressible. An incompressible material will cause volumetric locking that requires special treatment on element formulation. This limits the choice of element types for stress analysis on a nearly incompressible hyperelastic material to (up to release 5.5) first order solid elements, HYPER56 and HYPER58, structural element SHELL181, and second order solid elements HYPER74 and HYPER158. Even with the proper choice of the element type, a legitimate concern is the proper quantity for Poisson's ratio. Incompressibility means Poison's ratio is equal to 0.5. However, linear elastic theory cannot support this value.

It is useful to write the basic definition of some important terms in hyperelasticity theory before we go further. Given the principal stretches at any deformation state of a material point as, λ_1, λ_2, and λ_3, the strain invariants are defined as,

$$\begin{aligned} I_1 &= \lambda_1^2 + \lambda_2^2 + \lambda_3^2 \\ I_2 &= \lambda_1^2\lambda_2^2 + \lambda_2^2\lambda_3^2 + \lambda_3^2\lambda_1^2 \, . \\ J &= \lambda_1\lambda_2\lambda_3 \end{aligned} \tag{1}$$

Upon eliminating the volumetric strain, the deviatoric strain invariants are written as,

$$\overline{I_1} = I_1 / \sqrt[3]{J}; \qquad \overline{I_2} = I_2 / \sqrt[3]{J} \, . \tag{2}$$

With these quantities defined, the five-term Mooney-Rivlin model is written as [3],

$$U = \sum_{i+j=1}^{2} \mathbf{C}_{ij} \left(\overline{I}_1 - 3\right)^i \left(\overline{I}_2 - 3\right)^j + \frac{\kappa}{2}\left(J-1\right)^2 . \tag{3}$$

Once this function is determined, *i.e*., $\mathbf{C}_{ij}$s and κ have been fit from experimental data, the hyperelastic material model is defined. For incompressible material, the deviatoric strain invariants are approximately equal to the original strain invariants. Thus the deviatoric strain invariants are simply referred to as strain invariants in the latter contents.

Even though the ANSYS theory manual uses the same notation for $\mathbf{C}_{ij}$s as they appear in equation (3), ANSYS GUI uses a different notation. Relations between these coefficients are tabled below.

ANSYS GUI	C_1	C_2	C_3	C_4	C_5
Equation (3)	$\mathbf{C}_{10}$	$\mathbf{C}_{01}$	$\mathbf{C}_{20}$	$\mathbf{C}_{11}$	$\mathbf{C}_{02}$
ANSYS GUI	C_6	C_7	C_8	C_9	
Equation (3)	$\mathbf{C}_{30}$	$\mathbf{C}_{21}$	$\mathbf{C}_{12}$	$\mathbf{C}_{03}$	

2.2. ANSYS Warning Messages on Material Model

ANSYS did a wonderful job in curve fitting the material model to our material test data. However, most of the time before a FEA solution was performed, a warning message such as $\mathbf{C}_{20}$ or $\mathbf{C}_{20}$ + $\mathbf{C}_{11}$ + $\mathbf{C}_{02}$ is not positive or $\mathbf{C}_{02}$ is not negative was displayed. We propose that these warnings are generally not of great concern. To understand the meaning of these warning messages, it is useful to write the stress strain relations on the three recommended experiments for calibrating the constitutive model.

2.2.1. Planar (pure shear) loading

The so-called pure shear test is actually analogous to a plane strain condition in which one of the transverse directions is constrained so that there is no deformation along this direction. In this case, we have,

$$\lambda_1 = \lambda_S,\ \lambda_2 = 1,\ \lambda_3 = \lambda_S^{-1}\,. \tag{4}$$

The deviatoric strain invariants are written as,

$$I_1 = I_2 = 1 + \lambda_S^2 + \lambda_S^{-2}\,. \tag{5}$$

The engineering stress and stretch relation is,

$$\begin{aligned} T_B = 2\left(\lambda_S - \lambda_B^{-3}\right)[&\mathbf{C}_{10} + \mathbf{C}_{01} \\ &+2\left(\mathbf{C}_{20} + \mathbf{C}_{11} + \mathbf{C}_{02}\right)\left(I_1 - 3\right)] \end{aligned}\,. \tag{6}$$

As the use of a hyperelastic material model is mainly for cases of large strain, it is important to make sure that the dominant term is stable at large strain. When λ_S is large, the stress will be dominated by the last term of equation (6) that contains I_1. Clearly if $\mathbf{C}_{20}$+$\mathbf{C}_{11}$+$\mathbf{C}_{02}$ is not positive, the above stress-strain relation gives rise to a negative tangential stiffness at large strain. The negative tangential stiffness violates Drucker's stability postulation, and therefore cannot be modeled by current finite element formulations. We reason this is why ANSYS issues a warning message when $\mathbf{C}_{20}$+$\mathbf{C}_{11}$+$\mathbf{C}_{02}$ is not positive.

2.2.2. Uniaxial loading

Under uniaxial loading,

$$\lambda_1 = \lambda_U,\ \lambda_2 = \lambda_3 = 1/\sqrt{\lambda_U}\,. \tag{7}$$

The strain invariants are written as,

$$I_1 = \lambda_U^2 + 2\lambda_U^{-1};\quad I_2 = \lambda_U^{-2} + 2\lambda_U\,. \tag{8}$$

The engineering stress-stretch relation is written as,

$$\begin{aligned} T_U = 2\left(1 - \lambda_U^{-3}\right)[&\mathbf{C}_{10}\lambda_U + \mathbf{C}_{01} \\ &+\left(2\mathbf{C}_{20}\lambda_U + \mathbf{C}_{11}\right)\left(I_1 - 3\right) \\ &+\left(2\mathbf{C}_{02} + \mathbf{C}_{11}\lambda_U\right)\left(I_2 - 3\right)] \end{aligned}\,. \tag{9}$$

Taking the limit of equation (9), one can immediately show that when λ_U is large, T_U can simply be approximated as,

$$T_U = 4\mathbf{C}_{20}\lambda_U^3. \tag{10}$$

Clearly, it requires $\mathbf{C}_{20}$ to be positive. Once again, we think this is why ANSYS issues a warning message when $\mathbf{C}_{20}$ is not positive.

2.2.3. Equibiaxial loading

Under this condition, we have,

$$\lambda_1 = \lambda_2 = \lambda_B,\ \lambda_3 = \lambda_B^{-2}\,. \tag{11}$$

The deviatoric strain invariants are written as,

$$I_1 = 2\lambda_B^2 + \lambda_B^{-4};\quad I_2 = 2\lambda_B^{-2} + \lambda_B^4\,. \tag{12}$$

The engineering stress and stretch relation is,

$$\begin{aligned} T_B = 2\left(\lambda_B - \lambda_B^{-5}\right)[&\mathbf{C}_{10} + \mathbf{C}_{01}\lambda_B^2 \\ &+\left(2\mathbf{C}_{20} + \mathbf{C}_{11}\lambda_B^2\right)\left(I_1 - 3\right) \\ &+\left(2\mathbf{C}_{02}\lambda_B^2 + \mathbf{C}_{11}\right)\left(I_2 - 3\right)] \end{aligned}\,. \tag{13}$$

If one agrees with our logic in the previous two special cases, one can see that $\mathbf{C}_{02}$ has to be positive. This is inconsistent with what the ANSYS message indicates. We do not have a simple explanation for this particular warning message. We cannot find an explanation of any of these three warning messages from ANSYS either.

Regardless, we propose there is little concern about any of these warning messages. The coefficients are obtained in the first place by curve fitting the experimental data. Therefore, there is no reason to expect them to satisfy these stated conditions. Two of the three warning messages are focused on mathematical limits. The mathematical limits are far enough away from most engineering applications to be of little concern. A stretch of 1.5, corresponding to 50% of engineering strain, is considered mathematically small, yet is a large strain for most engineering applications. Furthermore, even if the warning messages do not appear, the material model can still be unstable. In one case, we selected material constants such that there were no warning messages issued, but the stability check in ANSYS 5.6 indicated that the material model could still be unstable. Therefore, such warning messages can be ignored.

2.3. Poisson's Ratio and Incompressibility

Incompressibility is purely a mathematical simplification. It leads to many theoretical simplifications in constitutive model formulation and simple stress analyses. However, numerically, it is not easy to handle. The most common treatment in finite element analysis is to introduce a penalty (hydrostatic) term [last term in equation (3)] in strain energy (stress) formulation to enforce the incompressible condition of J = 1. This normally requires the selection of a large number for the penalty coefficient (bulk modulus). The relation of Poisson's ratio and bulk modulus is derived below.

By evaluating the derivative of uniaxial stress with respect to uniaxial strain [equation (9)] at λ_U =1, the Young's modulus at zero strain takes the form,

$$E = 6(\mathbf{C}_{10}+\mathbf{C}_{01}). \quad (14)$$

As a result, the bulk modulus, κ, can be written as a function of shear modulus and the Poisson's ratio. The formula is,

$$\kappa = \frac{2(\mathbf{C}_{10}+\mathbf{C}_{01})}{(1-2\nu)}. \quad (15)$$

One can tell from this formula, as Poisson's ratio approaches 0.5, the bulk modulus κ (penalty coefficient) of the material approaches infinity. In theory, when κ goes to infinity, the material is fully incompressible and the condition of J = 1 is numerically enforced. However, in reality, there is no such material that is fully incompressible. ANSYS 5.6 suggests a value of 0.4999, but does not explain the reason [1]. Ideally, a hydrostatic pressure experiment can be performed to measure the bulk modulus κ and the analyst can use equation (15) to find the precise Poisson's ratio. However, in most engineering applications, slight variation of the Poisson's ratio does not affect the stress analysis results. This is consistent with the penalty method, i.e., there exists a point when the selection of the bulk modulus κ or Poisson's ratio does not affect the stress strain relation. This particular value of the Poisson's ratio varies from application to application. Generally speaking, Poisson's ratio must be increased to enforce the incompressibility, as the hydrostatic compression on a structure increases. Computing time is increased as the Poisson's ratio approaches the value of 0.5. From practice, we found that a Poisson's ratio of 0.475 is sufficient for our applications.

2.4. Single Element Check

Even though one can fit the testing data nicely in ANSYS, there is no guarantee that the best fit to the testing data leads to a stable material model. This is true even if data from all three recommended material tests are available. The analyst may still run into divergence problems and see the nodes fly around in the ANSYS graphics window. In ANSYS 5.6, the program is smart enough to perform a stability check for the material model and report the stability limits under uniaxial and biaxial loading conditions. But, the planar loading condition is not checked. Furthermore, there is not a straightforward manner to get a stable material model from the test data in the first place. An iterative technique of curve fitting and stability checking may be necessary. The analyst might consider sacrificing fit accuracy to gain a stable material model. Nevertheless, even if the analyst believes a stable material model has been obtained, we recommend a simple approach to double check the material model and the selection of the Poisson's ratio. Stress analyses should be performed on a single element in six different loading cases: tension and compression of uniaxial, equibiaxial, and planar. Only then should the model be used in stress analyses of the intended structures. Upon completion of stress analyses of the intended structures, assumptions of curve fitting accuracy should be re-evaluated. The analyst and designer must be willing to consider iterating the entire process.

One additional issue is that for experimental simplicity, stress and strain are measured as their engineering values, yet ANSYS outputs true stress and logarithmic strain. The relation between engineering stress and true stress in simple loading cases is written as,

$$\sigma_E = \sigma_T/\lambda, \quad (16)$$

where λ is the principal stretch along the principal stress direction. The logarithmic strain can be easily converted to engineering strain through the formula,

$$\varepsilon_E = \exp(\varepsilon_T) - 1. \quad (17)$$

When using the single element stress analyses to double-check the material model, it is very convenient to use a unit cubic element to do the analysis. This way, by tracking the load and displacement history of the unit element, one automatically has the engineering stress and engineering strain responses.

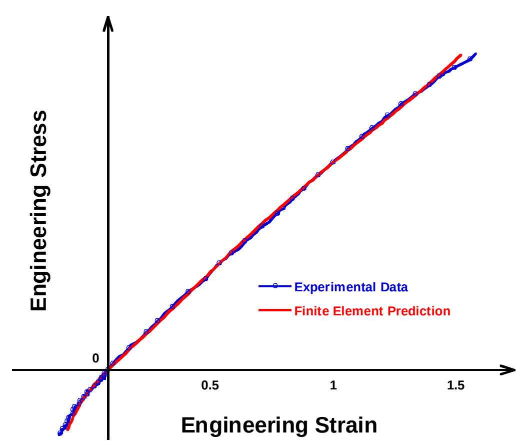

Figure 2. Comparison of a typical stress stretch plot from ANSYS and experiments.

We strongly suggest that the analyst graphically compare the data from the ANSYS single element analyses against the experimental data. A typical plot of a uniaxial loading case is shown in Figure 2. There is a very close fit to the experimental data up to engineering strain of 1.5. Beyond this strain, the material model is still stable, but the fit departs from the experimental data. The analyst can make assumptions of the appropriate limit to the curve-fitting regime.

3. CASE STUDY

The above stable material model is used to analyze a quarter of a notched fatigue specimen shown in Figure 3 to obtain a precise prediction of the stress and life relation (S-N curve). Also shown in figure 3 is the finite element mesh and constraints. A small elemental aspect ratio is used with element concentration on the potentially largest deformation area. HYPER58 is used in this analysis.

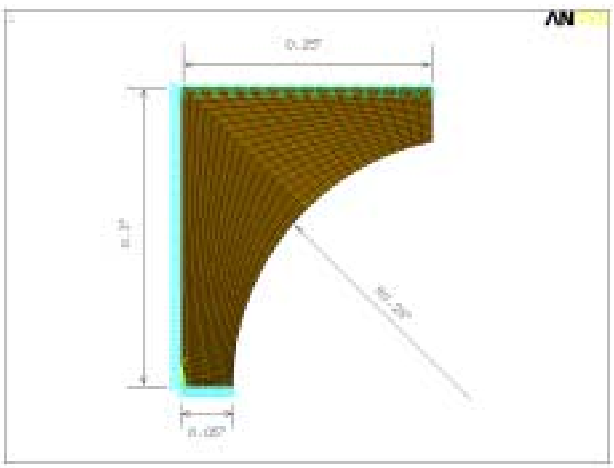

Figure 3. Front view of the FEA model on the fatigue specimen with uniform thickness of 0.01".

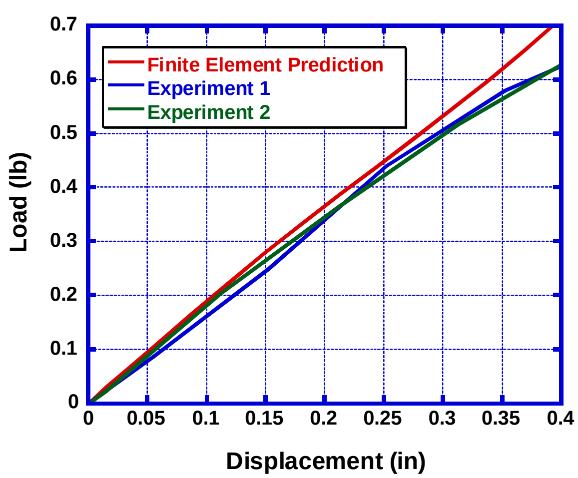

Figure 4. Comparison of load-displacement history between analysis and experiment.

Figure 4 shows the comparison of load displacement history from FEA and experiments. FEA agrees well with the experiments at relatively low loading and diverges at the higher loading levels. Some of these discrepancies are due to the difficulties in experimentation. Others are due to the deviation of the material model description. Neither is an issue at the relatively low stress region. This relatively low stress region is where we are most interested in our application. Thus we are confident with the results.

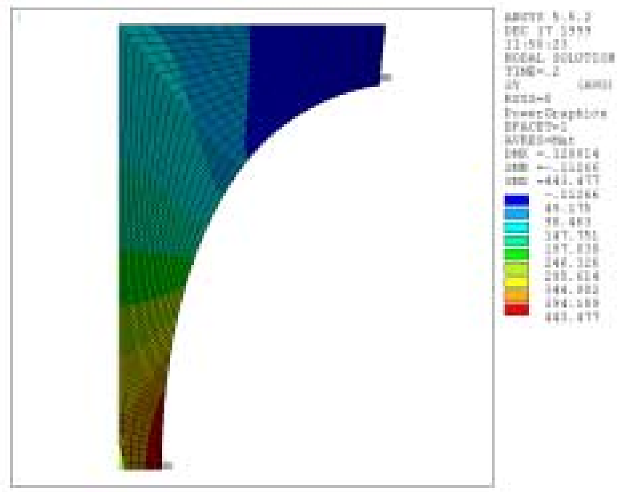

Figure 5. Longitudinal normal stress plot contour.

Figure 5 shows the longitudinal normal stress contour plot at a total displacement of 0.12" on the deformed elements. Near the notch tip, the elements have been stretched to a larger aspect ratio. There is a stress concentration at the notch tip. Stress concentration on a hyperelastic material depends on deformed geometry and material response. It is very hard to identify the correct stress values analytically even for the simple geometry shown in Figure 3. Therefore the FEA is the only way to precisely predict the correct S-N curve.

Our target application is far more complicated than this simple example. However, with confidence in the material model we are able to focus our attention on resolving other issues, such as mesh distortion and structural instability, through the techniques appropriate for traditional materials. We have had success in stress analysis in these complicated applications.

4. CONCLUSION

Despite the complex application nature of flexible prosthetic heart valves, which involve nonlinear geometry and nonlinear material responses, we are able to perform stress analysis successfully. In using ANSYS to simulate such complicated hyperelastic problems, we found that understanding the fundamentals of the hyperelastic theory is very helpful. We propose that certain warning messages may be ignored. We appreciate the improvement in ANSYS 5.6 that provides a check on the stability of the material model. We recommend a single element test to confirm the material model. We also draw attention to the difference in stress and strain definitions from typical experimental data and the analysis output.

ACKNOWLEDGMENT

The authors would like to thank their colleague Mr. Edward Sarnowski for providing the experimental data and some helpful discussion in this article.

REFERENCES:

[1]. ANSYS structural analysis guide for release 5.6.
[2]. ANSYS structural nonlinearities training manual for release 5.5.
[3]. ANSYS theory manual.
[4]. ANSYS element reference manual for release 5.6.
[5]. Malvern, L.E., (1969), *Introduction to the Mechanics of a Continuous Medium*, Prentice-Hall, New York.
[6]. Kaliske, M. and Rothert, H., (1997), "On the finite element implementation of rubber-like materials at finite strains", Engineering Computations, Vol. 14, No. 2, pp216-232.