Science & Technology Development Journal, 24(SI1):SI43-SI50

Open Access Full Text Article

Research Article

Finite element analysis for three-dimensional hyper-elastic problems

Trong Duc Nguyen[1,2], Trong Tran Hieu Huynh[1,2], Nha Thanh Nguyen[1,2,*], Hien Thi My Nguyen[3], Thien Tich Truong[1,2]

[1]*Department of Engineering Mechanics, Faculty of Applied Sciences, Ho Chi Minh City University of Technology, 268 Ly Thuong Kiet Street, District 10, Ho Chi Minh City, Viet Nam*

[2]*Vietnam National University Ho Chi Minh City, Linh Trung Ward, Thu Duc District, Ho Chi Minh City, Viet Nam*

[3]*Institute of Transport and Environment Research, Ho Chi Minh City University of Transport, 2 Vo Oanh Street, Ward 25, Binh Thanh District, Ho Chi Minh City, Viet Nam*

Correspondence

Nha Thanh Nguyen, Department of Engineering Mechanics, Faculty of Applied Sciences, Ho Chi Minh City University of Technology, 268 Ly Thuong Kiet Street, District 10, Ho Chi Minh City, Viet Nam

Vietnam National University Ho Chi Minh City, Linh Trung Ward, Thu Duc District, Ho Chi Minh City, Viet Nam

Email: nhanguyen@hcmut.edu.vn

History

- Received: 2021-08-31
- Accepted: 2021-12-13
- Published: 2022-28-02

DOI : 10.32508/stdj.v24iSI1.3797

ABSTRACT

Hyper-elastic materials are special elastic materials that the stress that can be derived by the strain energy density function. The most attractive property of these rubber-like materials is their ability to undergo large strains with small loads and recover their initial shape after unloading. Hyper-elastic materials are widely used in engineering, such as tires, elastomeric bearing pads, belt drives, rail pads, and so on. Due to the large strain state in most the applications of this type of material, hyper-elastic behavior is often considered infinite deformation analysis. *Compared with linear material, analysis for hyper-elastic material with nonlinearities is much complicated because both material,* and geometric nonlinearity must be considered. Along with the development of computer and numerical methods, the finite element method (FEM) is a powerful numerical computation tool that can help to solve engineering problems in many fields such as structural, thermal, electronic, and biomedical analyses. Actually, most practical engineering problems appear in three-dimensional (3D) states, especially in non-linear analyses. So, developing an effective numerical method for 3D non-linear problems is very necessary. In this study, a finite element approach with an 8-node hexahedron element is presented to analyze large deformation problems of *hyper-elastic* material. A computing program is built with Matlab language to simulate the non-linear behavior of *hyper-elastic 3D models.* The compressible neo-Hookean is used as the constitutive model for 3D hyper-elastic problems, and the total Lagrange formulation is applied to discretize large deformation problems. T-he standard Newton-Raphson algorithm with a constant load step is chosen as the iteration method to obtain the non-linear solutions. To verify the accuracy of the numerical algorithm and program, the obtained results are compared with the reference solutions given by the Ansys program.

Key words: hyper-elastic, large deformation, 3D finite element method, non-linear analysis

INTRODUCTION

HYPER-ELASTIC material is the special class of non-linear materials with specific characters: they respond elastically even when subjected to large deformations (geometric nonlinearity) and show a highly non-linear stress-strain relation (material nonlinearity). For example, rubber and rubber-like materials are typical hyper-elastic materials. They are important and widely used in industrial fields such as building, automotive, electronics, and petroleum [1–4].

Numerical analysis for hyper-elastic material is difficult and challenging because of nonlinearities. Several constitutive models developed for hyper-elastic materials such as Neo-Hooken, Mooney-Rivlin, Ogden… [5]. In these non-linear materials, the relations between stress and strain are given by the strain energy functions. To perform analysis for the behavior of these materials, a non-linear solver with several iterations must be applied.

P*roblems relevant to hyper-elastic material* play an important role in industries. R. Hassani et al. [6] use a numerical solution technique named variational differential quadrature (VDQ) to analyze large deformation of 2D hyper-elastic bodies based on the compressible non-linear elasticity. F. Peng [7] and partners use FEM to perform simulations of hyper-elastic material fracture behavior at large deformation. With analytical solutions and FEM, A. Almasi [8] and partners investigated thermomechanical of hyper-elastic thick-walled cylindrical pressure vessels. Guangdi Hu and P. Wriggers [9] studied steady-state rolling contact of the hyper-elastic model. Shahab Sahraee and Farzam Dadgar-Rad [10] discussed large deformation of fully incompressible hyper-elastic curved beams.

The purpose of this study is to present a three-dimensional finite element approach for non-linear analysis of hyper-elastic problems. The computational program built with Matlab language can do the calculations for the behaviors of hyper-elastic mod-

Cite this article : Nguyen T D, Huynh T T H, Nguyen N T, Nguyen H T M, Truong T T. **Finite element analysis for three-dimensional hyper-elastic problems**. *Sci. Tech. Dev. J.;* 24(SI1):SI43-SI50.

els such as displacement, strain, and stress. The obtained results from the computational program are compared with those of the Ansys program to verify the accuracy of the built program. The paper is organized as follows: After this introduction, the material constitutive law is presented in the second section. The next part shows the formulations of the finite element approach for 3D problem. Numerical examples and results are given in the following section. After that, some discussions are given and finally, the conclusion is given the last section.

CONSTITUTIVE EQUATIONS FOR HYPER-ELASTIC NEO-HOOKEAN MODEL

The constitutive law for anisotropic hyper-elastic material is defined by an equation relating the strain energy density of the material to the deformation gradient or the three invariants of the strain tensor. For the generalized Neo-Hookean model, the strain energy function is given as

$$U(C) = \frac{\kappa}{2}(\ln J)^2 + \frac{\mu}{2}(I_1 - 2\ln J - 3) \tag{1}$$

where κ and μ are the bulk modulus and the shear modulus, respectively. I_1 is the first invariant of the right Cauchy-Green tensor $C = F^T F$ and the J parameter relates with C tensor:

$$I_1 = trC;\ J = detF = \sqrt{detC} \tag{2}$$

In compressible Neo-Hookean material, the second Piola–Kirchhoff stress is given by the constitutive law for non-linear stress-strain relation

$$S = \kappa \ln J (C)^{-1} + \mu\left(I - C^{-1}\right) \tag{3}$$

where I is the unit tensor

THREE-DIMENSIONAL FINITE ELEMENT FORMULATIONS FOR NON-LINEAR ANALYSIS

Weak form and discretized formulations

Consider a 3D continuum domain Ω of a hyper-elastic body as presented in Figure 1. The body is subjected to a traction t* on boundary Γ_t; it has a body force $\bar{b}$ and a prescribed displacement $\bar{u}$ on boundary Γ_u. The general weak form formulation for non-linear problem defined in the undeformed configuration is written as

$$\int_\Omega S : \nabla\delta u dV - \int_\Omega \delta u \cdot \bar{b} dV - \int_{\Gamma_t} \delta u \cdot t^* d\Gamma = 0 \tag{4}$$

where δu is the vector of test functions.

The displacement field u is approximated by using the vector of shape functions N and nodal displacement vector of element u^e. The approximated displacement, the variational of displacement, and the gradient of test functions are given as below

$$u = Nu^e;\ \delta u = N\delta u^e;\ \nabla\delta u = B\delta u^e \tag{5}$$

where matrix B is the B-operator that contents the derivatives of shape functions with respect to the initial configuration. Substituting (5) to (4), the weak form can be obtained as follow

$$\int_\Omega B^T S dV - \int_\Omega N^T \bar{b} dV - \int_{\Gamma_t} N^T t^* d\Gamma = 0 \tag{6}$$

The weak form can be rewritten by the residual vector R that stands for the difference between internal force F_{int} and external force F_{ext} in each step of the Newton-Raphson algorithm.

$$R(u) = F_{int} - F_{ext} \tag{7}$$

$$F_{ext} = \int_\Omega N^T \bar{b} dV - \int_{\Gamma_t} N^T t^* d\Gamma \tag{8}$$

$$F_{int} = \int_\Omega B^T S dV \tag{9}$$

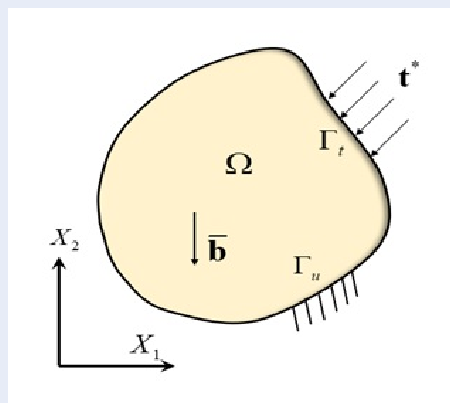

Figure 1: Hyper-elastic 3D body with boundary conditions

Perform the linearlization procedure to Eqs. (6) and (7), two non-linear parts for finite deformation analysis are derived

$$\left(\int_\Omega \triangle B^T \sigma dV + \int_\Omega \triangle B^T \triangle\sigma dV\right)\triangle\sigma = K_{\tan\triangle u} \tag{10}$$

where $K_{\tan}$ is the tangent stiffness matrix that includes the geometric non-linear and material non-linear parts of the problem

$$K_{\tan = \int_\Omega G^T M G dV + \int_\Omega B^T \triangle C^E B dV} \tag{11}$$

In Eq. (11), matrices M and G are defined at each node i in an element as following

$$G^i = \begin{bmatrix} I_{3\times3}\dfrac{\partial\phi_i}{\partial x} \\ I_{3\times3}\dfrac{\partial\phi_i}{\partial y} \\ I_{3\times3}\dfrac{\partial\phi_i}{\partial z} \end{bmatrix} \quad and\ M = \sigma\otimes I_{3\times3} \tag{12}$$

where $I_{3\times3}$ is the (3×3) identity matrix

Hexahedron elements in natural coordinates

Hexahedron elements are described in the expansion of quadrangular elements, in natural coordinates $\xi_j = [-1,1]$ $(j = 1,2,3)$ that are assembled in the natural coordinate vector $\xi_j = \begin{bmatrix}\xi & \eta & \zeta\end{bmatrix}^T$.

In Figure 2, natural coordinates make up the master or unit cube of 3D elements on which ansatz functions of arbitrarily shaped hexahedron elements can be generated. On the left side, the element domain is given, and the strain components are defined corresponding to the three natural coordinates.

Shape functions for eight-node hexahedron element

Consider a 3D domain, which is meshed to create several hexahedron elements. Each element has eight nodes and six surfaces; its nodes are numbered by 1, 2, 3, 4 and 5, 6, 7, 8 in a counterclockwise manner, as shown in Figure 1.

There are three degrees of freedom (DOFs) at one node, so the element has total of 24 DOFs. Using the natural coordinate system , eight shape functions for a hexahedron element is given as

$$\begin{aligned}
N_1 &= \frac{1}{8}(1-\xi)(1-\eta)(1-\zeta)\\
N_2 &= \frac{1}{8}(1+\xi)(1-\eta)(1-\zeta)\\
N_3 &= \frac{1}{8}(1+\xi)(1+\eta)(1-\zeta)\\
N_4 &= \frac{1}{8}(1-\xi)(1+\eta)(1-\zeta)\\
N_5 &= \frac{1}{8}(1-\xi)(1-\eta)(1+\zeta)\\
N_6 &= \frac{1}{8}(1+\xi)(1-\eta)(1+\zeta)\\
N_7 &= \frac{1}{8}(1+\xi)(1+\eta)(1+\zeta)\\
N_8 &= \frac{1}{8}(1-\xi)(1+\eta)(1+\zeta)
\end{aligned} \tag{13}$$

NUMERICAL EXAMPLES

Problem 1: A 3D block under tensile load

The first problem deals with a 3D solid hyper-elastic model, as described in Figure 3. The dimensions are $R_2 = 2$ and $R_3 = 3$. The compressible Neo-Hookean model is applied for this problem, and the material parameters are assumed as shear modulus $\mu = 1Pa$ and bulk modulus $\kappa = 10Pa$. The displacements on x and z directions of face ABCD are constrained. Similarly, the displacements on y and z directions of face IJGH are fixed. A unit force $F = 1N$ is applied to face EHGF on x-direction. The model includes 117 nodes with 48 hexahedron elements.

The non-linear problem is solved by Matlab code with a standard Newton-Raphson algorithm with 15 uniform load steps. To verify the built code for 3D non-linear analysis, the same model is also performed with the Ansys program. The obtained result of displacement in x-direction is shown in Figure 4 in comparison with solution from Ansys with the same mesh. The deformation shape is very similar between the two solutions.

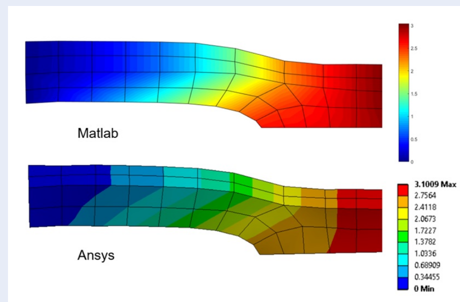

Figure 4: Deformation in x-direction: Matlab (top), Ansys (bottom)

Table I shows the overall error of displacements between Matlab and Ansys on the whole model. The formulation for error is $\varepsilon = ||u_{Matlab} - u_{Ansys}|| / ||u_{Ansys}|| \times 100\%$. The obtained results are acceptable.

Plots in Figure 5 display the distributions of normal stress in x direction. The maximum of normal stress σ_x given by Matlab is $2.35Pa$, and this of Ansys is $2.302Pa$, good accuracy is obtained.

Problem 2: 3D trapezoid under shearing load

In this example, a 3D hyper-elastic block with 8 points is considered, as shown in Figure 6. Displacements in x and y directions of face ABCD are constrained while displacement in the z direction of face ADHE is fixed. A force of 1 N is applied on the face EFGH in the y direction (upward). Like the first problem, the compressible Neo-Hookean model is chosen for

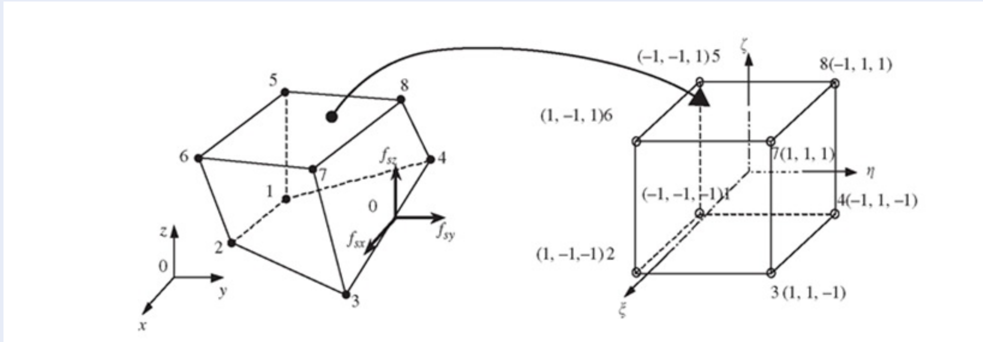

Figure 2: Generation of 8 – node volume element

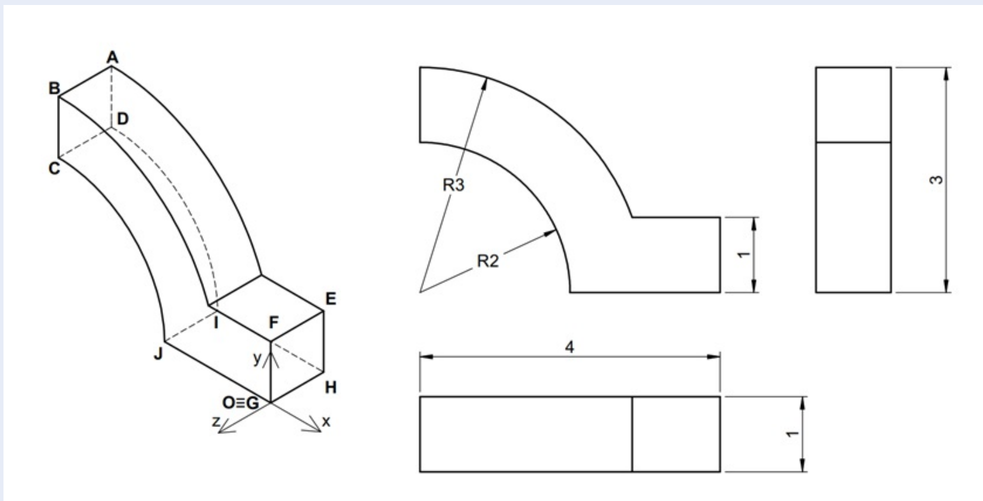

Figure 3: Geometry description for problem 1

Table 1: The displacements error between Matlab and Ansys

	u_x	u_y	u_{total}
Error (%)	1.82	4.69	2.65

this example with shear modulus $\mu = 1Pa$ and bulk modulus $\kappa = 10Pa$.

In this example, the 3D model has meshed with 726 nodes and 500 hexahedron elements. The mesh is shown in Figure 7 in the front view. The problem is solved by Matlab code and Ansys program with 30 steps. Figure 8 shows the displacement result of the model in the applied force direction. The result from the Ansys program is plotted on the left side, and the right side is this from Matlab software. Both solutions are very similar in deformation shape. The overall displacement error in y y-direction and total displacement on the whole model are 2.68% and 4.36%, respectively. Figure 9 shows the normal stress results in the applied force direction. The maximum of normal stress σ_y given by Matlab is $1.95Pa$, and this result of Ansys is $1.889Pa$, the exact of Matlab code is credible. Several meshes are chosen for the computations to investigate the convergence of the obtained results. Data in Table II shows the solutions of maximum vertical displacement and normal stress of the model concerning three sets of mesh density.

Problem 3: 3D curved beam under bending

The last problem deals with a 3D curved beam model under bending load, as displayed in Figure 10. To

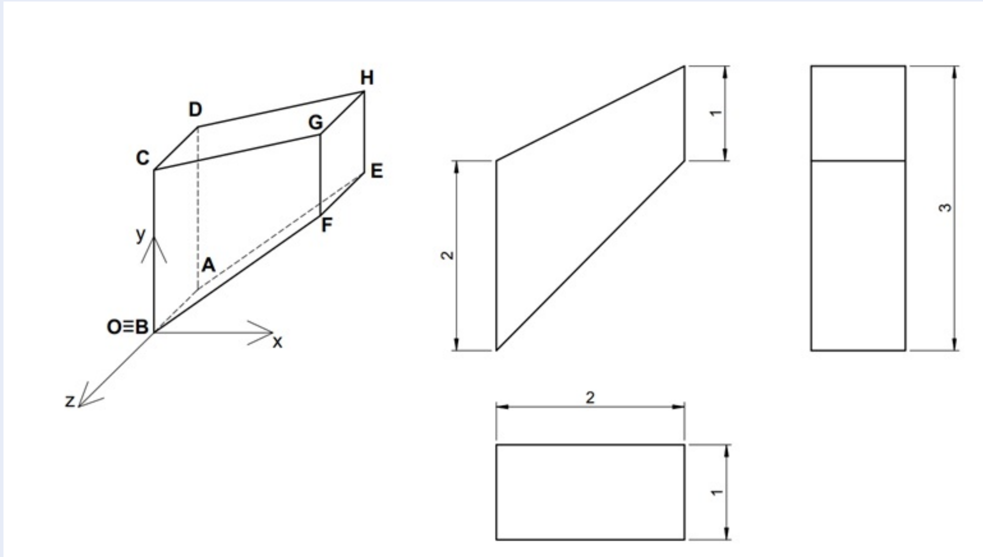

Figure 6: Geometry description for problem 2

Table 2: The convergence of vertical displacement and normal stress results

Number of nodes	u_y^{max}	σ_y^{max} (Pa)
726	2.10	1.95
1350	2.08	1.90
2646	2.07	1.93

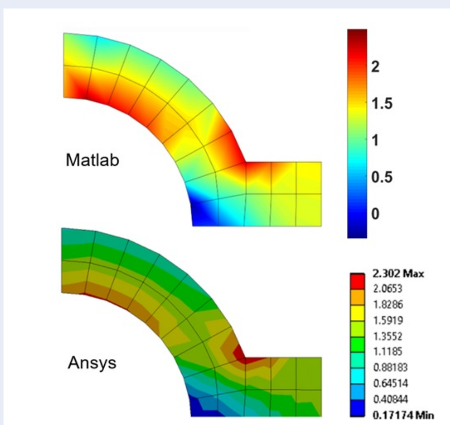

Figure 5: Normal stress in the x direction (Pa)

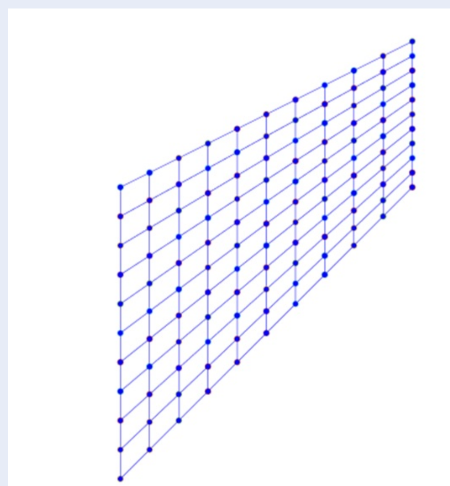

Figure 7: Element model of problem 2 (front view)

constrain the model, face ABCD is fixed. A compression force is applied on face EFGH; components of the force are -0.005 N along x-direction, -0.005 N long y-direction and 0 N along z-direction. Figure 11 shows mesh from the front view. There are 1525 nodes and 960 hexahedron elements are used for this simulation.

The hyper-elastic material properties are the same as the previous problem.

The problem is solved by Matlab code and Ansys program with 15 uniform load steps. The comparison of

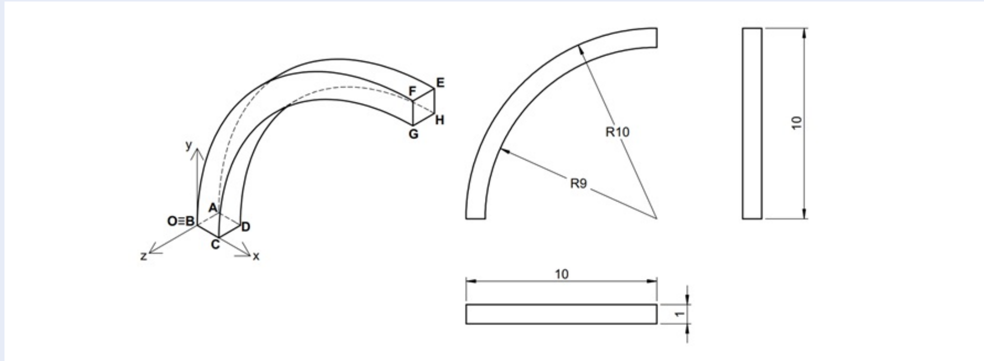

Figure 10: Geometry description for curved beam model

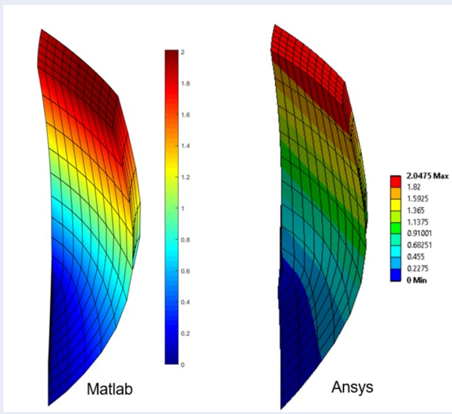

Figure 8: Deformation of the model, color indicates the displacement in y-direction

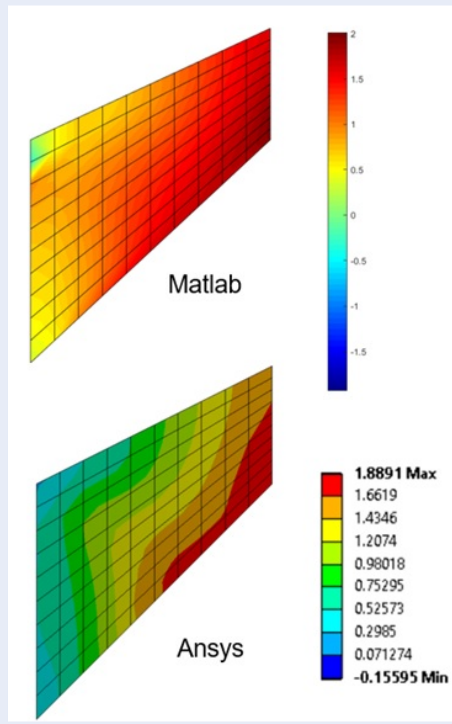

Figure 9: Normal stress in the y direction (Pa)

deformation result of the curved beam between Matlab code and Ansys program is shown in Figure 12; the color indicates the displacement in x direction. An acceptable result on deformed shape is achieved. The overall displacement error in y direction and total displacement on the whole model are 3.63% and 0.03%, respectively. The results obtained using the present 3D Matlab code match well with the references solution.

To verify the stress results, Figure 13 shows the comparison of normal stress in x-direction. The maximum value σ_x given by Matlab code is 0.29Pa, and this result of Ansys is 0.277Pa, the accuracy of Matlab code can be assured.

DISCUSSIONS

Phần biện luận quá ngắn Không đúng chuẩn một bản thảo khoa học Đề nghị tác giả bổ sung. Cả phần biện luận chỉ đúng có 2 câu

According to the numerical results of the three examples, the difference of displacement results between the present Matlab code and Ansys is very small. The obtained results of normal stress are also acceptable due to the errors are smaller than 5%. The deformations of 3D models are visualized to show the finite strain behavior of hyper-elastic solid. The convergence of the displacement and stress solutions are also investigated and the mesh independent of these solutions is insured. The built code is good enough for 3D non-linear finite element analysis.

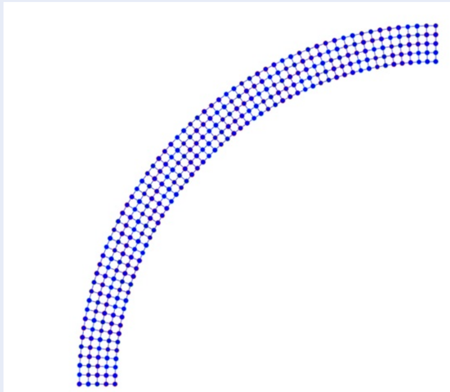

Figure 11: The mesh of curved beam model (front view)

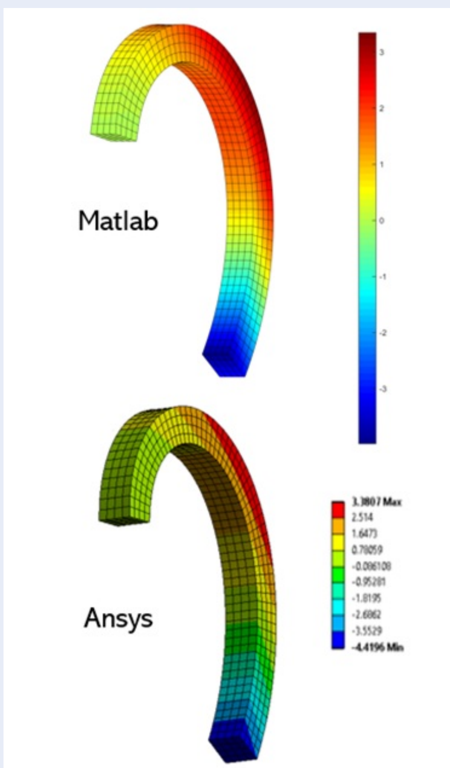

Figure 12: Deformation result of curved beam

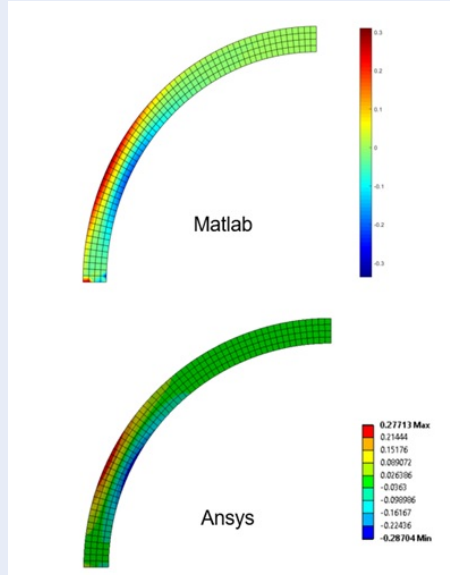

Figure 13: Distributions of normal stress in x-direction

CONCLUSION

The study presents three non-linear 3D problems of hyper-elastic materials with different models and boundary conditions to investigate the accuracy of the proposed 3D finite element approach. The obtained results are verified with the solution given by the commercial program. From the theories and algorithms in non-linear analysis for hyper-elastic materials based on the 3D finite element method, computational programs are built using Matlab languages to solve three-dimensional problems for hyperelastic bodies. Generally, all the results given by the Matlab program are good enough to be used in the analysis behavior of hyper-elastic bodies. However, it is necessary to continue improving the program to optimize the performance. The program also needs to be developed for more complicated problems such as 3D non-linear dynamic behavior of solid and 3D non-linear contact analysis.

ACKNOWLEDGMENT

This research is funded by the Ho Chi Minh City University of Technology – VNU-HCM, under grant number SVCQ–2020–KHUD–47. For this study, the authors acknowledge the support of time and facilities from Ho Chi Minh City University of Technology (HCMUT), VNU-HCM.

ABBREVIATIONS

FEM: Finite Element Method

3D: Three dimension

CONFLICT OF INTEREST

Group of authors declare that this manuscript is original, has not been published before and there is no conflict of interest in publishing the paper.

AUTHOR CONTRIBUTION

Trong Duc Nguyen works as the algorithm developer for the numerical program and the manuscript editor. Trong Tran Hieu Huynh plays the role of program developer and verify the solution.

Nha Thanh Nguyen and Hien Thi My Nguyen take part in the work of gathering data, checking the numerical results and manuscript.

Thien Tich Truong is the supervisor of the group, he also contributes ideas for the proposed method.

REFERENCES

1. Liu F, et al. The influence of rubber layer on the response of fluid-filled container due to high-velocity impact. Composite Structures. 2018;183:671–681. Available from: https://doi.org/10.1016/j.compstruct.2017.09.005.
2. Xue C, et al. Hyperelastic characteristics of graphene natural rubber composites and reinforcement and toughening mechanisms at multi-scale. Composite Structures. 2019;228:111365. Available from: https://doi.org/10.1016/j.compstruct.2019.111365.
3. Bairán JM, Cladera A. Collapse of a precast concrete beam for a light roof. Importance of elastomeric bearing pads in the element's stability. Engineering Failure Analysis. 2014;39:188–199. Available from: https://doi.org/10.1016/j.engfailanal.2014.02.001.
4. Ge X, et al. Effect of distributed support of rail pad on vertical vehicle-track interactions. Construction and Building Materials. 2020;262:120607. Available from: https://doi.org/10.1016/j.conbuildmat.2020.120607.
5. Kim NH. Introduction to Nonlinear Finite Element Analysis. Springer US, New York. 2015;.
6. Hassani R, Ansari R, Rouhi H. Large deformation analysis of 2D hyperelastic bodies based on the compressible non-linear elasticity: A numerical variational method. International Journal of Non-Linear Mechanics. 2019;116:39–54. Available from: https://doi.org/10.1016/j.ijnonlinmec.2019.05.003.
7. Peng F, et al. Phase field simulation for fracture behavior of hyperelastic material at large deformation based on edge-based smoothed finite element method. Engineering Fracture Mechanics. 2020;238:107233. Available from: https://doi.org/10.1016/j.engfracmech.2020.107233.
8. Almasi A, Baghani M, Moallemi A. Thermomechanical analysis of hyperelastic thick-walled cylindrical pressure vessels, analytical solutions and FEM. International Journal of Mechanical Sciences. 2017;130:426–436. Available from: https://doi.org/10.1016/j.ijmecsci.2017.06.033.
9. Hu G, Wriggers P. On the adaptive finite element method of steady-state rolling contact for hyperelasticity in finite deformations. Computer Methods in Applied Mechanics and Engineering. 2002;191(13):1333–1348. Available from: https://doi.org/10.1016/S0045-7825(01)00326-7.
10. Farzam DR, et al. Large deformation analysis of fully incompressible hyperelastic curved beams. Applied Mathematical Modelling. 2021;93:89–100. Available from: https://doi.org/10.1016/j.apm.2020.12.001.

Tạp chí Phát triển Khoa học và Công nghệ
Đại học Quốc gia Tp. Hồ Chí Minh

Tạp chí Phát triển Khoa học và Công nghệ
ISSN: 1859-0128
Hình thức xuất bản: In và trực tuyến
Hình thức truy cập: Truy cập mở (Open Access)
Ngôn ngữ bài báo: Tiếng Anh
Tỉ lệ chấp nhận đăng 2021: 72%
Phí xuất bản: Miễn phí
Thời gian phản biện: 43 ngày
Lập chỉ mục (Indexed): Google Scholar, Scilit
SCAN ME

Tạp chí Phát triển Khoa học và Công nghệ - Khoa học Tự nhiên
ISSN: 2588-106X
Hình thức xuất bản: In & trực tuyến
Hình thức truy cập: Truy cập mở
Ngôn ngữ bài báo: Tiếng Việt
Tỉ lệ chấp nhận đăng 2021: 75%
Phí xuất bản: Miễn phí
Thời gian phản biện: 30-45 ngày
Lập chỉ mục (Indexed): Google Scholar, Scilit
SCAN ME

Tạp chí Phát triển Khoa học và Công nghệ - Kĩ thuật và Công nghệ
ISSN: 2615-9872
Hình thức xuất bản: In & trực tuyến
Hình thức truy cập: Truy cập mở
Ngôn ngữ bài báo: Tiếng Việt
Tỉ lệ chấp nhận đăng 2021: 61%
Phí xuất bản: Miễn phí
Thời gian phản biện: 50 ngày
Lập chỉ mục (Indexed): Google Scholar, Scilit
SCAN ME

Tạp chí Phát triển Khoa học và Công nghệ - Kinh tế-Luật và Quản lý
ISSN: 2588-1051
Hình thức xuất bản: In & trực tuyến
Hình thức truy cập: Truy cập mở
Ngôn ngữ bài báo: Tiếng Việt
Tỉ lệ chấp nhận đăng 2021: 65%
Phí xuất bản: Miễn phí
Thời gian phản biện: 45 ngày
Lập chỉ mục (Indexed): Google Scholar, Scilit
SCAN ME

Tạp chí Phát triển Khoa học và Công nghệ - Khoa học Xã hội và Nhân văn
ISSN: 2588-1043
Hình thức xuất bản: In & trực tuyến
Hình thức truy cập: Truy cập mở
Ngôn ngữ bài báo: Tiếng Việt
Tỉ lệ chấp nhận đăng 2021: 62%
Phí xuất bản: Miễn thu phí đối với tác giả là CBVC của ĐHKHXHNV, ĐHQG-HCM; Tác giả khác: 500.000 VNĐ/bài
Thời gian phản biện: 75 ngày
Lập chỉ mục (Indexed): Google Scholar, Scilit
SCAN ME

Tạp chí Phát triển Khoa học và Công nghệ - Khoa học Trái đất và Môi trường
ISSN: **2588-1078**
Hình thức xuất bản: In & trực tuyến
Hình thức truy cập: Truy cập mở
Ngôn ngữ bài báo: Tiếng Việt và tiếng Anh
Tỉ lệ chấp nhận đăng 2021: 87%
Phí xuất bản: liên hệ tòa soạn
Thời gian phản biện: 45 ngày
Lập chỉ mục (Indexed): Google Scholar, Scilit
SCAN ME

Tạp chí Phát triển Khoa học và Công nghệ - Khoa học Sức khỏe
ISSN: 2734-9446
Hình thức xuất bản: In & trực tuyến
Hình thức truy cập: Truy cập mở
Ngôn ngữ bài báo: Tiếng Việt
Tỉ lệ chấp nhận đăng 2021: 70%
Phí xuất bản: Miễn phí
Thời gian phản biện: 30 ngày
Lập chỉ mục (Indexed): Google Scholar, Scilit
SCAN ME

Tạp chí Phát triển Khoa học và Công nghệ, Đại học Quốc gia Tp.HCM
25 năm xuất bản học thuật (1997-2022)
Tòa soạn: Nhà điều hành Đại học Quốc gia Tp.HCM, P. Linh Trung, TP. Thủ Đức, TP. HCM
Email: stdj@vnuhcm.edu.vn; tcptkhcn@vnuhcm.edu.vn; Website: http://www.scienceandtechnology.com.vn