TESTING SAFETY FACTOR OF FLEXIBLE GEAR RIM IN HARMONIC GEAR DRIVE BY THE FINITE ELEMENT METHOD

Le Thi Thu Thuy*

Abstract: *In wave transmission, the flexible gear (flex-spline) is a complex stage that has a decisive effect on the working capacity and durability of the transmission. In the absence of finite element software, the Russians came up with formulas to determine the factor of safety for flexible gears. The question is how this coefficient will change when using finite element software. This paper has the purpose of determining the external force acting on the flexible gear rim by performing an estimate to calculate the external force, then using this mechanical and external force to specify the safety factor of the flexible gear on the finite element software. The insignificant differences between the traditional calculation and the modern calculation suggest the reliability of both. Nowadays, with the wide applications of industrial robots, the harmonic drive is very common, in which the problem mentioned here is one of the key points of design.*

Keywords: Harmonic drive; Flex-spline; FEM, External force; Transposition; Safety factor.

1. INTRODUCTION

Strain wave gear systems with different textures will behave differently. Here shows the shear wave forming principle of the simplest reducer: single-stage, two-wave type [1, 2, 5].

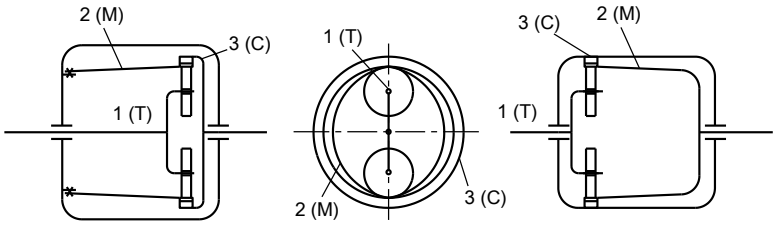


Figure 1. *Schematic of single-stage, two-wave drive.*

Basically a single-stage harmonic drive consists of the following components:

- Circular spline;
- Flexible gear (flex-spline);
- Wave generator;

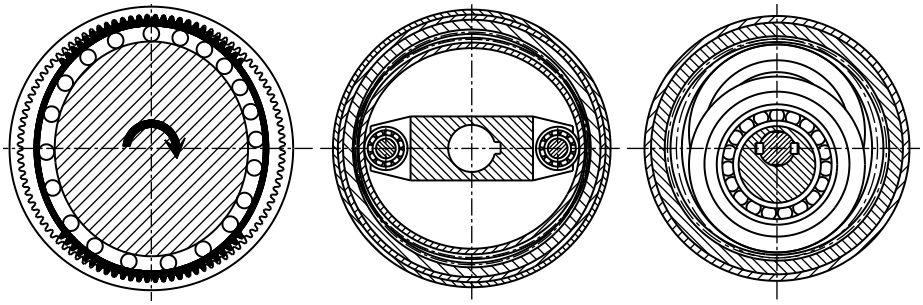


Figure 2. *Some popular wave generators based on mechanical principles.*

a. Flexible ball bearings and cam, b. Hard ball bearings and link, c. Eccentric cam and hard ball bearings

The wave generator consists of two rollers symmetrically arranged on the link to deform the flex spline. The Flex spline has a form of a cylindrical cup attached to the motor shaft (or fixed to the box cover). The bottom of the cup is thick and rigid but the side is very thin and has

external teeth. As the flex spline works, the contact points between it and the rollers are constantly changing. The circular spline is an internal gear that is fixed (or attached to the speed reducer output shaft). With the elastic deformation state, the flex spline wrapped on the outside of the wave generator creates two symmetrical meshing regions with the entire tooth height. Conversely, in the smallest deformed regions, teeth completely separate and do not mesh.

Due to the working characteristics, the flexible gear has different load-bearing areas. This leads to the difficult calculation of the safety factor, such as the bearing condition of the gear rim in figure 3.

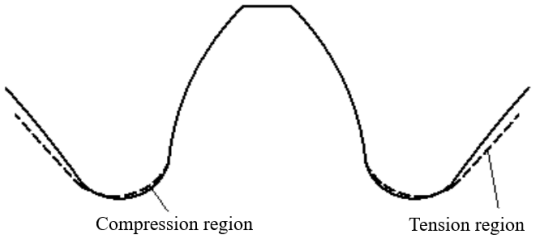


Figure 3. *load-bearing situation on the flexible gear rim when working.*

According to documents [2, 4, 5], the mathematical models that allow the determination of the safety factor have been fully showed. Retesting these indicators on a specific situation is what will be addressed in the next sections of this article.

2. MODEL OF DETERMINING EXTERNAL FORCES AFFECTING ON FLEXIBLE GEAR RIM

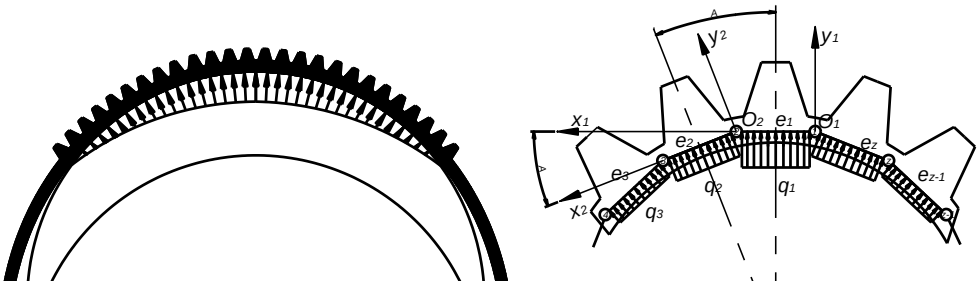


Figure 4. *Assumptions about the external force distribution rule on the gear rim.*

The finite element method (FEM) is based on dividing the continuous geometric model into discrete elements and replacing them with displacement functions. The geometry model is replaced with the node model [9, 10, 13]. The characteristics of the system including the node displacement, the node force and the stiffness (excluding the derivatives of displacement) are described in a relationship by a system of algebraic equations. When described in the form of a matrix, the relationship has the form:

$$\left[\overline{\overline{K}}\right].[\delta]=[Q] \tag{1}$$

Where:

$\left[\overline{\overline{K}}\right]$: Hardness matrix of the system of elements;

$[\delta]$: Displacement matrix of all nodes in the model;

$[Q]$: The node force matrix of all the nodes in the model.

Flexible gears in the bearing state can be assumed that most of the strain falls into the tooth root since that is where the material concentration is lowest compared to other points. According to FEM, a tooth can be considered as an element. Ignore the difference in the two-headed distribution force to consider that each element is subjected to an evenly distributed force. At the node between the two elements, there is a jump of distributed force as shown in figure 4.

With the mechanical parameters of the material and the geometric correlation of the elements on the flex spline rim after deformation, there is enough basis to build the hardness matrix of the element system.

After the flex-spline is fitted with the wave generator, the shape of the elastic line can be determined using a three-dimensional coordinate measuring machine (CMM) (figure 5). Based on the assumptions about the distribution of the elements above, it is possible to determine the matrix of displacement elements compared with before deformation, which is the matrix $[\delta]$ in equation (1).

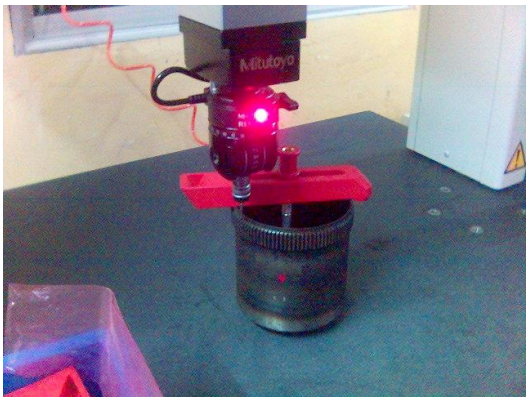


Figure 5. *Determining node displacement matrix by CMM machine at laboratory TNUT.*

When using the wave generator with a disk form, the entire flexible tooth rim model has two symmetrical axes, so it is not necessary to consider them all. But it is necessary to model how the part in question shows the links with the rest. The following model is given for consideration:

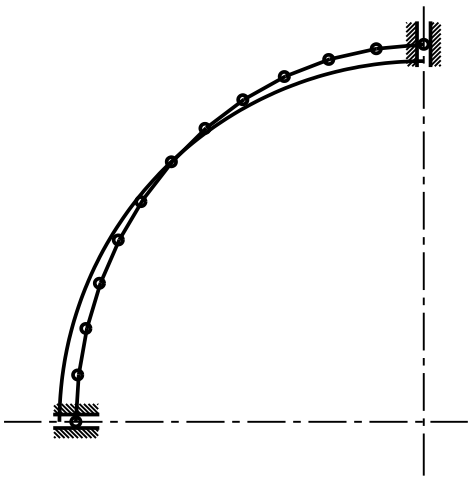


Figure 6. *Flex-spline model with two symmetrical axes.*

To show the alignment with the rest while the wave generator meshes the flex-spline rim, the first and last element of the model has only vertical and horizontal displacements. These two elements are described as a revolute joint and a translational joint with the translational direction coinciding with the direction of the symmetry axis. This generates a constraining force at the node with the removed part.

The rest of the elements are shown as having two hinged ends. These hinge joints are in space so each node has three remaining displacements: vertical, horizontal, and angular displacement of the bar.

The stiffness matrix of the system in the reference frame is established as follows:

$\mathbf{K}_{11}^{1}$	$\mathbf{K}_{12}^{1}$				
$\mathbf{K}_{21}^{1}$	$\mathbf{K}_{22}^{1}+\mathbf{K}_{22}^{2}$	$\mathbf{K}_{23}^{2}$			
	$\mathbf{K}_{32}^{2}$	$\mathbf{K}_{33}^{2}+\mathbf{K}_{33}^{3}$			
			$\ddots$		
				$\mathbf{K}_{n,n}^{n-1}+\mathbf{K}_{n,n}^{n}$	$\mathbf{K}_{n,n+1}^{n}$
				$\mathbf{K}_{n+1,n}^{n}$	$\mathbf{K}_{n+1,n+1}^{n}$

(2)

At this point, it is qualified to determine the external force acting on the soft tooth rim according to the model (1).

3. COMPARING TRADITIONAL MODEL AND FEM

Considering the study object is flex-spline z = 98 teeth, m = 1;

**** Test according to the traditional model*** [14, 15, 16]:

The gear rim of flex-spline is made of material 40XC, hardness HB = 300, M_{Me} = 120 (kg.m); i = 50. The flex-spline is active with L = d_{cC}; q_{Me} = 0,15; [p] = 600 (kg/cm^2); pressure angle $\alpha_{\partial} = 20^0$. Combining the geometry data of the rim has been determined when calculating the kinematic: D_{cC} = 100 (mm); m = 1 (mm); m_y = 0.9615 (mm); ◆=1; $\xi = 2.165$; r_{dt} = 47.13 (mm).

Determine the minimum normal stress acting on the bottom land of tooth of the flex-spline: $\sigma_z = \dfrac{\delta E}{m_y z_{Me}(0{,}1366\dfrac{z_{Me}}{k_\Delta}-1)} = \dfrac{0{,}3.2{,}1.10^6}{0{,}9615.98(0{,}1366\dfrac{98}{0{,}8197}-1)} = 427(kg/cm^2)$.

Value of $\lambda^{'}$ determined by the graph is 2,75.

$$\sigma_{\max} = \sigma_z \lambda^{'} = 427.2{,}75 = 1174{,}25(kg/cm^2)$$

$$\sigma_{\min} = -\frac{\delta E \alpha^{'}_{\sigma}}{z_{Me} m_y (0{,}1366\dfrac{z_{Me}}{k_\Delta}-1)} = -\frac{0{,}3.2{,}1.10^6.1{,}2}{98.0{,}9615(0{,}1366\dfrac{98}{0{,}8197}-1)} = -512(kg/cm^2)$$

The value of torsional stress in the bottom land of the flexible gear is determined by the formula: $\tau_{\max} = \frac{2M_{Me}}{\pi d^2_{cMe}\delta} = \frac{2.12000}{3,14.9,8^2.0,3} = 265,2(kg/cm^2)$.

Frequency and cyclic stress is calculated as:

$$\sigma_a = \frac{1}{2}(\sigma_{\max} - \sigma_{\min}) = \frac{1}{2}(1174,25 + 512) = 843,125(kg/cm^2)$$

$$\sigma_m = \frac{1}{2}(\sigma_{\max} + \sigma_{\min}) = \frac{1}{2}(1174,25\text{-}512) = 331,125(\text{kg/cm}^2)$$

$$\tau_a = \tau_m = \frac{1}{2}\tau_{\max} = \frac{265,2}{2} = 132,6(kg/cm^2)$$

The safety factor by normal stress is determined by the following formula:

(With material 40XC, $\sigma_{-1} = 4600(kg/cm^2); \tau_{-1} = 2700(kg/cm^2)$)

$$n_\sigma = \frac{\sigma_{-1}}{\frac{\sigma_a}{\varepsilon_\sigma \beta_\sigma} + \sigma_m \psi_\sigma} = \frac{4600}{\frac{843,125}{0,6.0,9} + 331,125.0,1} = 4,885$$

$$n_\tau = \frac{\tau_{-1}}{0,5\tau_{\max}(\frac{\alpha'_\tau}{\varepsilon\beta_\tau} + \psi_\tau)} = \frac{2700}{0,5.265,2(\frac{1,95}{0,68.0,9} + 0,05)} = 6,292$$

So the total safety factor will be equal to:

$$n = \frac{n_\sigma n_\tau}{\sqrt{n_\sigma^2 + 0,7n_\tau^2}} = \frac{4,885.6,292}{\sqrt{4,485^2 + 0,7.6,292^2}} = 4,279$$

This safety factor is greater than 1.5, it satisfies the requirements of working durability and the calculation is finished.

**** Test with Cosmos Design Star*** [11-13]:

To do the test we need to build a 3D model of the rim on the AutoCad Mechanical Destop, export the file as (*.iges), or (*.igs), import the model into the Cosmos Design Star environment, and do the following steps:

- Determine material grades through physical and mechanical parameters.

- Applying force based on result of the node force.

- Control the displacement as when working.

- Discrete the continuous model into FEM model.

- Run an interactive solving program.

- Choose test standards.

- Result evaluation.

The software allows to check the safety factor according to three basic criteria:

- *Mohr – Coulomb stress* FOS = 4.4 FOS(factor of Safey)

Test formula: $\frac{\sigma_1}{\sigma_{TensiveLimit}} + \frac{\sigma_2}{\sigma_{CompressiveLimit}} < 1$

- *Maximum Von Mises stress* FOS = 4.4

Test formula: $\frac{\sigma_{Vonmises}}{\sigma_{Limit}} < 1$

- *Maximum shear stress (Tresca)* FOS = 4.4

Test formula: $\frac{\tau_{\max}}{0,5\sigma_{Limit}} < 1$

The test results here for the safety factor are all (4,4) - quite close to the calculation result according to the traditional model (4,279). The specific images of the results are as shown in figure 7:

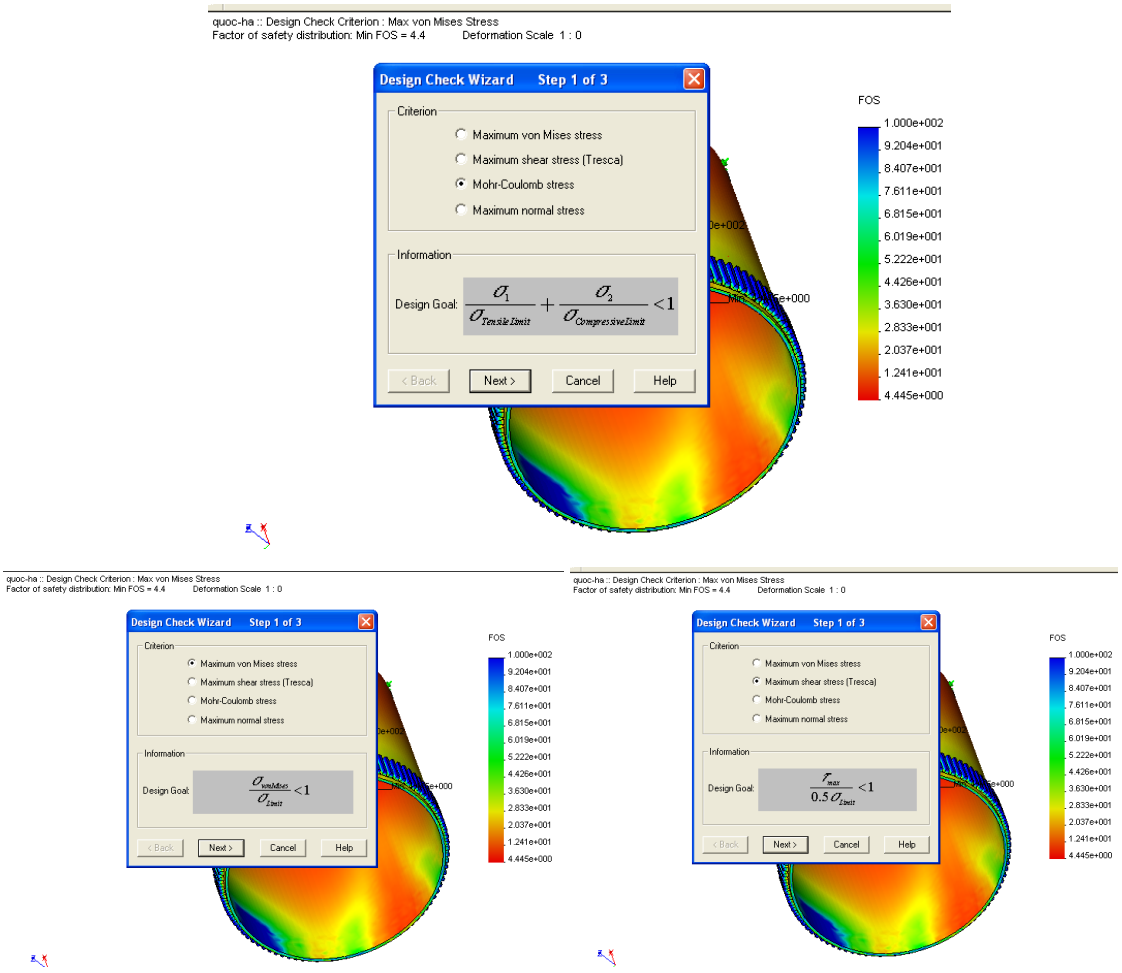


Figure 7. *Durable safety factor according to Cosmos Design Star with different standards.*

4. CONCLUSIONS

Through the verification process as done above, it can be seen that the flex-spline has been designed to meet the requirements of durability: while this coefficient in the flex-spline only needs to be greater than 1.5, it is over 4 here. In addition, the stability factor calculated according to the traditional model is quite close to the finite element model used by the author, which shows the reliability of the calculation method. In fact, the product has been loaded and tested at Diezel Song Cong Factory and meets the requirements for durability.

Acknowledgement: *Thank you to Thai Nguyen University of Technology (TNUT) of Vietnam for sponsoring this research.*

REFERENCES

[1]. Wallace Bentley, "*Sketches of Engine and Machine Details: A Reference Book for the Use of Engineers, Draughtsmen, and Mechanics, and a Text Book for Machine Construction and Drawing Classes*", Forgotten Books, ISBN13 9781330582923 (2018).

[2]. Yague-Spaude Eloy, Gonzalez-Perez Ignacio, Fuentes-Aznar Alfonso, "*Stress analysis of strain wave gear drives with four different geometries of wave generator*", Meccanica, Springer Netherlands, DOI: 10.1007/s11012-020-01248-5 (2020).

[3]. F. C. Park and K. M. Lynch, "*Introduction to robotics: mechanics, planning, and control*", Springer, ISSN: 05849667, (2014).

[4]. Paweł Weroński, Yi Jiang, Steen Rasmussen, "*Application of molecular dynamics computer simulations in the design of a minimal self-replicating molecular machine*", Complexity, John Wiley and Sons, Volume 13, Issue 4, DOI: 10.1002/cplx.20212 (2008).

[5]. Li Shuting, "*Diaphragm stress analysis and fatigue strength evaluation of the flex-spline, a very thin-walled spur gear used in the strain wave gearing*", Mechanism and Machine Theory Volume: 104 Issues: 1-16, Elsevier Science, DOI: 10.1016/j.mechmachtheory.2016.05.020, (2016).

[6]. Suk-Hwan Suh, Seong-Kyoon Kang, Dae-Hyuk Chung, Ian Stroud, "*Theory and Design of CNC Systems*", Springer, DOI 10.1007/978-1-84800-336-1 (2008).

[7]. Yongshun Zhao; Xingquan Zhang; Xianbing Liu; Kazuo Yamazaki, "*Geometric modeling of the linear motor driven electrical discharge machining (EDM) die-sinking process*", International Journal of Machine Tools and Manufacture, Elsevier Science, DOI: 10.1016/j.ijmachtools.2003.09.003 (2004).

[8]. Narayanan, G. Sankara, Vasudevan, Durairaj, "*A Mathematical Model for Wire Cut Electrical Discharge Machine Parameters Using Artificial Neural Network*", Advanced Materials Research, DOI: 10.4028/www.scientific.net/AMR.984-985.9 (2014).

[9]. Nguyễn văn Phái, Vũ văn Khiêm, "*Phương pháp phần tử hữu hạn thực hành trong cơ học*", Nhà xuất bản Giáo Dục, (2001).

[10]. T. Strouboulis, K. Copps, I. Babuška, "*The generalized finite element method: an example of its implementation and illustration of its performance*", International Journal for Numerical Methods in Engineering, Volume 47, Issue 8, DOI: 10.1002/(sici)1097-0207(20000320)47:8<1401::aid-nme835>3.0.co;2-8 (2000).

[11]. Nguyễn Văn Phái, "*Giải bài toán cơ kỹ thuật bằng chương trình Ansys*", NXB khoa học và kĩ thuật, (2003).

[12]. Đinh Bá Trụ, "*Hướng dẫn sử dụng Ansys, chương trình phần mềm thiết kế mô phỏng bằng phương pháp phần tử hữu hạn*", NXB khoa học và kĩ thuật, (2000).

[13]. "*Ansys và mô phỏng số trong công nghiệp bằng phần tử hữu hạn*" - NXB khoa học và kĩ thuật, (2003).

[14]. G. Pintaude; F.G. Bernardes; M.M. Santos; A. Sinatora; E. Albertin, "*Mild and severe wear of steels and cast irons in sliding abrasion*", Wear, Elsevier Science, Volume 267, Issue 1-4, DOI: 10.1016/j.wear.2008.12.099 (2009).

[15]. Heng Liang Zhang; Z. J. Zhang, "*On-line calculation models of temperature and thermal stresses in a 2-D axis-symmetric object during heat treating*", Heat Transfer - Asian Research, John Wiley and Sons, Volume 40, Issue 6, DOI: 10.1002/htj.20359 (2011).

[16]. S. P. Srivastava, "*Developments in Lubricant Technology (Srivastava/Developments in Lubricant Technology): Lubrication, Friction, and Wear*", Volume 10.1002/9781118907948, DOI: 10.1002/9781118907948.ch6 (2014).

TÓM TẮT

KIỂM NGHIỆM HỆ SỐ AN TOÀN VÀNH BÁNH RĂNG MỀM TRONG BỘ TRUYỀN SÓNG BẰNG PHƯƠNG PHÁP PHẦN TỬ HỮU HẠN

Trong truyền động sóng, bánh răng mềm là khâu chịu lực phức tạp có ảnh hưởng quyết định đến khả năng làm việc và tuổi bền của bộ truyền. Khi chưa có các phần mềm phần tử hữu hạn, người Nga đã đưa ra các công thức để xác định hệ số an toàn của bánh răng mềm. Câu hỏi đặt ra ở đây là hệ số này sẽ thay đổi như thế nào khi ứng dụng các phần mềm theo phương pháp phần tử hữu hạn. Bài báo này có mục đích xác định ngoại lực tác dụng lên vành bánh răng mềm bằng cách thực hiện một phép ước lượng để tính toán ngoại lực, từ đó sử dụng cơ tính và ngoại lực này để định rõ hệ số an toàn của bánh răng mềm trên phần mềm phần tử hữu hạn. Các chênh lệch không đáng kể giữa cách tính truyền thống và cách tính hiện đại cho thấy sự tin cậy của cả hai cách làm. Ngày nay, với các ứng dụng rộng rãi của robot công nghiệp, sự trở lại của truyền động sóng là rất phổ biến, trong đó, bài toán đề cập ở đây là một trong những mấu chốt quan trọng khi thiết kế.

Từ khóa: Bộ truyền bánh răng sóng; Bánh răng mềm; FEM; Ngoại lực; Chuyển vị; Hệ số an toàn.

Received date, 15th March, 2021
Revised manuscript, 09h August, 2021
Published, 05th September, 2021

Author affiliations:
Thai Nguyen University of Technology.
*Corresponding author: hanthuyngoc@tnut.edu.vn.