International Journal of Engineering Inventions
e-ISSN: 2278-7461, p-ISSN: 2319-6491
Volume 13, Issue 5 [May. 2024] PP: 199-203

Analyze the effect of mesh size in the finite element model on the resonant frequency of the ultrasonic vibration device

Ha Bach Tu[1], Hoang Anh Quang[1*]
[1] *Faculty of Mechanical Engineering, Thai Nguyen University of Technology, Thai Nguyen, 250000, Vietnam.*

ABSTRACT: *Nowadays, ultrasonic vibration devices have been widely used in life such as in industry, medicine, and construction. To optimally design these structures, finite element analysis was used in the design process. The transducer is the most important device in the ultrasonic vibration system. To optimally design these structures, finite element analysis was used in the design process. In this paper, a study on the effect of grid size on the resonant frequency of the transducer is carried out. The results show that the mesh size of the element in the FEM model has a clear influence on the resonant frequency of the transducer.*
KEYWORDS: mesh size, simulation, transducer, resonant frequency, amplitude

---

Date of Submission: 09-05-2024 Date of acceptance: 21-05-2024

I. INTRODUCTION

Ultrasonic vibration is a vibration with a high frequency above 20 kHz. The applications of ultrasonic vibrations are divided into two main groups: The groups with small amplitude and high frequency and those with high amplitude and low frequency. Ultrasonic vibrations with small amplitudes usually have an energy density ranging from 0.1 to 1 W/cm2, and the frequency is usually several MHz or more. This type of ultrasound is often used in medical imaging and non-destructive examination of defects. There have been many domestic research projects in this field, such as research in medicine [1], research on seafood detection using ultrasound [2], and assessment of the uniformity of concrete in construction [3]. High-amplitude ultrasonic vibrations are often used in industrial production to permanently change the affected object's physical, chemical, or biological properties. Power intensity is usually from a few tens to several hundred W or several kW, frequency from 20 to several hundred kHz, even up to MHz as in cleaning and chemical treatment.

In recent years, ultrasonic vibrations have been used more and more commonly in many industrial production sectors, such as mechanical cutting and machining; ultrasonic cleaning; ultrasonic welding; lubricating, reduced friction; car manufacturing; and food processing. Ultrasonic vibration technology to assist in the machining and cutting process has shown many outstanding advantages, such as reducing cutting force, increasing tool durability, improving machined surface quality, and cutting many types. materials that are difficult to machine. The design process of ultrasonic vibration equipment has been using the finite element method. In the study of R. MahdaVinejad [4], the converter was modeled using the finite element method to evaluate the theoretical calculation results. Through this, it is confirmed that the finite element model and experimental testing are very close to each other, so this method is considered a suitable basis for the design of the converter.

In the ultrasonic field, FEA is used to predict the vibration characteristics of a horn and its ultrasonic assembly before it is manufactured [5]. Modal analysis is often the first step of analysis. When identifying individual vibration modes, will provide information about the limits of frequency used to work. It is extremely important in terms of time and cost of designing, manufacturing, and using the horn. Iulian Stănăşel et al. used integrated CAD/CAE application and analysis relationships to design a horn for use in an ultrasonic installation for welding copper wire cables. FEM analysis aims to analyze and identify its vibration modes, thereby providing information about the use of frequencies to work with positive implications in terms of time and cost of horn design. In another study, Mathieson Andrew succeeded in modifying the design parameters to create an ultrasound device suitable for use in bone surgery [6] based on the design of an existing Langevin probe. In addition, other studies [7–10] also focus on designing ultrasound probes for different purposes.

Finite element analysis has been applied in calculating the ultrasonic transducer design. However, building finite element models still has many issues that need attention and research. In this study, a finite element model of an ultrasonic vibrating head was built to simulate the operation of an ultrasonic transducer. Including the influence of mesh size on the resonant frequency results.

II. THEORETICAL ANALYSIS

The research object in this study is an ultrasound probe with the structure shown in Figure 1 [11, 12]. The resonant frequency in working mode is 20 kHz, and the longitudinal oscillation amplitude is 10 µm. Component dimensions and material specifications are given in Tables 1 and Table 2.

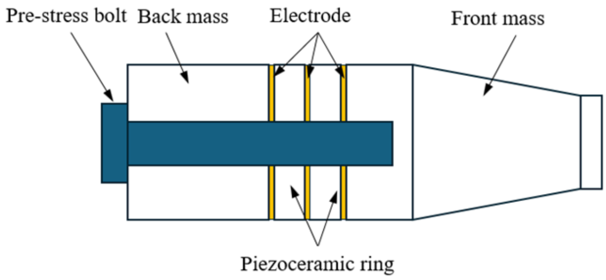

Fig. 1 Experimental setup scheme

Table 1. The dimensions of ultrasonic transducer parts

Component	External diameter (mm)	Bore size (mm)	Length (mm)	Material
Back mass	38	12	36	Steel
Piezoelectric ceramic ring	38	16	8	PZT4
Pre-stress bolt	12		60	Titanium
Electrode sheet	38	16	0.5	Copper
Front mass (Cylindrical segment 1)	38	12	15	Titanium
Front mass (Conical segment)			45	Titanium
Front mass (Cylindrical segment 2)	20		10	Titanium

Table 2. Material parameters

Material	Density ($kg.m^{-3}$)	Elastic modulus (GPa)	Poisson ratio
PZT-4	7600	65	0.31
Copper	8900	115	0.31
Titanium	4418	110	0.34
Steel	7800	209	0.3

In the structure of the transducer, the ceramic ring plays a very important role in converting electrical energy into mechanical energy. Therefore, the organic element model needs to provide complete and accurate parameter values of this material. Linear electrical behavior of ceramic materials.

$D = \varepsilon E \Rightarrow D_i = \varepsilon_{ij} E_j$ (1)

Where, D is the electric charge density displacement, ε is permittivity, E is electric field strength. The Hooke's law for elastic materials:

$S = sT \Rightarrow S_{ij} = s_{ijkl} T_{kl}$ (2)

Where, S is strain, s is compliance under short-circuit conditions T is stress. These relations may be combined into so-called coupled equations, of which the strain-charge form is:

$$\begin{aligned} S &= sT + d^t E \\ D &= dT + \varepsilon E \end{aligned} \quad (3)$$

Where, d is matrix for the direct piezoelectric effect. The piezoelectric coefficients can be defined as follows

$$d_{ij} = \left(\frac{\partial D_i}{\partial T_j}\right)^E = \left(\frac{\partial S_i}{\partial E_j}\right)^T ; \; e_{ij} = \left(\frac{\partial D_i}{\partial S_j}\right)^E = -\left(\frac{\partial T_j}{\partial E_i}\right)^T$$

$$g_{ij} = -\left(\frac{\partial E_i}{\partial T_j}\right)^D = \left(\frac{\partial S_j}{\partial D_i}\right)^T ; \; h_{ij} = \left(\frac{\partial E_i}{\partial S_j}\right)^D = -\left(\frac{\partial T_j}{\partial D_i}\right)^S \quad (4)$$

Relationships between electrical and elastic mechanical behavior

Strain – Charge Form: $S = s_E T + d^t E; D = d.T + \varepsilon_T E$ (5)

Strain – Charge Form: $T = c_E S - e^t E; D = e.S + \varepsilon_s E$ (6)

Strain-Voltage Form: $S = s_D T + g^t D; E = -g.T + \varepsilon_T^{-1} D$ (7)

Strain-Voltage Form: $T = c_D S - q^t D; E = -q.S + \varepsilon_S^{-1} D$ (8)

The linear relationship between the mechanical and electrical fields in the matrix is provided in Abaqus

$$[d] = \begin{bmatrix} d_{1\,11} & d_{1\,22} & d_{1\,33} & d_{1\,12} & d_{1\,13} & d_{1\,14} \\ d_{2\,11} & d_{2\,22} & d_{2\,33} & d_{2\,12} & d_{2\,13} & d_{2\,14} \\ d_{3\,11} & d_{3\,22} & d_{3\,33} & d_{3\,12} & d_{3\,13} & d_{3\,14} \end{bmatrix} (9)$$

$$[e] = \begin{bmatrix} e_{1\,11} & e_{1\,22} & e_{1\,33} & e_{1\,12} & e_{1\,13} & e_{1\,14} \\ e_{2\,11} & e_{2\,22} & e_{2\,33} & e_{2\,12} & e_{2\,13} & e_{2\,14} \\ e_{3\,11} & e_{3\,22} & e_{3\,33} & e_{3\,12} & e_{3\,13} & e_{3\,14} \end{bmatrix} (10)$$

where $d_{1\,23}$, $d_{2\,13}$, $d_{3\,11}$, $d_{3\,22}$, and $d_{3\,33}$ are provided by the manufacturer. Besides, the constantpiezoelectric charge [e] is also calculated according to the formula 4 [15].

$$[e] = [d][D] (11)$$

With D_{ijkl} is a fourth-order elastic tensor of the elastic stiffness parameter evaluated at a constant electric field

$$[D] = \begin{bmatrix} D_{1111} & D_{1122} & D_{1133} & 0 & 0 & 0 \\ & D_{2222} & D_{2233} & 0 & 0 & 0 \\ & & D_{3333} & 0 & 0 & 0 \\ & & & D_{1212} & 0 & 0 \\ & \text{symmetric} & & & D_{1313} & 0 \\ & & & & & D_{2323} \end{bmatrix} \quad (13)$$

III. NUMERICAL SIMULATION ANALYSIS

The finite element model is built in Abaqus software as shown in Figure 2. As mentioned above, the PZT4 ceramic material parameters are shown in Table 1. Meshing and using the appropriate element type can greatly affects the results as well as CPU time in numerical simulation. A small mesh will lead to a more accurate solution. However, when the mesh is made smaller, the calculation time increases and requires a large computer with a large configuration. Therefore, choosing the size and type of element is very important.

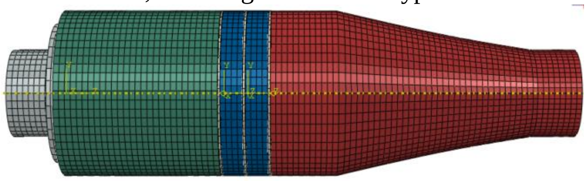

Figure 2. FEM model of the transducer

Table 3. Piezoelectric elastic properties of material

D_{1111}	D_{1122}	D_{1133}	D_{2233}	D_{3333}
9.22E+10	-6.75E+09	2.58E+10	2.58E+10	8.59E+10
D_{1212}	D_{1313}	D_{2323}	D_{2222}	
4.65E+10	4.65E+10	4.65E+10	9.34E+10	

Table 4. Charge constants matrix in the strain of material

$d_{1\,11}$	$d_{1\,22}$	$d_{1\,33}$	$d_{1\,12}$	$d_{1\,13}$	$d_{1\,23}$	$d_{2\,11}$	$d_{2\,22}$	$d_{2\,33}$
0.0E+00	0.0E+00	0.0E+00	0.0E+00	0.0E+00	3.65E-10	0.0E+00	0.0E+00	0.0E+00
$d_{2\,12}$	$d_{2\,13}$	$d_{2\,23}$	$d_{3\,11}$	$d_{3\,22}$	$d_{3\,33}$	$d_{3\,12}$	$d_{3\,13}$	$d_{3\,23}$
0.0E+00	3.65E-10	0.0E+00	-9.5E-11	-9.5E-11	2.35E-10	0.0E+00	0.0E+00	0.0E+00

Table 5. Piezoelectric charge of material

$e_{1\,11}$	$e_{1\,22}$	$e_{1\,33}$	$e_{1\,12}$	$e_{1\,13}$	$e_{1\,23}$	$e_{2\,11}$	$e_{2\,22}$	$e_{2\,33}$
0.0E+00	0.0E+00	0.0E+00	0.0E+00	0.0E+00	1.67E+01	0.0E+00	0.0E+00	0.0E+00
$e_{2\,12}$	$e_{2\,13}$	$e_{2\,23}$	$e_{3\,11}$	$e_{3\,22}$	$e_{3\,33}$	$e_{3\,12}$	$e_{3\,13}$	$e_{3\,23}$
0.0E+00	1.67e+01	0.0E+00	- 1.97E+00	- 1.97E+00	1.570E+01	0.0E+00	0.0E+00	0.0E+00

In Abaqus, the 8-node element (C3D8R) is commonly used for conventional materials. The C3D8E element has the same mechanical properties as the C3D8R element but has additional piezoelectric properties so it is used for ceramic materials. The mesh sizes selected for the model are 1 mm, 1.5 mm, 2 mm, 2.5 mm, and 3 mm respectively.

A series of numerical simulations (Abaqus/Frequency) were performed to determine the resonance frequency and vibration mode of the ultrasonic transducer. The obtained results are shown in Figure 3

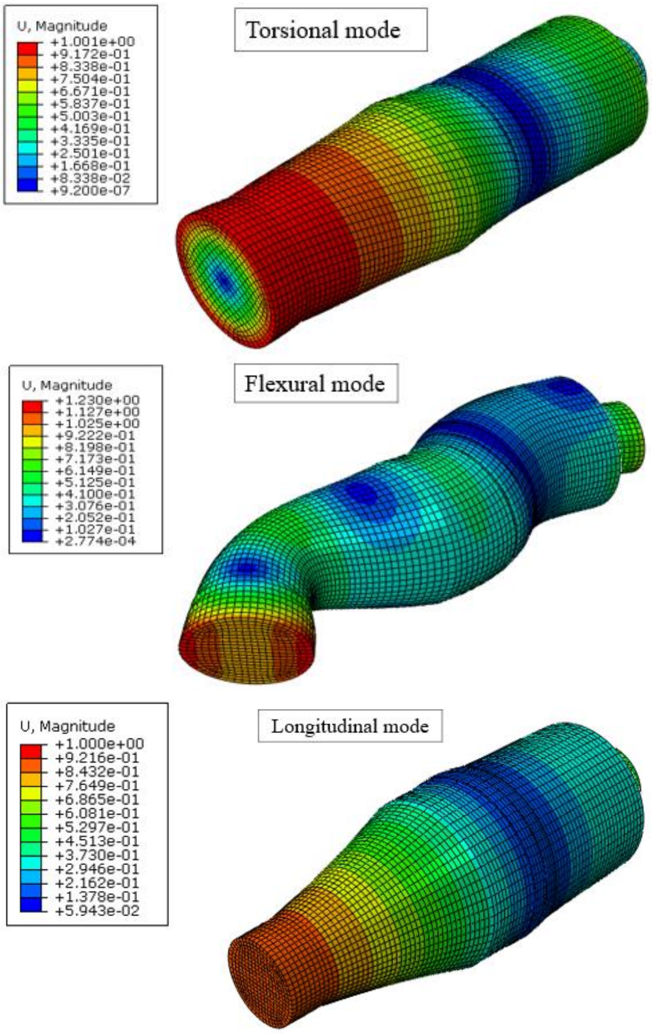

Figure 3. Modal analysis of the ultrasonic transducer (mesh size 1.5 mm)

Table 6. Compare the mode and natural frequency of the transducer with different mesh sizes

Mesh size (mm)	Natural frequency (Hz)			Time CPU (s)
	Mode 1	Mode 2	Mode 3	
1	16657	17482	19989	2128
1.5	16862	18388	20242	1899
2	17216	19253	21783	1382
2.5	18328	20056	23178	953
3	21717	23748	25278	667

Table 6 shows the resonant frequency values corresponding to different mesh sizes. Also shows the calculation time at different mesh sizes. The results show that the mesh size has a great influence on the resonance frequency. Comparing the calculation time, it is easy to see that the smaller the grid size, the larger the calculation time.

IV. CONCLUSION

In this study, a finite element model of an ultrasonic transducer is built to study the effect of mesh size on the resonant frequency. Besides, it should be noted that calculating the values of ceramic material parameters in the model also greatly affects the numerical simulation results. The results show that surveying and selecting the mesh size for the FEM model is essential before performing any numerical simulation.

REFRENCES

[1]. NguyễnViệtHoavàVũHồngTuân, "Vaitròcủasiêuâmtrướcsinhtrongquảnlýbệnhtắctáràngbẩmsinhtại BệnhviệnViệtĐứcgiaiđoạn 2010 - 2015," Tạpchí Y họcViệt Nam, vol. 465, no. 2, pp. 95–98, 2018.

[2]. L. T. Dũng, N. D. Tâm, and T. V. Chương, "NghiênCứuPhươngPhápPhụcHồiBiếnTửÁpĐiệnTrongĐầuDòCủaMáyĐoSâu ,DòCá," TạpchíKhoahọc - Côngnghệthủysản, vol. 2016, no. 1, pp. 26–31, 2016.

[3]. N. T. Hieu, "Khảosáthệsốbiếnđộngvậntốcxungsiêuâmápdụngtrongđánhgiáđộđồngnhấtcủabêtông," TạpchíGiaothôngvậntải, vol. 58, no. 10, pp. 45–49, 2017.

[4]. R. Mahdavinejad, "Finite element dimensional design and modeling of an ultrasonic transducer," Iran. J. Sci. Technol. Trans. B Eng., vol. 29, no. 2, pp. 253–263, 2005.

[5]. K. O'Shea, "Enhanced vibration control of ultrasonic tooling using finite element analysis," Am. Soc. Mech. Eng. Des. Eng. Div., vol. 37, pp. 259–265, 1991.

[6]. A. C. Mathieson, "Nonlinear CharacterisationOf Power Ultrasonic Devices Used In Bone Surgery," pp. 1–217, 2012, [Online]. Available: http://theses.gla.ac.uk/3135/.

[7]. V. D. Luong, P. T. M. Duong, T. B. N. Nguyen, N. K. Ngo, T. H. Nguyen, and V. Du Nguyen, "Dynamic Response of High-Power Ultrasonic System Based on Finite Element Modeling of Piezoelectric," J. Appl. Eng. Sci., vol. 21, no. 3, pp. 859–871, 2023.

[8]. K. Ngo-Nhu et al., "A new algorithm to calculate complex material parameters in piezoelectric stacks," Lat. Am. J. Solids Struct., vol. 20, no. 3, 2023.

[9]. M. V. Guiman and I. C. Roca, "A New Approach on Vibrating Horns Design," Shock Vib., vol. 2017, 2017.

[10]. E. Carrera, S. Valvano, and G. M. Kulikov, "Multilayered plate elements with node-dependent kinematics for electro-mechanical problems," Int. J. Smart Nano Mater., vol. 9, no. 4, pp. 279–317, 2018.

[11]. J. van Randerrat and R. E. Setterington, "Piezoelectric ceramics. Applications book," Philips Aplication Book. p. 202, 1974.

[12]. S. Sherrit, B. P. Dolgin, Y. Bar-Cohen, D. Pal, J. Kroh, and T. Peterson, "Modeling of horns for sonic/ultrasonic applications," in Proceedings of the IEEE Ultrasonics Symposium, 1999, vol. 1, pp. 647–651.