International Journal of Current Engineering and Technology
E-ISSN 2277 – 4106, P-ISSN 2347 – 5161

Available at http://inpressco.com/category/ijcet

Research Article

Calculation of Natural Frequencies of Micro-Motor by using Finite Element Method

Hatem R. Wasmi[#] Mohammad Qasim Abdullah[#], Jasim Mohammad Abid[#] and Omar Ali Jassim[#*]

[#]University of Baghdad, Mechanical Dept., Iraq

Accepted 03 Nov 2016, Available online 05 Nov 2016, **Vol.6, No.6 (Dec 2016)**

Abstract

In this work, it was used the ANSYS Program to get the natural frequency, where the results are showing the first eight natural frequencies and which are tabulated in Table (2). The table also shows results obtained from using triangular elements as well as a finer mesh of quadrilateral elements. The results were obtained using 384 quadrilateral elements, it wasn't different so much from those that use 1280 elements. This again shows that the triangular elements are less accurate than the quadrilateral elements. Note that the mode shapes obtained in the three analyses are nearly the same.

Keywords: *Micro-motor, Natural frequency, ANSYS, Finite Element.*

Introduction

During the case study, it was analyzed another MEMs device: a common micro-actuator in the form of a side drive electrostatic micro-motor, as shown in Figure (1)[G.R Liu and SS Quek, 2013]. The natural frequencies and mode shapes were examined of the micro-motor. The natural frequencies are as known properties of a system, and it is important to study the natural frequencies and corresponding mode shapes of the system, because if the forcing frequency is applied to the system near to or at the natural frequency, resonance will occur, and it will be associated with very large amplitude due to the vibrations that their results might be disastrous in some situations. In this case study, the flexural vibration modes of the rotor of the micro-motor will be analyzed.

Figure 1 SEM image of an electrostatic micro-motor with eight rotor and 12 stator poles

*Corresponding author **Omar Ali Jassim** and **Jasim Mohammad Abid** are Ph.D. student; **Hatem R. Wasmi** and **Mohammad Qasim Abdullah** are working as Professors

System Modeling

The geometry of the micro-motor's rotor will be the same as that of Figure (2) and the elastic properties will remain unchanged using the properties in Table (1) [Timoshenko, S. P. and Goodier, J. N.,1970] .

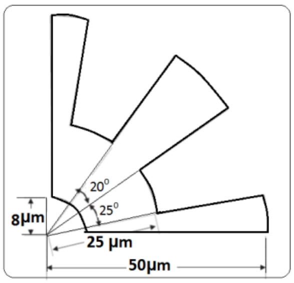

Figure 2 Plan view (2D) of a quarter of micro-motor rotor

Table1 Elastic properties of polysilicon [Timoshenko, S. P. and Goodier, J. N.,1970]

Young's Modulus, E	169 GPa
Poisson's ratio, ν	0.262
Density, ρ	2300 kgm−3

To show the mode shapes more clearly, the rotor was modeled as a whole rather than as a symmetrical quarter model. However, using a quarter model is still possible, but one has to take note of symmetrical and anti-symmetrical modes. Figure (3) shows the finite element model of the micro-motor containing 480 nodes and 384 elements [Liu, G. R., 2002].

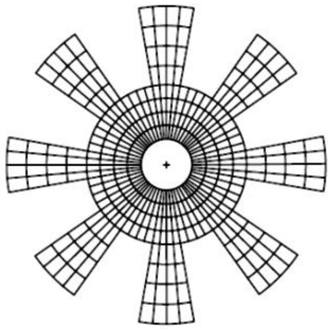

Figure 3 Finite element mesh using 2D, four nodal shell elements

The flexural vibration modes were studied, the plate elements discussed in this case ought to be used [Petyt, M.,1990]. However, as mentioned earlier in this study, most commercial finite element packages, including ANSYS, do not allow the use of pure plate elements. Therefore, shell elements will be utilized here for meshing up the model of the micro-motor. 2D, four nodal shell elements (S4) are used. Recall that each shell element has three translational degrees of freedom and three rotational degrees of freedom, and it is actually a superposition of a plate element with a 2D solid element. Hence, to obtain just the flexural modes, we would need to constrain the degrees of freedom corresponding to the *x* translational displacement and the *y* translational displacement, as well as the rotation about the *z* axis. This would leave each shell element with the three degrees of freedom of a plate element. As before, the nodes along the edge of the centre hole will be constrained to be fixed. Since we are interested in the natural frequencies, there will be no external forces on the rotor.

Solution Process

Looking at the mesh in Figure (3) one can see that quadrilateral shell elements are used. Therefore, the equations for a linear, quadrilateral shell element must be formulated by ANSYS. As before, the formulation of the element matrices would require information from the nodal cards and the element connectivity cards. The element type used here is S4, representing four nodal shell elements. There are other types of shell elements available in the ANSYS element library.

After the nodal and element cards, next to be considered would be the property and material cards. The properties for the shell element used here must be defined, which in this case includes the material used and the thickness of the shell elements [Cook, R. D.,1995,].

The boundary (BC) cards then define the boundary conditions on the model. In this work, the flexural vibration modes of the motor were obtained only, hence the components of displacements in the plane of the motor are not actually required. As mentioned, this is the important characteristic of the plate elements. Therefore, DOFs 1, 2 and 6 corresponding to the *x* and *y* displacements, and rotation about the *z* axis, are constrained. The other boundary condition would be the constraining of the displacements of the nodes at the centre of the motor.

Without the need to define any external loadings, the control cards then define the type of analysis (ANSYS) would carry out [Y. Nakasone and S. Yoshimoto,2006]. ANSYS uses the sub-space iteration scheme by default to evaluate the eigenvalues of the equation of motion. This method is a very effective method of determining a number of lowest eigenvalues and corresponding eigenvectors for a very large system of several thousand DOFs. Finally, the output control cards define the necessary output required by the analyst.

Result and Discussion

Using the input file above, an eigen value extraction is carried out in ANSYS [Y. Nakasone and S. Yoshimoto,2006]. The output is extracted from the ANSYS results file showing the first eight natural frequencies and tabulated in Table (2) . The table also shows results obtained from using triangular elements as well as a finer mesh of quadrilateral elements. It is interesting to note that for certain modes, the eigen values and hence the frequencies are repetitive with the previous one. This is due to the symmetry of the circular rotor structure. For example, modes 1 and 2 have the same frequency, and looking at their corresponding mode shapes in Figures (4) and (5) , respectively, one would notice that they are actually of the same shape but bending at a plane 90◦ from each other. As such, many consider this as one single mode. Therefore, though eight eigen modes are extracted, it is effectively equivalent to only five eigen modes. However, to be consistent with the result file from ANSYS, all the modes extracted will be shown here. Figure (6) to (11) show the other mode shapes from this analysis. Remember that, since the in-plane displacements are already constrained, these modes are only the flexural modes of the rotor.

Table 2 Natural frequencies obtained from analyses

Mode	Natural frequencies (MHz)		
	768 triangular elements with 480 nodes	384 quadrilateral elements with 480 nodes	1280 quadrilateral elements with 1472 nodes
1	7.67	5.08	4.86
2	7.67	5.08	4.86
3	7.87	7.44	7.41
4	10.58	8.52	8.30
5	10.58	8.52	8.30
6	13.84	11.69	11.44
7	13.84	11.69	11.44
8	14.86	12.45	12.17

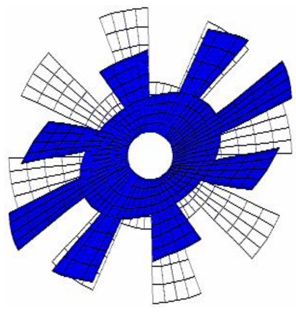

Figure 4 Mode 1

Comparing the natural frequencies obtained using 768 triangular elements with those obtained using the quadrilateral elements [Zienkiewicz, O. C., and Taylor , 2000], one can see that the frequencies are generally higher using the triangular elements. Note that for the same number of nodes, using the quadrilateral elements requires half the number of elements [Zienkiewicz, O. C., and Taylor, 2000].

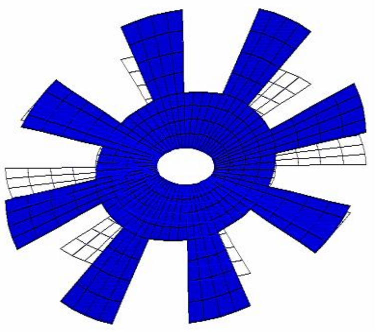

Figure 5 Mode 2

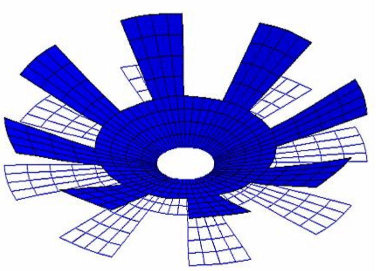

Figure 6 Mode 3

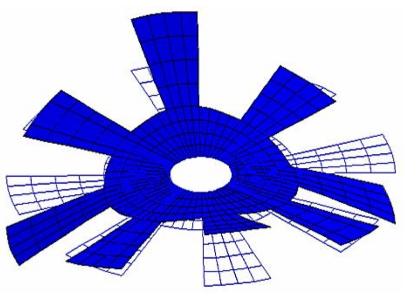

Figure 7 Mode 4

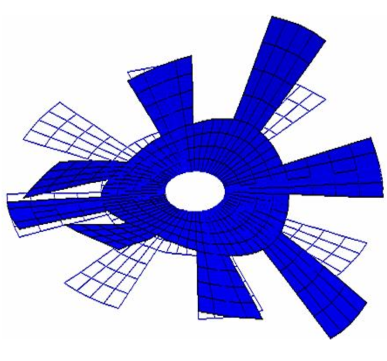

Figure 8 Mode 5

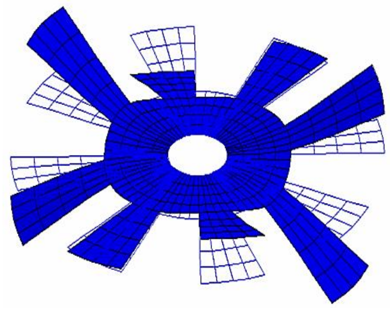

Figure 9 Mode 6

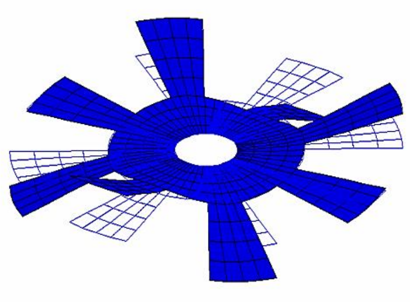

Figure 10 Mode 7

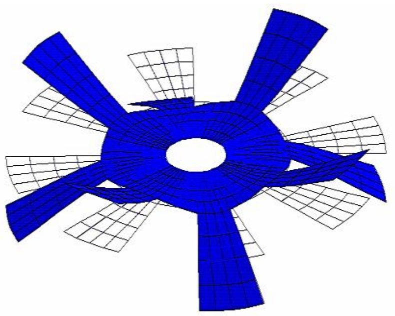

Figure 11 Mode 8

The results obtained using 384 quadrilateral elements do not differ much from those that use 1280 elements. This again shows that the triangular elements are less accurate than the quadrilateral elements. Note that the mode shapes obtained in the three analyses are the same.

References

G.R Liu and SS Quek, 2013, The Finite Element Method, A practical course

Timoshenko, S. P. and Goodier, J. N.,1970, Theory of Elasticity, 3rd edition, McGraw-Hall, New York.

Liu, G. R., 2002, Mesh Free Methods: Moving Beyond the Finite Element Method, CRC Press, Boca Raton.

Petyt, M.,1990, *Introduction to Finite Element Vibration Analysis*, Cambridge University Press, Cambridge

Cook, R. D., 1995, Finite Element Modeling for Stress Analysis, John Wiley & Sons, Inc.

Zienkiewicz, O. C., and Taylor , 2000, The Finite Element Method, 5th edition, Butterworth Heinemann.

Y. Nakasone and S. Yoshimoto,2006, Engineering Analysis With ANSYS Software