한국기계가공학회지, 제14권, 제2호, pp.127～133(2015.4) ISSN 1598~6721(Print)
Journal of the Korean Society of Manufacturing Process Engineers, Vol.14 No.2, pp.127～133(2015.4)ISSN2288-0771(Online)

http://dx/doi.org/10.14775/ksmpe.2015.14.2.127

Automation of One-Dimensional Finite Element Analysis of a Direct-Connection Spindle System of Machine Tools Using ANSYS

Jin-Woo Choi[*,#]
[*]School of Mechanical Engineering Technology, Yeungnam University College

ANSYS를 활용한 공작기계 직결주축 시스템의 1차원 유한요소해석 자동화

최진우[*,#]
[*]영남이공대학교 기계계열
(Received 23 March 2015; received in revised form 1 April 2015; accepted 3 April 2015)

ABSTRACT

In this study, an analytical model was developed for one-dimensional finite element analysis (1D FEA) of a spindle system of machine tools and then implemented to automate the FEA as a tool. FEA, with its vibration characteristics such as natural frequencies and modes, was performed using the universal FEA software ANSYS. VBA of EXCEL was used to provide the programming environment for its implementation. This enabled graphic user interfaces (GUIs) to be developed to allow interactions of users with the tool and, in addition, an EXCEL spreadsheet to be linked with the tool for data arrangement. The language of ANSYS was used to develop a code to perform the FEA. It generates an analytical model of the spindle system based on the information at the GUIs and subsequently performs the FEA based on the model. Automation helps identify the near-optimal design of the spindle system with minimum time and efforts.

Keywords : Finite Element Analysis(1차원 유한요소해석), Automation(자동화), Spindle System(주축 시스템), Optimal Design(최적설계), ANSYS(안시스)

1. Introduction

As recent machine tools have required high speed, high efficiency, and high precision, they require their spindle system to be improved in terms of technical performance including dynamic stiffness and precision[1]. The spindle system consists mainly of a shaft, an arrangement of bearings, a housing, and a motor and thus its performance is highly affected by the key components. The technical performance, especially, the dynamic stiffness, has a great influence on its cutting accuracy and material removal rate[2,3]. Again, the dynamic stiffness is based on mass, static stiffness, and damping and, in addition, the mass and the static stiffness determine natural frequencies and modes of vibration. Accordingly, design of a spindle system needs to pursue high natural frequencies

Corresponding Author : jinwoochoi@ync.ac.kr
Tel: +82-53-650-9223, Fax: +82-53-625-0861

leading to a high dynamic stiffness and consequently cutting performance of accuracy and removal volume.

The technical performance of spindle systems has been evaluated with finite element analysis (FEA)[3-6]. The performance included thermal characteristics and stiffness including natural frequency. A proper analytical model was developed for the FEA.

A spindle system under design can be evaluated using one, two, or three dimensional (3D) FEA. While 3D FEA uses a solid model of the spindle system, 1D FEA needs to converts the solid model into the 1D beams which have a length and a cross-section. However, 1D FEA needs less time of computation due to the relatively smaller number of nodes for FEA than 3D FEA does.

In this research, an analytical model was developed for 1D FEA of vibration of a direct-connection spindle system under design and implemented into automation of the FEA as a tool. The spindle system is composed mainly of a shaft, bearings and a cutting tool for the vibration analysis. The shaft was converted into beam elements, the bearings were into springs, and the cutting tool was into a mass element connected with the tip of the shaft. All the elements belong to the universal software of FEA, ANSYS[7], used in this research.

The computer languages of VBA (Visual Basic for Application)[8] of EXCEL[9] and APDL(ANSYS Parametric Design Language) were used for the implementation into the automation. VBA provided the programming environment to develop the tool and APDL allowed the analytical model to be automatically constructed in ANSYS. The tool uses EXCEL for data arrangement function such as a chart to display the section of the shaft. A user can interact with the tool using the graphic user interfaces (GUIs) developed in this research. This automation can be helpful for identification of an optimal design of the spindle system by rapidly performing design changes and consequently FEA in minimum time and efforts. Another advantage of the automation is a reduction in human error in construction of the FE model.

2. Development of an analytical model

2.1 Direct-connection spindle system

Fig. 1 presents configuration of a typical direct-connection spindle system[6]. It is constructed of the shaft directly connected with the motor using the coupling. The shaft is supported with the bearings and clamps the cutting tool at its tip. As shown in Fig. 1, the motor is assembled with the housing or a head frame.

The dynamic stiffness of the spindle system is heavily determined with the shaft, the bearings, and the cutting tool. The dynamic stiffness includes natural frequencies and vibration modes. The motor and the housing do not considerably affect the natural frequency because they are fastened to the head frame or the housing. These influential components should be converted into appropriate FEs for construction of an analytical model of the direct-connection spindle system.

2.2 Development of a analytical model

Fig. 2 shows an analytical model developed in this research for FEA of the direct-connection spindle system in Fig. 1. Its main components are replaced with appropriate finite elements. The shaft is meshed into the beam elements using length and tubular cross-section. The bearings in Fig. 1 are converted into spring elements in X, Y, and Z directions, respectively. The tool is into a point-mass element and then connected with the tip of the shaft with a rigid coupling element.

Boundary constraints are applied to the elements of the main components connected with the housing. The end node of the spring elements is constrained in both translation and rotation in X, Y, and Z directions, respectively. The end node of the shaft is constrained in

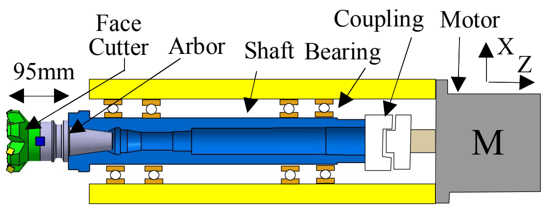


Fig. 1 Configuration of a spindle system

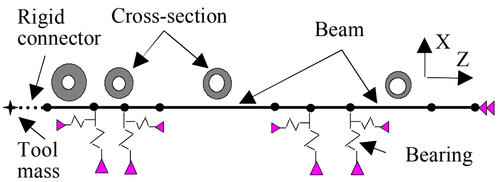


Fig. 2 Analytical model in 1D for the spindle system

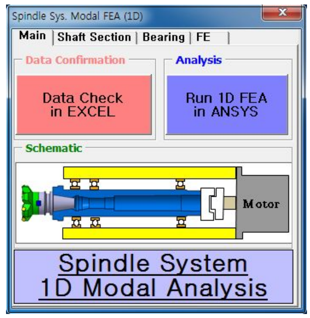
Z rotation as it is connected with the motor with a coupling as shown in Fig. 1. These boundary constraints prevent the six rigid motions from occurring for FEA of the natural frequencies and modes of the spindle system.

3. Implementation into FEA automation

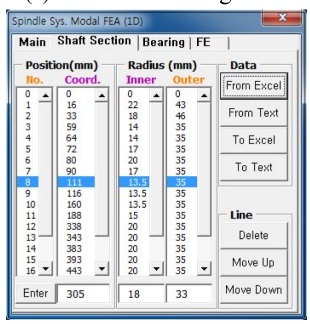
3.1 Development of graphic user interfaces

A user may place commands and receive responses through graphic user interfaces (GUIs) developed for the automation as shown in Fig. 3. With use of 'Multi-Page' of VBA, the GUIs could be designed to be small in size in order not to interrupt the user's vision.

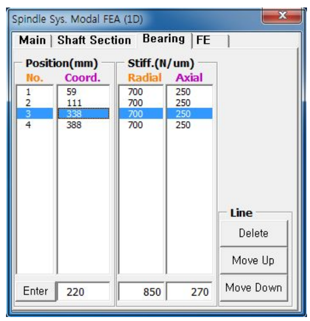
The GUI in Fig. 3 (a) launches an EXCEL spreadsheet to display the input data in the other GUIs for confirmation, for example, of the section configuration in a chart as shown in Fig. 4. It also launches ANSYS environment to conduct FEA. The shaft of a spindle system, especially, its longitudinal section and bearings are defined the GUIs in Fig. 3 (b) and (c), respectively. The GUI in Fig. 3 (d) requires element properties to be input for the shaft and the cutting tool. The user may determine the type and the size of the beam elements at the GUI.



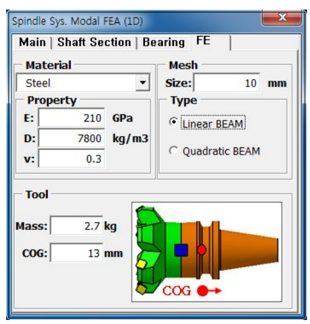
(a) GUI for running FEA



(b) GUI for definition of a shaft



(c) GUI for definition of bearings



(d) GUI for element characteristics

Fig. 3 Graphic user interfaces

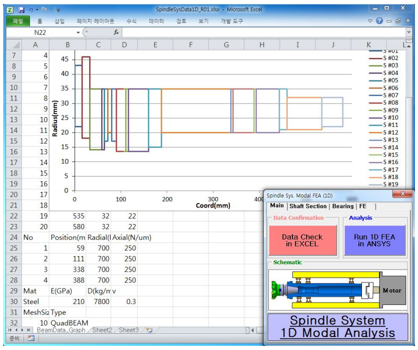


Fig. 4 Spreadsheet to display the input data

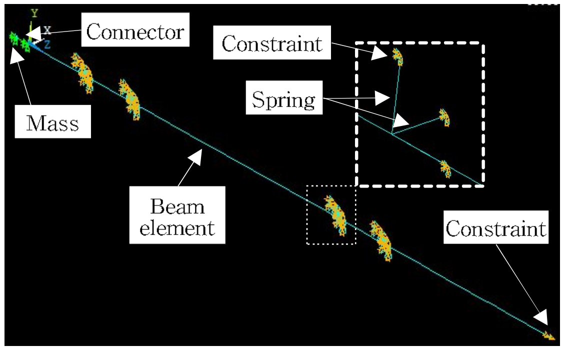


Fig. 5 FE model for analysis of the spindle system

The GUIs provide default values for the mesh and the material at the GUIs. Crome-Mo steel is normally used for the shaft of precision systems. Spindle systems of machine tools are representative precision systems. The size of the beam elements of the shaft is automatically determined with consideration of the whole length of the shaft. The type of beam element is determined with the size. Linear beam element is recommendable for a small length of beams to reduce computation time of FEA. These defaults are helpful for novice analysis engineers such as design engineers to perform FEA of a spindle system and, besides, can automate the FEA.

3.2 Implementation of the analytical model

The analytical model developed in Fig. 2 was implemented for vibration analysis in ANSYS Classic using APDL (ANSYS Parametric Design Language). The code was written in APDL to automate the FEA of the spindle system by constructing the geometric model and the analytical model in the ANSYS environment based on the data defined in the GUIs in Fig. 3.

The code generates the analytical model, shown in Fig. 5, of the spindle system in Fig. 1. First, it reads the data generated from the GUIs. Key points are made and then connected into lines representing the shaft defined. The lines are meshed into beam elements of ‘BEAM188’ of ANSYS with the tubular cross-sections defined at the GUI in Fig. 3 (b).

Nodes are created at the location for the bearings and then connected with each of their corresponding nodes on the shaft into spring elements of ‘COMBIN14’. Bearing stiffness are added to the spring elements in each direction. A node is generated at the location of its COG (Center Of Gravity) defined at the GUI in Fig. 3 (d) in order to make the mass element of the cutting tool. The mass element is defined with the mass of the tool and then connected with the node of the shaft tip with the rigid connecting element of ‘CP’. The elements constructed are shown in Fig. 5.

Boundary constraints are applied to the FE model. The end node of the spring elements are fixed in X, Y, and Z directions of translation and rotation, respectively. In addition, the node located at the shaft end is also constrained in rotation in Z direction. These boundary constraints prevent the FE model of the spindle system from the six rigid motions whose frequency is approximately 0Hz. Stiffness and mass matrix is constructed based on the FE model and then solved into node displacements. The vibration modes of the spindle system are visualized at each of the corresponding natural frequencies.

4. Application of the automation

The automation was applied to FEA of the spindle system shown in Fig. 1 for the purpose of validating the tool in usefulness. It automatically generated points and then connected them into lines based on the data at the GUI in Fig. 3 (b). They represented the shaft and then were meshed into the beam elements of the size of 10mm with the cross-sections shown in Fig. 6. A mass element was made for the cutting tool of 2.7kg and connected with the node at the shaft tip with the rigid connector shown in Fig. 5. Spring elements were made to represent the bearings of the spindle system. Boundary constraints were applied to the nodes of the spring elements and the shaft end as shown in Fig. 5. Computation was conducted to produce node displacements and the vibration modes of the spindle system were displayed for each natural frequency. The tool automates all the processes including the construction of the geometric model of points and lines and of the FE model.

Table 1 presents the FEA results of the natural frequencies and modes of the spindle system. The first vibration mode is that the shaft bends up and down about the last bearing at the frequency of 1,220Hz. This is because of the great length of the rear part of the shaft from the last bearing considering the bending stiffness determined with the cross-section and the mass of the rear part. The second mode is that the front part of the shaft vibrates at 2,409Hz due to mainly the cutting tool. This mode is considered to be slightly affected by the radial stiffness of 700N/μm[6] of the first bearing.

At the third mode, the shaft bends about the center between the second and the third bearings at the frequency of 3,659Hz showing a typical mode of the first bending of a beam. The fourth mode of the spindle system includes the two bendings about the center and the rear part of the shaft at 3,689Hz. At the fifth mode, the rear part of the shaft bends as a typical bending of a long beam. This fifth natural frequency is 5,574Hz. This mode is due to the great length of the rear part of the shaft from the last bearing like the first vibration.

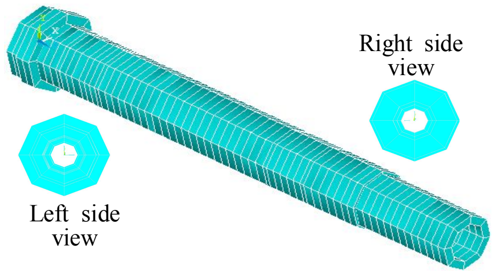
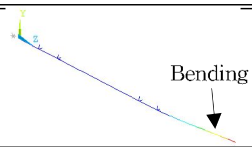
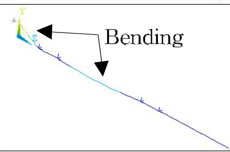
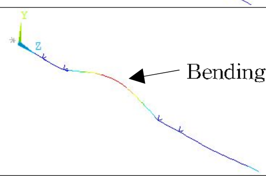
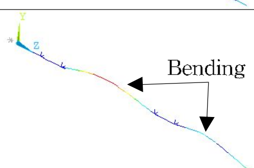
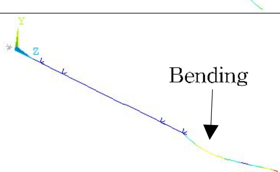


Fig. 6 Display of the cross-section of beam elements

Table 1 Natural frequencies and modes

Mode	Frequency (Hz)	Mode Shape
1	1220	
2	2409	
3	3659	
4	3689	
5	5574	

The spindle system is expected to sustain the face cutting of up to 24,000RPM (Revolution Per Minute) with a face cutter of 6 inserts against resonance. This cutting condition produces the periodic cutting force of 2,400Hz to the spindle system but the second frequency is over 2,409Hz. The cutting force does not affect the spindle system at the first mode because it occurs only at the rear part. The periodic force has direct influence on the second mode. Chattering in cutting is not considered in this research.

This automation enables a user to perform multiple design modifications and then FEA in search of an optimal design of a spindle system based on high natural frequencies leading to high dynamic stiffness. The number and the location of bearings and the configuration of the shaft have a large influence on the natural frequencies. Moreover, the other factors such as the mass of the cutting tool also affects the frequencies. This tool can help to make design modifications of a spindle system and consequently to perform the FEA of its vibration in minimum time as it automates this processes of design and analysis. This automation is also expected to save a great amount of time and efforts to perform both of design and FEA and besides, to prevent human errors from occurring in the processes.

5. Conclusion

An analytical model was developed for 1D FEA of the spindle system of machine tools and then implemented into automation of the FEA in the universal FEA program, ANSYS. The analysis evaluates the spindle system in vibration characteristics of modes and natural frequencies. Diverse types of finite elements were used for the analytical model. Beam element of one dimension was used for the shaft, mass element of zero dimension was for the cutting tool, spring element of one dimension was for the bearings, and coupling element of one dimension was for rigid connection between the cutting tool and the shaft.

The automation was implemented into a tool in the programming environment of VBA of EXCEL and thus it can make use of an EXCEL spreadsheet for data arrangement such as a chart. A user can interact with the tool through the graphic user interfaces (GUIs) developed for the tool to receive commands and to provide responses. APDL was used to develop a code for automation of the FEA in ANSYS. It generates geometric elements of the spindle system and then analytical elements based on the information defined at the GUIs and, finally, executes computation for the FEA.

This automation enables evaluation of a spindle system under design to be rapidly performed and thus design engineers to find out a near optimal design of the spindle system. This automation can make a great reduction in time, efforts, and human errors to perform both of design and FEA.

References

1. Baek, I. H., Kim, J. S., Jeon, U. C., Kim, N. K., Choi, M. S., and Lee, D. W., Machine Tools; Driving Mechanism and Design Principle, Ch. 5, Cheongmoongag, South Korea, 2007.
2. Weck, M., Handbook of Machine Tools, Vol. 1 (English translation), Ch. 1-3, John Wiley, New York, 1984.
3. Abele, E., Altintas Y. and Brecher C., “Machine tool spindle units”, CIRP, Vol. 59, pp. 781-802, 2010.
4. Lim, J. S., Chung, W. J. and Lee, C. M., “Selection of Bearing Position for Improving Static and Dynamic Stiffness of 40,000rpm High-speed Spindle”, J of KSMPE, Vol. 8, No. 1, pp. 10-17, 2009.
5. Jian, J., Ko, T. J. and Kim, H. S., “An Analysis of the Thermal Deformation of the Main Spindle for Small and Precision Lathe”, J of KSMPE, Vol. 6, No. 1, pp. 43-49, 2007.
6. Lee, C. H. and Choi J. W., “Evaluation of Thermal Characteristics of a Direct-Connection Spindle Using Finite Element Co-Analysis”, Journal of the KSMTE, Vol. 22, No. 2 228-234.

2013.
7. ANSYS Mechanical, ANSYS V14.5 User Manual, SAS IP, Inc., 2012.
8. Swartzfager, G., Chandak, R. and Alvarez, S., Visual Basic 6 Object-Oriented Programming, Coriolis Group Books, Ch. 1-5, 1998.
9. EXCEL, EXCEL Documentation, Microsoft, 2010.