

A study on automation of modal analysis of a spindle system of machine tools using ANSYS

Bong-Gu Lee¹, Jin-Woo Choi^{*}

¹School of Mechanical Engineering Technology, Yeungnam University College

ANSYS를 활용한 공작기계 주축 시스템의 진동 모드 해석 자동화에 관한 연구

이봉구¹, 최진우^{*}

¹영남이공대학교 기계계열

Abstract An analytical model was developed in this study and then implemented into a tool for automation of FEA (Finite Element Analysis) of a spindle system for natural frequencies and modes in the universal FEA software, ANSYS. VBA of EXCEL was used for the implementation. It allowed graphic user interfaces (GUIs) to be developed for a user to interact with the tool and, in addition, an EXCEL spreadsheet to be used for data arrangement. A code was developed in the language of ANSYS to generate the geometric model of the spindle system, sequentially to construct the analytical model based on the information in the GUIs, and finally to perform computation for the FEA. Its automation of the model generation and analysis can help to identify a near optimal design of the spindle system under design in minimum time and efforts.

요약 본 연구에서는 범용 유한요소해석 소프트웨어인 ANSYS를 사용하여 공작기계 스피들 시스템의 유한요소 해석 자동화를 위해서 해석 모델을 개발하고, 그것을 툴로 구현하였다. 그 구현을 위해 EXCEL의 VBA를 사용하였으며, 이로 인해서 사용자가 툴과 상호교류할 수 있도록 그래픽 인터페이스를 개발할 수 있었고, 데이터 정렬을 위해서 EXCEL의 spreadsheet를 사용할 수 있었다. ANSYS 언어를 사용하여 코드를 개발하였으며, 이는 인터페이스에 입력된 정보를 사용하여 형상 모델을 만들고, 순차적으로 해석 모델을 만들며, 마지막으로 유한요소해석 연산을 수행한다. 모델 생성과 해석의 자동화는 최소한의 시간과 노력으로 설계 중인 스피들 시스템의 근사 최적 설계를 발견하도록 도울 수 있을 것이다.

Key Words : Analytical model, ANSYS, EXCEL, Finite element analysis, Spindle system design, Tool development

1. Introduction

The spindle system of a machine tool is composed 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 including cutting accuracy and the material removal rate is highly determined with

dynamic stiffness of the spindle system[1,2]. Again, the dynamic stiffness is a function of mass, static stiffness, and damping and, besides, the mass and the static stiffness lead to natural frequencies and modes. Accordingly, the spindle system needs to be designed to have high natural frequencies leading to a high dynamic stiffness and then cutting performance of accuracy and removal volume.

^{*}Corresponding Author : Jin-Woo Choi (Yeungnam Univ. College)

Tel: +82-53-650-9223 email: jinwoochoi@ync.ac.kr

Received March 20, 2015

Revised April 7, 2015

Accepted April 9, 2015

Published April 30, 2015

Finite element analysis (FEA) has been applied to evaluation of the performance of a spindle system[3,4,5] such as thermal characteristics and stiffness including natural frequency. The use of FEA requires an appropriate analytical model to be developed. If the spindle system is designed into three-dimensional (3D) configuration, it would be evaluated with 3D FEA using solid elements. If it is simply designed with a line representing the shaft and circles representing its cross-sections, 1D FEA can be used with beam elements. It tends to be designed in 3D at the detail design stage and then evaluated in detail.

In this study, an analytical model was developed for 3D FEA of a spindle system and implemented into a tool to automate the FEA. The spindle is constructed mainly a shaft, bearings, and a cutting tool for the analysis. The shaft is meshed into solid elements, bearings are converted into springs, and the tool is into a point mass connected with the tip of the shaft.

VBA (Visual Basic for Application)[6] of EXCEL[7] and APDL (ANSYS Parametric Design Language) were used for the implementation. ANSYS[8] is a universal software of FEA. EXCEL is linked with the tool for use of its data arrangement function such as a chart to display the section of the shaft. Graphic user interfaces (GUIs) were developed to allow a user to use the tool with ease. This tool is designed to help design engineers to identify an optimal design of the spindle system by performing design changes and consequently FEA in minimum time and efforts.

2. Development of an analytical model

2.1 Structure of a spindle system

[Fig. 1] shows a typical direct-connection spindle system[5] which has the shaft directly connected with the motor with the coupling. The cutting tool is clamped at the tip of the shaft supported with the bearings and the motor is fastened with the housing or the head frame as shown in [Fig. 1]. The shaft, the bearings, and the tool heavily affect dynamic stiffness,

for example, natural frequency of the spindle system. The motor and the housing do not have a considerable influence on the natural frequency because they are fixed to the head frame. These main components should be converted into appropriate elements to make an analytical model of the spindle system.

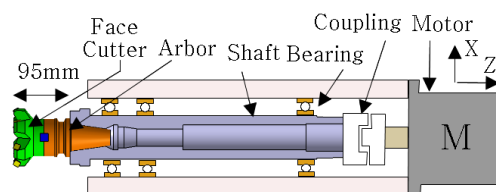


Fig. 1. A typical direct-connection spindle system

2.2 Development of an analytical model

[Fig. 2] presents an analytical model developed in this study for automation of the FEA of the spindle system in [Fig. 1]. Its components are converted into proper finite elements. The shaft of the spindle system is divided into solid elements. The bearings, whose stiffness is mainly considered in modal analysis, in [Fig. 1] are replaced with spring elements in X, Y, and Z directions[5]. Rigid bars are used to connect the face of the bearing seats with each of the corresponding springs at the bearing center point. The tool is replaced with a point-mass element and is also connected with rigid bars.

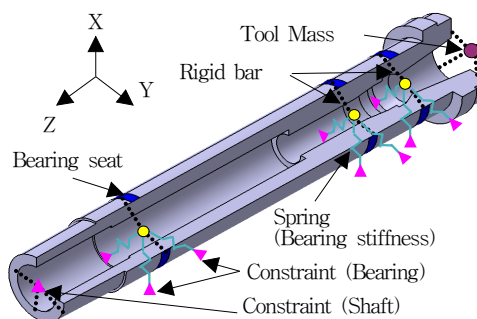


Fig. 2. Modal analysis model for the spindle system

Boundary constraints are applied to the elements of the components fixed with the housing. The spring elements are fixed in translation in X, Y, and Z

directions, respectively. The end of the shaft is fixed in Z rotation as it is coupled with the motor. These boundary constraints lead to prevention of the six rigid motions of the spindle system.

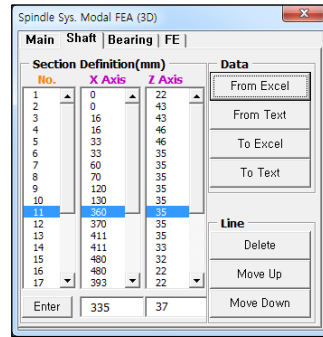
3. Development of an automation tool

3.1 Development of graphic user interfaces

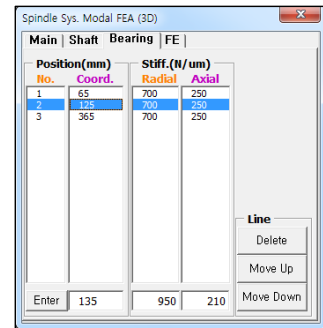
As shown in [Fig. 3], graphic user interfaces (GUIs) were developed for a user to place commands and receive responses. ‘Multi-Page’ of VBA allowed the GUIs to be designed to be small in size in order not to interrupt his/her vision.

Input data is defined in the GUIs in [Fig. 3] (b), (c), and (d). The GUI in [Fig. 3] (a) displays the input data into an EXCEL spreadsheet for confirmation of, for example, the section configuration in a chart as shown in [Fig. 4]. It also runs FEA in ANSYS. The GUIs in [Fig. 3] (b) and (c) are used to define the section of the shaft of a spindle system and bearings, respectively. The GUI in [Fig. 3] (d) requires FE properties to be input for the material of the shaft and the cutting tool. The user can select element type and size at the GUI.

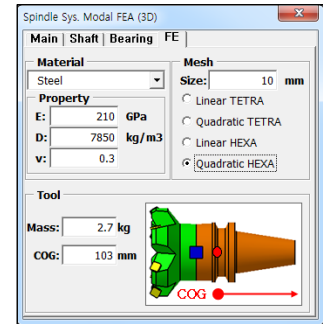
Default values are provided for the mesh and the material at the GUIs. Crome-Mo steel is normally used for the shaft of a precision system such as spindle systems of machine tools. Quadratic hexagonal element is recommendable for element type. The element size is automatically determined with the minimum thickness of the shaft section. These defaults are considered helpful to run the FEA.



(b)

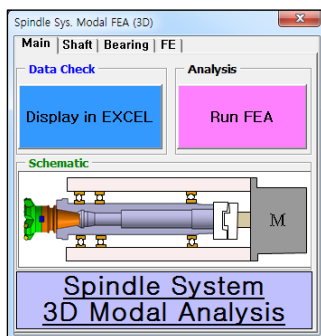


(c)



(d)

Fig. 3. Graphic user interfaces for FEA automation (a)GUI for analysis execution (b)GUI for shaft section (c)GUI for bearings (d)GUI for FE properties



(a)

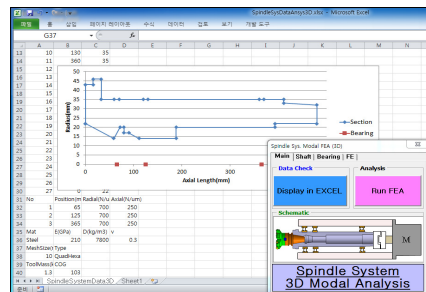


Fig. 4. Display of input data in EXCEL

3.2 Implementation of the analytical model

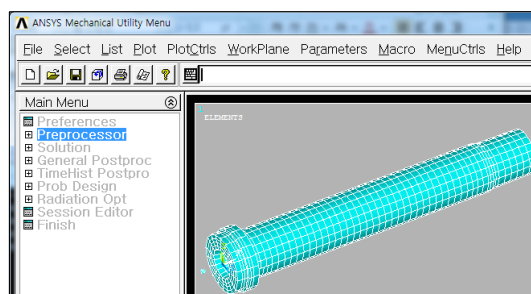
The analytical model for vibration modes, developed as shown in [Fig. 2], was implemented in ANSYS using APDL (ANSYS Parametric Design Language). The code written in APDL automates the FEA of the spindle system defined in the GUIs in [Fig. 3]. It makes a geometric model and then constructs the analytical model based on the geometric model.

It reads the data generated from the GUIs. A section is made and then rotated into the solid of the shaft, which is meshed into solid elements in quadratic hexagon. Nodes are created at the location for the bearings, the cutting tool, and the boundary constraint of the shaft and then rigidly connected with the elements of their corresponding faces such as bearing seats. Spring elements are made at the nodes of the bearing centers. A point mass is made at the node of the COG (Center Of Gravity) of the tool.

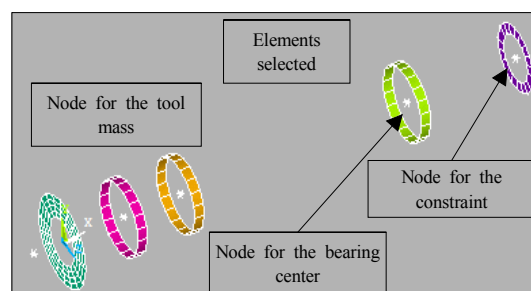
The other node of the springs are fixed in X, Y, and Z directions of translation, respectively and the node located at the shaft end is fixed in Z direction of rotation. These constraints remove the six rigid motions of the spindle system for the FEA of vibration mode, whose frequency is usually 0Hz. Stiffness and mass matrices are constructed and then solved into node displacements. The vibration modes of the spindle system are displayed with their natural frequency.

4. Application of the automation tool

The spindle system, shown in [Fig. 1], was used as an example to validate the tool in usefulness of the automation of FEA. It automatically generated a shaft and then meshed it into solid elements of quadratic hexagon in 10mm shown in [Fig. 5] (a). Rigid connections were made between the nodes and the elements selected as shown in [Fig. 5] (b) for spring elements of the bearings and the point-mass of the cutting tool. Boundary constraints were applied at the nodes of the bearing centers and of the shaft end.



(a)



(b)

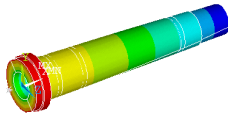
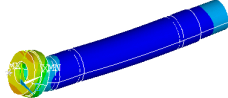
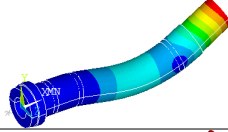
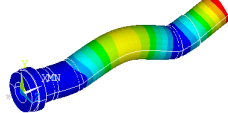
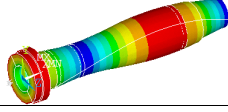
Fig. 5. Analytical model developed at FEA workbench
(a)Element generation in AYSYS
(b)Node connections

[Table 1] presents natural frequencies and modes of the spindle system designed by the tool. The first mode is that the shaft revolves in Z direction at the frequency of 1,344Hz. This is because of the masses of the shaft and of the cutting tool against the constraint at the shaft end. The second mode is that the shaft vibrates at the first bearing at 1,543Hz due to the mass of the cutting tool. The radial stiffness of $700\text{N}/\mu\text{m}[5]$ of the first bearing is thought to affect the mode to a slight degree. In the third mode, the shaft bends about the center between the second and the third bearings. Its frequency is 2,080Hz showing a typical mode of the first bending of a beam. The fourth mode of the spindle system is the second bending of its shaft at the frequency of 3,716Hz. The shaft swells in and out at the center between the second and the third bearings in the fifth mode of 4,300Hz. This mode is due to the thickness of the shaft.

The spindle system is considered to sustain the face cutting of up to 12,000RPM (Revolution Per Minute) with a face cutter of 6 inserts. Under this cutting

condition, it is exposed to periodic cutting forces of 1,200Hz but the first frequency is over 1,344Hz. Therefore, there would be no resonance for the cutting with the spindle system.

Table 1. Natural frequencies and modes

Mode	Frequency (Hz)	Mode Shape
1	1,344	
2	1,543	
3	2,080	
4	3,716	
5	4,300	

This tool is helpful to perform numerous design changes and then FEA for identification of an optimal design of a spindle system. Bearing count and location affect dynamic stiffness of the spindle system and also the shaft configuration has a large influence on the stiffness due to its mass or bending stiffness or both. The tool can help to re-design the shaft rapidly in geometry and, consequently, to perform the FEA of vibration modes of the spindle system as it automates this process. This automation is expected to make a large reduction in time and efforts to perform both of design and FEA.

5. Conclusion

In this study, an analytical model was developed and then implemented into a tool to automate FEA of a

spindle system in the universal FEA program, ANSYS. The FEA evaluates vibration modes and natural frequencies. The analytical model was constructed of diverse types of finite elements. A point element of zero dimension was used for the mass of a cutting tool, spring elements of one dimension were for bearings, and solid elements of three dimension were for the shaft meshed into quadratic hexagons.

VBA of EXCEL provided the programming environment for the implementation of the tool and thus it can use functions of an EXCEL spreadsheet such as a chart. Graphic user interfaces (GUIs) were developed for the tool to receive commands from a user and to provide responses. A code was developed in APDL for the automation of the FEA in ANSYS. It generates the geometric model of the spindle system and then the analytical model based on the information in the GUIs.

This tool provides rapid evaluation of a spindle system under design and thus it is helpful for design engineers to identify a near optimal design of the spindle system by automating the processes of design change and consequently FEA in minimum time and efforts.

References

- [1] M. Weck, Handbook of Machine Tools, Vol. 1 (English translation), Ch. 1-3, *John Wiley*, New York, 1984.
- [2] E. Abele, Y. Altintas, C. Brecher, "Machine tool spindle units", *CIRP*, Vol. 59, pp. 781-802, 2010.
DOI: <http://dx.doi.org/10.1016/j.cirp.2010.05.002>
- [3] S. C. Jung, K. W. Kim, "A study on Finite Element Analysis of Tool Deformation in End Milling", *Journal of the Korea Academia-Industrial Cooperation Society*, Vol. 6, No. 1, pp. 83-86, 2005.
- [4] J. K. Lee, "Evaluation of the Influence of Blast Vibration on Machine Tool Accuracy", *Journal of the Korea Academia-Industrial Cooperation Society*, Vol. 15, No. 8, pp. 4790-4795, 2014.
DOI: <http://dx.doi.org/10.5762/KAIS.2014.15.8.4790>
- [5] C. H. Lee, J. W. Choi, "Evaluation of Thermal Characteristics of a Direct-Connection Spindle Using Finite Element Co-Analysis", *Journal of the Korean*

Society of Manufacturing Technology Engineers, Vol. 22, No. 2, pp. 228-234. 2013.

DOI: <http://dx.doi.org/10.7735/ksmte.2013.22.2.228>

- [6] G. Swartzfager, R. Chandak, S. Alvarez, Visual Basic 6 Object-Oriented Programming, *Coriolis Group Books*, 1998.
 - [7] Microsoft Office Professional, EXCEL Documentation, Microsoft, 2010.
 - [8] ANSYS Mechanical V14.5, *ANSYS V14.5 Documentation*, SAS IP, Inc., 2012.
-

Bong-Gu Lee

[Regular member]



- Feb. 2003 : Yonsei Univ., Mech. Eng., MS.
- Feb. 2009 : Yonsei Univ., Mech. Eng., PhD.
- Mar. 2000 ~ Jul. 2003 : KITECH, Research Engineer
- Feb. 2012 ~ Mar. 2009 : Daerim College, Professor
- Mar. 2012 - current : Yeungnam University College, Sch. of Mech. Eng. Tech., Prof.

<Research Interests>

Precision Machining, CAD/CAM, Hybrid Machining

Jin-Woo Choi

[Regular member]



- Feb. 2001 : Kyungnam Univ., Mech. Eng, MS
- Dec. 2006 : UNSW., Mech. Eng., PhD
- Aug. 2008 ~ Feb. 2014 : Hyundai-Wia Corp., Engineer
- Mar. 2014 ~ current : Yeungnam University College, Sch. of Mech. Eng. Tech., Prof.

<Research Interests>

Knowledge-Based Engineering, Design Automation, Machine Tools