Journal of the Korea Academia-Industrial
cooperation Society
Vol. 16, No. 4 pp. 2350-2355, 2015

http://dx.doi.org/10.5762/KAIS.2015.16.4.2350
ISSN 1975-4701 / eISSN 2288-4688

Development of a tool to automate finite element analysis of a spindle system of machine tools

Jin-Woo Choi[1*]
[1]**School of Mechanical Engineering Technology, Yeungnam University College**

공작기계 주축 시스템의 유한요소해석 자동화를 위한 툴 개발

최진우[1*]
[1]영남이공대학교 기계계열

Abstract A tool was developed in this research for automation of one-dimensional finite element analysis (1D FEA) for design of a machine tool spindle system composed mainly of a shaft and bearings. As it is based on object-oriented programing, it uses the objects of a CAD system. It requires minimum data to be input to define the spindle system such as shaft cross-sections and bearing stiffness. Then, it automatically generates the geometric model based on the data and then, converts it into the FE model of 1D beams and springs. The graphic user interfaces developed allow a user to interact with the tool. This tool can be applied to identification of a near optimal design of the spindle system in minimum time and efforts by automating the FEA process with numerous design changes.

요 약 본 연구에서는 축과 베어링으로 구성된 공작기계 주축 설계를 위한 1차원 유한요소 해석을 자동화하기 위한 툴을 개발하였습니다. 객체지향 프로그래밍을 기반으로 하기 때문에, CAD 시스템의 객체를 사용할 수 있습니다. 스핀들 시스템을 정의하기 위한 축의 단면과 베어링 강성과 같은 최소한의 데이터를 입력할 필요가 있으며, 그 데이터를 기반으로 형상 모델을 먼저 만들고, 그리고, 1차원 빔과 스프링 요소로 구성된 유한요소 모델로 변환합니다. 본 툴을 위해서 개발된 사용자 인터페이스는 사용자가 툴과 상호교류할 수 있도록 도와줍니다. 본 툴은 다수의 설계 변경과 그 후에 수행되는 유한요소해석 과정을 자동화함으로써 최소한의 시간과 노력으로 공작기계 주축 시스템의 근사 최적 설계를 발견할 수 있도록 해줍니다.

Key Words : Tool development, Finite element analysis, Spindle system, Optimal design, Graphic user interface

1. Introduction

A spindle system is one of the main units of a machine tool to machine a material into a part. It should perform both a heavy cutting for large volume removal and a fine cutting for high cutting accuracy[1,2]. In order to achieve high speed, high efficiency, and high precision, recently, the spindle has been designed to have a high technical performance including dynamic stiffness and high precision[2]. A spindle system is composed of a shaft, an arrangement of bearings, and a housing and the bearings and the shaft have a relatively greater influence on its performance such as cutting accuracy and the removal efficiency.

Generally, a spindle needs to be evaluated in stiffness at the design stage. A simple way for the evaluation is to use the theoretical equation using simply supported beam[1]. It includes bearing stiffness and a uniform cross-section beam. It is limited to application to a spindle with multiple cross-sections of

---

[*]Corresponding Author : Jin-Woo Choi (Yeungnam Univ. College)
Tel: +82-53-650-9223 email: jinwoochoi@ync.ac.kr

Received March 11, 2015 Revised April 7, 2015
Accepted April 9, 2015 Published April 30, 2015

its shaft and more than two bearings. Finite element analysis (FEA) has been widely applied to the evaluation of, for example, spindle designs in stiffness and thermal characteristics as well[2-4].

One-dimensional (1D) FEA using beam elements takes less time in analysis computation than three-dimensional(3D) FEA using solid elements. As it is necessary to run FEA of multiple designs of a spindle for numerous times in order to identify an optimal design, the time for the FEA needs to be minimized for rapid evaluation. Conceptual design of a spindle is not as fully determined at its detail design in which its shaft is fully determined in configuration such as its cross-section. Thus, 1D FEA is appropriate to rapid identification of an optimal design and stiffness evaluation of a spindle at the conceptual design stage.

In this research, a tool was developed to automate 1D FEA to evaluate stiffness of a spindle under design. It requires a user to input minimum information on the spindle such as bearing stiffness and shaft radii. Some research has been carried out to develop tools for FEA automation with only a shaft and bearing position. They cannot consider cutting position and bearing stiffness which has a large influence on the static stiffness of the spindle[1-2]. Graphic user interfaces (GUIs) were developed for the tool to allow a user to enter the minimum information on the spindle design.

The CAD system allows boundary conditions to be applied on only geometries such as points or lines for FEA. Finite elements(FEs) or nodes cannot be selected to accommodate loads or boundary constraints. All the boundary conditions and element properties imposed on geometry are transferred directly to its FE model. The tool can construct links to compensate for the shortage in the conversion. For example, two nodes cannot be merged into one for element combination. An additional link is made to combine the two geometric lines which are converted into beam elements. The tool can automate the link generation and the imposition of boundary conditions.

The programming environment was Visual Basic for Application (VBA)[5] embeded in the computer-aided design (CAD) system, CATIA [6]. It can use the objects of the CATIA developed in object-oriented programming technique[7] and they are included in the tool to automate both geometry generation and FEA at the corresponding workbenches of the CAD system. The automation is expected to help even design engineers to perform FEA, usually done by analysis engineers, in order to identify an optimal design of the spindle. It has the advantage to reduce much time and efforts to perform FEA for the spindle.

2. Development of an automation tool

2.1 Configuration of a spindle system

[Fig. 1] shows a typical direct-connection spindle system[8], which is normally composed of a shaft, bearings, a housing, and a motor. The spindle system is connected directly with the motor with a coupling. Thus the motor has little influence on static stiffness of the spindle system as it is fixed with the housing. The cutting tool, especially, its cutting location, affects the stiffness due to the bending moment proportional to the distance from the cutting location to the shaft tip. The stiffness of the spindle system is determined mainly with the shaft, bearing stiffness and location, and cutting location.

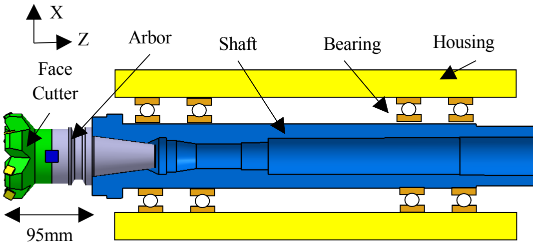


Fig. 1. Configuration of a spindle system

2.2 Development of a analytical model

[Fig. 2] shows an analytical model developed for FEA of the spindle in [Fig. 1] The shaft in [Fig. 1] is converted into beams with multiple cross-sections, the bearings are into the springs in the radial and axial directions, and the cutting tool is into a rigid connection with the tip of the shaft in order to apply the cutting force of 1,000N for static stiffness. The bearing seats are replaced with geometric lines, each of which is deformable and bound with each corresponding spring. The lines are divided into beam elements supported by the spring elements. The springs are fixed in X, Y, and Z directions in translation, respectively, and the shaft is fixed at the end in Z rotation as shown in [Fig. 2]. These boundary constraints prevent rigid motion in all directions in translation and rotations. The tool, developed in this research, automates the conversion process including the spring binding to construct the FE model shown in [Fig. 2].

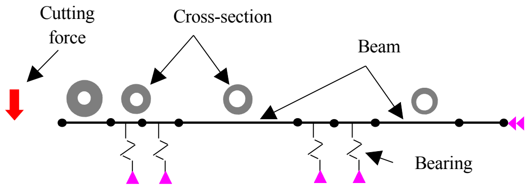
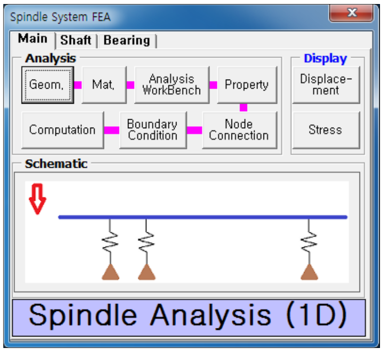


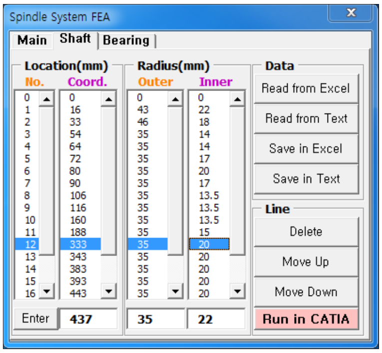
Fig. 2. Analytical model for the spindle system

2.2 Development of graphic user interfaces

[Fig. 3] shows graphic user interfaces(GUIs) developed for the automation tool in this research. They allow a user to interact with the tool by placing commands and receiving responses. 'Multi-Page' of VBA allowed the GUIs to be designed to be small to contain minimum information for FEA in order to prevent the user's vision to the main GUI of the CAD system.

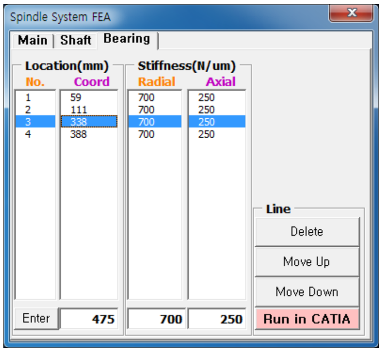


(a)



Location(mm) No.	Location(mm) Coord.	Radius(mm) Outer	Radius(mm) Inner
0	0	0	0
1	16	43	22
2	33	46	18
3	54	35	14
4	64	35	14
5	72	35	17
6	80	35	20
7	90	35	17
8	106	35	13.5
9	116	35	13.5
10	160	35	13.5
11	188	35	15
12	333	35	20
13	343	35	20
14	383	35	20
15	393	35	20
16	443	35	20
Enter	437	35	22

(b)



Location(mm) No.	Location(mm) Coord	Stiffness(N/um) Radial	Stiffness(N/um) Axial
1	59	700	250
2	111	700	250
3	338	700	250
4	388	700	250
Enter	475	700	250

(c)

Fig. 3. Analytical model for the spindle system
(a)GUI for analysis performance (b)GUI for shaft definition (c) GUI for bearing definition

The GUI in [Fig. 3] (a) governs generation of geometric and analytical models using the data input in those in [Fig. 3] (b) and (c). It also displays FEA results at the workbench of the CAD system. A shaft is defined with multiple lines and cross-sections. The lines are constructed with two points whose coordinate is defined in the GUI in [Fig. 3] (b). The interface in [Fig. 3] (c) accommodates bearing information including position and stiffness.

2.3 Procedure of the FEA automation

Fig. 4 shows the procedure of the FEA automation. A user inputs the data required to define a shaft with coordinates and sections and bearings with their positions and stiffness for a spindle. Points and lines are generated for the shaft and the bearings based on the data in the geometry modeling workbench. Material is applied manually. Use of 'drag and drop' is made to apply material to the shaft geometry from CATIA material interface. The geometries, lines and points, are transferred to the FEA workbench and then the lines are meshed into finite elements.

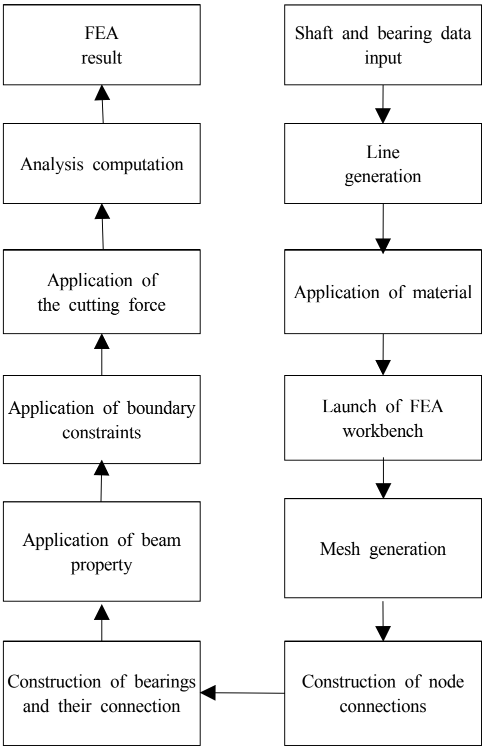


Fig. 4. Procedure of 1D FEA automation

The two nodes generated at the end of each of two neighboring lines need to be combined into one node in order to represent one connected shaft and, therefore, prevent computation singularity. As the FEA workbench does not allow two nodes to be merged into one for element combination by re-numbering nodes, it is necessary to develop an algorithm to make rigid combination of the two nodes of two neighboring lines. This process of rigid connection generation is repeated until all lines are connected together.

Springs are made for the bearings and connected with the lines of the shaft segments supported by the bearings. Beam property including sections is applied to the lines and boundary condition of constraints and a load is applied at the bearing springs and at the cutting point. The cutting force of 1,000 N is applied in X direction as it is a linear analysis to obtain the static stiffness of the spindle of interest.

All analysis data applied to the geometric elements is automatically transferred to its corresponding finite elements before the computation leading to node displacements. The result of FEA such as displacement or stress is displayed at the FEA workbench.

2.4 Structure of the automation tool

[Fig. 5] shows the structure of the FEA automation tool based on its modules and the peripheral systems. The modules and the systems take the responsibility of performing the analysis automation for a spindle system. The graphic user interfaces which are connected with the other modules allow a user to interact with the tool. The module of geometry generation makes the geometric elements to be used for FEA with the data provided from the module of data management. Then, it transfers the geometric elements from the geometry workbench to the FEA workbench for construction of an FE model. The module of FEA execution receives FEA data from data management and constructs the analytical model. The tool exchanges some text data with the spreadsheet program, EXCEL[9].

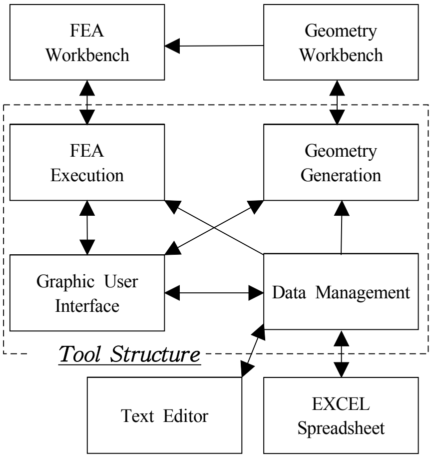


Fig. 5. Structure of the automation tool

3. Application of the automation tool

The tool was used to carry out an FEA for the spindle system, shown in [Fig. 1], defined with the data in [Fig. 3] (b) and (c). The tool constructed the geometry and then the analytical model, shown in [Fig. 5], for the FEA. Multiple containers were generated to store the data used for the FEA of the spindle system made mainly of the shaft and the bearings. [Fig. 7] shows the data of FE properties for node connections and bearings.

[Fig. 8] shows the displacement of the shaft and the bearings after the FEA completion. ‘B’ in [Fig. 8] represents a bearing. The maximum displacement occurred at the tip of the shaft since the load was applied at the cutting point connected with the shaft tip. The static stiffness at the tip can be obtained to be 260.4N/μm (=1,000N/0.00384mm) with the load and the displacement. It can be seen from [Fig. 8] that the displacement of bearings, especially, ‘B #1’ makes a large contribution to that at the tip. It is necessary that the number of bearings and their location are properly determined to achieve a high stiffness.

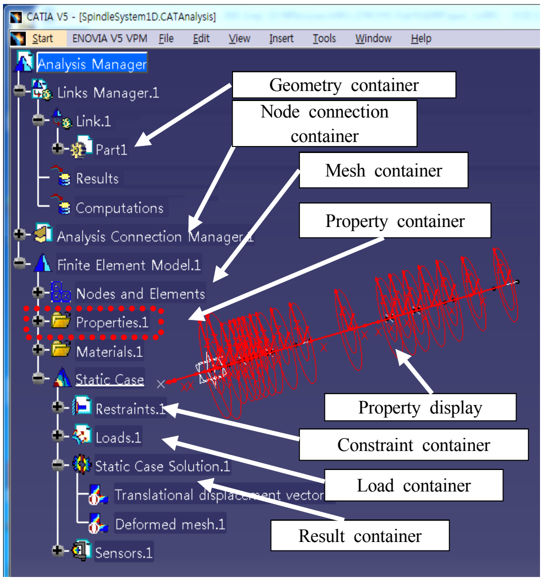


Fig. 6. Analytical model developed at FEA workbench

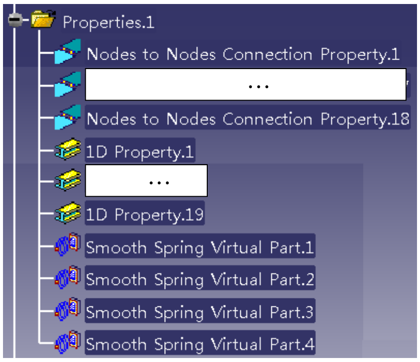


Fig. 7. Analytical data in ‘Property’ container

The displacement at the segments between the second and the third bearings is relatively smaller meaning that the shaft is highly stiff against the load and, therefore, can be smaller in diameter leading to a reduction in its weight and the size of bearings.

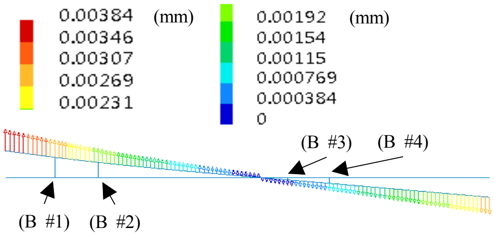


Fig. 8. Displacement of the spindle system

[Table 1] presents the reaction force occurring at the bearing points with the boundary constraints applied. The sum of the reaction forces is -1,000N which is the same magnitude with the load applied. It can be seen from the reaction forces in [Table 1] that the two rear bearings do not make a great contribution against the load applied and, therefore, one of the two can be removed leading to a reduction in bearing cost and assembly time.

Table 1. Reaction force at the bearing points

Bearing	B #1	B #2	B #3	B #4	Sum
Reaction Force(N)	-730.5	-546.8	76.1	201.2	-1,000

If this process is repeated with design changes, a near optimal design can be identified. The design changes include different diameters of the shaft, bearing stiffness, and bearing count. First, a design change is made and then is evaluated based on stiffness, weight, and manufacturing cost as well.

4. Conclusion

In this research, a tool was developed in the programming environment of a CAD system using its objects. The development focused on automation of one dimensional finite element analysis with the minimum data to define a machine tool spindle system constructed mainly of a shaft and bearings. The modules of the tool took the responsibility for generating geometric and analytical models at the workbenches of the CAD system to perform the FEA. Graphic user interfaces, developed for the tool, allows the tool to interact with users. They are linked with each of the modules to transfer the data required for the FEA to the modules.

This tool can play the role to identify a near optimal design for the spindle system with multiple design changes and, consequently, their analysis. The stiffness and position of the bearings can be optimized based on high stiffness and also cross-sections of the shaft of the spindle can be changed in order to have, for example, a minimum mass. Due to input of minimum data, this tool is expected to allow even a design engineer to find out an optimal design with nearly full automation of the FEA process 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, p781-802, 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] G. Swartzfager, R. Chandak, S. Alvarez, Visual Basic 6 Object-Oriented Programming, Coriolis Group Books, 1998.

[6] CATIA V5, CATIA V5 R18 Documentation, Dassault Systemses, 2007.

[7] J. K. Won, Object Oriented Programming with Examples, Global, 2013.

[8] 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. 2 228-234. 2013.
DOI: http://dx.doi.org/10.7735/ksmte.2013.22.2.228

[9] Microsoft Office Professional, EXCEL Documentation, Microsoft, 2010.

Jin-Woo Choi [Regular member]

- Feb. 2001 : Kyungnam Univ., Eng, MS
- Dec. 2006 : UNSW., 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