http://dx.doi.org/10.7735/ksmte.2015.24.2.172

J. Korean Soc. Manuf. Technol. Eng.
ISSN 2283-4846(Online) / ISSN 2233-6036(Print)

Development of a Tool to Automate One-Dimensional Finite Element Analysis of Machine Tool Spindles

Jin-Woo Choi[a*]

[a] School of Mechanical Engineering Technology, Yeungnam University College, 170, Hyunchung-ro, Nam-gu, Daegu 705-703, Korea

ARTICLE INFO

Article history:
Received 4 February 2015
Revised 3 April 2015
Accepted 4 April 2015

Keywords:
Tool development
Finite element analysis
Machine tool spindles
Optimal design
Graphic user interface
Object-oriented programing

ABSTRACT

In this research, a tool was developed to automate one-dimensional finite element analysis (1D FEA) for design of a machine tool spindle. Based on object-oriented programing, this tool employs the objects of a CAD system to construct a geometric model and then to convert it into the FE model of 1D beams at the workbenches of the CAD system with minimum data to define the spindle such as bearing positions and cross-sections of the shaft. Graphic user interfaces were developed for users to interact with the tool. This tool is helpful in identifying a near optimal design of the spindle with the automation of the FEA process with numerous design changes in minimum time and efforts. It is also expected to allow even design engineers to perform the FEA in search of an optimal design of the machine tool spindle.

1. Introduction

A spindle is one of the core elements determining the performance of a machine tool as it heavily affects its cutting accuracy and efficiency. As recent machine tools have required high speed, high efficiency, and high precision, their spindle needs to be improved in terms of performance including stiffness and precision[1]. It is composed mainly of a shaft, a set of bearings, and a housing. The shaft and bearings have relatively a larger influence on the accuracy and efficiency.

A spindle needs to be designed to have low mass and high stiffness to improve dynamic stiffness and, therefore, cutting performance. When the type of bearings and their position are fixed, the shaft determines the dynamic stiffness. The static stiffness of the shaft can be improved as its diameter becomes large but its mass becomes increased. The position of the bearings is also much influential on the static stiffness of the spindle.

Generally, a spindle composed of the shaft and the bearings should be evaluated in stiffness at the design stage. A simple means for the evaluation is to use the theoretical equation using simply supported beam[1]. It is difficult to apply the equation to the spindle with multiple cross-sections of its shaft and more than two bearings because of its application to simple beams. Finite element analysis (FEA) has been used to evaluate spindle designs in stiffness and thermal characteristics[2-6].

The FEA requires much knowledge to define the FE model with definition of element property and boundary condition.

* Corresponding author. Tel.: +82-53-650-9223
Fax: +82-53-625-0861
E-mail address: jinwoochoi@ync.ac.kr (Jin-Woo Choi).

Mesh generation is relatively easier to be carried out rather than the definition of property and boundary condition. However, design engineers do not have sufficient knowledge on FEA. Normally, analysis engineers perform the structural FEA on behalf of the design engineers. If a design change occurs, this mutual process is repeated. This mutual process takes much time and efforts. Accordingly, it is necessary to reduce the time and efforts to run the FEA of spindle designs.

FEA using beam elements of one dimension (1D) takes less time in analysis computation but more time in definition of element property than that using solid elements of three dimensions. A spindle is not fully determined in configuration of its cross-section at the conceptual design but fully determined at the detail design stage. Thus, one-dimensional elements seem to be appropriate to conceptual design of a spindle. However, the time for the property definition needs to be reduced to minimize the total FEA time.

CATIA[7] program used in this research, provide various engineering environments, such as ‘Workbenches’, for FEA, manufacturing, and programming based on the geometry model constructed internally. Besides, the programming environment allows for use of the objects generated in an Object-Oriented Programming (OOP)[8] computer language in development. CATIA has Visual Basic for Application (VBA)[9] to access many of its objects regarding geometric modeling, FEA and others.

In this research, an analysis tool was developed to automate 1D FEA with minimum information on a spindle to evaluate its stiffness. The information includes the definition of its shaft in beam and the position of its bearings. This automation tool was programmed in VBA embedded in the CATIA in order to use its geometric and analytical data. Graphic User Interfaces (GUIs) were developed to allow a user to interact with the tool. It is expected that this analysis tool helps even design engineers to perform FEA in order to identify an optimal design of the spindle. It is expected that this tool would reduce time and efforts to perform FEA for the spindle.

2. Development of an analysis tool

2.1 Spindle configuration

A spindle normally consists of a shaft, bearings, a housing, a cooling jacket, and a motor. The connection between the shaft and the motor determines the type of the spindle. For example, if the shaft is inserted at the center of the motor, it is a built-in spindle and if they are connected directly by a coupling, it is a direct connection spindle[6] as shown in Fig. 1.

The shaft and the bearings need to be designed in section and position to enable their assembly of the spindle to have maximum technical performance. The spindle can be converted into an analytical model with beams of tubular cross-sections representing the shaft and multiple supports representing bearings in position as shown in Fig. 2. The beams and the bearing positions need to be optimized for high stiffness and low mass.

2.2 Structure of the automation tool

Fig. 3 shows the structure of the tool including its modules

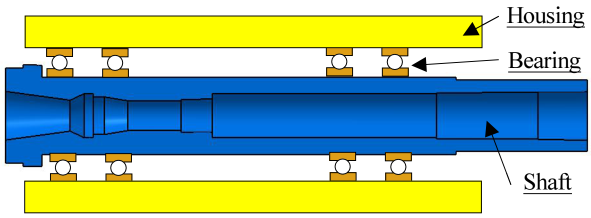


Fig. 1 Configuration of a spindle

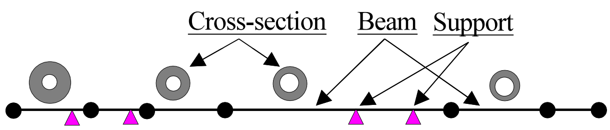


Fig. 2 Analytical model of the spindle shaft

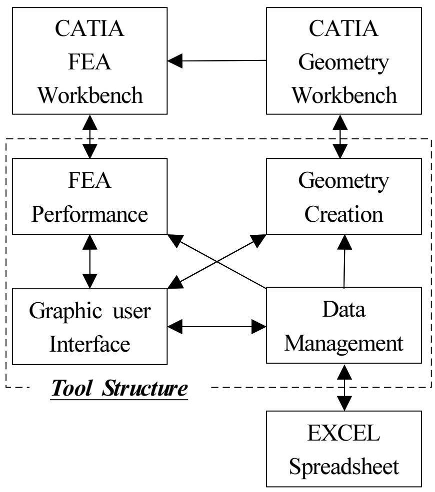


Fig. 3 Structure of the tool developed

and the peripheral systems. The modules and the systems play their own roles to perform the analysis automation for a spindle. A user interacts with the tool through its graphic user interfaces which are connected with the other modules. The module of geometry creation generates the geometric elements to be used for FEA based on the data provided from the module of data management. Then, it transfers the geometry at CATIA geometry workbench to the FEA workbench for an FE model. The module of FEA performance receives FEA data from data management and constructs the analytical model at the FEA workbench. The tool exchanges data with the spreadsheet program, EXCEL[10].

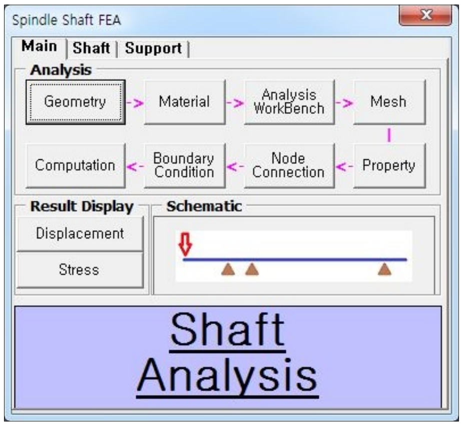
2.3 Development of graphic user interfaces (GUIs)

Fig. 4 shows user interfaces developed for the automation tool in this research. They allow a user to communicate with the tool by placing commands and receiving responses. The GUI in Fig. 4(a) helps to create the geometric model and the FE model using the shaft data defined at the GUI in Fig. 4(b). It executes the FEA and displays its result such as displacement and stress.

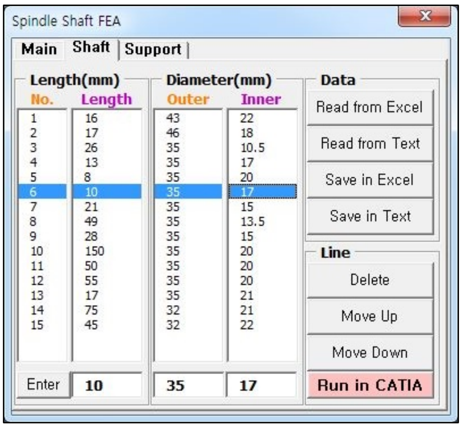
Boundary condition is imposed on the geometric model in the FEA workbench. The load of 1,000 N is applied to the tip of the shaft and the boundary constraint is to the shaft at the position defined at the GUI in Fig. 4(c). The magnitude of the load is not important to evaluate the static stiffness as it is a linear static analysis. The boundary condition of the load and the constraint is based on nodes of the FE model of the shaft. The shaft is defined with length, outer diameter, and inner diameter alone. If a certain segment of the shaft has a variable cross-section, it needs to break down into smaller beam elements with different cross-sections to compensate for the cross-section variation.

2.4 Automation limitation

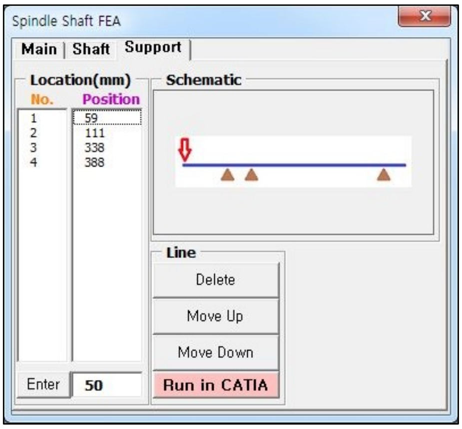
The tool has limitations to full automation of the process of the FEA since it cannot access all the objects of CATIA. VBA has a limited access to the objects such as the objects of material imposition in this research. The material needs to be defined manually for the shaft by ‘drag and drop’ function on CATIA material interface. In order to help with the manual definition, advice in text format is provided for the material



(a) GUI for analysis execution



(b) GUI for shaft definition



(c) GUI for bearing position

Fig. 4 Development of graphic user interfaces (GUIs)

definition.

The tool could fully automate the process of the FEA if the full accessibility for all CATIA objects is obtained. The commercial programming environment, CATIA Application Architecture - Rapid Application Development Environment (CAA-RADE) enables an independent application tool to be developed to be embedded in CATIA.

2.5 Automation process for FEA

A user defines the section of the shaft of a spindle and the bearing positions either at the GUIs in Fig. 4(b) and (c) or externally, for example, at an EXCEL spreadsheet. The tool makes the geometry model using lines, representing segments of the shaft, at geometry workbench and allows the user to directly apply the material of, mostly, steel to the model of the shaft definition. It launches the FEA workbench and defines the beam element size of the lines. The property is generated using the data of the shaft at Fig. 4(b) and then applied to each of the lines because the FEA workbench allows analytical data to be applied only to geometry even though finite elements and nodes are generated.

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 there is the accessability limitation with the combination, an algorithm was developed to make rigid connection between the two ends of the two neighboring lines. This connection process is repeated until all lines are connected together.

Boundary condition is applied with the load and the boundary constraints with the bearing positions in Fig. 4(c) and then, the computation is performed. The result of the analysis is displayed for displacement or stress. This FEA automation process can be repeated with different cross-sections of the shaft and different locations of bearings in search for an optimal design of the shaft with, for example, minimum mass and high stiffness.

3. Application to FEA of a Spindle

The tool was used to carry out an FEA for the spindle, shown in Fig. 1, defined with the data in Figs. 4(b) and (c). The span between the front and the rear bearings, located at 59 mm and 388 mm as shown in Fig. 4(c), is 329 mm. The tool generated the geometry and then the analytical model, shown in Fig. 5, for the FEA. Multiple containers were made to include the data used for the FEA of the spindle.

Fig. 6 shows the displacement of the spindle shaft after the FEA execution. The maximum displacement occurred at the tip of the shaft since the load was applied at the tip. The static stiffness can be obtained with the load and the displacement to be 813 N/μm (= 1,000 N/0.00123 mm). The displacement at the segments between the second and the third bearings is relatively much smaller meaning that the shaft is highly stiff against the load and, therefore, can be smaller in diameter leading to a reduction in its mass and the size of bearings.

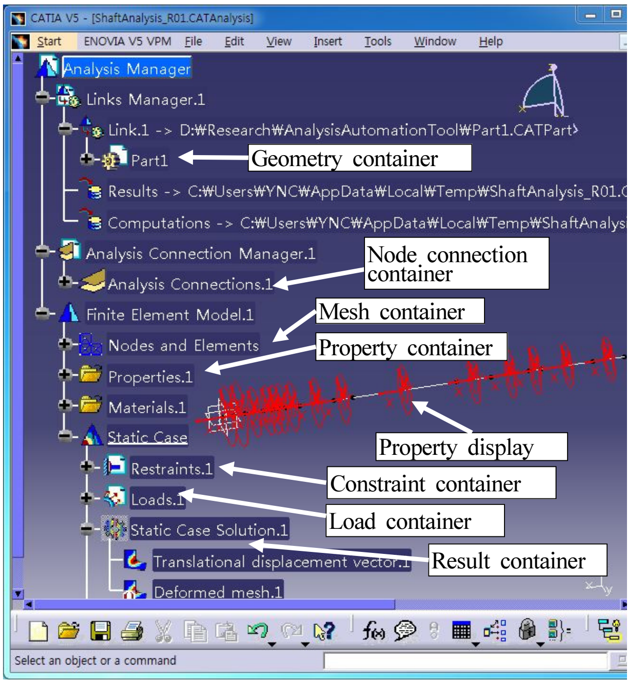


Fig. 5 Generation of an analytical model for FEA of a spindle

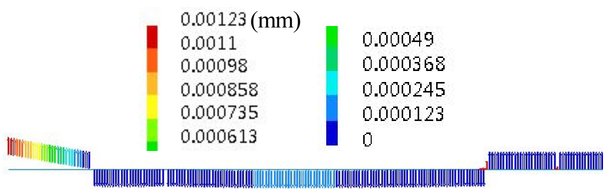


Fig. 6 Displacement of the shaft of the 4-bearing spindle

The reaction force were -1,933 N at the first bearing position at 59 mm distance from the tip, 877 N at the second, 101 N at the third, and -45 N at the last at 388 mm distance. The sum of the reaction forces leads to -1,000 N which is the same magnitude with the load applied. It can be seen from the reaction forces that the two rear bearings do not sustain much load and, therefore, one of the two can be removed to save the bearing cost and assembly time.

Fig. 7 shows the displacement of the spindle shaft supported by three bearings. The third bearing was removed from the shaft in Fig. 6. The static stiffness is the same with the one in Fig. 6 due to the same maximum displacement. However,

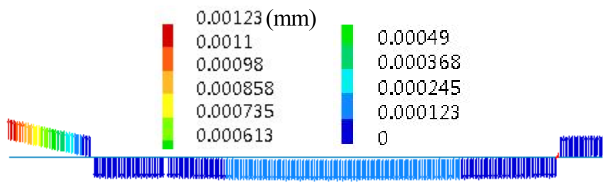


Fig. 7 Displacement of the shaft of the 3-bearing spindle

the 3-bearing shaft is slightly more deformed at the middle part than the 4-bearings shaft. The reaction forces were -1,964 N, 933 N, and 31 N at the bearings. Removal of the third bearing has no influence on the static stiffness and a slight influence on the overall deformation and reaction forces. The 3-bearing spindle could be better in design in terms of saving the bearing cost and assembly time.

A near optimal design can be found if this process is repeated with design changes. First, a design change is made and then is evaluated based on stiffness, mass, and manufacturing cost as well.

4. Conclusion

In this research, a tool was developed in the programming environment of a CAD system with the use of the objects of the CAD system. It was designed to automate one-dimensional finite element analysis with the minimum data to define a machine tool spindle. The modules of the tool played the role of generating geometric and analytical models at the workbenches of the CAD system to perform the FEA. Graphic user interfaces, developed for the tool, allow a user to interact with the tool. They are linked with the modules to help the data required for the FEA to flow among the modules and the CAD system.

This tool playes the role to identify a near optimal design for the spindle by repeating the analysis process with design changes. The position of the bearings can be optimized based on high stiffness and also cross-sections of the shaft of the spindle can be varied in order to have a minimum mass. This tool can allow even a design engineer to identify an optimal design with nearly full automation of the FEA process in minimum time and efforts.

References

[1] Weck, M., 1984, Handbook of Machine Tools, Vol. 1 (English translation), John Wiley, New York, Ch. 1-3.

[1] Baek, I. H., Kim, J. S., Jeon, U. C., Kim, N. K., Choi, M. S., Lee, D. W., 2007, Machine Tools; Driving Mechanism and Design Principle, Cheongmoongag, South Korea, Ch. 5.

[2] Abele, E., Altintas, Y., Brecher, C., 2010, Machine tool Spindle Units, CIRP, 59:2 781-802.

[3] Cao, Y., Altintas, Y., 2007, Modeling of Spindle-bearing and Machine Tool Systems for Virtual Simulation of Milling Operations, International Journal of Machine Tools & Manufacture, 47:9 1342-1350.

[4] Hwang, Y. K., Chung, W. J., Lee, C. M., 2007, Static Stiffness Characteristics of Main Spindle Interface using Finite Element Method, KSMTE, 16:1 40-46.

[5] Kim, S. I., Cho, J. W., 2004, Thermal Characteristic Analysis of a High-speed Horizontal Machining Center with Built-in Motor and Linear Motors, Transactions of the Korean Society of Machine Tool Engineers, 13:5 30-37.

[6] Lee, C. H., Choi, J. W., 2013, Evaluation of Thermal Characteristics of a Direct-connection Spindle Using Finite Element Co-analysis, KSMTE, 22:2 228-234.

[7] CATIA V5 R18, 2007, CATIA V5 R18, Dassault Systemses.

[8] Won, J. K., 2013, Object Oriented Programming with Examples, Global, South Korea.

[9] Swartzfager, G., Chandak, R., Alvarez, S., 1998, Visual Basic 6 Object-oriented Programming, Coriolis Group Books, USA.

[10] Microsoft Office Professional, 2010, EXCEL, Microsoft.