http://dx.doi.org/10.7735/ksmte.2015.24.2.166

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

Development of an Automation Tool for the Three-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	11	February	2015
Revised	28	February	2015
Accepted	6	April	2015

Keywords:
Automation tool development
3D finite element analysis
Machine tool spindle
Optimal design
Graphic user interface
Object-oriented programming

ABSTRACT

In this study, an automation tool was developed for rapid evaluation of machine tool spindle designs with automated three-dimensional finite element analysis (3D FEA) using solid elements. The tool performs FEA with the minimum data of point coordinates to define the section of the spindle shaft and bearing positions. Using object-oriented programming techniques, the tool was implemented in the programming environment of a CAD system to make use of its objects. Its modules were constructed with the objects to generate the geometric model and then to convert it into the FE model of 3D solid elements at the workbenches of the CAD system using the point data. Graphic user interfaces were developed to allow users to interact with the tool. This tool is helpful for identification of a near optimal design of the spindle based on, for example, stiffness with multiple design changes and then FEAs.

1. Introduction

A spindle is the main unit cutting materials into the part required. It is exposed to heavy cutting for high volume removal and fine cutting for high cutting accuracy[1,2]. Recently, high speed, high efficiency, and high precision have been required for machine tools and, therefore, the spindle has been required to be designed a high technical performance including dynamic stiffness and precision for cutting[2]. A shaft, an arrangement of bearings, and a housing are the core components to construct the spindle units and thus highly affect its performance such as cutting accuracy and the removal efficiency.

A spindle needs to be designed to pursuing the high performance to improve dynamic stiffness and, therefore, cutting volume and precision. The dynamic stiffness is highly related to mass and static stiffness. Thus, it is necessary to design the major components into low mass and high static stiffness.

Finite element analysis (FEA) has been widely applied to evaluation of spindles in performance such as stiffness or thermal characteristics at the design stage[3-7]. All components of the spindle including its shaft is fully designed into three dimensional (3D) configuration in a computer-aided design (CAD) system and, therefore, can be evaluated based on 3D FEA using solid elements. 3D FEA has the advantage that the 3D model does not need to be converted into 1D finite elements of beams with different cross-sections. If a certain segment of the shaft has a variable cross-section in 1D FEA, it is necessary to divide it into smaller beam elements with

---

* Corresponding author. Tel.: +82-53-650-9223
Fax: +82-53-625-0861

E-mail address: jinwoochoi@ync.ac.kr (Jin-Woo Choi).

different cross-sections for compensation of the cross-section variation.

A standard model needs to be developed for FEA of a spindle at the design stage. Much knowledge is required for the FEA, for example, to define element property and boundary condition. Generally, design engineers have less knowledge and experience in performing the FEA than analysis engineers and, therefore, the analysis engineers tend to carry out the FEA on behalf of the design engineers. If a FE model is standardized, the design engineers can perform FEA without much efforts. Accordingly, it is necessary to develop a FE model for 3D FEA of a spindle at the detail design stage and to implement it into a tool to automate the FEA to save time and efforts.

Many CAD systems have diverse engineering capabilities for geometric modeling, FEA, manufacturing, and programming. The CAD system, CATIA[8], used in this research, has many ‘Workbenches’ for the engineering performance. In addition, many of the CAD systems were constructed based on object-oriented programming (OOP)[9] and allow their objects to be used externally for development of software tools. CATIA has the programming environment, Visual Basic for Application (VBA)[10], to access many of its objects regarding geometric modeling, FEA and others and macro functionality as well. The two scripting languages have a different level of accessibility to the CATIA objects.

In this research, a tool was developed for automation of 3D FEA of a spindle geometrically designed in 3D. Mainly, its shaft is evaluated in stiffness with bearing supports. A standardized FE model was developed and implemented into the automation tool in VBA embedded in the CATIA in order to use its objects to control geometric and analytical data. Graphic user interfaces (GUIs) were developed for the tool to interact with users. It is expected to help even design engineers to perform FEA in search of an optimal design of the spindle, especially, its shaft. Besides, it would reduce an amount of time and efforts for evaluation of the spindle.

2. Development of a Tool for 3D FEA

2.1 Development of an analytical model for a spindle

Fig. 1 shows a typical spindle composed mainly of a shaft, bearings, and a housing. A driving source, normally, a motor, is connected with the shaft for revolution. A direct-connection spindle[6] is combination of a shaft and a motor with a coupling in a row. The motor does not much affect the static stiffness of the spindle as it is fixed with the housing and therefore, the shaft is the main component to determine the stiffness.

Fig. 2 shows an analytical model developed for FEA of the shaft supported with the bearings in Fig. 1. A bearing seat is constructed with a face for definition of bearing location and then a center point is made at the center of each of the bearings to apply the boundary constraint for the FEA. The point is fixed in X, Y, and Z directions in translation and Z direction in rotation, respectively, and rigidly connected with the bearing seat. In spite of rigid connection, the bearing seat is deformable in order to allow for shaft bending at the bearing support. The cutting force is applied at the tip face of the shaft with the rigid connection. The tetrahedral element of 10 nodes is selected and its size is half of the minimum thickness in the section for analysis accuracy.

2.2 Graphic user interfaces (GUIs)

Graphic user interfaces, shown in Fig. 3, were developed for the automation tool in this research. They are used to allow a user to interact with the tool with placing commands

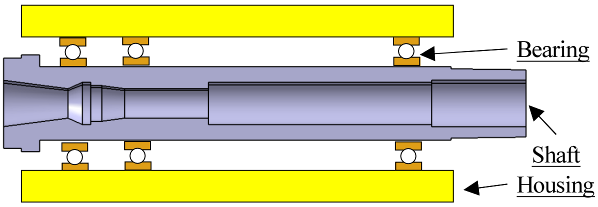


Fig. 1 Configuration of a spindle

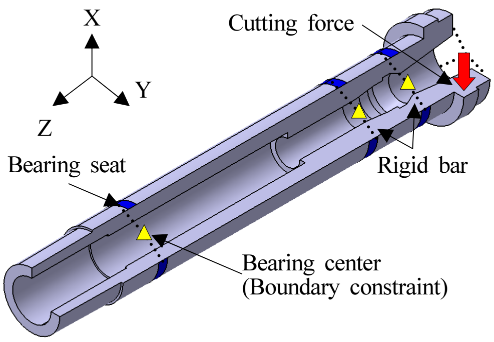
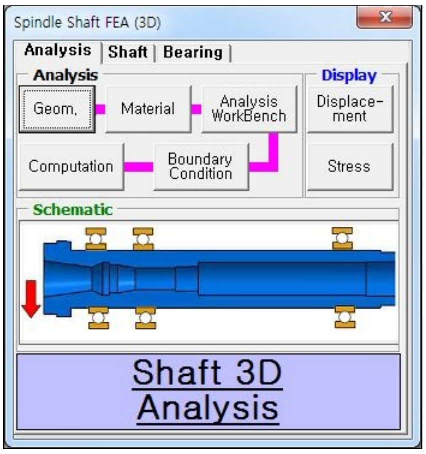
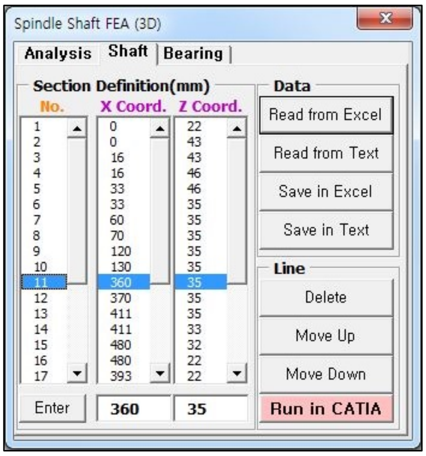


Fig. 2 Analytical model of the spindle

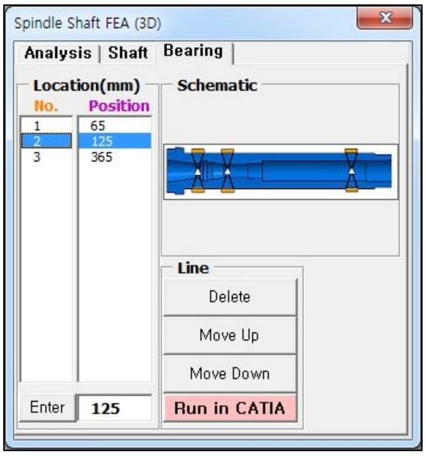


(a) GUI for analysis performance



No.	X Coord.	Z Coord.
1	0	22
2	0	43
3	16	43
4	16	46
5	33	46
6	33	35
7	60	35
8	70	35
9	120	35
10	130	35
11	360	35
12	370	35
13	411	35
14	411	33
15	480	32
16	480	22
17	393	22
Enter	360	35

(b) GUI for shaft section definition



No.	Position
1	65
2	125
3	365
Enter	125

(c) GUI for bearing position

Fig. 3 Development of graphic user interfaces (GUIs)

and receiving responses. Using ‘Multi-Page’, The GUIs were designed to be small to have minimum information for FEA in order to prevent the GUI from covering the main GUI of the CAD system, CATIA. Data for the shaft section and bearings can be entered and changed easily with multiple buttons in Fig. 3(b) and (c). Images are also provided for understanding of the spindle definition.

Table 1 Macro code to provoke the FEA workbench

```
Dim arrayOfVariantOfShort1(0);
arrayOfVariantOfShort1(0) = 0;
analysisManager1.ImportDefineFile "Part1.CATPart",
"CATAnalysisImport", arrayOfVariantOfShort1;
```

Table 2 Procedure of selection of a face with its corresponding line

Step #1: Determine a face for rigid connection
Step #2: Search for its constructing line
Step #3: Search for the points to construct the line
Step #4: Identify the names of the line and the points
Step #5: Selection of the connection face with the names

2.3 Algorithms for the automation

Some algorithms were developed for the automation of the FEA. Although embedded in the CAD system, VBA cannot access all of its objects. The system has ‘Macro’ function to allow a user to record a series of tasks in script and run it for repetition. ‘Macro’ has a better accessibility to certain objects. The object, *arrayOfVariantOfShort1* in Table 1, cannot be used in VBA to bring in the FEA workbench but in macro. A macro was written in script and an algorithm was implemented into the tool in order to execute the macro externally to automate the process of the FEA.

A code was written to select the face for rigid connection of the points for the bearings and the cutting force. The FEA workbench does not allow the boundary condition to be applied to finite elements or nodes. Accordingly, their corresponding face should be identified automatically. The code used the rule of geometry generation of the CAD system with geometric naming. The code memorizes the point numbers, defined at the GUI in Fig. 3(b), and use them to identify the cylindrical face made with revolution of its corresponding line made of two points as shown in Table 2 and Fig. 4 showing the solid generated with the section of the poly-line defined with multiple points contained in ‘SectionDefinition’.

2.4 Procedure of the FEA automation

Fig. 5 shows the procedure of the FEA automation. A user

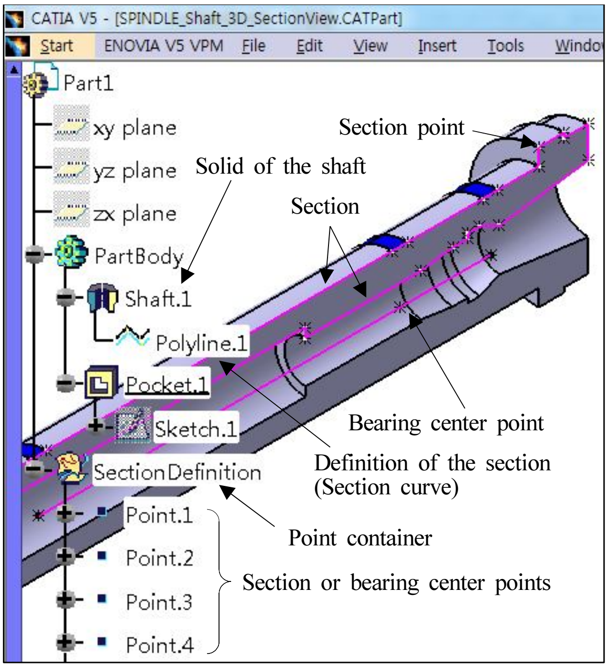


Fig. 4 Points for the shaft section and bearings

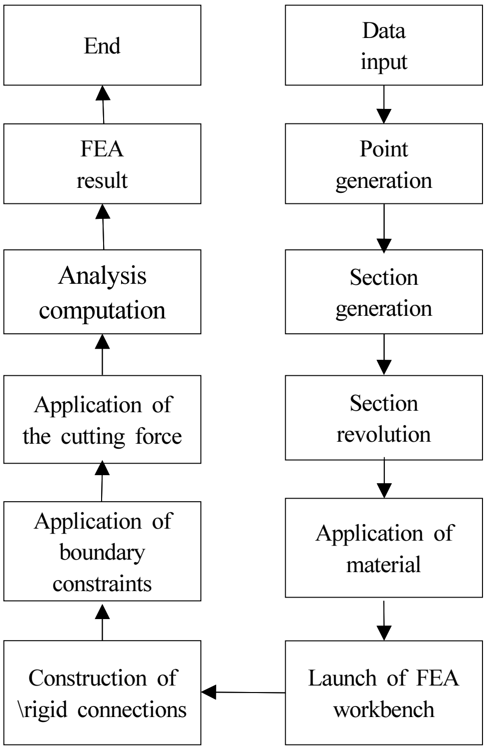


Fig. 5 Procedure of the FEA automation

inputs the data required to define the section of the shaft and the bearing position for a spindle. Points are generated based on the data in the geometry modeling workbench. A section is made by connecting the points into a polyline and then is rotated into the solid of a spindle shaft shown in Fig. 4.

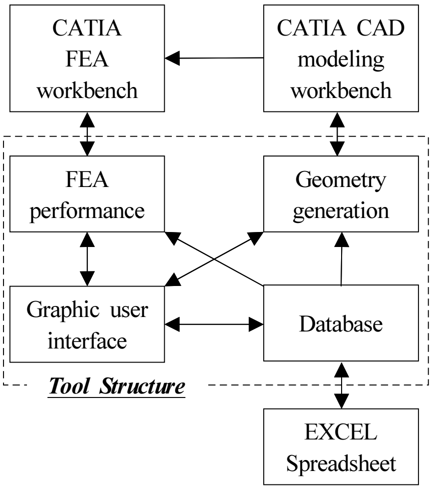


Fig. 6 Modules of the tool developed

Material is applied manually using ‘drag and drop’ function on CATIA material interface. The FEA workbench is launched and the rigid connections are made between points and their corresponding face in order to apply the boundary condition to the points. The boundary constraints in X, Y, and Z directions, respectively, are imposed on the bearing points and the cutting force of -1,000 N is applied in X direction to the load point. As these points are connected with their corresponding faces, the boundary condition applied is transferred to the faces. The magnitude of the cutting force is not influential on the static stiffness of the shaft in a linear static analysis. The elements to be used is automatically determined in size and type at FEA workbench of CATIA. A tetrahedral element based on 10 nodes is selected with the size determined with the section thickness. Computation process is run to mesh the solid of the shaft and then to solve the FE stiffness matrix into displacements. The static stiffness is evaluated with the FEA result of the cutting force and its corresponding displacement.

2.5 Structure of the tool

The FEA procedure in Fig. 5 was implemented into the modules, shown in Fig. 6, of the tool developed in this research. The structure of the tool is similar to that of other automation tools because they automate the task, for example, FEA in this research, usually performed manually. The modules play their own role for the FEA automation.

Connected with the modules, the GUIs allow a user to interact with the tool. The module of ‘Geometry generation’ makes the shaft in 3D and its bases of corresponding points and lines at the CAD modeling workbench and that of ‘FEA performance’ constructs the FE model for the shaft, runs the computation, and displays the result of the FEA at the FEA workbench. ‘Database’ plays the role to provide data for the other modules and, in addition, is connected with the spreadsheet program, EXCEL[11] for data exchange.

3. Application to 3D FEA of a Spindle

The spindle, shown in Fig. 1, was used to validate the usefulness of the tool which automatically carried out the tasks of geometry generation and FEA. As shown in Fig. 7, the tool made multiple containers to include the FE model of boundary constraints, properties, and others. Ten-node tetrahedral elements of 5 mm in size were included in the mesh container. CATIA automatically registered all the containers in the tree shown in Fig. 7. In addition, it constructed the rigid connections at the bearing supports for the boundary constrains.

The displacement of the spindle shaft was obtained after the FEA execution as shown in Fig. 7. As the cutting force of -1,000 N was applied at the point rigidly connected with the front face, the maximum displacement occurred at the front face of the shaft. It can be seen that the front part before the first bearing support is much deflected but the middle one between the first and the second bearings and the rear part between the second and the third bearings is little deflected because it acts as a continuous beam with three supports.

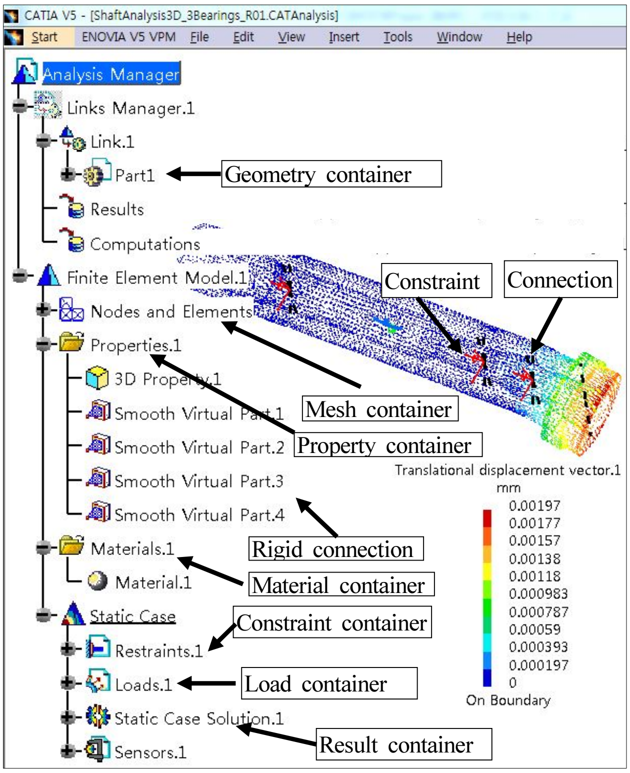


Fig. 7 FEA of a shaft with three supports

FEA was carried out for a shaft supported with two bearing as shown in Fig. 8. The boundary condition including the cutting force was the same with that for the shaft in Fig. 7. The maximum displacement occurred at the same location. It can be seen from Fig. 8 that the shaft is much deflected throughout the whole part because it acts as a simply supported beam with two bearings. The middle part is much more deflected than the one in Fig. 7.

The reaction forces were obtained from the FEA and presented in Table 3. ‘RF’ represents reaction force in Table 3. The sum of the reaction forces at each of the shafts has the same magnitude with the cutting force applied but the opposite sign. It is seen that the maximum reaction force occurs at the front bearing because of the location of the cutting force. It implies that the front bearing needs to have a high stiffness to prevent a large bearing displacement.

The displacement of the shaft with two bearings is much greater than the one with three by 1.74 times and thus its static stiffness is also much higher. It is better to have three bearings than to have two bearings in order to increase the stiffness of a shaft. The number of bearings and their location can be

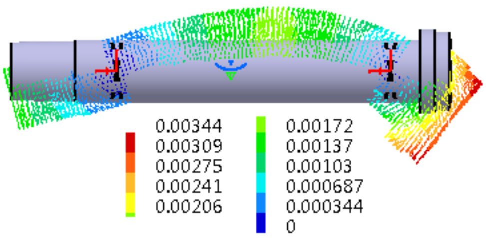


Fig. 8 Displacement of a shaft with two supports (Unit: mm)

Table 3 Reaction forces for the shafts

No. of supports	RF #01 (N)	RF #02 (N)	RF #03 (N)	Max. displacem-ent (mm)	Static stiffness ($\times10^3$N/mm)
2 bearings	1,216	-	-216	0.00344	291
3 bearings	1,977	-951	-26	0.00197	508

determined with more FEAs in search of the shaft of high static stiffness. The tool can reduce much of the time and efforts to perform the FEAs by automation. However, an optimal number of bearing supports depends on objectives such as stiffness or manufacturing cost. An increase in the number of bearings leads to an increase in material cost and assembly cost leading to a higher manufacturing cost.

A near optimal design of the shaft can be identified based on stiffness with multiple design changes such as different sections or different locations of bearings. The tool can be useful for rapid evaluation of the stiffness of the shaft by automating the FEA process and, therefore, it enables even design engineers to make design changes and run FEAs with ease and, therefore, to perform design optimization based on an objective, say, stiffness.

4. Conclusion

In this research, a tool was developed to automate finite element analysis of a machine tool spindle designed in three dimensions. It was implemented in the programming environment of a CAD system with use of its objects. It requires a design engineer to input minimum data to run FEAs. They are the point positions to define the section of the spindle shaft and bearing locations. The tool substitutes for an analysis engineer that usually perform the FEA on behalf of the design engineer.

Its modules take the responsibility to generate the geometric model in 3D and an FE model at the workbenches of the CAD system to perform the FEA. Graphic user interfaces were developed for the tool interact with users. They are connected with the modules to exchange the data used for the FEA.

This tool can be used for identification of a near optimal design of the spindle based on an objective such as static stiffness or mass. The optimization can be implemented by repeating design changes and FEAs for the spindle. It provides rapid evaluation of each design requiring the minimum geometric data and, therefore, allows the section of the spindle shaft and the position of the bearings to be designed to be near-optimal based on, for example, high stiffness.

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.