한국기계가공학회지, 제18권, 제6호, pp.19∼25(2019.06) ISSN 1598-6721(Print)
Journal of the Korean Society of Manufacturing Process Engineers, Vol. 18, No. 6, pp.19~25(2019.06) ISSN 2288-0771(Online)

https://doi.org/10.14775/ksmpe.2019.18.6.019

# Development of a Tool for Automation of Finite Element Analysis of a Shaft-Bearing System of Machine Tools

Jin-Woo Choi[*,#], Gi-Young Kang[**]
[*]Department of Mechanical Computer-Aided Design, Korea Polytechnics VII,
[**]Dong-In High School

# 공작기계 회전축-베어링 시스템의 유한요소해석 자동화를 위한 툴 개발

최진우[*,#] 강기영[**]
[*]컴퓨터응용기계설계과, 한국폴리텍VII, [**]동인고등학교

(Received 29 April 2019; received in revised form 7 May 2019; accepted 12 May 2019)

## ABSTRACT

We have developed a tool that uses finite element analysis (FEA) to rapidly evaluate a shaft-bearing system of machine tools. We extracted commercial data on suitable clamping units and defined the inner profile of the shaft to avoid needing direct user input to define the profile. We use a splitting algorithm to convert the shaft into beam elements with two diameters and length. To validate the tool, we used it to design and evaluate a shaft-bearing system and found that our tool automated the construction of an FE system model in a commercial FEA package as well as the static stiffness evaluation; both tasks were completed in seconds, demonstrating a significant reduction from the minutes normally required to complete these tasks manually.

**Key words :** Finite Element Analysis(유한요소해석), Tool Development(툴 개발), Shaft-Bearing System(회전축-베어링 시스템), Automation(자동화), Splitting Algorithm(분할 알고리듬)

## 1. Introduction

Modern businesses require high-speed, high-efficiency, and high-precision machine tools to produce machined parts with a high-quality finish at high volumes[1,2]. These needs have led to better-performing machine tools with improved dynamic stiffness, which requires high dynamic stiffness in the spindle and feeding system[2,3].

The spindle is a core component of machine tools and affects the cutting accuracy and efficiency[2,3]. The main components of a spindle are the shaft, bearings, and motor. Since the spindle stiffness is heavily affected by the shaft and the bearings[4], the shaft-bearing system needs to be designed for low mass and high stiffness.

The mass and stiffness of the shaft are

---

# Corresponding Author : jinwoochoi@kopo.ac.kr
Tel: +82-53-260-1173, Fax: +82-55-260-1175

determined by the thickness between the inner and outer diameters. The outer diameter is constrained by the diameter of the bearings while the inner diameter is limited by the clamping unit configuration[5]. The bearing position also strongly affects the stiffness of the shaft-bearing system.

Mechanical engineers spend a great deal of time and effort to optimize the shaft-bearing system. The design process typically involves designing a candidate system, converting the system into an analytical model, and evaluating the stiffness. The engineer repeats this process until a satisfactory design is identified, which requires design and evaluation expertise. Designers determine the internal configuration of the shaft with a clamping unit and typically evaluate stiffness using finite element analysis (FEA) since the system with bearings is statically indeterminate[2].

To reduce cost and improve cycle time, engineers need a tool to rapidly design and evaluate different systems. Although previous researchers have developed automated FEA[1] tools, that software still required manual input to definite the shaft, which is time-consuming and requires knowledge of the internal configuration of the clamping unit; an impractical configuration leads to an unacceptable shaft design.

In our research, we developed a tool to rapidly design and evaluate a shaft-bearing system. We extracted the data we needed from a commercial source[5] and implemented the different definitions of inner shaft profiles. We use an algorithm to divide the shaft into the beam elements, including inner and outer diameter and length, which allows automated construction of the finite element (FE) model and automated evaluation of static stiffness in the commercial FEA tool ANSYS.

We developed the tool using Visual Basic .NET (VB)[7] and APDL (ANSYS Parametric Design Language)[6]. VB provides the programming environment to develop the graphical user interface (GUI) and implement the shaft division algorithm; the GUI allows a user to interact with the tool. We used APDL to implement the automatic construction of the FE model in ANSYS.

Our tool enables the user to rapidly find a near-optimal design of the shaft-bearing system by designing and evaluating many systems in seconds. Another advantage of our tool is that it prevents human error in the shaft design since we constrain the design to feasible commercial inputs and construct the FE model with a beam generation algorithm.

## 2. Automated tool development

### 2.1 Shaft definition

For the tool we describe in this paper, we used commercial data[5] to define the shaft of the spindle system. Fig. 1 shows the configuration of a typical built-in motor spindle system, which is composed of a shaft, a built-in motor with a rotor and a stator, a set of bearings, and a clamping unit. The inner configuration of the shaft is determined mainly by the clamping unit while the outer configuration is determined mainly by the bearings. We extracted data for the tool holder type, HSK A, with sizes of 50, 63, 80, 100, and 160 from engineering drawings of the clamping units[5]. We implemented this data in our tool to automatically definite the shaft profile.

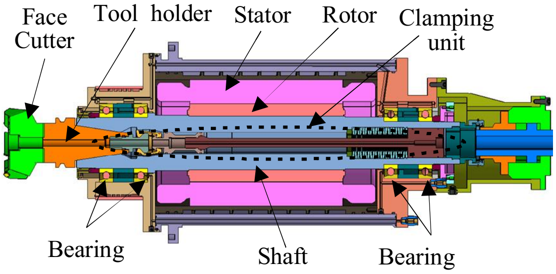


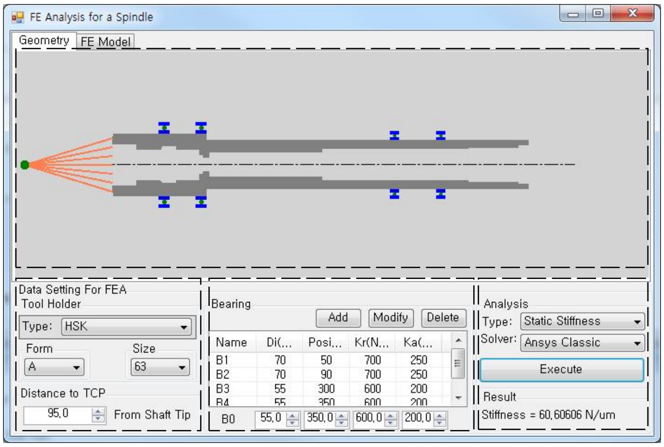
**Fig. 1 Configuration of a spindle system**

## 2.2 GUI development

We developed a GUI to allow users to interact with the tool. The main window, shown in Fig. 2(a), provides options to execute the design and evaluate the defined spindle. The other window, shown in Fig. 2(b), presents the diameters and lengths of beam elements. The user can check the finite elements almost in real time as our tool rapidly converts the shaft into the corresponding beam elements.

The main window contains four panes. The first pane shows the section view of the shaft and bearings. The second pane shows the tool holder and allows the user to select the size and distance from the shaft to the tool center point (TCP) for the imposed cutting force. The third pane allows the user to define the inner diameter ('Di' in the GUI), position, radial stiffness ('Kr'), and axial stiffness ('Ka') of the bearings. The last pane runs the FEA and displays the results.

The inner profile is based on the manufacturer-provided clamping unit and the outer profile is determined by the size and position of the bearings defined by the user, since the outer diameter varies according to the bearing diameter. The length of the shaft should accommodate the

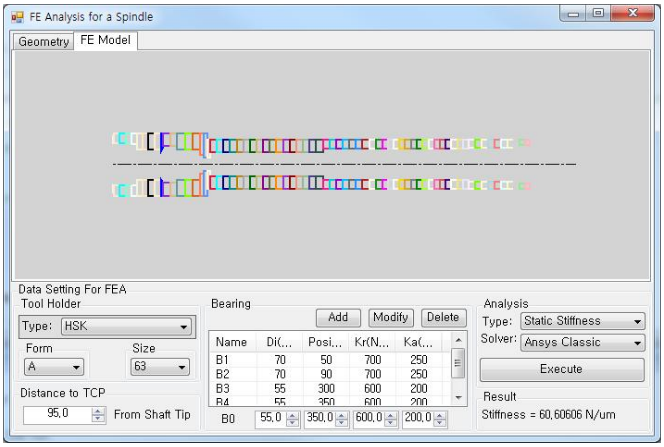


Type: HSK
Form: A
Size: 63
Distance to TCP: 95,0 From Shaft Tip

| Name | Di(... | Posi... | Kr(N... | Ka(... |
|---|---|---|---|---|
| B1 | 70 | 50 | 700 | 250 |
| B2 | 70 | 90 | 700 | 250 |
| B3 | 55 | 300 | 600 | 200 |
| B4 | 55 | 350 | 600 | 200 |
| B0 | 55,0 | 350,0 | 600,0 | 200,0 |

Type: Static Stiffness
Solver: Ansys Classic
Stiffness = 60,60606 N/um

(a) Main GUI for a shaft-bearing system



Type: HSK
Form: A
Size: 63
Distance to TCP: 95,0 From Shaft Tip

| Name | Di(... | Posi... | Kr(N... | Ka(... |
|---|---|---|---|---|
| B1 | 70 | 50 | 700 | 250 |
| B2 | 70 | 90 | 700 | 250 |
| B3 | 55 | 300 | 600 | 200 |
| B4 | 55 | 350 | 600 | 200 |
| B0 | 55,0 | 350,0 | 600,0 | 200,0 |

Type: Static Stiffness
Solver: Ansys Classic
Stiffness = 60,60606 N/um

(b) GUI for beam elements

**Fig. 2 Graphic user interfaces**

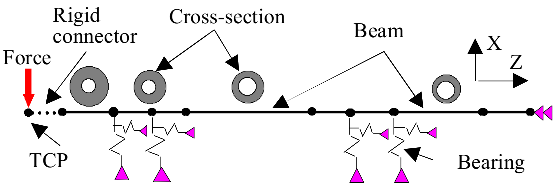


**Fig. 3 Analytical model for the shaft-bearing system**

clamping unit and the bearings.

## 2.3 Analytical model implementation

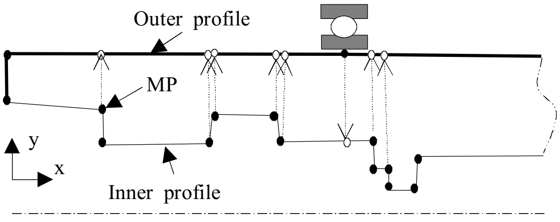
The FE model, shown in Fig. 3, is based on the models from Ref. 1 that were developed for a one-dimensional FE model of a spindle. Our tool constructs an FE model for the shaft-bearing system in Fig. 1(a) using the code developed in APDL to construct the FE model in ANSYS. For the shaft, we construct beam elements with length and tubular cross-section and, for the bearings, we construct spring elements in X, Y, and Z directions. This model sets up a TCP node where the cutting force is imposed and connects the TCP with the tip of the shaft using a rigid coupling element.

The FE model includes boundary constraints of the spindle system in all six degrees of freedom (DOF). One constraint attaches to the end node of the spring elements in both translation and rotation for all six DOF while the other constraint attaches to the end node of the shaft in rotation about the Z axis.

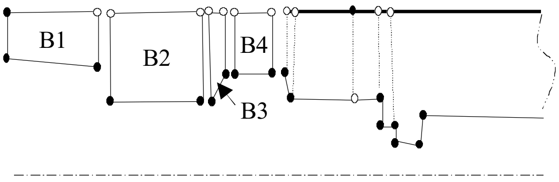
We chose to model the shaft with Crome-Mo steel, which is normally used for the shafts of similar precision systems[1]. The distance from the shaft tip to the TCP is 95mm for a face cutter using the HSK-A-63 tool holder.

## 3. Modeling a shaft with beams

The shaft is defined by the inner profile and the outer profile, as shown in Fig. 4. The inner shaft is based on the manufacturer's engineering drawing of a clamping unit suitable for the spindle system. The outer shaft definition uses the bearings chosen by the user.



(a) Shaft profile (●: Old point, ○: New point)



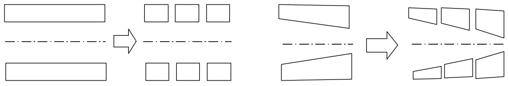
(b) Quadrilateral bars (B: Bar)

**Fig. 4 Division of the shaft profile into quadrilateral bars**

The outer and inner profiles start at the same X position. Successive points follow the inner radius of the bearing position in the Y direction and the distance from the shaft tip in the X direction. The profile ends at the starting X position of the inner profile and, in the Y direction, ends at the inner radius of the last bearing. The points are connected together to define the outer profile.

We developed a splitting algorithm to convert a shaft into beam elements. Fig. 3(a) shows how we define the shaft as inner and outer profiles with points. To address areas without any available points, we extrapolate a new point from an existing point on the opposite profile in the same X direction. We repeat this process until we reach the last point of each profile.

Fig. 3 (b) shows how we construct quadrilateral bars from the four vertices and create adjacent bars. A point like MP, for example, demonstrates how we neglect two points at the same X position



(a) Long bar (b) Non-rectangular bar

**Fig. 5 Division of a bar into small beams elements**

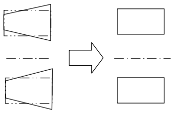


**Fig. 6 Conversion of a varied bar into a beam element**

when defining the position of a quadrilateral bar like B2. We repeat this process if the bar contains the end point of each profile.

Our tool splits long or diameter-varied bars into small-length bars as shown in Fig. 5(a) and 5(b). Bars that are too long are split further and, if the user varies the diameter of a bar, the bar is split into at least three small bars unless its length is too small. A small rectangular bar is directly converted into a beam element using the inner and outer diameters and the length. A varied bar uses the average of the starting and ending outer and inner diameters, as shown in Fig. 6.

## 4. Application of the automation

We validated the tool developed in this research by analyzing a shaft-bearing system. We used four bearings for the shaft-bearing system of the tool holder, HSK-A-63, as shown in Fig. 2(a). The data in Table 1 shows the bearing parameters. The tool automatically constructed an FE model of the system in ANSYS, performed the FEA, and extracted the stiffness from the FEA output into the GUI in Fig. 1(a).

The tool only required 4.38 seconds for one FEA, demonstrating that it can successfully perform rapid evaluations of the shaft-bearing system and help to identify a near-optimal design out of numerous design candidates. The FE model, shown in Fig. 7, has 83 nodes and 82 beam elements for the shaft, as shown in Fig. 8, and for the bearing spring type. We applied boundary constraints to the end of the springs and the end of the shaft to restrain the six DOFs.

**Table 1 Data for the bearings**

| Bearing | Di (mm) | Position (mm) | Kr (N/µm) | Ka (N/µm) |
|---|---|---|---|---|
| B1 | 70 | 50 | 700 | 250 |
| B2 | 70 | 90 | 700 | 250 |
| B3 | 55 | 300 | 600 | 200 |
| B4 | 55 | 350 | 600 | 200 |

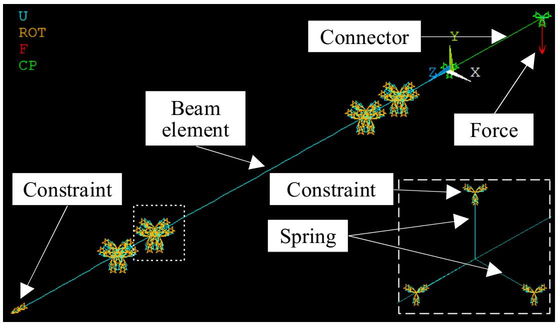


**Fig. 7 FE model for a shaft-bearing system**

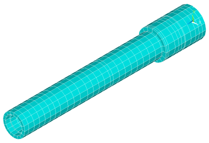


**Fig. 8 Shaft made of beam elements**

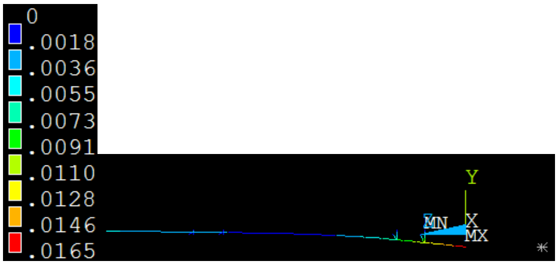


**Fig. 9 Displacement of FEA for the shaft-bearing system**

Fig. 9 presents the FEA spindle displacement of the FE model shown in Fig. 7. With a force of 1,000N

Table 2 Data for definition of the first bearing

| Design candidate | Di (mm) | Position (mm) | Kr (N/μm) | Ka (N/μm) | Stiffness (N/μm) |
|---|---|---|---|---|---|
| 1 | 70 | 50 | 700 | 250 | 60.6 |
| 2 | 70 | 45 | 700 | 250 | 64.6 |
| 3 | 70 | 55 | 700 | 250 | 58.5 |
| 4 | 75 | 50 | 700 | 250 | 62.2 |
| 5 | 65 | 50 | 700 | 250 | 58.6 |
| 6 | 70 | 50 | 800 | 250 | 71.1 |
| 7 | 70 | 50 | 900 | 250 | 77.2 |

applied to the TCP when connected to the shaft tip, the model predicted a maximum displacement of 0.0165mm at the TCP. The static stiffness calculated from this force and displacement is 60.6N/μm.

We created several design candidates (see Table 2) and evaluated static stiffness to identify the effects of bearing parameters such as the inner diameter. The second bearing’s diameter and the stiffness parameters (Kr, Ka) are the same as the first one since they make a set.

The design candidates 1, 2, and 3 in Table 2 show that static stiffness of the system is maximized when the first bearing is located close to the shaft tip. As the first bearing and shaft diameter is increased, the stiffness only increases slightly. Likewise, although an increase in the diameter from 70mm to 75mm led to 1.6N/μm increase, the larger diameter increases the mass of the shaft and negatively affects dynamic stiffness. The radial stiffness of the bearing, however, has a large influence on the system stiffness: increasing the bearing stiffness by 14.3%, from 700N/μm[8] to 800N/μm, raised the static stiffness by 17.3%, from 60.6N/μm to 71.1N/μm. The increase in bearing stiffness can be obtained computing the increase of the bearing preload after assembly.

## 5. Conclusion

In this paper, we describe a software tool that rapidly designs and evaluates a shaft-bearing system within seconds. To ensure that the tool uses feasible parameters for a suitable clamping when it automatically defines the inner profile of a shaft, we extracted and implemented commercial data in the tool; this built-in intelligence should prevent users from designing impractical shafts. We also developed a splitting algorithm to divide the shaft into small segments that can be readily converted into beam elements, defined only with the two diameters and length. When we applied the tool to design and evaluate a four-bearing system, the tool only required 4.38 seconds to automatically construct an FE model in ANSYS and evaluate its static stiffness.

We summarize our findings below:

1) The tool can rapidly design and evaluate a system
2) The tool facilitates the design and evaluation, even for novice engineers, by implementing commercial data before constructing the FE model
3) The tool can help identify a near-optimal design of the system based on the static stiffness
4) The radial stiffness of the first and second bearings has a greater influence on the static stiffness of the system than the location of the first bearing or the shaft diameter

## References

1. Choi, J. W., “Automation of One-Dimensional Finite Element Analysis of a Direct-Connection Spindle System of Machine Tools Using ANSYS”, Journal of the Korean Society of Manufacturing Process Engineers, Vol. 14, No. 2, pp. 127-133, 2015.
2. Abele, E., Altintas Y. and Brecher C., “Machine tool spindle units”, CIRP, Vol. 59, pp. 781-802, 2010.
3. Weck, M., Handbook of Machine Tools, Vol. 1 (English translation), Ch. 1-3, John Wiley, New York, 1984.
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", Journal of the Korean Society of Manufacturing Process Engineers, Vol. 8, No. 1, pp. 10-17, 2009.

5. Ott-Jacob Gmbh & Co. Spanntechnik, Modular Clamping Technology, 2017 (Technical Brochure).
6. ANSYS Mechanical, ANSYS V14.5 User Manual, SAS IP, Inc., 2012.
7. Liberty J., Learning Visual Basic .NET, O'Reilly, 2003.
8. Lee, C. H. and Choi, J. W., "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, p. 228-234, 2013.