Method for automated building of spindle thermal model with use of CAE system

S V Kamenev

Orenburg State University, 13, Pobedy Av., Orenburg, 460018, Russia

E-mail: kamenev_sergey@mail.ru

Abstract. The spindle is one of the most important units of the metal-cutting machine tool. Its performance is critical to minimize the machining error, especially the thermal error. Various methods are applied to improve the thermal behaviour of spindle units. One of the most important methods is mathematical modelling based on the finite element analysis. The most common approach for its realization is the use of CAE systems. This approach, however, is not capable to address the number of important effects that need to be taken into consideration for proper simulation. In the present article, the authors propose the solution to overcome these disadvantages using automated thermal model building for the spindle unit utilizing the CAE system ANSYS.

1. Introduction

Machining accuracy of a metal-cutting machine tool is crucial for its quality. The machine tool accuracy is quantitatively characterized by the total machining error that consists of several components one of which is thermal error. Thermal error of machining is determined by the thermal deformations associated with redistribution of heat generated by the various sources during machine operating. According to different researches the thermal error can reach up to 70 % of total machining error [1-2] and have a major impact on the accuracy of machined workpiece during a finishing machining under conditions of the machine tool is not subjected by the large cutting forces.

At the same time one of the main sources of heat generation is modern spindle units with rolling bearings. Their speed is constantly growing and leads to significant heat generation in the bearings due to friction power losses. For this reason the machine tool manufacturers seek a way to estimate the thermal behaviour of the spindle unit on the design stage to minimize undesirable effects. An overview of related work has shown that the most common approach for such thermal behaviour preliminary estimation is computer aided engineering of a spindle unit using CAE systems based on the finite element method [1-4].

Generally the specified approach suggests execution of consistent steps that involves building a solid geometrical model for spindle unit, finite element discretization of this model, defining initial and boundary conditions, and performing calculations and analysis of the obtained results. The implementation of these steps, however, involves a number of problems, the most critical of which can be formulated as follows:

- Considerable labour cost for the preparation of the spindle mathematical model because of high complexity of initial and boundary conditions that require a large amount of preliminary manual calculations based on empirical formulas and reference data.

- Invariance of the boundary conditions imposed on the model that prevents scalability if the spindle performance parameters (for instance, rotational speed) related to the model are changed. That results in duplicating boundary conditions related work.
- High qualification of the computing engineer which in the circumstances of machine-tool building enterprise might not be sufficient to develop an adequate spindle model and full engineering analysis.

2. Modelling methodology

As a solution of the mentioned problems, a method for building the spindle thermal model based on the specialization of the CAE system ANSYS using scripted solution on built-in programming language APDL (Ansys Programming Design Language) can be proposed. In this case, the process of model preparation can be greatly simplified due to the automated execution of the sequence of steps shown in Figure 1.

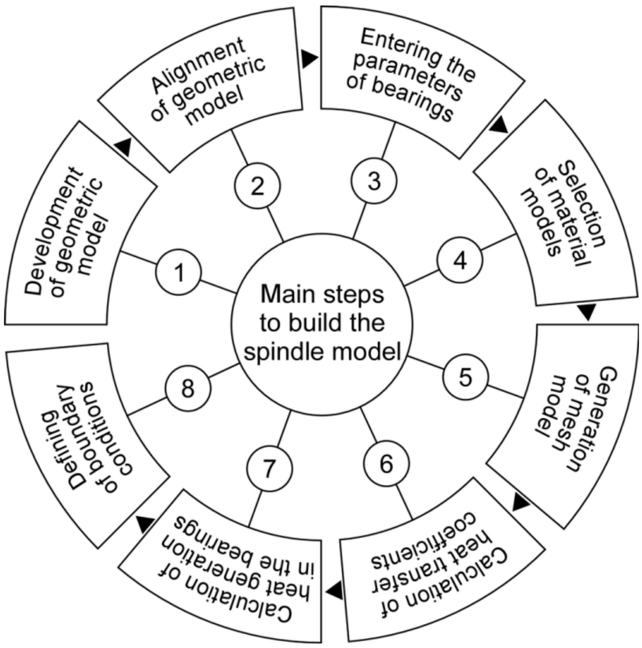

Figure 1. The sequence for building the spindle thermal model

As it shown in Figure 1, the beginning stage of modelling is a building of spindle geometry that can be obtained both by means of an ANSYS preprocessor and imported from an external CAD system.
The proper execution of the script assumes the use of the spindle assembly model that includes the solid models of the spindle and all parts and units directly linked to spindle (bearings, gears, bushings etc.). At the same time the model shall meet the following conditions:

- All components of the model must be the axisymmetric bodies of revolution that have common axis.
- The normal distance between the mated surfaces of components must be equal to zero.
- The geometry of the model for each of the rolling bearing must be formed solely by its ring which is mated with spindle upon the mandatory existence of a raceway on this ring.
- Only components with total area of their surfaces sufficient to affect the heat exchange between other components and the environment must be presented in the model.

For example, in the case of the spindle unit for machining center 400V (manufactured by JSC «Sterlitamak M. T. E») which design is shown in Figure 2, the corresponding geometric model that meets the specified requirements is shown in Figure 3. Both models are presented in these figures in the sectional view for a better understanding.

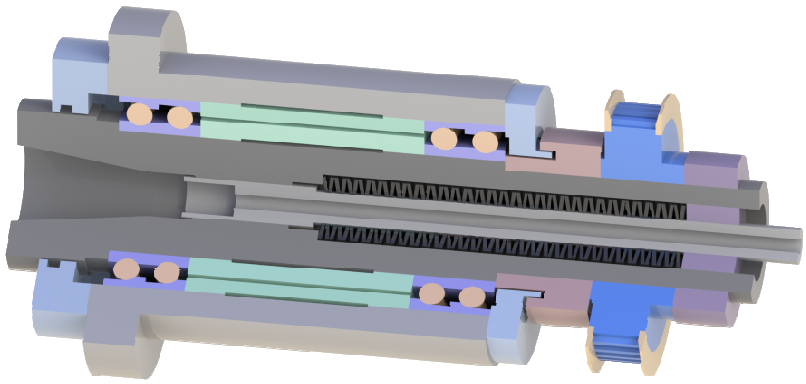

Figure 2. The design of the spindle unit for machining center 400V

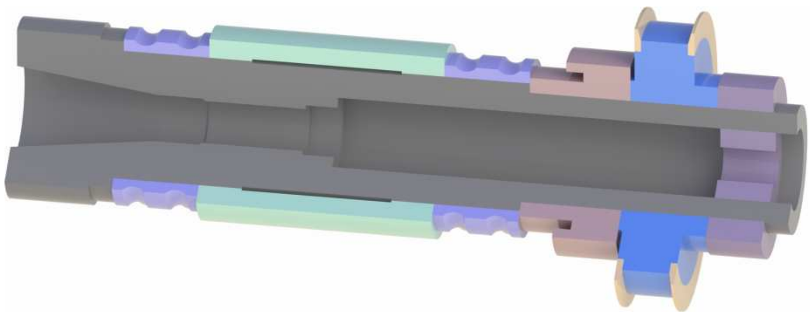

Figure 3. The geometric model of the spindle unit developed for thermal modelling

This geometric model allows one to apply the modelling script that automates all subsequent stages of modelling. The first stage is the procedure of model alignment, which consists of automatic assignment to certain position and orientation of the spindle in the current coordinate system, irrespectively to initial model configuration imported into the CAE system ANSYS (or created in it). The procedure is performed in a fully automatic mode and its basis is the calculation of mass center coordinates for all surfaces of the geometric model.

The further process of the model building is realized on the assumption that the joints between the mated parts of the spindle unit are ideal. For this reason, at the current stage of modelling an automatic Boolean «gluing» of the spindle's mated parts is performed. The result of this gluing is the surfaces that define boundary of two bodies and belonging simultaneously to both bodies. The assumption of ideal joints makes possible to significantly simplify the spindle model and provide acceptable simulation results.

In the next stage of the model building, it is required to perform interactively a number of actions that allows one to define the parameters of the rolling bearings. These actions include:

- Input of the total number of bearings mounted on the spindle.
- Interactive selection of the solid model component that corresponds to each bearing ring.
- Input parameters for each bearing which are the bearing type code, the bearing series code, the mean bearing diameter, the number of rows of rolling elements, the contact angle, the static load carrying capacity and the axial load.

In addition to the above-mentioned parameters, the full definition of the bearing also includes the value of radial force that is acting on the bearing. For radial and angular contact bearings this value is determined automatically by calculating the reactions in the spindle supports due to the radial cutting force. The radial load on the thrust bearings is assumed to be zero by default.

Calculation of the reactions is performed with the use of a special beam finite element model that is generated automatically according to the progress of script execution. The basis for its generation is the axial dimensions of the spindle unit. An example of such model obtained for the spindle design under discussion is shown in Figure 4.

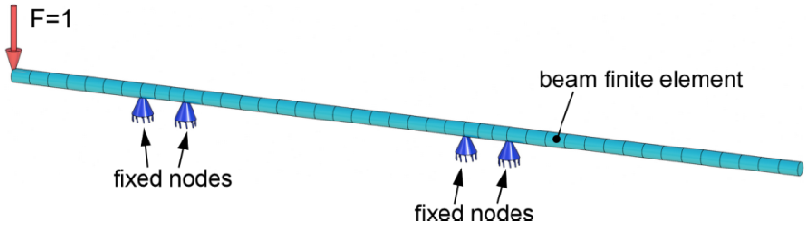

Figure 4. The beam model of the spindle used for the calculation of reactions in the bearings

The calculated reactions are stored in the model database as coefficients whose sum is unity. In the future, these coefficients are used to determine the actual radial loads acting on the bearings by multiplying these coefficients by a specified value of radial cutting force.

The next stage of the script involves the automatic determination of the material models that is characterizing the heat-transfer properties of the spindle parts. By default, ten material models are created with characteristics corresponding to the various grades of structural steel commonly used in the design practice of spindle units. Each material is assumed to be isotropic with temperature dependent thermal conductivity, density, and specific heat. Unique numerical code is assigned to the materials in the model for easier identification.

Special multistep interactive procedure is used to designate materials to the parts of the spindle unit. Each step includes the interactive input of the required material model number and subsequent manual selection of the corresponding parts of the spindle unit. Depending on the assigned materials, the parts are automatically stored in the named components that allow easy replacement of particular material.

The stage of the mesh model generation is implemented in a fully automated mode beginning with Boolean dividing the spindle geometry by plane passing through spindle axis. It results in set of flat surfaces that used as a basis for the generation of the 2D finite element mesh using a standard type Shell57 element that available in the CAE system ANSYS. The average size of the elements is estimated automatically on the basis of the total section area. This size is controlled by the corresponding parameters. The 2D mesh obtained for the spindle design under consideration is presented in Figure 5.

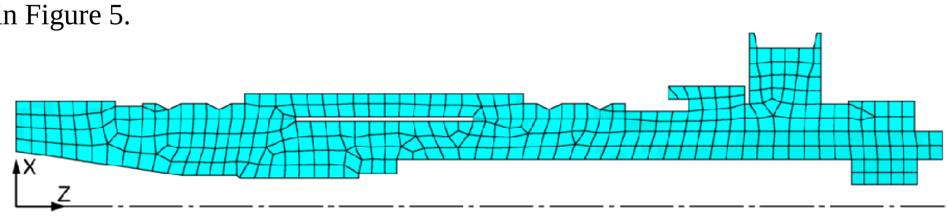

Figure 5. The 2D mesh in the axial section of the model

The generated 2D mesh serves as the basis for generating a 3D mesh of solid elements (Solid 70) by means of a sweeping around the spindle axis. To control the angular spacing of the mesh elements a special parameter is introduced. In the process of 3D mesh generation, the 2D elements are automatically deleted due to their further uselessness. In the present case the 3D mesh model of spindle will look like one shown in Figure 6.

Then, the block of statements within the script that are responsible for the calculation of the heat transfer coefficients on the free surfaces of the spindle is provided. Automatic identification of free cylindrical (conical) and end surfaces, where conditions for convective heat transfer must be specified, is performed before the direct calculation of the coefficients. Based on the results of the identification, two data sets with unique identifiers and geometric parameters of the surfaces are created in the model database. In the case of cylindrical (conical) surfaces, these parameters are the mean radius of their circular edges. In the case of the end surfaces – the inner and outer radius of their boundary circular edges.

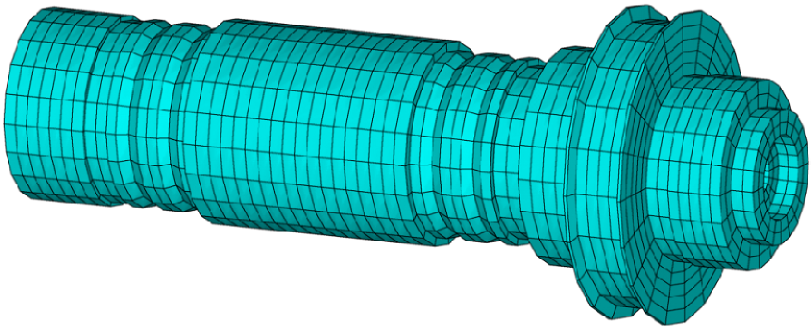

Figure 6. The 3D finite element mesh

The identified surfaces are used later for imposing boundary conditions that defines convective heat transfer to the environment. These conditions are defined quantitatively by the values of heat transfer coefficients that calculated automatically in the relevant block of the script.

For this purpose, the values of the spindle speed and ambient temperature must be entered into the model database in advance using interactive interface. The value of the ambient temperature also determines the default initial temperature of the spindle unit. Thereafter, with respect to the temperature specified for the environment (air) a set of parameters characterizing its thermal properties (density, kinematic viscosity, thermal conductivity, specific heat and coefficient of cubical expansion) is calculated. The automated calculation of their values is implemented based on the equations presented in the reference [5].

The calculated values, in turn, are used for the estimation of several similarity criteria, including the Prandtl number, the Reynolds number, the Grashof number and the Nusselt number. At the same time the values of the Nusselt number for cylindrical and flat end surfaces are calculated using the empirical formulas presented in the references [6-7]. On the basis of the obtained values for each heat sink surface of the spindle, the heat transfer coefficient is automatically calculated and is written to the relevant data array.

The next block of statements in the script is related to the identification of heat generating surfaces and the calculation of the heat fluxes. As the heat sources, the surfaces corresponding to the raceways of the bearing rings are considered. The search of these surfaces is performed in automatic mode on the base of consecutive exclusion of the surfaces belonging to the each model of the bearing ring.

The next step after the identification of the heat generating surfaces is the determination of heat sources parameters which are the rolling bearings. The starting point is to select interactively the type of lubrication system for bearings. The selection can be made from four systems (grease, oil mist, oil bath and oil jet) that are coded by the numerical identifiers.

Then, in accordance with the selected type of lubrication system the lubricant type is selected interactively. Selection of two lubricant types is possible: greases and liquid lubricants. Each type includes several existing lubricant specifications defined by entering the appropriate numeric code. Temperature dependence of the kinematic viscosity is considered as main characteristic of the selected lubricant. The values of this function are calculated automatically based on the properties of various lubricants and empirical formulas presented in the references [4, 8].

The obtained values are used for the calculating of heat fluxes in the bearings which is performed through the estimation of friction torque. At present, for the estimation of friction torque in the rolling bearing the different mathematical models are used, but the most common model is the Palmgren model [9] as proven by the review of various references.

According to this model the total friction torque is the sum of the two components, namely, load friction torque and viscous friction torque. Their values for each bearing are calculated automatically in the process of script execution on the basis of previously defined parameters using the formulas and reference data that presented in the reference [9]. The value of radial cutting force is interactively entered into the script for the calculation of radial loads affecting the bearings. After determining the values of total friction torques, the heat fluxes are calculated and stored to the relevant arrays as the values functionally dependent on temperature. The number of these arrays containing the values of heat load corresponds to the number of bearings.

At the final stage of the model building the prepared arrays with the values of heat transfer coefficients and heat fluxes are used for the automated generation of boundary conditions on the surfaces of geometrical model. During this process the interface surfaces of the mated parts are automatically excluded. This procedure completes the process of automated building of the spindle model. The obtained model can be used in further analysis of spindle thermal behaviour in the CAE system ANSYS.

3. Conclusions

The presented approach to build the thermal model of spindle units allows one to highlight the number of advantages that makes the approach superior to similar methods proposed by the other authors. They include:

- Minimizing the time taken to build the model, e.g., for the spindle unit under consideration, the total time of the model building is equal to about 3 minutes (when using the computer Intel Core i5 3.4 GHz, RAM 8 GB) while in the conventional approach it will be no less than a few hours.
- Exclusion of the errors related to the determination of heat transfer coefficients and heat fluxes for the plenty of surfaces that may occur when the conventional approach is used (preliminary manual calculation or with the involvement of auxiliary software tools).
- Avoiding the need to use the large volumes of the reference books required to define all required conditions for convective heat transfer and the heat-transfer properties of materials.
- As a consequence of three aforementioned items, the reducing demands qualification of the computing engineer conducting the thermal modelling of the spindle unit in the CAE-system.
- Possibility of quick modification of the obtained spindle thermal model for the realization of other important engineering analysis tasks, such as static, modal and harmonic analysis.
- Good agreement of modelling results obtained on the basis of the proposed approach with the results of experiments. For example, for the spindle unit being considered, the modelling error for temperature at the spindle nose that settled during 8 hours of its continuous work with a rotational speed equal to 3000 rpm did not exceed 4.5%.

References

[1] Deng X, Fu J, He Y, Chen Z and Lin F 2011 *Proc. of 10th Int. Symp. of Measurement Technology and Intelligent Instruments (June 29-July 2, 2011 Daejeon)* (Daejeon: Lab. for Precision Engineering and Metrology, KAIST) p 253
[2] Ramesh B S and Dhamotharan S 2014 Simulation of temperature distributions of CNC lathe headstock assembly *Int. J. of Mech. & Mechatronics Engr.* **14(6)** 99–103
[3] Li Y, Zhao W, Wu W and Lu B 2017 Boundary conditions optimization of spindle thermal error analysis and thermal key points selection based on inverse heat conduction *Int. J. Adv. Manuf. Technol.* **90** 2803–2812
[4] Kim J D, Zverev I A and Lee K B 2010 Thermal model of high-speed spindle *Intell. Info. Mgmt.* **2(5)** 306–315
[5] Dixon J C 2007 *The Shock Absorber Handbook* 2nd ed (Chichester: John Wiley & Sons, Ltd.) p 432
[6] Howey D A, Childs P R N and Holmes A S Air-gap convection in rotating electrical machines 2012 *IEEE Trans. Ind. Electron.* **59(3)** 1367–1375
[7] Kreith F, Manglik R M and Bohn M S 2011 *Principles of Heat Transfer* 6th ed (Boston: Cengage Learning, Inc.) p 696
[8] Knezevic D and Savic V 2006 Mathematical modeling of changing of dynamic viscosity, as a function of temperature and pressure, of mineral oils for hydraulic systems *Mech. Engr.* **4(1)** 27–34

[9] Harris T A 2006 *Essential Concepts of Bearing Technology* (Boca Raton: Taylor & Francis Group) p 376