Development and Validation of a Simulation Model for Thermal Analysis of Structure with Built-in Cooling Systems

[1]Siddharth Akki , [2]Balappa B U
[1,2]Ramaiah University of Applied Sciences, Bangalore, India

Abstract : A representative Spindle structure is fabricated wherein variable heat flux can be induced using a band-heater while coolant oil is circulated in the structure. Provisions have been made to alter various cooling parameters such as flow rate, coolant oil, cooling jacket and examine the efficiency of the cooling system. Also, a numerical model of the representative spindle structure is built using a commercial CFD software, Fluent, and validated against temperature measurements at critical points in the structure. Parametric study is then performed to decide on the optimal set of parameters which yield the highest cooling efficiency and reduce the required experimentation. The details of the experimentation, the validation of the numerical model results will be provided in the further in the paper.

IndexTerms **- Conjugated heat transfer; Spindle heat transfer; thermal analysis of spindle; High speed spindle**

INTRODUCTION

The present era of rapid development of technologies, there is an ever increasing need to manufacture products which are more accurate, takes less time to produce and achieve higher energy efficiencies. As far as metal cutting technology is concerned, this requirement can be realized through higher metal removal rate in precise machine tools, which in turn is possible through high speed machining. High speed machining not only improves the manufacturing efficiency but also leads to better surface finish and in general results in lower cutting forces. From a machine tool designer perspective, high speed machining calls for a combined high speed-torque characteristics of the motor. Although motors with gear arrangements can be used, a motorized spindle (with a built-in motor) eliminates the need for power transmission elements in addition to providing better stability at high speeds. Thus, motorized spindles have now become an industry-accepted norm for high speed machining. Motorized spindle is one of the critical sub-assemblies of a machine tool and its mechanical design is quite complicated as separate provisions need to be made to cool the spindle from heat generation through motor in addition to the bearings. This heat generation is typically non-uniform along the spindle shaft and results in axial and/or radial thermal distortion. In order to achieve higher accuracy and repeatability of machined product, the thermal distortion of the spindle shaft needs to be regulated, which can be partly achieved through the optimization of the cooling system. However considering the complexity of a typical motorized spindle, the present work is aimed at cooling system optimization of a simpler representative structure in a mixed experimental-numerical framework.

Literary survey

Studies has been carry out to understand the how thermal characteristic of spindle is evaluated through experiment and numerical method, and some of them are listed below. J.F. Zhang, P.F. Feng, Z.J. Wu, D.W. Yu and C. Chen [1] presented thermal analysis in the tool head stock with the spindle structure. The model accounted of conducting the thermal structure optimization and controlling temperature levels, it can increase the machining accuracy and reduce the thermal deformation. Further Syath Abuthakeer.S, Mohanram P.V and Mohan Kumar G [2] explained about the dynamic and thermal analysis of high speed motorized spindle, the thermally induced preload has been estimated from the thermal analysis for 10000 rpm with corresponding stiffness has been studied. C.H. Chien, J.Y. Jang [3] analyzed numerically and experimentally the 3D fluid motion and distribution of temperature in the motorized spindle in a channel. Extension of the C.H. Chien, J.Y. Jang [3] work M.Karthikeyan[4] has explained about the CFD analysis of cooling channels of high speed motorized spindle with built in cooling system. J.S. Jayakumar [5] and Shiva Kumar, K.Vasudev Karanth[6] explained about the difference in the helical coil and straight coil of coolant passage, in order to obtain the high heat transfer per unit volume and to enhance the heat transfer coefficient on the inner surface. In the process the outcome indicated that helical coil showed 11% rise in the heat transfer rate than the straight coil. Then Nusselt number increased by 10% in the simulation, whereas there is pressure drop in coils shape of helical than straight coil.

Mathematical Analysis

The model of the representative spindle structure with the built in cooling system is shown in the Fig. 1 The distribution of the heat per unit volume is at the center of the sleeve at two different places of length 20mm with equal distribution of heat. The structure is operated at ambient temperature at Ambient = 23°C to 27°C. Hence heat is removed by two major type’s i.e. natural convection and by forced convection of the spin oil 12 coolant. There are some assumptions made to reduce the efforts:

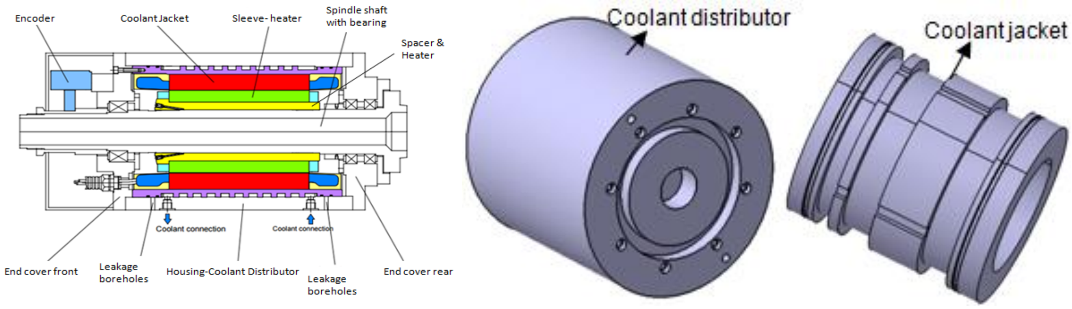

Fig. 1 High speed spindle with built in cooling system

- The effect of radiations are ignored
- The fluid is assumed to be incompressible with constant physical properties
- The flow is considered to be turbulent with no viscous dissipation and with the effect of turbulent intensity
- The material of structure is assumed to be isotropic and independent of temperature
- The effect of heat generation is assumed to be constant during the experiment

Numerical methods

In the present work the model is create using Catia software. Grid generation using HyperMesh with the lower level geometry i.e. bottom-up approach is used. It is mainly divided into two steps, in the initially step face mesh is created and later the transformation of those mesh is into volume mesh. Finite volume method is developed using elements with the base size of 6mm. Further with the solver settings provided. In the first stage, face mesh is applied using the tri elements as 2D mesh. Further and unstructured grid is used to generate triangular elements with pre-defined element size. Quality check is employed on the generated mesh to check the validity of the mesh. Flow and turbulent governing equation is solved using SIMPLE algorithm under pressure-velocity coupling. Solution convergence is controlled using default value of under relaxation factors and discritization methods. Convergence criteria part is traced in the different monitor controls to study the variation of solution. Results of control numerical transformation of the governing equations are converted into simulations in the post processing stage.

Result of flow analysis is discussed in the next chapter. Iteration is repeated for varying velocities to study the fluid flow behaviour and determine the region of critical velocity.Boundary conditions used for the model are as shown in the Table 1. 0.0657 Kg/s corresponds to 5 liters/min with the fluid density of 889 Kg/m-3. Further the analysis is carried out with 15lts/min with the 225W and 460W.

Table 1 Boundary conditions

Boundary condition & type	Magnitude	Turbulence parameters	
		Turbulence Intensity %	Hydraulic Diameter
Inlet- Mass flow rate	0.0657 Kg/s	4.123	10 mm
Outlet- pressure	101325 pascals	4.123	10 mm
Heat transfer co-efficient	15 (approximated)	--	

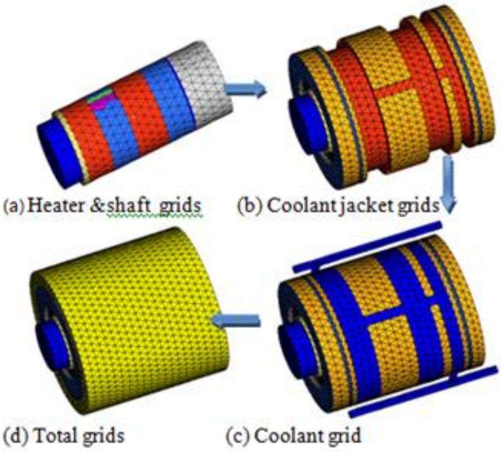
(a) Heater &shaft grids
(b) Coolant jacket grids
(d) Total grids
(c) Coolant grid

Fig 2 Computational grid system

Experimental set up

The schematic diagram of the experimental set up of the representative model of the spindle structure is as shown in the Fig 3. The spindle housing is heated by the heater which is a representative model of the bearing and motorized spindle. The heaters are shown in the Fig. 3 inserted around the shaft at the centre region at the inner walls of the sleeve of the spindle housing. The temperature distributions of the structure were locations embedded in the spindle along the axial direction and inclined to touch the sleeve, the location of the RTD sensors are decided based on the critical temperature generation in the system. The outer surface of the structure is exposed to the ambient temperature were heat transfer coefficient is calculated. Electric power was supplied to the cable by the heater source multi-output DC power supply. The working medium of the chillier unit is spin oil-12 which is near to ambient temperature. The oil temperature is controlled by the chillier unit which will vary of ± 3 °K as the chiller unit gets start when the temperature increase above the ambient temperature. Both inlet and outlet temperature are measured with the RTDs with the accuracy of ±0.05 K. Volumetric flow rate is measured in liters/min at the inlet of the coolant jacket. All the data are collected manually and taken in the data entry sheet. The whole set up is as shown in the Fig 3.

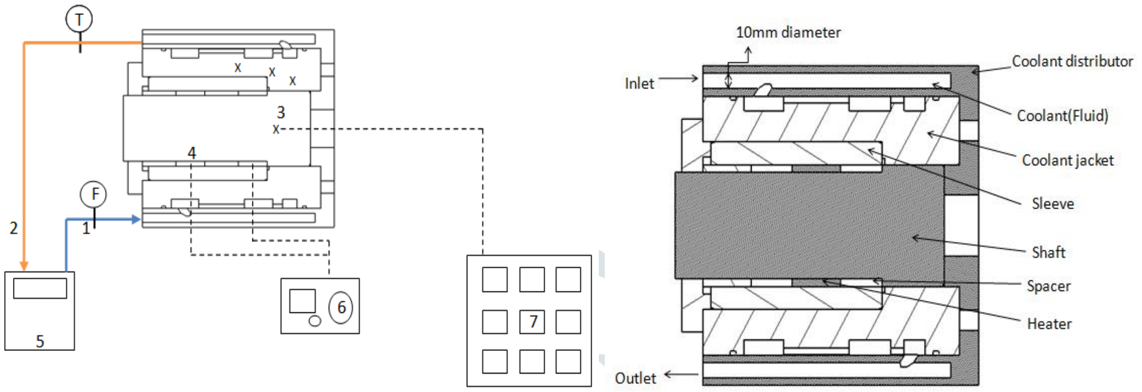

Fig 3 Schematic diagram of the experimental set up

1. Coolant inlet F: Flow measuring section
2. Coolant outlet T: Temperature measuring section
3. RTD sensors 4. Heaters
5. Chiller unit 6. Power supply regulator
7. Digital temperature indicators

Results and discussion

Numerical analysis of the velocity, temperature, pressure, for the coolant with heat source of 225W, mass flow rate of 0.22 Kg/sec, the temperature of the fluid at inlet is 300 °K and pressure of STP 101325 Pa. The coolant passed is at the temperature of 300 K, heat transfer coefficient as a boundary exposed is assigned 10-25 W/m^2.In the structure there is only one inlet and one outlet, but the outlet can be diverted into two different locations. The analysis result shows that heat is majorly concentrated at the center of the spindle. This is due to the fact that the heat source of the heat generation i.e. bearing in real application is connected to the spindle. The results of the heat transfer is as given in the Fig. 4.where the maximum temperature has reached to 312.07 °K from 296 °K and the maximum temperature is near the source and spindle center. Thus it shows that increasing heat source also increases the spindle structure temperature. The Fig. 5. shows contour with the inlet mass flow rate, as 0.22 Kg/s. Velocity is high at the place of divergence from the inlets and as the fluid particle enters the pipe of diameter of 10mm, it gets energized by the varying cross-sectional area of pipe and results with generation of high velocities at the sharp edges or sharp counters. With 0.22 Kg/s mass flow rate, the fluid particles enriches to 1.86 m/s of velocity. Further the maximum pressure at the inlet is 5808 Pascal and the pressure reduces at the outlet as it will be exposed to the ambient pressure mentioned which is as shown in the Fig 5. The results indicated that the temperature distribution is in good agreement with the experimental results up to 6%.The present design of the inlet is inclined in an angle of obtuse to the inlet direction of the flow of the fluid which causes the fluid to reverse back at the starting of the channel. This can be reduced by changing the angle of the inlet to acute angle as shown in the Fig 6.To carry out parametric study to increase the cooling efficiency some of the parameters are been considered which can change the cooling rate effectively without changing design of the structure.

Mass flow rate: The effect of change in mass flow rate on the outlet temperature has been shown in the Fig 7. It can be clearly observed that as the mass flow rate of the fluid is decreased the outlet temperature is increased

Fluid Change: Winsorlube- L245X has high change in temperature for inlet to outlet which means it can permit high heat transfer which is as shown in the Fig 7.

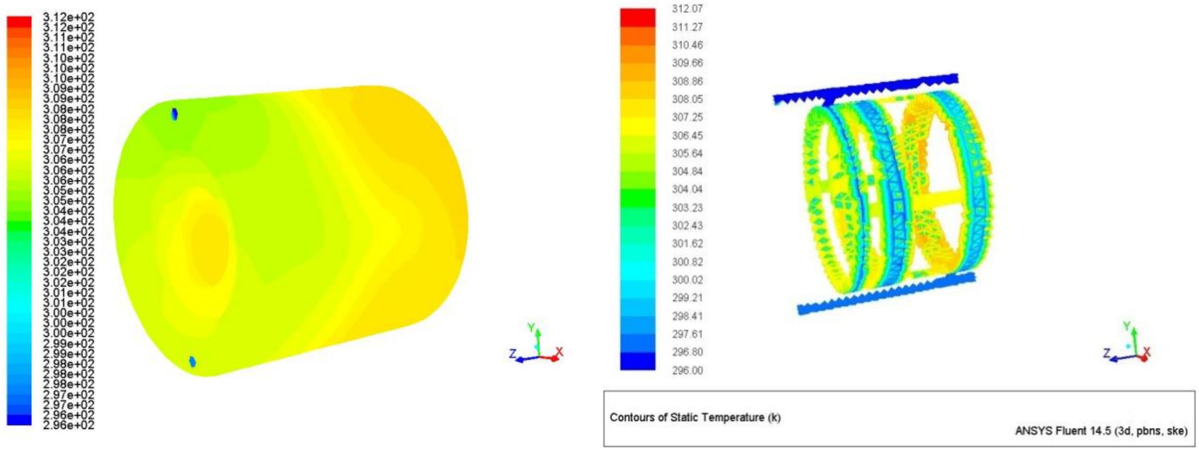

Fig. 4 Temperature profiles

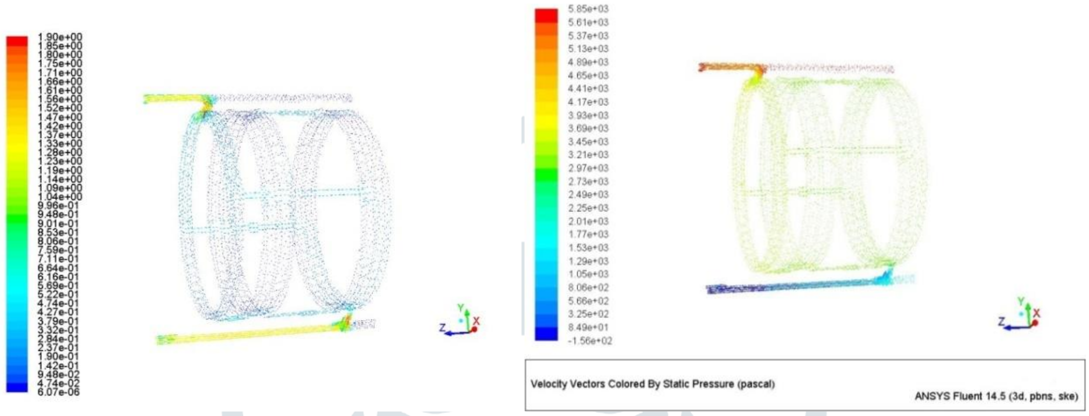

Fig. 5 Velocity and pressure profiles

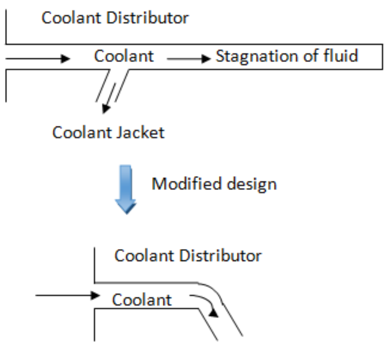

Fig. 6 Change in the design of flow path

Table 2 Comparisons of Experimental and Numerical results

Sensors Numbers	Results		Error In %
	Experimental	Numerical	
1	34.6	36.43253	5.29633
2	34.8	36.43732	4.70494
3	34	35.54471	4.54326
4	34.1	36.23969	6.27475
5	33.4	34.02734	1.87826
6	33	34.60782	4.87218
7	33.7	34.88629	3.52015
8	33.8	34.91214	3.29036
9	33.5	34.2009	2.09224
10	34.7	34.90338	0.58611
11	34.7	35.28848	1.69591
12	35.9	35.29691	1.679916
13	37.5	35.40881	5.5765
Inlet	23	23	-
Outlet	24.3284	25	2.79509

Table 2 shows temperature results of the experiment and simulated model in Fluent which is for 225W and 5 liters/min. Further same procedure is followed to carry out validation for 255 W and 15 litres/min, 460 W and 5 litres/min, 460 W and 15 litres/min

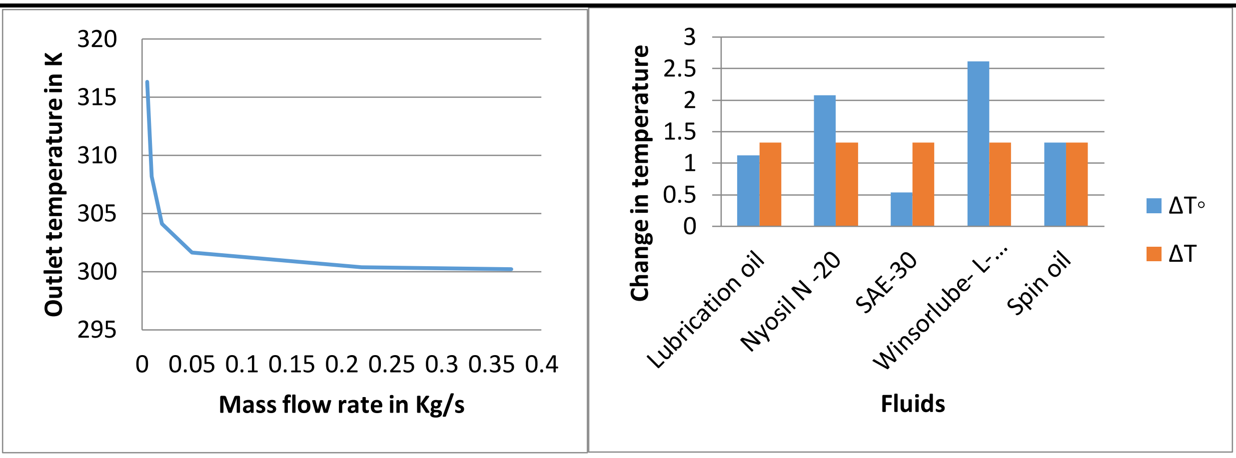

Fig. 7 Change in mass flow rate and various fluid effects

Conclusions

One of the main adversaries in machine tools is heat generation. A prime source for generating heat in a machine is machine tool spindle. Before integral spindle, most of the spindles were driven by either gearboxes or belts to a motor. Integral spindles are used on machining centres, grinding centres and most other machine tool variations. In precision machining applications, integrating the electric motor into the spindle creates a heat source which must be controlled. The cutting tool's position is affected when the spindle dimensions change due to thermal growth as the cutting tool is in direct contact with the spindle. Fluid as the cooling medium is used around the outside of the housing in integral spindles as cooling jacket to reduce the heat distribution. It was demonstrated that heat was concentrated near the spindle centre. The effect of different heat source and flow rate of 255 W and 5 liters/min, 255 W and 15 liters/min, 460 W and 5 liters/min, 460 W and 15 liters/min respectively are examined in details. The results indicated that the temperature distribution is in good agreement with the experimental results up to 6%. It also shows that the change in flow rate, different fluid and properties, viscosity of the fluid significantly affects the heat transfer in the structure. It was observed that decrease in the flow rate and viscosity of the fluid there is increase in temperature difference between inlet and outlet of the coolant.

References

[1] J.F. Zhang, P.F. Feng, Z.J. Wu, D.W. Yu and C. Chen “Thermal structure design and analysis of a machine tool headstock” Mechanika Vol 19(4), 2013 pp478-485.

[2] Syath Abuthakeer.S, Mohanram P.V and Mohan Kumar G “Dynamic and thermal analysis of high speed motorized spindle” International Journal of Applied Engineering Research (Dindigul) Vol 1, No 4, 2011 pp864-882.

[3] C.H. Chien, J.Y. Jang “3-D numerical and experimental analysis of a built-in motorized high-speed spindle with helical water cooling channel” Science Direct: Applied Thermal Engineering Vol 28, 2008 pp 2327–2336.

[4] K.MadhanMuthuGanesh, Dr.A.K.Shaik Dawood, N.V.Kamalesh and M.Karthikeyan [“CFD Analysis of cooling channels in built-in motorized high speed spindle” Engineering Science and Technology: An International Journal (India), Vol.2, No. 2, April 2012 pp238-244.

[5] J.S. Jayakumar, S.M. Mahajani, J.C. Mandal, P.K. Vijayan, Rohidas Bhoi”Experimental and CFD estimation of heat transfer in helically coiled heat exchangers” ScienceDirect : Chemical Engineering Research and Design Vol 86, 2008 pp221–232

[6] Shiva Kumar, K.Vasudev Karanth “ Numerical Analysis of a Helical Coiled Heat Exchanger using CFD” International Journal of Thermal Technologies(India) Vol.3(4), Dec 2013 pp126-130