Special Interest Group Meeting on Thermal Issues
Laboratory for Machine Tools and Production Engineering (WZL) of
RWTH Aachen, Germany, February 2020
www.euspen.eu

Multi-phase numerical simulation of coolant flow around the cutting tool

Michael Bräunig[1], Lukáš Topinka[1], Joachim Regel[1], Matthias Putz[1,2]

[1] *Chemnitz University of Technology, Institute for Machine Tools and Production Processes, Professorship Machine Tool Design and Forming Technology, Chemnitz, Germany*
[2] *Fraunhofer Institute for Machine Tools and Forming Technology IWU, Chemnitz, Germany*

michael.braeunig@mb.tu-chemnitz.de

Abstract
The thermal behavior has a significant effect on the manufacturing accuracy. For many machining tasks, the use of cooling lubricant is essential in order to achieve possible cooling and lubricating effect in the cutting zone. Its reduction due to resource-efficient or ecological efforts has an impact on the temperature field in the machining tool, which affects the thermal behavior of the frame structures and can cause machining inaccuracies. This article describes the CFD-model for simulation multi-phase flow of liquid coolant around the cutting tool and the prediction of the temperature field. With experimental investigations, the simulation model is verified and checked for plausibility.

Keywords: CFD-Simulation, thermal investigation, machine tools, tool cooling simulation, multi-phase simulation

1. Introduction

The accuracy of metal cutting machine tools is essential for most of the industrial production. The majority of materials used for machining application are negatively affected by temperature in terms of both tool life and manufacturing accuracy. Coolants are very common in this field but according to an industry survey, at least 50 % of enterprises also use dry machining [1]. Previous studies on effects of air cooling on the tool structure [2,3] shows that the temperature field in the tool with air cooling can be reduced by up to 50% in comparison to machining without any coolant. But for many manufacturing operations are coolant lubricants are still indispensable, as they provide not just a cooling effect in the cutting zone, but also less friction and adhesion between tool and workpiece during cutting operations. Delivery of the liquid coolant into the inner part of tool teeth and the cutting zone can be difficult due to high pressure and high rotational speed of tool as consequence of highspeed machining. Moreover, depending on the tool rotation speed, a significant concentration of fog particles can be evaporated into the environment, which can increase health risks [4] and dangerous for the environment. Therefore, the investigation with focus on their effect and efficiency is necessary.

This work describes the multi-phase simulation model for system consisting of restricted area and machining parts around the cutting zone. This simulation-based investigation was carried out on the rotary milling tool. A numerical model approximates the flow of liquid coolant around the milling tool and makes it possible to evaluate the efficiency of the use of coolant lubricant in order to specific working conditions. Further part concerns with the investigation of heat fluxes through the cutting tool, which is essential to be able to evaluate the cooling effect of the liquid coolants on the cutting tool.

2. The Simulation model

A cutting tool is assembled of many different parts. As the numerical methods for computational fluid dynamics (CFD) are highly demanding on time and computational performance, it is necessary to use area, which is closed to the point of investigation. In this case, only the parts immediately nearby to the cutting zone were chosen and modelled. The 3D CAD model therefore consists of simplified chuck and the four blades cutting tool. The virtual space represents the environment around the tool and its chuck and there is a nozzle for liquid coolant supply. Both the meshing and the calculation were carried out in Ansys Workbench by using CFX.

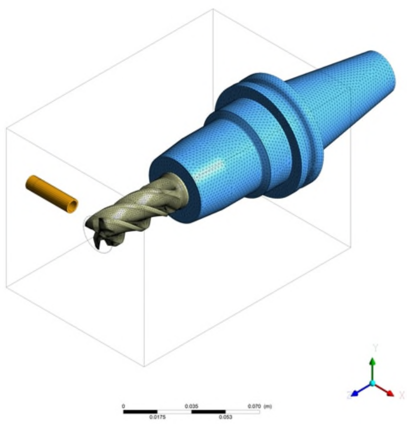

Figure 1. 3D CAD model of the system

For numerical calculation of fluid dynamics, every part of simulation model must be defined as a domain and the connection between them is described as an interface. Both the domains and the interfaces must be set up with its properties, such as Domain Type, Material, Heat transfer, etc., as they describe the interaction and function of the simulation system. This simulation model consists of six domains and 9 interfaces. There are three solid domains representing milling tool, tool chuck and nozzle for coolant supply. Domains for tool and its chuck are set as rotary with defined rotational speed corresponding to speed used in real experiments. For these two domains is also specified Heat Transfer for option Thermal Energy, which enables further evaluation of cooling effect. Other three domains are then described as Fluid, respectively in multi-phase system representing air and the liquid coolant, with corresponding Surface Tension Coefficient between these two phases. The first *Airvolume* represents the volume space in restricted area of the simulation model. Although it is modelled as a restricted area, it is defined as an open system, so the exchange is enabled with its surrounding environment. Because it is impossible to rotate domain, which is not symmetric and perpendicular to the direction of rotation, there was second *Airvolume 2* developed. This was created as a shape of cylinder, which fills the space of inner part of tool teeth. Third *Airvolume 3* then represents a small space between the cutting tool and tool chuck. The flow of air and coolant in the simulation is turbulent, these Fluid domains are calculated with the shear stress transport (SST) turbulence model. In these domains, the option Total Energy was used for the Heat Transfer. Buoyancy was applied for simulate of the gravitational force causing curving of coolant flow. The schematic built, which shows individual domains and their connections with a 3D model of its real equivalent, is showed in Figure 2.

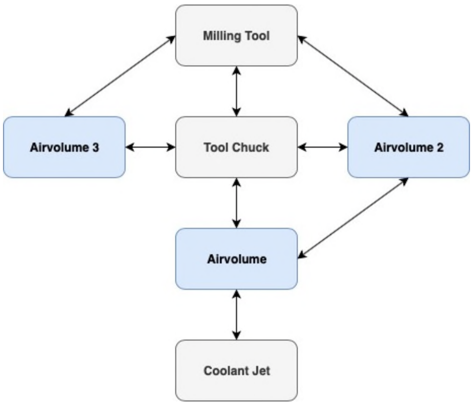

Figure 2. Scheme of domains of the system

3. Evaluation of simulation

To be able to verify the simulation model, the experimental investigations were carried out on a test stand specified particularly for thermal analyses. The tool and the chuck were mounted in a motor spindle. This spindle is fixed on a machine tool table with supporting element. Testing stand has also enclosure shields, so it is possible to isolate working space from its environment and secure better temperature measurement conditions. For recording the coolant flow, there was used a camera with two positions, above the tool and from top to the cutting edges of the tool.

The results of experimental investigation in Figure 3 shows the differences of the liquid coolant flow around the cutting tool for different conditions. Starting with the tool without rotational, a main part of coolant flows around the tool and continues further under curvature caused by gravitational force. There are also other flows, which direction varies by the time. With rotation tool in small rotational speeds, there can be seen change of flow to the field of droplets departing by centrifugal force. With increasing the rotational speed of the tool, the field of droplets changes into the fog of small coolant particles.

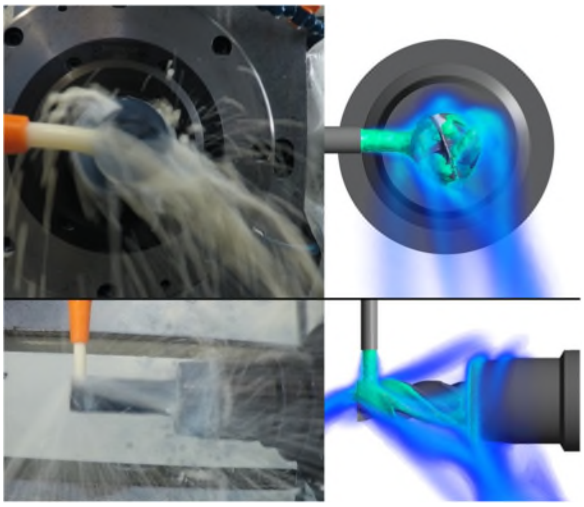

Figure 4. Comparison of real and simulated coolant flow

These instabilities show difficulties of simulation. As it is impossible to predict the direction of some parts of coolant flow after the contact with tool, such as field of droplets, the results of CFD simulation does not fully correspond with the real experiments. Also, the simulation cannot show particles smaller that the dimension of mesh. This can be influenced by changing the dimension of mesh, but it would also rapidly increase calculation time. The point of interest is therefore focused on the main flow. As shows the Figure 4, there can be seen slight inequalities, which can be caused by many reasons, such as small change of physical properties of liquid coolant cause by absorption of air humidity, difference between the surface tension of dry and wet surface or problems with convergency of turbulent models. On the other hand, these slight inequalities should have not an impact on the simulation of cooling effect.

4. Thermal simulation

For the evaluation of heat transfer, it is necessary to define material and thermodynamic properties of every domain. There must also be defined two heat transfer variables. First take its position as the tool chuck is clamped into the spindle. It is provided on the surface of ISO Taper cone with value already found in previous studies. The second important definition is the heat source in this system. There was used a simplified surface of the cutting edges with value corresponding with the induction system, which was carried out in the experimental measurement. The value was therefore set as Heat Flux variable into the simplified surface of cutting edges as shows Figure 5.

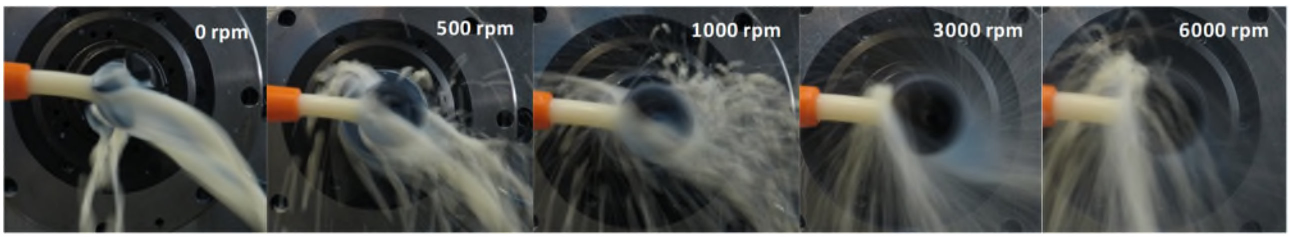

Figure 3. Coolant flow around the cutting tool.

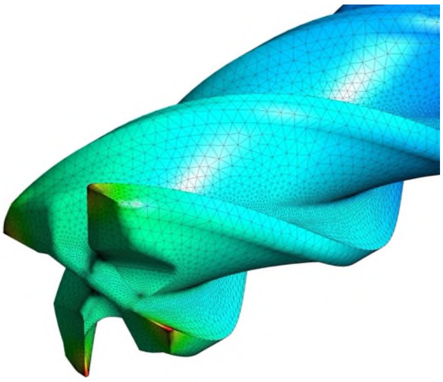

Figure 5. Temperature field of the tool with heat source

For the validation of temperature field through the tool, there was carried out the experiment with the composition of temperature sensors and induction system described in [5]. The temperature results of the simulation coincident with slight differences to the measured values on real milling tool. There are also many factors, which influence calculated temperature field in tool and its chuck. The significant role plays the accuracy of turbulence and flow of the coolant and also the fact, that sensors placed on the surface of cutting tool are not isolated from environment and can therefore measure some inequalities while contact with liquid coolant, which has significantly higher value of *Specific Heat Capacity* than material of the milling tool itself.

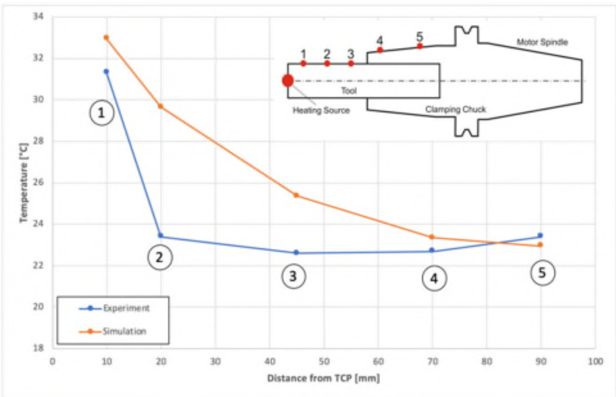

Figure 6. Temperature fields from experiment and simulation at 3000rpm

5. Summary and outlook

Numerical flow simulations provide a better understanding of how the coolant flows around the cutting tool and its effect on the temperature field in the tool. Simulating coolant flow around a rotating tool reveals difficulties caused mostly by turbulence and dispersion of fluid into the environment as a function of rotational speed of the cutting tool. Taking into account the simulation of small particles of liquid spreading into the environment is therefore difficult and increases demands on computational power and accuracy of the simulation itself. However, it is still possible to simulate the temperature field caused by the heating of the cutting surfaces with certain precision and to make the use of coolant more efficient.

There is still a need of research into the fluid flow around the rotating tool and thus refining the existing model, which will allow a better simulation of the fluid flow and thus a better understanding of the cooling of the tools by the use of coolants. Further, it is necessary to assess how much liquid actually reaches the tool and thus contributes indeed to the actual cooling of the tool and not only to cooling the surroundings or the chip removal function. In order to refine the evaluation of the temperature field in the tool, it is also necessary to determine the influence of the isolation of the temperature sensor on the surface of the tool against the liquid with a significantly higher coefficient of thermal capacity compared to the material of the tool itself on the measured values during the experiment.

Acknowledgement

The work is funded by the German Research Foundation – Project-ID 174223256 – TRR 96. The authors thanks for the support.

References

[1] Putz M, Richter C, Regel J and Bräunig M 2018 Industrial consideration of thermal issues in machine tools *Prod. Eng. Res. Devel.* **12** 723–736

[2] Perri G M, Bräunig M, Gironimo G D, Putz M, Tarallo A and Wittstock V 2016 Numerical modelling and analysis of the influence of an air cooling system on a milling machine in virtual environment *Int. J. Adv. Manuf. Technol.* **86** 1853–1864

[3] Putz M, Wittstock V, Semmler U and Bräunig M 2015 Simulation-based thermal investigation of the cutting tool in the environment of single-phase fluxes *Int. J. Adv. Manuf. Technol.* **83** 117-121

[4] Bennett EO, Bennett DL 1987 Minimizing human exposure to chemicals in metal working fluids *Lubr. Eng.* **43** 169–175

[5] Bräunig M, Regel J, Glänzel J, Putz M 2019 Effects of cooling lubricant on the thermal regime in the working space of machine tools *Procedia Manufacturing* **33** 327-334