euspen's 22nd International Conference & Exhibition, Geneva, CH, May/June 2022
www.euspen.eu

Modelling of the coolant flow behaviour in the narrow-closed cutting gap during sawing with internal coolant supply

Christian Menze[1], Jiawen Xiang[1], Hans-Christian Möhring[1], Jan Stegmann[2], Stephan Kabelac[2], Alexander Tismer[3], Jonas Wack[3], Stefan Riedelbauch[3]

[1]*Institute for Machine Tools, University of Stuttgart, Holzgartenstrasse 17, 70174 Stuttgart*
[2]*Institute of Thermodynamics, Leibniz University Hannover, An der Universität 1, 30823 Garbsen*
[3]*Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart, Pfaffenwaldring 10, 70569 Stuttgart*

Christian.menze@ifw.uni-stuttgart.de

Abstract
An important process in the manufacturing of semi-finished products is the sawing process. The chip formation takes place in a narrow, closed cutting gap. In the circular sawing process, there are no mechanisms for chip removal - for example, in comparison to flutes in drilling. The chip formed is enclosed in the chip space between the tool and the workpiece until the saw tooth exits the workpiece. The cutting tools are exposed to high thermal and mechanical loads, which require the use of cooling lubricants for heat dissipation, friction reduction between the material and the tool, and chip transport out of the cutting zone. Usually, flooding lubrication is used for sawing. However, depending on the shading of the cutting zone by the workpiece, tool or accumulating chips, only a fraction of the used coolant lubricant reaches the gaps between the chip, tool and workpiece. An internal coolant supply (ICS) through the tool system can be used to inject the coolant into the cutting zone in a targeted and controlled manner. For the process conditions in sawing, it is still unclarified which quantity of cooling lubricant, under which pressure and in which injection direction has to be applied to achieve an optimal cooling, lubricating and transport effect. The use of computational fluid dynamics (CFD) supports the investigation of a sawing process with internal coolant supply. This paper presents the modelling of the fluid mechanical process of the inflowing coolant inside the chip space in the narrow cutting gap. This includes the definition of the system boundaries. A porous boundary condition was used to set a controlled pressure loss within the chip space. The focus of this paper is on the meshing and the selection of a suitable turbulence model. The k-omega shear stress transport (SST) proved to be the most suitable model for this application.

Simulation, Fluid, Cutting

1. Introduction

Circular sawing is an important process to produce semi-finished products. The technology is characterised by using rotating, disc-shaped and multi-bladed tools with geometrically determined cutting edges (saw teeth). In the circular sawing process, there are no mechanisms for chip removal - compared to flutes in drilling. The chip formed is enclosed in the chip space between the tool and the workpiece until the saw tooth exits the workpiece. The cutting process takes place in a closed narrow cutting gap. Therefore, a targeted cooling and lubrication of the cutting process by external flood lubrication is difficult. By using an internal coolant supply (ICS), the chip formation process can be provided with metalworking fluid directly inside the chip space in the cutting gap.

The cooling effect of water-based emulsions using the high heat capacity and thermal conductivity of water is investigated in [1]. Furthermore, recent studies show that internal high-pressure lubrication influences the chip formation and the thermo-mechanical tool load [2-3]. While the strategies for ICS of the cutting process have been established for modern turning, milling, and drilling processes [4-5], there is considerable need for research of circular sawing. For the process conditions in sawing, it is still unclear what quantity of coolant must be introduced into the process zones under which pressure and in which direction to achieve an optimal cooling, lubrication, and transport effect.

Because of the small scale mechanism and the complex thermo-mechanical and thermo-fluid interactions, in-situ cutting investigations with metalworking fluid are extremely difficult. Therefore recent research is often made with computer-based simulation analysis of the multiphysical phenomena in wet cutting using computational fluid dynamics (CFD). Klocke et al. [6] modelled the chip-fluid interaction in a wet turning process with a coupled eulerian-lagrangian approach in ABAQUS. Pervaiz [7] used the results of a 2D-cutting simulation in DEFORM as input for a fluid dynamic simulation in ANSYS CFX. A similar approach was used in [8], where the flow of the cooling liquid in a drill was studied. By exporting the model from DEFORM and after repairing irregularities in the chip model, the authors performed a CFD-analysis and concluded, that the flow of fluid to the main cutting areas was insufficient. Uhlmann et al. [9] demonstrated the simulation of orthogonal cutting with the integration of a metalworking fluid by the finite-pointset-method. The results show a good agreement of the cutting force between simulation and experiment. However, the simulation of the interactions of cutting fluids and processes in production engineering is still in an early stage. In the following, the 2D numerical modelling of the fluid dynamics of a metalworking fluid in the chip space in the narrow closed cutting gap in sawing is presented.

2. Numerical modelling of the flow dynamics in the chip space

To simulate the fluid dynamics in the cutting space with ICS during sawing, first the real physical process has to be analysed and then abstracted and modelled. This includes the definition and creation of the simulation domain, the boundary conditions, the mesh and the selection of the numerical solver.

2.1. Experimental investigation

In circular sawing, the process kinematic consists of a combined rotational and translational motion. Consequently, it is difficult to observe a single cutting edge as it moves continuously in the actual circular sawing process. The circular sawing process is therefore reduced to a static analogy experiment, here.

A special experimental set-up is used to investigate the flow behaviour inside the closed cutting gap (Figure 1a-c). For the experiment, a saw segment was cut out of a circular saw blade and fixed in a holder. The saw blade segment has a blade width of 2.5 mm in which a bore hole with a diameter of 1.5 mm for the supply of cooling lubricant is drilled.

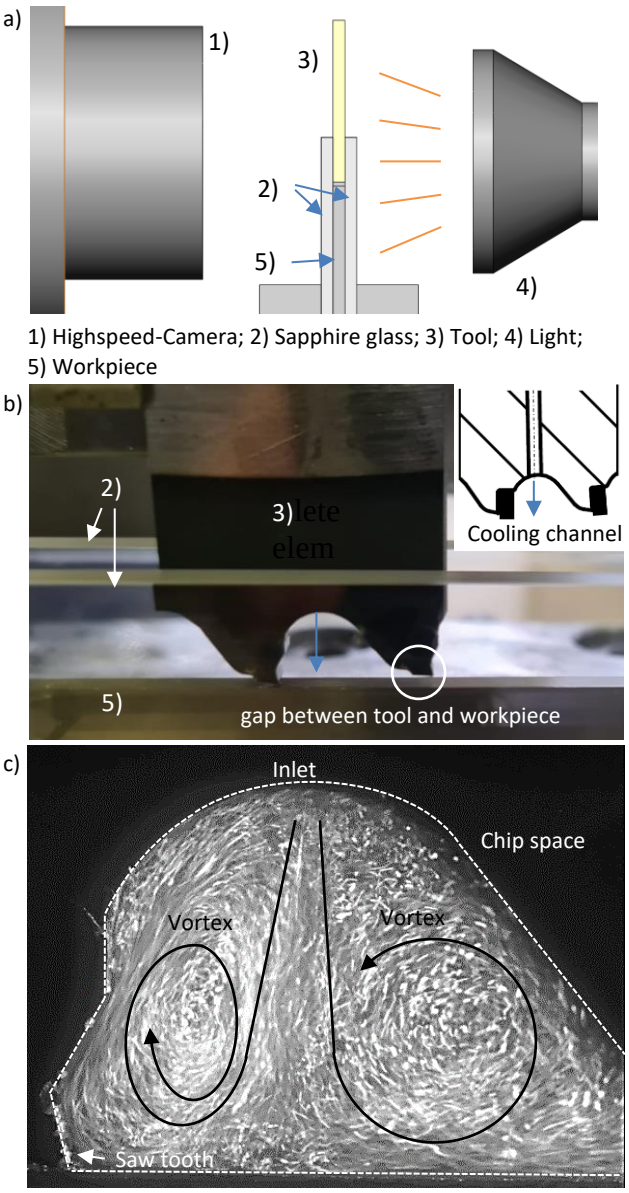

Figure 1. a) Shematic Set-up; b) Saw blade segment in the artificial cutting gap; c) Recording of the flow dynamic in the chip space (system-pressure 3 bar)

To visualise the fluid flow, the cutting gap is artificially closed with sapphire glass windows (Figure 1a-b).The cooling lubricant is injected into the chip space via the ICS using a pressure accumulator. For quantitative and qualitative analysis of the flow characteristics, polyamide particles with a density of 1.016 g/cm³ are inserted into the fluid as tracer particles (Figure 1c). The flow and the particle movement are recorded with a high-speed camera of the type Os8 - S3 from Imaging Solutions GmbH and synchronised lighting.

Figure 1c shows an instantaneous image of the flow inside the chip space. The experiments were carried out under different system pressures (p = 2 and 10 bar). The flow rate is 1 l/min and 3 l/min. A characteristic flow with two vortices is formed even under different boundary conditions. The fluid flows out of the bore hole into the chip space and hits the workpiece. There, the flow splits into two directions. The flow on the left in the picture forms a vortex together with the fluid flowing downstream. The fluid flowing to the right forms a second rotating vortex. The fluid can exit the chip space through two outlets. The first outlet results from the curvature of the saw blade. In the static analogy test, the left tooth in Fig. 1c was aligned in its cutting position. Since the workpiece is not curved like the cut out section of the circular sawing tool, there is an opening between the previous tooth (on the right) and the workpiece from which the fluid can flow out. Nevertheless, most of the fluid flows along the gap between the kerf and the saw blade.

2.2. Simulation domain and boundary conditions

For simulating the fluid dynamics in the narrow-closed cutting gap in sawing, OpenFOAM is used here. In the here presented studies, the chip formation is not considered. Within the ongoing research work, the evolving geometry of the chip will be incorporated. The final goal is to couple the structural simulation of the chip forming process [10] and the fluid simulation.

The model asymmetry of the here regarded computational domain (Figure 2) makes the creation of a structured mesh challenging. An unstructured mesh ensures flexibility. Thus, the choice of an unstructured mesh here serves as a preliminary investigation. The local refinement of the mesh should be small enough to accurately capture the key surfaces, sharp corners, slits and other details of the geometric model that have an impact on the simulation. In addition, no-slip walls in turbulent simulations need a mesh refinement to enable the special treatment of boundary layer effects by using wall functions.

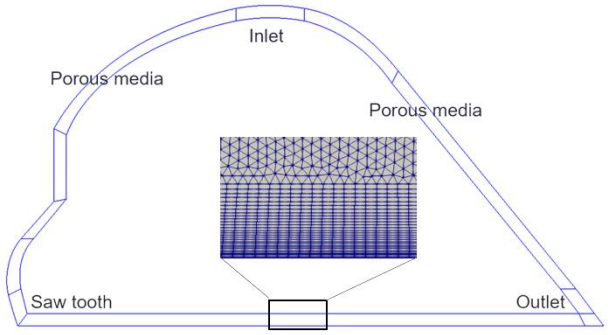

Figure 2. Division of the computational domain.

In flow field simulations a finer mesh is beneficiary for regions where the physical quantities change drastically. Therefore, it is good practice to have a rough estimate of the flow characteristics in the fluid domain before creating the mesh. For example, the number of mesh cells close to the wall should be higher than in the central area. As shown in Figure 2, a reasonable division of the computational domain facilitates the generation of structured meshes. The boundary area and the

workpiece are divided into quadrilateral shapes conforming to the contour curve. It is possible to refine the mesh in these areas using transfinite interpolation of the open source software gmsh. The boundary conditions for the pressure, velocity and turbulence fields at the inlet and outlet as well as at the wall are defined in Table 1. It should be noted that the inletOutlet boundary condition acts as a normal zeroGradient condition for outflow, but prohibits inflow.

Table 1. Boundary conditions at the inlet, outlet and wall.

Boundary	Pressure	Velocity	Turbulence Field
Inlet	zeroGradient	fixedValue	fixedValue
Outlet	fixedValue	inletOutlet	inletOutlet
Wall	zeroGradient	fixedValue	WallFunction

In general, near wall modeling is arguably the most problematic area in turbulence modeling. The turbulent flow near a flat wall can be divided into viscous sublayer, buffer layer, log-law region and outer layer. Therefore, a suitable near-wall model is necessary when dealing with wall-bounded turbulent flows. A commonly employed modeling procedure is the logarithmic wall treatment. These functions are widely used in industry and may be found in almost all commercial and in-house CFD codes [11]. The here considered Reynolds numbers are between 20.000 and 60.000. In this range, a flow is generally assumed to be turbulent. However, compared to many typical CFD-applications such as turbomachinery or aerospace engineering the considered Reynold numbers are comparably low. Due to the the overall small geometrical dimension of the model, the Reynolds number does not clearly indicate a turbulent flow. Therefore, also laminar simulations were performed and compared to the turbulent simulations. A comparison of integral values has shown that both simulation approaches give comparable results. But it is clear that a direct simulation based on the non averaged Navier-Stokes equations for a sufficient low Reynolds number show much more details, e.g. vortices, of the flow.

The most popular turbulent model is the k-Epsilon. It is a high Reynolds model and achieves good results in free flows. The model is less suitable for modelling the viscous sublayer and in the laminar-turbulent transition. The k-omega model, on the other hand, is a good model for near-wall flow processes, but does not reproduce free flows sufficiently well. The Shear Stress Model (SST) is a hybrid model and combines the advantages of both models. It is therefore a good compromise between free flow and near-wall flow processes. For this reason, the k-Omega SST model is used for further modelling.

During the sawing process, respectively in the static test, the cooling lubricant escapes through the small gap between the tool and the workpiece (in the experiment the sapphire glass). Preliminary investigations showed that a consideration of the upper boundary of the computational domain, representing the sawing tool, (Figure 2) as a sealed wall leads to a flow characteristic in the simulation that does not correctly reproduce the experimental highspeed-camera recordings. That means that a special boundary condition at the upper boundary of the simulation domain is needed to allow a certain pressure loss and mass transfer. Therefore, this disturbed outflow is modelled by declaring a porous media close to the boundary. The porous media is applied to a specific cell zone (Figure 2) of the mesh. According to the definition of porous boundary conditions in OpenFOAM, a larger porous media coefficient indicates a more significant pressure difference at the cell zone resulting in more resistance. The application in the simulation is to simplify and simulate the effect of the structure on the flow region in the geometric model for different machining conditions. For example, the difference in thickness between the blade and the side cutting edge of a circular saw can be also a factor in defining the porous boundary in the simulation. The larger the thickness difference, the easier it is for the cutting fluid to exit the cutting space from the boundary during the machining process. It should be noted that the simulation is 2-dimensional here. Therefore, the disturbed outlet has to be modelled. In prospective 3D-simulations, the gap between tool and workpiece can be modelled explicitly.

2.3. Numerical solver

The incompressible Navier-Stokes equation is given by

$$\nabla \cdot \mathbf{U} = 0 \tag{2.1}$$

$$\frac{\partial \mathbf{U}}{\partial t} + \nabla \cdot (\mathbf{UU}) = -\nabla \frac{p}{\rho} + \nabla \cdot (\nu \nabla \mathbf{U}) \tag{2.2}$$

where ρ is the density, $\mathbf{U}$ is the velocity, p is the pressure, ν is the kinematic viscosity. The Reynolds averaging method (RANS) is currently the most widely used method in engineering to solve fluid dynamic problems. For RANS, mean and fluctuation components of the Navier Stokes equations are splitted, as shown in the equation

$$u_i = \bar{u}_i + \dot{u}_i \tag{2.3}$$

These instantaneous flow variables are incorporated into the continuity and momentum equation [12], for single-phase flow which is given by

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial X_i}(\rho u_i) = 0 \tag{2.4}$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_i}(\rho u_i u_j) = \frac{\partial \rho}{\partial x_i} + \frac{\partial}{\partial x_j} A + \frac{\partial}{\partial x_i}(-\rho \overline{\bar{u}_\iota \dot{u}_\iota})$$

$$A = \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_i}{\partial x_i}\right)\right] \tag{2.5}$$

During the filling process of the chip space with cutting fluid, the volume fraction of the liquid phase is increasing, and the volume fraction of the gas phase is decreasing. In this gas-liquid mixing process, the movement of air bubbles after coolant injection and the tracking of the gas-liquid interface are the main objects of interest. Therefore, the Volume of Fluid (VOF) model is a suitable model for this particular two-phase flow simulation.

3. Simulation results

Figure 3a shows the velocity profile of the simulated flow. The inlet was defined as Dirichlet boundary condition with a velocity of 10 m/s. It can be seen that the flow splits up after hitting the workpiece (lower boundary) and two vortices occur. This flow behaviour agrees with the high-speed camera recordings. Without the boundary condition of the porous medium, such a flow did not occur, even when varying the boundary conditions at the inlet and outlet.

As shown in Figure 3b, the pressure distribution in the cutting space is also closely related to the flow state of the coolant. In other words, the pressure is increased in the area where the coolant touches the wall of the cutting space and, consequently, changes its direction. This is in line with observations of stagnation areas.

In addition, the CFD simulation can also predict the turbulent kinetic energy in the turbulent behaviour of the fluid. As shown in Figure 3c, the flow behaviour of the coolant ingested from the inlet still satisfies the characteristics of pipe flow, with low

turbulent kinetic energy. However, when the coolant collides with the workpiece and forms two distinct vortices in the cutting space, the turbulent kinetic energy increases significantly.

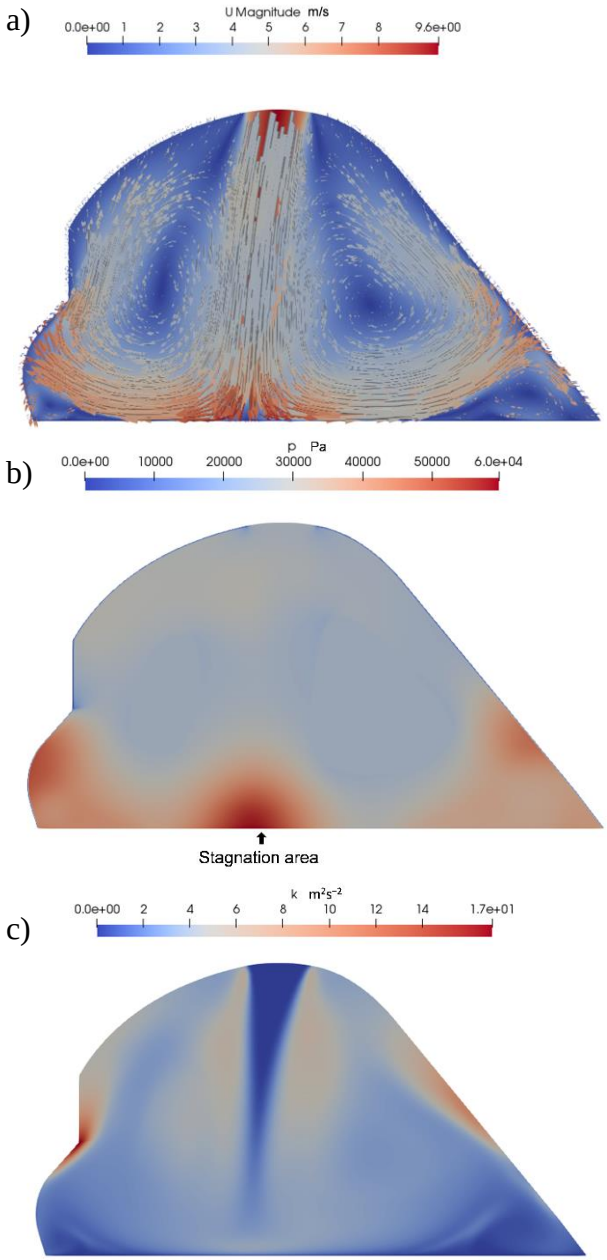

Figure 3. a) Vector diagram of velocity colored by magnitude of velocity; b) Prediction of the pressure in cutting space colored by pressure; c) Prediction of the turbulent kinetic energy k in cutting space.

4. Conclusion and outlook

In this work, a two-dimensional fluid dynamics model of the cutting space in circular sawing is created and important geometric features are preserved. A particular challenge in creating the simulation model was to reproduce the actual flow behaviour. With conventional inlet and outlet boundary conditions, the flow dynamics observed in the experiment could not be reproduced. By integrating a special boundary condition, the porous medium, the outflow of the fluid from the chip space along the saw blade was reproduced and the characteristic vortex structure from the experiment could be simulated. This made it possible to reduce the 3-dimensional flow case to a 2-dimensional problem.

For the simulation, the RANS method was used. For this, a suitable turbulence model was selected. The considered Reynolds numbers are relatively low compared to common applications. Therefore, the k-Omega SST turbulence model was selected, as it is suitable for a large number of flow problems. For this purpose, the boundary layer at the walls was provided with an appropriate mesh refinement.

However, often even simple flow cases have a 3-dimensional character [13]. Consequently, in a next step, the chip space of the sawtooth is extended to a 3-dimensional simulation. In this way, it can be shown how far the 2-dimensional simulation with the corresponding boundary conditions can reproduce the flow behaviour compared to the 3D model.

Acknowledgment

The authors appreciate the funding of this work within the Priority Program 2231 “Efficient cooling, lubrication and transportation – coupled mechanical and fluid-dynamical simulation methods for efficient production processes (FLUSIMPRO)” by the German Research Foundation (DFG) – project number 439925537.

References

[1] Brinksmeier E, Meyer D, Huesmann-Cordes AG, Herrmann C. Metalworking fluids—Mechanisms and performance. *CIRP Annals 2015*; 64(2):605–28.

[2] Liu H, Peng B, Meurer M, Schraknepper D, Bergs T. Three-dimensional multi-physical modelling of the influence of the cutting fluid on the chip formation process. *Procedia CIRP 2021*; 102:216–21.

[3] Courbon C, Sajn V, Kramar D, Rech J, Kosel F, Kopac J. Investigation of machining performance in high pressure jet assisted turning of Inconel 718: A numerical model. *Journal of Materials Processing Technology 2011*; 211(11):1834–51.

[4] Klocke F, Sangermann H, Krämer A, Lung D. Influence of a High-Pressure Lubricoolant Supply on Thermo-Mechanical Tool Load and Tool Wear Behaviour in the Turning of Aerospace Materials. Proceedings of the Institution of Mechanical Engineers, Part B: *Journal of Engineering Manufacture 2011*; 225(1):52–61.

[5] Sangermann H. Hochdruck-Kühlschmierstoffzufuhr in der Zerspannung. *Dissertation RWTH Aachen*, ISBN: 978-3-86359-148-9, Apprimus-Verl. 2013

[6] Klocke F, Döbbeler B, Peng B, Lakner T. FE-simulation of the Cutting Process under Consideration of Cutting Fluid. *Procedia CIRP 2017*; 58(11):341–6.

[7] Pervaiz S, Deiab I, Wahba EM, Rashid A, Nicolescu M. A Coupled FE and CFD Approach to Predict the Cutting Tool Temperature Profile in Machining. *Procedia CIRP 2014*; 17:750–4.

[8] Oezkaya E, Biermann D. A new reverse engineering method to combine FEM and CFD simulation three-dimensional insight into the chipping zone during the drilling of Inconel 718 with internal cooling. *Machining Science and Technology 2018*; 22(6):881–98.

[9] Uhlmann E, Barth E, Seifarth T, Höchel M, Kuhnert J, Eisenträger A. Simulation of metal cutting with cutting fluid using the Finite-Pointset-Method. *Procedia CIRP 2021*; 101:98–101.

[10] Menze C, Wegert R, Reeber T, Erhardt F, Moehring H-C, Stegmann J. Numerical methods for the simulation of segmented chips and experimental validation in machining of TI-6AL-4V. *MM SJ 2021*; 2021(5):5052–60.

[11] Tu, J. Computational fluid dynamics. A practical approach. *Butterworth-Heinemann*, Amsterdam. 2018.

[12] Wilcox DC. Reassessment of the scale-determining equation for advanced turbulence models. AIAA Journal 1988; 26(11):1299–310.

[13] Breuer, M. Large eddy simulation of the subcritical flow past a circular cylinder: numerical and modeling aspects. *International Journal For Numerical Methods in Fluids*. 1998; 28(9), 1281–1302