CFD simulation for internal coolant channel design of tapping tools to reduce tool wear

D. Biermann (1)*, E. Oezkaya

Institute of Machining Technology, TU Dortmund University, 44227 Dortmund, Germany

ARTICLE INFO

Keywords:
Simulation
Tapping
Optimization

ABSTRACT

This paper presents the analysis and controlled modification of the coolant flow in tapping processes by means of Computational Fluid Dynamics (CFD). First, a conventional straight flute tapping tool was analyzed and the results of the CFD simulation show, that the cutting edges are not sufficiently supplied with coolant. Therefore, the design of the internal coolant channels was modified based on these simulation results. To validate the CFD simulation, experimental tests were performed, using an optimized tool. The applied modifications lead to a reduction of the tool wear and an increase of the tool’s performance of about 36% was achieved.

© 2017 Published by Elsevier Ltd on behalf of CIRP.

1. Introduction and background

Product requirements increase in almost every industrial sector. In addition, great importance is attached to a high rate of production, high quality standards, environment protection and low energy as well as resource consumption. Special attention should be paid towards tapping, since it is often the final process step and damaging a component at the end of the process chain would result in a very expensive rework [1]. The tapping tools must be able to withstand high cutting forces, a large heat dissipation rate at the cutting edges and a very good chip removal are mandatory. Therefore, high-performance taps need to be designed to fulfill these requirements, i.e. they are coated, geometrically adapted and equipped with internal coolant channels. Experimental investigations of the flow and distribution of pressure of the cutting fluid cannot be performed, due to the kinematical inaccessibility of the tapping process. For that reason, it has not yet been possible to establish specific tool modifications to enhance tool performance and to optimize the usage of coolants. Based on the work of Beer et al., this paper presents a CFD simulation of the flow distribution and pressure of coolants while tapping, depending on the arrangement of internal coolant channels [2].

1.1. CFD analysis in tapping

A literature review has shown, that a CFD simulation for evaluating the distribution of coolant during tapping has not yet been done. A likely reason is, that CFD simulations in contrast to FEM simulations are not yet well-established in machining technology. A lot of challenges arise when modelling the fluid, e.g. the geometrical boundary conditions, resistance coefficients, driving forces and interactions between different processes as well as the input values of the fluid model. Furthermore, the correct selection of turbulence models and types of meshing are of primary importance for representing a realistic process. The requirements, that the used fluid models need to fulfill, increase even more with the high complexity of the tapping tools. Especially in the areas of the boundary layers of the tap tooth, the meshing of smaller dimensioned elements leads to major challenges. Moreover, a 3D simulation has to be used for tapping, since the torque arises about the whole chamfer length. Compared to a 2D simulation, the risk of numerical errors and the computing time increase when using a 3D simulation, due to the high number of mesh elements. The fluid cannot be modelled using axially symmetrical 2D reduction. As mentioned above, another complex problem is, that the flow distribution and pressure of the coolants cannot be measured. Therefore, a CFD simulation is a powerful method to investigate the behaviour of the coolant at the cutting edge and to extend the process knowledge for tapping [3,4].

1.2. CFD analysis of the vortex intensity

Vortices are known as a flow with a circular and rotary route, dissipating energy. Thereby, the vortices are formed in different sizes, interacting with each other and reacting sensitively in accordance with the initial and boundary conditions. So the vortices are irregular and chaotic. The three-dimensional and stochastic character shows the completely developed turbulence. This formation is strongly dependent on the time. Vortices are often described as regions of high vorticity, but no universal definition for vortex intensity exists [5]. Problems resulting from the different definitions have been exposed by several researchers [6–8]. During the last three decades, a variety of methods has been proposed to identify vortices. Kolář gives a short overview of the
most important methods [9]. Zemliak and Mastorakis elaborate different methods and their fundamental properties together with corresponding criteria [10]. For CFD simulations, the q-criterion from Hunt and the $\lambda_2$-method are well established [6,11]. Both of these methods are proposed for rotation invariant vortices for flow visualization in the work of Günther et al. [12]. In this paper, the q-criterion and the software ANSYS CFX were used.

2. Fluid model and CFD simulation

Fig. 1 shows the fluid model. The internal coolant channel passes through the centre of the tool and branches out over the three areas (A-A). Using the arrangement of the internal coolant channel and the tapping conditions for the workpiece, the inlet and outlet ambience can be defined and the CFD fluid model is prepared. For calculating the flow area, solid body and fluid body were separated and the interfaces between tool, workpiece and fluid were defined. Another important criterion for the exact modelling of the fluid and the boundary conditions, summarized in Table 1, is the correct choice of a turbulence model.

![Reference tapping tool and reference fluid model](image)

**Table 1** Boundary conditions for straight flute tapping of through holes.

<table>
<thead>
<tr>
<th>Boundary condition</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Transient: total time/time step</td>
<td>Time $t_0 = 10 s/\tau_0 = 0.1 s$</td>
</tr>
<tr>
<td>Water (measured)</td>
<td>Working temperature $T_w = 298.15 K$</td>
</tr>
<tr>
<td>Inlet: coolant supply (measured)</td>
<td>Pressure $P_{\text{inlet}} = 48.8$ bar</td>
</tr>
<tr>
<td>Outflow: opening – ambience</td>
<td>Mass flux $m = 0.1934$ kg s$^{-1}$</td>
</tr>
<tr>
<td>Interface 1: workpiece/coolant (measured)</td>
<td>Roughness $R_z \leq 0.01$ mm</td>
</tr>
<tr>
<td>Interface 2: fluid/tool</td>
<td>Smooth conduit</td>
</tr>
<tr>
<td>Number of elements</td>
<td>Element Approx. 2 million</td>
</tr>
<tr>
<td>Infiltration-layer: 1st layer</td>
<td>Length $0.0005$ mm</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>Type $k-\omega$-SST</td>
</tr>
</tbody>
</table>

The SST turbulence model combines the benefits of the $k-\varepsilon$ and $k-\omega$ model and is ideal for describing the turbulence in the near-wall area and in the centre area (logarithmic range) [2,13]. The SST turbulence model behaves like the $k-\omega$ model with a function $A_1$ and the transformed $k-\varepsilon$ model with $(1 - A_1)$. These turbulence models are rewritten in terms of $k$ and $\omega$, resulting in the SST model including the $k$-equation (4) and $\omega$-equation (5) [14,15]:

$$\frac{\partial}{\partial t} k + \frac{\partial}{\partial x_j} (\rho u_j k) = \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_1}{\sigma_{k1}} \frac{\partial k}{\partial x_j} \right] + P_k - \beta \rho \omega$$

$$\frac{\partial}{\partial t} \omega + \frac{\partial}{\partial x_j} (\rho u_j \omega) = \frac{\partial}{\partial x_j} \left[ \frac{\omega}{\sigma_{\omega 2}} \frac{\partial \omega}{\partial x_j} \right] + \frac{2}{3} \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_1}{\sigma_{\omega 1}} \frac{\partial \omega}{\partial x_j} \right]$$

The function $A_1$ results in the value 1 in the near-wall region, so that the SST turbulence model is reduced to the $k-\omega$ model. In the logarithmic range, the function results in the value 0, so the parameters of the $k-\omega$-model are used. Between these two regions, a combination of both parameter sets is adapted. The function $A_1$ is defined as:

$$A_1 = \tanh \left[ \min \left( \frac{1}{\sqrt{\frac{k}{500 \mu_0}}, \frac{4 \rho \sigma_{k1} k}{CD_{\text{inlet}} \rho}} \right) \right]$$

The term $CD_{\text{inlet}}$ defined in Eq. (7), describes the cross-diffusion term from Eq. (5), while $y$ is the distance to the wall:

$$CD_{\text{inlet}} = \max \left( \frac{2 \rho}{\rho_0 \alpha_{\text{inlet}} \omega} \left( \frac{\partial k}{\partial x_j} \right) \frac{\partial \omega}{\partial x_j} \right) \cdot 10^{-20}$$

The turbulent viscosity $\mu_1$ can be calculated in terms of $k$ and $\omega$ as:

$$\mu_1 = \frac{\rho \alpha_1 k}{\max \left( \alpha_1 \partial \omega / \partial y, \alpha_2 \partial \omega / \partial y \right)}$$

A further function $A_2$ is used for the limitation of the turbulent viscosity and results in values between zero and one:

$$A_2 = \tanh \left[ \max \left( 2 \frac{\sqrt{k}}{0.09 \omega}, \frac{500 \mu_0}{\gamma \omega} \right) \right]$$

The factor $\alpha_1$ in Eq. (8) describes the proportionality of the shear stresses $u_i$ and $u_j$ to the turbulent kinetic energy $k$. The parameters of the SST-turbulence model are provided in Table 2.

**Table 2** $k-\omega$-SST closure constants.

<table>
<thead>
<tr>
<th>$\beta'$</th>
<th>$\phi_1$</th>
<th>$\beta_1$</th>
<th>$\sigma_1$</th>
<th>$\sigma_{\omega 1}$</th>
<th>$\phi_2$</th>
<th>$\beta_2$</th>
<th>$\sigma_2$</th>
<th>$\sigma_{\omega 2}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>5/9</td>
<td>0.075</td>
<td>2</td>
<td>0.5</td>
<td>0.44</td>
<td>0.0828</td>
<td>1</td>
<td>0.856</td>
</tr>
</tbody>
</table>

2.1. Meshing and CFD result of the standard tool model

The fluid model depicted in Fig. 2 was separated into the front part with the larger diameter (A) and into the centre of the internal cooling channel (B). The irregular areas of the flow (C) were meshed with tetrahedral elements.

![Meshing of the fluid model for the tapping process](image)

**Fig. 3** Fluid velocity of the reference tool model.

Please cite this article in press as: Biermann D, Oezkaya E. CFD simulation for internal coolant channel design of tapping tools to reduce tool wear. CIRP Annals - Manufacturing Technology (2017), http://dx.doi.org/10.1016/j.cirp.2017.04.024
دریافت فوری متن کامل مقاله

امکان دانلود نسخه تمام متن مقالات انگلیسی
امکان دانلود نسخه ترجمه شده مقالات
پذیرش سفارش ترجمه تخصصی
امکان جستجو در آرشیو جامعی از صدها موضوع و هزاران مقاله
امکان دانلود رایگان ۲ صفحه اول هر مقاله
امکان پرداخت اینترنتی با کلیه کارت های عضو شتاب
دانلود فوری مقاله پس از پرداخت آنلاین
پشتیبانی کامل خرید با بهره مندی از سیستم هوشمند رهگیری سفارشات