Cooling Study of Baffles Integration in the Molding Industry

B. Noailles[1], S. Meunier[1], V. Bruyere[2]

[1]RocTool, Savoie Technolac, Module R, France
[2]SIMTEC, France

Abstract

Introduction: In the molding industry, high productivity rate, low energy consumption, large 3D parts, and homogeneous temperature distribution are the main targets. The 3iTech® inductive technology developed by RocTool ensures both good temperature homogeneity and short heating time.

Conventionally, to guarantee efficient cooling, a turbulent water flow is directly integrated into the mold. Ideally, to cope with the temperature homogeneity requirements for 3D shaped parts, cooling lines are expected to follow the molding surface at a constant distance. Additive manufacturing techniques which generate conformal integration are of high interest for small parts while drilling operations are still the best options for large parts. As part of drilling operations, the so called baffle geometry is particularly well suited when cooling complex geometries. The heat transfer coefficient (htc), which has been extensively studied in the literature for simple configuration such as cylindrical tube, is not defined for such complex geometries and requires specific numerical studies. In the present study, the CFD turbulent model has been developed in collaboration with SIMTEC.

As illustrated in the Figure 1, a specific baffle integration is studied through the evaluation of the turbulent flow passing through the cooling line represented in blue.

Use of COMSOL Multiphysics:
The turbulent flow is solved as a stationary study independently from the mold temperature. To ensure an accurate estimation of the heat exchanges at the interface between the turbulent water flow and the solid, the Low-Reynolds number k-epsilon model is used. This model does not take into account any wall function and solves the velocity of the flow in the whole domain.

Boundary layers are introduced to implement specific meshing parameters allowing to refine the viscous layer and the buffer layer ensuring the accuracy of the results. The size of the mesh is dependent upon the characteristics of the flow such as the Prandtl number and the Reynolds number.

Taking into account the thermal initial distribution of the mold after the heating step and accounting for the velocity distribution, the heat transfer in fluid model is used to evaluate the

impact of the turbulent water flow on the temperature distribution of the mold according to the time.

Results: The velocity field and the thermal distribution are respectively illustrated in the Figure 2 and 3. The heat transfer coefficient estimated along the cooling line is shown in the Figure 4.

Conclusions: The use of COMSOL Multiphysics software enables accurate estimation of the heat transfer coefficient in a complex geometry allowing to optimize cooling integration in the molding industry.

Reference

• Launder, B. E. and Sharma, B. I. (1974), "Application of the Energy-Dissipation Model of Turbulence to the Calculation of Flow Near a Spinning Disc", Letters in Heat and Mass Transfer, Vol. 1, No. 2, pp. 131-138.

• Comparison Between Honeycomb and Fin Heat Exchangers, P. Gateau , P. Namy , and N. Huc, COMSOL European Conference 2011

Figures used in the abstract

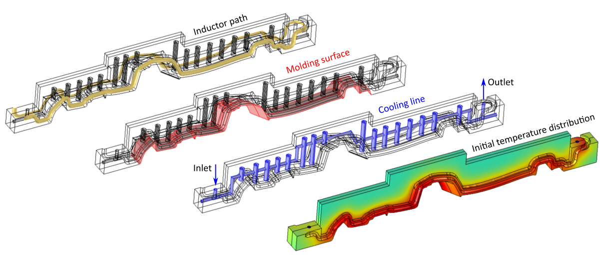

Figure 1

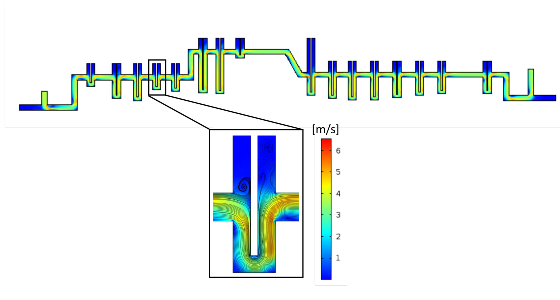

Figure 2

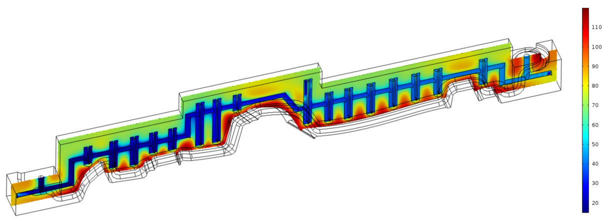

Figure 3

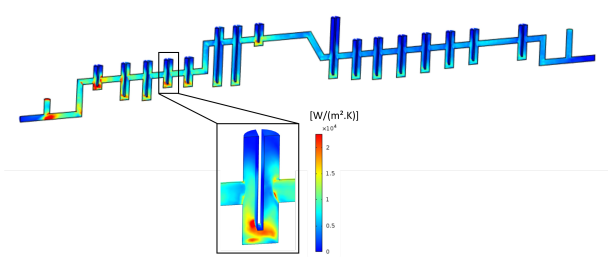

Figure 4: Distribution of the heat transfer coefficient