Proceedings of the 4th World Congress on Momentum, Heat and Mass Transfer (MHMT'19)
Rome, Italy – April, 2019
Paper No. ENFHT 115
DOI: 10.11159/enfht19.115

Numerical investigation of spray cooling in OpenFOAM

Sarah Winstanley[1], Alexander Haban[2], Bernd Platzer[1], Martin Fehlbier[2], Daniel Zipplies[1]
[1]Professorship Technical Thermodynamics, Chemnitz University of Technology
Reichenhainer Straße 70, 09126 Chemnitz, Germany
sarah.winstanley@mb.tu-chemnitz.de; bernd.platzer@mb.tu-chemnitz.de; daniel.zipplies@mb.tu-chemnitz.de
[2]Department of Casting Technology, University of Kassel
Kurt-Wolters-Straße 3, 34125 Kassel, Germany
alex.haban@uni-kassel.de; fehlbier@uni-kassel.de

Extended Abstract

Spray cooling is a technique, which is applied in numerous industrial processes. These include continuous casting, where it is used as a secondary cooling process, the cooling of electronic systems, which offer limited surface area available for cooling, as well as the high-pressure die casting process.

For casting processes an in-depth knowledge of spray cooling is desired, because it promotes the construction of a die, which is thermally balanced. This will lead to an improved quality of the casting as well as to lowered costs. Although the heat removal by spray strongly influences the cooling and therefore the structure of the casting, but the process and the characteristics are not completely understood. The investigations presented here aim for the development of a spray cooling solver in OpenFOAM, which is capable of calculating the heat transfer from a substrate over the range of single phase cooling up to the film boiling regime. The solver also aims to include the formation of a liquid film depending on the spray parameters and boundary conditions and heat conduction in the solid substrate.

The developed solver includes a carrier fluid in Eulerian formulation, a particle cloud in Lagrangian formulation, a wall interaction model for the particles impinging on the wall [1], a wall film model in Finite-Area-Method [2] which is extended with the energy equation, a heat transfer model [3], and an implementation describing the conjugate heat transfer between the fluid and solid domains. The present paper aims to verify and validate the solver against experimental data, which are chosen to be in the range of the practical die casting process.

In order to examine the developed solver, four test cases are set up as experiments. These are transferred to a simulation model with the aim of executing the numerical investigation. The thermal conditions of the solid are varied between the different cases. The spray parameters are chosen to be identical for the four cases.

For evaluating the numerical solver, the results of the simulations are compared to the experimental results. The measured heat transfer coefficients are compared to the values determined numerically. Additionally, high-speed recordings of the spray impacting on the hot substrate are matched with the results obtained by the simulation.

References

[1] C. Bai, H. Rusche and A. Gosman, "Modeling of gasoline spray impingement," *Atomization and Spray,* vol. 12, no. 1-3, 2002.
[2] Z. Tukovic and H. Jasak, "Simulation of thin liquid film flow using openfoam finite area method," in *4th OpenFOAM Workshop*, Montreal, Canada, 2009.
[3] J. Wendelstorf, K. Spitzer and R. Wendelstorf, "Spray water cooling heat transfer at high temperatures and liquid mass fluxes," *International Journal of Heat and Mass Transfer,* vol. 51, no. 19, pp. 4902 - 4910, 2008.