

CFD simulation of the multiphase heat transfer during the quenching process

Corresponding author:

Jan Novosád, jan.novosad@vuts.cz, VÚTS, a.s

Co-authors:

Pavel Peukert, Norbert Pomp, Pavel Klouček

Abstract:

The paper presents the results of the CFD simulation of quenching process for the ring shape sample. The aim of the work is to develop and validate the methodology for multiphase CFD simulation including the boiling during the quenching process. CFD simulation is provided in ANSYS Fluent. The Lee model is used for modelling the phase change during the quenching process. The first step consider the simulation of cooling of the ring sample when the correct model parameters will be found. Validation of results is performed by comparison with experimental data. Experimental was realized inside the own designed quenching bath device filled with quenching polymer. The general description of the experimental setup is included in the paper. The CFD results are cooling curves, i.e. variation of solid temperature on time. The Lee model parameters especially the evaporation frequency was tuned. Thin polymer film on the solid surface was considered to bring the results closer to the experimental data. The comparison between experiment and CFD shows very good agreement for higher temperatures, which covers the boiling stage. On the other hand for lower temperatures worse match of results was found caused probably by the sensitivity on the inlet velocity profile settings. Some recommendations for future work were defined.

Key words:

Multiphase, CFD, Fluent, quenching, polymer