Numerical and experimental study of spray cooling of a heated metal surface

M.R. Guechi*, P. Desevaux, P. Baucour
FEMTO-ST Institute, Energy Department, 2 avenue Jean Moulin, F-90000 Belfort, France
mguechi@univ-fcomte.fr, pdesevau@univ-fcomte.fr and philippe.baucour@univ-fcomte.fr

Abstract

The spraying of an impinging jet is an effective way to cool heated surfaces. The objective of this study is to develop a numerical model to predict the heat transfer with phase change between a hot plate surface and a two-phase impinging jet. Different two-phase modeling approaches (Lagrangian and Eulerian methods) are compared. The influence of the spray nozzle operating conditions (pressure, flow rate, droplets size) and of the distance between the nozzle exit and the surface impact is analyzed. The numerical results are compared with measurements obtained on an experimental test bench. The confrontation numerical/experimental is carried out by comparing the distribution of temperature at the surface of the plate and the heat transfer coefficient. This comparison shows that it is the Eulerian model which seems most capable to take into account the evaporation of the droplets in contact with the heated plate and consequently, which gives results more in agreement with the experiments. However, the simulation performed with this model show a strong dependence of the results to the turbulence model used.

Introduction

This work falls under a research project aiming at improving the cooling of electric motors of great power using an impinging jet spray. A computational approach by CFD is planned in order to evaluate the effectiveness of the spray cooling and to dimension and optimize the cooling system. For that, it is necessary to develop a numerical model making it possible to predict the heat transfer with phase change between a heated surface and a two-phase impinging jet and to validate the CFD results against experiments.

Experimental Setup

A test bench designed to enable the experimental study of the spray cooling of a heated metal surface has been constructed. It consists of a heating system, a spray nozzle and a data acquisition system. The spraying surface corresponds to the top surface of a copper block cylinder heated by means of a 400 W cartridge heater. 12 J-type thermocouples were embedded at various depths below the heater surface to provide the temperature gradient and temperature profile within the copper block cylinder. The test bench enables the investigation of the spray cooling process for various operating conditions (different heat fluxes, different distances between the spray nozzle and the hot surface).

Numerical approach

The numerical model was implemented in a commercial CFD code. Different two-phase modeling approaches (Lagrangian and Eulerian methods) and several RANS turbulence models were tested.

Results and Discussion

The results obtained relate to the evolution of the local heat transfer coefficient and of the surface temperature along the plate. The computational results more in agreement with the experiments were obtained with the two-phase Eulerian model associated with the Realizable k-epsilon turbulence model. The CFD model was then used under various operating conditions of the spray system (water pressure, flow rate, droplets size) and for various conditions of heating. The influence of the distance between the nozzle exit and the surface impact was also analyzed. The results obtained show strong variations of the surface cooling with the spraying system operating conditions and the comparison between simulations and experiments shows the capability of the model to correctly predict the phase change heat transfer during the spray cooling. The model developed must allow, in a short time, the optimization of the spraying system before its implementation on an electric motor.

*Corresponding author : mguechi@univ-fcomte.fr