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 heat dissipation and the cooling of electric motors of great power (more than 30 kW) intended for a new generation of electric cars. Indeed, the cooling of the electric motors is generally obtained by natural or forced convection modes [1, 2]. The need for designing electric motors with higher specific powers obliges to consider other more effective cooling solutions, in particular to cool the high temperature parts of the motor (as the coil winding heads). The use of impinging jet sprays is proved to be an effective way to cool high temperature surfaces and has been previously used for cooling electronic components [3] or for metal quenching [4, 5].

Spray cooling consists of a stream of fluid droplets (mixture of gas and fine liquid droplets) impacting on the surface to cool. On contact with the hot surface, the droplets evaporate during the boiling process and the resulting phase change considerably increases the heat transfer between the impinging jet and the surface [6, 7].

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 first 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.

The CFD simulation of two-phase or multiphase flows consists in solving the fluid dynamics equations and in coupling the problems between the phases by jump relations at the interfaces taking into account the exchanges of mass, energy and momentum between the phases. Currently there are two approaches for the numerical calculation of multiphase flows: the Euler-Lagrange approach and the Euler-Euler approach [9-10].

In the Euler-Euler approach, the different phases are treated mathematically as interpenetrating continua and by introducing the concept of phase volume fraction. Conservation equations for each phase are derived to obtain a set of equations, which have similar structure for all phases. These equations are closed by providing constitutive relations that are generally obtained from empirical information. The Euler-Euler approach uses the notion of interfacial area concentration which is defined as the area of interfaces between two phases per unit mixture volume. This approach allows for heat and mass transfer between phases but does not seek to determine the properties of each particle present in the flow but to calculate local properties of the multiphase flow.

In the Euler-Lagrange approach (Discrete Phase Model), the fluid phase is treated as a continuum by solving the time-averaged Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of particles, bubbles, or droplets through the calculated flow field. The dispersed phase can exchange momentum, mass, and energy with the fluid phase. In this approach, the particle or droplet trajectories are computed individually at specified intervals during the fluid phase calculation. These modeling approaches have been compared in a previous work carried out by the present authors [12]. This previous study has in particular shown the strong dependence of the results to the model of turbulence used and to the size of the droplets forming the spray. The present study offers a more rigorous validation of the CFD model developed by confronting the numerical simulations with the measurements obtained on a test bench, in the case of the spray cooling of a hot metal plate which can represent part of an electric motor.

The influence of the distance between the spray nozzle exit and the surface impact is also analyzed. This paper is organized as follows. The following section describes the experimental setup and its instrumentation. The next section presents the CFD model. The last section discusses the results obtained and in particular the comparison between the numerical and experimental results.

Experimental Setup

The test bench designed to enable the experimental study of the spray cooling of a heated metal surface is presented in figure 1. It consists of a heating system, a spraying system and a data acquisition system. The spraying surface corresponds to the top surface of a copper block cylinder of 60 mm in diameter and 75 mm in height. This block is heated by means of a 400 W cartridge heater positioned in the lower part of the cylinder as shown in figure 2. The cartridge heater is supplied with a variac allowing the control of the voltage and therefore the control of the heat output. The entire copper block (except the top surface) is wrapped by thermal insulation material to minimize heat loss and to guarantee unidirectional heat conduction (in the cylinder axial direction).

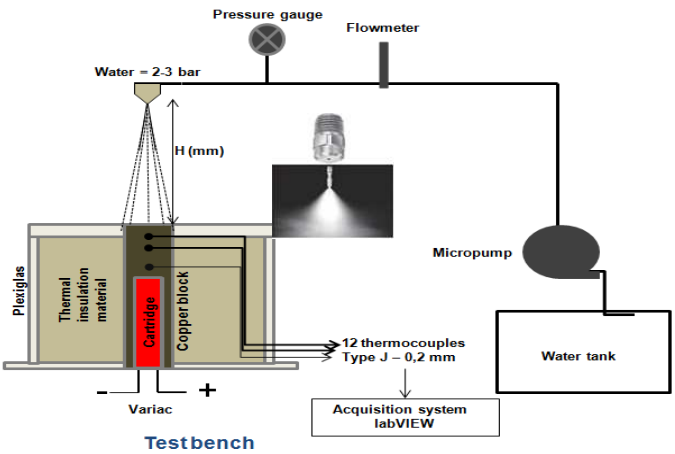

Figure 1 Spray cooling test bench

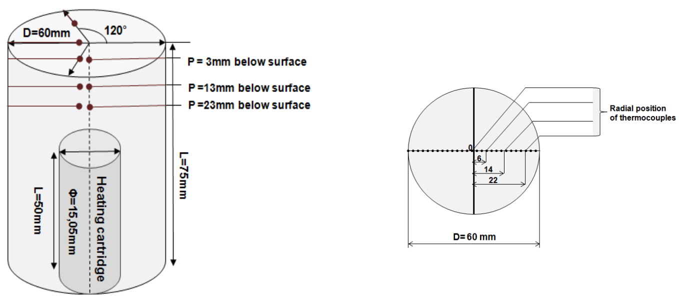

Figure 2 Heater assembly and thermocouples implementation

A spray nozzle (Fine Spray Hydraulic Atomizing-type1/4M-SS2 - spraying system) with a spray angle of 70° and an orifice diameter of 0.71 mm is used. The spray nozzle is supplied with water at 10 bar maximum pressure by means of a micro pump. As specified by the manufacturer, the spray nozzle used produces water droplets of about 214 μm Sauter Mean Diameter at a supply pressure of 3 bar. A flow meter (Brooks Instruments – R6-15B) and a Bourdon manometer are used to measure the flow rate and the inlet pressure of water, respectively. During our experiments, the water flow rate lies between 109 ml/min and 130 ml/min. The supply pressure of water is comprised between 2 and 3 bar and the water inlet temperature is maintained at 40°C. 12 J-type (iron-constantant) thermocouples, each having a 0.2 mm bead diameter, were embedded at various depths below the heater surface to provide the temperature gradient and temperature profile within the copper block cylinder. The detailed implementation of thermocouples is shown in figure 2. The heat flux was calculated using the one-dimensional Fourier's law of heat conduction and the surface temperature was determined by linear extrapolation of the measured temperatures. The thermocouple signals were collected using a high-density thermocouple module (NI9213 module) connected to a PC running LabVIEW.

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). During the present experiments, the spray nozzle was positioned above and perpendicular to the horizontally placed heated surface.

Numerical approach

The computation has been carried out using the commercial CFD software ANSYS-FLUENT. The steady state simulation uses the pressure based solver, which employs an implicit pressure-correction scheme and decouples the momentum and energy equations. The SIMPLE algorithm is used to couple the pressure and velocity. Second order upwind scheme is selected for spatial discretization of the convective terms.

The turbulence of the flow, which plays a dominating role in two-phase flows, is taken into account using different RANS models (Standard k-epsilon, Realizable k-epsilon and RNG k-epsilon models).

The two-phase flow phenomenon can be modeled in several ways In the CFD code ANSYS-FLUENT. The two-phase flow models used in this study are the Eulerian model (Euler-Euler approach) and the Discrete Phase Model (Euler-Lagrange approach).

In the present simulations, the Eulerian model uses the interfacial area concentration model of Hibiki-Ishii [11]. In the DPM simulation, the spray-wall interaction is taken into account via the wall film model [11]. According to Naber and Reitz [8], this model is appropriate for high-temperature walls where no significant liquid film is formed, and in high-Weber-number impacts where the spray acts as a jet. The droplet breakup regime is considered using the atomization regime.

Geometry and mesh have been generated using the pre-processor GAMBIT. Figure 3 shows the 2D axi-symmetric computational domain and the boundary conditions used. The bottom of the domain corresponds to the top of the heating cartridge. The spray nozzle exit is located in the top left corner of the domain. The distance H between the spray nozzle exit and the impact surface is variable during this study. The computational domain has been discretized to fine cells to conduct the simulation. Figure 3 shows the computational grid which contains approximately 40000 quadrilateral elements. To accurately predict temperature gradients and heat transfer, the cells have been clustered towards the impact surface to obtain appropriate y+ value less than 1.

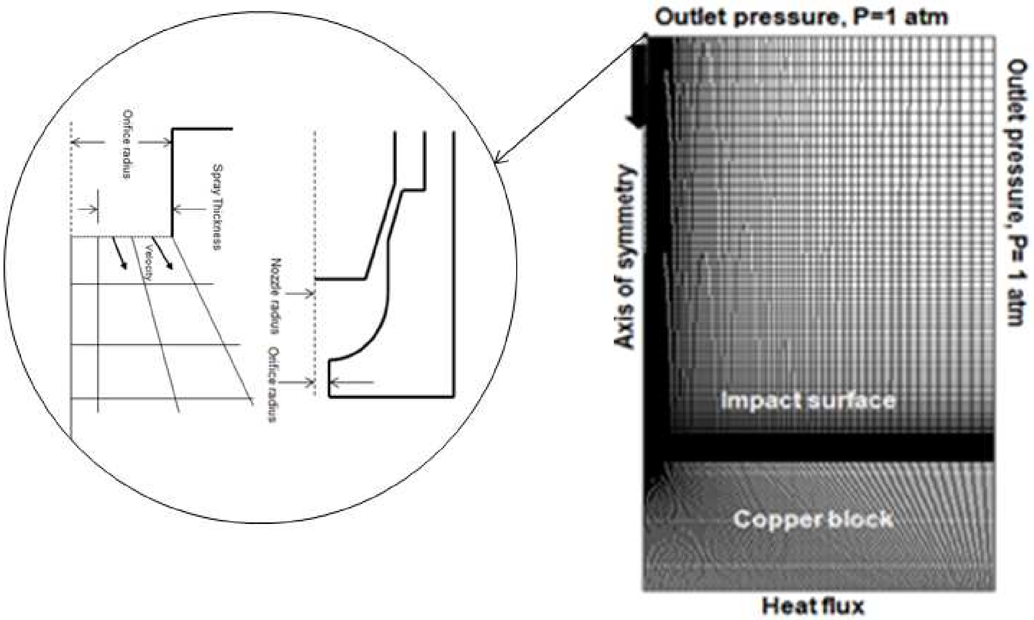

Figure 3 Computational domain and mesh grid

Results and Discussion

The results obtained relate primarily to the evolution of the local heat transfer coefficient and of the surface temperature along the plate.

A comparison of the DPM and Eulerian two-phase models was carried out initially. Figure 4 compares the evolutions of the coefficient of the heat transfer coefficient obtained experimentally and from simulations by using the same turbulence model (i.e the Realizable k-epsilon model). It may be noted that it is the Eulerian model which gives results more in agreement with the experiments. The difference between the measured and calculated values of heat transfer coefficient does not exceed 12%. Our Discrete Phase Model does not seem able to correctly predict the phase change which occurs on the surface of the plate and strongly underestimates the heat transfer. These results are confirmed by the radial distributions of the temperature at the upper surface of the plate shown in figure 5. One notes a very good agreement between the numerical results obtained with the Eulerian model and the experiments. It is thus the Eulerian model which is used in the continuation of this study.

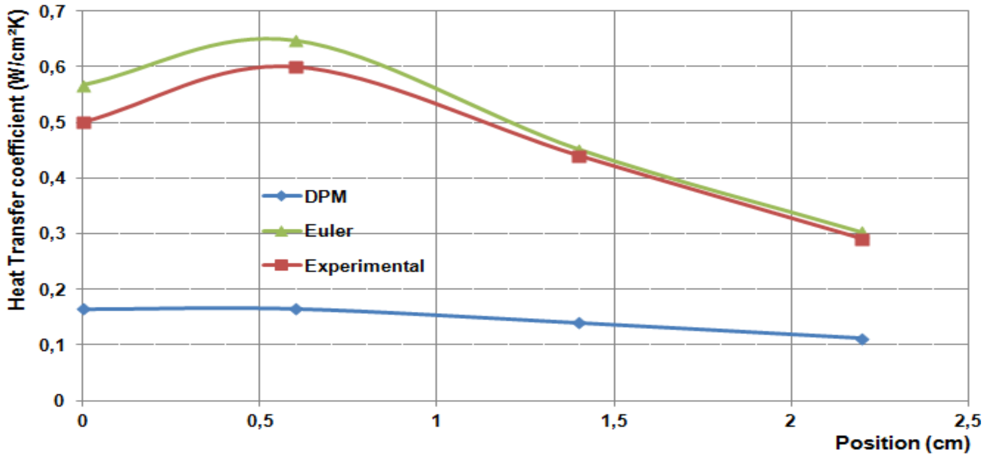

Figure 4 Radial evolution of heat transfer coefficient for DPM and Eulerian models
(Water pressure = 3 bar, Water flow rate = 131 ml/min,
Droplets mean diameter = 214 µm, Heating power = 300 W, H = 50 mm)

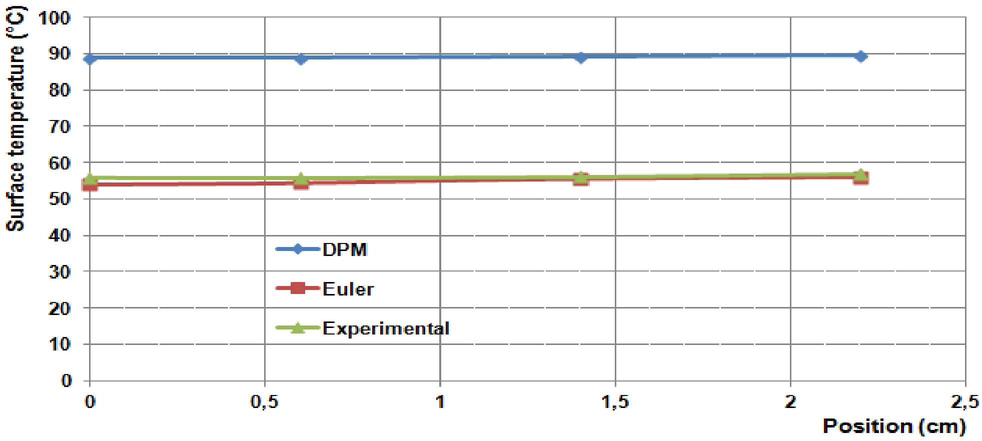

Figure 5 Radial evolution of surface temperature for DPM and Eulerian models
(Water pressure = 3 bar, Water flow rate = 131 ml/min),
(Droplets mean diameter = 214 µm, Heating power = 300 W, H = 50 mm)

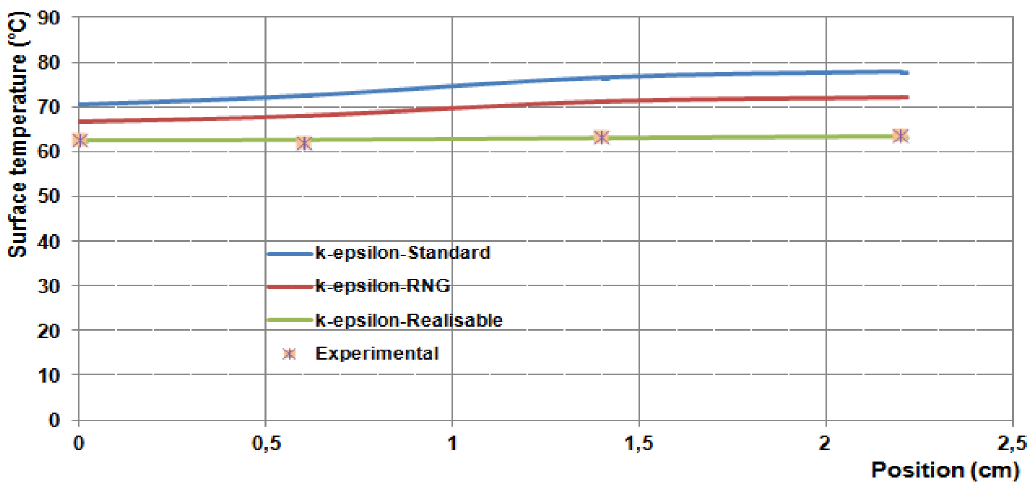

Figure 6 Radial evolution of surface temperature for different turbulence models
(Water pressure = 3 bar, Water flow rate = 131 ml/min),
(Droplets mean diameter = 214 µm, Heating power = 366 W, H = 50 mm)

Figure 6 illustrates the influence of the turbulence model used on the numerical prediction of heat transfer. It compares the radial distribution of the surface temperature obtained by using three RANS models. The differences between the different simulations are significant and it is the Realizable k-epsilon model, mainly dedicated to the flows with strong pressure gradients and recirculation regions, which gives the results closest to the experimental results.

It is thus the Realizable k-epsilon model which is used in the rest of this study, and in particular for the analysis of the influence of some parameters (such as the heating power and the spray nozzle distance) and for various operating conditions of the spraying system. Results obtained are presented in figures 7 to 12.

The results obtained while varying the heating power of the cartridge from 250 to 366 W (figure 8) show a very good experimental / numerical agreement and therefore confirm the capacity of the model CFD used to correctly predict the boiling heat transfer which occurs on the impact surface.

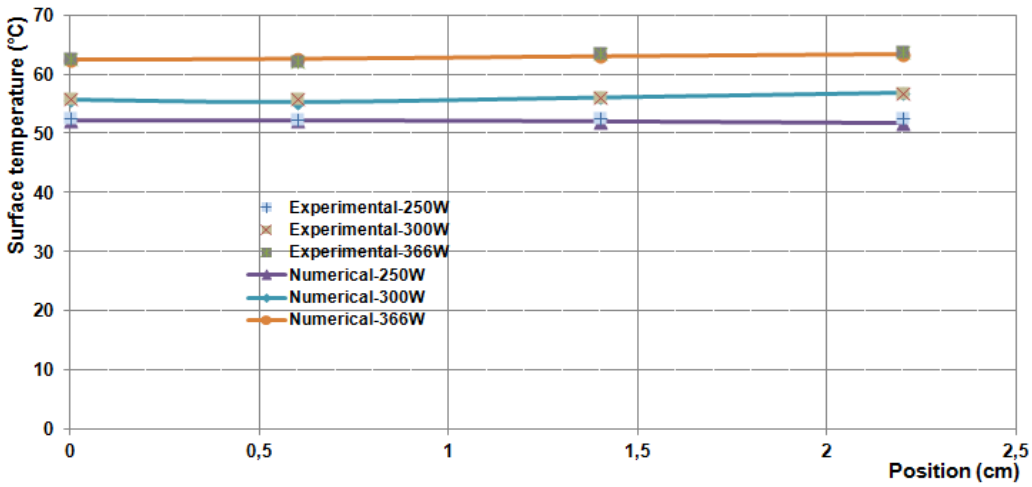

Figure 7 Radial evolution of surface temperature for different heating powers
(Water pressure = 3 bar, Water flow rate = 131 ml/min),
(Droplets mean diameter = 214 µm, H = 50 mm)

Figures 8 and 9 show the impact of the distance H between the nozzle and the impact surface on the cooling by spray. The numerical and experimental tests were carried out for several values of the distance H ranging between 10 and 70 mm. The agreement between the numerical and experimental is good, especially concerning the mean surface temperature (Figure 9). One notes a significant impact of the distance H on the cooling performance and an optimal distance (in the neighborhoods of 40 mm on our device) seems to emerge. At this distance, the angle of spray allows the total cover of the surface of impact by the spray jet.

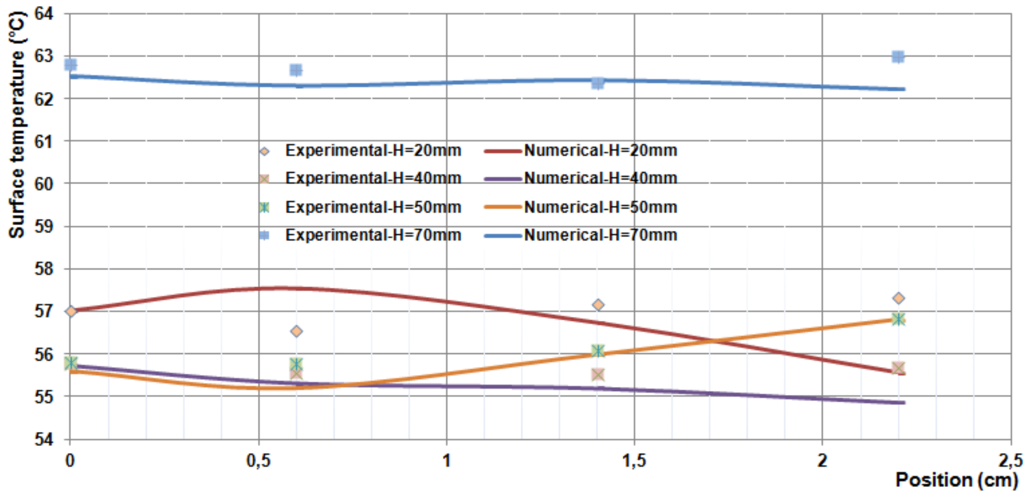

Figure 8 Radial evolution of surface temperature for different spray nozzle distances H
(Water pressure = 3 bar, Water flow rate = 131 ml/min),
(Droplets mean diameter = 214 µm, Heating power = 300 W)

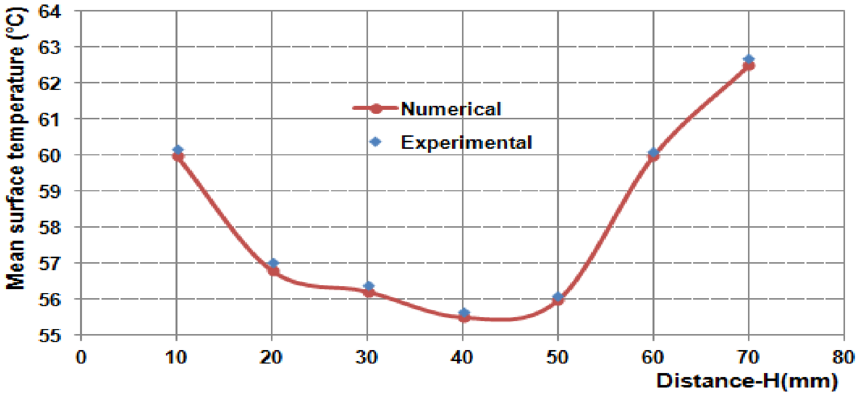

Figure 9 Evolution of mean surface temperature with the spray nozzle distance H (Water pressure = 3 bar, Water flow rate = 131 ml/min), (Droplets mean diameter = 214 µm, Heating power = 300 W)

Figures 10 and 11 presents results obtained experimentally and numerically under various operating conditions of the spray system, with a spray nozzle distance H of 50 mm comprised in the optimal range. Three cases of operation were considered while varying the supply pressure of the spray nozzle between 2 and 3 bar. As the supply pressure conditions the water flow rate and the droplets size, the characteristics of the spray under the operating conditions tested are summarized in table 1.

	Water presssure (bar)	Flow rate (ml/min)	Droplet Sauter Mean Diameter (µm)
Case 1	2	109	237
Case 2	2.5	119	224
Case 3	3	131	214

Table 1 Spray characteristics and operating conditions

The results obtained show strong variations of the surface cooling with the spraying system operating conditions. The distributions of temperature on the surface of the plate (figure 10) show a clear improvement of the cooling performance when the flow rate increases. A reduction in approximately 5°C of the mean surface temperature is observed both numerically and experimentally when the flow rate passes from 109 to 131 ml/min. This observation is confirmed by the evolutions of the heat transfer coefficient along the surface presented on figure 11. Once again, the agreement between the numerical and experimental results is satisfactory

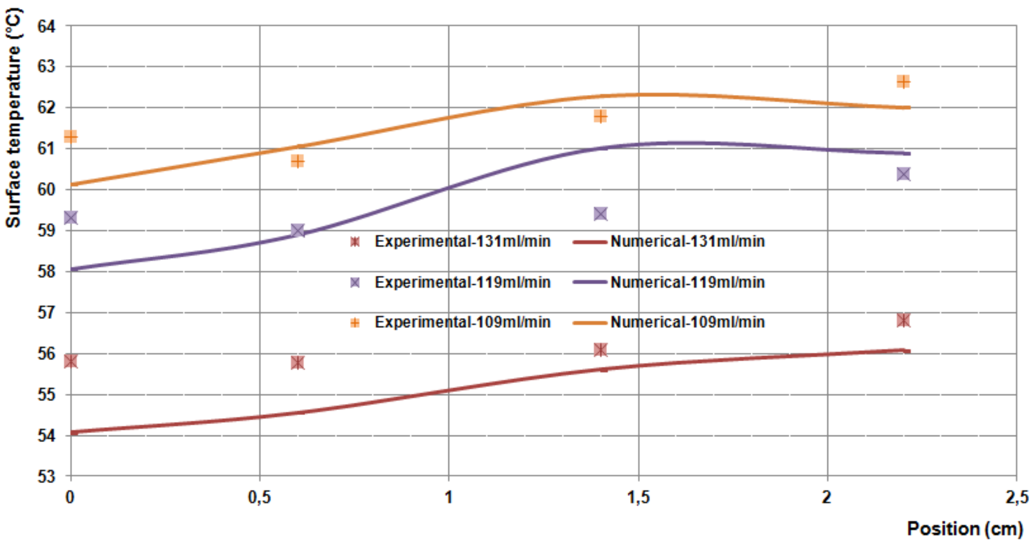

Figure 10 Radial evolution of surface temperature for different water flow rates (Heating power = 300 W, H = 50 mm)

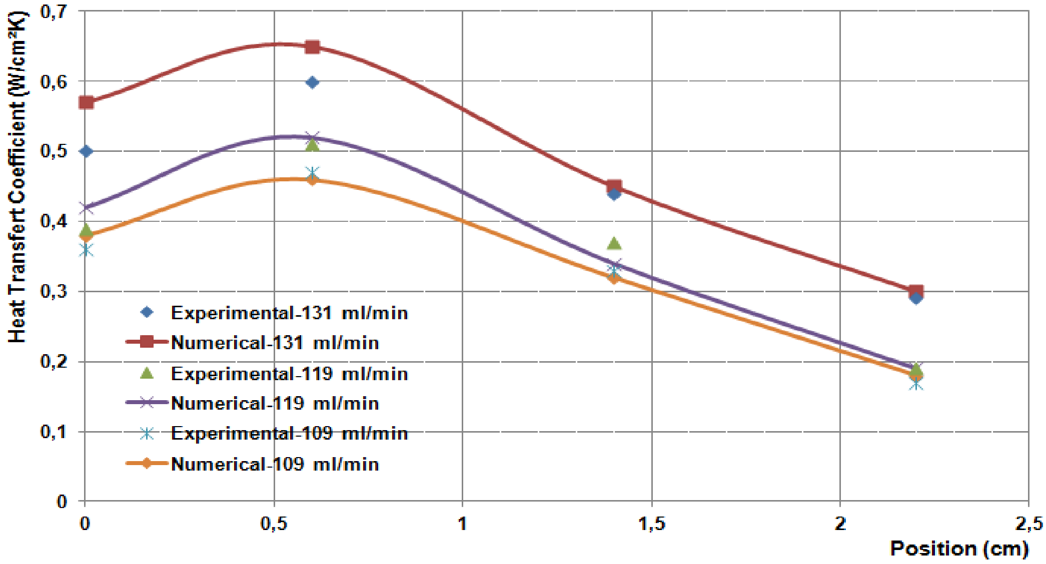

Figure 11 Radial evolution of heat transfer coefficient for different water flow rates (Heating power = 300 W, H = 50 mm)

Summary and conclusions

In order to correctly dimension a spraying system for cooling electric motors, a numerical model making it possible to predict the heat transfer with phase change between a heated surface and a two-phase impinging jet was developed. This model was implemented in a commercial CFD code and validated against experiments carried out on an experimental test bench specifically dedicated for this purpose. Different two-phase modeling approaches (Lagrangian and Eulerian methods) and several RANS turbulence models were tested. 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 and for various conditions of heating.

It may be noted that due to the low heat flux densities used, the surface temperature remains significantly below the vaporization temperature of water. This underlines the minor role played by evaporation on the surface in our operating conditions. However, the comparison between simulations and experiments shows the capability of the model to correctly predict the phase change heat transfer during the spray cooling. It must allow, in a short time, the optimization of the spraying system before its implementation on an electric motor.

References

[1] Bertin Y., Refroidissement des machines électriques tournantes, Technique de l'ingénieur, D3460, (1999).

[2] Farsane K., Desevaux P. and Panday P.K., Experimental study of the cooling of a closed type electric motor, Applied Thermal Engineering, 20: 1321-1334 (2000).

[3] Hsieh C.C., Yao S.C., Evaporative heat transfer characteristics of a water spray on micro-structured silicon surfaces, International Journal of Heat and Mass Transfer, 49, 962 -974, 2006

[4] Hall, D. D, Issam Mudawar, Experimental and numerical study of quenching complex-shaped me tallic alloys with multiple, overlapping sprays , *International journal of heat and mass transfer* 38 (7): 1201–1216 (1995).

[5] Puschmann F., Specht E., and Schmidt J., Local distribution of the heat transfer in water spray quenching, International Conference on Continuous Casting of Non-Ferrous Metals, Proceedings of the DGM Frankfurt, 101-107, 2000.

[6] Hsieh, S. S, T. C Fan, H. H Tsai, Spray cooling characteristics of water and R-134a. Part I: nucleate boiling, *International Journal of Heat and Mass Transfer* 47 (26): 5703–5712 (2004a).

[7] Hsieh, S. S, T. C Fan, H. H Tsai., Spray cooling characteristics of water and R-134a. Part II: transient cooling , *International Journal of Heat and Mass Transfer* 47 (26): 5713–5724 (2004b)

[8] Naber J.D., Reitz, R.D., Modeling Engine Spray/Wall Impingement, *SAE paper 880107* (1988)

[9] A, Vallet, Burluka AA, Borghi R, Development of a Eulerian model for the atomization of a liquid jet, *Atomization and Sprays* 11 (6): 619–642 (2001).
[10] Nijdam, J.J., B. Guo, D.F. Fletcher, et T.A.G. Langrish, Lagrangian and Eulerian models for simulating turbulent dispersion and coalescence of droplets within a spray, *Applied mathematical modelling* 30 (11) 1196–1211 (2006).
[11] ANSYS FLUENT, www.ansys.com/fluent,
[12] Guéchi M, Desevaux P, Baucour P, Etude numérique du refroidissement d'une surface métallique par pulvérisation (spray cooling), 20ème Congrès Français de Mécanique, Besançon 2011