ISBN: 978-3-95450-250-9 ISSN: 2673-5520 doi:10.18429/JACoW-MEDSI2023-THPPP016

NUMERICAL AND EXPERIMENTAL STUDIES TO EVALUATE THE CONSERVATIVE FACTOR OF THE CONVECTIVE HEAT TRANSFER COEFFICIENT APPLIED TO THE DESIGN OF COMPONENTS IN PARTICLE ACCELERATORS

M. Quispe†, J. J. Casas, C. Colldelram, M. Sanchez, ALBA-CELLS, Cerdanyola del Vallès, Spain
R. Capdevila, M. Rabasa, G. Raush, ESEIAAT, Terrassa, Spain
S. Grozavu, Universidad Politecnica de Madrid, Madrid, Spain
H. Bello, La Romanica, Barberà del Vallès, Sabadell, Spain

Abstract

The fluid boundary condition applied to the design of components in particle accelerators is calculated as a global variable through experimental correlations coming from the literature. This variable, defined as the convective heat transfer coefficient, is calculated using the conventional correlations of Dittus and Boelter (1930), Sieder and Tate (1936), Petukhov (1970), Gnielinski (1976), among others. Although the designs based on these correlations work properly, the hypothesis of the present study proposes that the effectiveness of these approximations is due to the existence of a significant and unknown conservative factor between the real phenomenon and the global variable. To quantify this conservative factor, this work presents research based on Computational Fluid Dynamics (CFD) and experimental studies. In particular, recent investigations carried out at ALBA confirm in a preliminary way our hypotheses for circular pipes under fully and non-fully developed flow conditions. The conclusions of this work indicate that we could dissipate the required heat with a flowrate lower than that obtained by applying the conventional experimental correlations.

INTRODUCTION

Nowadays, in particle accelerator engineering and in engineering in general, numerical simulations, such as FEA (Finite Element Analysis) and CFD (Computational Fluid Dynamics), are decisive to approve the viability of a proposed design. Although its importance is recognized, it is also known that the results of numerical simulations have a strong dependence on the precision and good approximation of other variables such as the geometric model, physical properties, boundary conditions, etc. In this context, the content of this work is oriented to the study of one of the boundary conditions commonly used in design: the convective heat transfer coefficient (h) for internal flow in cooling channels. In particular, at ALBA we are studying the conservative factors inherent in the "h" coefficient, currently obtained from experimental correlations reported in the literature. Our main working hypothesis considers the existence of a significant and not yet quantified conservative factor in the calculation of the "h" coefficient. The results of this study will be relevant for the design of the new components of ALBA II, our current project to become a fourth-generation accelerator. From the point of view of accelerator engineering, we will have the challenge of designing the new components for higher power densities, compared to ALBA I. In this new scenario, it will be important to know with a better precision the value of the "h" coefficient to avoid oversizing in the new designs.

For general engineering applications, the "h" coefficient is obtained from experimental correlations from the literature, reported by authors such as Dittus and Boelter (1930), Sieder and Tate (1936), Petukhov (1970), Gnielinski (1976) [1], among others. The design engineer must choose between those authors to obtain this coefficient, whose value is not unique depending on the selected experimental correlation. For example, for a hypothetical case of water at 23 ºC circulating in a pipe with an internal diameter of 10 mm and assuming a dissipation of 7 kW in the water, there is a difference of approximately 10 % between the values of "h" calculated with the correlations of Dittus – Boelter and Petukhov.

Another aspect to highlight of the "h" coefficient is the condition of its approximation: the experimental correlations have been formulated for thermal and hydraulic fully developed flow conditions. In real applications of particle accelerators, we rarely have fully developed flow, because the geometries are small in size, such as the cooling channels of mirrors, monochromators, front end masks, radiation absorbers, etc. These real geometries would increase the unknown conservative factor with respect to the case of fully developed flow, according to our hypothesis. On the other hand, from the point of view of the real phenomenon, conventional correlations assume homogeneity of the coefficient along the cooling channel, which is not true because this variable has local behaviour and its distribution is influenced by the geometry of the channel, by the flow conditions (especially for transient and turbulent cases), and by the temperature of the fluid.

In the same line of research, another variable to study is the approximation of the hydraulic diameter concept. In many applications we are forced to design cooling channels with non-circular cross sections. In these cases, the application of the hydraulic diameter concept suggested by conventional references introduces, in our opinion, a new conservative factor with respect to the case of a circular tube.

The investigations of this paper are based on CFD calculations, Heat Transfer (HT) simulations, and preliminary experimental studies in setups developed at ALBA. The HT simulation approximates the heat transfer in the fluid

†mquispe@cells.es

ISBN: 978-3-95450-250-9 ISSN: 2673-5520 doi:10.18429/JACoW-MEDSI2023-THPPP016

using the "*h*" coefficient, calculated through experimental correlations reported in the literature [1]. The HT and CFD cases have been carried out using the MECHANICAL and FLUENT modules of ANSYS WORKBENCH [2], respectively.

CIRCULAR CHANNELS: CFD STUDIES

Model Description

Two pipes with an internal diameter of 8 and 10 mm have been studied [3, 4], both 0.5 m lengths. The heat flux applied to the surface are assumed to be constant values of 80 and 12.55 kW/m^2, respectively (Fig. 1a). At the inlet, water at 23 ºC and a velocity range < 4 m/s are fixed.

O-grid structured mesh has been applied along all the wall boundaries inside the fluid, inflation layers has been introduced for a smooth transition of the mesh until reaching a $y+\approx 1$ [5] (Fig. 1b). Also, a grid convergence study of three levels of mesh refinement has been performed, implementing the convergence Python program provided as a part of the NASA Examining Spatial (Grid) Convergence tutorial [6]. The highest refinement has generated a mesh of around 2.5 million elements.

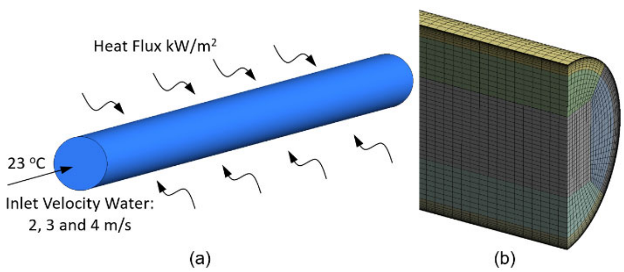

Figure 1: (a) Pipe simplified model, (b) Mesh detail.

A group of viscous models have been tested such as the $k-\omega$ Shear-Stress Transport (SST), $k-\omega$ Standard, the Realizable k-ε with Scalable Wall Functions (RKE ScWF), the Realizable k-ε with Enhanced Wall Treatment (RKE EWT) and the Transition SST [7]. The results have been compared with the Darcy–Weisbach equation for pressure drop and the Power-Law equation for the developed velocity profile, in order to select the most accurate. For this comparative study, according to studies carry out by [3] and [4], the authors agree on the better performance of the $k-\omega$ Shear-Stress Transport (SST) model.

For the studies, the Nusselt number $Nu = hD/k$ is computed, where D is the diameter of the channel and k the thermal conductivity of the fluid. The "*h*" coefficient is computed using the Newton's law of cooling $h = Q/(A(Tw\text{-}Tf))$, where Q is the heat transfer rate across the area A, Tw is the wall temperature and Tf the fluid bulk temperature. Tf is derived from the rate of flow of enthalpy divided by the rate of heat flux through a cross section, like defined in Neale's study [8], which can be calculated in ANSYS FLUENT as the Mass Flow Average of the temperature of a transversal area of the fluid.

Results

For the case of pipe internal diameter 8 mm, the results from Fig. 2 suggest that experimental benchmark coefficients are conservative. For instance, comparing CFD results with Dittus and Boelter's correlation, an increase between 12.6 and 13.8 % of the convective heat transfer coefficient has been found.

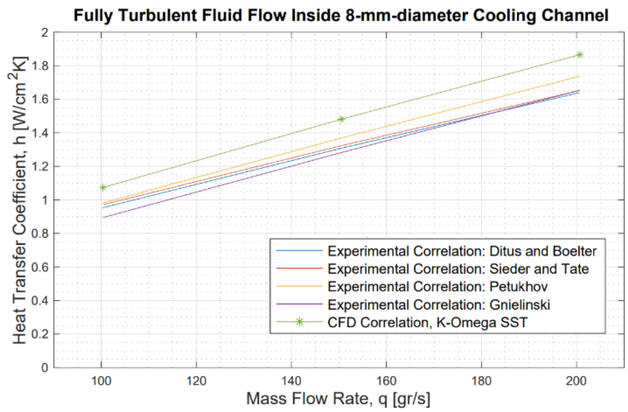

Figure 2: Experimental correlations of the "*h*" coefficients contrasted to the CFD values for circular channel flow 8 mm diameter.

For a pipe of 10 mm inner diameter, the different viscous models offer quite different results between themselves (Fig. 3). Compared with Dittus – Boelter, the average variation with the models are 25.9 %, 20.1 % and –3.4 % for the k-ω SST, RKE EWT and RKE ScWF, respectively. This last model deserves special attention for new studies, because its discrepancy is significant with respect to the other turbulence models.

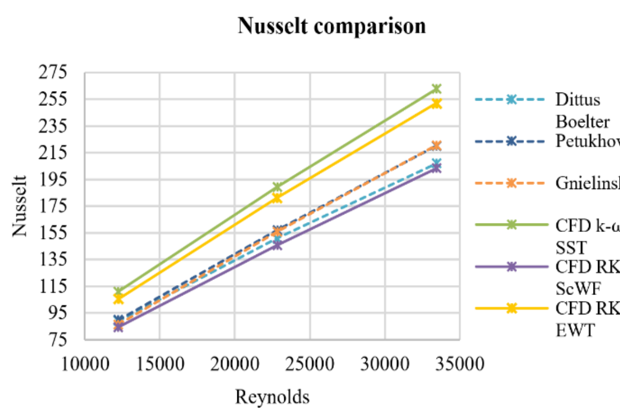

Figure 3: Comparison of Nusselt calculated by CFD to experimental correlations for channel 10 mm diameter.

The effect of the heat flux is also studied. The CFD Nusselt number (Nu CFD) is calculated based on the k-ω SST model, at 3 m/s inlet velocity and different heat fluxes. The results presented in Table 1 show that higher heat flux values the higher differences of the Nu CFD compared to the experimental correlations.

ISBN: 978-3-95450-250-9 ISSN: 2673-5520 doi:10.18429/JACoW-MEDSI2023-THPPP016

Table 1: Increase of Nu CFD (K-ω SST) for Different Heat Fluxes in Respect to Experimental Correlations

Heat Flux (W/m^2)	ΔNu CFD - Dittus Boelter	ΔNu CFD - Petukhov	ΔNu CFD - Gnielinski
170000	29.38 %	22.07 %	21.70 %
125464	26.86 %	19.31 %	19.03 %
80000	24.23 %	17.20 %	17.00 %

MIRROR: CFD, HT, AND EXPERIMENTAL STUDIES

Model Description

This section presents numerical and experimental studies for non-fully developed flow conditions. The model reproduces a mirror with internal cooling channel. The geometry consists in an Al 6082 T6 block of 60×60×150 mm with a 10 mm diameter hole where the water flows through (Fig. 4 a). As boundary conditions, heat flux on the top surface (35.85 and 43.4 W) and fluid velocities of 1 and 2 m/s are imposed. For the experiment, heat flux is applied using two 65×11 mm heater foils (Fig. 4 b).

Three thermocouples type K are placed to measure surface temperature, as shown in Fig. 4b, and insulation is achieved using an aluminium foil layer underneath a fibre glass wool layer. The experimental setup used was developed at ALBA for hydraulic and thermal testing. It allows the user to regulate the flow rate and inlet temperature [9].

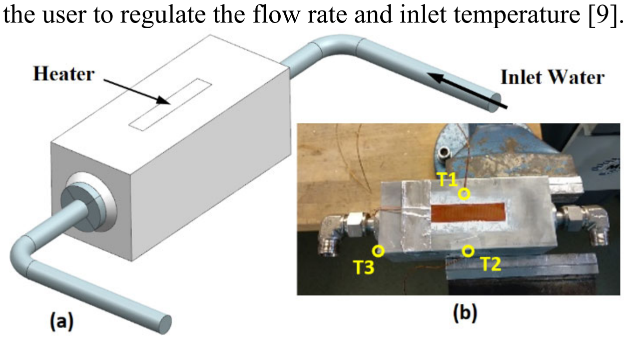

Figure 4: (a) Mirror simplified model, (b) Details of temperature sensors and heater for experiment.

For the CFD model, a mesh study is performed for three meshes (1.6M, 2.8M and 4M) with a $y+\approx 1$. HT simulations have also been carried out by applying the "*h*" coefficients, calculated at the average temperature of the fluid in the cooling channel.

Results

Figure 5 shows the results of three cases studied for different heat fluxes and inlet velocities. The temperature results applying the HT simulation are higher than the temperature results calculated with the CFD models described in the previous section. On the other hand, the results obtained with the RKE ScWF model are closer to the temperatures using the HT simulations. The results obtained with the RKE EWT and k-ω SST models show almost exactly the same temperature results, this behaviour is reproduced for all velocities and heat fluxes studied. The CFD results, applying the k-ω SST and RKE EWT models, are generally closer to the experimental results compared to the results obtained with the HT simulations.

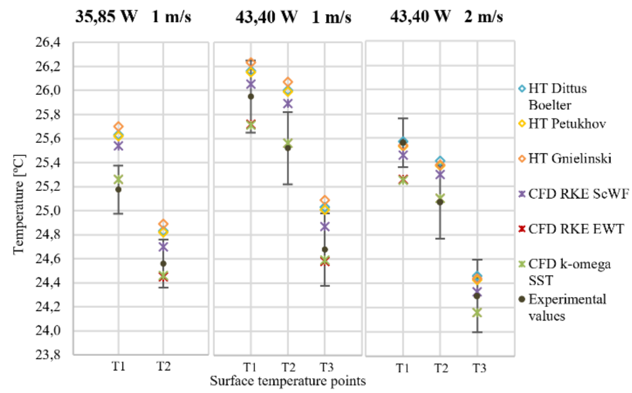

Figure 5: Results of numerical simulations and experimental studies for three study conditions.

Figure 6 shows the distributions of temperatures and velocities for the CFD model, considering the k-ω SST turbulence model, inlet velocity into the tube of 1 m/s and the value of the heat flux equals 43.4 Watts.

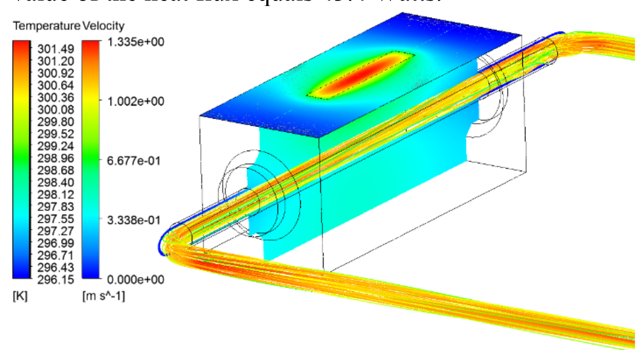

Figure 6: Velocity and temperature distributions for the case CFD k-ω SST, 43.4 W, and 1 m/s inlet.

CONCLUSIONS & FUTURE WORK

For the case of fully developed flow conditions in pipes with an internal diameter 8 mm, the CFD calculations (applying the k-ω SST viscous model) confirm the existence of an average conservative value of 14 % in the conventional convective heat transfer coefficient (taking as reference the Dittus and Boelter correlation). For the pipe of 10 mm diameter, average variations of 25.9 % and 20.1 % are obtained when the "*h*" coefficient (based on the Dittus and Boelter correlation) is compared with CFD calculations (based on the models of turbulence k-ω SST and RKE EWT, respectively). Then, this second diameter also confirms the existence of the conservative factor.

It has been found that the conservative factor is also affected by the heat flux condition: the conservative factor increases as the heat flux condition increases.

The experimental results for the proposed case also confirm the existence of the conservative value for non-fully developed flow conditions. However, to have a definitive conclusion, it is recommended to carry out similar experiments subject to higher heat fluxes.

ISBN: 978-3-95450-250-9 ISSN: 2673-5520 doi:10.18429/JACoW-MEDSI2023-THPPP016

REFERENCES

[1] W. M. Rohsenow, J. P. Hartnett, and Y. I. Cho, *Handbook of heat transfer*, McGraw-Hill Education, 1998. doi:10.1016/s0278-6125(99)90104-8

[2] ANSYS, https://www.ansys.com

[3] S. Grozavu *et al.*, "CFD Studies of the Convective Heat Transfer Coefficients and Pressure Drops in Geometries Applied to Water Cooling Channels of the Crotch Absorbers of ALBA Synchrotron Light Source", in *Proc. IPAC'22*, Bangkok, Thailand, Jun. 2022, pp. 2887-2890. doi:10.18429/JACoW-IPAC2022-THPOTK050

[4] M. Rabasa *et al.*, "CFD Studies and Experimental Validation of the Convective Heat Transfer Coefficient in Non-Fully Developed Flows Applied to Conventional Geometries Used in Particle Accelerators", in *Proc. IPAC'23*, Venice, Italy, May. 2023, pp. 4912-4915. doi:10.18429/JACoW-IPAC2023-THPM010

[5] SimScale CAE Forum, https://www.simscale.com/forum/t/what-is-y-yplus/82394

[6] NASA Examining Spatial (Grid) Convergence, https://www.grc.nasa.gov/www/wind/valid/tutorial/spatconv.html

[7] "Turbulence", in ANSYS FLUENT 12.0 Theory Guide, ANSYS Inc., 2009.

[8] A. Neale, D. Derome, B. Blocken, and J. Carmeliet, "CFD calculation of convective heat transfer coefficients and validation– Part I: Laminar flow", in *Proc. IEA Annex 41 working meeting*, Kyoto, Japan, Apr. 2006.

[9] M. Quispe, "Modular Setup for Thermal Hydraulic Measurements at ALBA Storage Ring", unpublished.