ISBN: 978-3-95450-231-8 ISSN: 2673-5490 doi: 10.18429/JACoW-IPAC2023-THPM010

CFD STUDIES AND EXPERIMENTAL VALIDATION OF THE CONVECTIVE HEAT TRANSFER COEFFICIENT IN NON-FULLY DEVELOPED FLOWS APPLIED TO CONVENTIONAL GEOMETRIES USED IN PARTICLE ACCELERATORS

M. Rabasa, R. Capdevila, G. Raush, School of Industrial,
Aerospace and Audiovisual Engineering of Terrassa (ESEIAAT), Terrassa, Spain
J. Casas, C. Colldelram, M. Quispe, M. Sanchez, ALBA-CELLS Synchrotron,
Cerdanyola del Vallès, Spain
H. Bello, La Romanica, Barberà del Vallès, Spain

Abstract

In the field of Particle Accelerators engineering, the design of the cooling channels of its components has been extensively based on experimental correlations for the calculation of convective heat transfer coefficients. In this scenario, this work is focused on studying whether the experimental correlations are conservative when the flow is turbulent in fully developed and non-fully developed regions.

For this research, simulation models have been developed for turbulent flows in fully developed and non-fully developed regions, all of them for cooling channels with a 10 mm inner diameter. In the first case, for a circular channel, turbulence models have been studied, and comparative studies with respect to experimental correlations and previous studies performed at ALBA have been carried out. Simulation models based on the coefficients obtained from experimentally observed correlations, CFD models and an experimental validation of a mirror with inside cooling, have been performed in the second case.

INTRODUCTION

Nowadays, the values of different experimental correlations for the Nusselt number (Dittus - Boelter, Sieder - Tate, Petukhov, Gnielinski, among others [1]), are widely used in the design of cooling systems in particle accelerator geometries. Recent studies at ALBA synchrotron indicate that these values are lower compared to CFD results in conventional channels for turbulent fully developed flows [2]. On the other hand, in real applications, the dimensions of the cooling channels do not correspond, in general, to fully developed flow geometries.

The objective of this research is to perform Heat Transfer (HT) and Computational Fluid Dynamics (CFD) simulations of existing geometries in ALBA Synchrotron and the experimental validation of the results, in order to study the convective heat transfer coefficient and see if the values of existing experimental correlations are conservative in comparison with the CFD.

A first geometry in fully developed flow conditions will be studied to be compared with experimental correlations and a second geometry in non-fully developed conditions will be simulated and experimentally validated.

METHODOLOGY

In favour of achieving the objective, two phases are developed in this research. For the numerical simulations, ANSYS FLUENT and ANSYS MECHANICAL tools have been used.

Phase 1: CFD simulations of a circular channel tube with fully developed turbulent flow and a constant heat flux. It is a reproduction of Grozavu's first stage with a different diameter [2]. The aim of this phase is to develop a meshing strategy and test different Reynolds-Averaged Navier–Stokes (RANS) viscous models to be applied in phase 2b.

Phase 2a: Experimentation of a mirror absorber geometry with inside cooling, applying a constant heat flux in non-fully developed turbulent flow.

Phase 2b: Simulations of the experimented cases in phase 2a. CFD simulations, including the fluid and solid parts, and HT simulations of the solid body, using the Steady State Thermal package, where convection based on the experimental correlations is applied as a boundary condition in the inside cooling channel of the absorber.

Phase 1: Circular Channel CFD

The geometry is a 10 mm interior diameter smooth pipe with a 0.5 m length, above hydrodynamic and thermal entry lengths, with a constant arbitrary heat flux of 125464 W/m^2 and 23 ºC as inlet water temperature.

A grid study is performed with four different meshes (2.6M, 1.8M, 1.3M and 0.7M) with a dimensionless wall distance $y^+ \approx 1$ [3] and a fluid velocity of 3 m/s, as it is ALBA's maximum fluid flow, with the k-ω SST model [4] and good convergence in all cases.

A viscosity model study with the previous conditions is carried out using the k-ω SST, the Realiazable k-ε with Scalable Wall Functions (RKE ScWF), the Realiazable k-ε with Enhanced Wall Treatment (RKE EWT) and the Transition SST models [4]. RKE models show slight sensitivity to the mesh so the finest one is used for these models.

For correlations comparison purposes, the Nusselt value, $Nu = hD/k$, where h is the heat transfer coefficient, D the diameter of the channel and k the thermal conductivity, is computed as an average between pipe length x=0.45 m and x=0.5 m. h is computed using Newton's law of cooling,

ISBN: 978-3-95450-231-8 ISSN: 2673-5490 doi: 10.18429/JACoW-IPAC2023-THPM010

$h = Q/\left(A\left(T_w - T_f\right)\right)$, where Q is the heat transfer rate across an area A, T_w the wall temperature and T_f the fluid bulk temperature. T_f is derived from the rate of flow of enthalpy divided by the rate of heat flow through a cross section, like defined in Neale's studio [5], which can be obtained in ANSYS FLUENT as the Mass Flow Average of the temperature of a transversal area of the fluid [2].

Phase 2a: Mirror Experimentation

The tested mirror consists in an aluminium 6082 T6 bloc of 60×60×150 mm with a 10 mm diameter hole where the refrigerating fluid flows through. To recreate the effect of the synchrotron radiation, a heat flux is applied to the top of the model using a 65×11 mm heater foil. Three thermocouples type K are placed to measure surface temperature, as shown in Fig. 1, and insulation is achieved using an aluminium foil layer underneath a fibre glass wool layer.

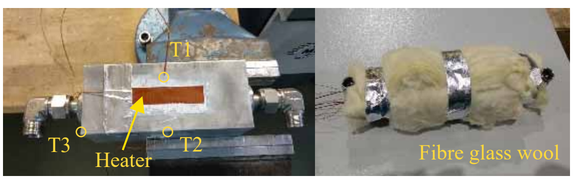

Figure 1: Experimental model.

The experimental setup used was developed at ALBA for hydraulic and thermal testing. It allows the user to set flow rate and inlet temperature, as well as read inlet and outlet temperature values [6].

Despite having tried different strategies to regulate inlet water temperature to a fixed setpoint, there is an oscillation of 2 ºC around the desired 23 ºC temperature. With measures every two seconds, a small sample of 20 points is randomly selected within a ±0.2 ºC interval of the average inlet temperature value. For these samples, the average values are computed for inlet, outlet and surface temperatures, as well as flow rate. The 'two sigma rule' is used to report the uncertainty of the results [7].

Phase 2b: Mirror CFD and HT Simulations

Figure 2 shows the geometry for the simulations of the experimented model. The CFD includes the elbows and part of the pipe, whereas the HT geometry is just the bloc.

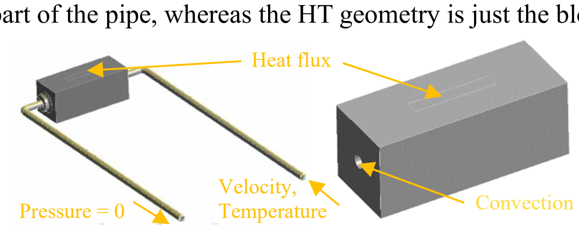

Figure 2: CFD (left) and HT (right) geometries and boundary conditions.

For the CFD model, a grid study is performed for three meshes (1.6M, 2.8M and 4M) with a $y^+ \approx 1$ and the RKE ScWF model [4], which was the most sensitive to mesh in Phase 1. The boundary conditions values, such as inlet temperature, heat flux and flow rate, are based on the experimental results for each case. The best three viscosity models of Phase 1 are used in this one.

A 0.2M mesh is used in the HT simulations. For the boundary conditions, a heat flux is applied on the top of the model and convection inside the cooling channel, with the heat transfer coefficient and fluid bulk temperature as parameters. All values are based or calculated using the experimental results, with fluid properties at average temperature between inlet and outlet.

RESULTS

Phase 1: Circular Channel CFD

Results for the grid study show almost the same values for pressure drop, wall shear stress and velocity and temperature profiles. The coarsest mesh shows slightly different results for wall temperature. The 1.3M mesh is used for further simulations.

For the viscous model study, hydraulic results and temperature profiles are similar. Compared to Darcy-Weisbach [8] and the velocity profiles with the 'power law', the 'logarithmic law' and the 'law of the wall' [9], the k-ω SST and the RKE EWT models perform better. Wall temperature results differ substantially between models and therefore Nusselt values. The Transition SST model is discarded as it results in variations above 40% in comparison to Dittus-Boelter, Petukhov and Gnielinski correlations.

Three different inlet velocity cases are simulated and the respective computed Nusselt values are compared to experimental correlation results, as shown in Fig. 3.

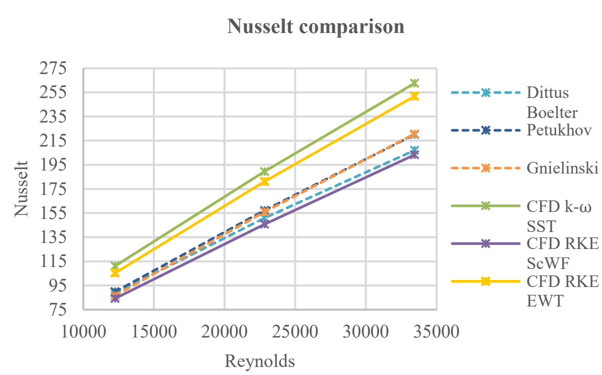

Figure 3: Comparison of Nusselt obtained CFD values to experimental correlations [1] for the same case.

Observing the values, the different viscous models offer quite different results between themselves. Compared with Dittus Boelter, the most conservative correlation and the one usually used at ALBA, 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. It can be concluded that the RKE EWT gives more accurate results for the Nusselt computation with respect to the k-omega SST model. For the RKE ScWF, although more conservative, the values are significantly closer to the correlations in comparison with the other models.

Simulations with the k-ω SST model, 3 m/s inlet velocity and different heat fluxes are performed to be compared

ISBN: 978-3-95450-231-8 ISSN: 2673-5490 doi: 10.18429/JACoW-IPAC2023-THPM010

with Grozavu's first stage case, with only the diameter as difference [2]. The results for the 80000 W/m^2, 8 mm pipe showed an increase of around 12 % with Dittus Boelter [2], whereas in this case, the difference for the 10 mm pipe, observed in Table 1, is around 24 %. Moreover, higher heat flux values result in higher differences of the Nusselt compared to the experimental correlations.

Table 1: Increase of CFD Computed Nusselt 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%

Phase 2: Mirror Experimentation and Simulations

For the CFD grid study, results for surface temperature differ slightly in the decimals with a relative error between the 2.8M and the 4M mesh of 0.005%. The first mesh is chosen for the simulations.

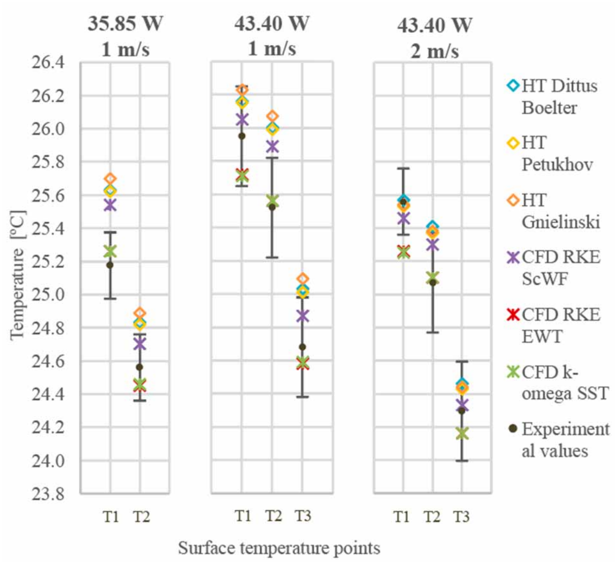

Figure 4: Simulations and experimental results comparison for surface temperature values in Phase 2.

Figure 4 shows three cases for different heat fluxes and inlet velocity. As a general behaviour the HT models based on experimental correlations for the convective heat transfer coefficient obtain higher temperatures than the CFD, which implies less heat transfer with the fluid and therefore a lower heat transfer coefficient. These results are in accordance with the values obtained for the Nusselt in Phase 1 for the RKE EWT and the k-ω SST models, as the CFD obtained higher values. Moreover, the RKE ScWF, which was the model closer to the correlations with the fully developed circular channel case, shows a similar nature in these ones, although in this case the heat transfer is higher than the correlations. The RKE EWT and the k-ω SST, which had similar results in Phase 1, show almost exactly the same temperature results in these simulations, despite differences in entry velocity profiles. An example of the simulation results is shown in Fig. 5.

The CFD models, specially the k-ω SST and the RKE EWT, are generally closer to the experimental results compared with the HT simulations. However, given the uncertainty of the experimental results, almost all simulations are in accordance with them in some cases, especially when there is a higher heat transference like the 2 m/s case. It can also be observed that the T1 thermocouple, which was placed after the 35.85 W test for more information, gets higher temperature results in contrast with the simulations than the other sensors. This difference could be due to the proximity with the heater and the inaccuracies that come with the modelling of the heat flux in the simulations.

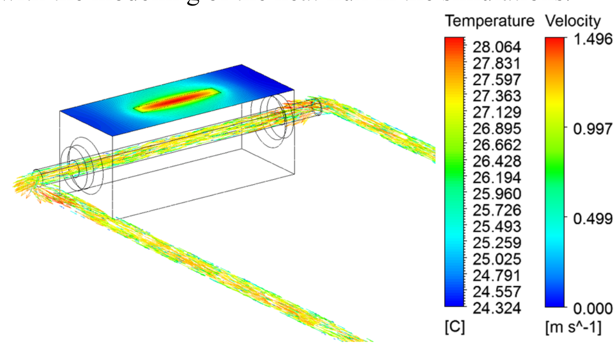

Figure 5: CFD k-ω SST, 43.4 W, 1 m/s case, velocity and top temperature distributions.

CONCLUSIONS & FUTURE WORK

In respect to the results of Phase 1, it can be concluded that the different viscous models have really different behaviours concerning wall temperature and thus heat transfer, generally predicting significantly higher heat transfer coefficients compared to the experimental correlations. Nusselt values also seem to be affected by the pipe diameter and heat flux, with higher values for these parameters resulting in higher differences of the Nusselt with the correlations. Future research in this topic would be interesting.

In Phase 2, despite the equipment being of really good quality, the bad regulation of the inlet temperature could result in inaccuracies for the experimental results that cannot be properly quantified and are to be improved in future work. CFD models results show higher heat transfer than HT simulations and are closer to the experimental values. However, given the really close results of all the simulations, it is necessary to experiment with higher power to reduce the impact of the uncertainties. Experiments with the same power applied to different sized areas, thus changing heat flux values, would also be interesting. In this case, all simulations are really close to experimental values, hence with lower power and heat fluxes the extra computational effort spent on CFD simulations is probably not necessary. Nevertheless, if the hypothesis is confirmed for further experimentation, the implications, both technical and environmental, are really positive, as lower fluid flow would be necessary to dissipate the same amount of heat, resulting in less power and resources needed and higher life of the components.

ISBN: 978-3-95450-231-8 ISSN: 2673-5490 doi: 10.18429/JACoW-IPAC2023-THPM010

REFERENCES

[1] W. M. Rohsenow, J. P. Hartnett, Y. I. Cho, *Handbook of heat transfer*, McGraw-Hill Education, 1998.

[2] 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. 13th International Particle Accelerator Conf. (IPAC'22)*, Bangkok, Thailand, Jun. 2022, pp. 2887-2890. `doi:10.18429/JACoW-IPAC2022-THPOTK050`

[3] LEAP Australia, `https://www.computationalfluiddynamics.com.au/y-plus_part1_understanding-the-physics-of-boundary-layers/`

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

[5] 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*, Apr. 2006.

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

[7] H.W. Coleman, W.G. Steele, *Experimentation, validation, and uncertainty analysis for engineers.* 3rd ed. Wiley, 2009.

[8] F.M. White, *Fluid Mechanics*. 8th ed. New York: McGraw-Hill Education, 2016.

[9] N.M.C. Martins, N.H.G Carriço, H.M. Ramos, D.I.C. Covas. "Velocity-distribution in pressurized pipe flow using CFD: Accuracy and mesh analysis", *Computers & Fluids,* vol. 105, p. 218-230, 2014. `doi.org/10.1016/j.compfluid.2014.09.031`