CFD Modeling of Supercritical Water Heat Transfer in a Vertical Bare Tube Upward Flow

Dr. Vladimir Agranat

Applied Computational Fluid Dynamics Analysis, Thornhill, Ontario, Canada
E-mail: vlad@acfda.org Web: www.acfda.org

and

Dr. Igor Pioro

Faculty of Energy Systems and Nuclear Science
University of Ontario Institute of Technology, North Oshawa, Ontario, Canada
Email: igor.pioro@uoit.ca Web: www.nuclear.uoit.ca

Summary

Computational Fluid Dynamics (CFD) modeling of supercritical water (SCW) heat transfer has been used in nuclear engineering for predicting the heat transfer coefficients and better understanding of SCW heat transfer mechanism for more than 40 years. A recent review of these studies is given in [1] where some advances and shortcomings in this field are described.

In this summary, the first experience of applying the general-purpose CFD software, PHOENICS-2009, to the modeling of the two typical SCW heat transfer cases has been briefly described. A vertical upward flow in a bare tube with the inner diameter of 10 mm and the heated length of 4 m, which was studied experimentally in [2] at a pressure of 24 MPa, has been numerically simulated at the inlet temperature of 350 °C and the mass flux value of 1000 kg/(m^2s). The two modeling cases are considered: the tube wall heat flux is equal to 681 kW/m^2 in the first case and it is equal to 581 kW/m^2 in the second case.

The cylindrical coordinate system (X, Y, Z) is applied and it is assumed that there are no changes of fluid flow and heat transfer characteristics in the angular X-direction. As a result, a simplified 2D formulation is applied and the sizes of the computational domain in radial and axial directions are 5 mm and 5 m respectively. In the CFD model, the tube length is extended by 1 m upstream to create the fully developed turbulent profiles of velocity and temperature at the beginning of the heated tube length (at Z = 1 m). A computational grid of 40x90 cells was used in the majority of simulation runs. The grid was made significantly finer near the tube wall: the first layer of near-wall cells was located at a distance of 5.E-6 m from the wall. A few turbulence models were tested but only results obtained with the use of LVEL turbulence model [3] are described in this summary.

The physical properties of SCW at a pressure of 24 MPa (density, specific heat, thermal conductivity, kinematic viscosity and Prandtl number) were read during the CFD runs

from the separate files obtained for each individual property using the Excel table provided by UOIT (NIST data). At each computational cell, the above physical properties were calculated based on the local enthalpy value and using the linear interpolation.

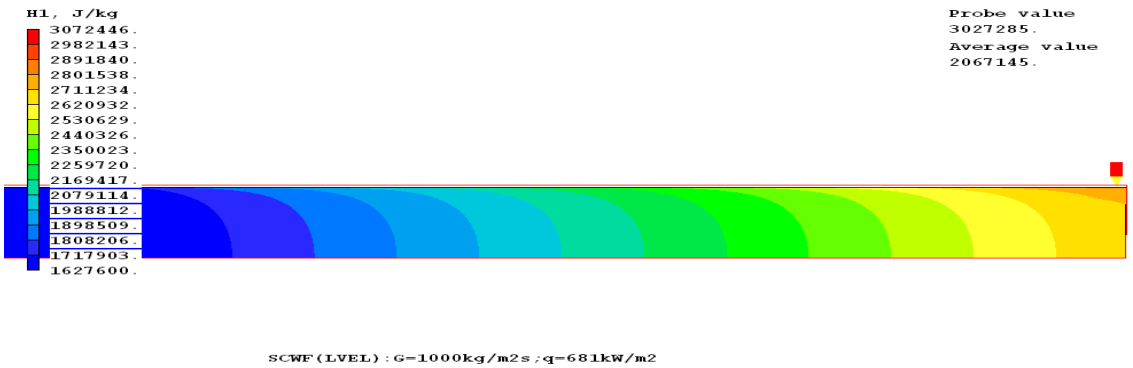
The 2D distributions of pressure, velocity components, enthalpy, temperature and physical properties have been predicted in the two modeling cases. Also, the heat transfer coefficient (HTC), the bulk enthalpy (HAVE), the wall temperature (TWAL) and the bulk temperature (TAVE) were calculated and compared with their experimental values.

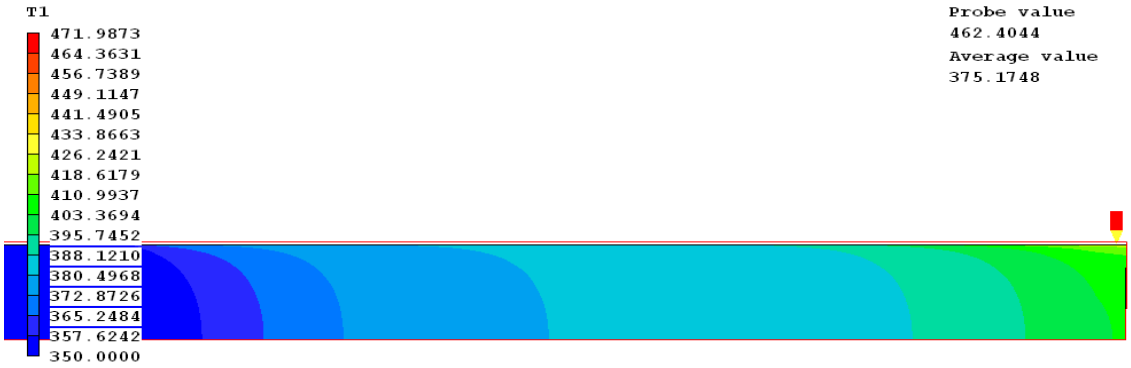
It has been found that the PHOENICS predictions of HTC, TWAL and TAVE with the use of LVEL turbulence model agree mostly well with experimental data in both modeling cases considered. However, there are greater discrepancies between the predicted and measured values of HTC and TWAL in the first half of the heated tube length. More research is needed in order to test the different turbulence models and various numerical grid settings. In particular, the proper selection and optimization of turbulence model, grid parameters and numerical relaxation factors are required.

It is proposed to create a customized (robust, accurate and user-friendly) SCW CFD tool using the PHOENICS software as a framework and adding some additional input and output interface for user convenience. This tool (after its validation) could be applied for predicting the SCW heat transfer in various simple and complex geometry cases.

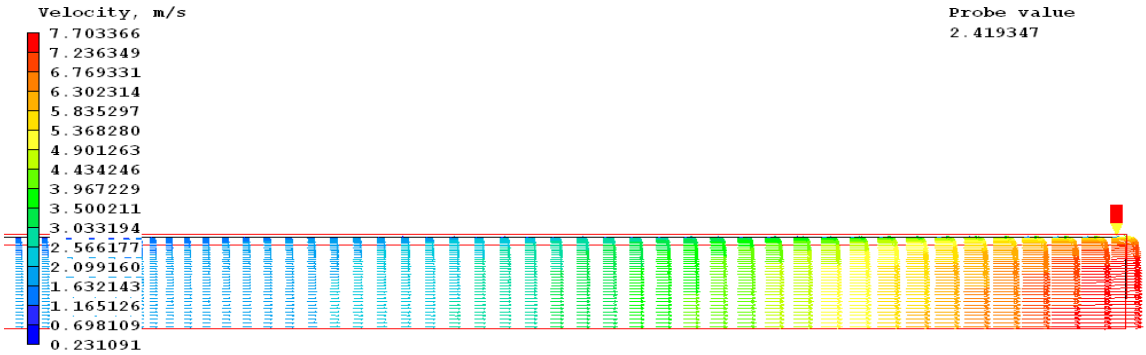
1. Simulation results in case 1 (T_{in} = 350 °C, G=1000 kg/(m^2s), q=681 kW/m^2)

Figure 1 shows the 2D contours of enthalpy (H1), temperature (T1), axial velocity (W1), density (RHO1), laminar kinematic viscosity (ENUL) and laminar Prandtl number (RPL1), which are predicted in the first modeling case. Due to a large decrease in density near the heated wall, there is a significant acceleration of flow along the heated tube length and a strong buoyancy effect is observed. The dependence of PRL1 on axial Z-coordinate is non-monotonic: there is a local maximum in the tube area where the local fluid temperatures are close to the pseudo-critical temperature of 381 °C.

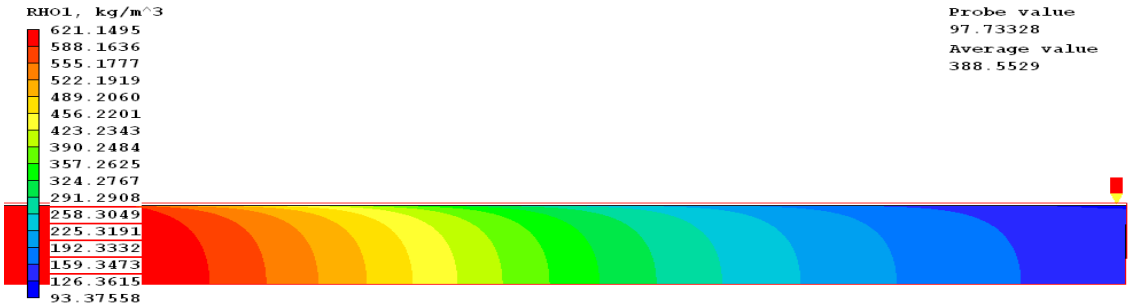




SCWF(LVEL):G=1000kg/m2s;q=681kW/m2



SCWF(LVEL):G=1000kg/m2s;q=681kW/m2



SCWF(LVEL):G=1000kg/m2s;q=681kW/m2

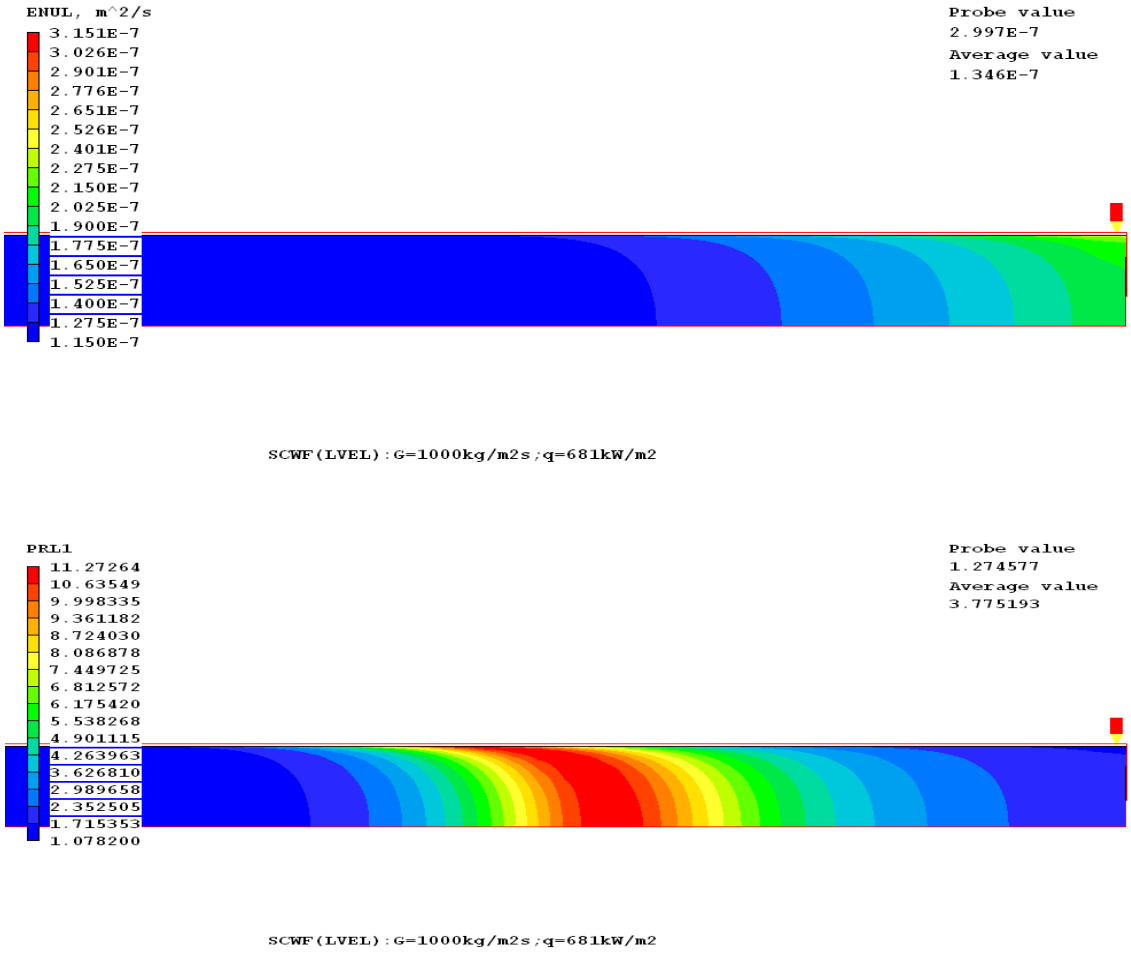


Figure 1. Contours of enthalpy, temperature, velocity, density, laminar kinematic viscosity and laminar Prandtl number predicted in case 1.

Dependences of heat transfer coefficient (HTC), bulk enthalpy (HAVE), wall temperature (TWAL) and bulk temperature (TAVE) on the axial distance, Z, have been predicted with PHOENICS and compared with the experimental data [2]. The predicted values agree mostly well with the experimental values shown in [2] in Figure 9. However, there are greater deviations in the first half of the heated length. Figure 2 shows the HTC predictions with the two different grid settings.

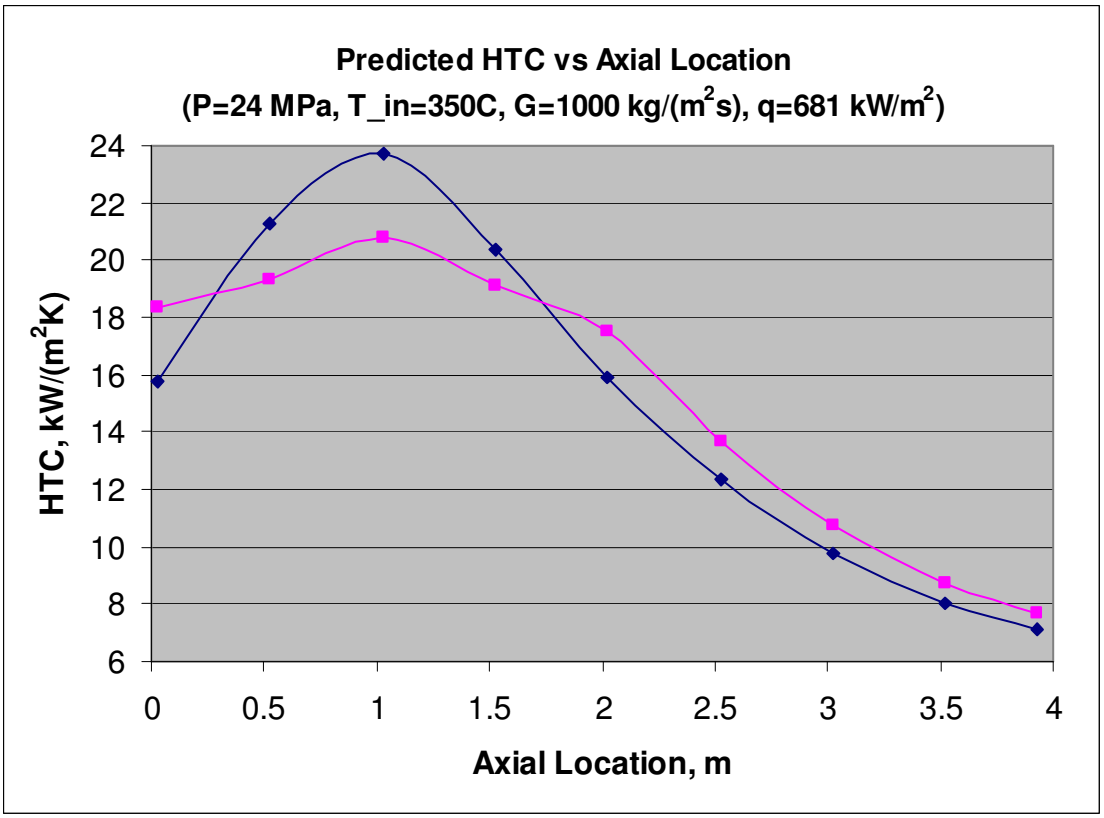


Figure 2. Predicted dependence of heat transfer coefficient (HTC) on axial location along the heated tube length (Z-1) in the first modeling case (two different grids are used for comparison).

2. Simulation results in case 2 (T_{in} = 350 °C, G=1000 kg/(m^{2}s), q=581 kW/m^2)

The 2D contours of enthalpy (H1), temperature (T1), axial velocity (W1), density (RHO1), laminar kinematic viscosity (ENUL) and laminar Prandtl number (RPL1) were predicted. As in the first modeling case, due to a large decrease in density near the heated wall, there is a significant acceleration of flow along the heated tube length and a strong buoyancy effect is observed. Also, the dependence of PRL1 on axial Z-coordinate is non-monotonic: there is a local maximum in the tube area where the local fluid temperatures are close to the pseudo-critical temperature of 381 °C.

Dependences of heat transfer coefficient (HTC), bulk enthalpy (HAVE), wall temperature (TWAL) and bulk temperature (TAVE) on the axial distance, Z, have been predicted and compared with the experimental data [2]. The predicted values agree mostly well with the experimental values shown in [2] in Figure 7. As in the first case, there are greater deviations in the first half of the heated length. Figure 3 shows the HTC predictions.

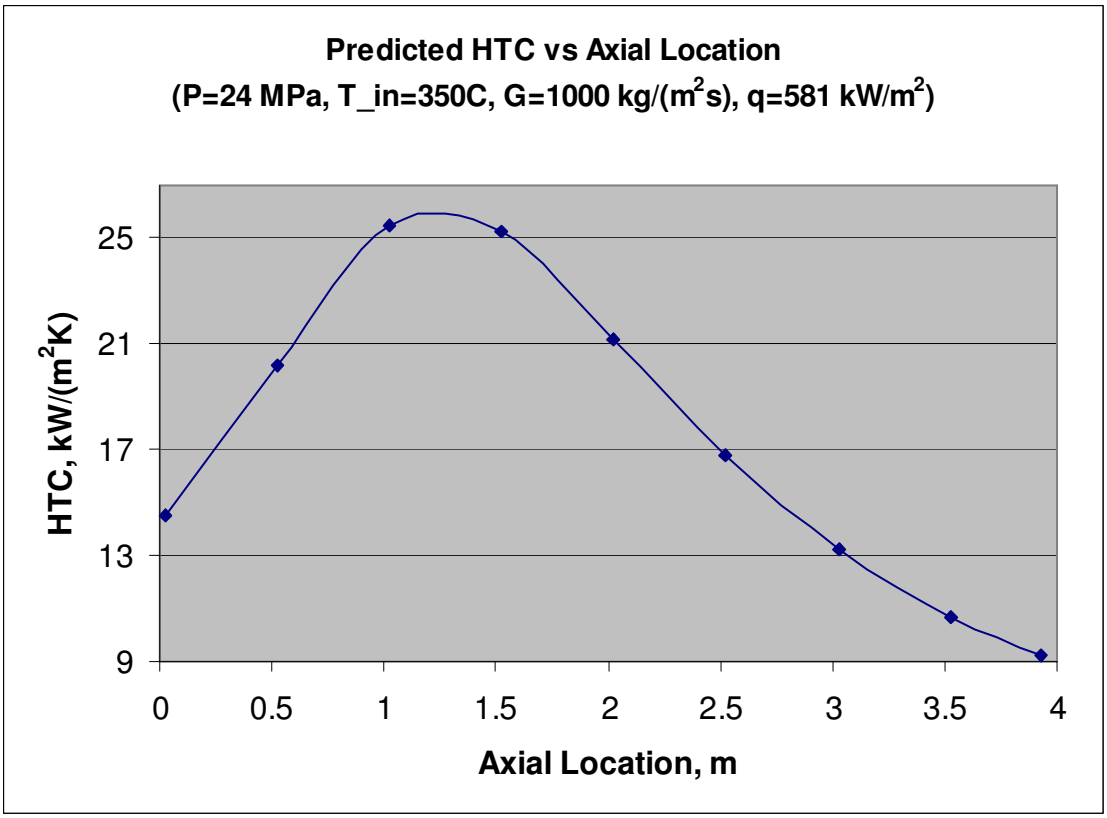


Figure 3. Predicted dependence of heat transfer coefficient (HTC) on axial location along the heated tube length (Z-1) in the second modeling case.

References

1. Cheng, X., Kuang, B., and Yang, Y.H., 2007. Numerical analysis of heat transfer in supercritical water cooled flow channels. Nuclear Engineering and Design, 237, 240-252.
2. Mokry, S., Farah, A., King, K., Gupta, S., Pioro, I., and Kirillov, P., 2009. Development of supercritical water heat-transfer correlation for vertical bare tubes, Proc. International Conference on Nuclear Energy for New Europe 2009, Bled, Slovenia, September 14-17, pp. 210.1-210.13.
3. http://www.cham.co.uk/phoenics/d_polis/d_lecs/general/turb.htm.

Acknowledgements

Authors would like to thank Dr. Michael Malin of CHAM Ltd. and Dr. Sergei Zhubrin of ACFDA for their technical support and helpful advices.