A Computational Mesh Sensitivity Study for Forced Convective Heat Transfer to Water at Supercritical Conditions in Tubes

D. McClure[1*] and A. Rashkovan[1]
[1] McMaster University, Hamilton, Ontario, Canada
*(mcclud@mcmaster.ca)

A Master's Level Submission

Summary

A computational mesh sensitivity study for turbulent flow and heat transfer of supercritical water in a straight heated pipe was performed using the CFD software STAR-CCM+ 6.06.017. Computed results were validated against experimental results of [1]. The study was done using the SST k-ω turbulence model due to its previous success in predicting wall temperatures in flows of this type [2, 3]. This study presents recommendations for the meshing of the near wall region, specifically, near wall node placement, axial refinement and radial growth ratio (RGR) are discussed. An appropriate mesh was chosen to have a y+ < 0.3, a RGR = 1.19 and an axial refinement of 7.2×10^{-4}m.

1. Introduction

The need for plentiful, affordable, clean energy has never been in greater demand. It is the goal of the Generation IV International Forum (GIF) to provide research and development "to support a generation of innovative nuclear energy systems" [4]. Under the umbrella of the GIF's research and development there exist multiple innovative reactor concepts. Canada has chosen to focus on the design of a Supercritical Water Reactor, where water at pressures above the critical pressure (P = 22.1 MPa) is used as a coolant.

A fluid at supercritical conditions does not change phase as a fluid at subcritical conditions does. However, a distinct change in physical properties occurs as the supercritical fluid crosses the "pseudocritical" temperature, T_{pc}, which is defined as the temperature at which the specific heat capacity of the fluid reaches a maximum. As the temperature is increased through T_{pc}, the density, thermal conductivity and dynamic viscosity of the fluid decrease drastically, while the specific heat capacity goes through a sharp maximum.

Due to the sharp variation of physical properties that are typical of supercritical fluids, the heat transfer mechanisms are quite complex. Three heat transfer regimes are identified, normal heat transfer, enhanced heat transfer where the heat transfer coefficient (HTC) has a larger value than that of normal heat transfer and deteriorated heat transfer (HTD) where the HTC has a smaller value than that of normal heat transfer [5]. Due to the complex nature of the heat transfer mechanisms, Dittus Boelter type correlations have been unsuccessful in predicting the HTC for supercritical flows [6].

To overcome the weakness of current correlation prediction methods, computational fluid dynamics (CFD) can be used with the ultimate goal of providing a piece of the thermal-hydraulics (TH) component for coupled simulations featuring neutronics and TH for the purpose of licensing a SCWR design. However, prior to this, CFD procedures must be rigorously verified and validated against experimental data on heat transfer to supercritical fluids. This includes studies focused on choosing an appropriate turbulence model as well as mesh requirements, specifically in areas where physical property gradients are large.

Visser et al [2] used FLUENT 6.2 to model upward flow of CO_2 with experimental data provided by [7]. The k-ε turbulence model was found to outperform the SST k-ω model at high mass fluxes while SST k-ω performed better at low mass fluxes. Further, SST k-ω was found capable of predicting flow structure changes associated with HTD. The enhanced wall treatment was used with y+ < 1.

Zhu [3] used CFX 12.0 to model all modes of heat transfer for low and high mass fluxes using experimental data from [1][8] and [9]. SST k-ω was found to be superior to RNG k-ε. A mesh of y+ < 0.1 was used.

The planned rod bundle nominal flow rate Reynolds number is expected to be on the order of 10^5 and higher. Hence, fine mesh is expected to be needed near the wall. To make the full rod bundle calculation time acceptable, it is important to define the optimal mesh requirement. The literature gives y+ recommendations but omits important parameters such as axial refinement, the effect of high aspect ratio and optimal RGR. It is the purpose of this paper to provide recommendations on y+ requirements, axial refinement and RGR.

2. Model Definition

The experiments of [1] are that of upward flow of water in a tube with constant heat flux applied at the wall. The experimental conditions are summarized in Table 1.

Pressure (p)	24.5 MPa
Mass Flux (G)	1260 kg/m^{2}s
Heat Flux (q)	465 kW/m^2
Diameter (d)	7.5 mm
Enthalpy Range (h)	1.4787 – 2.65 MJ/kg

Table 1 Experimental conditions to be modelled

The computations were done on a 2D axisymmetric mesh using a hydrodynamic fully developed attached inlet profile that has been calculated previously. The inlet temperature was held at 600K, corresponding to an inlet enthalpy of 1.4787 MJ/kg. The outlet plane was defined as a pressure outlet and an adiabatic section was attached downstream of the heated one in order to avoid the outlet boundary condition having impact on the flow. Physical properties were linearly interpolated between the data from the National Institute of Standards and Technology.

In order to create a mesh for the tube three variables must be defined, the distance from the wall that the first node is to be placed (y+), the ratio at which that distance increases as nodes are placed towards the bulk of the flow (RGR) and the degree of axial refinement. For a given y+ and RGR, the axial aspect ratio (axial refinement) will determine the final mesh size. The mesh parameters studied are shown in Table 2.

y+	< 0.15, 0.3, 1
RGR	1.1, 1.19, 1.28
Axial refinement (distance between axial nodes)	$4.5\text{x}10^{-5}$ m, $1.8\text{x}10^{-4}$ m, $7.2\text{x}10^{-4}$ m

Table 2 Mesh parameters studied

3. Results and Discussion

Axial refinement sensitivity was examined using a y+ < 1 and RGR = 1.28. It was found that results were independent of axial refinement, as seen in Figure 1 (left). Three of the simulations used constant axial refinement, while an additional simulation was done that was meshed very finely in the axial region where T_w was expected to pass through T_{pc}. Therefore, an axial refinement of $7.2\text{x}10^{-4}$m was chosen as optimal in order to reduce computational effort.

RGR sensitivity was examined using a y+ < 0.3 and axial refinement of $1.8\text{x}10^{-4}$m. It was found that results changed slightly from a RGR of 1.28 to 1.1, with a RGR of 1.19 giving results in between, as seen in Figure 1 (right). Therefore, a RGR of 1.19 was chosen as optimal to balance computational effort and accuracy.

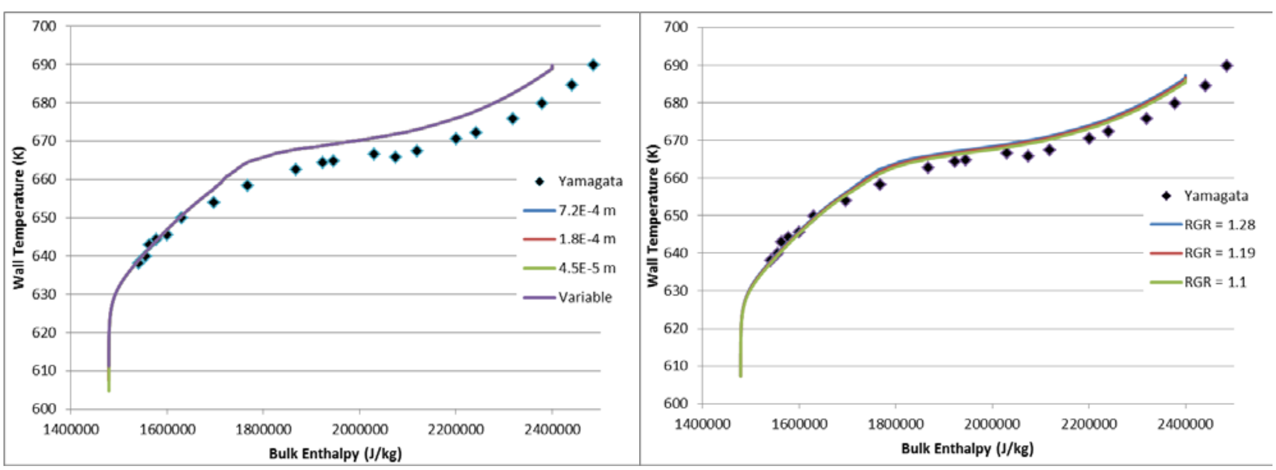

Figure 1 Axial refinement sensitivity performed with y+ < 1, RGR = 1.28 (left), RGR sensitivity performed with y+ < 0.3, $7.2\text{x}10^{-4}$ m (right)

y+ sensitivity was examined using a RGR = 1.19 and an axial refinement of $7.2\text{x}10^{-4}$m. It was found that the results were quite insensitive at y+ < 0.3, as seen in Figure 2 (left). Therefore, y+ < 0.3 was chosen as optimal to achieve close agreement to experimental data, slightly reduce computational effort and reduce the exceedingly high aspect ratios that become present as the near wall mesh becomes very fine.

The effect of high aspect ratio was examined in order to determine if the solution was affected as the near wall mesh became extremely fine in the radial direction while the axial refinement remained quite coarse. This was done using a y+ < 0.15 and RGR = 1.28. It was found that, although the aspect ratio increased to 3600, the results were independent of aspect ratio, as seen in Figure 2 (right). Therefore, the study can continue on with the knowledge that aspect ratios on the order of a few thousand are acceptable.

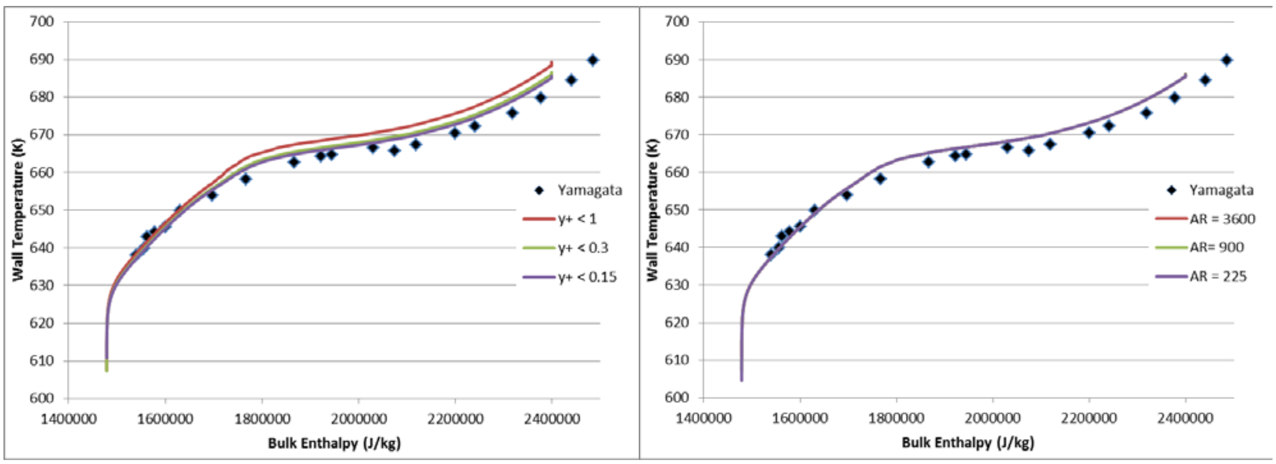

Figure 2 Y+ sensitivity performed with RGR = 1.19, 7.2x10^{-4}m (left), Effect of high aspect ratio performed with y+ < 0.15, RGR = 1.28 (right)

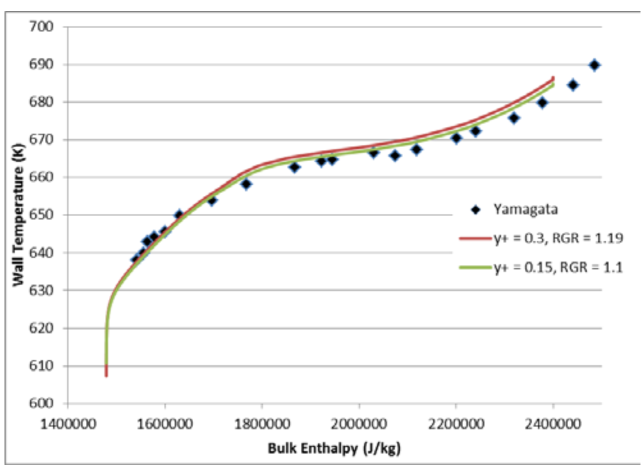

Figure 3 The optimal mesh results compared to the mesh that gave the best results

Figure 3 shows the results obtained on the optimal mesh compared to the mesh that gave the best results. The optimal mesh is that of y+ < 0.3, RGR=1.19 and an axial refinement of 7.2x10^{-4}m. The mesh that gave the best results is that of y+ < 0.15, RGR=1.1 and an axial refinement of 7.2x10^{-4}m. The savings by using the optimal mesh are approximately 50% in terms of computation time. What is gained in computation economics is lost only slightly in an over prediction of wall temperature (an under prediction of HTC) as the thermal boundary layer finishes development. This can be seen at approximately h=1 500 000 J/kg in Figure 3.

4. Conclusion

The present study provides a guideline in near wall mesh generation for forced convective heat transfer to water at supercritical conditions. The results were shown to be sensitive to y+ as well as RGR but were insensitive to axial refinement. In parameters that were sensitive, a compromise was made between solution accuracy and computational effort.

As seen in the literature the SST k-ω turbulence model performed well against the experimental data in [1]. The results of this study can now be used as a step towards establishing a verified and validated CFD procedure in the modelling of a fuel bundle.

5. Acknowledgements

Funding to the Canada Gen-IV National Program was provided by Natural Resources Canada through the Office of Energy Research and Development, Atomic Energy of Canada Limited, and Natural Sciences and Engineering Research Council of Canada. The authors would also like to acknowledgement the contribution of Dr. D.R. Novog.

6. References

[1] K. Yamagata, K. Nishikawa, S. Hasegaw, T. Fuji, S. Yoshida, “Forced convective heat transfer to supercritical water flowing in tubes”, International Journal of Heat and Mass Transfer, Vol. 15, 1972, pp. 2575–2593.

[2] D. Visser, J. Lycklama a Nijeholt, F. Roelofs, “CFD predictions of heat transfer in the super critical flow regime,” Proceedings of ICAPP '08, Anaheim, CA, USA, 2008.

[3] Y. Zhu, “Numerical Investigation of the Flow and Heat Transfer within the Core Cooling Channel of a Supercritical Water Reactor”, Dissertation, Universitat Stuttgart, 2010.

[4] DoE, U.S., “A technology roadmap for generation IV nuclear energy systems”, Nuclear Energy Research Advisory Committee and the Generation IV International Forum, 2002.

[5] I.L. Pioro, R.B. Duffey, “Experimental heat transfer in supercritical water flowing inside channels (survey)”, Nuclear Engineering Design, Vol. 235, 2005, pp. 2407–2430.

[6] J.Y. Yoo, “The Turbulent Flows of Supercritical Fluids with Heat Transfer”, Annual Review of Fluid Mechanics, Vol. 45, 2013, pp. 495-525.

[7] H. Kim, Y. Bae, H. Kim, J.H. Song, and B. Cho, “Experimental Investigation on the Heat Transfer Characteristics in a Vertical Upward Flow of Supercritical CO2”, Proceedings of ICAPP '06, Reno, NV, USA, 2006.

[8] A. Ornatskii, L. Glushchenko, and S. Kalachev, “Heat transfer with rising and falling flows of water in tubes of small diameter at supercritical pressures,” Thermal Engineering, vol. 18, 1971, pp. 137-141.

[9] M. Shitsman, “Impairment of the heat transfer at supercritical pressures,” High Temperatures, vol. 1, 1963, pp. 267-275.