CFD SIMULATION OF SUPERCRITICAL FLOW AND HEAT TRANSFER IN A THREE ROD WIRE WRAPPED BUNDLE

K. Podila, Y.F. Rao
Canadian Nuclear Laboratories,
Chalk River, Ontario, Canada, K0J 1J0
krishna.podila@cnl.ca; yanfei.rao@cnl.ca

ABSTRACT

This paper presents the results from an ongoing effort for assessing the CFD capability in simulating supercritical fluid flows in rod bundles. In the present study, a test from three-rod wire wrapped bundle experiment performed at University of Ottawa using CO_2 at 8.6 MPa was simulated using CFD. Five turbulence models were compared to assess their capability in predicting the sheath temperature variation along the length of the heated bundle. The mesh refinement was performed to minimize the effect of mesh sizes on the CFD predicted results. Using a correct mesh size and an appropriate turbulence model, the CFD model was then assessed against measurements. The CFD model used in this study correctly predicted the experimental trends of sheath temperature variation along the heated length of the bundle. However, the exact degree of temperature increase was under predicted by up to 15˚C.

KEYWORDS
CFD, Supercritical flows, heat transfer, Thermalhydraulics, Rod bundles

1. INTRODUCTION

Because of its versatility and widespread usage for simulating single-phase flow phenomena, CFD is being used by the Canadian Nuclear Laboratories for the development of Canadian SCWR fuel bundle concept. Since the flow in the SCWR concept is similar to that of the single-phase flow, i.e., it does not involve phasic interfaces, it is expected that the application of CFD methodology can provide useful information, especially in the boundary layer region, for the SCWR design. However, supercritical (SC) flow exhibits sharp variations in the fluid properties when its temperature crosses the threshold pseudo-critical temperature limit [1]. This presents a unique challenge for the existing turbulence models to capture the heat transfer deterioration, a key phenomenon that differentiates SC flows from the subcritical flows [2].

A “fit-to-purpose” turbulence model for SC flows does not exist. Also, the Canadian SCWR project within the GEN-IV framework lead by CNL is currently in its conceptual design phase, for which the relevant experimental data are currently not available. Hence, in order to gain confidence in the choice of turbulence model for simulating the new fuel bundle design, it is imperative that the assessment of the CFD predictions against experimental heat-transfer data for bare bundle subassemblies and bundle subassemblies with spacers be performed. In an attempt to test the turbulence models, experiments available from the open literature are simulated by Atomic Energy of Canada Limited, such as the ones from IPPE in Russia [3] on a vertically oriented “seven-rod bare bundle” cooled with supercritical Freon -12 [4].

Due to limited availability of experiments for SC flows in bare bundle assemblies and bundle assemblies with spacers, the majority of the previous analyses in literature have been devoted to smooth pipes.

However, the flow physics in the bundles differ significantly from those in smooth pipes [5], which necessitate the need to have access to experiments in bundle geometries especially with spacers to test the capabilities of the existing turbulence models to predict the flow and heat transfer characteristics in SC flows.

As a result of the ongoing co-operation of CNL with universities to better understand the heat transfer phenomenon at SC flow conditions, experiments were performed at University of Ottawa in a vertical up-flow loop comprising of three-rod wire wrapped fuel bundle cooled with supercritical CO_2 [6]. The scope of the current paper is set to simulate the three-rod wire wrapped fuel bundle assembly of UO using the commercial CFD code STAR-CCM+ version 9.02.007 and assess the ability of CFD to predict the experiments. The current work forms a part of overall broad objective of developing the CFD capability for simulating SC flows in fuel bundles and determining the suitability of the existing turbulence models in predicting heat transfer in SC flows including heat transfer deterioration for the Canadian SCWR design. In this work, STAR-CCM+ [7] CFD software is chosen because of its powerful meshing abilities and its flexibility in software licensing. STAR-CCM+ has been extensively used and tested by investigators [9, 10] and consortiums [11,12] in the past to simulate rod bundles, thereby confirming its suitability for the current application. Details of the test section, loop instrumentation and the data acquisition system can be found in [6].

2. COMPUTATIONAL MODEL DEVELOPMENT

2.1. CAD Model

The fuel bundle geometry consists of three elements with wire wrap on the surface of the fuel rod with a pitch of 200 mm. The surface for the sub-assembly geometries was created using ANSYS Design Modeller [8], with the model dimensions listed in Table I. The fuel element model developed in CAD was then exported into STAR-CCM+ for meshing the fluid domain. Figure 1 presents the fluid model of the wire-wrapped sub-assembly which is subsequently used for meshing. A design simplification similar to the ones presented in Podila and Rao [13] and by investigators at Idaho National Labs [14] was adopted in this investigation to avoid a mesh singularity introduced due to the contact of the wire on the fuel rod. The cylindrical wire wrap was approximated by the semi-cylindrical cross section with equivalent cross sectional area as shown in Figure 2 for improved solution convergence and ease of meshing. As a result of this simplification, a gap of ~0.18 mm was introduced, which might slightly change the flow pattern. However, this simplification is necessary to avoid meshing singularity.

Table I Geometric Model Dimensions used for Developing CAD (from Eter [6])

Parameter	Dimensions
Pressure tube diameter	25.4 mm
Fuel rod diameter	10 mm
Fuel rod pitch circle diameter	13.164 mm
Wire wrap nominal diameter	1.1 mm
Wire pitch	200 mm
Number of wire pitches simulated	7.5
Length of simulated domain	1500 mm

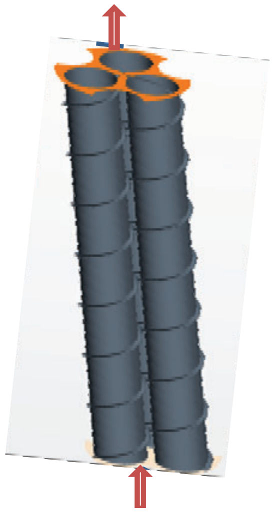

Figure 1 Computational Model Development in CAD (Pressure Tube Not Shown to Display the Wire Wrap)

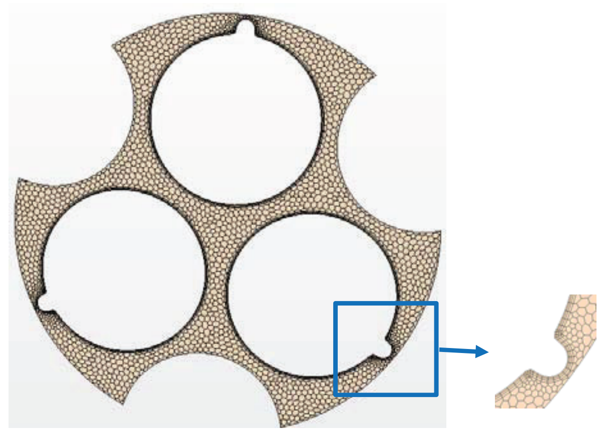

Figure 2 A Typical Computational Mesh Used for this Investigation

2.2. Mesh Generation

Even after the geometric simplification for the contact of the wire and the fuel rod (discussed earlier), meshing the wire wrapped bundle configuration poses to be a challenge. The tight-lattice wire-wrap geometry exhibits unique geometry disfeaturing especially at the point of contact of the wire and the fuel rod. In order to avoid this disfeaturing of the geometry and skewed faces, the wires were subjected to an individual mesh control that explicitly facilitated the specification of surface mesh size on the wires. Due to the nature of geometry that comprises of wire spacers helically wound along the fuel rod axis, polyhedral cells with prism layers were used to mesh the computational domain. Although hexahedral and tetrahedral cells could have been used to mesh the same configuration, they cannot be stretched as much as polyhedral cells. Further, polyhedral cells have more neighbour cells compared to hexahedral or tetrahedral cells thereby allowing gradients to be better estimated, which in turn leads to a more stable solution. Prism layers were applied only to heated sections (three wire wrapped fuel rods of the geometry). In order to capture the variation of physical properties in the boundary layer and possible HTD, five boundary layers were used on the wire wrapped fuel rods and the first node point was set at 43μm away from the wall (Figure 2).

2.3. Solution Approach and Models Used

In the current CFD simulation, the Reynolds Averaged Navier-Stokes equations, along with the conservation equations for energy, mass, and turbulence, were solved simultaneously. The thermo physical properties for CO_2 at 8.6 MPa were obtained from NIST online data base [15] and were implemented in `STAR-CCM+` solver. Similar to the previous investigations [4, 5, 13], properties of SCW were assumed to be only temperature dependent. The dependency on the pressure is usually small and hence is neglected in the current investigation. In the simulations, the entrance and exit of the flow channel were modeled with mass flow inlet and pressure outlet boundary conditions. The fuel rods and pressure tube were set as solid walls with no-slip conditions.

In the experiments conducted at UO, the test section comprising of three rod wire wrapped bundle (as 1500-mm Inconel 600 pipe of 0.036 ohm electrical resistance) was electrically heated by a rectified DC power supply having a maximum voltage of 60 V DC and a maximum current of 2833. For the CFD simulations, uniform heat flux boundary condition was imposed on the surface of the wire wrapped rods. The effect of conjugated heat transfer was not considered for this investigation.

The equations were solved using a steady-state segregated solver. Considering the recommendation made in ASME CFD numerical accuracy guidelines [16], all the equations were solved using the 2nd order differential schemes. The URF values for flow, pressure and energy were set to values of 0.7, 0.3 and 0.99 respectively. Convergence was monitored for each run and the solution was iterated till the residuals dropped at least by three orders of magnitude or less at completion and fluctuated in a steady manner. The suitable turbulence model for simulating the three-rod geometry was chosen based on sensitivity analysis of the available turbulence models to predict experiments (refer to Section 3). The testing of turbulence models was made in conjunction with an all y^+ wall treatment approach to account for the less resolved near-wall region adjacent to the pressure tube. The all y^+ wall treatment (a blended approach) uses a wall function that is automatically applied if the local y^+ value is insufficient to support a low y^+ representation. Table II lists the inlet fluid velocity, temperature, pressure, and heat fluxes on the fuel elements used for the current CFD simulation. The water equivalent pressure and temperature for the conditions simulated (Table II) in this study are 25.76 MPa and 173˚C respectively.

Table II Test conditions used for the current investigation

Operating fluid	CO_2
Mass flow	0.2163 kg/s
Inlet temperature	17.6 ˚C
Heat flux on fuel rods	124.6 kW/m^2
System pressure	8.6 MPa
Pseudocritical temperature of CO_2	38.07 ˚C

3. SENSITIVITY TO TURBULENCE MODELS AND EFFECT OF MESH SIZE

Five turbulence models: SST k-ω, Lien k-ε, Reynolds Stress Model, Realizable k-ε and AKN k-ε were tested in this study to assess their capability in predicting the measured wall temperatures over the heated length of the three-rod wire wrapped bundle. The details on each of the respective turbulence models are discussed in the STAR-CCM+ user manual [7]. For all the results presented in this paper, the circumferential positions defined in Figure 3 were used for comparing CFD predictions with experiments.

As seen in Figure 4, close to the inlet of the channel, SST k-ω model predicted the experiments reasonably well compared to the other four turbulence models, i.e. the standard turbulence models and their variants based on dissipation rate, and the RSM. Consequently, the SST k-ω model was used for carrying out further analysis i.e. sensitivity to computational mesh and assessment of the CFD predictions with measurements. It is interesting to note that all the tested models except the SST k-ω model resulted in over-prediction of wall temperatures close to the inlet (at 0.2m from inlet). Additionally all the turbulence models except the SST k-ω model resulted in similar values for the temperature predictions at 0.2 m (see Figure 4). The difference was further reduced amongst the predictions along the length of the bundle especially after 0.7 m from the inlet. Based on the experimental data, the deteriorated heat transfer (DHT) is expected to occur between 0.9~1.2 m, while all the simulations including SST k-ω model [17] failed to predict the peak there. The small sharp ramp ups in the predicted temperatures correspond to the wired-wrap spot which narrows the gap between two bundle rods.

The mesh sensitivity analysis was performed to check if the number of meshes used for the study is adequate and errors were not introduced in predicting the experiments as a result of using incorrect mesh count. It should be noted that the near wall meshing and wall y^+ values were not changed. The cell count for the refined mesh was approximately tripled to that of the base case mesh (refer to Figure 4) to check for the sensitivity of the mesh count on the CFD predictions. As seen in Figure 4 the prediction of the temperature along the length by the refined mesh was similar to that by the base case mesh. Hence further assessment against measurements at three individual rods and four different circumferential positions was performed using the base case mesh with approximately 12 million cells.

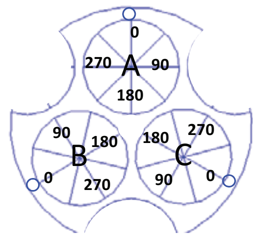

Figure 3 Circumferential Positions (Θ) on the Rods used for the assessment of the CFD Predictions

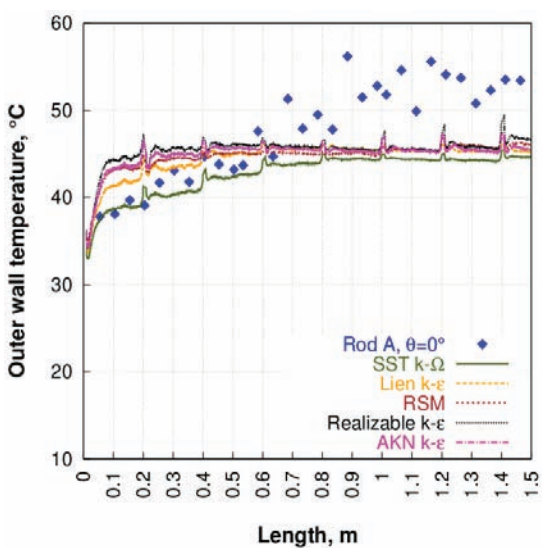

Figure 4 Variation of Temperature with Turbulence Models for Rod A at 0 Degrees Along the Heated Length of the Three-Rod Wire Wrapped Bundle Test Section

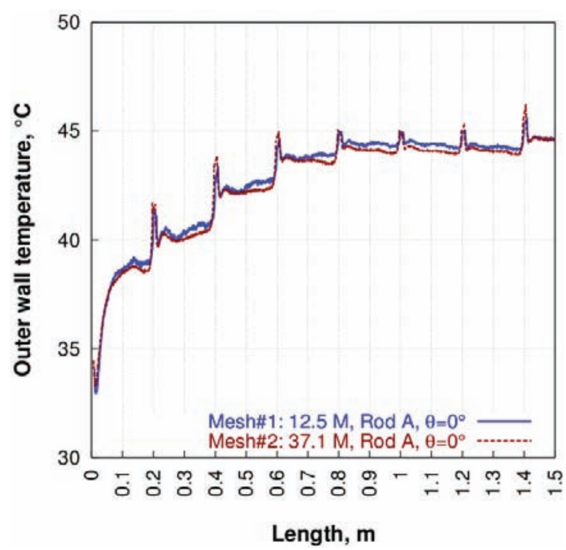

Figure 5 Variation of Temperature with Mesh Size for Rod A at 0 Degrees along the Heated Length of the Three-Rod Wire Wrapped Bundle Test Section

Table III Test conditions used for the current investigation

Base case mesh (mesh#1)	12,522,038 polyhedral cells
Refined mesh (mesh#2)	37,147,461 polyhedral cells

4. ASSESSMENT OF THE CFD PREDICTIONS

In this section, a detailed assessment of the CFD predictions for the three-rods at four different circumferential angles are presented using the correct mesh count and an appropriate turbulence model that was earlier identified in section 3. The overall temperature variation for the three-rod wire wrapped bundle can be seen in Figure 6. A maximum of 52˚C was predicted for the simulated test condition listed in Table II. It should be noted that, the occurrence of maximum temperature (peak temperature) can be primarily observed at the point of contact between the wire and the fuel rod, as a result of neglecting the sheath conduction model. The introduction of wire wraps leads to higher turbulence intensity; as a result the temperatures on the surface of fuel rod become more homogeneous circumferentially. CFD under predicted the experimentally reported measurements along the heated length for the three fuel rods at four different circumferential rod positions of 0, 90, 180 and 270 degrees respectively (Figure 7 through Figure 9). Although, CFD was able to capture the general trends for the variation of temperature along the heated length, it failed to capture the exact quantitative value of the temperature increase along the length of the bundle especially after 0.7m. It should be noted that, the experimentally measured temperatures have peaks and valleys which were not in correlation with the wire pitch on the fuel rod, there by leading to an unexplainable scatter of the experimental data along the length of the bundle. However, consistently for the three-rods at four circumferential positions, it can be observed that the temperature gradually increases along the length. In addition, based on the measurements, occurrence of the heat transfer deterioration may exist close to the exit of the bundle. However, the predictions from CFD exhibited a gradual rise in temperature up to 0.7 m, and further increase of temperature beyond 0.7m was not captured in this study. For Rod C, the temperature variation until 0.5 m of length from the inlet was not reported by the experiments. Hence in Figure 9 the CFD predictions were plotted starting from 0.5m till the end of the heated length of bundle, 1.5m.

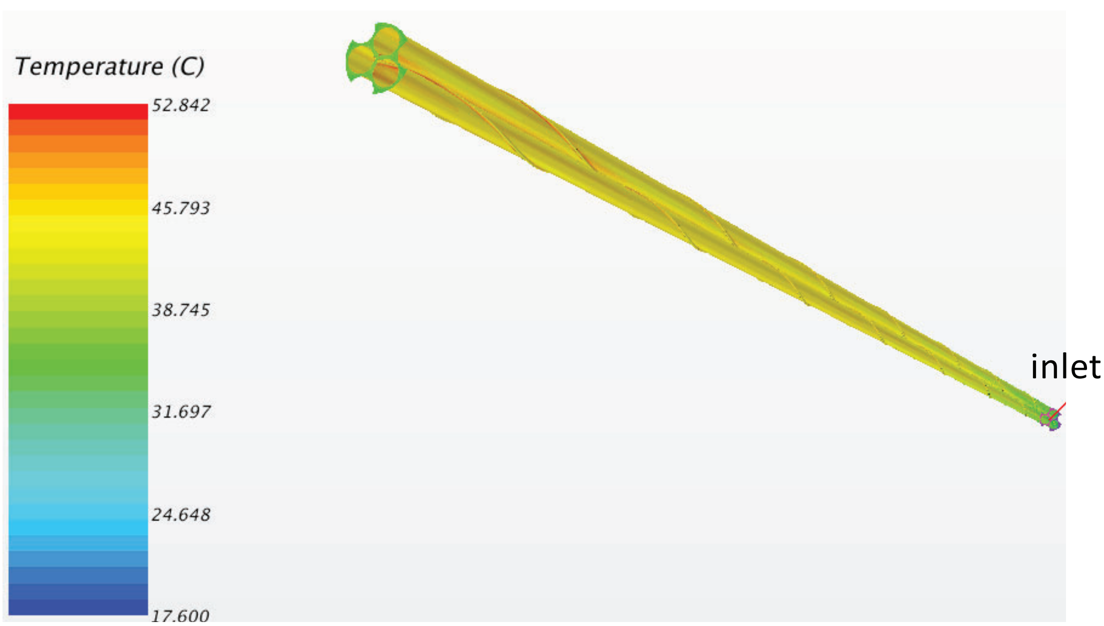

Figure 6 Variation of Temperature on the Three-Rod Wire Wrapped Bundle

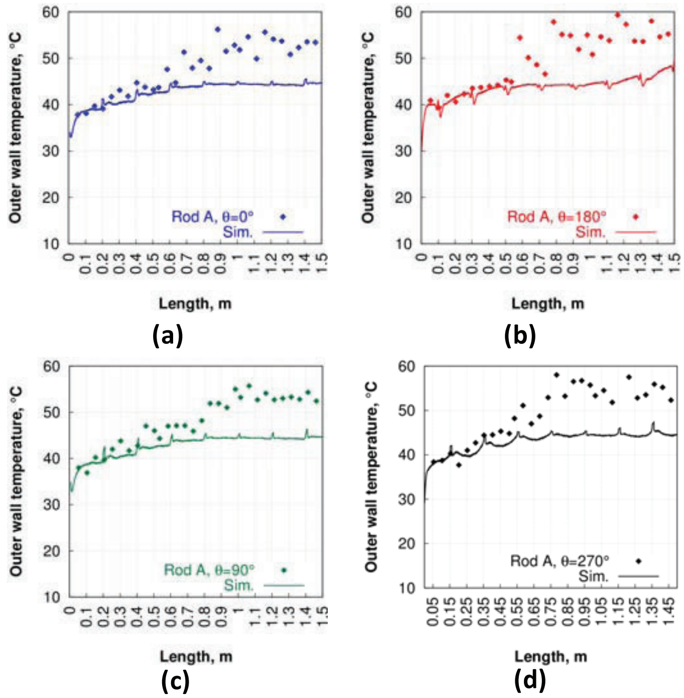

Figure 7 Variation of temperature for Rod A at; (a) θ=0˚, (b) θ=90˚, (c) θ=180˚, (d) θ=270˚ along the heated length of the three-rod wire wrapped bundle test section

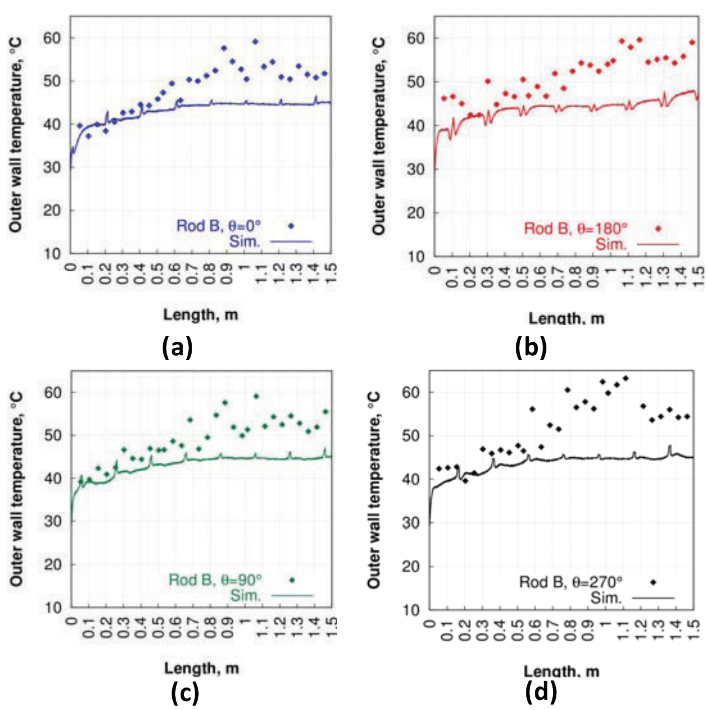

Figure 8 Variation of temperature for Rod B at; (a) θ=0˚, (b) θ=90˚, (c) θ=180˚, (d) θ=270˚ along the heated length of the three-rod wire wrapped bundle test section

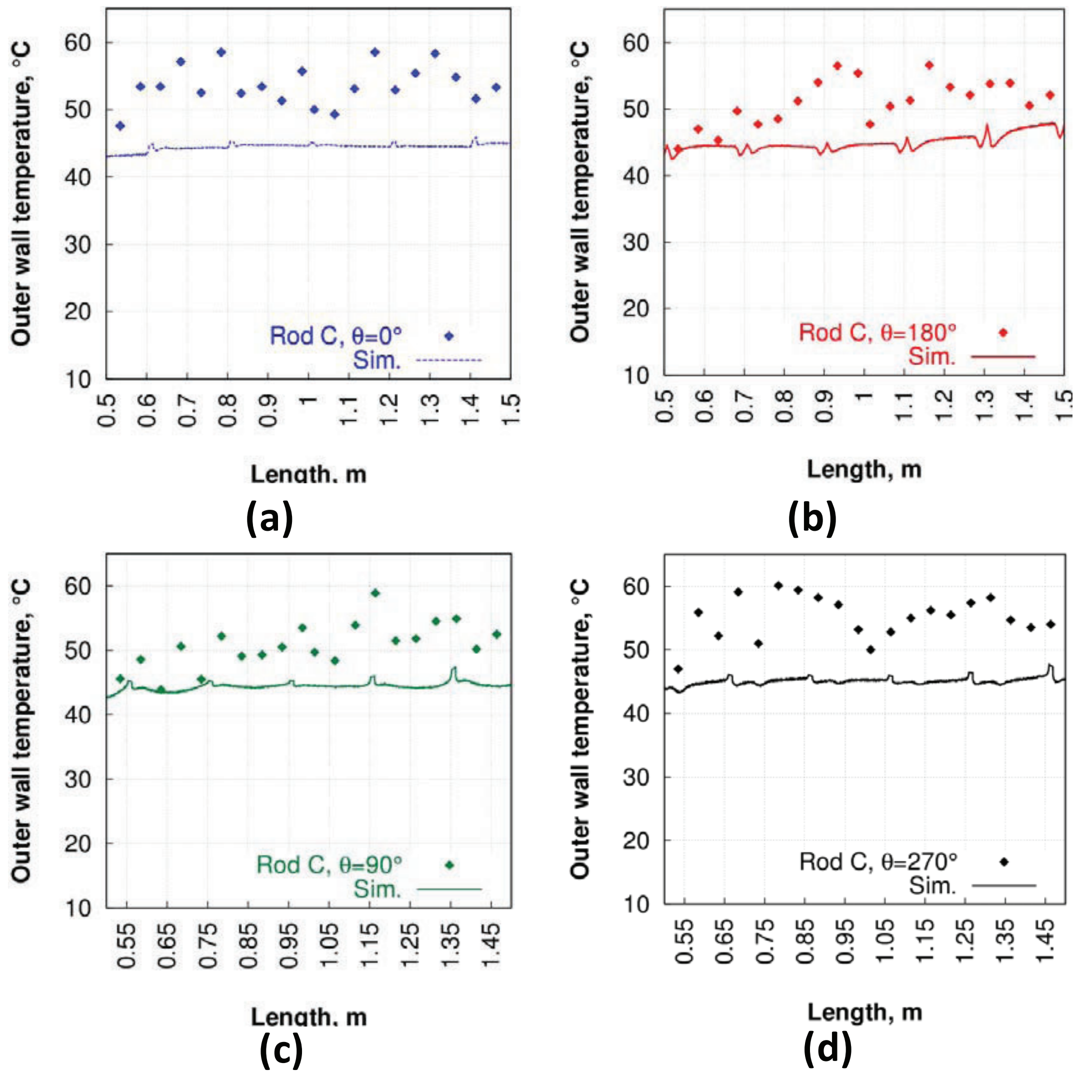

Figure 9 Variation of temperature for Rod C at; (a) θ=0˚, (b) θ=90˚, (c) θ=180˚, (d) θ=270˚ along the heated length of the three-rod wire wrapped bundle test section

5. CONCLUSIONS

Based on the CFD analysis of the UO three-rod wire wrapped bundle experiments, it can be concluded that:

1. For the test condition simulated in this study, significant differences were not predicted amongst the five turbulence models except for the SST k-ω model which resulted in close agreements with the experiments till 0.7m along the heated length of the bundle.
2. The CFD simulations predicted the trend in temperature rise along the heated length but failed to capture the exact temperature rise reported in the experiments.
3. The validity of the computational model developed for a single experimental test condition should be further tested for other available experimental runs made by UO for the three-rod wire wrapped bundle.

NOMENCLATURE

ACRONYMS

AKN	Abe, Kondoh, Nagano turbulence model
CO_2	Carbon dioxide
ASME	American Society of Mechanical Engineers
CAD	Computer Aided Design
CNL	Canadian Nuclear Laboratories
CFD	Computational Fluid Dynamics
GEN IV	Generation IV Program
HTD	Heat Transfer Deterioration
DHT	Deteriorated Heat Transfer
IPPE	Institute of Physics and Power Engineering
NIST	National Institute of Standards and Technology
SC	Supercritical Flows
SCWR	Supercritical Water-cooled Reactor
SST	Shear Stress Tensor
UO	University of Ottawa

GREEK SYMBOLS

Θ	Circumferential angle

ACKNOWLEDGMENTS

Financial support from the CNL Generation IV Energy Technologies Program is gratefully acknowledged.

REFERENCES

1. I.L. Pioro, R.B. Duffey, *Heat Transfer and Hydraulic Resistance at Supercritical Pressures in Power Engineering Applications*, ASME Press, New York (2007).
2. M. Angelucci, W. Ambrosini, N. Forgione, "Numerical Estimation of Wall Friction Ratio Near the Pseudo-Critical Point With CFD Models", *Nuclear Engineering and Design* **264**, pp. 71-79 (2013).
3. P.L. Kirillov, A.N.Opanasenko, R. Pomet'ko, A.S. Shelegov, "Experimental Study of Heat Transfer on Rod Bundle at Supercritical Parameters of Freon-12", Federal Agency for Atomic Energy State Scientific Center of RF-Institute for Physics and Power Engineering Named after A.I. Leypunsky IPPE, Obninsk, Russia (2006).
4. X. Huang, K. Podila, Y.F. Rao, "CFD Simulation of Vertical 7-Element Bundle Cooled With Supercritical Freon-12", *AECL Nuclear Review*, pp. 17-28 (2014).
5. D. Palko and H Anglart, "Theoretical and Numerical Study of Heat Transfer Deterioration in High Performance Light Water Reactor", *Science and Technology of Nuclear Installations* **2008**, pp 1-5 (2008).
6. A. Eter, "Heat Transfer Experiments in a Three-Rod Bundle Cooled by Supercritical and High Pressure Subcritical CO_2 Phase I Results of Initial Series of Tests", Technical Report UO-MCG-SC-2014, pp. 1-50 (2014).
7. CD-Adapco, STAR-CCM+ Software package, Users Manual, Version 9.02.007, New York USA, (2015).

8. ANSYS® CFD, Design Modeller Help System, Release 15.0 ANSYS Inc., Canonsburg USA, (2015).
9. E.A. Amoako, W. Ambrosini, "Developing a CFD Methodology for the Analysis of Flow Stability in Heated Channels with Fluids at Supercritical Pressures", *Annals of Nuclear Energy* **54**, pp. 251-262 (2013).
10. R. Wigeland, K.D. Hamman, "Fast Reactor Subassembly Design Modifications for Increasing Electricity Generation Efficiency", *Proceedings of Global 2009*, Paris, France, September 6-12, 2009 (2009).
11. "CASL", http://www.casl.gov/index.shtml, Last Accessed on March 2015.
12. "THINS", http://www.ifrt.kit.edu/thins/, Last Accessed on September 2013.
13. K. Podila, Y.F. Rao, "Assessment of CFD for the Simulation of Canadian SCWR Bundle with Wire Wraps", *Progress in Nuclear Energy* **77**, pp. 373-380 (2014).
14. K.D. Hamman, R.A. Berry, "A CFD M&S Process for Fast Reactor Fuel Assemblies", *Proceedings of the Experiments and CFD Code Applications to Nuclear Reactor Safety (XCFD4NRS 2008)*, Grenoble, France, September 10-12, 2008 (2008).
15. E.W. Lemmon, M.O. McLinden, D.G. Friend, "Thermophysical Properties of Fluid Systems", In: P.J. Linstrom, and G.W. Mallard, (Eds.), NIST Chemistry WebBook, NIST Standard Reference Database Number 69, National Institute of Standards and Technology, Gaithensburg, Maryland, USA retrieved September 2014.
16. ASME V&V 20 Committee, Standard for Verification and Validation in Computational Fluid Dynamics and Heat Transfer, ASME, New York, (2009).
17. F.R. Menter, "Two-Equation Eddy-Viscosity Turbulence Modeling for Engineering Applications", AIAA Journal **32**, pp. 1598-1605 (1994).