Numerical Simulation of Heat Transfer in Vertical Fuel Bundles of 37-element Canadian SCWR

Huirui Han [1] , Chao Zhang [1] and Jin Jiang [2]

[1] *Department of Mechanical and Materials Engineering, The University of Western Ontario*
1151 Richmond Street, London, ON, Canada, N6A5B9

[2] *Department of Electrical and Computer Engineering, The University of Western Ontario*
1151 Richmond Street, London, ON, Canada, N6A5B9

Email: *hhan55@uwo.ca*

ABSTRACT

Supercritical water cooled reactor was proposed as one of the Generation IV nuclear systems. Although many researches for heat transfer of supercritical water in circle channels have been conducted, there is still lack of prediction of heat transfer in transient process of tube bundles used in supercritical nuclear reactors. Besides, since there are multiple fuel rods, supercritical water is external flow not internal flow as in circle channels. This will cause the difference of fluid flow and heat transfer on cladding surface. In this work, a 37-element Canadian SCWR vertically arranged fuel bundle is simulated numerically. The thermo-hydraulic behaviors of the supercritical water in the SCWR fuel bundle are predicted. For nuclear power plant safety, the cladding surface temperature is a very important parameter. The results in this work show that the maximum cladding temperature is lower than the limit when the load is 100%. In the 3D transient process simulation, a step increase is introduced and test how long the system will return to steady state. This can provide references for future control system design.

NOMENCLATURE

Abbreviations

3D	three dimensional
CFD	Computational Fluid Dynamics
RSM	Reynolds Stress Model
SCWR	Supercritical Water-Cooled Reactor

General symbols

C_{ij}	convection term
C_p	specific heat capacity, J/kg·K
$D_{T,ij}$	turbulent diffusion
$D_{L,ij}$	molecular diffusion
G_{ij}	buoyancy production
p	pressure, Pa
P_{ij}	stress production
Pr_t	turbulent Prandtl number
S	heat sources
T	temperature, K
u	velocity, m/s
ρ	density, kg/m^3
μ	dynamic viscosity, $Pa \cdot s$
λ	thermal conductivity, $W/m \cdot K$
μ_t	turbulent viscosity, $Pa \cdot s$
ϕ_{ij}	pressure strain
ε_{ij}	dissipation

1. INTRODUCTION

In the Generation IV nuclear reactor concepts proposed in 2002, the Supercritical Water-Cooled Reactor (SCWR) is the only one that uses light water as coolant. CANDU series are heavy pressurized reactors. Compared with nowadays reactors, SCWR system configuration size is smaller. Also, since water is heated by reactor core to supercritical steam, then directly does work through turbine, thermal efficiency can be increased to 40% [2]. Typical Canadian SCWR is shown in Figure 1. Moderator for

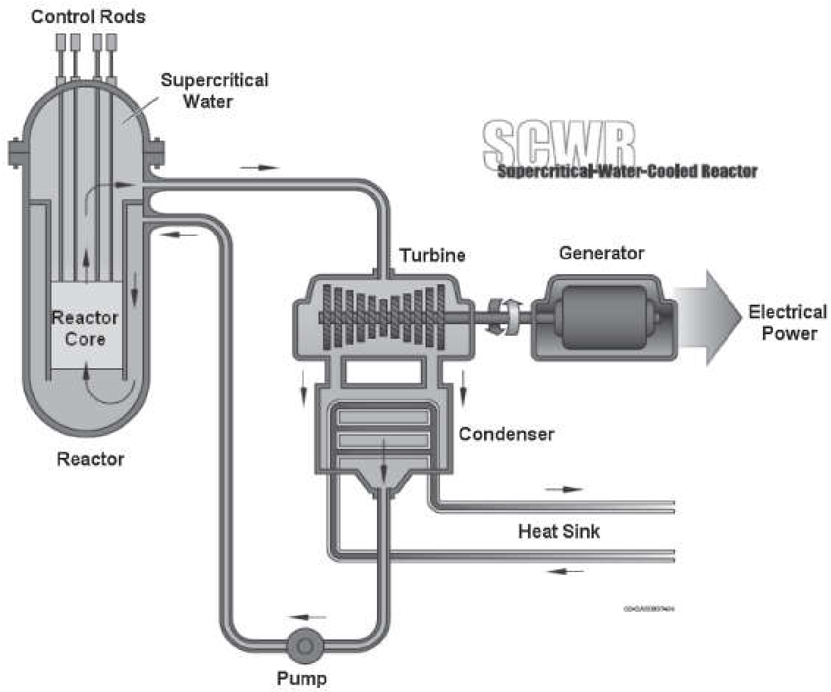

Figure1 [1] Configuration of Supercritical Water-cooled Reactor System

Table 1 Specifications of CANDU-SCWR

Spectrum	Thermal
Moderator	Heavy water
Coolant	Light water
Thermal Power	2540MW
Flow rate	1320kg/s
Number of channels	336
Electric power	1200MW
Efficiency	48%
Fuel	UO2/TH
Enrichment	4%
Inlet temperature	350℃
Outlet temperature	625℃
Cladding temperature	〈850℃
Heated length	5m
Operating pressure	25MPa
Calandria diameter	4m

the reactor is heavy water and coolant is light water. The specifications for Canadian SCWR [3] is shown in Table 1. Because of the abrupt properties change of supercritical water around pseudocritical point, heat transfer deterioration is easy to happen. Since Canadian SCWR is still in conceptual design, there are no experimental data now for flow and heat transfer in fuel bundles. Numerical method is highly recommended to predict fluid flow and heat transfer of supercritical water in fuel bundles. This paper will focus on predicting thermo-hydraulic behavior of supercritical water in fuel bundles in transient process. Because of the specialty of supercritical water, heat transfer deterioration may occur on cladding surface. This will affect safety of whole reactor system. Azih et al. [4] studied the convective heat transfer of supercritical fluids without gravity, and obtained the effects of Reynolds number, boundary layer and other factors on heat transfer. Zeng et al. [5] also conducted computational fluid dynamics simulation in circle channel. The consideration of gravity and thermal expansion acceleration effect in turbulence model under subcritical simulation is put forward. This can be helpful for investigation of Canadian SCWR startup process. Zhang et al. [6] proposed a new supercritical fluid flow model to better deal with the physical instability properties of the supercritical fluid around the pseudo-critical point. Rohit [7] also simulated heat transfer of supercritical water in circle channels using different turbulence models. And the results are compared with experimental data to check fitting degree. However, these previous researchers mainly investigated heat transfer and fluid flow in single channel. This may ignore effects of interaction between fuel rods. And in fact, the supercritical water in fuel bundles is external flow around fuel rods, not internal flow in channels. Yina [8] simulated heat transfer and flow of supercritical water in a 37-element horizontal arranged SCWR in steady state. The nuclear power plant work procedures are complicated, it is hard to include all points in numerical simulation. However, there is still lack of 3D numerical simulation for supercritical water in vertical fuel bundles. In this work, we will investigate heat transfer of supercritical water in vertical fuel bundles with multi fuel rods in transient process.

2. GOVERNING EQUATIONS AND NUMERICAL MODELS

In this study, a 37-element fuel bundle [9] is chosen for numerical transient simulation. Fuel rods are averagely distributed in the fuel bundle. The fuel bundle is vertically arranged and works under 100% load. Since the symmetry of fuel bundles, only a quarter of the fuel bundle is included in this simulation. And the heated length is reduced to 1m to save simulation time. The properties of supercritical water, such as density, specific heat, thermal conductivity, viscosity, are from Wagner [10]. The pseudocritcal point region is about 384.9℃ for 25MPa[11]. The governing equations for this three-dimensional flow and heat transfer are conservations of continuity, momentum, energy, which are shown in Reynolds averaged form [12]:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho \overline{u_i})}{\partial x_i}=0 \tag{1}$$

$$\frac{\partial(\rho \overline{u_i})}{\partial t}+\frac{\partial(\rho \overline{u_i}\,\overline{u_j})}{\partial x_j}=-\frac{\partial \overline{p}}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu\frac{\partial \overline{u_i}}{\partial x_j}-\rho\overline{u_i' u_j'}\right) \tag{2}$$

$$\frac{\partial(\rho C_p T)}{\partial t}+\frac{\partial}{\partial x_i}(\overline{u_i}\rho C_p T)=\frac{\partial}{\partial x_i}\left[\left(\lambda+\frac{C_p\mu_t}{Pr_t}\right)\frac{\partial T}{\partial x_i}\right]+S \tag{3}$$

Zhang found the numerical results calculated by Reynolds Stress Model agree well with experimental data for vertical tubes[13]. Thus, the Reynolds stress model (RSM) with enhancement wall function is used for solving the Reynolds stress term in this simulation. Thermal and full buoyancy effects, viscous heating are all included. The transport equation of RSM is as follows[12]:

$$\underbrace{\frac{\partial}{\partial t}(\rho\overline{u_i' u_j'})}_{\text{Local Time Derivative}}+\underbrace{\frac{\partial}{\partial x_k}(\rho u_k\overline{u_i' u_j'})}_{C_{ij}}=$$
$$\underbrace{-\frac{\partial}{\partial x_k}\left[\rho\overline{u_i' u_j' u_k'}+\overline{p'(\delta_{kj}u_i'+\delta_{ik}u_j')}\right]}_{D_{T,ij}}+\underbrace{\frac{\partial}{\partial x_k}\left[\mu\frac{\partial}{\partial x_k}(\overline{u_i' u_j'})\right]}_{D_{L,ij}}$$
$$\underbrace{-\rho\left(\overline{u_i' u_k'}\frac{\partial u_j}{\partial x_k}+\overline{u_j' u_k'}\frac{\partial u_i}{\partial x_k}\right)}_{P_{ij}}\underbrace{-\rho\beta(g_i\overline{u_j'\theta}+g_j\overline{u_i'\theta})}_{G_{ij}}$$
$$\underbrace{+\overline{p'\left(\frac{\partial u_i'}{\partial x_j}+\frac{\partial u_j'}{\partial x_i}\right)}}_{\phi_{ij}}\underbrace{-2\mu\overline{\frac{\partial u_i'}{\partial x_k}\frac{\partial u_j'}{\partial x_k}}}_{\varepsilon_{ij}}+\underbrace{S_{user}}_{\text{User defined source term}} \tag{4}$$

The simulation parameters for the fuel bundle is shown in Table 2 [8]. The computational domain is in Figure 2 and the mesh is in Figure3. ANSYS Fluent 15.0 is used to solve equations. The SIMPLEC scheme is selected for pressure correction, and QUICK method is considered for conducting spatial discretization. The convergence criteria for continuum, momentum, and turbulence parameters are all 10^{-6}, but for energy equation is 10^{-9}. The heat flux of fuel rod cladding surface is $10^{6} W / m^{2}$ averagely. The reference pressure is 25MPa. 334290 nodes give grid independence for this simulation. Time step size is 0.01s and the max iteration for each time step is 20. Boundary conditions are as follows:

Inlet: Here, velocity-inlet is chosen for the inlet type, inlet velocity is 1.18m/s and inlet temperature is 623.15K. Turbulence intensity is 6%, and hydraulic diameter is 7.416mm based on Table 2;

Outlet: Outflow is selected;

Walls: For the walls, here we consider they are smooth walls with no-slip conditions.

Table 2 Parameters for the fuel bundle simulation

Fuel Rod Diameter/mm	13.081
Fuel Bundle Diameter/mm	103.378
Length/mm	1000
First Ring Diameter/mm	29.769
Second Ring Diameter/mm	57.506
Third Ring Diameter/mm	86.614

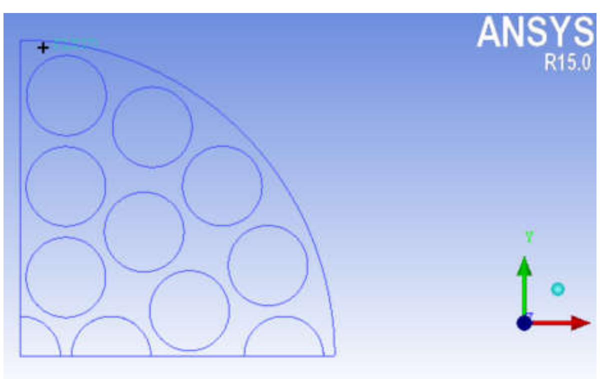

Figure 2 Computational domain

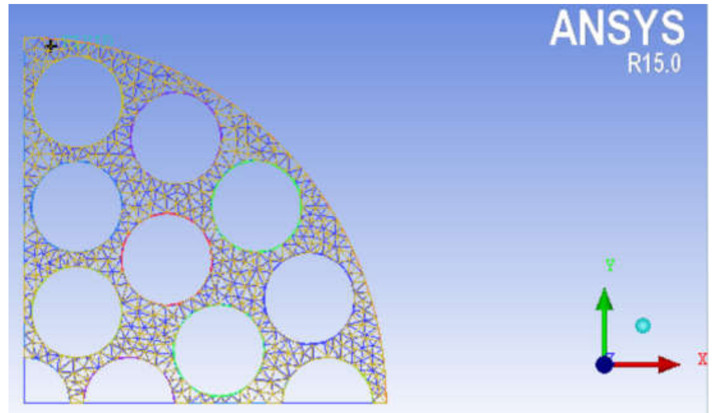

Figure 3 Mesh

3. RESULTS AND DISCUSSIONS

Figure 4 shows the cladding surface temperature at t=3s. It is shown that the maximum value is below the limit when the load is 100%. Figure5 (a) and 5(b) show the inlet and outlet mass flow rate, outlet temperature of the fuel bundle variation with time when inlet mass flow rate decrease 10% at 3s. It can be seen that the flow is very unstable at first when supercritical water is heated around the pseudocritical point. Inlet and outlet mass flow rate tends to be uniform when t=2s. At the same time, the outlet temperature also increases to a peak value and then gradually remain stable before the step decrease inserted. A 10% step decrease of inlet mass flow rate is introduced when t=3s. Then the outlet mass flow rate also decrease and reach the inlet value quickly. The outlet temperature just increase about 5℃ and then becomes stable again. In this 10s transient process, the system can return to a relatively steady state quickly after perturbances are introduced. Figure 5(c) shows the cross section view of the cladding surface temperature distribution of fuel rods at t=10s. For each fuel rod, the cladding surface temperature distribution is not uniform. Higher temperatures always appear on the location area between each two fuel rods. The maximum cladding temperature shown in Figure 5(c) can be up to about 1250K, which is higher than the design value in Table 1. This may cause safety accident in nuclear power plant, especially when the nuclear power unit is in shut-down process.

For another condition, the wall heat flux decrease 10% at t=3s, these three factors variations are shown in Figure 6(a) and 6(b). Trends are the same before 3s. However, when the 10% step decreased is inserted at

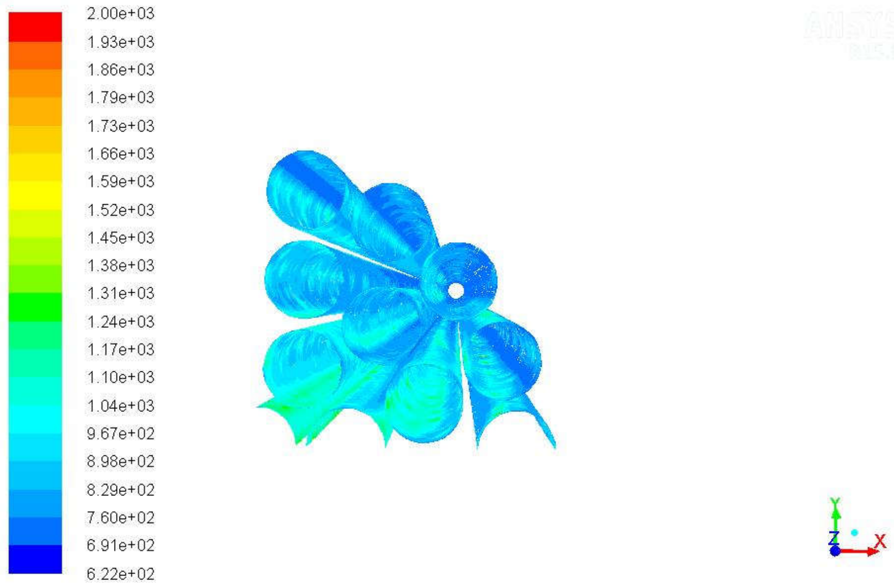

Figure 4 Cladding surface distributions at t=3s

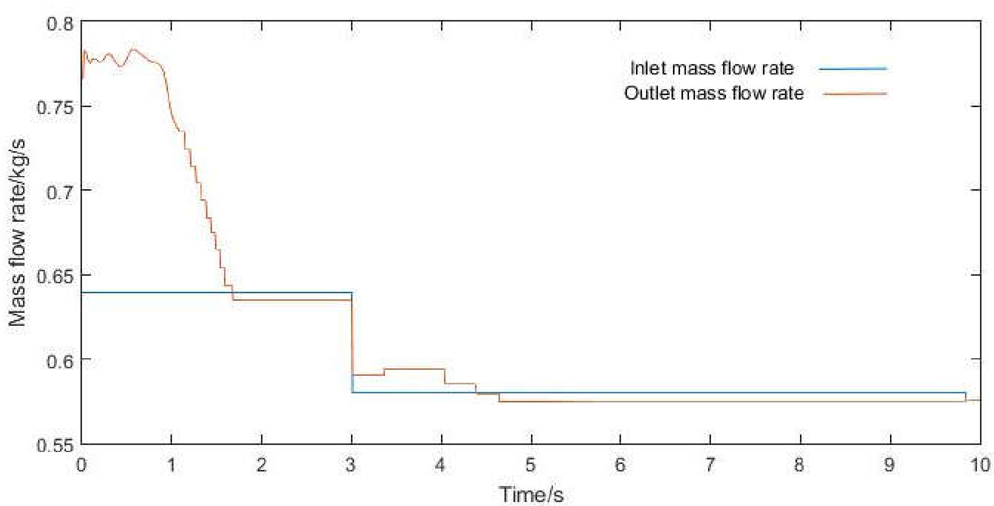

Figure 5(a) Inlet and outlet mass mass flow rate variation with time

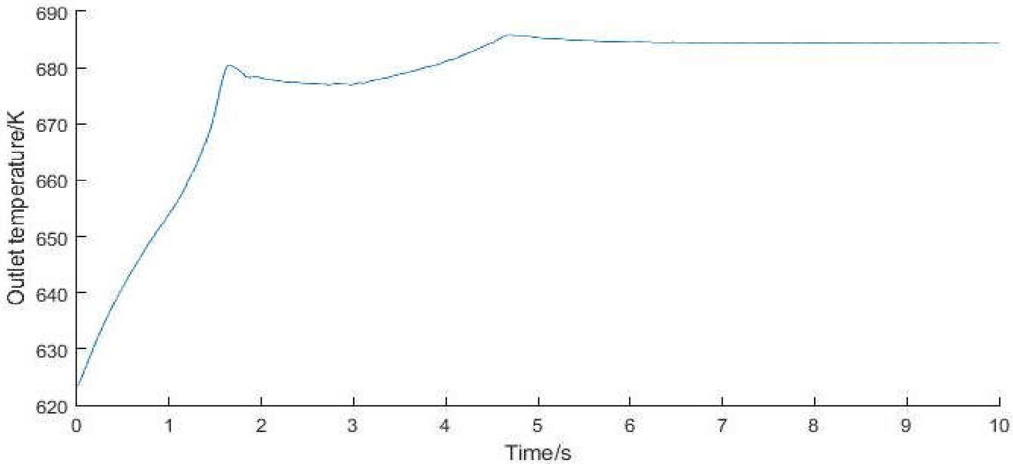

Figure 5(b) Outlet temperature ot the fuel bundle variation with time

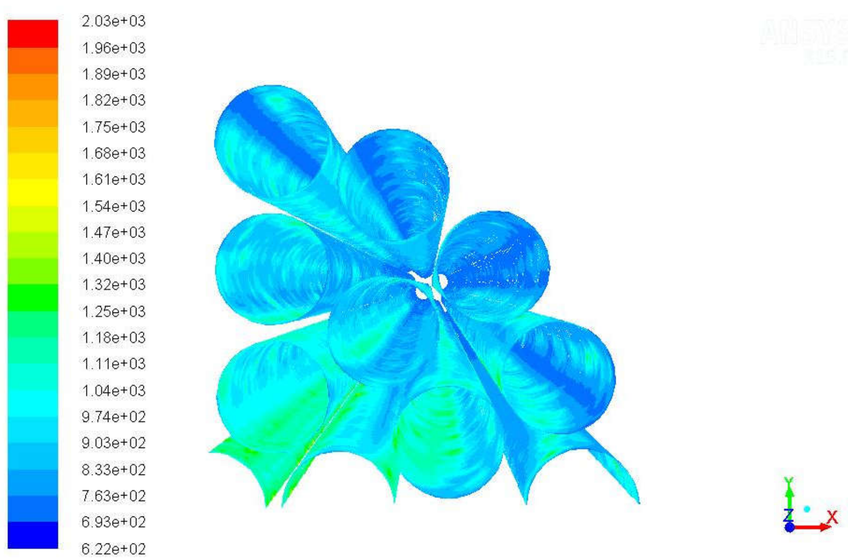

Figure 5(c) Cladding surface temperature(K) distributions

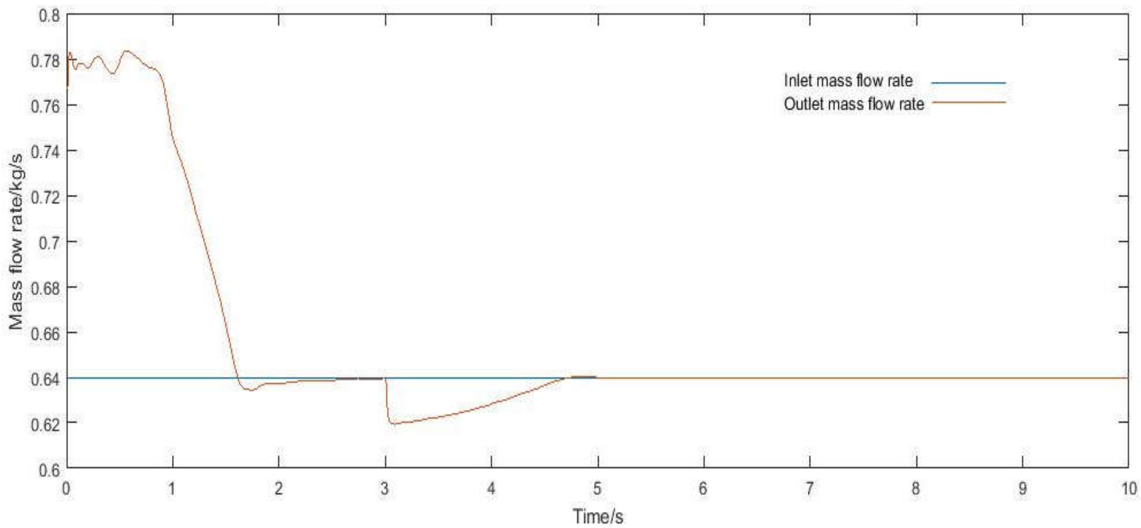

Figure 6(a) Inlet and outlet mass mass flow rate variation with time

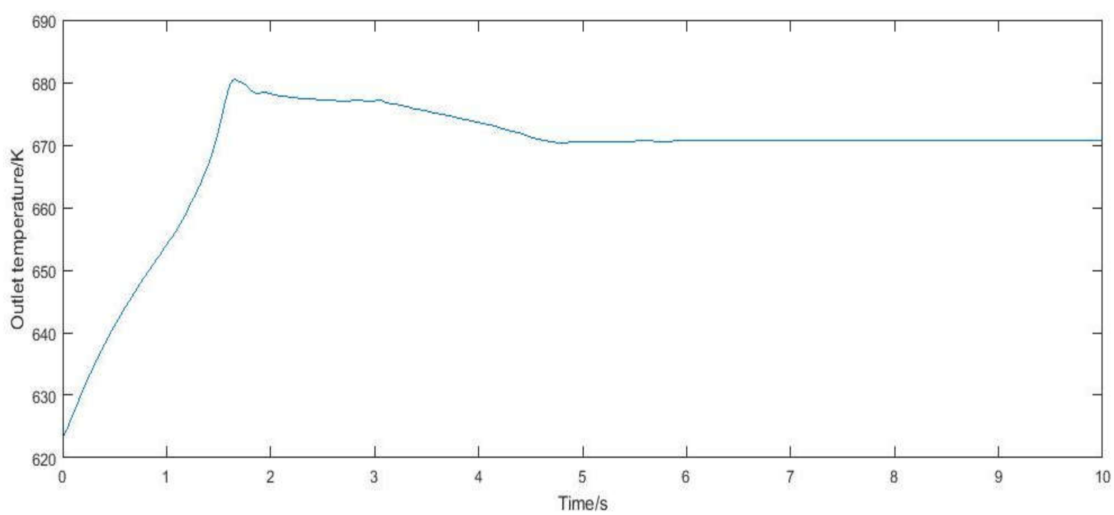

Figure 6(b) Outlet temperature of the fuel bundle variation with time

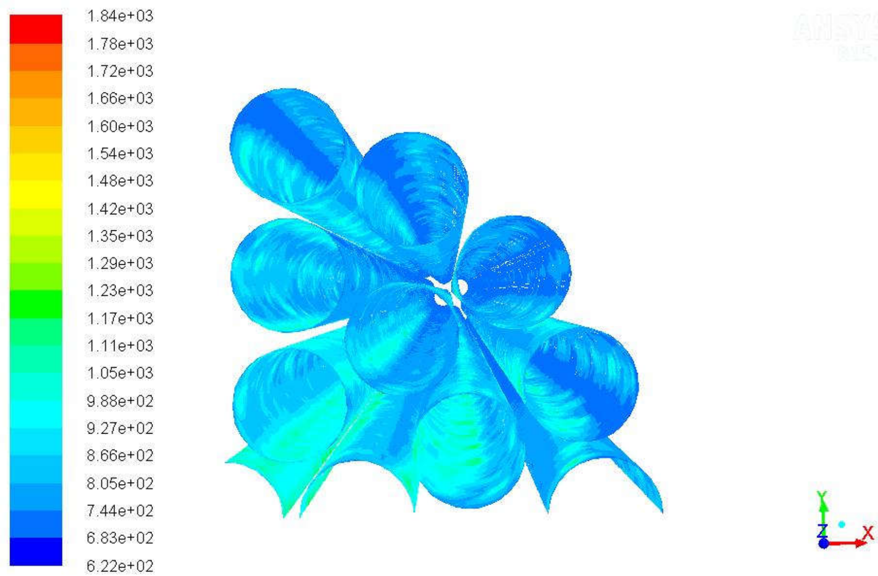

Figure 6(c) Cladding surface temperature(K) distributions

3s, the outlet mass flow rate present a sharp decrease along with. This is mainly due to the sharp density change around the pseudocritical point when the heat flux decreases. Figure 6(c) shows the cladding surface temperature distributions at t=10s when heat flux decrease 10% at t=3s.Although the fuel rod cladding surface temperature is still uniform, the maximum value shown is below the design value.

4. Conclusions

Transient flow and heat transfer behaviors of supercritical water in the fuel bundle perturbed under two conditions at the design point are investigated in this paper. The flow is not steady at the beginning because of the special property of supercritical water. When perturbances are inserted, the system can return to steady state in a short time. However, the cladding surface temperature distribution is not uniform for each fuel rod and the maximum value is above the design value when the system return to relatively stable. This should be controlled to make sure safety when a nuclear reactor is normally working. Since the Canadian SCWR is still in conceptual design, experimental data are needed in the future to validate simulation results.

Acknowledgements

The organizing committee would like to thank all authors for their submissions to CFD2017.

References

[1] US NERAC and GIF, A technology roadmap for generation IV nuclear energy systems, The U.S. DOE nuclear energy research advisory committee and the generation IV international forum, 2002.

[2] C.K.Chowand, H.F.Khartabil. Conceptual fuel channel designs for CANDU-SCWR. *Journal of Nuclear Engineering and Technology*, 40:139-146,2007.

[3] L Leung. Overview of SCWR Concepts. In IAEA-SJTU Course on Science and Technology of SCWRs ,Shanghai, China, 2013.

[4] Azih C, Brinkerhoff J R, Yaras M I. Direct numerical simulation of convective heat transfer in a zero pressure gradient boundary layer with supercritical water. *Journal of Thermal Science*, 21(1): 49–59,2012.

[5] X. K. Zeng, Y. L. Li, X Yan, et al. CFD method application in supercritical water cooled reactor thermohydraulic research. *Jounal of Nuclear Engineering*, 34(2): 114–120,2013.

[6] Y F Zhang, H X Li, L X Li, *et al*. A new model for studying the density wave instabilities of

supercritical water flows in tubes. *Journal of Applied Thermal Engineering*,75:397-409,2015.

[7] R. V. Maitri, G. Dutta, C. Zhang and J. Jiang. Numerical Analysis of the Deterioration of Heat Transfer Phenomenon in Supercritical Water Flow in Circular Tubes Using Different Turbulence Models. 22nd Annual Conference of the CFD Society of Canada, Toronto, Canada, 2014

[8] Y. N. Zhang, C. Zhang, J. Jiang, Numerical simulation of heat transfer of supercritical water in SCWR, 17th Annual Conference of the CFD Society of Canada, Ottawa, Canada, 2009

[9] Heavy Water Reactors: Status and Projected Development, Technical Report Series No. 407, International Atomic Energy Agency, 2002

[10] W. Wagner, A. Kruse, Properties of Water and Steam: The Industrial Standard IAPWS-IF 97 for the Thermodynamic Properties and Supplementary Equations for Other Properties, Springer, New York ,1998

[11] I.Pioro, S.Mokry, Thermophysical Properties at Critical and Supercritical Pressures, Heat Transfer:574-592,2011

[12] 14.0 ANSYS fluent theory guide. ANSYS inc, 2011

[13] Y. N. Zhang, C. Zhang, J. Jiang, Numerical simulation of heat transfer of supercritical fluids in circular tubes using different turbulence models, *Journal of Nuclear Science and Technology*,48(3):366-373,2010