NURETH14-551

CRITICAL HEAT FLUX IN CANDU MODERATOR FOLLOWING A PRESSURE TUBE TO CALANDRIA TUBE CONTACT - PART I

A. BEHDADI, J.C. LUXAT

McMaster University, Engineering Physics Department, Hamilton, Ontario, Canada L8S 4L7
e-mail: behdada@mcmaster.ca; luxatj@mcmaster.ca

ABSTRACT

Heavy water moderator surrounding each fuel channel is one of the important features in CANDU reactors that act as a heat sink for the fuel in the situations where other means of heat removal fail. In the critical break LOCA scenario, fuel cooling becomes severely degraded due to rapid flow reduction in the affected flow pass of the heat transport system. This can result in pressure tubes experiencing significant heat-up while coolant pressure is still high, thereby causing uniform thermal creep strain (ballooning) of the pressure tube (PT) into contact with its calandria tube (CT). The contact of the hot PT with the CT causes rapid redistribution of stored heat from the PT to CT and a large spike in heat flux from the CT to the moderator fluid. For lower subcooling conditions of the moderator, this heat flux spike can cause dryout of the CT. This can detrimentally affect channel integrity if the CT post-dryout temperature becomes sufficiently high to result in thermal creep strain deformation. The focus of this research is to develop a mechanistic model to predict Critical Heat Flux (CHF) on the CT surface following a contact with its pressure tube. A COMSOL multi-physics model using a two-dimensional transient fluid-thermal analysis of the CT surface undergoing heat up is used to predict flow and temperature profile on the CT surface. A mechanistic CHF model is to be proposed based on a concept of wall dry patch formation, prevention of rewetting and subsequent dry patch spreading.

Keywords: CHF, Subcooled pool boiling, Mechanistic multi-physics modeling, CANDU Moderator

1. INTRODUCTION

A loss of Coolant Accident (LOCA) caused by a break in one of the primary heat transport system pipes is one of the most safety significant accidents in water reactors and can be the precursor to more serious accidents. The CANDU reactor safety research program in Canada has a strong focus on developing and verifying computer models that predict the reactor process and safety systems accurately during accident situations. Heavy water moderator surrounding each fuel channel is one of the important features in CANDU reactors that act as a heat sink for the fuel in the situations where other means of heat removal fail. During postulated LOCAs, for a particular break size and location referred to as critical break LOCA, the coolant flow through a portion of the reactor core decreases rapidly to very low flow rates before main heat transport pump head degradation causes increased reverse flows through the affected channels and reestablishes fuel cooling. Subsequently, emergency coolant injection then ensures sustained fuel cooling.

In the critical break LOCA scenario, fuel cooling becomes severely degraded due to rapid flow reduction in the affected flow pass of the heat transport system. This can result in pressure tubes experiencing significant heat-up while coolant pressure is still high, thereby causing uniform thermal creep strain (ballooning) of the pressure tube (PT) into contact with its calandria tube (CT). The contact of the hot PT with the CT causes rapid redistribution of stored heat from the PT to CT and a large spike in heat flux from the CT to the moderator fluid. For lower subcooling conditions of the moderator, this heat flux spike can cause dryout of the CT. This can detrimentally affect channel integrity if the CT post-dryout temperature becomes sufficiently high to result in thermal creep strain deformation. The focus of this research is to develop a mechanistic model to predict Critical Heat Flux (CHF) on the CT surface following a contact with its pressure tube. A COMSOL multi-physics model using a two-dimensional transient fluid-thermal analysis of the CT surface undergoing nucleation is used to predict fluid velocity and temperature on the outside wall of a CT.

The computer programs developed to assess the effect of heat transfer on the CT surface after a postulated PT/CT ballooning contact such as WALLR[6] and CATHENA[5] are one-dimensional codes and predict film boiling completely around the circumference of the CT. This does not happen in the experiments. In the experiments the calandria tube surface had patches of film boiling surrounded by patches of nucleate boiling. Therefore in this study the simulation has been performed in two dimension. In addition, in other studies in 2D or 3D, the whole calandria has been modeled. However, it is a concern that when PT/CT contact occurs in a channel, local moderator boiling may lead to CT dryout and may affect the flow around the CTs at higher elevations. This can detrimentally affect channel integrity if the CT post-dryout temperature becomes sufficiently high to result in thermal creep strain deformation. According to the importance of local prediction of flow and temperature distribution around a single channel, this study has been performed in order to investigate the local thermalhydraulic conditions on the calandria tube wall that will influence potential dryout. This problem has been analyzed using the general purpose finite element code COMSOL.

In this paper, the CANDU moderator flow and temperature distribution have been studied by developing a Computational Fluid Dynamics (CFD) model. The model predicts the fluid velocity and temperature distribution around a channel in normal operation and also after a postulated ballooning deformation of the pressure tube (PT) into contact with its calandria tube (CT). The present research is focused on establishing the limits for dryout occurrence on the CTs after PT/CT contact. CT dryout may occur due to the large spike of heat flux to the moderator after contact. The CT post-dryout temperature may become sufficiently high to result the thermal creep strain deformation and affect the channel integrity. In this study two different turbulent models, standard $k-\varepsilon$ and $k-\omega$, have been used and compared in order to consider turbulence in moderator flow. Governing equations have been solved by the finite element software package COMSOL. The buoyancy driven natural convection, the local moderator subcooling, fluid velocity, wall temperature and heat flux has been analyzed in the model. The flow pattern and temperature distribution predicted by both turbulent models indicate a greater tendency for film boiling to occur at low subcoolings and at the top or bottom of the CT.

2. MODEL DESCRIPTION

In this model, a single phase fluid inside the moderator and a uniform PT/CT contact has been considered. During normal operation, heat can be deposited to the moderator in two different ways. The first one is by direct heating of neutrons, decay heat from fission products and gamma rays. The second way, which accounts for a small portion of the total heat load, is by heat convection from the surface of fuel channels. The total heat load to the moderator is taken to be 103 MW (about 103% of full power) consisting of 98.7 MW by volumetric direct heating and 4.3 MW by convective heat from fuel channel surface[7]. The convective heat transfer to the moderator is assumed to be uniformly with axial direction. In the postulated PT/CT contact, the heat load to the moderator fluid is governed by the conduction and convection heat transfer through the solid walls and the fluid surrounding the channel.

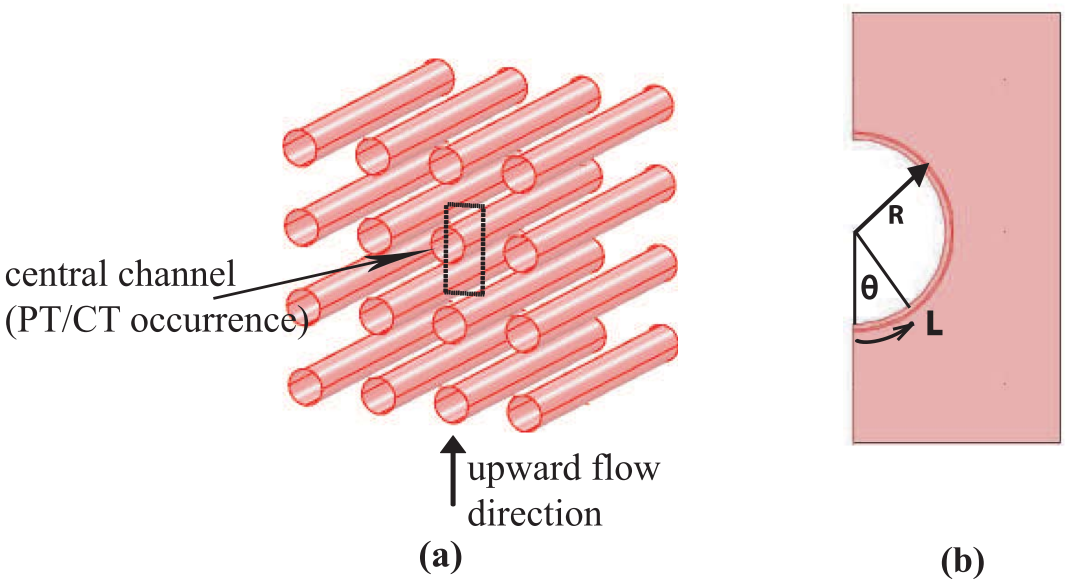

Figure 1. a) part of calandria tubes in 3D, b) the model geometry in 2D

The geometry is depicted in Figure 1-a. The arrow indicates the flow direction for upward flow. Neglecting any end effects from the walls of the vessel, the solution is constant in the direction of the tubes and therefore the model is reduced to a 2D domain. The dashed line marks the model region in 2D which is shown in Figure 1-b. L denotes the arc-length associated with the angle θ on the surface of the central CT in which PT/CT contact is postulated to occur.

3. GOVERNING EQUATIONS

The standard $k - \epsilon$ or $k - \omega$ turbulence model associated with a logarithmic wall function is used to predict the turbulence. The governing equations are:

- Reynolds Averaged Navier-Stokes (RANS) equations in the heavy water moderator domain:

$$\frac{D\rho}{Dt} + \rho\nabla.\mathbf{u} = 0 \tag{1}$$

$$\rho\frac{D\mathbf{u}}{Dt} = \mathbf{F} - \nabla p + (\mu + \mu_T)[\nabla^2\mathbf{u} + \frac{1}{3}\nabla(\nabla.\mathbf{u})] \tag{2}$$

- Heat transport equation in the water domain and the solid tube walls (CT thickness):

$$\rho C_p\frac{DT}{Dt} = \nabla.((\mathrm{k} + \mathrm{k}_T)\nabla T) + Q \tag{3}$$

Where,

$\mathbf{u}$ is the velocity field, $[m/s]$
$\mathbf{F}$ is body force, $[N/m^3]$
p is pressure, $[Pa]$
ρ is fluid density, $[kg/m^3]$
μ is dynamic viscosity, $[Pa.s]$
∇ is vector differential operator energy, $[m^2/s^3]$
t is time, $[s]$
C_p is specific heat capacity, $[J/(kg.K)]$
Q is the heat source, $[W/m^3]$
I is the identity matrix.

The heat source is the volumetric heat flux by direct heating of neutrons, decay heat from fission products and gamma rays. Free convection as the result of the density gradient, is added to the momentum balance in the body force term.

For Navier-Stokes equations in the fluid domain the specified boundary conditions are:

- A pressure difference between inlet and outlet given by the mass flow.
- Normal flow and stream-wise periodic conditions for velocity, at the inlet and outlet.
- Symmetry boundary condition at the region borders.

For the heat transport equation, the boundary conditions are specified as:

- Moderator temperature with different subcoolings at the inlet.
- Convection dominated transport at the outlet of the domain.

- At the inner surface of calandria tube a heat flux associated with normal operation is implemented for steady state case which is applied as the initial condition for the transient case. After a postulated PT/CT contact, the applied heat flux is increased with time until the saturation point is reached in the fluid.
- The borders of the domain are considered to be symmetric.

In order to compute turbulent flows successfully, some consideration is needed during the mesh generation. Turbulence through the spatially-varying effective viscosity plays an important role in the transport of mean momentum and other parameters. Therefore, in order to obtain more accurate results it is important to make sure that turbulence quantities in complex turbulent flows are properly resolved. The fluid velocity, the heat transfer coefficient and consequently the temperature at the boundaries are very sensitive to the number of mesh. The mesh size was made small enough until no significant change was observed in the results by further refinement.

4. RESULTS AND DISCUSSION

To obtain the initial values for the transient case, a steady-state problem has been solved first. In the steady-state case the channel is in the normal operation. In the transient case, the applied heat flux is increased with time until the saturation point is reached in the fluid. The minimum time step used in the model is $0.001s$. By further decreasing in time step, no significant change was observed. Heat is implemented uniformly to the inner surface of the tube and increases exponentially by time. After $1s$ it reaches to its maximum value at which the maximum fluid temperature around the cylinder is near saturation. The transient results are shown separately for $k - \epsilon$ and $k - \omega$ models.

The temperature distribution and velocity streamlines for an inlet flow velocity of $0.5m/s$ are shown in Figure 2 and 3 for upward and downward flows, respectively. Three different areas for the velocity field can be clearly seen: 1) the stagnation point in the front, 2) the recirculation zone in the rear behind the cylinder, 3) high velocity regions in the middle of the cylinder.

Figure 2 shows the velocity streamlines for upward flow obtained by both $k - \epsilon$ and $k - \omega$ turbulent models. It can be seen that $k - \omega$ model predicts a larger wake in the downstream region. The same behavior was observed for downward flow showing in Figure 3.

Temperature distribution of the fluid around the CT is shown in Figure 4 and Figure 5 for different subcoolings of upward and downward flow, respectively. Maximum temperature in the fluid is considered to be less than saturation. For each subcooling starting at the stagnation point ($\theta = 0$ and $L = 0$ for upward flow and $\theta = 180$ and $L = 0.21$ for downward flow), temperature decreases with increasing arc-length due to increase in velocity and heat transfer coefficient. However by developing the laminar boundary layer on the CT surface, the heat transfer coefficient decreases and temperature increases to its maximum value considered in this model. Eventually, separation occurs (at $L = 0.16$ for upward flow and at $L = 0.04$ for downward flow) and fluid temperature declines due to the increase in heat transfer coefficient as a result of the considerable mixing associated with the wake region. However, the variations

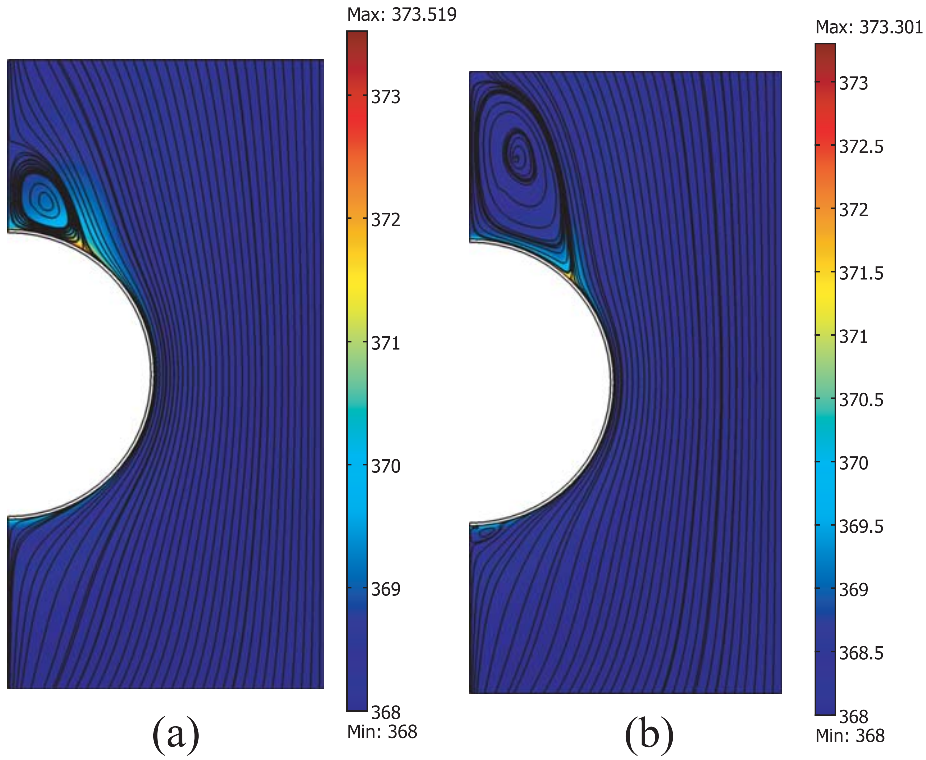

Figure 2. Velocity streamlines for upward flow, (a) $k - \epsilon$, (b) $k - \omega$ model

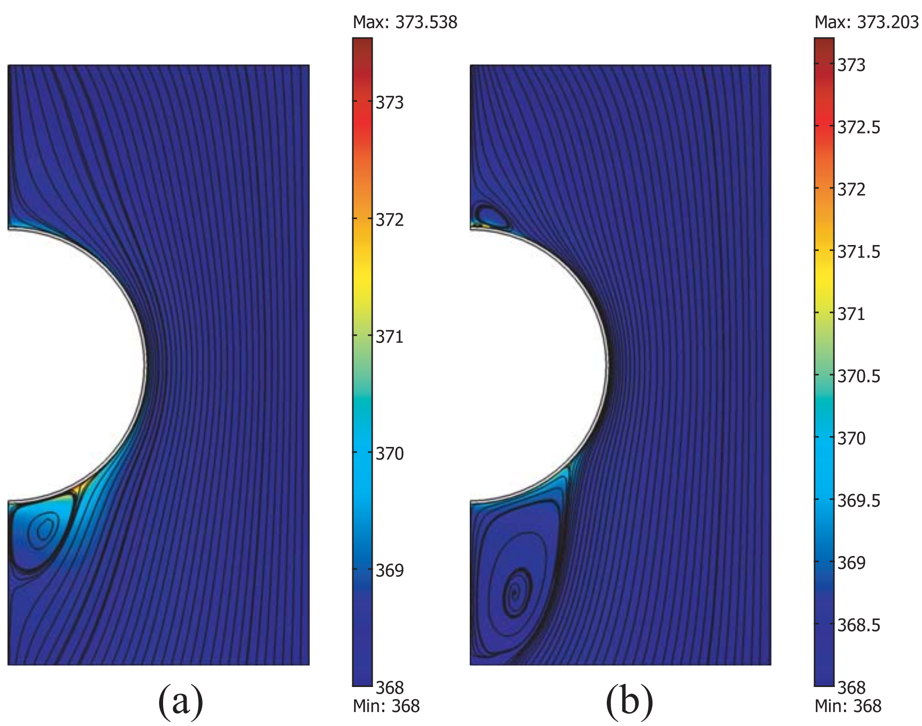

Figure 3. Velocity streamlines for downward flow, (a) $k - \epsilon$, (b) $k - \omega$ model

in local fluid temperature become smaller as subcooling reduces. Qualitatively, this indicates a greater tendency for vapor film to extend around the CT surface as subcooling decreases. This signifies the importance of moderator fluid subcooling in the situation of critical break LOCA inside the CANDU reactor. As subcooling increases the region of high fluid temperature becomes increasingly localized which will result in higher likelihood of quenching. This makes the spreading of drypatches around the CT more difficult.

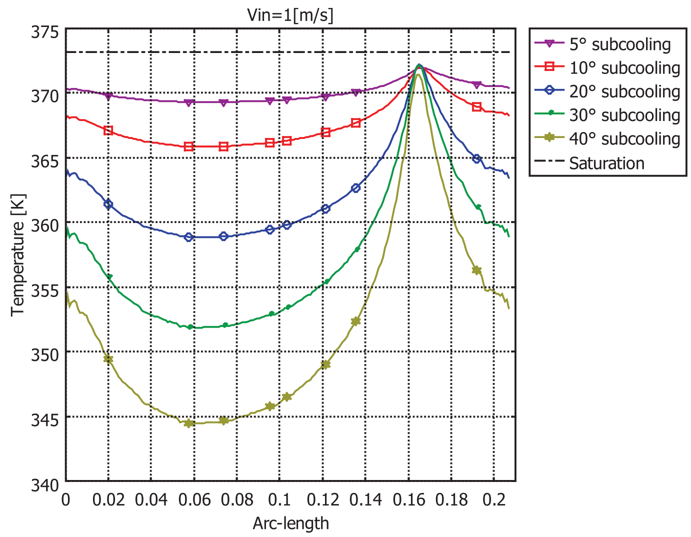

Figure 4. Local temperature around CT for upward flow by $k - \epsilon$ model

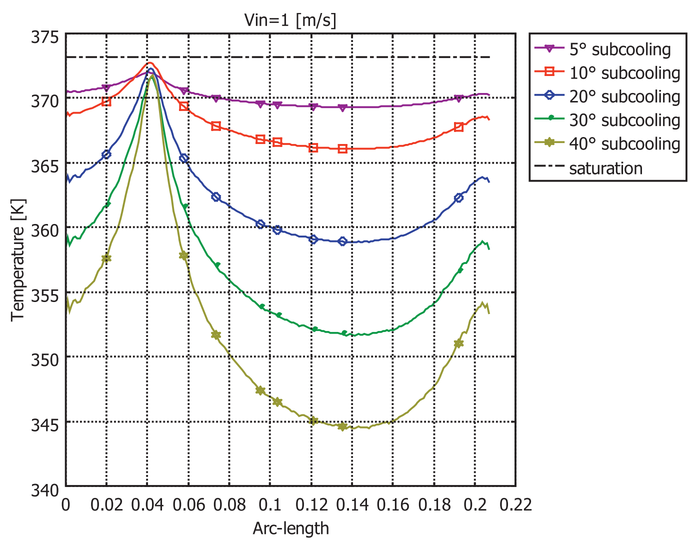

Figure 5. Local temperature around CT for downward flow by $k - \epsilon$ model

Temperature distribution around the CT for different subcoolings obtained by $k - \omega$ model is shown in Figures 6 and 7 for upward and downward flows respectively. Comparing Figures 4 and 6 indicates that in the $k - \omega$ predictions the temperature decreases sharply after the

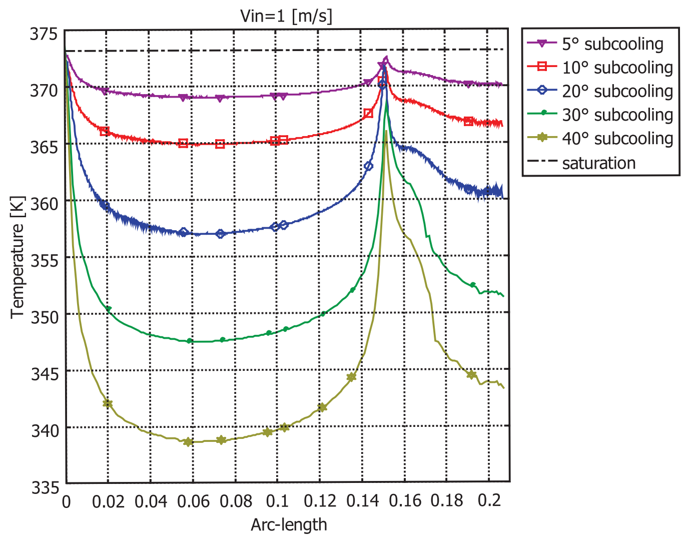

Figure 6. Local temperature around CT for upward flow by $k - \omega$ model

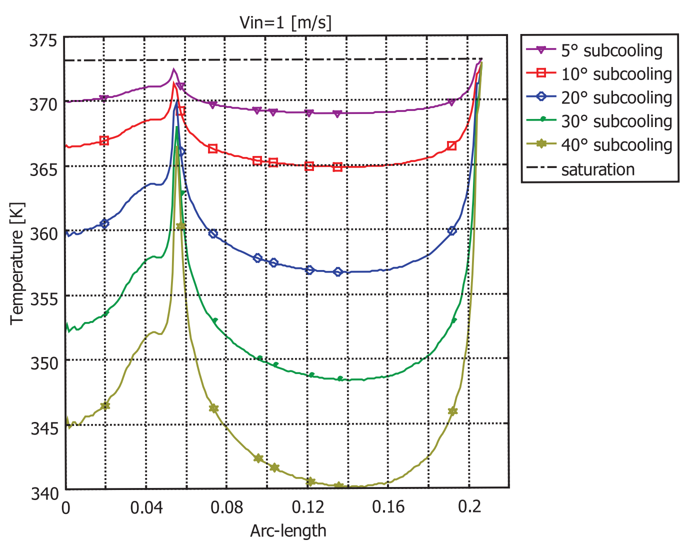

Figure 7. Local temperature around CT for downward flow by $k - \omega$ model

stagnation point from its highest value and it is near constant around the cylinder before the separation point at which the temperature again increases sharply. After the separation point, in the wake region again the temperature decreases due to the flow recirculation in the wake region. Although the same behavior was observed by the $k-\epsilon$ model but the temperature decreases smoothly from the stagnation point and again increases smoothly to reach to the separation point. In fact the high temperature regions predicted by $k-\epsilon$ model are not as localized as the one predicted by $k-\omega$ model.

For both turbulent models, by decreasing the inlet velocity, the overall temperature becomes higher especially in downward flow where the effect of buoyancy forces become significant. At very low velocity ($0.2 m/s$), there is a relatively high temperature increase in the wake region for both upward and downward flows.

5. FUTURE WORK

In almost all of the existing CHF models, the critical heat flux was treated constant and uniform over the entire heating surface which is a reasonable assumption for upward facing surfaces. The main goal in this research is to develop a theoretical model based on the above arguments for a horizontal cylinder with dimensions of CANDU calandria tubes. In the case of a cylinder with diameter of CANDU calandria tube, it should be noted that as vapor bubbles grow and depart from the heating surface, they tend to flow upward along the circumference. According to the study of Cheung and Haddad [3] for saturated pool boiling on a downward facing curved heating surface, this results into a two phase boundary layer development along the surface of the cylinder. The formation of this boundary layer flow driven by buoyancy, may affect the boiling process and the local CHF limit. In fact the local critical heat flux could be substantially modified by the flow leading to a significant spatial variation on the CHF values along the heating surface in the flow direction. In spite of its practical importance, very little is known about the CHF under this condition. A boundary layer analysis will be performed treating the two-phase motion as an external buoyancy driven flow to determine the liquid supply rate and thus the local critical heat flux. The effect of subcooled conditions inside the CANDU moderator will be analyzed, as well.

6. CONCLUSION

The focus of this research was to establish a Computational Fluid Dynamics (CFD) model for predicting the moderator flow field and temperature distribution around one single channel to investigate the potential of dryout occurrence on the CT surface following a PT/CT contact. Buoyancy forces due to density variations has been taken into account and the fluid is considered to be single phase. Two different turbulent models $k-\epsilon$ and $k-\omega$ have been used separately to predict the fluid turbulence and the obtained results were compared together. The model clearly indicates the wake region behind the cylinder. It also shows the stagnation region in front of the cylinder at which the velocity is zero and it can be concluded that the stagnated flow can result to accumulation of the bubbles and consequent stable vapor film generation at this region. Some differences were observed in the predictions of the two turbulence models. $k - \omega$ model predicts the highest temperature at the stagnation point in front of the cylinder while $k - \epsilon$ model predicts the highest temperature at the separation point on the cylinder. The predicted local temperature variations with subcooling illustrates a greater tendency for vapor film to extend around the CT surface at low subcooling for both $k - \omega$ and $k - \epsilon$ models. This shows the importance of moderator subcooling in preventing the vapor film to extend on the CT surface. The high temperature regions predicted by $k - \epsilon$ model are not as localized as the one predicted by $k - \omega$ model.

ACKNOWLEDGMENTS

The financial support of the Natural Science and Engineering Research Council (NSERC) and the University Network of Excellence in Nuclear Engineering (UNENE) is gratefully acknowledged.

REFERENCES

1. A. Behdadi, J.C., Luxat, "Comparison of Turbulent Models for CANDU Moderator following a Pressure Tube to Calandria Tube contact", *31st Annual Conference of the Canadian Nuclear Society, Montreal, Canada*, (2010).

2. A. Behdadi, J.C., Luxat, "2D Modeling of Moderator Flow and Temperature Distribution around a Single Channel after Pressure Tube/Calandria Tube Contact", *30th Annual Conference of the Canadian Nuclear Society, Calgary, Canada*, (2009).

3. F.B. Cheung, K.H., Haddad, "Steady-State Observation of Critical heat Flux Phenomenon on a Downward Facing Hemispherical Surface", *Proceedings of the 23rd water reactor safety meeting*, (1997).

4. COMSOL Chemical Engineering Module User's Guide, http://www.comsol.com, (2008).

5. H.Z. Fan, Z. Bilanivic, "Use of CATHENA to Model Calandria-Tube/Moderator Heat Transfer After Pressure Tube/Calandria tube Ballooning Contact" *CNS 6th International Conference on Simulation Methods in Nuclear Engineering, Montreal, Quebec, Canada*, (2004).

6. G.E. Gillespie,"An Experimental Investigation of Heat Transfer From a Reactor Fuel Channel to Surrounding Water", *2nd Annual Conference of the Canadian Nuclear Society 1981*, p 157-163, (1981).

7. M. Kim, S. Yu and H. Kim,"Analysis on Fluid Flow and Heat Transfer inside Calandria vessel of CANDU-6 using CFD", *Nuclear Engineering and Design*, v236, n11, p1155-64, (2005).