ADVANCED CFD SIMULATIONS OF TURBULENT FLOWS AROUND APPENDAGES IN CANDU FUEL BUNDLES

F. Abbasian, G. I. Hadaller and R. A. Fortman
Stern Laboratories Inc.,
(Hamilton, Ontario, Canada L8H 3L3, (905) 548-5328, fabbasian@sternlab.com)

ABSTRACT- Computational Fluid Dynamics (CFD) was used to simulate the coolant flow in a modified 37-element CANDU fuel bundle, in order to investigate the effects of the appendages on the flow field. First, a subchannel model was created to qualitatively analyze the capabilities of different turbulence models such as k.ε, Reynolds Normalization Group (RNG), Shear Stress Transport (SST) and Large Eddy Simulation (LES). Then, the turbulence model with the acceptable quality was used to investigate the effects of positioning appendages, normally used in CANDU 37-element Critical Heat Flux (CHF) experiments, on the flow field.

It was concluded that the RNG and SST models both show improvements over the k.ε method by predicting cross flow rates closer to those predicted by the LES model. Also the turbulence effects in the k.ε model dissipate quickly downstream of the appendages, while in the RNG and SST models appear at longer distances similar to the LES model. The RNG method simulation time was relatively feasible and as a result was chosen for the bundle model simulations. In the bundle model simulations it was shown that the tunnel spacers and leaf springs, used to position the bundles inside the pressure tubes in the experiments, have no measureable dominant effects on the flow field. The flow disturbances are localized and disappear at relatively short streamwise distances.

Introduction

The coolant flow inside the nuclear reactor core is affected by the unique structure of fuel bundles. The tubular structure of fuel rods and the irregular geometry of the appendages account for important characteristics such as turbulence mixing, inter-subchannel mass and kinetic energy transfer and enthalpy balance. It is imperative to understand such features in order to make improvements to critical power limitations as they directly affect the reactor thermalhydraulics.

In this paper, a CFD analysis is presented to investigate the capacity of different turbulence models in simulating the flow field inside CANDU fuel bundles. The study is concentrated on the non-uniform flows generated downstream of the appendages and the capability of each model to more accurately predict the flow pattern along the bundle. The turbulence model comparisons were made using a subchannel model. Then a suitable method was used to perform a CFD analysis on the flow field inside bundle J, the bundle number 10 inside a CANDU 37-element fuel channel. The purpose of the study is to compare the bundles used in the CHF experiments performed in Stern Labs to the real bundles used in the reactors. In the experimental fuel bundles, normally some positioning appendages are used that do not exist in the reactors.

The simulation results are used to confirm whether or not such structural discrepancy would affect the flow field.

The flow patterns inside fuel bundles have been extensively studied that have been presented in numerous papers. While a thorough literature review is beyond the scope of this work, the papers presented by Rowe et al [1] and Moller [2] are likely the most the relevant to the subject of this paper. The great experimental work presented by [1,2] showed correlations between the geometry effects, such as gap-to-diameter ratios in densely packed rod bundles, and the turbulence intensities. The latter work stepped forward by presenting a characterization of turbulence anisotropy and the fact that the azimuthal and axial (streamwise) turbulence patterns behave differently near and away from the walls. Such findings are particularly important when studying the intersubchannel mixing effects on the wall surface heat transfer calculations.

As performing representative experiments is often expensive and time consuming, CFD simulations are used nowadays to help understand the flow behavior inside the fuel bundles. The authors have previously presented different CFD applications in the CANDU reactors [3,4]. The present work is aiming to focus on turbulence modeling and capturing the effects caused by the bundle appendages.

1. Subchannel Model

1.1 Computational model

In order to investigate turbulence model capabilities, a subchannel model was used to provide a finer mesh proportional to the appendages sizes. The subchannel model is extracted from a modified CANDU 37-element bundle geometry as shown in Figure 1.

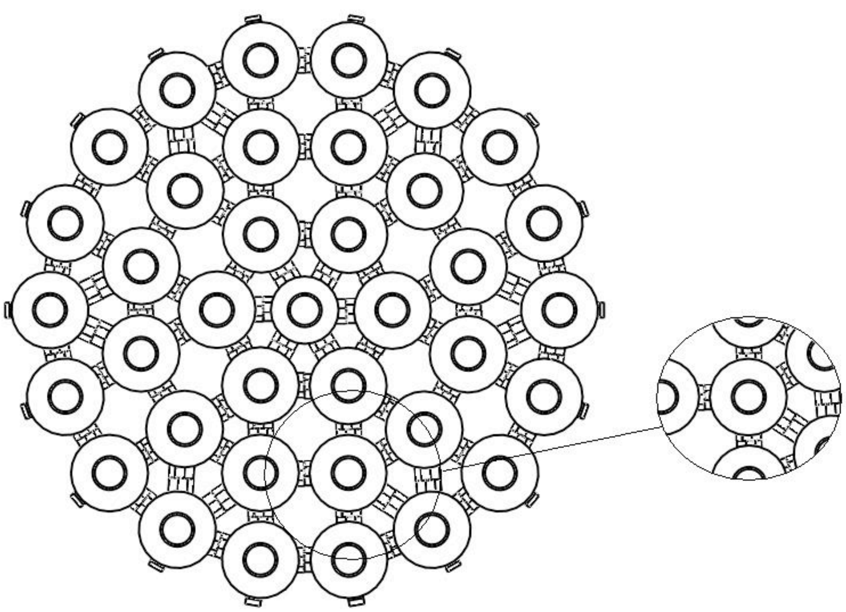

Figure 1 The subchannel position in the modified CANDU 37-element bundle

The computational domain and cross-sectional mesh are shown in Figure 2. The dimensionless distance from the wall (y^+) is close to unity at the wall boundaries. The total number of volume cells is about 6.2 million. Typical single-phase scheme with 3D continuity and Navier-Stokes (N-S) equations are used in the models. The inlet and outlet mass flow and pressure boundary conditions at an outlet pressure of 9 MPa and a plane-averaged velocity of 3.5 m/s were defined for the model.

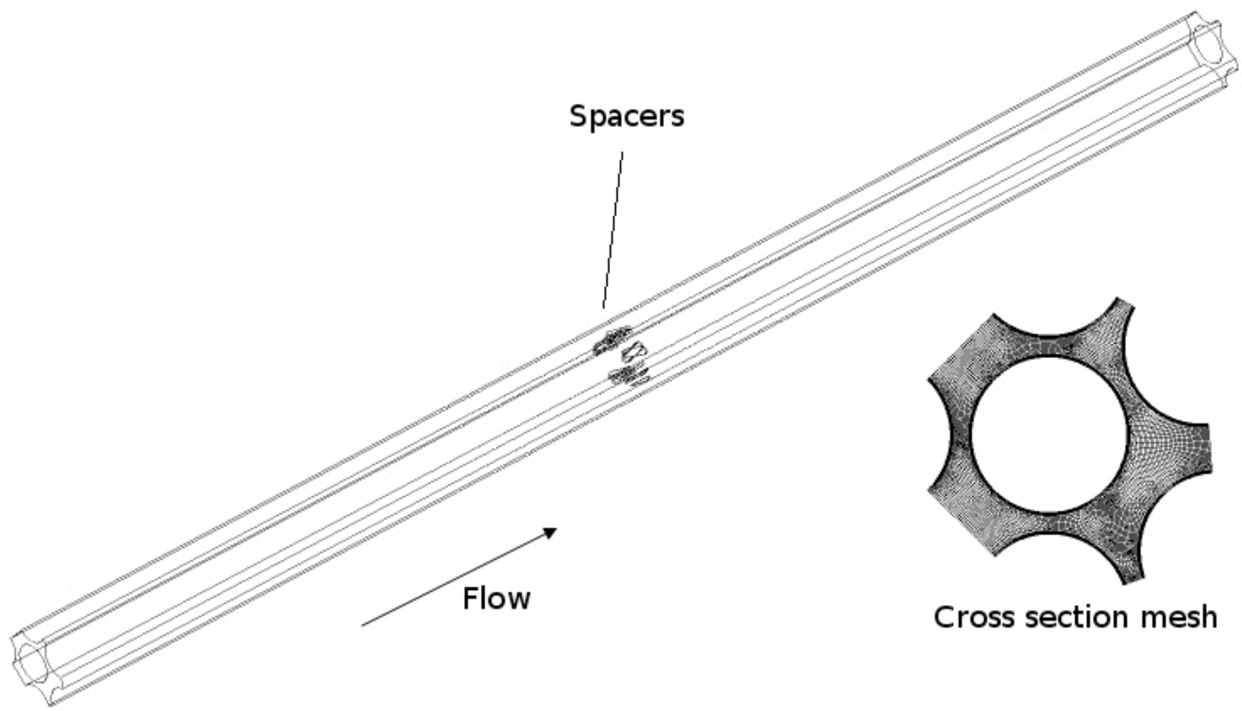

Figure 2 The subchannel computational model and the cross section mesh

1.2 Turbulence methods

The turbulence models are chosen based on the application and complexity of each method. In Reynolds Averaged Navier-Stokes (RANS) methods such as k.ε and k.ω, the Navier-Stokes equations are decomposed to the averaged and fluctuating flow field variables and rearranged in terms of turbulence kinetic energy (k) and turbulence dissipation rate (ε) or ω, that is specific dissipation rate (ε/k) in case of the k.ω method. The fluctuating parts of the N-S equations are defined as Reynolds stresses and are determined through Boussinesq hypothesis. The relationship between these two characteristics (k and ε or k and ω) is defined using turbulence viscosity and is used as the closing equation. Both k.ε and k.ω models are considered to be the typical Reynolds averaging methods with a variety of industry applications. Their relatively lower computational demands often make them a good candidate for various problems, however, when more details of rather complex flow fields are required, more advanced methods may be needed.

The Reynolds Normalization Group (RNG) method [5] is a refined k.ε method. In this method the closing equation for the turbulence viscosity is in the form of a differential equation that accounts for the Reynolds number effects, such that, at high Reynolds numbers the equation becomes similar to the one provided by the basic k.ε model. As a result, the low-Reynolds number effect is better captured by the RNG method. The RNG method is suitable for flow fields with a wide range of Reynolds numbers and swirling flow regimes. Another advanced method is the Shear Stress Transport (SST) method [6] that is a modification to the k.ω model. In this method, using a blending function the model switches between the k.ε and k.ω equations depending on the proximity to the wall regions. This model more accurately predicts the wall shear effects and realistic pressure gradients. Finally in LES, which unlike the previous methods is not a RANS approach, N-S equations are filtered and the large scale eddies (or the large scale turbulence) are directly resolved and the small scale eddies are modeled using one of the RANS methods. As the large-scale turbulence has greater effects on the flow field, LES is more capable of predicting the realistic turbulence effects compared to the RANS methods. It goes without saying that any improvement or refinement on the turbulence modeling will be at the cost of higher simulation time and convergence problems. The purpose of this study is comparing the ups and down of these methods.

1.3 Subchannel model results

Parallel processing was used for all the simulation runs as the high number of elements and rigorous computations associated with the advanced turbulence models make it almost impossible to provide a solution with a single-processor system within a reasonable time. The simulation time of each model is given in Table 1. For all the simulation models the under-relaxation factors were set at lower values and gradually increased towards the convergence. For the LES model, the solution was initiated at a time step size of 0.0005 (s) and for all other models a steady state solution was used.

Table 1 Solution elapsed time at convergence (8-CPU parallel processing/64 Gb RAM)

Turbulence model	k.ε	RNG	SST	LES
Elapsed time (s)	16440	17670	27840	198720

The cross-flow contours are shown in Figure 3 and Figure 4 at two distances downstream of the spacer plane, $2D_h$ and $5D_h$, respectively. The ratio of cross flow (spanwise) velocity to the average streamwise velocity is shown. For the LES method, time-averaged values have been obtained to be compared with the steady-state results of other methods.

The comparison of the results shows that the k.ε method under-predicts the magnitude of the cross-flows. Also, it is evident that closer to the spacer plane, the swirling flow regimes tend to coalesce and form large-scale vortices in the k.ε model. This trend is different in the other three methods where more small-scale turbulence-driven flows are present. The structure and location of the vortices captured by the RNG and SST methods are similar, especially at the $2D_h$ location. Further downstream of the spacer plane ($5D_h$), it appears that the k.ε method fails to predict any significant cross flow values and the other three methods predict almost same values. It is believed that the RNG method has improved the k.ε model capability of predicting higher values for the cross flows, closer to those obtained by the SST and LES models. Furthermore, in the RNG method a longer zone is affected by the spacers compared to the k.ε method. The only method predicting the peak local cross values, as high as 71% at $2D_h$, is the LES method.

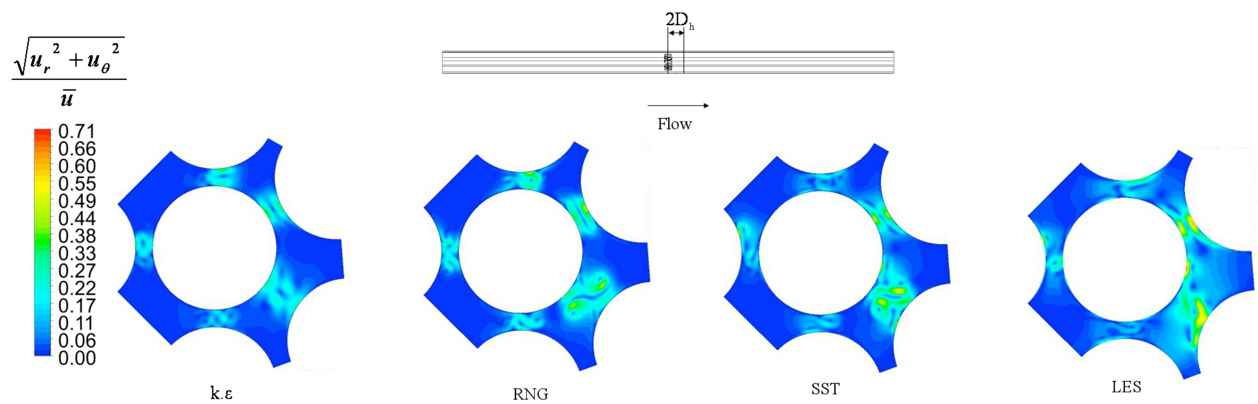

Figure 3 Cross flow ratio contours downstream of the spacer plane; u_r , u_θ and $\bar{u}$ are radial, tangential and average streamwise velocity components, D_h =hydraulic diameter

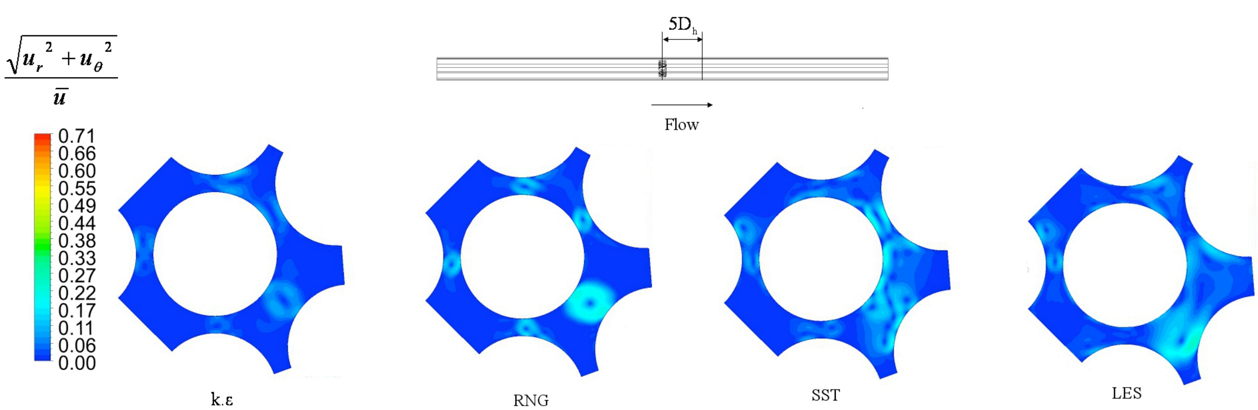

Figure 4 Cross flow ratio contours downstream of the spacer plane; u_r , u_θ and $\bar{u}$ are radial, tangential and average streamwise velocity components, D_h =hydraulic diameter

2. Bundle Model

After the results from the subchannel model were compared against each other and considering the simulation times given in Table 1, the RNG method was chosen as the most feasible approach for the bundle model.

2.1 Computational domain

The purpose of the bundle simulations is to investigate the effects of the bundle appendages on the flow field. In the Critical Heat Flux (CHF) experiments performed in Stern Laboratories Inc. since early 1980's, some appendages have been used to position the bundles at the center and bottom of the pressure tubes. The CFD models were developed for two different appendage types and the results were compared against the reactor case where no appendages of these kinds are used.

The three simulation models developed are shown in Figure 5. The bundle J or number 10 in the CANDU 37-element fuel string inside a 5.1% crept pressure tube is chosen for the analysis. The creep of the pressure tube is the result of the pressure stresses and neutron fluxes and happens as the fuel ages. The two typical creep profiles used in the CHF experiments are 3.3% and 5.1% (i.e. percentage of the diameter expansion with respect to the diameter at inlet). The 5.1% profile was chosen for the analysis as the appendages are largest in this creep profile and can cause strongest flow disturbances. Also, bundle J is the location with CHF occurrence and critical.

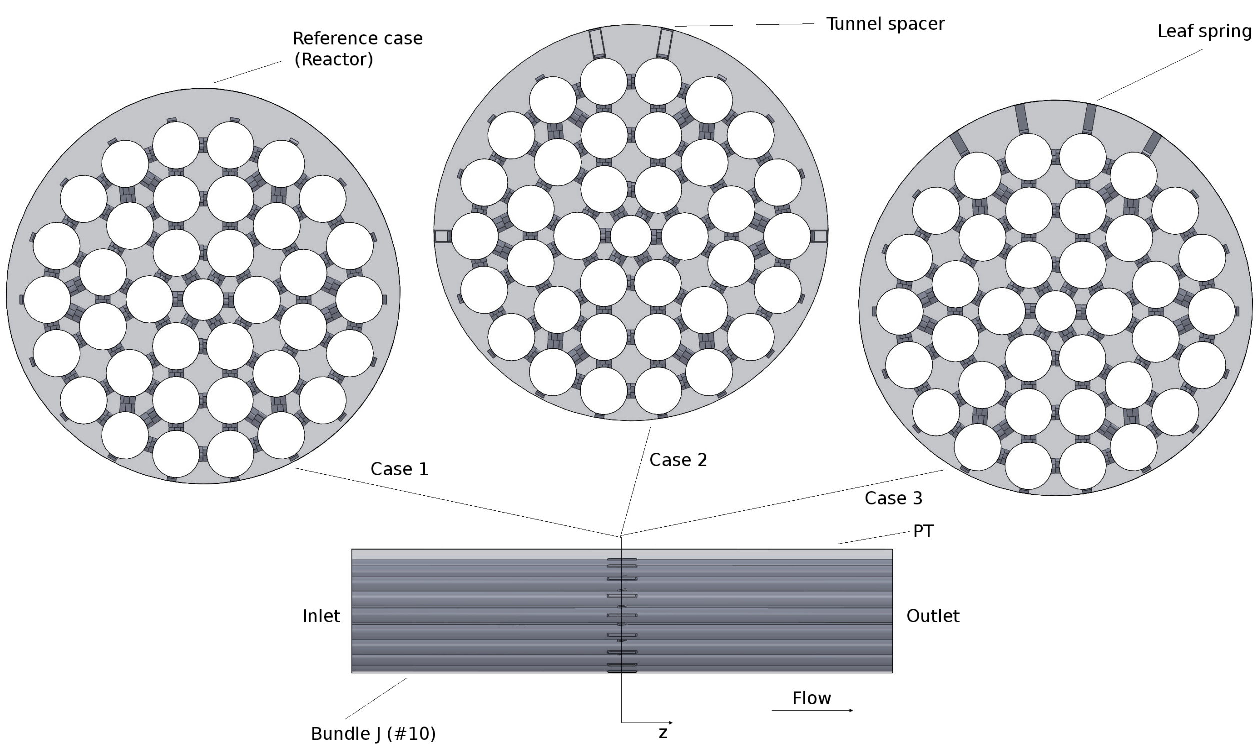

Figure 5 The bundle simulation model inside the pressure tube (PT)

The leaf spring appendages (case3) are the old design with the springs attached to the top four elements at the bundle mid-plane. The newer design, tunnel spacers shown by case 2, is a thin-wall rectangular channel attached to the top and side elements at the bundle mid-plane.

The bundle model boundary conditions are given in Table 2 and correspond to the operating conditions of Stern Labs single-phase heat balance tests, for which extensive experimental data are available for the future validation studies. The energy equations were added to the simulation models and local power was applied at the bundle J's elements. The total number of volume cells of the bundle model is about 22 million.

Table 2 The bundle model boundary conditions

Inlet temperature (^{o}C)	Outlet pressure (MPa)	Fuel string power (MW)	Flow rate (kg/s)
180	9	2	13.5

2.2 Bundle model simulation results

The flow field variables obtained from the converged solution are presented in this section. The local pressure variation along the bundle is shown in Figure 6. The location of the pressure plots are shown by the red dot at the bundle cross section that is located just downstream of the appendages. It is evident that the local pressure drop is more significant in the leaf spring model. This is due to the fact that the leaf springs cause flow obstruction as opposed to the tunnel spacers that cause minimal blockage. The stagnation downstream of the leaf spring leads to the pressure drop as observed in the plots. The plane-averaged pressure plots are shown in Figure 7, where lower effects on the pressure drops are observed. Although the tunnel spacer results are still closer to the reference case than the leaf springs, the overall pattern is the same for the three models, indicating a localized nature of the appendages effects.

The velocity overlay contours are shown in Figure 8. The absolute velocity difference between each case and the reference model is shown at four distances downstream of the appendages. The flow downstream of the leaf springs is clearly disturbed with an absolute difference value as high as 4 m/s. This shows the velocity field downstream of the leaf springs is majorly different than that of the reference case. On the other hand, the velocity difference between the tunnel spacer model and the reference case is minimal. For both the leaf spring and tunnel spacer models the flow disturbances fade away along the bundle. The velocity field can affect the surface heat transfer coefficient calculations and as a result the flow temperatures. The velocity and temperature overlay contours are shown in Figure 9 at 15 mm downstream of the appendages plane. The velocity disturbances have direct effects on the flow temperature at the exact same location of leaf springs as shown in the contours, while the tunnel spacer's effect on the flow temperature is small as expected. In general, the flow field is locally affected by the appendages and none of the designs has shown to cause dominant flow disturbances. The localized effects caused by the leaf springs are more pronounced at the vicinity of them due to the stronger flow obstruction.

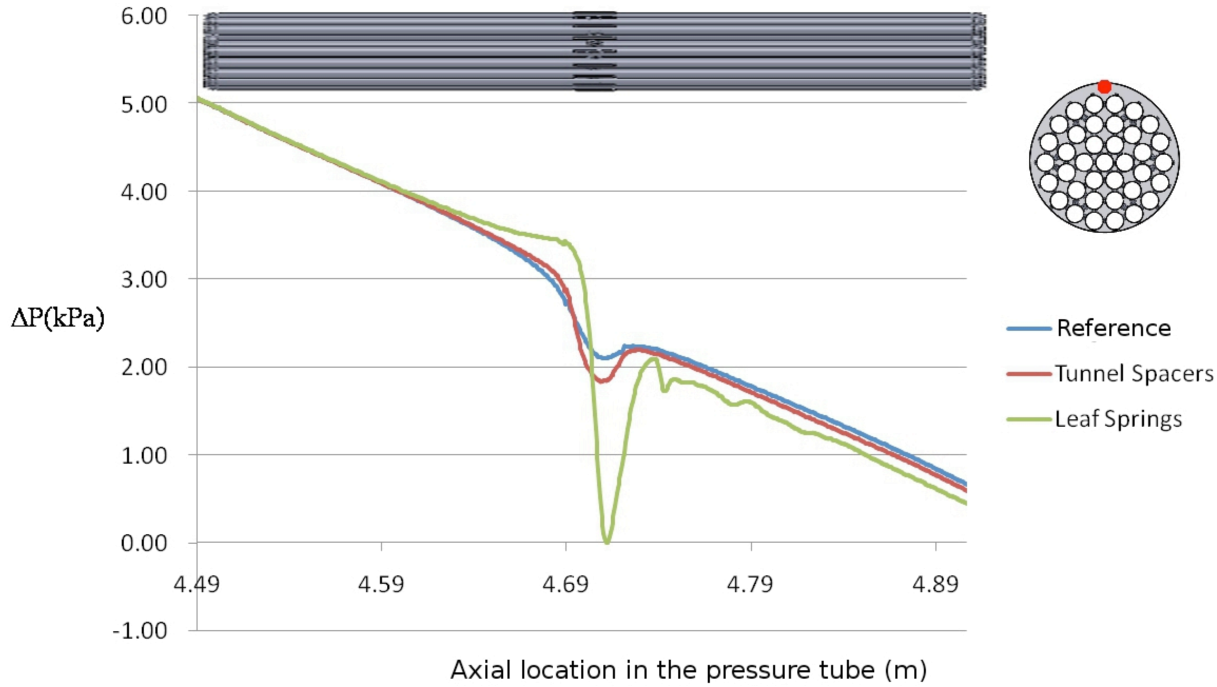

Figure 6 Local bundle pressure variations in proximity to the appendages

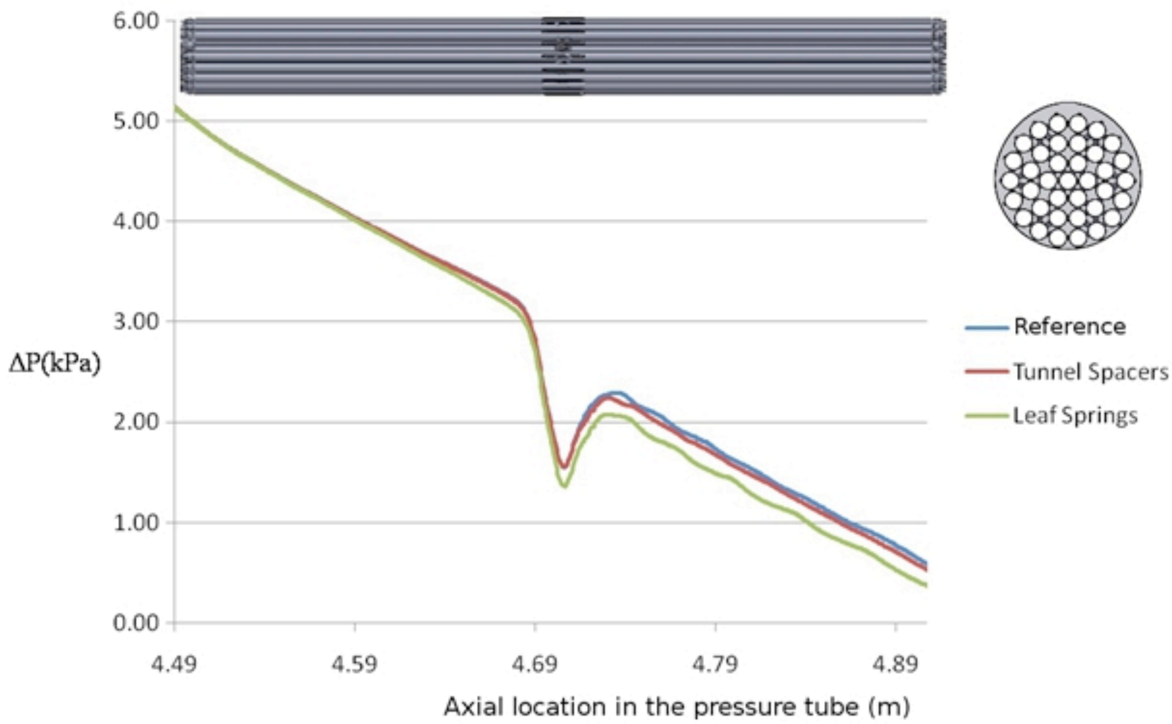

Figure 7 Plane-averaged bundle pressure plots

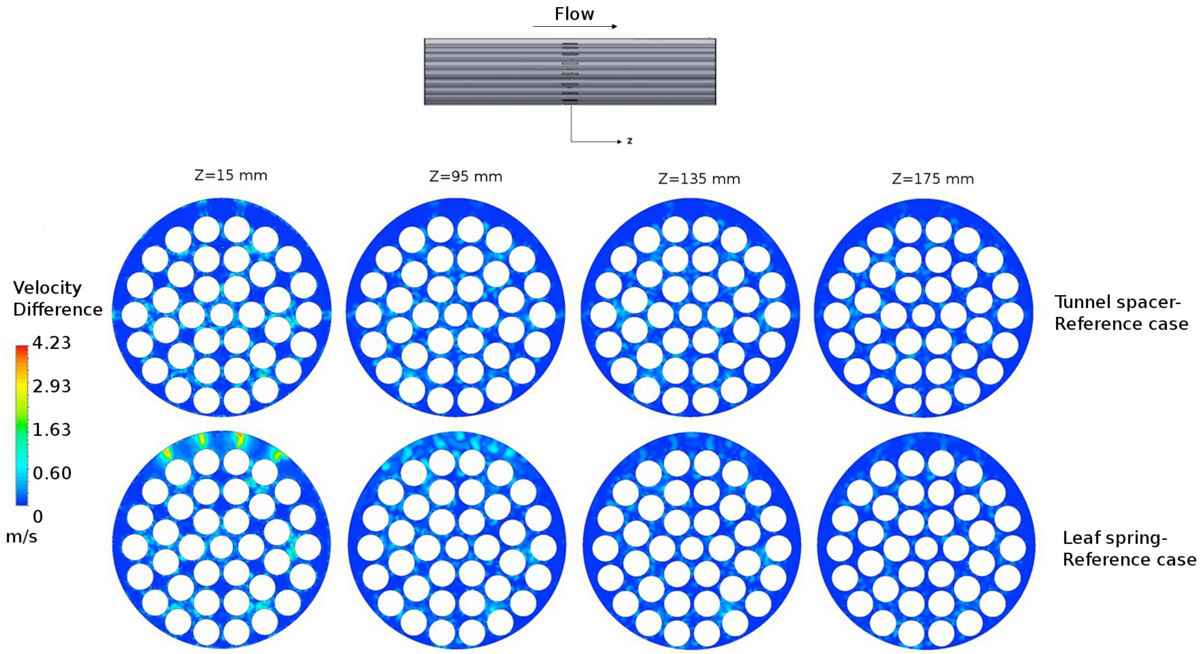

Figure 8 Velocity overlay contours; velocity difference downstream of the appendages

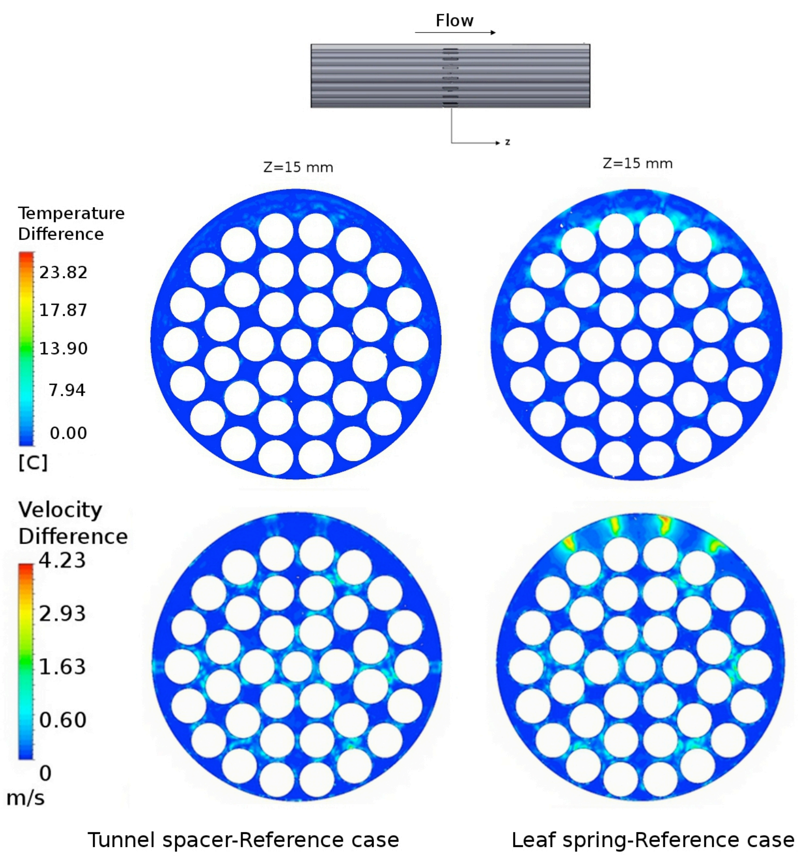

Figure 9 Velocity and temperature overlay contours downstream of the appendages

3. Conclusions

Turbulent flows inside the CANDU fuel bundles were simulated using CFD and the results were presented in this paper. A subchannel model was used to investigate the capabilities of four different turbulence models: k.ε, Reynolds Normalization Group (RNG), Shear Stress Transport (SST) and Large Eddy Simulation (LES). Both RNG and SST models are refined Reynolds Averaged Navier-Stokes (RANS) methods. It was found out that the k.ε model under-predicts the cross flow magnitudes and fails to capture smaller scale turbulence compared to the LES model. Moreover, the effects caused by the appendages disappeared at shorter distances downstream of the appendages in the k.ε model compared to the other three. The RNG and SST showed similar patterns. The RNG model prediction of cross flow magnitudes is closer to LES than k.ε, showing an improvement.

Since the solution time of the RNG model was reasonably lower than SST and LES, it was used for another simulation model for the CANDU modified 37-element bundle. In this model two cases were developed where two appendage types were used to position the bundle inside the pressure tube (normally used in CHF tests) and compared to a reference case where no positioning appendages are used (reactors). It was found out that none of the appendage designs (i.e. tunnel spacer and leaf spring) has any dominant effects on the flow field. That is likely due to their small size and low quantity compared to the hydraulic diameter. The local effects caused by the leaf springs are more pronounced compared to the tunnel spacers. These findings are particularly important to show how the experimental assembly represents the reactor case.

4. References

1 D. S. Rowe, B. M. Johnson and J. G. Kundsen, "Implications Concerning Rod Bundle Cross Flow Mixing Based on Measurements of Turbulent Flow Structure", Int. J. Heat Mass Transfer, Vol. 17, 1974, pp. 407–419.

2 S. V. Moller, "On Phenomena of Turbulent Flow Through Rod Bundles", Exp. Therm. Fluid Sci., Vol. 4, 1991, pp. 25–35.

3 F. Abbasian, S. D. Yu and J. Cao, "Experimental and Numerical Investigations of Three-Dimensional Turbulent Flow of Water Surrounding a CANDU Simulation Fuel Bundle Structure inside a Channel", Journal of Nuclear Engineering and Design, Vol. 239, No. 11, 2009, pp. 2224-2235.

4 F. Abbasian, G. I. Hadaller and R. A. Fortman, "CFD Simulations of the Single-Phase and Two-Phase Coolant Flow of Water inside the Original and Modified CANDU 37-Element Bundles", 11th International Conference on CANDU Fuel, Niagara Falls, Ontario, Canada, 2010 October 17-20.

5 V. Yakhot and S. A. Orszag, "Renormalization Group Analysis of Turbulence: I. Basic Theory", Journal of Scientific Computing, Vol. 1, No. 1, 1986, pp.1-51.

6 F. R. Menter, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications", AIAA Journal*,* Vol. 32, No.8, 1994, pp.1598-1605.