EXPERIMENTAL AND COMPUTATIONAL THERMALHYDRAULICS RESEARCH RELATED TO CANDU REACTOR OPERATION AND SAFETY

S. Tavoularis, D. Chang, E. Rind, G. Choueiri, Y. Liu, H. Shaban and E. Lessard
Department of Mechanical Engineering, University of Ottawa, Ontario, Canada

Abstract

This paper describes recent, ongoing and planned research projects at the University of Ottawa, whose objective is to enhance our knowledge of flow and heat transfer in CANDU rod bundles and header/feeder systems and to assist the Canadian nuclear industry in the analysis of operation and safety of CANDU components as well as in designing improved ones. Several experimental facilities are being developed, including a refractive-index matching flow loop for detailed measurements of flows in eccentric annuli and rod bundles, a large-scale, heated rod-bundle facility with air as medium, matching the Reynolds number of the CANDU core and suitable for the study of the effects of geometrical distortions (e.g., pressure tube creep, spacers and fuel element bow) and transients, and an air-water loop for the testing of the operation of wire-mesh sensors and the study of two-phase flows in simple header/feeder vessels. Extensive CFD work on similar topics is also been conducted in parallel with the experiments using the experimental results for its validation.

1. Introduction

Following decades without construction of new nuclear power plants in North America and Western Europe, recent trends have been pointing to a forthcoming flurry of nuclear power station construction, which the World Nuclear Association (WNA) has characteristically termed as “nuclear renaissance”. Nuclear power has been recognized as the only large-scale, continuous and reliable source of electricity that is capable of meeting the increasing demand in both industrialized and developing countries. Moreover, the political and economic leaders, as well as the general public, are becoming increasingly aware that, unlike fossil fuel plants, nuclear power generation does not release greenhouse gases into the atmosphere, thus not contributing to global warming. WNA estimates that 30 reactors are currently being built worldwide, an additional 90 reactors are planned for the next decade and 200 more are anticipated at a later time. Within Canada, serious discussions are in progress about building new power plants. In contrast to earlier designs of nuclear reactors, which were mainly the results of national efforts, future reactors are likely to be the product of international collaboration, as, for example, evidenced by the establishment of the Generation IV International Forum. Canada has committed itself to participating in and contributing to such programs. Moreover, considering that 15% of the total electric power in Canada and about half of that in Ontario are presently generated by nuclear plants, there is a continuing and ever increasing need for innovative ideas and coordinated efforts towards maintaining and updating the existing plants, thus extending their productive lifetime at a tremendous economic benefit. Universities in Canada have embraced the responsibility to participate actively in these efforts of the Canadian nuclear industry, directing the talents and skills of academic researchers towards resolving present and future challenges of the industry and, equally importantly, educating the engineers and scientists that will replace the current generation of nuclear workers, which is rapidly being depleted through retirement. As most universities in Canada, USA and elsewhere eliminated or drastically reduced their educational and research programs in Nuclear Engineering, graduates with this type of expertise are needed urgently.

Building on previous efforts, a new and extensive nuclear thermalhydraulics research program has been undertaken recently at the University of Ottawa. This program is aimed at producing original measurements and numerical simulations that will enhance the information and tools available to nuclear engineers in conceptualizing, optimizing, and analyzing the operation and safety of current and future nuclear reactors. It also provides hands-on education and training in nuclear reactor thermalhydraulics to several graduate students, postdoctoral fellows and other research personnel.

Canada has in the past played a leading role in the peaceful application of nuclear energy, with current CANDU reactors producing much of the country's electric power, whereas similar designs are used in several other countries. In recent years, however, the number of countries with advanced nuclear reactor design and manufacturing capabilities has increased. Besides USA, Russia, EU and Japan, South Korea and China have developed competitive nuclear reactor industries. The Canadian nuclear industry finds it increasingly harder to maintain its international status and to sign contracts for CANDU reactors with domestic and international customers. To satisfy the increasing power demand and maintain international competitiveness, it is imperative for future CANDU reactors that their efficiency be increased and their complexity and size be reduced, while also reducing the need for nuclear waste disposal. It is anticipated that the proposed work will contribute measurably to this goal.

The following sections outline some of the main experimental and computational research projects currently in progress at the University of Ottawa. In addition to these, a research program in support of the Canadian National Program on Generation IV Energy Technologies for the development of a Super Critical Water-cooled Reactor (SCWR) has also been undertaken. Progress in this research has been summarized elsewhere [1,2]. A step towards establishing a consensus of the nuclear reactor research community concerning the well-known large-scale vortex patterns in tightly packed rod bundle flows was made by the introduction of appropriate terminology for such patterns and guidelines for their accurate numerical simulation [3].

2. Flows in Rod Bundles

2.1 Research Objectives

The general goal of this research is to enhance our understanding and predictive capabilities of the complex coolant flow and heat transfer phenomena in the fuel channels ("rod bundles") that comprise the cores of the CANDU and other pressurized water nuclear reactors. Specific objectives of this research are i) validation of subchannel-code predictions of flow and enthalpy distribution in isothermal and heated multi-rod bundles for single-phase flows; ii) study of the effects of fuel bundle geometry changes on flow distribution and local heat transfer; iii) study of the effects of transients on flow distribution and heat transfer in rod bundles; and iv) analysis of two-phase flows using computational fluid dynamics. So far, work has focussed upon the understanding of flow structure and heat transfer in rod bundles, in refining previous computational methods for rod bundle flows and in establishing two unique and versatile experimental facilities for measurements in rod bundles.

2.2 Computational Work

A thorough numerical study of developing, isothermal, single-phase flow in a rectangular channel with a cylindrical core (see Figure 1) was recently completed [4]. This work has compared systematically results obtained using several numerical methods and boundary conditions and has assessed the capabilities, uncertainties and limitations of each approach so that it can serve as a guide for future work. A summary of the examined cases is given in Table 1. Simulations of developing flow were conducted using URANS (unsteady solutions of Reynolds-averaged Navier-Stokes equations) with different turbulence models, including RSM-ε (Reynolds stress model by Launder and Shima) and RSM-ω (Reynolds stress model by Wilcox); in addition, scale-adaptive simulations (SAS), improved delayed detached eddy simulations (IDDES), and segregated (or zonal) hybrid simulations, employing LES (large-eddy simulations) in a downstream sub-domain of the channel, were also carried out. Two sets of boundary conditions were used for the inlet velocity: uniform velocity and fully-developed mean velocity variation. The results have contributed significantly to our understanding of the accuracy of various CFD methods for isothermal flows in rod bundles. It was shown that simulations with the assumption of axial periodicity to reduce the size of the computational domain produced inferior predictions compared to developing flow simulations. It was also demonstrated that the use of fully-developed inlet velocity variation makes inferior predictions compared to the use of uniform inlet velocity, although the former shortens significantly the length of generation of the vortex street. Among the different turbulence modelling methods, the LES matched most closely the experimental results, particularly the characteristics of the gap vortex street as well as the time-averaged axial velocity and turbulent kinetic energy in the gap region. The LES analysis was the only one that predicted some fine turbulence structure embedded in the large-scale oscillations. Unsteady inviscid (Euler) simulations with a developed inlet velocity distribution were also conducted. These also predicted the onset of gap instability, which proves that this is an inviscid flow mechanism, associated with the azimuthally inflected velocity distribution. The predictions of this method differed very significantly from the measurements, and so unsteady Euler flow analysis is not recommended for rod bundle flows. Although the LES method showed a superior accuracy in predicting the characteristics of the gap vortex street and the time-averaged flow field in the present simplified rod-bundle flow, URANS with RSM, when properly applied, also had a fair accuracy indicating that URANS are still an acceptable choice for rod bundle CFD analysis. One should be encouraged to use LES instead, but only when it is possible to satisfy the well-established criteria that make LES what they are. The use of the segregated hybrid model is helpful in this respect, as it reduces the minimum channel length required for the LES analysis. If the recent growth of computer capabilities continues, it is possible that LES will eventually become the method of choice for the nuclear industry, but this time has not yet arrived.

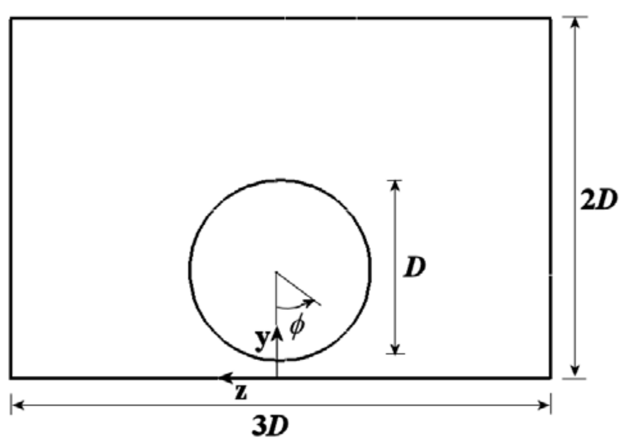


Figure 1. Sketch of the computational cross-section and definition of coordinates and dimensions.

Table 1. Summary of cases examined for isothermal flow in a simple rod-bundle-like channel

simulation type		inlet velocity		inlet turbulence intensity (%)	time-varying velocity algorithm	outlet boundary condition	domain length	convection scheme
		uniform	fully developed					
URANS with RSM-ε		√		3.0		pressure	108D	second-order upwind
URANS with RSM-ε			√	3.0		pressure	108D	second-order upwind
URANS with RSM-ε		√		10.0		pressure	108D	second-order upwind
URANS with RSM-ω		√		3.0		pressure	108D	second-order upwind
SAS		√		3.0	SSM	pressure	108D	bounded central-difference
IDDES		√		3.0	SSM	pressure	108D	bounded central-difference
segregated hybrid	SAS (up-stream)	√		3.0	SEM	interface	45D	bounded central-difference
	LES (down-stream)	interface			SEM	outflow	21D	bounded central-difference
Euler		√				pressure	108D	second-order upwind

2.3 Experimental Work

A recent publication describes some flow visualisation studies of the gap instability phenomenon in annular channels [5]. A much more detailed quantitative study of flows in annular channels and rod bundles is currently in progress in the newly constructed flow loop shown in Figure 2. Four test sections will be used, including three eccentric annular channels with diameter ratios 0.25, 0.5 and 0.75 and a simplified rod bundle (Figure 3). The fluid is a solution of ammonium thiocyanate (NH_4SCN), which matches the refractive index of the acrylic materials used for the test section, thus making flow visualisation and optical flow measurement techniques (LDV and PIV) possible. Each test section is enclosed within a circular channel with a square outer shape, 60″ long and with a diameter of 2″.

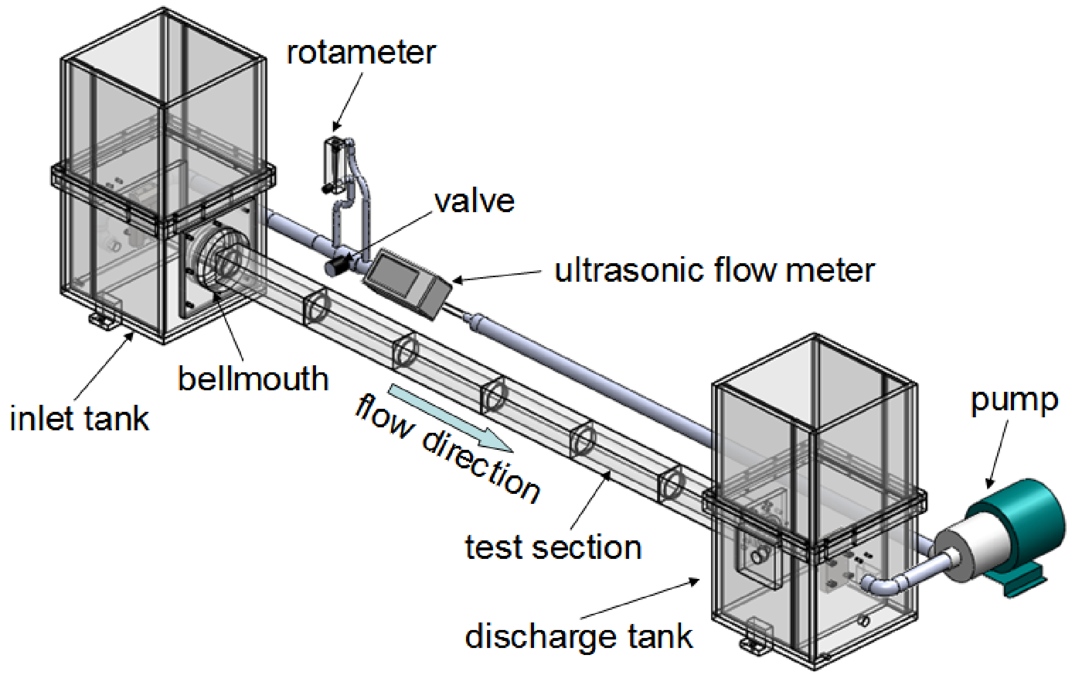


Figure 2. Sketch of the refractive index matching flow loop.

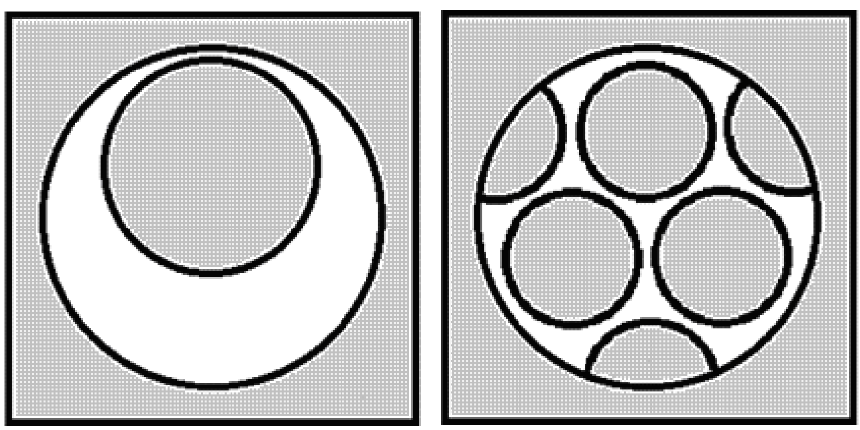


Figure 3. Cross sections of an eccentric annular test section and a simplified rod bundle.

Some preliminary LDV measurements have been conducted for the annulus with a diameter ratio of 0.50 for specific ranges of Reynolds numbers Re (based on the bulk velocity and the hydraulic diameter) and eccentricity $e = 1 - 2\delta/(D - d)$ (δ is the narrow gap width, and D and d are the diameters of the outer and inner cylinders, respectively). Axial and cross flow components of the velocity were measured at a streamwise position that was away from the inlet by approximately two thirds of the test section length and midway in the narrow gap. Spectral analysis was performed on the cross flow component of the velocity. The presence of strong cross-flow oscillations across the gap when the Reynolds number exceeded a critical value was clearly demonstrated by the existence of a well defined spectral peak at a characteristic dimensionless frequency (Strouhal number). For a fixed eccentricity, the Strouhal number increased with Reynolds number increasing within the relatively narrow range considered (Figure 4); it is expected that at some higher value Re and beyond, the Strouhal number will approach a constant asymptote.

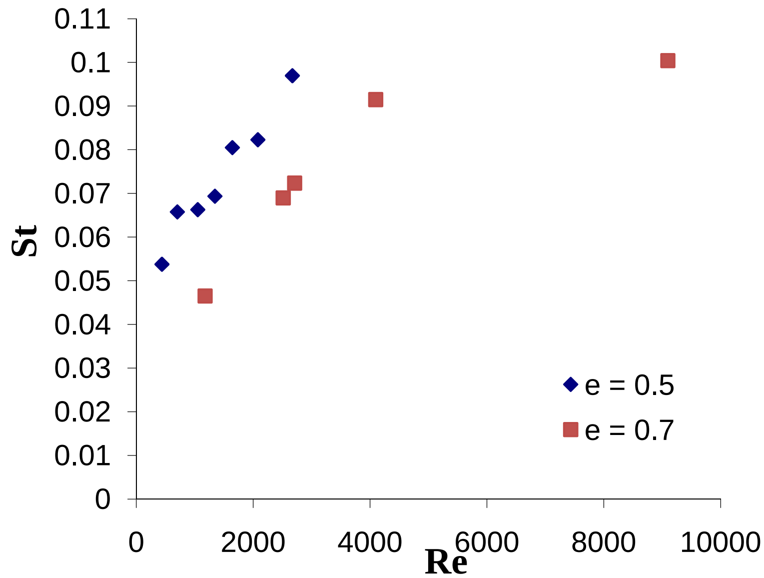


Figure 4. Strouhal number of gap vortex street vs. Reynolds number for d / D = 0.5 .

Additional pilot measurements were performed by keeping Re approximately equal to 2670 and varying the eccentricity. For $e < 0.4$, no obvious spectral peak was observed, which suggests that for these cases this Reynolds number was subcritical. This is demonstrated in Figure 5, which shows that the standard deviation of the cross-flow velocity fluctuations was relatively small for $e < 0.4$ but increased markedly with increasing e for $0.4 < e < 0.7$.

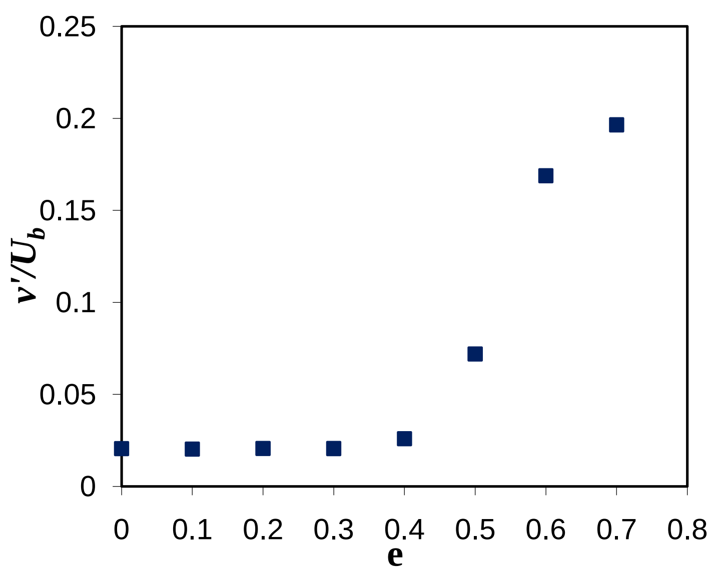


Figure 5. Standard deviation of the cross flow velocity normalized by the bulk fluid velocity versus eccentricity for a Reynolds number of 2670.

LDV measurements are in progress for wider ranges of Re and e to establish the dependence of the critical Reynolds number for instability on eccentricity and diameter ratio. Similar measurements will be taken to establish the conditions for transition to turbulence. In addition, measurements will be taken with planar PIV and stereo PIV to document the instantaneous flow structure. In addition to their contribution to our understanding of flow phenomena in rod-bundle-like systems, these experiments will be valuable for the validation of CFD simulations, particularly because they will report details that are essential for simulations, including inlet velocity and turbulence distributions.

A second experimental facility to be used in this project is a versatile rod bundle channel, which is a large-scale (12.9:1) model of a 60 deg sector of the CANDU 37-rod bundle, including an inlet endplate. A sketch of the facility is shown in Figure 6, where (1) is an air filter, used for removing suspended particles from the air, and an inlet contraction, used for the measurement of flow rate, (2) is a large blower, allowing the generation of flows with Reynolds numbers approximating those in the operating CANDU, (4) is a bypass duct with an automated flap (3) adjusting the bypass flow rate and thus allowing the generation of transient flows and flows at low Reynolds numbers, (5) is a plenum with a honeycomb and screens for flow quality management and upstream turbulence measurement, and (6) is the test section. A view of the test section is given in Figure 7; it contains six entire tubes and an one-sixth segment of a tube, which are models of CANDU fuel elements and are suspended in a wedge-like duct. The central outer rod is covered by a thin foil, which can be heated electrically to permit the measurement of the local heat transfer coefficient distribution in the surrounding subchannels. The same rod can be traversed azimuthally and radially to generate distrorted geometries. The upper enclosure of the test section, not shown in this figure, is a curved plate, modelling the inner surface of the pressure tube. Different plates can be used at different radial positions to simulate rod bundles with expanded pressure tubes (e.g., due to creep). The test section inclined walls are transparent to allow flow visualisation and optical measurements. Probes may be inserted in the test section from the downstream open end as well as from the top cover and acrylic sections of the side walls. Hot wire anemometry will be used as the main flow measurement technique because of its superior spatial and temporal resolutions by comparison to those of available non-intrusive methods. A limited number of PIV (particle image velocimetry) measurements has also been planned. The design of this facility and the selection of its instrumentations and controls have been completed. Construction is well in progress, all necessary instruments and components have been purchased, and the facility is expected to be commissioned during the summer 2011.

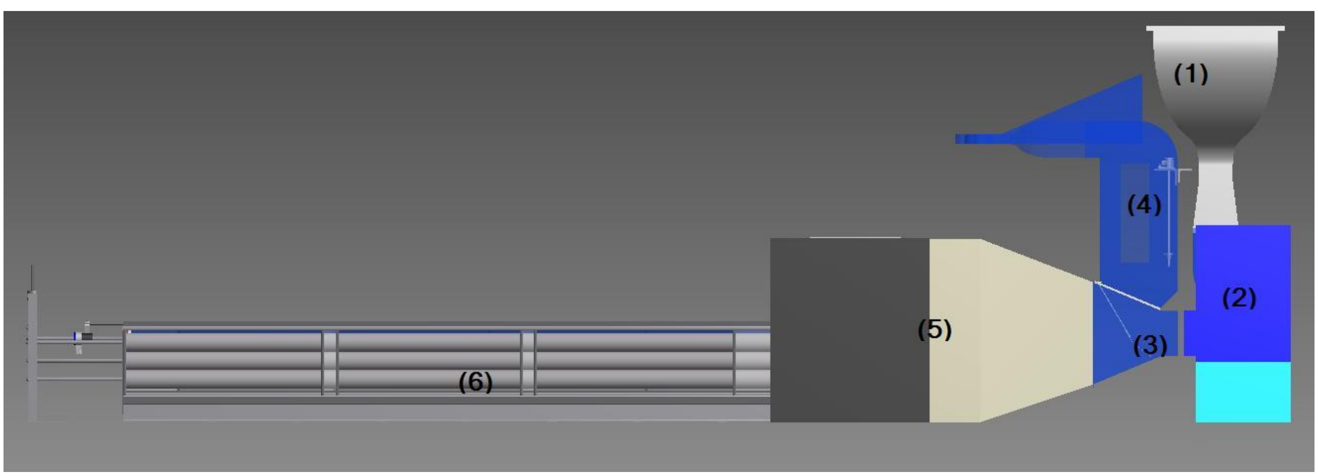


Figure 6. Large-scale rod bundle facility.

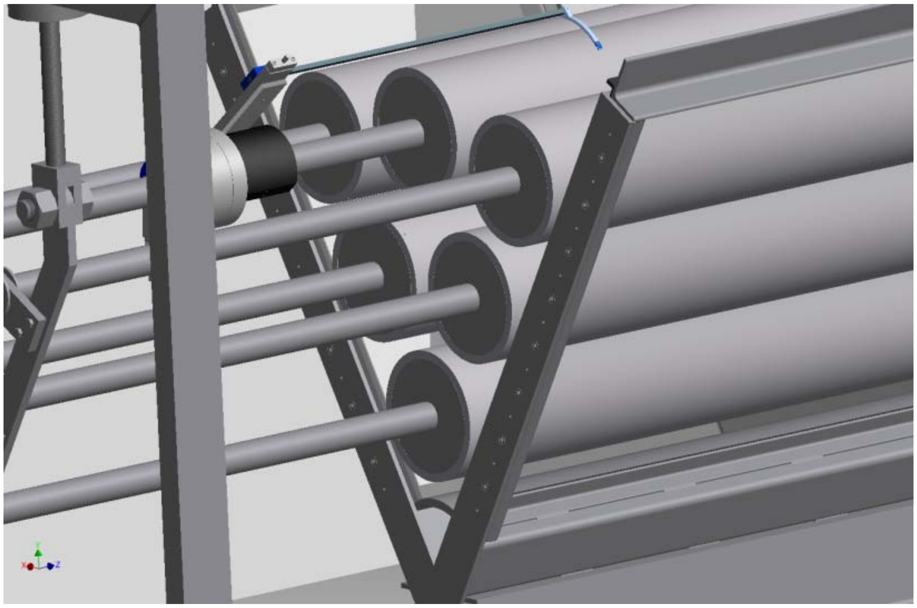


Figure 7. View of the test section from its downstream end.

3. Flows in Headers and Feeders

3.1 Background and Research Objectives

The complexities of two-phase flow arise from the highly irregular local flow structure, the existence of the interface between the two phases and the mass, momentum and heat transfer across the interface. For nuclear reactor design and safety analysis, reliable and accurate predictions of the two-phase flow parameters and interfacial structures are of great importance. Depending on the mass fluxes of the liquid and gas phases and on the geometric configuration, different flow regimes may be encountered. The most common flow regimes are bubbly flow, annular flow and slug flow. In bubbly flow, the gas or vapour phase is distributed as discrete bubbles in the continuous liquid phase. At the two extreme cases, the bubbles may be small and spherical or large with a spherical cap and a flat tail. In non-heated channel flows, once the superficial velocity of the gas exceeds a certain value, the liquid flows in a film near the wall and the gas flows as a continuous stream in the core. Large amplitude coherent waves are usually present on the surface of the film and the continuous break up of these waves forms a source for droplet entrainment which occurs in varying amounts in the central gas core. Although this topic is of great importance for thermal power plants and several other engineering systems, experimental studies in relevant configurations have appeared in the open literature only during the past decade. A comparison of the results of these studies amply demonstrates that flow characteristics in such complex systems depend strongly on the geometry and the operating conditions, which makes it impossible to draw general conclusions that can be extended from one case to another. The few published computational studies have demonstrated amply the difficulties of this approach. A goal of the present work is to determine the degree by which available computational codes are able to simulate air-water flows with sufficient engineering accuracy.

During loss of coolant accidents (LOCA), two-phase flow could occur in the CANDU headers and the distribution of the two phases is found to be uneven in the feeder pipes. This could mean that some fuel channels would receive little or no cooling, which may lead to fuel meltdown. The objective of this research is to evaluate the use of available experimental and computational fluid dynamics (CFD) methods for the study of two-phase flow characteristics in pipes and header-feeder systems relevant to CANDU nuclear power plants. The main experimental tasks are: i) to qualify and calibrate a wire-mesh sensor (WMS) for two-phase flow measurements by performing evaluation tests in adiabatic air-water pipe flow, and if possible also in steam-water pipe flow; ii) to quantify the measurement uncertainty of the wire-mesh sensor for measuring void fraction, interfacial area and gas velocity; and iii) to compile an experimental database of air-water flow characteristics in horizontal and vertical (upward and downward flow) pipes using the wire-mesh sensor. The main tasks of the computational analysis are: i) to perform CFD simulations of two-phase flows at the same inlet flow conditions and in the same geometries as those of the experiments and to compare the simulation results to the measurements to verify the applicability of CFD as a tool for two-phase flow simulation at the conditions and geometries of interest to nuclear safety analysis; ii) to perform CFD simulations of two-phase flows in a small-scale multi-bank header for which previous experimental results are available.

3.2 Experimental Work

The wire-mesh sensors will be tested in an entirely new, specially designed, recirculating water-air loop, which will be dedicated to this project. A schematic diagram of the flow loop is shown in Figure 8. Water contained within the main tank is pumped to the head tank, providing a head of approximately 4.5 m to the test section. The head tank is divided into two parts, separated by a dividing wall which acts as an overflow and maintains a constant head to the test section, whereas excess water is redirected back to the main tank. The maximum water flow velocity in the test section is estimated to be approximately 9 m/s. Water from the head tank is first supplied to the settling tank from which it enters the test section following mixing with compressed air injected just downstream of the test section inlet. The water and air flow rates will be measured by calibrated flowmeters and controlled separately to allow the generation of different two-phase flow regimes, including bubbly, slug and annular flows as well as transitional regimes. Figure 9 illustrates a preliminary design of the test section for the air-water pipe flow. This section is positioned downstream of a long pipe (4), which allows full development of the two-phase flow. The central piece contains two WMS (2) mounted on a special flange-spacer combination (3). On either side of this assembly, there are two short pipe sections (1) with thin walls made of fluorinated ethylene propylene (FEP, known under the trade name Teflon), having a refractive index that closely matches that of water, thus allowing undistorted optical access for LDV and PIV measurements. These sections are immersed in square viewing tanks with glass walls and containing still water.

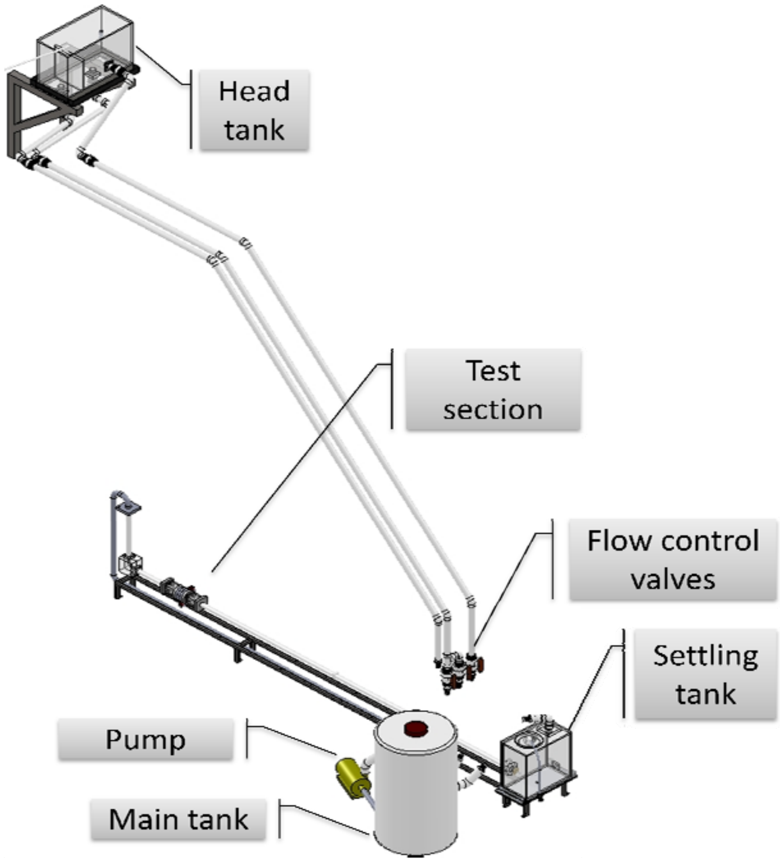


Figure 8. Schematic diagram of the water-air recirculating loop.

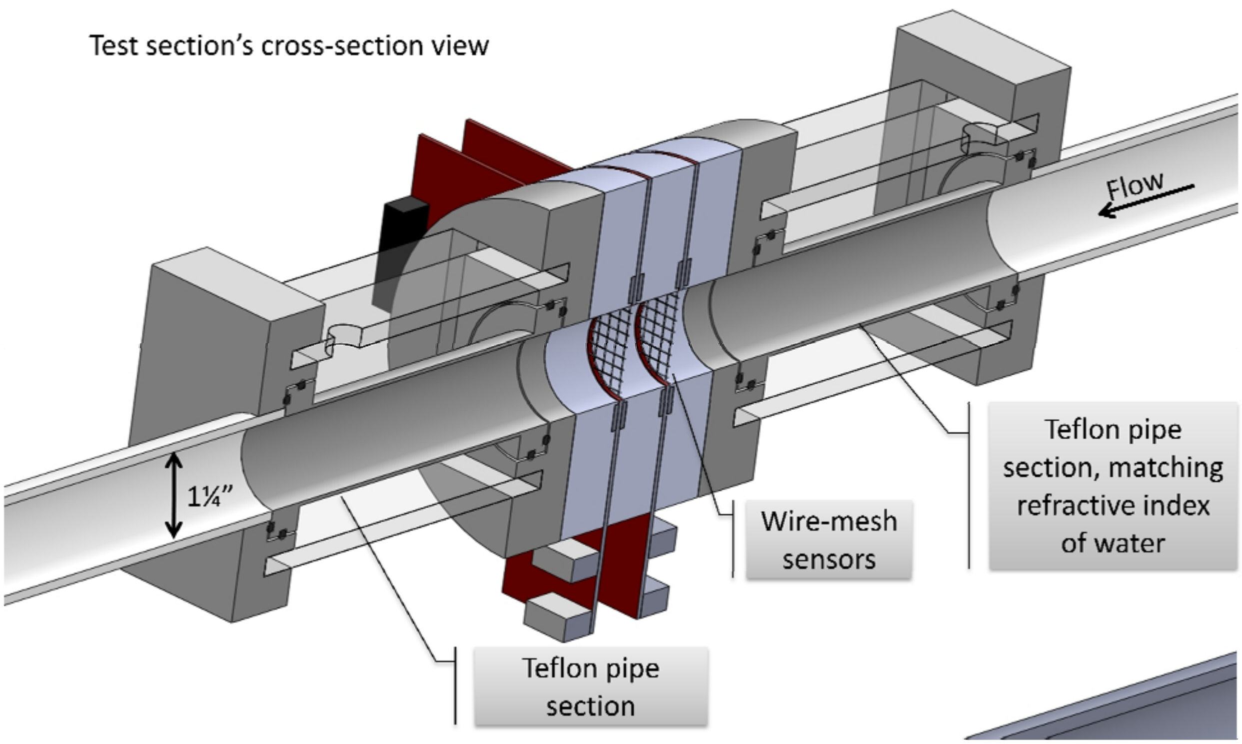


Figure 9. A cross-section of the test section with a double WMS.

Two 8 × 8 WMS (Figure 10) have been made available to this project, together with two matching acrylic flanges and an acrylic spacer ring. We are currently designing the adaptors required for the installation of the WMS in our test section. At present, effort is focussed at finalizing the assembly of the loop and designing and fabricating the remaining few components for the connection of the WMS.

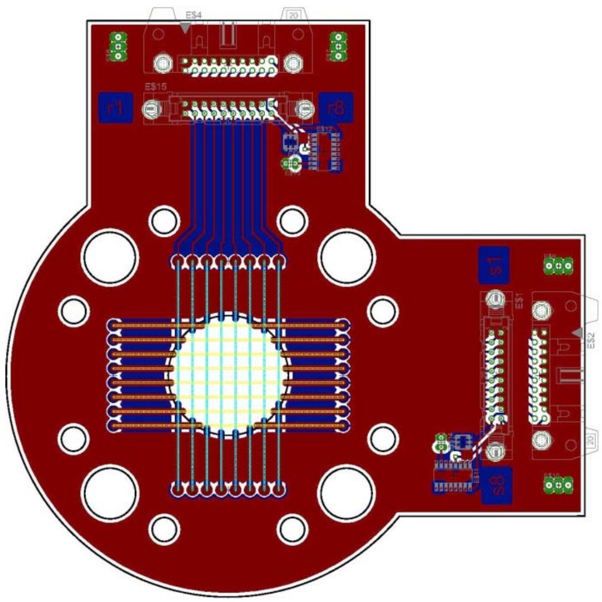


Figure 10. WMS.

3.3 Computational Work

This work has so far consisted of an extensive literature review of experimental and computational papers on liquid-gas flows in pipes, a selection of a few previous experiments which are suitable for the validation of computational methods, and a comparative evaluation of available CFD codes and methods. Most multiphase CFD codes are extensions of single-phase codes by the incorporation of models of inter-phase dynamics. Because multiphase flows are inherently unsteady, they require the use of time-consuming and computationally expensive time-dependent methods. To avoid possible solution instability, it is necessary to use small time steps, which requires long simulation times. The governing equations for fluid-fluid flow are the basis for most multiphase CFD codes. The most widely used multiphase models can be subdivided into three categories: Eulerian, Lagrangian and volume of fluid models. Eulerian models treat the dispersed phase(s) as a continuous fluid and solve for the average local volume fraction, and average velocity. A simplification of this model, often called the homogeneous or mixture model, solves for a single velocity field for both phases but may allow a small difference in their velocities by solving an additional slip velocity equation. Lagrangian models track a certain number of particles and solve for their velocity fields. Due to their high computational cost, they are limited by the number of particles they can track efficiently. Volume of fluid models are especially important in separated flows when the location and evolution of the interface are of interest. They solve a single set of equations for the fluids and an additional one for the interface. Among the many available multiphase CFD codes available, we have chosen to evaluate two commercial codes, CFX 12.1 and FLUENT 12.1, for which we maintain academic licences, and the open source code OpenFOAM 1.7.1. We have not yet finalized our conclusions and choices, as the work is in progress. The following paragraphs outline the work conducted so far and some lessons learned.

The first step of this work was to evaluate the capabilities of the three chosen CFD packages for simulating isothermal bubbly and slug flows in vertical pipes. The simulations were performed by simultaneously solving the conservation equations, as well as the turbulent kinetic energy and turbulent kinetic energy dissipation rate equations. The challenge in solving these equations for the described problem was to model the interphase momentum transfer mechanisms (interfacial forces). The interfacial forces considered in the present study include the drag force, the lift force, the wall lubrication force, and the turbulent dispersion force. The simulations were validated against the experimental results of Hibiki et al. [6]. The computational geometry was a 3D model of a vertical round pipe with a diameter of 50.8 mm and a length of 3061 mm. Superficial velocities of the gas and liquid phases at the inlet were set as 0.0556 and 0.491 m/s respectively, and the inlet bubble diameter was set as 2.5 mm. The models used in the different codes are summarised in Table 2. A comparison of the methodology and predictions of the three CFD packages for the bubbly flow simulations is summarized in Table 3.

Table 2. Models used in the CFD simulations of bubbly flow in a vertical pipe

CFD code	Multiphase model	Turbulence model
CFX12.1	Multiple Size Group (MUSIG) model	$k-\omega$ SST (Shear Stress Transport)
FLUENT12.1	Eulerian model with Interfacial Area Transport Equation (IATE)	$k-\varepsilon$
OpenFOAM1.7.1	twoPhaseEulerFoam (Eulerian model)	$k-\varepsilon$

Table 3 Methodology and results of CFD simulations of bubbly flow in a vertical pipe

			CFX 12.1	Fluent 12.1	OpenFoam 1.7.1
Interfacial forces	drag		5 implemented	2 implemented/ user-defined	5 implemented/ user-defined
	lift		3 implemented	user defined	user defined
	wall lubrication		3 implemented	user defined	user defined
	turbulent dispersion		2 implemented	user defined	user defined
Setting up simulation			easy	medium	hard
Convergence			quick	slow	quick
Implementation of additional models			not so convenient	fairly convenient	very flexible, but not very convenient
Post-processing			most convenient	convenient	convenient
Cost			licence fee	licence fee	free
Availability	at local clusters		yes	yes	yes
	at HPCVL		no	yes	yes
Results of tests	Void fraction	Z=6D	good	worse	best
		Z=53.5D	best	worse	good
	SMD	Z=6D	best	good	N/A
		Z=53.5D	best	good	N/A
	IAC	Z=6D	good	good	best
		Z=53.5D	best	good	worse
	Velocity of gas	Z=6D	good	good	best
		Z=53.5D	good	good	worse
	Velocity of liquid	Z=6D	good	good	good
		Z=53.5D	best	good	worse

In the bubbly flow regime, void fraction is one of the most important flow variables. Figure 11 compares the present simulation results of void fraction at two different axial locations, z/D = 6 and 53.5, with the measurements of Hibiki et al. [6] and two previous simulations, one using the MUSIG model and ABND approach (Cheung et al., [7, 8]) and a second one using the FLUENT-IATE approach (Sari et al. [9]). It has to be mentioned here that the label "Present OpenFOAM_Euler" represents the present simulation results obtained using the original twoPhaseEulerFoam without any modifications, whereas the label "Present OpenFOAM_Euler2" represents the results obtained using the original twoPhaseEulerFoam with user-defined interfacial forces models. In this case, the "wall peak" behaviour at z/D = 6 was successfully captured by all the codes. The best agreement between the predicted and measured value of void fraction is obtained by the OpenFOAM_Euler approach, whereas at z/D = 53.5, the OpenFOAM_Euler2 approach over-predicted the value of wall-peak. All other simulation methods under-predicted the value of wall-peak at this axial location. It can also be seen that the two OpenFOAM_Euler approaches show a dip in void fraction after the peak for both axial locations.

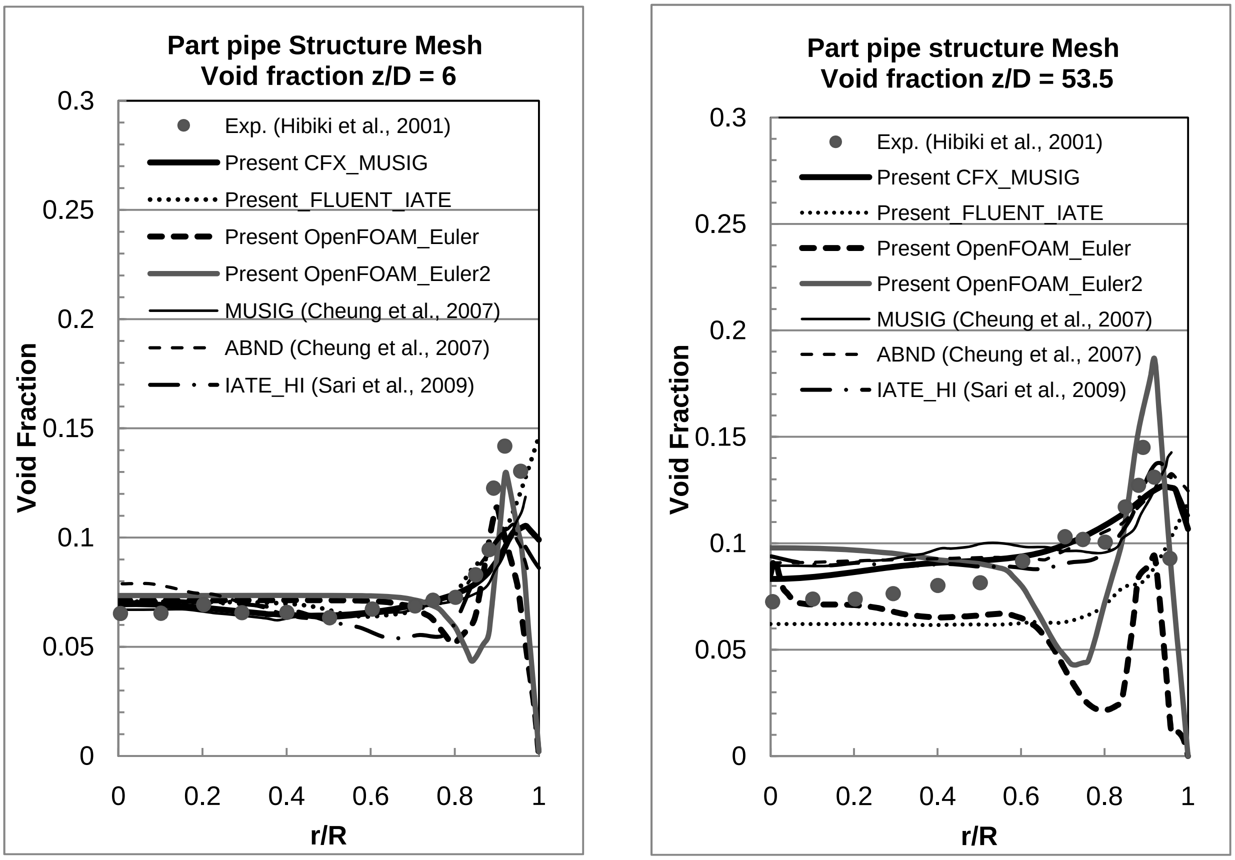


Figure 11: Comparison of present simulation results of void fraction with measurements by Hibiki et al. (2001) and previous simulation results.

Figure 12 compares the present predictions of the Sauter mean bubble diameter (SMD) with the measurements of Hibiki et al. and previous simulation results by Cheung et al. and Sari et al. at two different axial locations (*z/D* = 6 and 53.5). As reported by Hibiki et al., the SMD radial profiles were almost uniform and only showed a little increase in the vicinity of the wall. At *z/D* = 6, predictions of almost all models show fair agreement with the measurements, with the exception of the ABND of Cheung et al., which under-predicted the SMD. At *z/D* = 53.5, simulations using MUSIG model obtained overall better results than the other models, which under-predicted SMD. The better prediction ability of MUSIG (multiple size groups) model lies in that it can accurately capture the population balance of bubbles and get more accurate resolution of the dynamic changes of bubble size distribution.

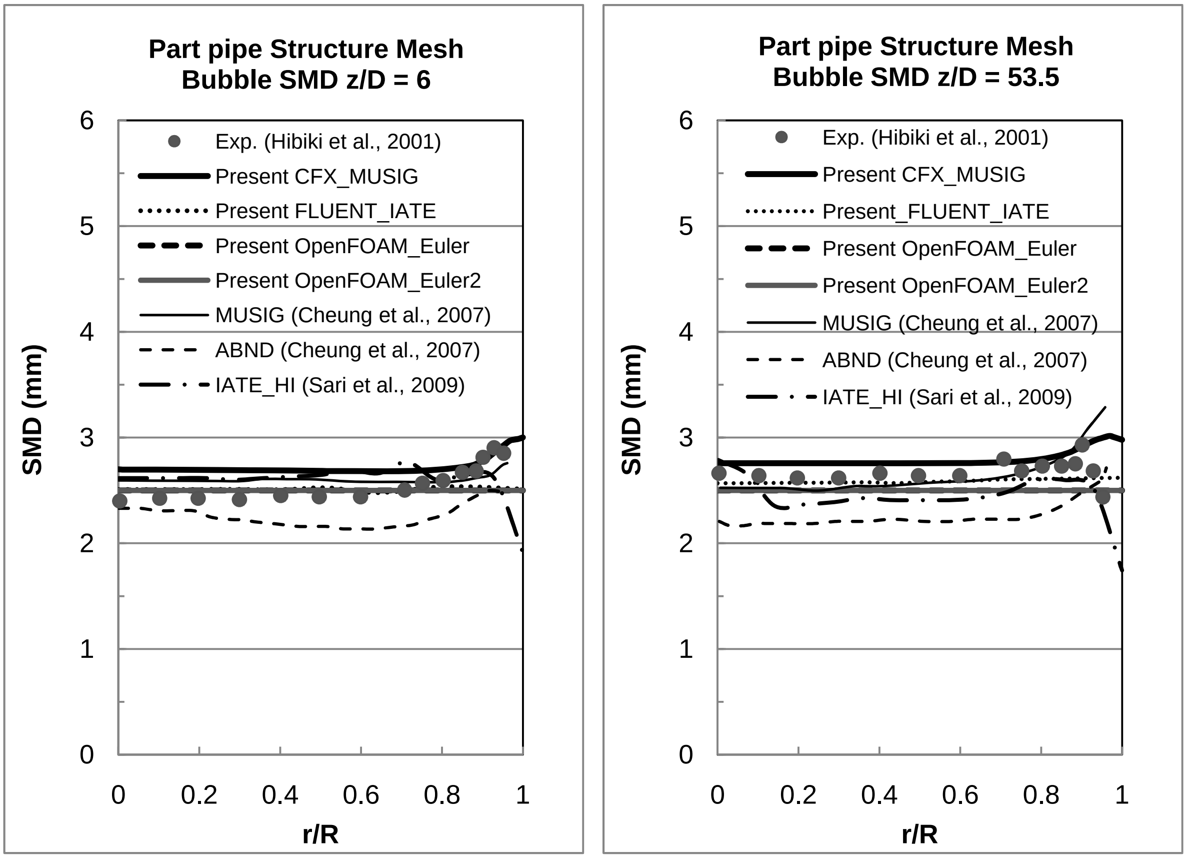
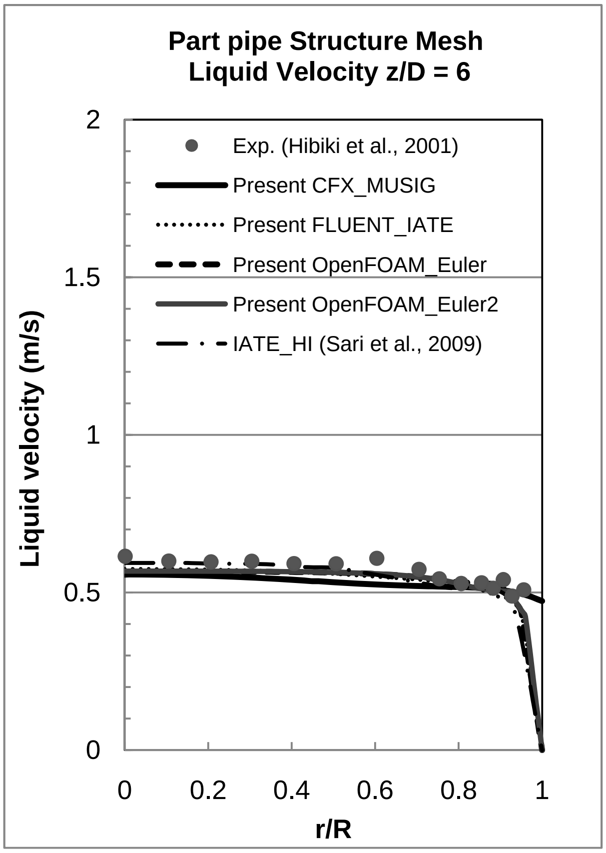


Figure 12: Comparison of present simulation results of SMD of bubbles with measurements by Hibiki et al. (2001) and previous simulation results.

The experimental data and present and previous simulation results of liquid phase velocity are illustrated in Figure 13. At z/D = 6, the velocity of the liquid phase was well predicted by all approaches, whereas, at z/D = 53.5, the measured data shows a core peak of liquid velocity which could not be captured by any of the methods.



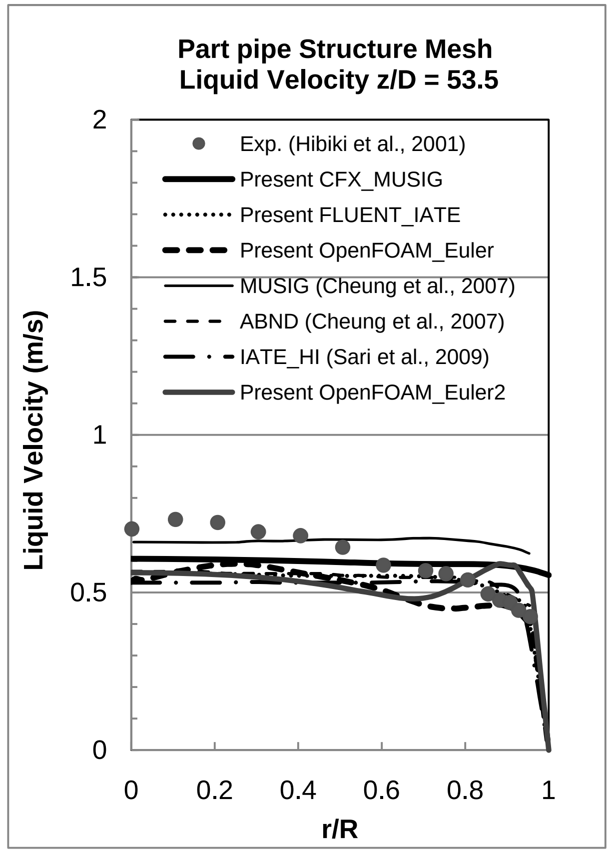


Figure 13: Comparison of present simulation results of liquid phase velocity with measurements by Hibiki et al. (2001) and previous simulation results.

Some general conclusions of these and similar tests are as follows.

CFX 12.1 has embedded the MUSIG model and several default models of interfacial forces. Thus, it is very convenient to set up the CFX_MUSIG simulation. Also, it is not difficult to obtain converged results and to post-process them. Although some discrepancies were found between predictions and measurements, the predictions of void fraction, SMD (Sauter mean diameter) of bubbles, and air and liquid velocities by the CFX-MUSIG model were generally fairly good. Another advantage noted by previous researches is that this approach can be conveniently extended to account for heat transfer. A limitation of the CFX code is the difficulty in implementing equations of IAC (interfacial area concentration) and other interfacial force models.

FLUENT 12.1 has two default models for drag force, but no default model for other interfacial forces, which must be implemented as source terms in the momentum equation by users through UDF (user-defined-functions). There are several options in FLUENT 12.1 to control the convergence of the simulation, which make it in this respect more difficult to use than CFX. On the other hand, the use of UDF in FLUENT offers more flexibility in integrating new models into the multiphase flow simulation. The FLUENT_IATE approach has a special model for IAC, and the agreement between simulated IAC and measured IAC is acceptable (within ±15 %). However, the IATE approach does not model SMD, and the predicted void fraction profile is not very good.

OpenFOAM has the advantage over the two commercial CFD packages that is free and entails no incremental cost for parallel processing. Also, because it is open-source, it offers much more flexibility for the simulation of multiphase flows. However, the environment of OpenFOAM is very different from these of the commercial CFD codes, which introduces a steep learning curve for new users. Also, for the solver of multiphase simulations in OpenFOAM, such as twoPhaseEulerFoam, bubbleFoam, etc., there is no default model for interfacial forces except the drag force, which further complicates the setting up process. The simulation results using twoPhaseEulerFoam with user defined interfacial forces show good agreement with measured data for an axial location which is near the pipe inlet, especially for void fraction, but poor agreement near the outlet. Overall, OpenFOAM needs more work for its default multiphase solvers, but it is more flexible in accepting new models and equations.

In summary, for isothermal bubbly flow simulation, CFX is the best overall choice. If more models for IAC (such as IATE) and other interfacial forces need to be considered, FLUENT and OpenFOAM are better choices as they offer more flexibility than CFX does. If cost of licences for large local parallel processing clusters is of concern, OpenFOAM would be preferable.

Similarly, comparisons of the procedures and results of simulations of slug flow in pipes using the same three CFD codes led to the following conclusions.

CFX 12.1 is very convenient when the default MUSIG model is implemented. It can provide overall better predictions of important characteristics of slug flow (and for bubbly-to-slug transient regime), such as void fraction, bubble size and velocity of two phases.

The Eulerian model together with IATE in **FLUENT** and the twoPhaseEulerFoam in **OpenFOAM** failed to predict the void fraction distribution, although both predicted acceptable velocities of gas and liquid phases. Although previous researchers used widely the VOF model in FLUENT to simulate the Taylor bubble in slug flow, it was presently found that the VOF model cannot capture the “core peak” of void fraction. This may be attributed to discrepancies between the prescribed inlet and initial flow conditions and the real flow conditions. In summary, for slug flow and bubbly-to-slug transient flow, the CFX_MUSIG model is the most convenient method to solve the entire flow field. FLUENT and OpenFOAM provide higher flexibility in integrating new multiphase models and interfacial force models.

A second independent evaluation of multiphase CFD codes was conducted by comparing predictions with experimental results in horizontal piping systems. Its conclusions are summarised as follows.

FLUENT: The following cases have been run in FLUENT: flows in horizontal and vertical pipes and in a 90 degree bend. Overall, it was found that numerical problems make it hard to get a converged solution in FLUENT. The final results captured the volume fraction distribution adequately but there were significant differences from the experimental results, particularly near the wall. Experiments show that a liquid film persists near the wall in horizontal pipes and bends but FLUENT does not capture this film. This is probably because of the absence of certain interphase force models, especially the wall lubrication force which pushes bubbles away from the wall. Moreover, introduction of certain other models inevitably leads to divergence. Refining the mesh does not solve this problem but introduces other inaccuracies because the wall y-plus becomes smaller than 30 and the standard k- ε model is not valid in this range. Previous authors testing FLUENT in 90 degree bends found similar results.

CFX: The cases run in CFX are 2D horizontal and vertical channels as well as 3D horizontal pipes. CFX was found to be easier to use from a numerical point of view. Convergence problems were not as frequent as when using FLUENT. A possible reason is that CFX uses a coupled solver for velocity, pressure and volume fraction while FLUENT uses a segregated solver and requires additional coupling algorithms, which are unstable and require very small time-steps to converge (10^{-8} seconds for FLUENT vs. 10^{-5} seconds for CFX). CFX has all the necessary interphase force models and its results are in good agreement with experiments.

OpenFOAM: OpenFOAM offers many solvers for multiphase flow. So far, the results seem to be comparable to those of CFX and the code does not face a lot of convergence problems. OpenFOAM has two main advantages: it is very customizable and in much more detail than the commercial software and it requires no licence fee. On the other hand, it also faces two minor problems: it is not at all user-friendly and has little to no documentation on most of the multiphase solvers.

In conclusion, the two independent comparisons of the capabilities of CFX, FLUENT and OpenFOAM for air-water simulations in pipes agree that FLUENT is less suitable than the other two for isothermal bubbly flows, because it is missing most of the interphase models and also has convergence problems. However, as we have not so far examined all features of the three codes, particularly the VOM, we cannot yet eliminate the use of any of them. It is possible that our analysis will demonstrate that adopting different approaches for different sections of a header/feeder system may be preferable to the use of a single model for the entire system.

4. Conclusion

A team of researchers at the University of Ottawa has undertaken an extensive experimental and computational research program with the objective to enhance our knowledge of flow and heat transfer in CANDU rod bundles and header/feeder systems. The results of this work are expected to assist AECL and the Canadian nuclear industry in the analysis of operation and safety of CANDU components as well as in designing improved ones. Newly established experimental facilities include i) a refractive-index matching flow loop for detailed measurements of flows in eccentric annuli and rod bundles, ii) a large-scale, heated rod-bundle facility with air as medium, matching the Reynolds number of the CANDU core and suitable for the study of the effects of geometrical distortions (e.g., pressure tube creep, spacers and fuel element bow) and transients, and iii) an air-water loop for the testing of the operation of wire-mesh sensors and the study of two-phase flows in simple header/feeder vessels. Extensive CFD work on similar topics is also been conducted in parallel with the experiments using the experimental results for its validation. Two commercial CFD codes and one open source code are being evaluated.

Acknowledgement

This research has been supported financially by NSERC, AECL, UNENE, HPCVL and Sun Microsystems Canada.

References

[1] H. Zahlan, D.C. Groeneveld and S. Tavoularis, 2011. Derivation of a Look-up Table for Transcritical Heat Transfer, 14th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-14), Toronto, Ontario, Canada, September 25-29, 2011.

[2] Jeddi, L., K. Jiang, S. Tavoularis and D. Groeneveld, 2011. Preliminary Tests at the University of Ottawa Supercritical CO2 Heat Transfer Facility, 5th International Symposium on Superctitical Water-Cooled Reactors (ISSCWR-5), Vancouver, British Columbia, Canada, March 13-16, 2011.

[3] Tavoularis, S., 2011. Rod bundle vortex networks, gap vortex streets, and gap instability: a nomenclature and some comments on available methodologies. Nucl. Eng. Des. 241, 2624-2626.

[4] D. Chang and S. Tavoularis, 2011. Numerical Simulations of Developing Flow and Vortex Street in a Rectangular Channel with a Cylindrical Core, 14th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-14), Toronto, Ontario, Canada, September 25-29, 2011.

[5] Piot, E. and S. Tavoularis, 2011. Gap Instability of Laminar Flows in Eccentric Annular Channels, Nucl. Eng. Des., doi:10.1016/j.nucengdes.2010.08.025.

[6] Hibiki, T., Ishii, M., Xiao, Z., 2001. Axial interfacial area transport of vertical bubbly flows, Int. J. Heat Mass Transfer 44, 1869–1888.

[7] Cheung, S.C.P., Yeoh, G.H., Tu, J.Y., 2007a. On the modelling of population balance in isothermal vertical bubbly flows- Average bubble number density approach. Chemical Engineering and Processing 46, 742–756.

[8] Cheung, S.C.P., Yeoh, G.H., Tu, J.Y., 2007b. On the numerical study of isothermal vertical bubbly flow using two population balance approach. Chemical Engineering Science 62, 4659–4674.

[9] Sari, S., Ergun, S., Barik, M., Kocar, C., Sokmen, C.N., 2009. Modeling of isothermal bubbly flow with interfacial area transport equation and bubble number density approach. Ann. Nucl. Energy 36 (2), 222–232.