ANALYSES OF FLUID FLOW AND HEAT TRANSFER INSIDE CALANDRIA VESSEL OF CANDU-6 REACTOR USING CFD

SEON-OH YU, MANWOONG KIM* and HHO-JUNG KIM
Korea Institute of Nuclear Safety
19 Guseong-dong, Yuseong-gu, Daejeon 305-338, Korea
*Corresponding author. E-mail : mwkim@kins.re.kr

Received January 28, 2005
Accepted for Publication July 8, 2005

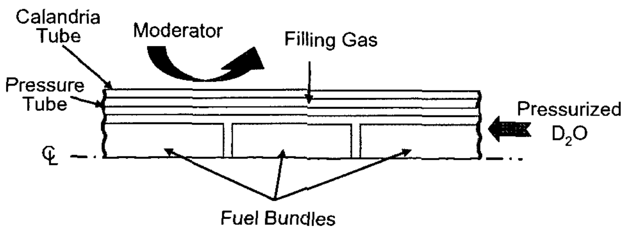
In a CANDU (CANada Deuterium Uranium) reactor, fuel channel integrity depends on the coolability of the moderator as an ultimate heat sink under transient conditions such as a loss of coolant accident (LOCA) with coincident loss of emergency core cooling (LOECC), as well as normal operating conditions. This study presents assessments of moderator thermal-hydraulic characteristics in the normal operating conditions and one transient condition for CANDU-6 reactors, using a general purpose three-dimensional computational fluid dynamics code. First, an optimized calculation scheme is obtained by many-sided comparisons of the predicted results with the related experimental data, and by evaluating the fluid flow and temperature distributions. Then, using the optimized scheme, analyses of real CANDU-6 in normal operating conditions and the transition condition have been performed. The present model successfully predicted the experimental results and also reasonably assessed the thermal-hydraulic characteristics of a real CANDU-6 with 380 fuel channels. A flow regime map with major parameters representing the flow pattern inside a calandria vessel has also proposed to be used as operational and/or regulatory guidelines.

KEYWORDS : CANDU-6 Reactor, Calandria Vessel, Moderator, CFD, Flow Regime Map

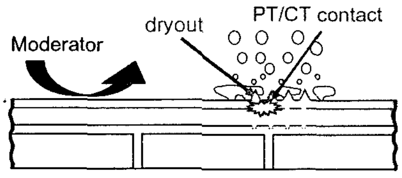
1. INTRODUCTION

Similar to a loss-of-coolant accident in a pressurized water-cooled reactor, LOCA in CANDU reactors can be precursors to fuel damage, which can have severe radiological consequences. However, the CANDU reactor has unique safety features with intrinsic safety related characteristics that distinguish it from other water-cooled thermal reactors. In particular, the heavy water moderator is continuously cooled, utilizing a heat sink for decay heat produced in the fuel if there is a LOCA with coincident failure of the emergency coolant injection system. Under such dual failure conditions, the hot pressure tube (PT) would deform and contact with the calandria tube (CT), providing an effective heat transfer path from the fuel to the moderator. Following contact between the hot PT and the relatively cold CT (referred to as PT/CT contact), there is a spike of the heat flux in the moderator surrounding the CT. This could in turn lead to sustained CT dryout, as shown in Fig. 1. The prevention of CT dryout resulting from PT/CT contact depends on available local moderator subcooling. Higher moderator temperature (or lower subcooling) would decrease the margin of the CTs to dryout. In addition, the fuel channel integrity is considered to depend on the coolability of the moderator as an ultimate heat sink.

In this regard, estimation of the local subcooling of the moderator inside the calandria vessel under normal operational condition and/or transient conditions is one of the major concerns in the CANDU safety analysis. Accordingly, the Canadian Nuclear Safety Commission (CNSC), a regulatory body in Canada, categorized the temperature prediction of the moderator for fuel channel integrity as a general action item and recommended that a series of experimental works be performed to verify the system evaluation codes in comparison with results of three-dimensional experimental data. In terms of predicting the moderator temperature in a CANDU reactor, some studies have been performed in the Canadian nuclear industry. Koroyannaski et al. [1] experimentally examined the flow phenomena formed by inlet flows and internal heating of a fluid in a calandria-like cylindrical vessel of the SPEL (Sheridan Park Engineering Laboratory) experimental facility. They observed three flow patterns inside the test vessel and their occurrence was dependent on the flowrate and heat load. Quaraishi [2] simulated the fluid flow and predicted temperature distributions of SPEL



(a) before



(b) after

Fig. 1. PT/CT Contact Phenomena

experiments using computational codes, 2DMOTH. Carlucci and Cheung [3] investigated the two-dimensional flow of an internally heated fluid in a circular vessel with two inlet nozzles at the sides and outlets at the bottom, and found that the flow pattern was determined by a combination of buoyancy and inertia forces. Collins [4, 5] carried out thermal hydraulic analyses for SPEL experiments and Wolsong units 2,3,4, respectively, using the PHOENICS code with a porous media assumption for the fuel channels.

Although some computational codes have been used to predict moderator temperature in some experimental studies and for real CANDU-6 nuclear power plants, they could not three-dimensionally simulate the flow behaviors inside calandria-like vessels. Furthermore, although some studies were performed with three-dimensional computational codes, the porous media approximation instead of real heaters or fuel channels located in the vessel was used. However, secondary flow, which plays an important role in fluid flow and heat transfer characteristics, can be observed among the horizontal fuel channels. Therefore, it is necessary to establish an analyses model that can simulate the flow behaviors and reasonably predict the temperature distributions inside a calandria vessel using the real geometry of fuel channels.

The objectives of this study are to establish an adequate theoretical basis for the models, and to analyze the fluid flow and temperature distributions inside the calandria vessel of a CANDU-6. The approach to this involves two steps. First, an optimized calculation scheme is obtained by various comparisons of the predicted results with related experimental data, and by evaluating the fluid flow and temperature distributions. Then, in the second step, using the optimized scheme, analyses for a real CANDU-6 under normal operating conditions and a transition condition are performed.

2. THERMAL-HYDRAULIC ANALYSES

2.1 Analysis Model

During the normal operation of a CANDU-6 reactor, there are two major flows inside the calandria vessel, i.e., buoyant fluid flow formed by internal heating and momentum fluid flow by the jet flows through inlet nozzles. The cold fluid is heated by with the heat generated not only by slowing down of neutrons and directed radiation heating of gamma rays in the vessel, but also by convective heat transfer from the fuel channel surface. As a large amount of heat load to the fluid is continuously transferred, the heated fluid flows upward due to the buoyancy force. Hence, it is noticeable that both the total heat load to the fluid and the inlet flow rate through the nozzles are the major parameters affecting the flow characteristics inside calandria vessel. However, the flow characteristics formed by the combination of heat load and inlet flow are very complicated to predict analytically under given operation conditions. Moreover, as the fluid passes among the fuel channels, the pressure drop and numerous secondary flows near the pressure tubes make precise and detailed analyses more difficult. Therefore, in this study, a numerical model for analyzing the thermal-hydraulic behaviors inside a calandria vessel, including the buoyancy and momentum forces, is considered.

All the horizontal heaters (fuel channels) in the calandria vessel are modeled such that they can be analyzed as a real geometry. This is pertinent because, in acting as flow resistance, they affect the heat transfer and the flow field. The fluid flow is assumed to be a steady, incompressible fluid flow. The flow is simulated using the standard k-ε turbulence model with a standard wall function to calculate the turbulent intensity and dissipation rate within the vessel. An adiabatic boundary condition is applied to the inner wall surface of the calandria vessel. The buoyancy effects are accounted for density change assumed as the Boussinesq approximation with a thermal expansion coefficient, β of 0.00054 K^{-1}. The conjugated heat transfer analysis method is used to calculate the asymmetry convective heat transfer on the fuel channel surface so as to adequately match the fluid-wall interface conditions. In order to solve the governing equations, a general purposed three-dimensional computational fluid dynamics code, FLUENT 6.0, is used. The computational domain is discretized into finite control volumes (or cells) using unstructured meshes for fine calculation except for in the vicinity of fuel channels and the reflector region. These are modeled with structured fine meshes for accurately calculating the turbulence generation and dissipation rate. In addition, several grid layers are applied to consider the wall effects of the fuel channels. In the calculation, the iteration is progressed until the absolute sum of dimensionless residuals in the numerical solution of governing equations is less than 10^{-3}.

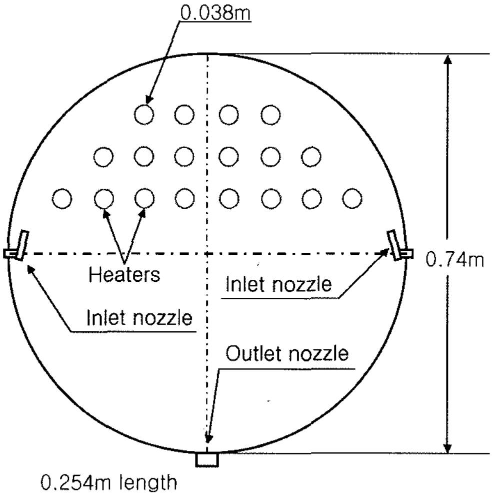
2.2 Analyses of SPEL Experiments

In order to experimentally investigate the flow phenomena in moderator inlet jets and internal heating in a calandria-like cylindrical vessel in a CANDU reactor, a series of experiments were performed at SPEL in Canada [1]. To verify the present analysis model, analyses of SPEL experiments have been performed.

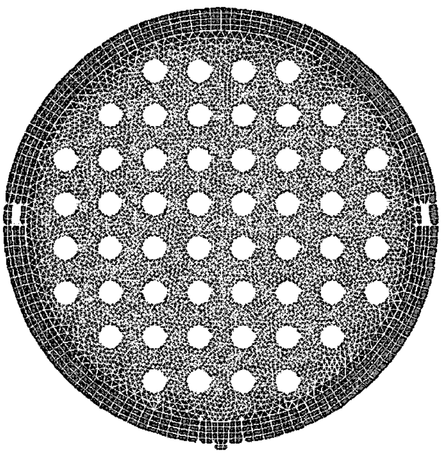
2.2.1 SPEL Experiment

The test vessel of SPEL is not a scaled facility of a real CANDU calandria vessel, but it has the features of a typical CANDU reactor, such as re-circulating jet induced flow, heating of the water by volumetric heat generation, and a matrix of horizontal tubes parallel to the cylindrical axis. The vessel consists of a transparent acrylic cylindrical shell with 0.737 m inner diameter and 0.254 m length. 52 copper tubes (38 mm outer diameter and 254 mm length) are arranged on a 75 mm square pattern and installed inside the tank. Two inlet nozzles were designed and installed along the horizontal centerline at each side of the vessel with an angle of 14° from the vertical direction. An outlet nozzle was also installed at the bottom of the vessel. High amperage, low voltage alternating current was passed via the tubes through the working fluid, which acted as a fluid resistor to generate heat from the copper tubes, which act as electrodes. A chemical flow visualization technique was employed to determine the predominant flow regime inside the vessel. Detailed temperature profiles inside the vessel were obtained using an optical fiber probe.

To simulate the SPEL experiments, all dimensions were

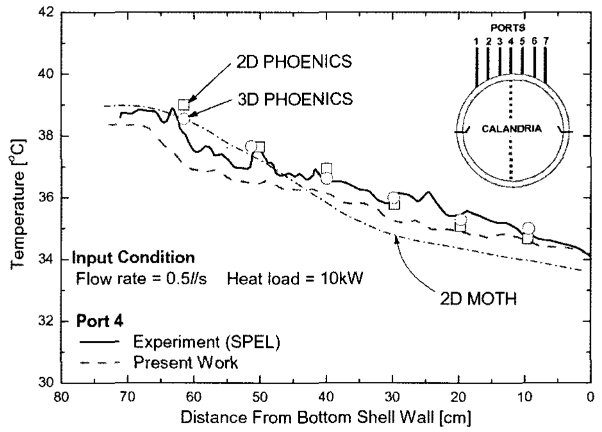


(a) SPEL experimental facility

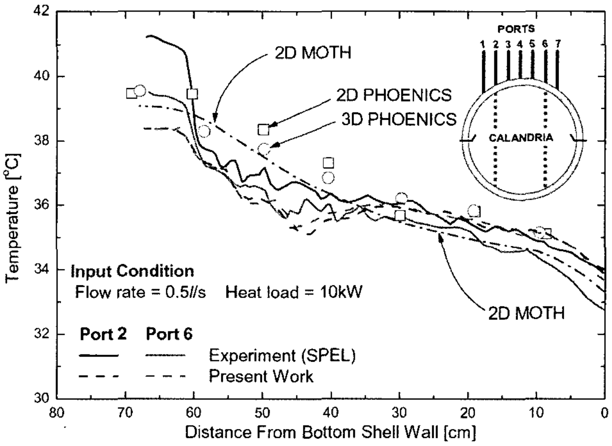


(b) mesh for calculation on central plane

Fig. 2. Schematic Diagram of SPEL Experiments and Calculation Domain

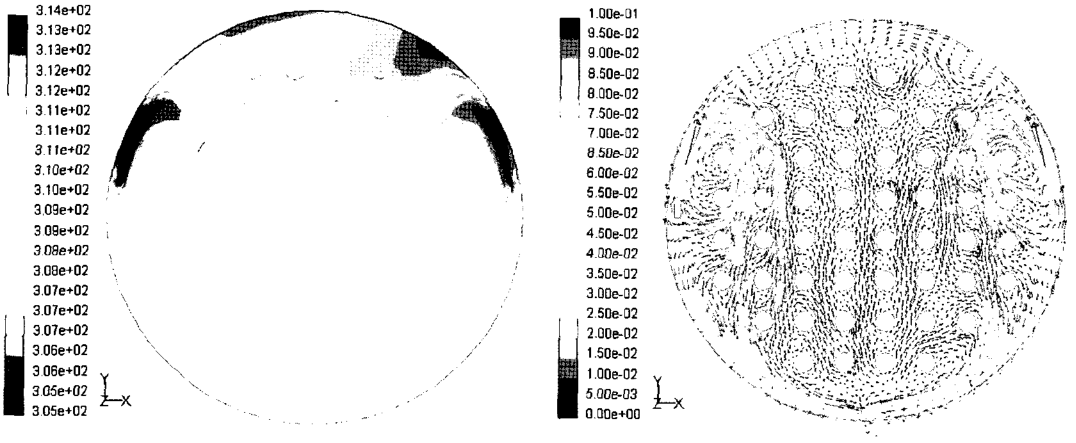


(a) along the port 4

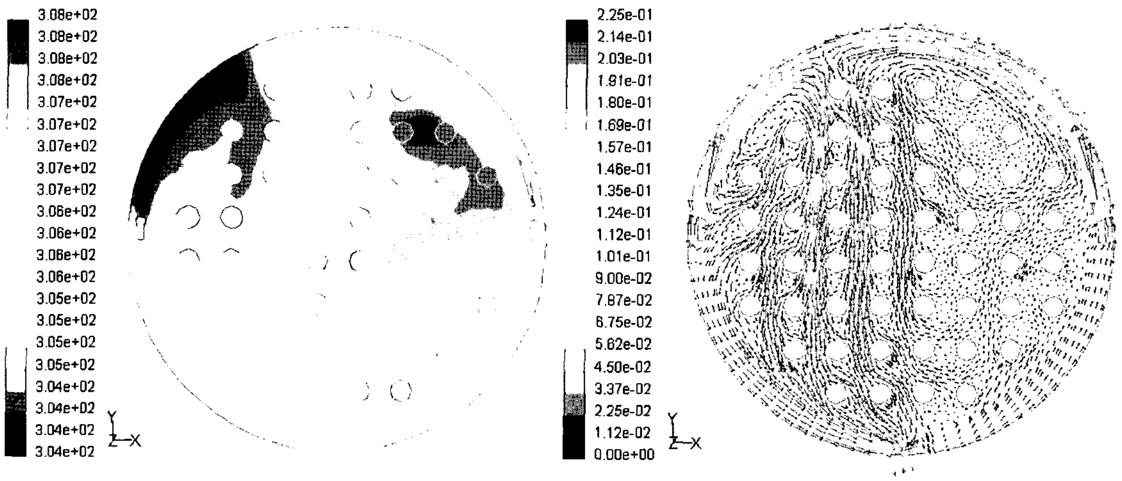


(b) along the ports 2 and 6

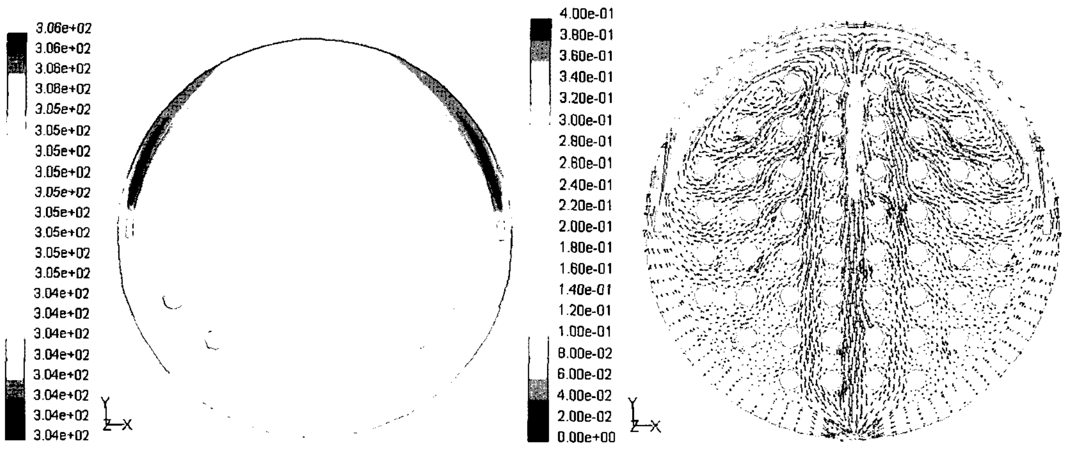
Fig. 3. Comparison of Experimental and Predicted Temperatures Along a Vertical Centerline



(a) buoyancy dominated flow (V_{in}=0.13m/s, T_{in}=30℃, Heat Load=10kW)



(b) mixed type flow (V_{in}=0.25m/s, T_{in}=30℃, Heat Load=10kW)



(c) momentum dominated flow (V_{in}=0.40m/s, T_{in}=30℃, Heat Load=10kW)

Fig. 4. Typical Flow Patterns for SPEL Experiments

set to the experimental apparatus, as shown in Fig. 2, and the number of cells was about 53,000 for a total volume of 0.3735 m^3 with unstructured meshes in the tube region and structured meshes in the outer region. These numbers were optimized in accordance with the mesh sensitivity analyses.

2.2.2 Results and Discussion

Numerical calculations were carried out for the calandria-like vessel of SPEL and the present analysis model predicted the fluid temperature reasonably, i.e., the maximum temperature inside the calandria-like tank is 40.3°C, similar to the SPEL experimental result of 41°C. Figure 3 shows comparisons of the experimental and computed temperatures along the vertical centerline. As shown in Fig 3(a), the temperature decreases slightly at the upper region and decreases gradually along the vertical centerline. The temperature predicted by 2DMOTH is underestimated at the bottom region of the calandria compared with that of the SPEL experiment. However, as shown in Fig. 3(b), the temperature decreases sharply due to the momentum of inlet flow at the regions of flow passage of the fluid. However, the temperature increases slightly due to the heat generation, and decreases again at the bottom region due to the forced convection. The above results indicate that the present model has the capability to properly analyze a fluid flow simultaneously subject to combined buoyancy and momentum forces.

The fluid flow inside the vessel is very complex due to the interaction between the momentum force generated by the inlet jets and the buoyancy force by heat load to the fluid. Hence, in order to identify the flow patterns formed inside the vessel with some major input conditions, some calculations have been performed. According to the computational results for the SPEL geometry, three flow patterns, e.g., momentum dominated flow, buoyancy dominated flow, and mixed type flow, respectively, were observed inside the calandria vessel, as shown in Fig. 4. It was also noticed that the onset conditions of these flow patterns mainly depend on the heat load and inlet flowrate. At low heat load and high flowrate, the effect of momentum force on the main stream inside the calandria vessel is more dominant than that of the buoyancy force. Although the entering fluid is heated by heat generation, the fluid flows toward the outlet due to the small amount of heat load and strong inlet jets. The two inlet flows encounter each other near the center region, and discharge through the outlet. In this condition, the high temperature area of the fluid is located at the near-bottom of the calandria, and the momentum dominates the flow forms. At high heat load and low flowrate, however, the effect of buoyancy force on the main stream becomes more significant than that of momentum force. Due to the strong heat generation, the entering cold fluid is heated enough to flow upward by the density gradient. It flows along the inner wall of the vessel and discharges through the outlet, producing secondary flows in the vicinity of the calandria wall and heaters. The high temperature area of the fluid is located at the upper region of the calandria, and a buoyancy-dominated flow is found. At a given heat load and flowrate, due to the force balance of momentum and buoyancy forces, mixed type flow forms and high temperature of the fluid are located at the center region.

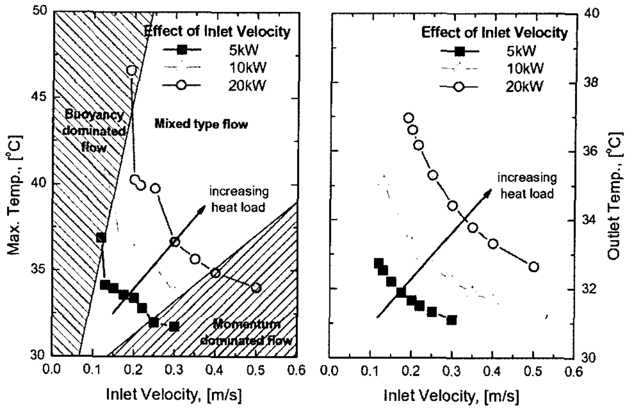


Fig. 5. Effect of Inlet Velocity on Temperatures

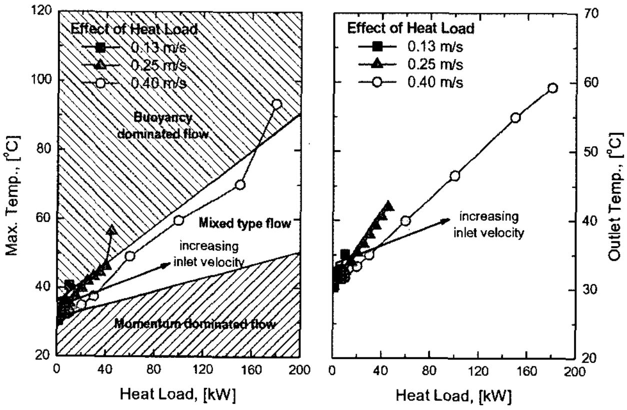


Fig. 6. Effect of Heat Load on Temperatures

It was found that the inlet velocity and heat load are the major parameters affecting the formation of flow patterns. Hence, in order to quantitatively elucidate the effects of the two major parameters, some sensitivity analyses were performed. In Fig. 5, as the inlet velocity is increased, the maximum and outlet temperatures are decreased due to the increasing effect of inlet jet momentum. It is also noticed that the ranges of inlet velocity where the mixed type flow is observed broaden as the heat load is increased. When the heat load is increased, the maximum and outlet temperatures are increased, as shown in Fig. 6. In particular, at a high moderator flow rate (momentum dominated flow), the

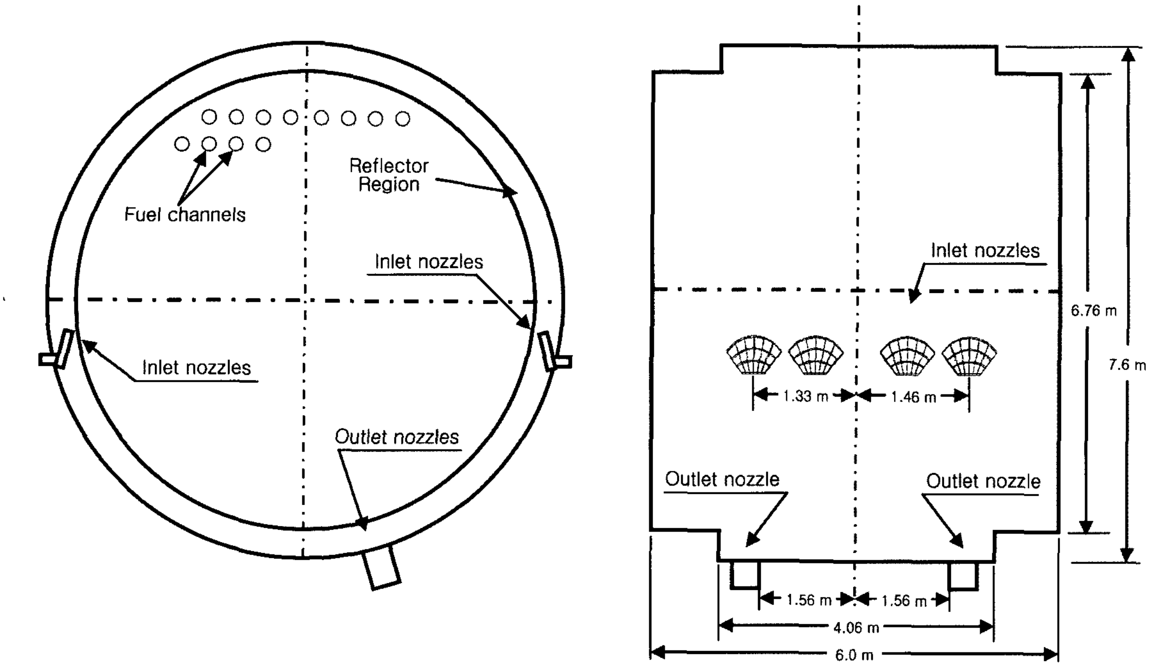
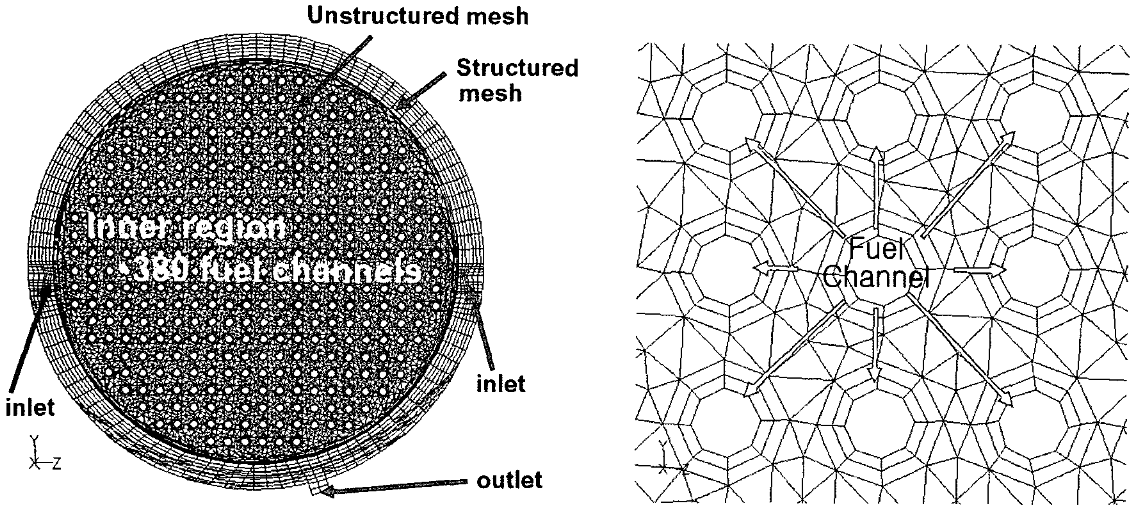


Fig. 7. Calandria Vessel of CANDU-6



(a)

(b)

Fig. 8. Mesh for Calculation of CANDU-6

difference between the maximum and outlet temperature is increased with the increased heat load.

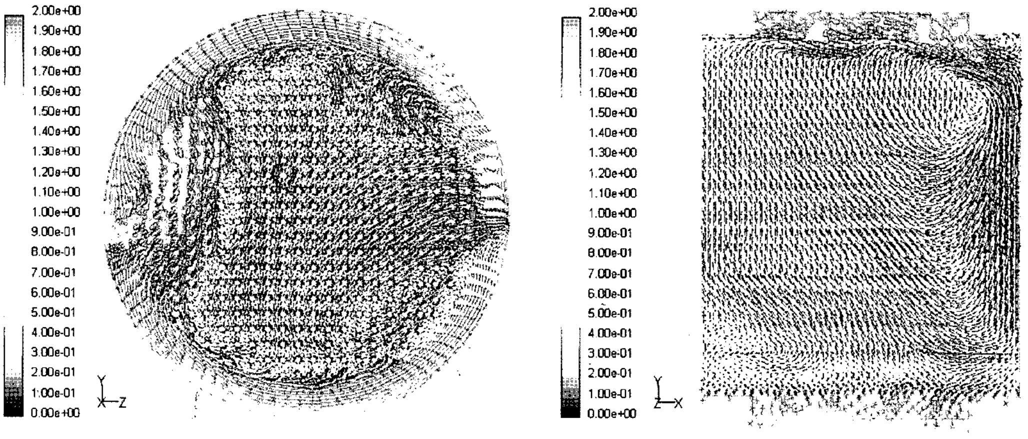
2.3. Analyses of CANDU-6

Analyses on the thermal hydraulic characteristics of the moderator inside a calandria vessel under the normal operating conditions of a real CANDU-6 reactor have been carried out using the present optimized model. In addition, transient analyses were also performed to estimate the fuel channel integrity during a LOCA with a LOECC.

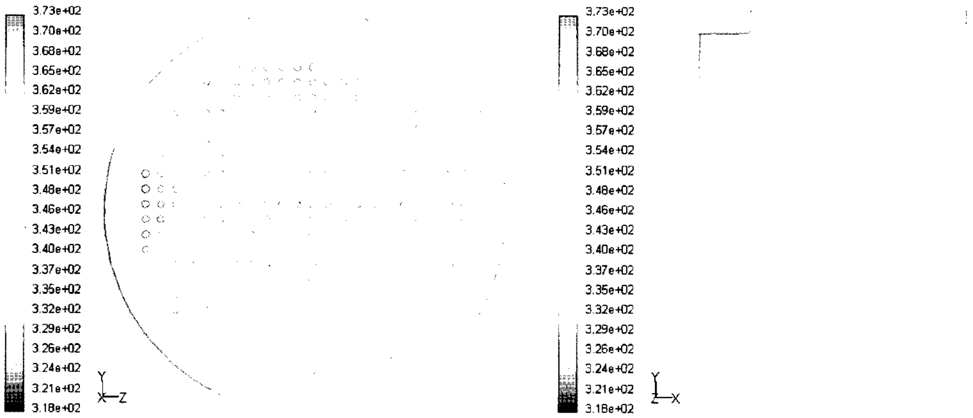
2.3.1 Calandria Vessel of CANDU-6 Reactor

A calandria vessel of CANDU-6 has the shape of a cylindrical tank, and is 6 m long and 7.6 m in diameter at its widest point. In the core region, there are 380 fuel channels with 0.131 m outer diameter, accounting for about 12 percent of the calandria vessel volume. These channels act as hydraulic resistance to fluid flow. As shown in Fig. 7, 8 inlet nozzles pointing upward at an angle of 14° from the vertical direction are installed at the middle of each sidewall of the calandria vessel symmetrically in the axial-center plane and asymmetrically axially. Two outlet ports are symmetrically located axially and asymmetrically placed in the axial-center plane.

Figure 8(a) shows the computational meshes on the axial-center plane to analyze the fluid dynamics and heat transfer characteristics inside the calandria vessel of the CANDU-6 reactor. In the core region, 380 fuel channels with 0.286 m of square pitch are simulated, and hence it is possible to observe the flow behaviors around the channels. The total cell number is about 830,000 for a total volume

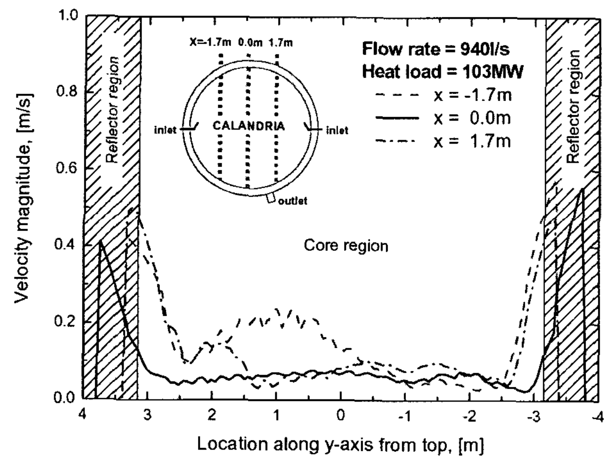


(a) velocity magnitude

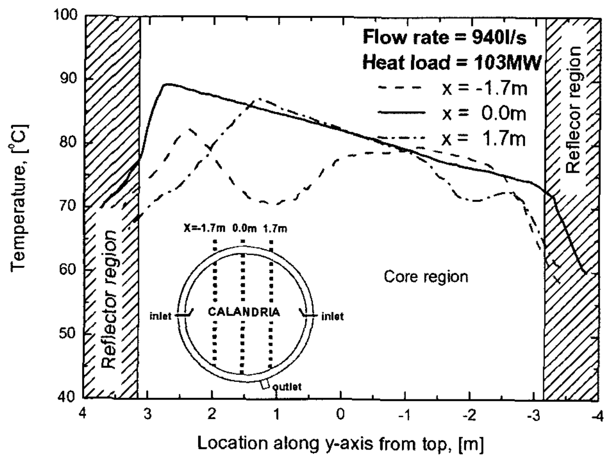


(b) temperature

Fig. 9. Velocity and Temperature Distributions in Normal Operating Condition



(a) velocity magnitude



(b) temperature

Fig. 10. Velocity and Temperature Profiles Along the Vertical Centerline in Normal Operating Condition

of about 965 m^3 with unstructured meshes in the core region and structured meshes in the vicinity of the fuel channels and reflector region in order to reduce calculation time. In addition, to consider the boundary layer effect over a fuel channel, 5 grid layers are averagely applied for a distance of 0.155m ($y^+ = y/\nu \cdot \sqrt{\tau_w/\rho}$ is up to about 1000) over each fuel channel, as shown in Fig. 8(b).

2.3.2 Results and Discussion

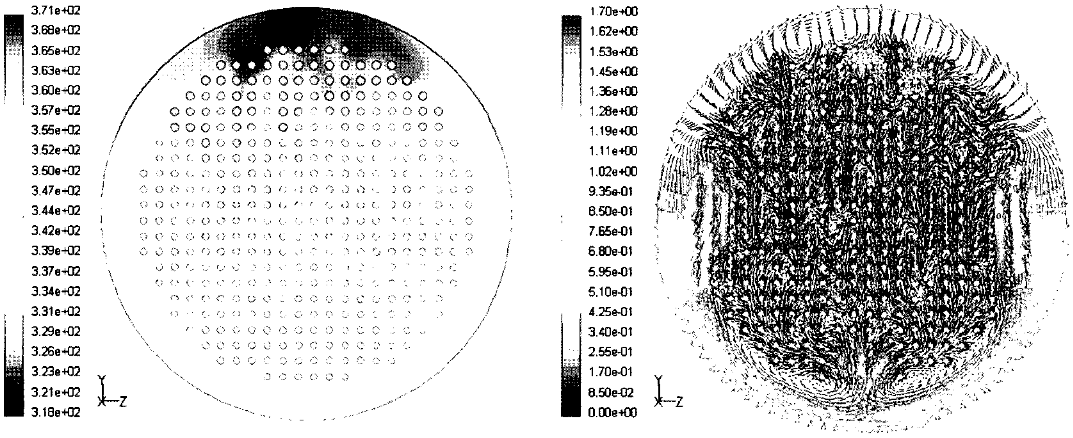
Steady-state condition: Normal Operation

Under the normal operating conditions of CANDU-6, most heat is deposited in the moderator by direct heating of neutrons, decay heat from fission products, and/or gamma rays, while heat by convection from the surface of the fuel channels accounts for a small portion of the total heat load. For a conservative analysis, 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 the fuel channel surface. In addition, it is assumed that the convective heat is transferred to the moderator uniformly in the axial direction. The total flow rate through the eight inlet nozzles is taken to be the design value of 0.94 m^3/s at the inlet temperature of 45°C. The thermal boundary condition of the calandria outer wall is conservatively assumed as adiabatic.

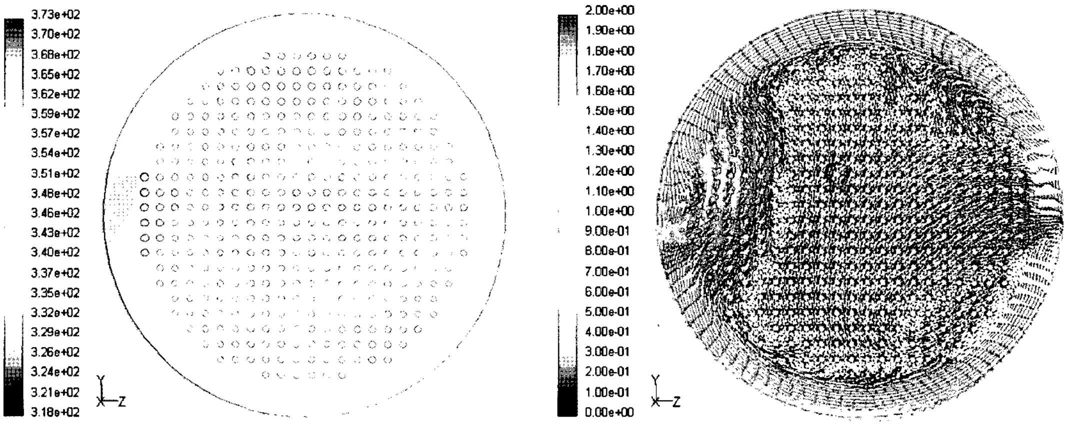
Under normal operating conditions, the predicted maximum, outlet, and average temperatures of the moderator are 95.10°C, 68.99°C, and 75.21°C, respectively, and no boiling is predicted. In particular, the present analysis model could properly predict the moderator outlet temperature compared with the recommended design value in the technical guides; the CANDU reactor should be operated with an outlet temperature of approximately 74°C and should not exceed 83°C.

Figure 9 shows the complex and asymmetric flow pattern inside the calandria vessel in the condition of normal operation, induced by a combination of the buoyancy and the inlet momentum forces and by the geometric effects, including fuel channels acting as flow resistance and asymmetric installation of outlet ports. The flow reverses its direction near the left inlet nozzles and proceeds down toward the outlets, and the temperature and velocity profiles show a steep change around the flow reversal region. Fluid heated in the core region flows upward by the buoyancy force, and changes its flow direction upon encountering the inlet cold flows at the circumferential upper region. The hottest spot is located at the upper region of the core region, which slightly leans to the right side. Therefore, the flow pattern in the normal operating condition of CANDU-6 is predicted to be mixed type flow with a combined buoyancy dominated flow and momentum dominated flow. In conflict with the assumption of uniform heat transfer from the fuel channels to the moderator, the predicted axial distributions of temperature are not uniform due to the combined effect of buoyancy force and momentum force.

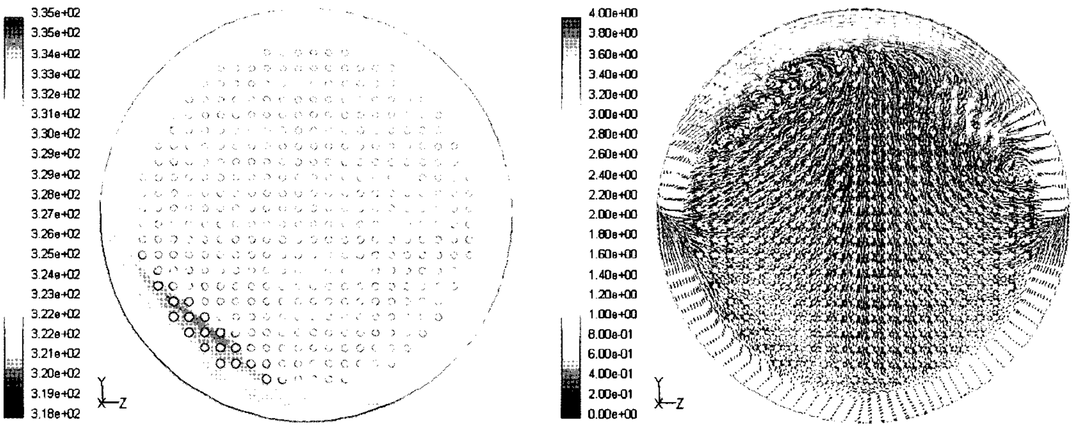
Figure 10 shows the temperature and velocity distributions along the vertical centerline of the calandria vessel. A moderator with relatively high temperature is placed in the upper region of the calandria vessel, due to the buoyancy force, and its temperature decreases gradually along the vertical line. Moreover, the two temperature profiles along the symmetric vertical centerlines are asymmetric with each other. The moderator velocity magnitude changes sharply near the wall of the calandria vessel, and hence the velocity gradient is concentrated in this region. The velocity in the core region is relatively small due to the hydraulic resistance, i.e., the fuel channels, as well as the force balance between buoyancy and momentum forces.



(a) buoyancy dominated flow (V_{in}=4.5m/s, Heat Load=103MW)



(b) mixed type flow (V_{in}=5.3m/s, Heat Load=103MW)



(c) momentum dominated flow (V_{in}=10.0m/s, Heat Load=103MW)

Fig. 11. Typical Flow Patterns for CANDU-6

In addition, secondary flows with small velocity among the fuel channels are found, and these play important roles in heat transfer and flow behaviors.

Three flow patterns inside the calandria vessel are also observed according to the variation of inlet velocity, and are similar to those of the SPEL experiments, as shown in Fig. 11. However, at the bottom region, the temperature and velocity distributions lean slightly toward the outlets due to the asymmetric installation of two outlets.

Transient Analyses: 35% RIH Break with LOECC

To perform the transient analyses, the transient condition after a 35% RIH (reactor inlet header) break LOCA with LOECC is considered. The initial condition for the transient analysis is the steady-state solution of normal operation of the CANDU-6 reactor. Figure 12 shows the typical profile of total heat load transferred to the moderator in this transient condition. The figure also shows numerous peaks of the total heat load between about 20 and 40 seconds due to PT/CT contact. Based on 40 seconds, the transient condition can be divided into a blowdown phase (up to 40 seconds) and a post-blowdown phase (after 40 seconds).

The heat load generated in the blow-down phase consists of two heat sources: (i) volumetric sources such as fission product decay heat and neutronic power, which are exponentially decreasing; and (ii) convective heat transfer by PT/CT contact in the critical pass. After the reactor trip is initiated, an increase of void generation due to the leakage of primary coolant during the LOCA without ECC injection induces a power pulse caused by void reactivity. A sharp peak of the heat load to the moderator takes place at about 1 second and the total amount of the heat load to moderator amounts to approximately 250 MW. However, the effect of the sudden heat load increase due to the power pulse is not significant, because the heat sink capability of the moderator is large enough compared to the effect of this pulse. As shown in Fig. 12, while a slight increase of the maximum temperature is observed due to the sudden increment of heat load at about 1 second, the flow is maintained in the same pattern as that of normal operation. During the time span of 1~20 seconds, PT/CT contact does not occur and the temperature of the moderator decreases continuously until PT/CT contact occurs, because of the decreasing heat load of volumetric heat sources. At approximately 20 seconds, a large amount of heat load generates from the fuel channels due to PT/CT contact and oscillates severely. The predicted maximum local temperature of the moderator reaches about 95.2°C, and thus the moderator can be maintained in a subcooled condition without boiling since the maximum temperature is lower than the saturation temperature. In the post-blowdown phase (after 40 seconds), although PT/CT contact could take place in each of the 190 channels of the broken loop, the total heat load is not large enough to cause the moderator to boil and the moderator temperature continuously decreases.

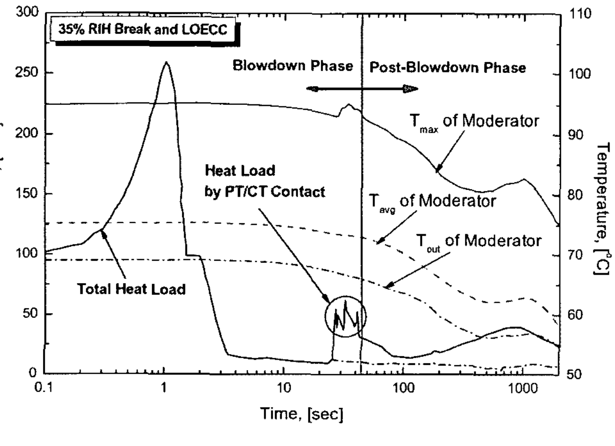


Fig. 12. Heat Load Profiles and Temperature Variation During Transient

3. PROPOSED FLOW REGIME MAP

According to the previous and present analyses, three moderator flow patterns, i.e., buoyancy dominated flow, mixed type flow, and momentum dominated flow, are generally observed in the calandria vessel. Among these flow patterns, the maximum temperature predicted in the buoyancy dominated flow is the highest. To ensure the safe operation of the CANDU reactor, it is necessary to determine the moderator flow regime map inside the calandria vessel, as this may be helpful to predict the flow pattern in a given condition in advance.

In order to develop the flow regime map, it is necessary to explore the main parameters representing the flow fields. In the flow fields inside the calandria, there are three forces that must be considered: the inertia force, viscous force, and buoyancy force. Therefore, two dimensionless numbers, Reynolds number (Re) and Archimedes number (Ar), can be considered to describe the flow regime map explicitly as follows;

$$\mathrm{Re} = \frac{\text{inertia force}}{\text{viscous force}} = \frac{\rho VD}{\mu}$$

and

$$Ar = \frac{Buoyance\ Force}{Inertia\ Force} = \frac{g\beta QD}{C_p \rho A V^3}$$

where

A = inlet area, m^2
C_p = specific heat, J/kg-K
D = calandria vessel, m

Q = Heat load, W
V = inlet velocity, m/s
β = thermal expansion coefficient, K^{-1}
μ = viscosity, Pa-s
ρ = density, kg/m^3

Figure 13 is the flow regime map for the SPEL experiments and also shows the comparison of the results of SPEL experiments and the present prediction. The present model predicted the experimental results reasonably well, as previously noted. Thus, it is considered that the present model can reasonably estimate a flow field where buoyancy and momentum forces coexist. According to the present and experimental results, it could be concluded that the transitions of flow patterns inside the calandria vessel can be described on the basis of a certain constant Ar number. For the CANDU-6 reactor, the flow patters under the normal operational conditions are predicted as a mixed type flow and are agree well with those of the CFD simulation. Accordingly, the proposed flow regime map could be applied to predict the flow patterns under some operational conditions.

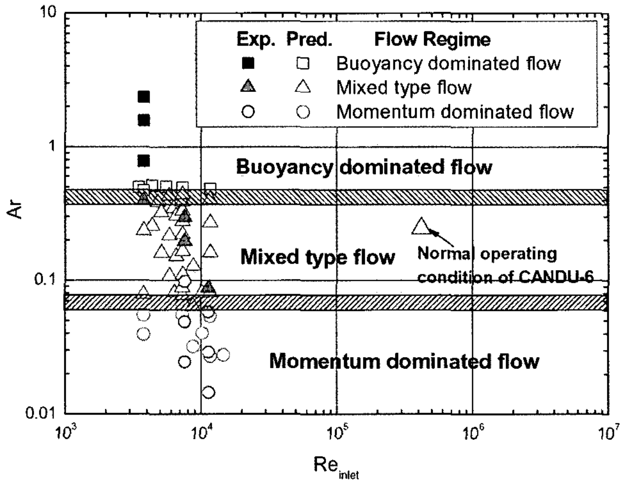


Fig. 13. Proposed Flow Regime Map

4. CONCLUSIONS

In order to investigate the thermal-hydraulic characteristics of moderator flow subject to two forces, i.e., momentum force by inlet jets and buoyancy force by heat load inside the calandria vessel of a CANDU-6 reactor, an analysis model has been established through comparison with experimental data. A series of numerical simulations has also been performed for the normal operating conditions and the transient condition of a 35% RIH break LOCA with LOECC. The real geometry of a CANDU-6 reactor with 380 fuel channels is simulated in this work in order to investigate the flow field near the fuel channels. The present model should prove useful in resolving the moderator temperature prediction item, which is the one of the generic action items issued by CNSC. The main conclusions of this study are as follows:

1. The comparison with the SPEL experimental data shows that the present model can reasonably predict the temperature distributions of the moderator, indicating that the present model has good capability to simultaneously analyze a fluid flow subject to buoyancy and momentum force.
2. It is found that the major parameters affecting the flow patterns are the inlet flow rate and heat load to the fluid. With the inlet flow rate, heat load, or both, three flow patterns can be predicted; that is, momentum dominated flow, mixed type flow, and buoyancy dominated flow.
3. In the normal operation of CANDU-6, due to the combined effect of momentum and buoyancy, a mixed type flow is predicted. Furthermore, many secondary flows with small velocity in the vicinity of the calandria wall and fuel channels are found and these play an important role in heat transfer.
4. In the transient condition of a 35% RIH break with LOECC, no boiling was predicted although there was a sharp peak of heat load at about 1 second and a large amount of heat load due to PT/CT contact. As such, since the moderator within the calandria vessel has sufficient coolability as the ultimate heat sink, the fuel channel integrity can be maintained and assured.
5. With the major parameters reasonably representing the flow pattern, a flow regime map has been proposed for the safe operation of the CANDU reactor. It can be applied to predict the flow patterns under some operational conditions.

ACKNOWLEDGEMENT

The authors would like to acknowledge the support of the Ministry of Science and Technology (MOST) of the Republic of Korea.

REFERENCES

[1] Koroyannakis, D, Hepworth, R. D. and Hendrie, G. An Experimental Study of Combined Natural and Forced Convection Flow in a Cylindrical Tank, AECL Report, TDVI-382 (1983).
[2] QURAISHI, M. S. Experimental Verification of 2DMOTH Computer Code Temperature Predictions, AECL Report, TDAI-353 (1985).
[3] CARLUCCI, L. N. and CHEUNG, I., "The Effects of Symmetric/Asymmetric Boundary Conditions on the Flow of an Internally Heated Fluid," Numerical Methods for Partial Differential Equations, 2, 47-61 (1986).

[4] COLLINS, W. M. Simulation of the SPEL Small Scale Moderator Experiments Using the General Purpose Fluid-flow, Heat Transfer Code PHOENICS, AECL Report, TTR-213 (1988).

[5] COLLINS W. M. PHOENICS2 Model Report for Wolsong 2/3/4 Moderator Circulation Analysis, Wolsong NPP 2/3/4, 86-03500-AR-053 (1995).