

## CFD Prediction of Two-phase Fluid Flow Characteristics Inside a Fuel Channel of a CANDU-6 Reactor

Manwoong Kim<sup>a</sup>, Yong-Seog Choi<sup>a</sup>, Hyun-Koon Kim<sup>a</sup>, Sung Hoon Choi<sup>b</sup>

<sup>a</sup> Korea Institute of Nuclear Safety, P.O.Box 114, Yuseong, Daejeon, Korea, [mwkim@kins.re.kr](mailto:mwkim@kins.re.kr)

<sup>b</sup> Advanced Technology Engineering Service, 184-1 Goro-3dong, Goro-gu, Seoul, Korea

### 1. Introduction

Fluid flow and heat transfer calculations for reactor fuel assemblies are typically based on either one-dimensional models for individual sub-channels or models of interconnected sub-channels in which cross flows are accounted for in a simplified manner. As for the fuel channel of a CANDU 6 reactor, the development of mechanistic models and methods for flows along fuel elements enclosed inside typical CANDU-6 fuel channel has encountered difficulties due to the modeling of local effects along the horizontal channel. The complexity of the configuration such as fuel elements, spacers, etc. inside the fuel channel affects various thermal-hydraulic characteristics. In particular, the fluid flow and phase changes along the fuel channel get much more complicated in the case of the CANDU fuel channels. Since the mechanisms governing two-phase pressure drop along the horizontal channel is still not fully satisfactory even for flows in smooth channels, the complexity of the CANDU-6 fuel channel geometry makes it even more difficult to predict the two-phase fluid flow characteristics. In other word, the increased pressure losses and the sub-cooled boiling may also change the void distribution along the channel. This is particularly important in CANDU-6 reactors, where small changes of the average void in the fuel channel directly affect, through the void reactivity feedback, the axial power distribution in the core. The recent progress in CFD methods has provided opportunities for using mechanistic multidimensional models reflecting the actual geometry of the fuel channel.

Therefore, the objectives of this study are: (i) to investigate a proposed sub-cooled boiling model developed at Rensselaer Polytechnic Institute and to apply against a experiment, the *Frigg Assembly* and (ii) to predict local distributions of flow fields for the actual fuel channel geometries of CANDU-6 reactors. The two-phase models used in the predictions have been numerically implemented using the FLUENT CFD computer code as a solver of the governing two-phase flow equations.

### 2. An Evaluation of a Frigg Assembly

The present analysis used the FT-6a assembly of the early FRIGG experiment as the reference geometry. A cross-sectional view of the FT-6a assembly is shown in Fig. 1. The test section consisted of six electrically heated rods,

placed in a cylindrical pressure vessel. The heated length of the test section was 4.22 m, the vessel internal diameter was 71 mm, and the heated rod diameter was 13.8 mm. The void fraction was measured both in axial and lateral directions using the  $\gamma$ -ray attenuation system. The experimental condition is shown in Table 1.

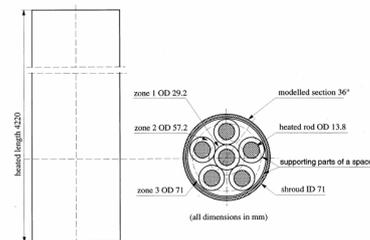


Fig. 1. Test section FT-6a of the FRIGG experiment

Table 1 Parameters of experiments used in validation.

Boiling liquid	water
Geometry	Vertical pipe with heated walls
System pressure	45 bar
Inner wall heat flux, $W/m^2$	0.57
Mass velocity, Re Numer	900 $kg/m^2/sec$ , 104210
section inlet	333 K
Wall material	Stainless steel

Computational mesh used for modeling of void distribution in test section FT-6a is as shown in Fig. 2 and the proposed sub-cooled boiling model was implemented in User Defined Function (UDF) of the FLUENT.

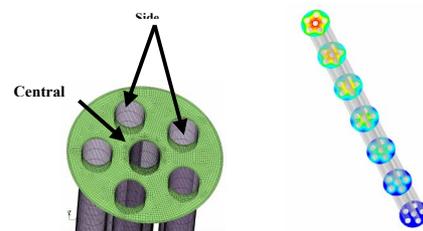


Fig. 2. Grid model and vapor void fraction development

Fig. 3 shows comparison of wall superheat  $T_w - T_s$  calculated and experiment. It indicates that “single phase”

greatly over-predicts superheat since it does not account for vaporization heat while introduction of the model could improve agreement. However, the model under-predicts the rate of vapor formation as shown in Fig. 3 shows comparison of liquid bulk sub-cooling where boiling model shows much better agreement with experiment than single phase model.

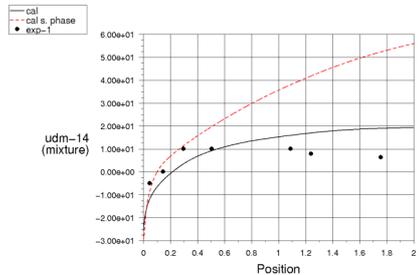


Fig. 3 Wall superheat vs. distance from inlet.

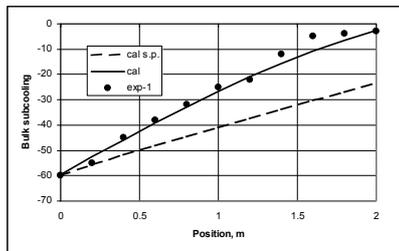


Fig. 4 Bulk sub-cooling vs. distance from inlet.

### 3. An Evaluation of Fuel Channel of CANDU-6

To investigate the convective sub-cooled boiling occurs inside the fuel channel under operating pressure, local distributions of flow fields and concentrations for both phases can be predicted for the actual fuel channel geometries of CANDU-6 reactors using multidimensional models of two-phase flow. The proposed sub-cooled boiling model used in the predictions has been numerically implemented using the UDF with the governing two-phase flow equations. The operating conditions are shown in Table 2.

Table 2 Operating parameters in validation.

Channel length	5.94 m
Diameter	0.10312 m
Diameter of rod	0.0131 m
Mass flowrate	28.281 kg/sec
Inlet temperature	541.163 K
Saturation temperature	586 K
Heat generation from fuel surface	function of region

Computational mesh used for modeling the fuel channel is as shown in Fig. 5.

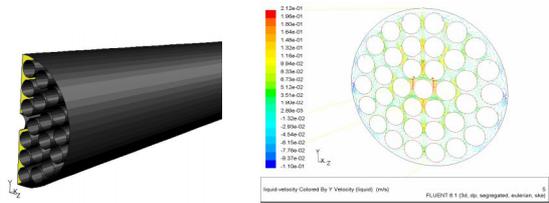


Fig. 5 Grid model and velocity distribution at the outlet

Fig. 6 shows comparison of void fraction between RELAP/CANDU and Fluent CFD predictions at the center and outer fuel elements. It indicates that the 3-D prediction using the proposed model is well agreed with that of RELAP/CANDU.

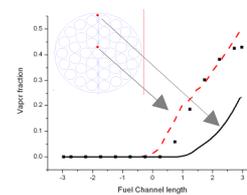


Fig. 6 Void fraction distribution

The comparison of temperature distribution between RELAP/CANDU and Fluent predictions at the center and outer fuel elements is also well agreed as shown in Fig. 7.

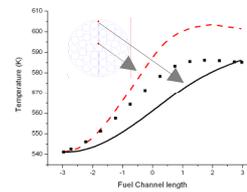


Fig. 7. Temperature distribution

### 4. Conclusion

To investigate the convective sub-cooled boiling occurs inside the fuel channel of the CANDU-6 reactor, the proposed sub-cooled boiling model applied against a experiment, the *Frigg Assembly* as well as predict local distributions of flow fields for the actual fuel channel geometries using the FLUENT CFD computer code.

### REFERENCES

[1] Podowski, M.Z. (1997) "Towards next generation multiphase models of nuclear thermal-hydraulics" *Proceedings of Eighth International Topical Meeting on Nuclear Reactor Thermal-Hydraulics, Kyoto, Japan*, Vol. I, pp. 53-68.

[2] Anglart, H. et al. (1997) "CFD prediction of flow and phase distribution in fuel assemblies with spacers" *Nuclear Engineering and Design* Vol. 177, pp. 215-228.