Khảo sát mô hình mô phỏng CFD đối với phân bố hệ số pha theo dọc kênh thực nghiệm ENTEK BM.

Hoàng Tân Hưng, Hoàng Minh Giang

Trung tâm Năng lượng hạt nhân, Viện Khoa học và Kỹ thuật hạt nhân
179 – Hoàng Quốc Việt, Cầu Giấy, Hà Nội
Hoangtanhung1991@gmail.com

Tóm tắt: Cho đến nay các mô hình mô phỏng dòng hai pha sử dụng trong CFD cũng chỉ có thể áp dụng tốt đối với một số các trường hợp cụ thể. Việc hiệu chỉnh mô hình mô phỏng nhằm thu được kết quả tính toán phù hợp với thực nghiệm cũng là công việc khó khăn. Báo cáo này trình bày mô phỏng CFD để tính phân bố hệ số pha hơi dọc theo kênh hệ thực nghiệm ENTEK BM có chiều dài 7m với điều kiến áp suất 3MP và 7 MPa. Điều hạn chế ở thực nghiện ENTEK BM là không có kết quả phần bố điểm theo các phương dọc và ngang để kiểm chứng mô hình mô phỏng. Tuy vậy việc so sánh kết quả mô phỏng trung bình hệ số pha hơi so với thực nghiệm và so với các phần mềm khác như CTF cũng phần nào phản ánh được sự phù hợp của các mô hình sử dụng trong CFD. Các phần mềm được sử dụng trong nghiên cứu ở đây là Ansys CFX và CTF.

Từ khóa: *CFD, ENTEK BM, ANSYS CFX, CTF, RPI.*

Investigation of void distribution along channel in ENTEK BM test using CFD simulation

Hoang Tan Hung, Hoang Minh Giang

Nuclear Power Center, Institute for Nuclear Science and Technology (INST)
179 – Hoang Quoc Viet, Nghia Do, Cau Giay, Ha Noi
Hoangtanhung1991@gmail.com

Abstract: Until now, two phase fluid models used in CFD simulation give only reasonable results in comparison with experiments for several specific cases. Calibration of necessary parameters in CFD models towards more reasonable results in comparison with those from experiment test is still challenged. This study presents CFD simulation for calculation void distribution along a vertical channel with 7 meter high in ENTEK BM test. The tests are performed in different pressures including 3MPa and 7 MPa. However, no radial nor along channel void distribution at local points are given from the tests in order to take for calibration several keys parameters in CFD models. Although, comparisons of CFD calculated results with averaged cross section void distribution along the channel from the tests as well as with results calculated from CTF code may be also presented a part of agreement between CFD models and experiment data. In this study, the codes Ansys CFX and CTF are used as CFD and component code, correspondently.

Keywords': *CFD, ENTEK BM, ANSYS CFX, CTF, RPI.*

I. Introduction

The multi-scale approach with different codes for reactor thermal hydraulics analysis was mentioned in the Ref. [1]. In this study, two different codes CTF (component code) and Ansys CFX (CFD code) are used to investigate void distribution along a vertical channel. As known, CTF is proven code and used long time in reactor core for thermal hydraulics analysis. Recently, CFD simulation for two phase flow such as CFX (Ansys CFX) for void fraction prediction provides with reasonable results in comparison with experiments data for several specific cases. However, calibration of necessary parameters in CFD models towards more reasonable results in comparison with those from experiment test is still challenged. This study presents CFD simulation for calculation void distribution along a vertical channel with 7 meters high in ENTEK BM test using CFX code. The tests are performed in different pressures including 3MPa and 7 MPa. However, no radial nor along channel void distribution at local points are given from the tests in order to take for calibration several keys parameters in CFD models. The ENTEK BM tests provide with only averaged cross section void fraction distributions along the channel at 10 locations. So that, these experiment data can be used for comparison with CFD simulation and if good agreements are achieved then some sub models chosen in CFX for boiling and condensation can be considered reasonable. Boiling and condensation models employed in CTF based on enthalpy change with heat transfer coefficients usually taken by Chen correlations [2] so that comparison of calculated results of void fraction between CTF and CFX code is also interested in term of verification of sub models using local phenomena in CFX.

II. The brief of ENTEK BM Test Facility

The BM Facility at the Research and Development Institute of Power Engineering models the force circulation circuit of RBMK type reactor. It includes many component fuel channel, steam separator, condenser, pump, and connecting piping. In CFD simulation we focus only the Heat Release Zone (HRZ, fuel channel – test section) for void simulation. Figure 1 shows a vertical and cross – section view of the HRZ. For cross – section view, the diameter are show in millimeters. The HRZ contains a 7 – rod bundle made by stain steel (X18H10T). All of rods are hollow with outer diameter of 13.5 mm, 1.25 mm wall thickness and 7 m length. The bundle is contained within a stain steel pressure tube (80 mm outer and 5 mm wall thickness) which is lined with a set of talcum chlorate thimbles (49 mm inner diameter and 10.5 mm wall thickness). The coolant flow area is 8.84×10^{-4} m^2 and hydraulic diameter is 7.84 mm. There are 20 honeycomb-type pin spacing grids along the HRZ, starting 30 mm from the beginning of the HRZ and repeated every 350 mm; these are similar to the spacers in the RBMK-1000 and have a hydraulic loss coefficient of 0.4 based on measurements.

Ten measured positions are located along the vertical channel (0.385, 0.948, 1.573, 2.322, 2.974, 4.010, 4.823, 5.448, 6.135, and 6.760 m from bottom of heated length). The measurements are taken by moving the detector during the test. The density was converted to a void fraction (ν) using the formula:

$$\nu = \frac{\rho_l - \rho_m}{\rho_l - \rho_v} \qquad (1)$$

Where ρ_l is the liquid phase coolant density at the measured (i.e., outlet) pressure and local coolant temperature calculated based on heating the coolant over the non-boiling length; ρ_v is the steam phase density at saturation for the test's pressure and ρ_m is mixture density.

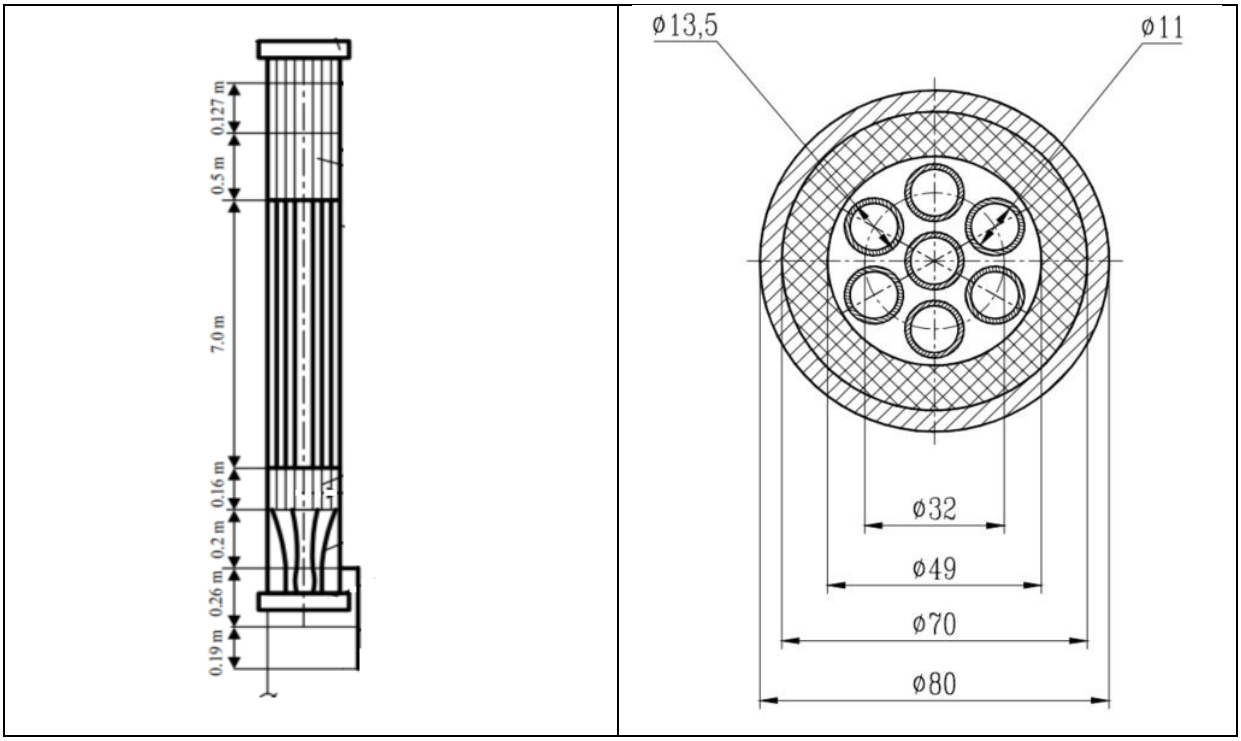

Figure 1: HRZ with vertical and cross – section view [4]

The uncertainties of the measurements for each parameter are following for all tests:

- Pressure at HRZ outlet: ±1.5 %;
- Coolant mass flow rate: ±0.0018 kg/s;
- Coolant temperature at HRZ inlet: ±1 K;
- Electrical power: ±2 kW;
- Void fraction: ±0.03;

The report [3] presents 25 tests together calculation results using RELAP5 code based on different four main parameters: pressure, mass flux, thermal power and coolant inlet temperature. In this study, six test cases as presented in Table 1 are investigated by CFX simulation. The CFX calculated void fractions are compared with those calculated by CTF mentioned in [2].

Six test cases are selected as following: three test cases in pressure condition of 3 MPa (T01, T04, and T014) and three other test cases in pressure of 7 MPa (T18, T24, T25). The input parameters using average value of these test cases are presented in Table 1.

Table 1: Input parameters of test cases [3]

Test No	P (MPa)	Gin (kg/s)	Q (kW)	Tin (K)
T01	3.12	0.437	300.3	387
T04	3.11	0.8816	297.6	451
T14	3.11	1.7644	511.3	476
T18	7.14	0.8816	302.4	525
T24	7.17	1.7749	519.6	528
T25	7.16	0.8849	632.1	454

III. The RPI Wall Boiling model

ANSYS CFX using RPI wall boiling model proposed by Kurul and Podowski, the heat flux (Q_w) from heated wall to the fluid is divided into three components [4]:

$$Q_w = Q_c + Q_e + Q_q \quad (2)$$

Where Q_c: the single – phase convection heat flux; Q_e: the evaporate heat flux and Q_q: the wall quenching heat flux. Each component is calculated by the equations:

$$Q_c = A_1 h_c(T_W - T_l) \quad (3)$$

$$Q_e = \dot{m}\left(h_{g,sat} - h_l\right) \quad (4)$$

$$Q_q = A_2 h_q(T_W - T_l) \quad (5)$$

where fraction A_2 is influenced by the vapor bubbles, formed on the heated wall, fraction A_1 is the rest of the heated wall surface, with $A_1 = 1 - A_2$; h_c is the turbulent heat transfer coefficient, which depends on the velocity field and on the near – wall grid cell size; T_W and T_l are the solid wall and the liquid at the wall temperatures; $\dot{m}$ is the evaporation mass transfer rate per unit wall area; $h_{g,sat}$ and are the specific enthalpies of the saturated vapor and sub-cooled liquid, respectively and h_q is the quenching heat transfer coefficient.

The area fraction values A_1 and A_2 play an important role in the heat – partitioning model. It is estimated the value of A_2 is the proportion of heated wall covered by nucleating bubbles:

$$A_2 = \min\left(\pi.\frac{a^2 d_w^2}{4}.n, 1\right) \quad (6)$$

Where d_w is the bubble departure diameter, n is the nucleation site density and a is an influence factor introduced by Kurul & Podowski and is assumed to be given by $a = 2$.

To closed model formulation for partitioning of the wall heat flux scheme, some closure models have to be provided. These are required following model parameters:

$$d_w = \min\left(d_{ref}.\exp\left(\frac{\Delta T_{sub}}{\Delta T_{ref}}\right), d_{max}\right) \quad (7)$$

The parameter of the model are dimensional (d_{max} =1.4 [mm], d_{ref} = 0.6 [mm], ΔT_{ref} = [45K]) and ΔT_{sub} refers to the local liquid subcooling. These model data are specific for the model application to nucleate subcooled boiling under pressurized conditions and need to be revised in case of model application to different operating conditions

n is the nucleation site density per unit wall area, given by Lemmert and Chawla model

$$n\ [m^2] = (m(\Delta T_{sup}[K]))^p \quad m = 210,\ p = 1.805 \quad (8)$$

IV. CFD Simulation Modeling

CFD geometry and Numerical meshes

Due to the symmetry structural of the HRZ, only 1/6th of the geometry will be simulated in the CFD simulation. To study mesh influence to simulation results, three meshes are generated with different refined levels. The averaged cross section void fraction calculated by mesh 1

different with those in mesh 2 and mesh 3 in which results between mesh 2 and mesh 3 are closed to each other. So that mesh 2 will be used for other cases.

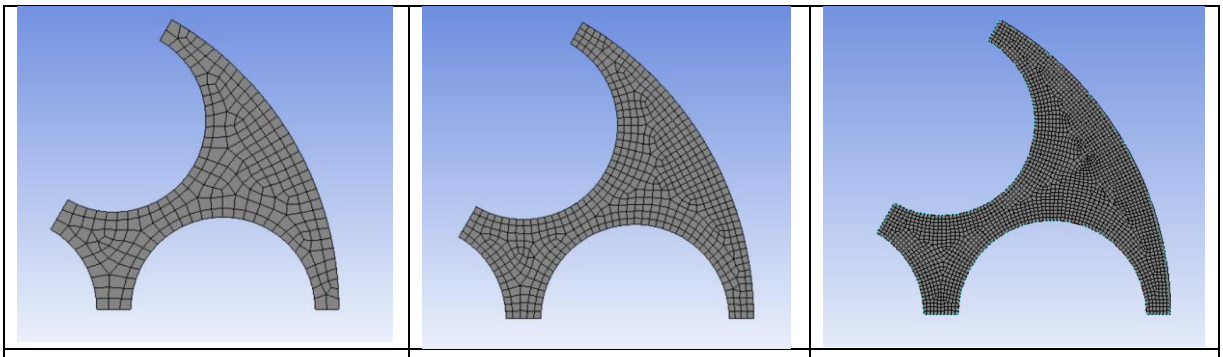

Figure 2: Mesh influence investigation

Table 2: Mesh influence investigation results for test case T01

Z (m)	Mesh 1	Mesh 2	Mesh 3
0.385	0.000	0.000	0.000
0.948	0.000	0.000	0.000
1.573	0.000	0.000	0.000
2.322	0.000	0.000	0.000
2.947	0.000	0.000	0.000
4.01	0.041	0.047	0.047
4.823	0.087	0.113	0.112
5.448	0.196	0.204	0.206
6.135	0.284	0.295	0.293
6.76	0.592	0.577	0.577

Wall boiling submodel setup

The wall boiling model can be set up by several of submodels and model parameters. As mentioned in the previous section, the most important parameters governing the heat partitioning model are the nucleation site density (n) and the bubble departure diameter (d_w). In this study, flowing settings of submodels for the wall boiling model have been used:

Table 3: Wall boiling model setup

Wall nucleation site density	Lemmert & Chawla model
Bubble departure diameter	Tolubinski and Kostanchuk model
Bubble detachment frequency	Terminal bubble rise velocity over the bubble departure diameter
Bubble waiting time	Tolubinski & Kostanchuk model, which sets the bubble waiting time to 80% of the time between bubble departures
Bubble diameter influence factor	Default value of 2.0
Liquid quenching heat transfer coefficient	Del Valle Kenning model
Fix Yplus for liquid subcooling	Default value of 250

V. Results and discussions

The results for void fraction computed using ANSYS CFX versus experiment distributions along the channel for six cases in two ranges of pressure are shown in Table 4. The

comparisons between void distribution calculation results of CFD and CTF [2] versus experiment data is shown in Figure 3. It is observed that, CFX void fraction distribution predictions are good agreement with distributions from experiment data in some case with almost deviation around 0.03 of void. Figure 3 shown that, with experiment void fraction less than 0.2, CFX simulation tends to give over prediction while CTF simulation tends to give under prediction. Especially, at same range of void fraction less than 0.2, with input pressure of 3 MPa, CTF simulation tends to give calculated void fraction with more accuracy than CFX results in comparison with experiment data and vice versa in pressure of 7 MPa. For the experiment void fraction greater than 0.2, both of CTF and CFX simulations tend to give over prediction. The deviations of calculation results from experiment data for both codes are nearly the same for pressure in 3MPa or 7 MPa.

The different simulation results between CTF and CFX codes for void fraction prediction can be explained as following. CTF code uses Chen's correlation to calculate the heat transfer in subcooled nucleate boiling region. Heat from the wall is transferred to liquid and then liquid enthalpy increases and phase change occurs when enthalpy exceeding saturated enthalpy. In CFX code, the wall heat flux is divided into three components and some of heat contributes to vapor production even though in subcooled nucleate boiling. Especially, CTF employed heat transfer coefficient as Chen's correlation which the calibrated pressure range from 0.17 to 3.5 MPa ([5], [6]) while most of wall boiling model in CFD has been derived for wall boiling processes under pressurized conditions so that CTF results give better void fraction predictions in range of pressure about 3 MPA and otherwise CFX give better results of void fraction prediction in range of pressure 7MPa.

Table 4. Comparisons between CFD simulation results with experiment distributions

	3 MPa						7 MPa					
Z(m)	$T01_{exp}$	$T01_{cal}$	$T04_{exp}$	$T04_{cal}$	$T14_{exp}$	$T14_{cal}$	$T18_{exp}$	$T18_{cal}$	$T24_{exp}$	$T24_{cal}$	$T25_{exp}$	$T25_{cal}$
0.385	0	0.000	0	0.000	0.002	0.000	0	0.000	0.021	0.000	0	0.000
0.948	0	0.000	0.006	0.000	0.001	0.000	0.003	0.000	0.008	0.000	0.004	0.000
1.573	0	0.000	0.015	0.000	0	0.001	0	0.000	0.004	0.000	0.003	0.000
2.322	0	0.000	0	0.000	0	0.046	0.009	0.018	0.002	0.017	0	0.000
2.947	0	0.000	0.002	0.001	0	0.193	0.089	0.085	0.066	0.074	0	0.003
4.01	0	0.047	0.002	0.109	0.24	0.371	0.275	0.208	0.278	0.223	0.065	0.127
4.823	0.027	0.113	0.043	0.230	0.484	0.516	0.405	0.313	0.435	0.326	0.197	0.247
5.448	0.178	0.204	0.136	0.287	0.594	0.632	0.485	0.428	0.468	0.408	0.332	0.307
6.135	0.493	0.292	0.299	0.359	0.646	0.721	0.553	0.549	0.484	0.495	0.511	0.401
6.76	0.635	0.577	0.472	0.496	0.718	0.773	0.612	0.625	0.534	0.564	0.65	0.540

With void fraction greater than 0.2, CTF code change heat transfer regime to saturated nucleate boiling, the bulk fluid has reached the saturated enthalpy, the heat from the wall almost provide for vapor production, the void fraction rapid increase close to experiment data.

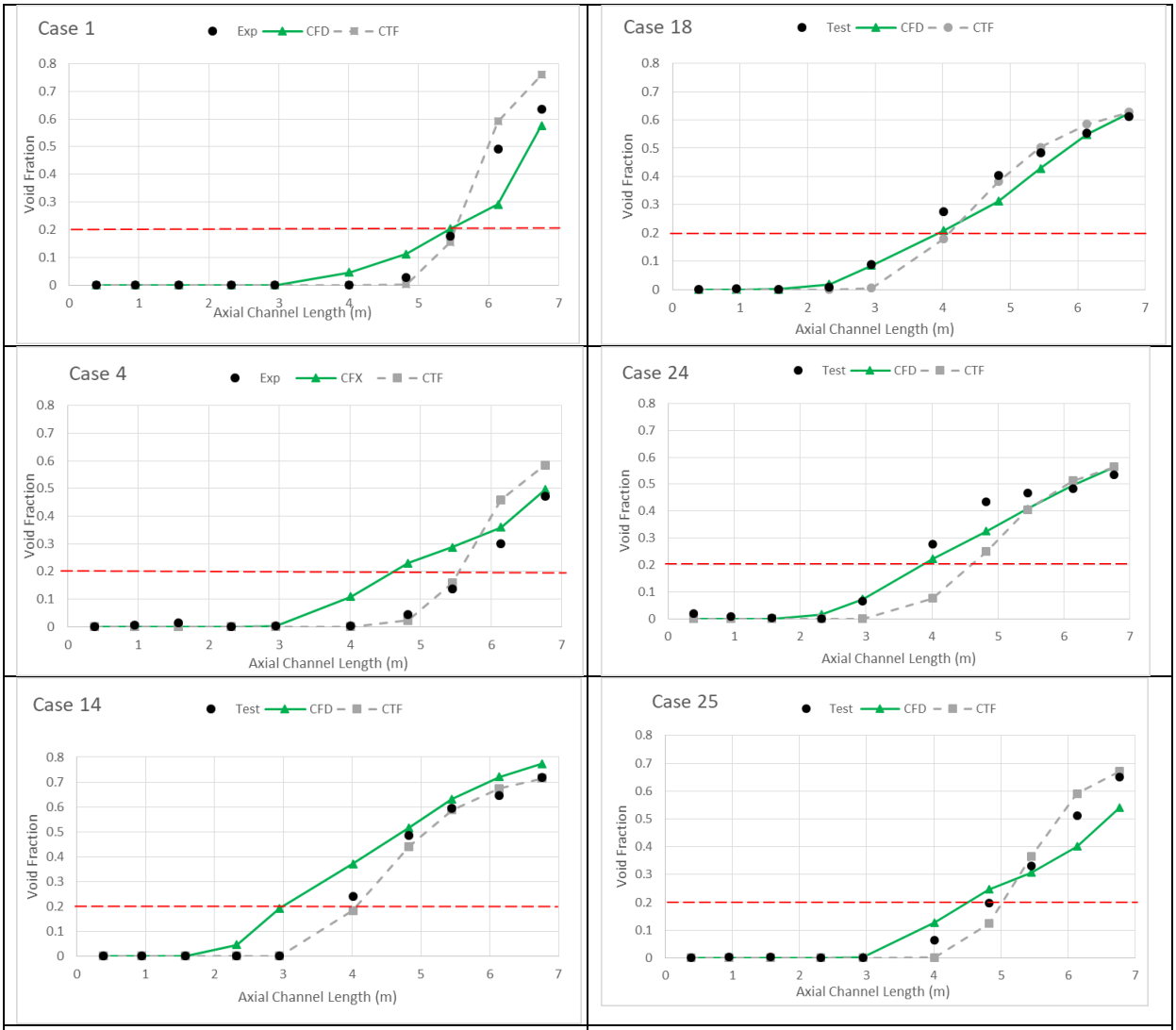

Figure 3. Comparisons between CFD and CTF simulation results with experiment distributions

As known, RPI wall boing model employed in CFX is developed for nucleate subcooled boiling. This wall boiling model may not give good void fraction prediction in case large void fraction in experiment. In case of large void existing around heated wall, the wall heat flux may be also transferred to the vapor due to convection. That means vapor can be existed in superheating condition. But in RPI model, vapor is always assumed existed in saturated condition everywhere, and no part of the wall heat flux is applied for vapor in superheating condition. This study also considers the radial channel temperature distribution at the highest measurement point of case 14 and case 24. The Figure 4 shows the radial channel temperature distribution with selected case. In both case, the calculated liquid temperature is higher than saturated temperature while void fractions are also greater than experiment measured values. It mean that RPI wall boiling model gives too much heat to liquid and the RPI model is not adequate to simulate experiment with large void fraction. An extension of RPI wall boiling model need to perform includes a fourth heat flux mechanism for convection of vapor phase as mentioned in [7].

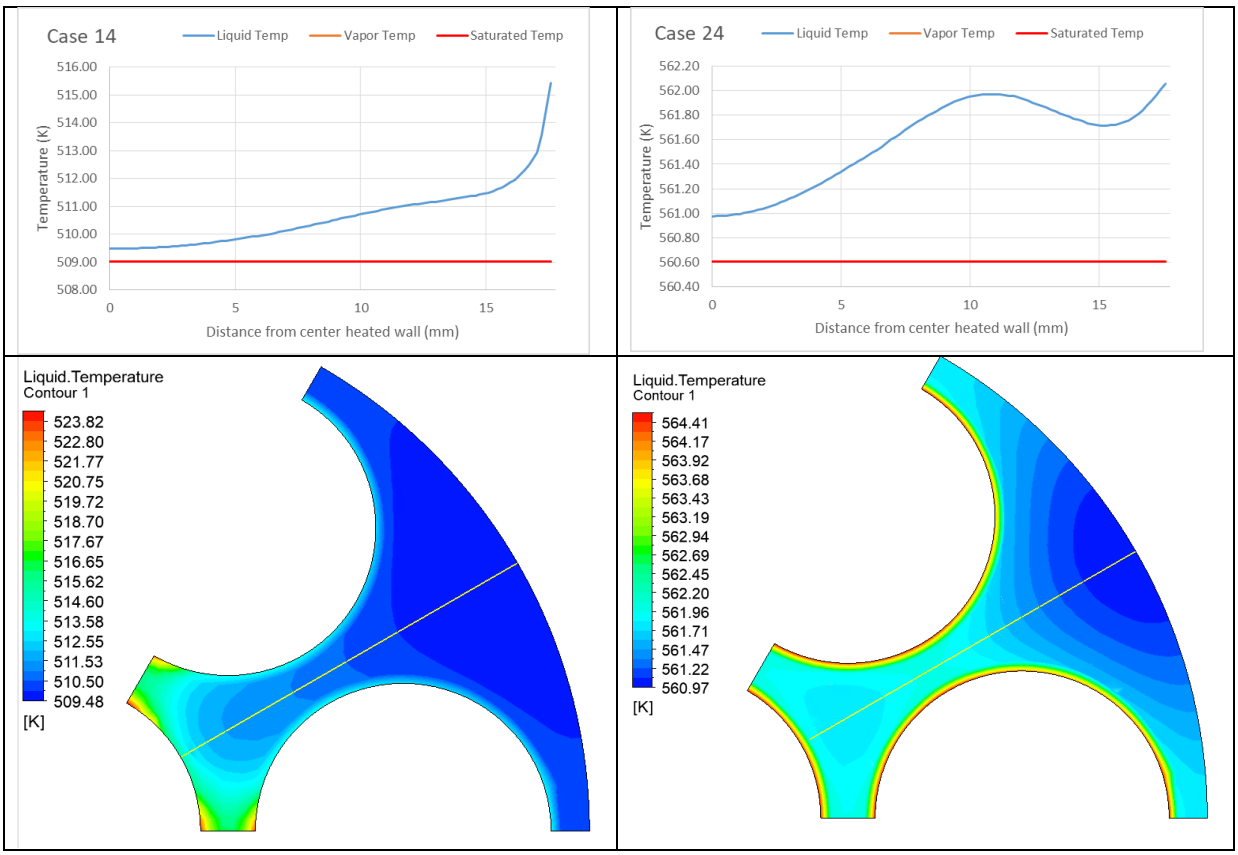

Figure 4: Temperature distribution at highest measurement point

Conclusions

In this study, some test cases of ENTEK BM facility is used to verify simulation model in CFX. In the vertical heated channel, experiment void fractions are developed to large values that are measured in 10 positions along whole channel with 7 meters high. Thus, simulation of two phase flow in this channel is very complicated due to uncertainties of parameters in several models. The multi-scale approach is used with CTF and CFX codes and the wall boiling model (RPI) is also mentioned.

The simulation results shown that with the experiment void fraction less than 0.2, CFX tends to give over prediction and CTF tends to give under prediction. For the experiment void fraction greater than 0.2, both of simulation tend to give over prediction. In comparison of simulation results to experiment data, it is observed that in test case with pressure of 3 MPa, CTF gives void fraction results more accuracy than CFX results and vice versa with pressure of 7 MPa. As explanation, the different results are depended on different boiling and condensation models in each code.

For investigation more detail of RPI wall boing model in CFX, two test cases is selected with large vapor measured. The results show that, the calculated liquid temperature is higher than saturated and vapor temperature while void fraction greater than measured values. It mean that RPI wall boiling model gives too much heat to liquid and the RPI model is not adequate to simulate experiment with large void fraction. The extension of RPI is needed to include a heat flux mechanism for convection of vapor phase in the future work.

References

[1] Dominique Bestion (2011) *Status and Perspective for a multi scale approach to Light Water Reactor thermal hydraulic simulation*. The 14th International Topical Meeting on Nuclear Reactor Thermal hydraulics, NURETH-14 Toronto, Ontario, Canada, September 25-30, 2011.

[2] Hoang Minh Giang, Hoang Tan Hung, Nguyen Phu Khanh, "Investigation of CTF void fraction prediction by RBMK experiment data", Nuclear Science and Technology, Vol.5, No.1, (2015).

[3] P. L. Garner, *RELAP5/MOD3.2 Analysis of INSC Standard Problem INSCSP-R7: Void Fraction Distribution over RBMK Fuel Channel Height for Experiments Performed in the ENTEK BM Test Facility*. United States International Nuclear Safety Center, Reactor Analysis and Engineering Division, Argonne National Laboratory, (2002).

[4] ANSYS Inc."ANSYS CFX 14.5. Theory users manual". Canonsburg, USA

[5] Robert K. Salko, Maria N. Avramova, *COBRA – TF Subchannel Thermal – Hydraulics Code (CTF) Theory Manual*, p.99, 2015.

[6] Neil E.Todreas, Muji S. Kazimi, Nuclear Systems I, p. 537 – 540, 1990.

[7] C. Lifante, Th. Frank and A. Burns, *Wall Boiling Modeling Extension Towards Critical Heat Flux*, The 15th International Topical Meeting on Nuclear Reactor Thermalhydraulics, NURETH-15 Pisa, Italy, May 12-15, 2013.