

ASSESSMENT OF INTERFACIAL DRAG AND TWO PHASE PRESSURE DROP MODELS OF TRACE CODE

S. L. Sharma, X. Yang, T. Hibiki, M. Ishii

School of Nuclear Engineering, Purdue University, West Lafayette, IN 47907-2017, USA

Abstract

The TRAC-RELAP Advanced Computational Engine (TRACE) is currently under development by the US Nuclear Regulatory Commission (USNRC) as a next-generation thermal-hydraulics safety analysis code for reactor systems. To assess the interfacial drag and two-phase pressure drop models of the code to be used for the reactor core, an assessment program was initiated by the US NRC. This paper summarizes the TRACE (V5.0P1) code assessment for the adiabatic air-water flow in an 8x8 rod bundle geometry experiment. For this purpose, PIPE Component of the TRACE code was used. A node sensitivity study was also performed to check the effect of node size on code results. Void fraction and pressure drop were the parameters used for the comparison. The range of void fraction covered for the assessment was from 0.22 to 0.88. Code predicted pressure drop and void fraction data compared well with the experimental data with mean absolute error of 13% and 16.5% respectively.

1. Introduction

The USNRC's system thermal-hydraulic analysis code TRACE (TRAC RELAP Advanced Computational Engine) is being developed to provide a best-estimate accident analysis capability for both operating pressurized and boiling water reactors as well as the next generation of evolutionary water reactor designs. The NRC has initiated an assessment program for the various models of the TRACE code. The TRACE code is based on the two fluid model [1]. The interfacial drag and two phase pressure drop models of a code decide void fraction, thus inventory in the core is predicted. Therefore, the accuracy of these models play a significant role in the thermal-hydraulics safety analysis of reactors.

The interfacial drag model of the code uses flow regime and geometric dependent correlations [1]. The interfacial drag force per unit volume implemented in the code is defined as

$$F_i''' = C_i V_r |V_r| \quad (1)$$

where C_i is the interfacial drag coefficient, and V_r is the relative velocity. For calculating interfacial drag, the TRACE code considers two broad flow regimes- "bubbly/slug" and "annular/mist". For the bubbly/slug flow regime (which includes three "bubbly" flow regimes — dispersed bubble, slug flow, and Taylor cap bubble), interfacial drag calculation is based on a drift flux formulation and includes specific correlations for both pipe and rod bundle geometry. In the case of annular/mist flow regime, the code considers interfacial drag for annular film as

well as for entrained drops. The transition between these two primary flow regimes is handled by a simple power law of the flow regime drag coefficients.

In the case of bubbly/slug flow regime, use of drift flux model for calculation of interfacial drag coefficient basically requires the specification of two parameters: void-weighted mean drift velocity, $\langle\langle v_{gj} \rangle\rangle$ and distribution parameter, C_0 . The code uses a unique specification of drift flux velocity for each of the bubbly/slug flow regimes as described below, whereas it uses one formulation for the distribution coefficient for all of the bubbly/slug regimes. The code uses a separate drift flux correlation for pipe and rod bundle geometry. In the case of dispersed bubbly pipe flow (void fraction, $\alpha \leq 0.2$) the code uses Ishii's churn-turbulent flow regime correlation [2] and for "cap/slug" regime (void fraction, $\alpha \geq 0.3$) uses Kataoka and Ishii correlation [3] in pipe flow. For transition in the void fraction range from 20 to 30%, the code uses a simple linear ramp as a function of void fraction between the limits of these two correlations. These correlations are provided within the PIPE component of the code. The code uses a single correlation for distribution parameter proposed by Ishii [2] for the entire bubbly/slug flow. For rod bundle geometry (like, in case of reactor core) the code provides another drift flux model option to users with component CHAN. It uses Bestion's drift velocity correlation [4] with constant distribution parameter value as 1.

The current pressure drop model in the code calculates the pressure drop across a grid spacer (i.e. local irreversible losses) by applying single K loss coefficient for both liquid and gas phase. Due to the importance of these models in the safety analysis calculations, it is necessary to perform an assessment of these models with two-phase experimental data.

2. Experimental Database

The schematic of the experimental test section used for generating the 1-D experimental database for the code assessment is shown in Figures 1 and 2. The test section consists of a stainless steel shell with square cross section of 140 mm inner dimension. Inside this shell are 64 stainless steel rods of diameter 10.3 mm and pitch of 16.7 mm simulating the fuel rods in a reactor system. Along the test section are seven instrumentation ports, the locations of which are given along the left side of Figure 1. Each of these locations houses an impedance void meter as well as a tap for the measurement of test section pressure. Void fraction measurements were taken at four locations: $z = 0.19$ m, 1.75 m, 1.94 m and 3.06 m. Grid spacers are located at seven locations along the test section as indicated on the right side of Figure 1.

The test matrix used for the assessment is shown in the Figure 3. The experiments were performed at 1 atm. The solid circles represent the experimental data. Color lines represent the flow regime transitions according to artificial neural network technique. The figure shows that the majority of the tests are concentrated in the transition region from bubbly flow to churn-turbulent flow. Details of the experimental setup and database can be found in the report submitted to USNRC [5].

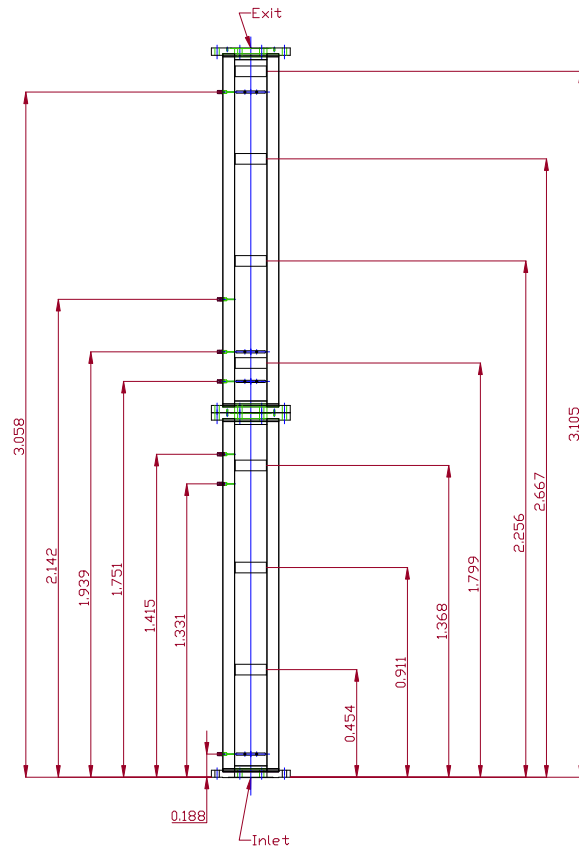


Figure 1 Schematic of test section indicating measurement locations and grid spacers.

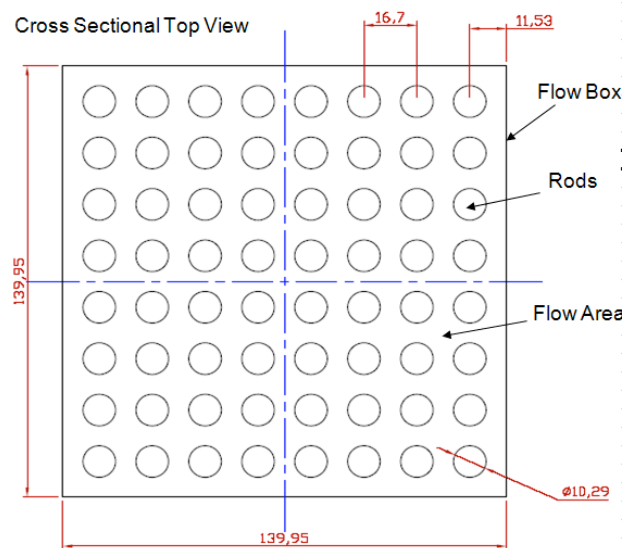


Figure 2 Cross-section of test facility.

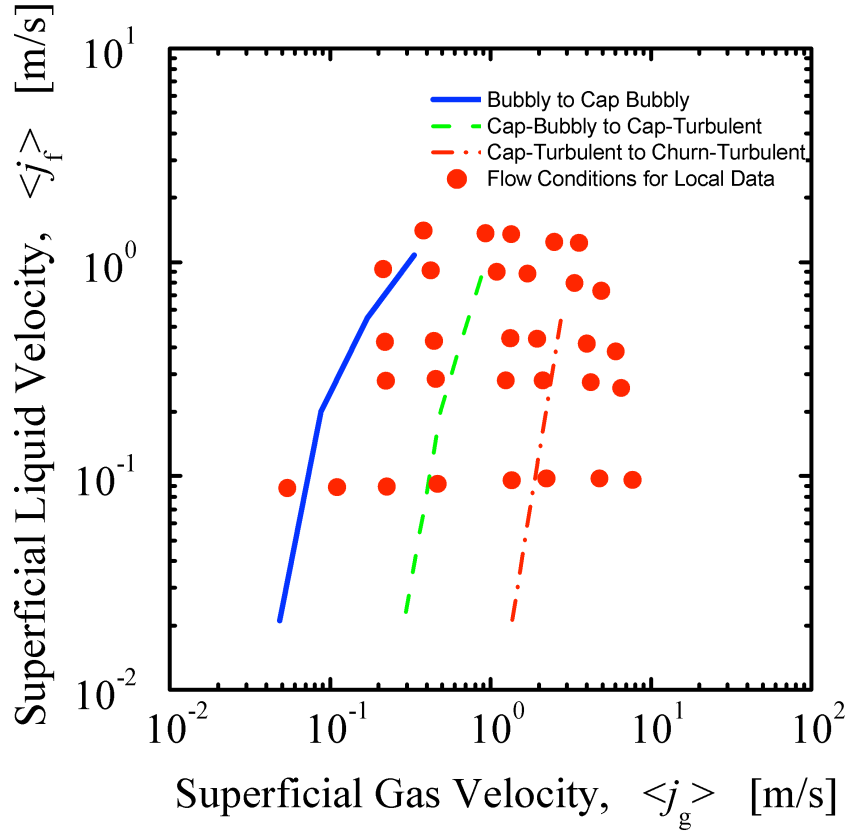


Figure 3 Test Matrix for code assessment.

3. Modelling and Nodalization in TRACE

There were two options for modeling the test section with the TRACE code. The first option was the CHAN component which uses Bestion's correlation for interfacial drag calculations in the bubbly-slug/cap bubbly flow regime. The other option was PIPE component which uses Ishii's (churn turbulent) correlation for dispersed bubbly flow and Kataoka-Ishii's correlation for slug/cap bubbly flow. In Bestion's correlation, the drift velocity is calculated as

$$\langle \langle v_{gj} \rangle \rangle = 0.188 \sqrt{\frac{g D_H \Delta \rho}{\rho_g}} \quad (2)$$

where $\Delta \rho = \rho_f - \rho_g$: density difference of liquid and gas, g : gravitational acceleration and D_H : hydraulic diameter.

The dependence of drift velocity on the gas density can lead to unrealistically high values for the drift velocity at low density ratios as shown in Figure 4. So it was decided to use the PIPE component for assessing the code.

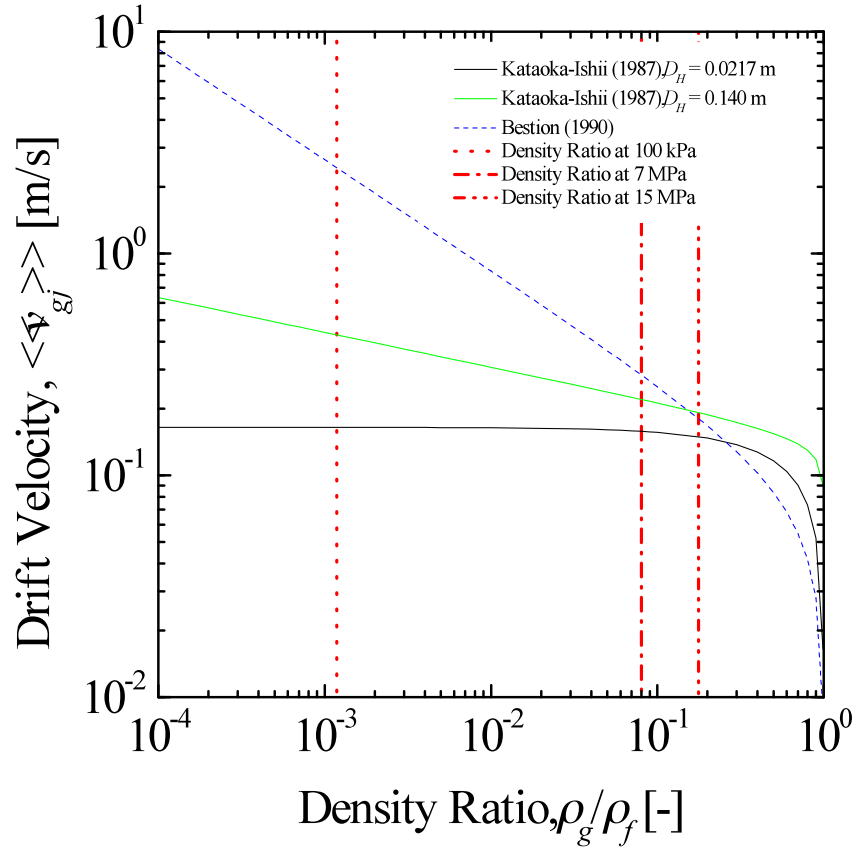


Figure 4 Variation of the drift velocity with density ratio

The nodalization diagram of the test section is provided in Figure 5. Fill and Break component elements of the code are used to simulate the inlet and outlet boundary conditions (B.C.) in the input deck. Inlet B.C. is specified as liquid velocity, gas velocity and void fraction. Outlet B.C. is specified as pressure. Based on node sensitivity (refer to Figure 6) it was decided to use 21 equal size cells for the test section. This configuration results in L_{node}/D_H ratio of 6.86 for each cell where the hydraulics diameter, D_H , for the test section is calculated as 0.0217 m. The internal junctions of the pipe with red dashed lines represent a spacer grid. The grid spacer loss is modeled by supplying a K_{loss} calculated from the following expression obtained experimentally from the single phase experiments.

$$K_{\text{loss}} = \frac{8.96}{Re^{0.0810}} \quad (3)$$

Here, for its application in a two phase flow case, the Reynolds number is calculated as

$$Re = \frac{D_H G_m}{\mu_f} = \frac{D_H (\rho_f \langle j_f \rangle + \rho_g \langle j_g \rangle)}{\mu_f} \quad (4)$$

where $\langle j_f \rangle$ and $\langle j_g \rangle$ are the superficial liquid and gas velocity respectively and μ_f is the liquid viscosity.

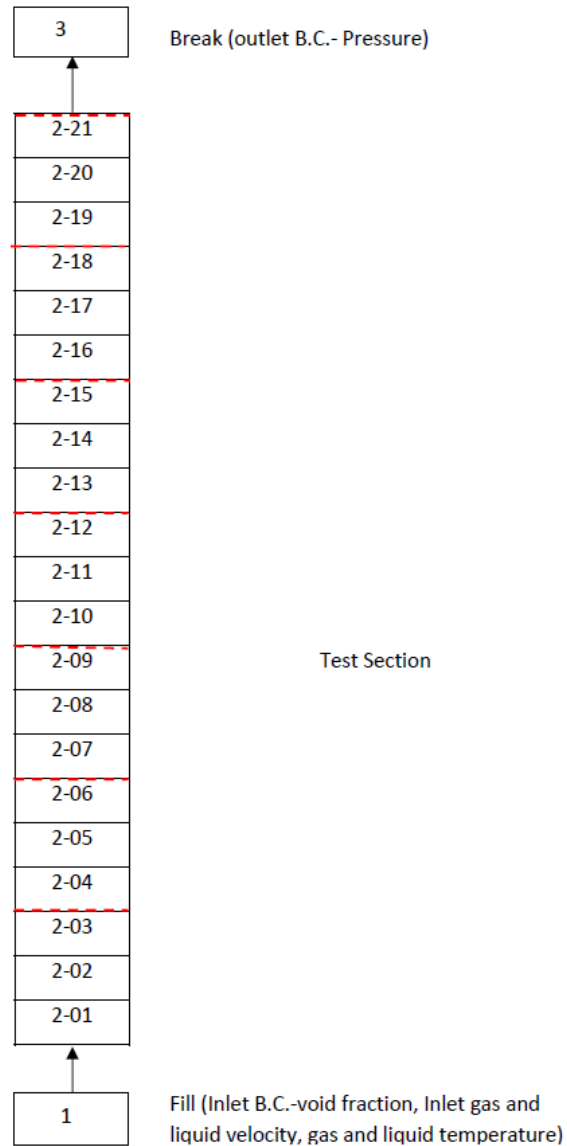


Figure 5 Nodalization diagram for the test section

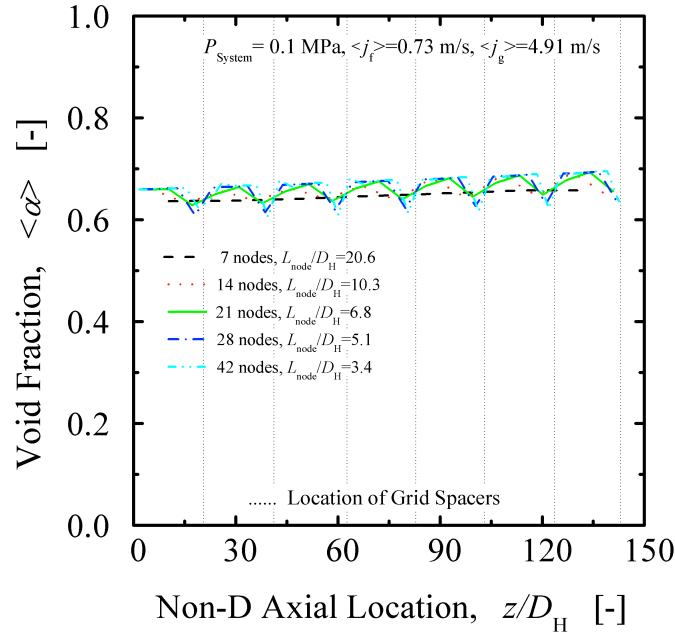


Figure 6 Node sensitivity analysis

4. Assessment Results and Discussion

Since the objective is to assess the interfacial drag model and two phase pressure drop models of TRACE, the assessment concentrated particularly on void fraction and pressure drop prediction. These parameters were available both experimentally and as TRACE code output.

The overall comparison result for the void fraction is shown in Figure 7. As the figure shows, most of the data agrees within $\pm 30\%$ with mean absolute error of 16.5%. The void fraction at the 5th port (shown by green solid triangles in the figure) shows large deviation. This measurement port is just after the grid spacer location. Code results for the effect of increasing gas flow rates for lowest and highest liquid velocity ($\langle j_f \rangle = 0.09 \text{ m/s}$, 1.32 m/s) are shown in Figures 8 and 9. Since the void fraction range is 0.22 to 0.88, so for most of the data Kataoka and Ishii's correlation has been used by the code. At lowest gas flow rates (i.e. at void fractions below 25%, bubbly flow regime), the code slightly under-predicts the void fractions for all liquid flow rates. As gas flow rate increases (cap-bubbly/cap-turbulent flow regime) the code starts over-predicting the measured data. However at the highest gas flow rates (void fraction above 0.8, towards churn-turbulent to annular transition) the prediction accuracy is good. For lower range of void fraction (corresponding to bubbly flows) and for higher range of void fraction (towards churn-turbulent to annular flow transition) the sub-channel length scale is most important. However for intermediate range of void fraction (corresponding to cap-bubbly/ cap-turbulent flow regime) the casing length scale dominates the flow. For the present assessment with hydraulic diameter of 0.0217 m, Kataoka and Ishii's correlation doesn't simulate the effect of the large scale casing (0.14 m). It has been seen from the analysis of experimental data that for the present value of hydraulic diameter (0.0217 m), which represents the sub-channel length scale, Kataoka and Ishii's correlation prediction is nearly the same as Ishii's churn flow correlation [5]. This result could be the reason for discrepancy at the intermediate range of void fraction.

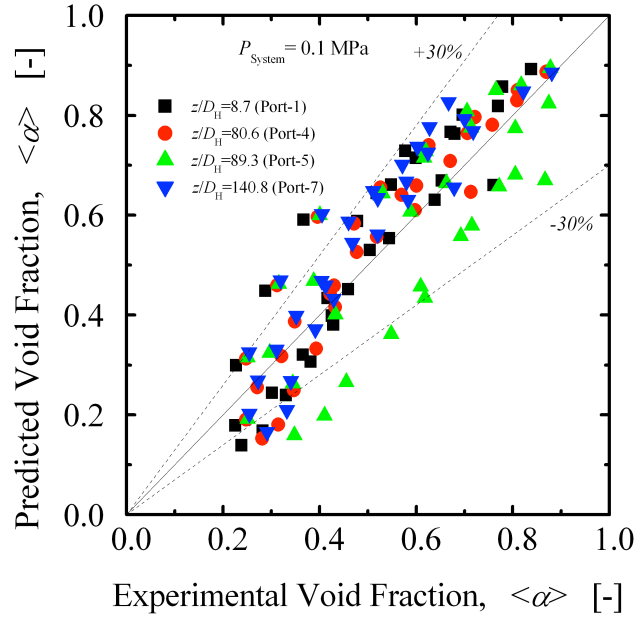


Figure 7 Overall comparisons of void fraction data

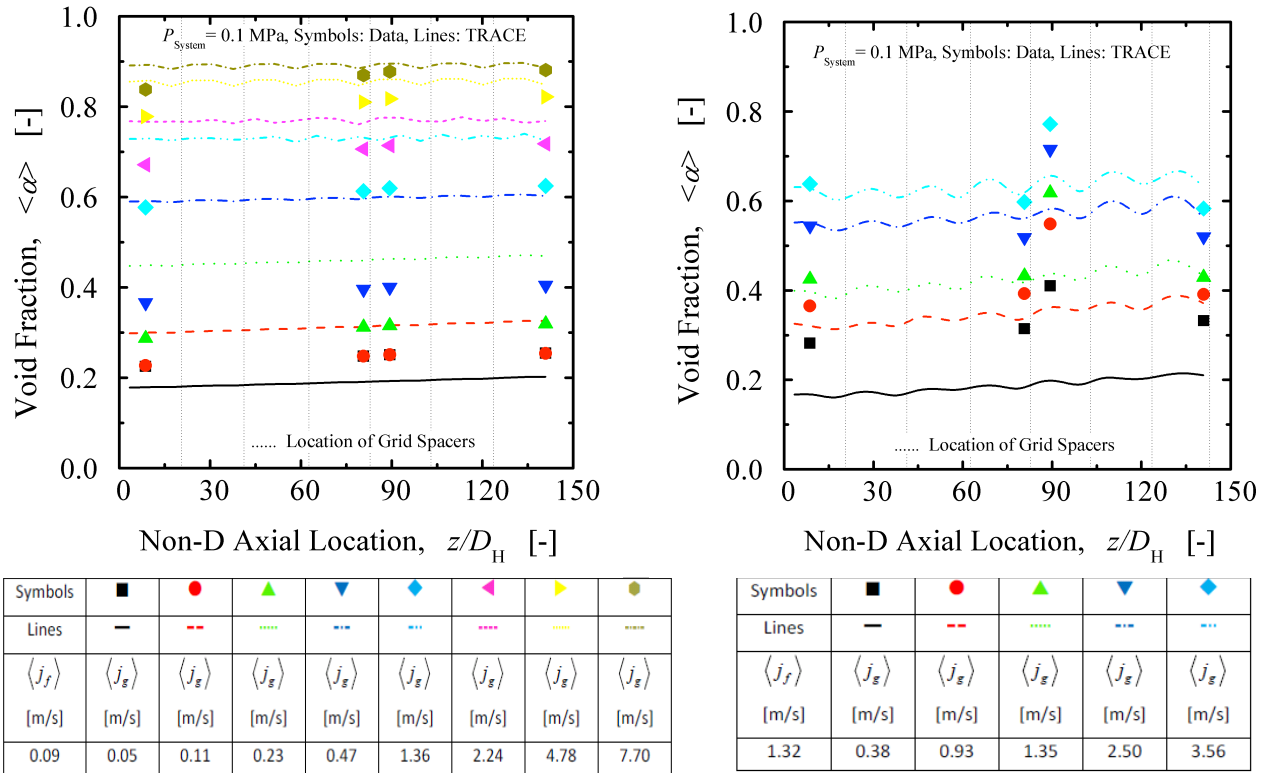


Figure 8 Assessment results for void fraction at (a) $\langle j_f \rangle = 0.09 \text{ m/s}$ (b) $\langle j_f \rangle = 1.32 \text{ m/s}$

At present there is no provision in the present version of the code to utilize the effect of these two different length scales. For the present assessment of the TRACE code, the hydraulic

diameter, D_H , of 0.0217 m which corresponds to sub-channel length scale was used. However, to see the effect of the large casing length scale, a few additional runs for the liquid flow rate, $\langle j_f \rangle = 0.09$ m/s, and in the intermediate range of void fraction were carried out with hydraulic diameter, $D_H = 0.14$ m. Results are as shown in Figure 9. It can be seen that in the intermediate range of void fraction (cap-bubbly and cap-turbulent flow regime) Kataoka and Ishii's correlation prediction is far better with the large casing length scale.

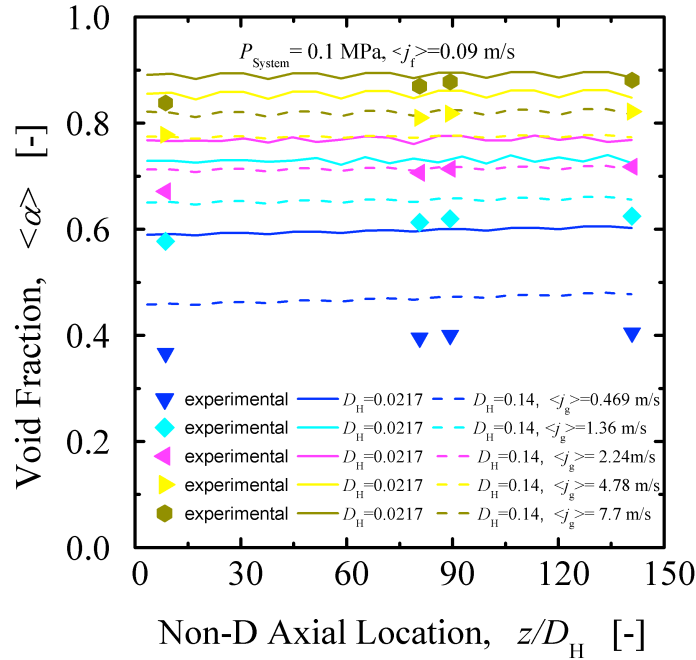


Figure 9 Effect of Length scale on code prediction for void fraction at $\langle j_f \rangle = 0.09$ m/s

The assessment results of void fraction in Figure 7 and 8 show that the code predicts slightly lower void fraction (i.e. higher slip) near the grid spacer locations resulting in wavy axial profile. With increasing gas flow rate this effect is more pronounced. Similar behavior was found by Nikitin et al [6] in their use of the TRACE code.

Figure 10 shows the pressure drop prediction for the present set of experimental data. As can be seen from the figure the overall prediction is good with mean absolute error of 13.5%. At low liquid flow rates the code under-predicted the experimental data. Code prediction improves with increasing liquid flow rate. Figures 11.a and 11.b shows the code results for the effect of increasing gas flow rates for lowest and highest liquid velocity ($\langle j_f \rangle = 0.09$ m/s, 1.32 m/s). At low liquid flow rates and low gas flow rate, pressure drop is predicted well, but as the gas flow rate increases, the code starts under-predicting the pressure drop. At much higher gas flow rates (churn-turbulent regime and towards transition to annular regime) prediction improves. However, with increasing liquid flow rate code prediction improves for all range of gas flow rates.

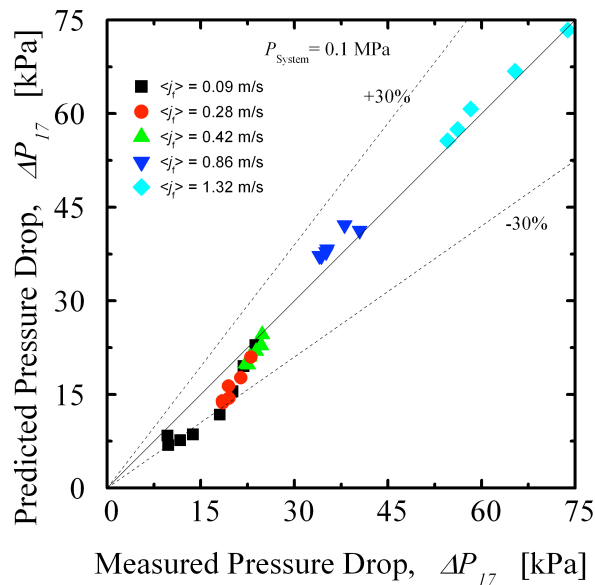


Figure 10 Overall comparisons of pressure drop data

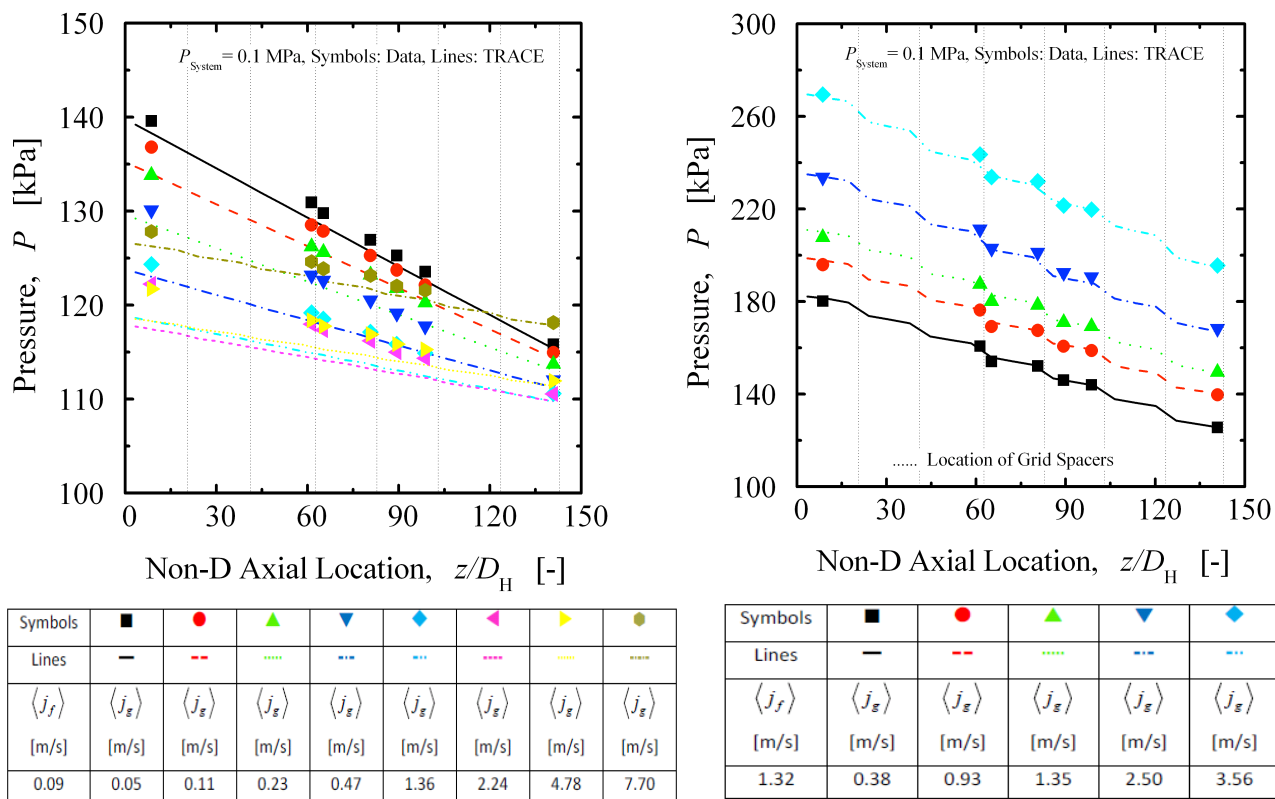


Figure 11 Assessment results for void fraction at (a) $\langle j_f \rangle = 0.09$ m/s (b) $\langle j_f \rangle = 1.32$ m/s

There could be two reasons for the discrepancy. The first reason is that the input value of the grid spacer loss coefficient is based on the measurable single-phase experimental data of Reynolds number ranging from 4000-32000 whereas the Reynolds number corresponding to the two-phase data at the lowest liquid flow rate is 1900, which is below the threshold for the laminar-turbulent transition. The other reason could be the way the present version of TRACE calculates the form loss across the grid spacer. The code applies the single-phase pressure loss coefficient to each phase to calculate form loss. A better way would be to use a two-phase multiplier along with single-phase loss coefficient [5]. It might be possible that the present approach of TRACE for calculating the two-phase form loss from the grid spacer might result in a high slip ratio near grid spacer locations which in turn causes a lower prediction in void fraction near grid spacers.

5. Conclusion

An assessment of the interfacial drag and two-phase pressure drop models of the TRACE code was performed for air-water flow test conditions in a rod bundle geometry. Since Bestion's correlation used in CHAN component of the code predicts too high drift velocity, it was decided to use PIPE component which uses Kataoka and Ishii's model. Node sensitivity was also carried out to see the effect of nodalization. Void fraction and pressure drop are the parameters used for the comparison.

The comparisons revealed that most of the void fraction predictions agree to within $\pm 30\%$ error. At low liquid flow rates and for intermediate range of void fraction (cap-bubbly/ cap-turbulent flow regime), the code over-predicts the void fraction but as liquid flow increases prediction improves. Over-prediction in the intermediate range of void fraction is attributed to the fact that the code uses only one hydraulic diameter in the calculation of void fraction. In this case the length scale is of the sub-channel of rod bundle geometry and doesn't utilize the ability of the Kataoka-Ishii's correlation to predict large length scale effect (due to casing dimension) which is important in intermediate range of void fraction. The Code predicts slightly lower void fraction near the location of grid spacer which results in its wavy axial profile. This effect gets more pronounced at high gas flow rates. Pressure drop prediction seems reasonable at high liquid flow rates. At low liquid flow rates with increasing gas flow rate, under-prediction of the pressure drop by the code is observed. This result suggests that a better model of the two-phase form loss could improve results. Also, the TRACE code uses Bestion's correlation for rod bundles (for CHAN component) which gives too high of a drift velocity. This indicates that implementation of a more accurate model could improve predictions of CHAN component.

6. Acknowledgements

This work was performed under the auspices of the U.S. Nuclear Regulatory Commission, Office of Nuclear Regulatory Research, through the Institute of Thermal Hydraulics at Purdue University.

7. References

- [1] TRACE V5.0 Theory Manual, Division of System Analysis, Office of Nuclear Regulatory Research, U.S. Nuclear Regulatory Commission, Washington, DC., 2008.

- [2] M. Ishii, “One-dimensional drift-flux model and constitutive equations for relative motion between phases in various flow regime”, ANL-77-47, USA, 1977.
- [3] I. Kataoka, M. Ishii, “Drift flux model for large diameter pipe and new correlation for pool void fraction”, *International Journal of Heat Mass Transfer*, Vol. 30, No. 9, 1987, pp. 1927-1939.
- [4] D. Bestion, “The physical closure laws in the CATHARE code”, *Nuclear Engineering and Design*, Vol. 124, 1990, pp. 229-245.
- [5] M. Ishii, T. Hibiki, Y. Liu, S. Paranjape, X. Yang, J.P. Schlegel., “Task order 2.-Subtask 2: void fraction and grid loss database”, Purdue University Report PU/NE-10-16, 2010.
- [6] K. Nikitin, J. Judd, G. M. Grandi, A. Manera, H. Ferroukhi, “Peach Bottom 2 Turbine Trip 2 Simulation by TRACE/S3K coupled code”, Proc. PHYSOR 2010 – Advances in Reactor Physics to Power the Nuclear Renaissance, Pittsburgh, Pennsylvania, USA, May 9-14, 2010.