Coupled Heat Transfer with Thermal Phase Change Using a Generalized Multiphase Modelling Approach

Hanan Aburema[1, a)], Bruce Hanson[1, b)], Michael Fairweather[1, c)] and Marco Colombo[2, d)]

[1] School of Chemical and Process Engineering, University of Leeds, Leeds, LS2 9JT, United Kingdom
[2] Department of Mechanical Engineering, University of Sheffield, Sheffield, S1 3JD, United Kingdom

[a)] pmhaa@leeds.ac.uk
[b)] b.c.hanson@leeds.ac.uk
[c)] m.fairweather@leeds.ac.uk
[d)] m.colombo@sheffield.ac.uk

Abstract. Multiphase flows enhance heat and mass transfer due to the large interfacial areas between the phases. However, these systems are complex, with diverse flow regimes depending on phase distribution patterns. In thermal hydraulic systems, these regimes range from dispersed to segregated/large interfaces. Traditional multiphase predictive models use either interface-averaging for dispersed flows or interface-resolving for large interfaces. However, these complex systems require a comprehensive model capable of addressing all flow regimes. To address this need, the Generalized Multifluid Modelling Approach (GEMMA) was developed in conjunction with OpenFOAM. GEMMA combines interface-resolving capabilities for large interfaces and reverts to standard multifluid methods for small or dispersed interfaces. This model was further enhanced in this work to predict phase-change heat transfer in turbulent flows, using specific heat transfer closures for each flow regime. Condensation correlations based on the surface renewal assumption and developed for condensation in stratified flows were applied in segregated/large interface regime to calculate the interfacial heat transfer coefficient, coupled with large eddy simulation (LES) to account for turbulence in the continuous phase. The accuracy of GEMMA is validated through comparison with experimental tests at the COndensation at Safety Injections (COSI) facility, which studies complex condensation phenomena during emergency core cooling injection in a simulated loss-of-coolant accident in a pressurized water reactor. The model successfully predicts the temperature distribution of the fluids under different water conditions and levels, with condensation correlations for large interfaces and LES for turbulent flow providing well-predicted temperature distributions in the COSI system. This work confirms GEMMA as a comprehensive predictive tool for heat transfer and thermal phase change in multiphase flows, capable of simulating complex multiscale flows of industrial interest.

INTRODUCTION

Multiphase thermo-fluid dynamics fundamentally involves the exchange of mass, momentum, and energy between phases at their interfaces. The efficiency of these transport mechanisms depends significantly on the interfacial area, which spans a large range of length scales. Historically, multiphase flows have been categorized into two distinct flow regimes: dispersed, such as gas bubbles or liquid droplets, and segregated/large regimes, such as stratified flows where fluids of different densities flow in distinct layers. In thermal hydraulic systems, smooth stratified flows are rare, and the interface is often turbulent due to phenomena like bubble trapping, displacement, and deposition of liquid droplets [1]. Accurately modelling these thermo-fluid dynamics is crucial as it enhances the accuracy and reliability of safety analyses, particularly in scenarios involving Loss-of-Coolant Accidents (LOCA), where accurate predictions of the fluid temperature distribution are crucial for evaluating structural integrity. This distribution is influenced by Direct Contact Condensation (DCC) between the cold water from the Emergency Core Cooling (ECC) system and the hot primary fluid. Although numerous experiments have been conducted to investigate this scenario in detail, it is challenging to obtain complete information due to the difficulty in measuring the shape and area of the interface.

Several studies have focused on simulating the conditions of the COndensation at Safety Injections (COSI) experiments to improve predictive models. Lakehal [2] proposed a Large Eddy and Interface Simulation (LEIS) model based on an interface tracking (level set) approach combined with the prediction of large-scale turbulence. The application of LEIS is limited to flows with moderate Reynolds numbers, and in the very large eddy simulation approach, the blending between unsteady Reynolds-Averaged Navier Stokes and large eddy simulation (LES) predictions is based on a filter width larger than the grid size, proportional to a characteristic length scale of the flow [3]. Mérigoux et al. [4] applied a Reynolds stress turbulence model to thermal free surface flows under the industrial conditions of the COSI experiments, coupled with the large interface model. Also, more recently, Raverdy [5] conducted sensitivity studies on DCC and turbulence models using the NEPTUNE CFD code, emphasizing the need for a low-turbulence heat transfer model and optimized numerical mesh.

Building on these efforts, the Generalized Multifluid Modelling Approach (GEMMA) has been further developed to predict phase-change heat transfer in turbulent flows for all flow regimes. Interphase heat transfer correlations, based on surface renewal theory, are used to predict the heat transfer coefficient at the large interface. Below, GEMMA is validated against different COSI experiments, showing strong agreement with experimental data and the predicted results of Mérigoux et al. [4], proving its capability to simulate complex multiscale flows in industrial applications.

MODEL DESCRIPTION

The GEMMA model extends the standard multifluid modelling framework, designed for small/dispersed interfaces in the OpenFOAM code, by integrating interface-resolving capabilities in cells with large interfaces. Within each computational cell, GEMMA selects between two approaches based on the grid's ability to resolve interface morphology. For regions without interfaces above the grid scale, the Eulerian-Eulerian method is employed, while a novel multifluid approach, simulating interface-resolving behaviour like the Volume of Fluid (VoF) method, is used for areas with large interfaces. A detailed description of the model can be found in [56. The GEMMA model has been rigorously evaluated against several fundamental test cases for adiabatic systems [6, 7], and the results demonstrated its comparable accuracy to the VoF approach for cases with large or segregated interfaces, while also exhibiting standard multifluid behaviour in dispersed flows. In this work, the model has been further developed to predict heat transfer during thermal phase changes, utilizing specific heat transfer closures for each flow regime in the modelling of DCC, where interfacial heat and mass transfer occur due to steam condensation on cold water surfaces. The Hughes and Duffey [8] correlation, coupled with LES, was implemented and applied in large interface regimes to address turbulent interactions in the continuous phase during phase-change heat transfer. Hughes and Duffey's [7] approach, based on Higbie's penetration theory, explains how turbulent eddies from the bulk fluid renew and exchange energy at the interface through molecular diffusion over a characteristic residence time [8, 9]. Integrating this with LES, which resolves large-scale turbulent structures while modelling smaller scales, enhances the model's accuracy by capturing turbulent fluctuations and their impact on the heat transfer dynamics at the phase interface.

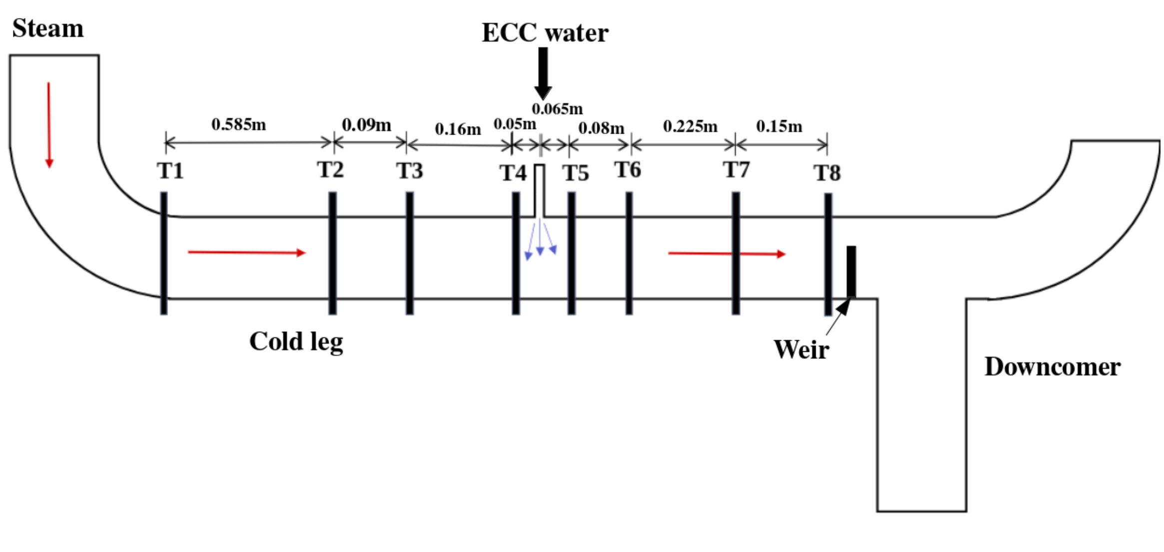
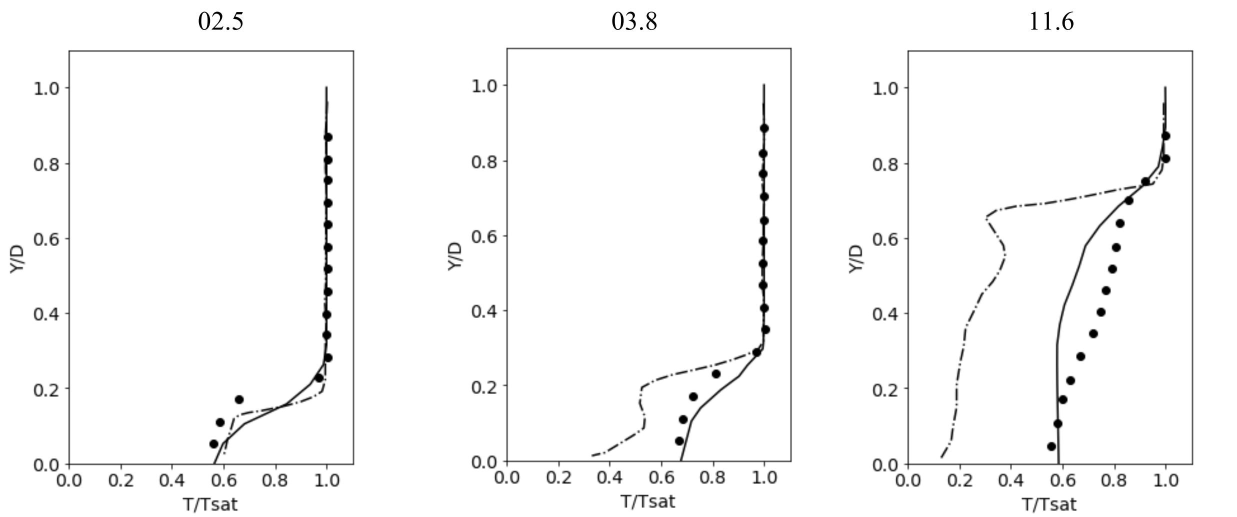


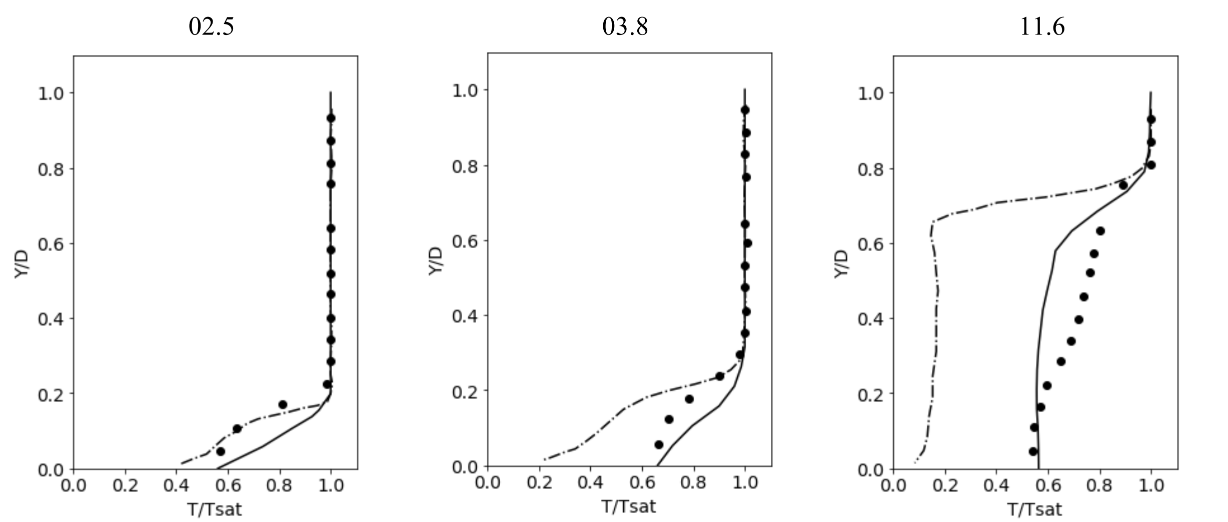
FIGURE 1. Schematic view of the COSI experiment with positions of thermocouple lines along the test section with a weir.

VALIDATION CASES

The COSI facility is designed to simulate LOCA scenarios through various steady-state and transient experiments [10]. This setup features a horizontal pipe representing the cold leg, an injection nozzle located at the centre of the test section, and a downcomer modelled by a cylindrical structure. Thermocouples are distributed along eight linear arrays in the cold leg: four before the ECC injection point and four between the ECC and a weir, as illustrated in Fig. 1. Steam is supplied to one end of the test section, while the opposite end can be either partially open or closed, depending on the specific test conditions. A weir can be installed downstream of the cold leg before the downcomer to control water levels. For model validation, three steady-state tests were selected: one involving a 0.6 diameter high weir (run 11.6) and two without a weir, differing in water and steam conditions (runs 02.5 and 03.8).



(a)



(b)

FIGURE 2. Normalized temperature distribution along the cold leg (a) before the water jet at T4, and (b) after the water jet at T5 for cases 02.5, 03.8 and 11.6; (•) experiment, (—) GEMMA model and (– · –) Mérigoux et al. [3].

RESULTS AND DISCUSSION

To evaluate the model's accuracy in predicting temperature distributions within the cold leg section during LOCA scenarios, as examined in the COSI experiments under different conditions, local temperature measurements are considered. The goal is to predict the temperature distribution across the section diameter at specific locations. Fig 2 shows model predictions of normalized temperature at each point across the diameter, vertically normalized by the pipe diameter, compared with measurements taken during the experiments near the water jet at points T4 and T5 (Fig. 1). Predictions at the remaining points are not included due to space limitations. In run 11.6, the water jet is under the free liquid surface due to the maintenance of a high water level by the presence of the weir, so in this case there is no direct condensation between the jet and steam. Although the temperature profile along the cold leg section is adequately predicted for this case, there is a notable challenge in replicating steep temperature gradients observed near the water jet region in the experiment. This discrepancy may stem from the complex interaction of local flow dynamics or turbulence patterns induced by the submerged jet. When the water jet is submerged, therefore, it can induce different water distribution patterns compared to when it is above the water level. Mérigoux et al. [3] noted similar issues, as shown by their predictions in Fig. 2, attributing them to an under-estimation of the turbulence intensity at the free surface when the steam velocity is high, leading to an under-estimation of steam condensation. In runs 02.5 and 03.8, the predicted temperatures closely approximate the experimental values, with slight underestimations observed in the temperature distribution after the jet at T5. Higher water flow rates in run 03.8 result in more consistent temperature profiles. Conversely, lower water flow rates in run 02.5 lead to underestimated temperature profiles in the jet region. Mérigoux et al. [3] found that their model predictions were more accurate at lower flow rates, suggesting that improvements in modelling the wave influence on condensation at the free surface are necessary. Nevertheless, the predictions of the GEMMA model shown in Fig. 2 illustrate good agreement with the experimental data used for validation purposes.

CONCLUSIONS

The Generalized Multifluid Modelling Approach developed in conjunction with OpenFOAM has proven effective in simulating complex multiphase flows and phase-change heat transfer. GEMMA integrates interface-resolving capabilities for large interfaces with traditional methods for dispersed flows, making it adaptable for different flow conditions typical in thermal hydraulic systems. Validated against data obtained at the COSI facility, GEMMA accurately predicts temperature distributions across various operational scenarios, closely matching experimental results and accurately predicting the steam-water interface. Overall, GEMMA is shown to be a reliable tool for predicting heat transfer and phase change in industrial multiphase flows, thereby enhancing its usefulness in performing safety assessments for nuclear reactor applications.

REFERENCES

1. P. Coste, A largeninterface model for two-phase.Nucl. Eng. Des. **255**, 38–50 (2013).
2. D. Lakehal, LEIS for the prediction of turbulent multifluid flows applied to thermal-hydraulics applications, Nucl. Eng. Des. **240**, 2096–2106 (2010).
3. D. Lakehal and M. Labois, A new modelling strategy for phase-change heat transfer in turbulent interfacial two-phase flow, Int. J. Multiph. Flow **37**, 627–639 (2011).
4. N. Mérigoux, J. Laviéville, S. Mimouni, M. Guingo and C. Baudry, Reynolds stress turbulence model applied to two-phase pressurized thermal shocks in nuclear powerplant, Nucl. Eng. Des. **299**, 201–213 (2016).
5. B. Raverdy, Two-phase CFD simulations of COSI tests and evaluation of condensation distribution. Nucl. Eng. Des. **416**, 112788 (2024).
6. A. De Santis, M. Colombo, B. C. Hanson and M. Fairweather, A generalized multiphase modelling approach for multiscale flows, J. Comp. Phys. **436**, 110321 (2021).
7. M. Colombo, A. De Santis, B. C. Hanson and M. Fairweather, Prediction of horizontal gas-liquid segregated flow regimes with an all flow regime multifluid model. Processes **10**, 920 (2022).
8. E. D. Hughes and R.B. Duffey, Direct contact condensation and momentum transfer in turbulent separated flows, Int. J. Multiph. Flow **17**, 599–619 (1991).
9. R. Higbie, The rate of absorption of a pure gas into a still liquid during short periods of exposure, AIChE Jl. **31**, 365–389 (1935).
10. A. Janicot and D. Bestion, Condensation modeling for ECC injection. Nucl. Eng. Des. **145**, 37–45 (1993).