Proceedings of the 18th UK Heat Transfer Conference
9-11 September 2024, Birmingham
UKHTC2024-035

SPECIAL SESSION: NUCLEAR THERMAL HYDRAULICS

MODELLING SPRAYS OF LIQUIDS INTO GASES AND VAPOURS

Amy B. Jardine[1*], Prof. Hector Iacovides[1], Dr. Timothy Craft[1]

[1]Thermo-Fluids Research Group, School of Engineering, University of Manchester, Manchester, M13 9PL, United Kingdom

1. ABSTRACT

This paper presents comparisons of Computational Fluid Dynamics (CFD) techniques for modelling the flow, heat and mass transfer of pressuriser sprays. Experimental data obtained in a pressuriser test-rig facility at the University of Manchester has been used to validate a range of Eulerian, multiphase modelling techniques with a focus on establishing methods for modelling sprays in an industrial context. The two fluid method was found to recreate the unique shape of sprays driven through a plain orifice with cross wires when applied to a RANS or LES approach, showing promise in the modelling of sprays in the nuclear industry.

2. INTRODUCTION

The pressuriser is a vessel partially filled with liquid water and steam, and it maintains the pressure of the primary circuit of a pressurised water reactor. Pressure is raised within the pressuriser vessel via steam generation, and is decreased by condensing the steam bubble in the vessel by spraying water from the cold leg of the circuit into the vessel. Computational fluid dynamics (CFD) can be used understand design functionality, in order to improve the accuracy of claims of nuclear power plant performance, safety, lifetime and costs. There is currently little data published on the modelling of pressuriser sprays. A pressuriser test-rig facility has been constructed at the University of Manchester in order to gain an insight on spray flow phenomena for a range of operating conditions relevant to the pressuriser spray. The spray nozzle of the pressuriser test rig is a simple orifice with cross wires, which has not been previously studied. Many spray modelling techniques published in the open scientific literature use highly simplifying, empirical models tailored to internal combustion sprays that are characteristically very different to that of the pressuriser. This is due to the relatively weak pressure gradients that drive the pressuriser spray, in comparison to those which drive the fuel sprays in an engine. Multiphase modelling techniques that are applicable to any flow problem involve the resolution of flow quantities, and are often not practical to implement in most industrial flow problems due to high computational cost and lead times. The aim of this work is to establish industrially-applicable multiphase techniques that can capture the flow, heat and mass transfer of the pressuriser test-rig sprays. Predictions of volume fraction, spray velocity, turbulence quantities, phase temperature and condensation rate are compared to the experimental test-rig data.

*Corresponding Author: amy.jardine@manchester.ac.uk

3. METHODOLOGY

In the pressuriser test-rig experiments conducted by Cooper [1], a plain orifice nozzle was chosen to produce sprays. The pressuriser test-rig is a chamber filled with humid air, into which cold water is injected. The experiments produced jets with little fragmentation in the relevant range of injection pressures, and little to no heat or mass transfer between the water and air was recorded. Cooper introduced cross wires to the nozzle, which aided the process of droplet production, as shown in Fig. 1, and importantly, the condensation of the humid air. In the sprays produced by the cross wires, four droplet streams were produced as well as a central stream along the nozzle axis. Although the cross wires promote atomisation, the sprays are still very different to those of internal combustion sprays. This work focuses on modelling sprays driven by a 100-400kPa pressure difference across 2mm-4mm plain orifice nozzles with cross wires.

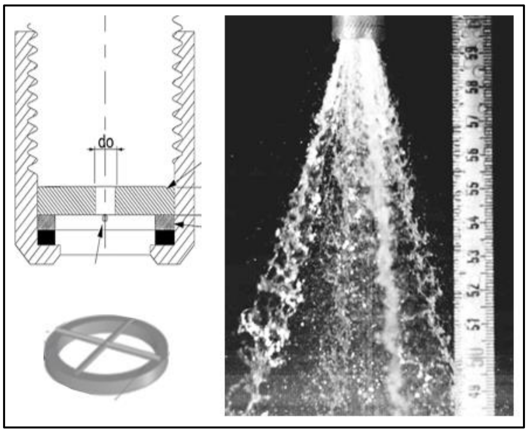

Fig. 1. Schematic of the plain orifice nozzle with cross wires and the 400kPa spray through a 4mm orifice.

A 3D mesh was constructed to replicate the geometry of the test rig. Using the commercial CFD code STAR-CCM+, two Eulerian multiphase modelling methods were tested: the Volume Of Fluid (VOF) method and the two-fluid approach. In the two-fluid method, flow transport equations are solved for each phase. The method is suitable for a wide range of flows, but the VOF method is a computationally less demanding, simplification of the two-fluid model. In the VOF, a single set of transport equations is solved to describe a two-phase flow on the condition that phase volume fractions occupying a cell sum to unity. VOF is suited to flows in which fluid interfaces are resolved by the computational grid. The use of Reynolds Average Navier-Stokes (RANS) and Large Eddy Simulation (LES) were compared, given the role which turbulence is known to play in the atomisation process. RANS models account for the effect of the fluctuating flow structures by solving for averaged quantities. This technique is widely used in industrial flow problems and is appropriate for the majority of applications. LES resolves some of fluctuating flow structures, and is often considered too time-consuming and computationally expensive to be employed in industrial flow problems. However, due to the role of turbulent eddies in the process of primary atomisation, it was considered possible that such eddies should be resolved to predict spray breakup. In the modelling of condensation in the test-rig, the two-fluid approach was used with a droplet evaporation model. In the two-fluid approach chosen, the entire liquid phase is assumed to be composed of droplets, and as such, models that were designed for dispersed flows were used to predict drag, heat and mass transfer between phases. Thus, this work explores the applicability of models designed for uniformly sized, spherical droplet sprays to flows with non-spherical liquid fragments with a range of sizes. The rate of condensation or evaporation of droplet flows involving gas mixtures is predicted based on the departure from equilibrium conditions.

4. RESULTS

The VOF method and the two-fluid method were applied to both RANS and large eddy simulations of the isothermal pressuriser test-rig flows. VOF was found to be unsuitable for capturing spray breakup, though it did show jet disturbances when LES was employed, implying that a significantly finer mesh may capture the splitting of jet into the four droplet streams observed in the experiments. However, employing an even finer mesh is not practical for industrial applications. The two-fluid RANS framework captured the spray topology well, but only when a grid and time step sizes comparable to those required by LES was employed.

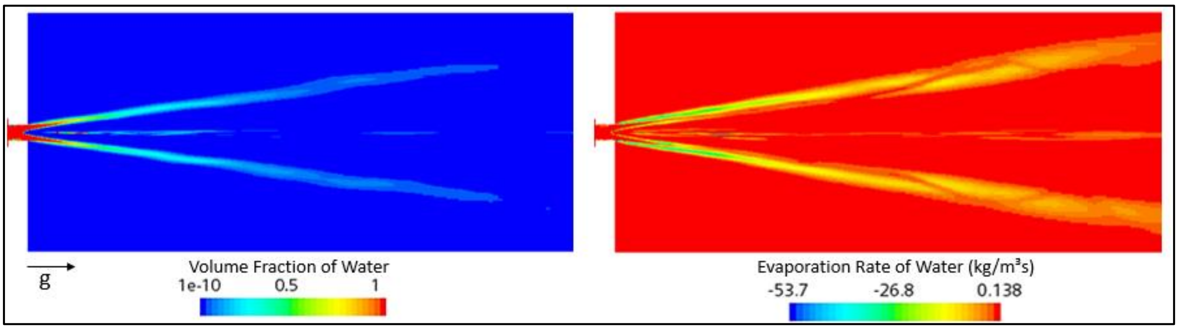

Fig. 2. Scenes of volume fraction field (left) and the evaporation rate of water (right) for the 400kPa spray through a 4mm orifice.

As LES is computationally cheaper than RANS for a given mesh and time step size due to the fewer transport equations, LES was used in the heat and mass transfer modelling of the pressuriser test-rig. Fig. 2 shows the volume fraction field (left) and evaporation rate (right). The figure shows a central plane through the 3D domain. The four radial droplet streams and the central stream of droplets that are observed in the pressuriser test rig experiments have been reproduced. However, as will be made clear in the conference presentation, the spray angle was consistently under-predicted in the computations for the range of operating conditions. In addition, the predicted spray angle increased only with orifice diameter, but not with driving pressure, as observed in the experiments. The negative values on the right hand scale of Fig. 2 represent the condensation rate of water.

Fig. 3 shows the decrease in test-rig chamber temperature over time for various orifice diameters and injection pressures. The behaviours of the computed sprays mimicked those of the experiments with the exception that the 4mm, 100kPa spray showed better cooling and condensing capabilities than the 2mm, 400kPa spray in reality, having a higher mass flow rate. In the two fluid approach, a constant diameter was defined based on a mean experimental value for each of the sprays. It is possible that the model is more sensitive to specified droplet diameter than the spray flow rate.

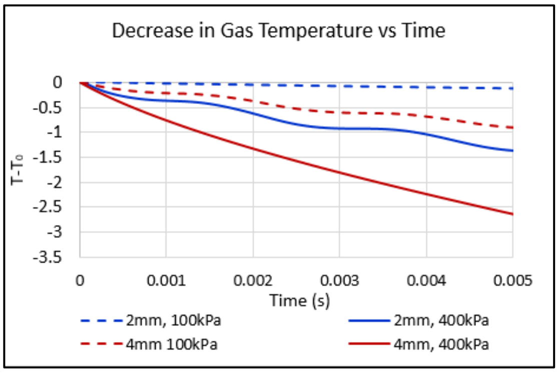

Fig. 3. Decrease in gas phase temperature vs time for various orifice diameters and injection pressures.

5. CONCLUSIONS

This abstract has presented the application of multiphase techniques to the modelling of liquid sprays that are not highly atomised in nature due to their modest driving pressures. The two fluid approach replicates the unique spray topology well, despite under-predicting spray angle. The work using evaporation models offers insight on the applicability of constant diameter droplet assumptions to low-atomised sprays. The conference presentation will reveal more on the behaviours of the multiphase modelling approaches for the range of injection pressures and nozzle diameters, with comparisons to behaviours observed in the test-rig experiments.

ACKNOWLEDGEMENTS

Acknowledgements are given to Rolls-Royce PLC, GREEN CDT and EPSRC for the support and funding of this work.

REFERENCES

[1] Cooper, D. *Internal communications,* Manchester, 2023.