Modelling of heat transfer and phase change for non-isothermal compressible two-phase flows using OpenFOAM integrated with machine learning.

Laila Abu-Farah[1], Gokul Siddarth Mani Sakthi[1], Natalie Germann[1]

[1]) Chair of Process Systems Engineering, Faculty 4 – Energy-, Process-, and Bio-Engineering, University of Stuttgart, Germany

The superheated steam dishwasher is a promising alternative to conventional dishwashers, especially in professional environments where hygienic safety and cleanliness are important, such as hospitals and restaurants. Further, it can significantly reduce water consumption and cleaning time without the use of chemical detergents. Computational fluid dynamics simulations of an idealized 3D dishwasher operated with superheated steam at 180 °C and 10 bar were carried out in our recent research work on innovative superheated steam dishwashers [1]. The heat transfer with phase change was investigated using the interThermalPhaseChangeFoam solver, which is a new two-phase VOF solver based on OpenFOAM [2]. This solver is coupled with bacteria inactivation kinetics, described using an Arrhenius-type equation. The simulation results confirmed that potentially harmful heat-resistant bacteria can be killed in a short time span of 25 s. One drawback of the InterThermalPhaseChangeFoam solver used in our simulations is that it assumes constant material propertites of the two phases and only accounts for compressibility effects due to phase change. To accurately predict the heat transfer and the compressibility effects, we would extend the current solver to a compressible flow case. Additional terms related to compressibility appear in the transport equations. To account for the dynamic changes in material properties due to pressure and temperature variations, thermodynamic library data will be integrated into a computational fluid dynamics (CFD) code. A polynomial approximation for the variation of thermodynamic properties of the materials with temperature and pressure using CoolProp library [3], an open-source thermophysics library, will be considered. The modified CFD solver is linked to real thermodynamic library data and combined with a machine learning algorithm to determine the temperature from the solution of the transport equations. An equation of state is no longer required to close the system of transport equations, and the temperature can be determined from the machine learning model using the solutions of the conservation equations at each time step. Machine learning model would be developed and trained with the data from the thermodynamic library, CoolProp for predicting the temperature from the data of total pressure, total enthalpy, and phase fractions. Machine learning model would be integrated with OpenFOAM by using python. Executing python codes of machine learning algorithms from OpenFOAM is carried out using an open-source library, pybind11 [4,5]. Integrating this library facilitates the transfer of field data (presure, enthalpy and phase fraction) from OpenFOAM to python and vice versa. This work explores the use of data-based models in multi-phase compressible flows and the development of a compressible solver for two-phase flow modelling which will help in accurately predicting the temperature and heat transfer due to condensation in many steam applications.

References

[1] L. Abu-Farah and N. Germann, “Simulations of thermal phase changes and bacterial inactivation in a superheated steam dishwasher”,*Physics of Fluids* , vol. 34, 085137, 2022. Featured Article.

[2] M. Nabil, Alexander S. Rattner, “InterThermalPhaseChangeFoam—A framework for two-phase flow simulations with thermally driven phase change,”*SoftwareX*, vol. 5, pp. 216-226, 2016.

[3] I. H. Bell, J. Wronski, S. Quoilin, and V. Lemort, “Pure and pseudo-pure fluid thermophysical property evaluation and the open-source thermophysical property library CoolProp.” *Industrial Engineering Chemistry Research* , vol. 53, no. 6, pp. 2498-2508, 2014.

[4] W. Jakob, J. Rhinelander, and D. Moldovan, “*pybind11 – seamless operability between c++11 and python* ”, 2017, https://github.com/pybind/pybind11.

[5] S. Rodriguez, and P. Cardiff. “A General Approach for Running Python Codes in OpenFOAM Using an Embedded PYBIND11 Python Interpreter”. *OpenFOAM Journal*, vol 2, 166–182. 2022.