Kalpa Publications in Computing

Volume XXX, 2022, Pages 5–7

Proceedings of V XoveTIC Conference. XoveTIC 2022

Implementation of a Solver to Simulate Industrial Galvanization

D. Barreiro-Villaverde[1,2], M. Lema[1], A. Gosset[3], and M.A Mendez[2]

[1] CITIC Research, Universidade da Coruña, A Coruña, Spain
david.barreiro1@udc.es
[2] von Karman Institute for Fluid Dynamics, Sint-Genesius-Rode, Belgium
miguel.alfonso.mendez@vki.ac.be
[3] CITENI, Universidade da Coruña, Ferrol, Spain
anne.gosset@udc.es

Abstract

In this work, we build a novel numerical approach to investigate the behaviour of the coating flow in the process of hot-dip galvanization. The latter involves a strong interaction between a liquid film transported by a moving strip and a high speed gas jet. The solver combines a high-fidelity model for the gas jet with a simplified model for the liquid film. The high-fidelity model is a solver that resolves the gas jet flow in OpenFOAM, an open-source CFD software built from C++ libraries. The model for the liquid film is an in-house finite volume solver implemented in Python. The communication between both models is handled with the plug-in coupling software preCICE.

1 Introduction

Thin liquid films interacting with gas flows are encountered in a variety of industrial processes: evaporators, distillation columns, coating processes etc. In all those applications, the interaction between the thin liquid layer and the gas flow plays a crucial role on the efficiency of the process. When the gas effect becomes stronger, the liquid film might deform in such a way that it affects the behavior of the gas flow. This happens for example with turbulent impinging gas jets on liquid films, sometimes giving rise to fascinating coupling phenomena [1].

The only way to resolve such a strong interaction between the liquid and the gas is through two-phase CFD simulations. However, those have a prohibitive computational cost for most industrial applications, as found in the case of the jet wiping process[1]. This is why in the last years, a lot of effort has been dedicated to the modelling of gas-liquid film flows based on long-wave models for the thin film; those are much less demanding in terms of computational resources because they are based on depth integrated equations. Lavalle et al.[4] coupled a similar long-wave model for the liquid film with a compressible Navier-Stokes solver for the gas phase. In a similar sheared film configuration, Bender et al.[2] used the turbulent stresses coming from a single-phase simulation of the gas flow to feed a long-wave model of the liquid flow. In this case, the model is one-way in the sense that there is no feedback of the film deformation on the gas flow.

In this work, we consider stronger gas-liquid interactions, in which the film deformation affects the gas flow. A long-wave model developed in the von Karman Institute[5] is coupled to a turbulent single-phase solver in OpenFOAM to resolve a challenging problem: the gas jet wiping of a liquid film dragged by a moving substrate, as represented in the 3D visualization in Figure 1a.

2 Numerical methods

The thin film solver, named BLEW, and implemented in Python, relies on the long-wavelength approximation. In this approach, the liquid flow equations are integrated across the film thickness, reformulating the problem as a function of thickness and flow rate [5]. The gas flow is resolved with a single-phase solver of the OpenFOAM libraries. The communication between both models is handled with the open-source plug-in preCICE[3].

The solvers are coupled explicitly at every time step: the stresses at the interface are extracted from the gas simulation and introduced in BLEW, which predicts the new thickness distribution to deform the numerical domain of the gas for the next time step. The gas jet therefore sees the interface as a solid wall at every time step, which solves many problems inherent to two-phase flow computations: diffuse interface, parasitic currents, turbulence damping at the interface, computational cost, etc.

The communication between solvers, usually referred to as participants, can be either serial using TCP/IP sockets or parallel using MPI ports. Typically, the high-fidelity model for the gas runs in parallel and the liquid film solver in serial in order to balance the computational load and speed up the computations. The spatial discretization in both participants does not need to be identical; therefore, several mapping strategies are available, i.e.: nearest neighbor, nearest projections and radial basis functions (RBF). We applied the nearest projections technique for simplicity and efficiency.

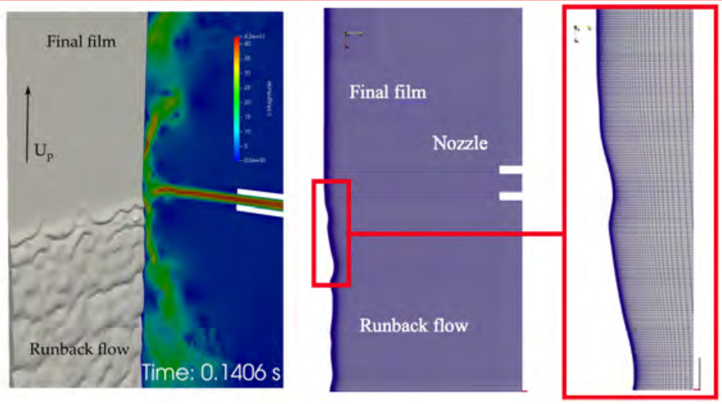
3 Results

The first test cases aim at proving that this segregated solver captures the two-way coupling nature of the problem. To this end, instead of using the liquid film solver, we introduce the time-dependent shape of the liquid from high-fidelity CFD simulations to deform the interface (left boundary), as shown in Figure 1b. In this situation, the jet was found to enter in an ample oscillating motion at a frequency of about 100 Hz, while it stays rigid when no deformation is imposed at the interface; therefore, proving that the model is capable to reproduce two-way coupling instabilities. Similarly, the liquid film solver has been subjected to the stresses from a single-phase simulation without mesh deformation, showing very good agreement with the CFD data.

Nevertheless, the full two-way model is currently under development. The mesh deformation leads to localized disturbances in the stresses at the interface that get amplified in subsequent time steps; thus, crashing the computations.

4 Conclusions

A novel numerical tool is proposed to solve strong liquid film-gas interactions; it is built by coupling an integral model for a liquid film and a standard Navier-Stokes solver of OpenFOAM. It is tested in 2D for the jet wiping process, giving preliminary good results in terms of film



(a) (b)

Figure 1: 3D view of the jet wiping process (a), and snapshot of the mesh deformation in this approach where the liquid film is replaced by a solid wall (b).

dynamics and CPU time. Data mapping, coupling schemes and specific features of both models will be investigated in order to apply the full two-way model for industrial galvanization.

5 Acknowledgments

D. Barreiro-Villaverde is financially supported by Xunta de Galicia with a pre-doctoral grant (ED481A-2020/018) and the research project is founded by Arcelor-Mittal. The authors also wish to thank the "Red Española de Supercomputación" for the computational resources attributed on FinisTerrae II and III (CESGA) (FI-2021-3-0012).

References

[1] D. Barreiro-Villaverde, A. Gosset, and M.A. Mendez. On the dynamics of jet wiping: Numerical simulations and modal analysis. *Physics of Fluids*, 33(6), 2021.

[2] Achim Bender, Alexander Stroh, Bettina Frohnapfel, Peter Stephan, and Tatiana Gambaryan-Roisman. Combined direct numerical simulation and long-wave simulation of a liquid film sheared by a turbulent gas flow in a channel. *Physics of Fluids*, 31(2):022103, feb 2019.

[3] Hans Joachim Bungartz, Florian Lindner, Bernhard Gatzhammer, Miriam Mehl, Klaudius Scheufele, Alexander Shukaev, and Benjamin Uekermann. preCICE – A fully parallel library for multi-physics surface coupling. *Computers and Fluids*, 141:250–258, dec 2016.

[4] G Lavalle, J.-P. Vila, G Blanchard, C Laurent, and F Charru. A numerical reduced model for thin liquid films sheared by a gas flow. *Journal of Computational Physics*, 301:119–140, 2015.

[5] M. A. Mendez, A. Gosset, and J. M. Buchlin. Experimental analysis of the stability of the jet wiping process, part II: Multiscale modal analysis of the gas jet-liquid film interaction. *Experimental Thermal and Fluid Science*, 106:48–67, sep 2019.