# Developing a Coupled Free-Surface Flow Solver using Interface Tracking Algorithm in foam-extend

**Behnam Cheraghi[1], Shidvash Vakilipour[2*]**

[1] Ph.D. Candidate, College of Interdisciplinary Science and Technology, University of Tehran, Tehran, Iran
[2] Associate Professor, College of Interdisciplinary Science and Technology, University of Tehran, Tehran, Iran

**ABSTRACT**

**In the present study, the implementation of a coupled implicit solver is presented for solving the free-surface flows. The implementation of the coupled pressure and velocity along with the default interface tracking algorithm has led to the creation of a solver equivalent to the basic foam-extend solver, which is called interTrackFoam. All of the features of the foam-extend platform can still be used i.e. mesh motion and parallel processing. Also, the libraries related to the block matrices available in foam-extend are used in the developed solver. The block matrix system is utilized as the basis of the coupled solver. The single-step solution of pressure and velocity is known as one of the main differences with the default solver. The ability of the developed solver to solve various test cases including a three-dimensional tank, solving the free-surface flow around an airfoil, and the flow passing over the ramp is demonstrated. The simultaneous solution capability has provided the possibility of reducing the number of iterations or considering a relatively higher time step related to solving the flow field. This aforementioned solver is the first step towards the velocity, pressure, temperature and species coupled solver with heat and mass transfer capability.**

**KEYWORDS**

**Coupled solver, free-surface flow, Interface tracking algorithm, foam-extend, OpenFOAM.**

[*] Corresponding Author: Email: vakilipour@ut.ac.ir

## 1. Introduction

One of the fundamental methods for studying two-phase flows involves analyzing fluid flow with a free-surface. In free-surface flows, one fluid is fully simulated, and the influence of the second fluid is modeled using specific boundary conditions at the interface. The interTrackFoam solver is well-known for its capabilities in solving various free-surface and two-phase problems. It operates using the PIMPLE flow solution algorithm and the surface displacement algorithm by Muzzafarija and Peric [1]. In problems where temperature and species equations are activated and mass and heat transfer occur at the free-surface, the boundary conditions of the free-surface and within the field can be highly interdependent. For instance, in the studies of Cheraghi and Vakilipour [3] and Havestini and Ormiston [3], the convergence rate of the problem is significantly reduced with segregated algorithms, which is why they have moved towards coupled algorithms [4]. The existence of two default solvers, i.e. interTrackFoam and pUCoupledFoam, has provided the possibility of developing a partially coupled solver that operates with the interface-tracking displacement algorithm. The interTrackFoam solver has a very high speed in solving free-surface flows. The only challenge to the performance of this solver arises when different phenomena with high physical coupling are involved in the problem.

Specifically, in the present study, a solver equivalent to the default solver, interTrackFoam, but with a coupled flow algorithm, has been developed and evaluated. The coupled algorithm involves solving velocity and pressure simultaneously and implicitly within a single system and matrix system, which is implemented using another default solver pUCoupled within the foam-extend. Initial efforts for the simultaneous pressure and velocity solution algorithm were initiated by Patankar [5], and further advanced by works from Schneider and Karimian [6], Darwish et al. [7], Vakilipour and Ormiston [8]. To verify the performance of this solver, various two-dimensional and three-dimensional test cases are considered. These include flow over a ramp, flow around an airfoil underneath the free-surface, and finally, flow within a three-dimensional tank. Comparisons between the results of both the default and developed solvers, as well as with results from other studies, show very good agreement and validate the performance of this solver. Furthermore, the coupled algorithm enhances the convergence rate and reduces the number of solution iterations, although in flow solution problems, the default segregated solver, i.e. interTrackFoam, demonstrates relatively higher computational speed.

## 2. Test Cases

Three test case are examined to evaluate the performance of the solver. The first example is flow over a ramp. The final free-surface shape is shown in Figure 1.

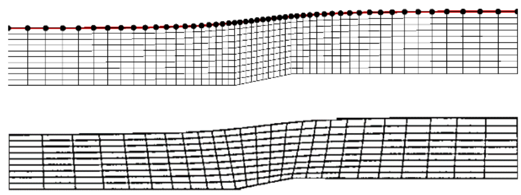

**Figure 1. Comparison of the final position of the free-surface, up) Default solver (black dots) and developed solver (red line), deown) Muzaferija and Peric [1]**

The second case is the hydrofoil problem [9]. The free-surface shape in comparison with the literature are shown in Figure 2.

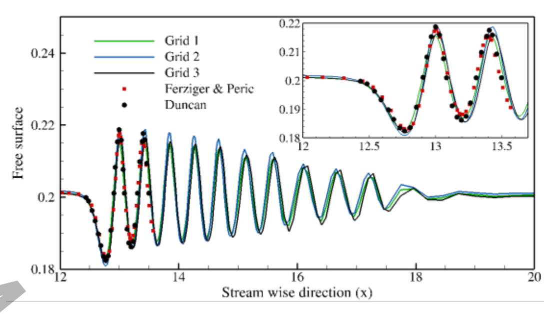


**Figure 2. Comparison of results for three grids with different resolutions and also comparison of free-surface waves with numerical and experimental results of [9]**

The third case is a three-dimensional flow in a cubical tank. In these test cases, accuracy, velocity and pressure distribution, convergence behavior and surface motion are investigated. The flow structure and velocity contours are presented in Figure 3.

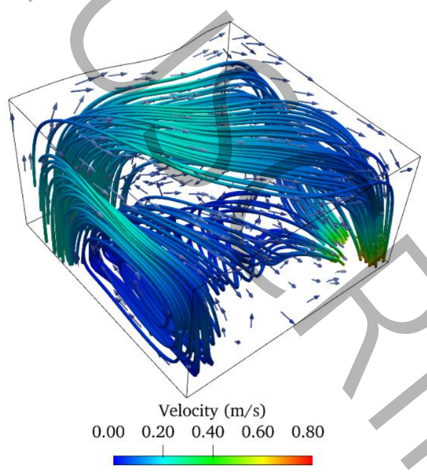


**Figure 3. Flow structure and velocity contour for the three-dimensional tank**

Also, the final shape of free-surface for the three-dimensional tank is shown at Figure 4.

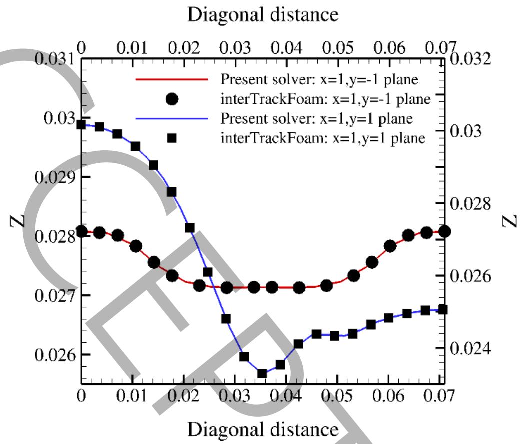


**Figure 4. : Comparison of the free-surface position in two inclined planes for three-dimensional tank**

## 3. Conclusion and Results

In the present study, the development, implementation, and comparison of results of a velocity-pressure coupled solver operating with the interface-tracking algorithm [1,10] are presented. This solver has been evaluated on various problems, yielding the following results:

- It can be asserted that the pressure equation limits the convergence rate of a single loop. In comparison to the default solver, the coupled solver shows approximately 10, 4, and 6 times fewer pressure solution iterations for ramp, hydrofoil, and 3D tank cases, respectively.
- The overall flow solution speed for the coupled solver relative to the default solver is 0.67, 2.2, and 1.24 times for the aforementioned cases. In other words, despite a higher number of pressure equation iterations, the interTrackFoam solver exhibits a more desirable computational speed.
- All object-oriented programming capabilities, including classes related to parallel programming, working with mesh displacement libraries, using functions of the freeSurface class, and working with functions of coupled solver classes, are preserved in this solver.
- For solving flows, the interTrackFoam solver is recommended as the preferred solver because it allows separate control over the number of iterations for each of the velocity or pressure equations, enabling optimization of the number of iterations while increasing computational speed. However, in the presence of strong coupling and physical complexity, this trend might reverse, or convergence challenges for this solver may arise.

## 4. References

[1] S. Muzaferija, M. Peri´ c, Computation of free-surface flows using the finite-volume method and moving grids, Numerical Heat Transfer, 32(4) (1997) 369-384.

[2] B. Cheraghi, S. Vakilipour, Developing an Interface Tracking Coupled Solver for Solving two Phase Flow Fields at Low Reynolds Numbers in foam-extend Platform, Sharif Journal of Mechanical Engineering, (2024), pp.-. (in Persian).

[3] R.A. Havestini, S.J. Ormiston, An elliptic numerical analysis of water vapour absorption into a falling film in vertical parallel plate channels, International Journal of Heat and Mass Transfer, 150 (2020) 119266.

[4] S. Vakilipour, M. Mohammadi, S. Ormiston, A fully coupled ALE interface tracking method for a pressure-based finite volume solver, Journal of Computational Physics, 427 (2021) 110054.

[5] S. Patankar, Numerical heat transfer and fluid flow, Taylor & Francis, 2018.

[6] S. Karimian, G. Schneider, Pressure-based computational method for compressible and incompressible flows, Journal of thermophysics and heat transfer, 8(2) (1994) 267-274.

[7] M. Darwish, I. Sraj, F. Moukalled, A coupled incompressible flow solver on structured grids, Numerical Heat Transfer, Part B: Fundamentals, 52(4) (2007) 353-371.

[8] S. Vakilipour, S. Ormiston, A coupled pressure-based co-located finite-volume solution method for natural-convection flows, Numerical Heat Transfer, Part B: Fundamentals, 61(2) (2012) 91-115.

[9] J.H. Ferziger, M. PeriC, Computational methods for fluid dynamics, in, Springer, 2002.

[10] Ž. Tuković, H. Jasak, A moving mesh finite volume interface tracking method for surface tension dominated interfacial fluid flow, Computers & fluids, 55 (2012) 70-84.