Summer Fellowship Report

On

Rayleigh-Taylor instability

Submitted by

Raj Niraj Patil

Under the guidance of

Prof. Shivasubramanian Gopalakrishnan
Mechanical Engineering Department
IIT Bombay

July 8, 2019

Acknowledgment

I thank Prof. Shivasubramanian Gopalakrishnan and Prof.Kannan M. Moudgalya for their invaluable guidance. At the same time, I would like to thank Mr Sathish Kanniappan and Ms Deepa Vedartham for their undying support.

These six weeks were one of the most prestigious weeks. I've learnt so much more about CFD, OpenFOAM and other FLOSS. I acknowledge the FOSSEE team for conducting this summer internship. The last but not least, I would like to thank the authority of IIT Bombay for letting us use the resources during this period and helping us from starting day of the internship till the end.

Contents

Chapter 1

Introduction

The RayleighTaylor instability, or RT instability (after Lord Rayleigh and G. I. Taylor), is an instability of an interface between two fluids of different densities which occurs when the lighter fluid is pushing the heavier fluid. Examples include the behavior of water suspended above oil in the gravity of Earth, mushroom clouds like those from volcanic eruptions and atmospheric nuclear explosions, supernova explosions in which expanding core gas is accelerated into denser shell gas, instabilities in plasma fusion reactors and inertial confinement fusion.

Water suspended atop oil is an everyday example of RayleighTaylor instability, and it may be modeled by two completely plane-parallel layers of immiscible fluid, the more dense on top of the less dense one and both subject to the Earth's gravity. The equilibrium here is unstable to any perturbations or disturbances of the interface: if a parcel of heavier fluid is displaced downward with an equal volume of lighter fluid displaced upwards, the potential energy of the configuration is lower than the initial state. Thus the disturbance will grow and lead to a further release of potential energy, as the more dense material moves down under the (effective) gravitational field, and the less dense material is further displaced upwards. This was the set-up as studied by Lord Rayleigh. The important insight by G. I. Taylor was his realisation that this situation is equivalent to the situation when the fluids are accelerated, with the less dense fluid accelerating into the more dense fluid. This occurs deep underwater on the surface of an expanding bubble and in a nuclear explosion.

As the RT instability develops, the initial perturbations progress from a linear growth phase into a non-linear growth phase, eventually developing "plumes" flowing upwards (in the gravitational buoyancy sense) and "spikes" falling downwards. In the linear phase, equations can be linearized and the amplitude of perturbations is growing exponentially with time. In the non-linear phase, perturbation amplitude is too large for the non-linear terms to be neglected. In general, the density disparity between the fluids determines the structure of the subsequent non-linear RT instability flows (assuming other variables such as surface tension and viscosity are negligible here). [1]

Chapter 2

Rayleigh-Taylor instability

2.1 Abstract

The report about the simulation of Reyleigh-Taylor instability of two fluids with density 1020 kg/m^3 and 800 kg/m^3 with heavier liquid being top on the lighter. The result obtained is good but not excellent. It could have been better but it needs high computational power. This case is solved with interFoam with k-Epsilon model. The result fairly agrees with experimental videos taken by several people.

2.2 Geometry & Meshing

It is 20cm x 20cm x 1cm cuboid with only single cell in z direction to make it 2D simulation. Top wall is named as topWall while remaining sides are named as walls. Meshing was done using blockMesh with all sides except front and back being wall and front and back side being empty.

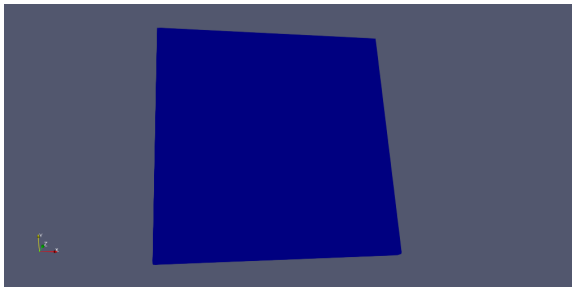

Fig 1: Mesh

2.3 Solver

This simulation is done using interFoam with K-Epsilon model. InterFoam solves for two incompressible immiscible fluids under isothermal conditions using a volume of fluid approach. It also allows us to use mesh motion, mesh topology changes and adaptive re-meshing. The solver solves the Navier Stokes equations for two incompressible, isothermal immiscible fluids. That means that the material properties

are constant in the region filled by one of the two fluid except at the interphase. It should be highlighted that interFoam applies the OpenFOAM specific algebraic VOF scheme called MULES (Multidimensional Universal Limiter with Explicit Solution)for the task of advecting the sharp interface. That it should be noted because of the MULES scheme has a number of desirable properties:

- It preserves the volume of fluid, i.e. it does not artificially create or destroy fluid.
- It keeps the so-called volume fraction field in the physically meaningful range-between 0 and 1.
- The interface stays sharp to within a few cell widths
- It works on unstructured meshes both in 2D and 3D
- It is efficient so only a minor fraction of the calculation time is spent on interfaceadvection. [2]

Concerning the solver algorithm used to solve the types of problems under consideration here, the PISO (Pressure Implicit with Splitting of Operators) loop was chosen.The main reason is that this algorithm works proper for transient calculations and for low Courant numbers (less than one). As it will be check, all the simulations run in the present thesis match with both approaches. The PISO approach was proposed by Issa [3] (1986). It consists on three steps for each time step and the iterations are only needed for the two last steps. One summary of this algorithm can be presented as [4]:

- Momentum predictor: The momentum equations are optionally solved using a best-guess pressure field to produce a best-guess velocity field. However, Issa [3] (1986) notes that while this momentum predictor step is formally required for the method, many applications do not require it, and can proceed straight to the second step using the previous time-steps velocity field.
- Using the previous velocities the pressure field is obtained. Therefore, thefirst estimate of the new pressure field is obtained.
- The velocity field is corrected using the new pressures.

Within each time step, additional equations for multi-phase flow are solved before the PISO algorithm, while other equations, such as turbulence models, are solved afterwards.

2.3.1 Equations

Continuity Equation

$$\frac{\partial u_j}{\partial x_j} = 0 \tag{2.1}$$

Momentum Equation

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_j u_i)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial(\tau_{tij} + \tau_{ij})}{\partial x_j} + \rho g_i + f_{\sigma i} \tag{2.2}$$

u represent the velocity, g_i the gravitational acceleration, p the pressure and τ_{ij} and $\tau_{t_{ij}}$ are the viscose and turbulent stresses. $f_{\sigma i}$, is the surface tension.

The density ρ is defined as follows:

$$\rho = \alpha\rho_1 + (1-\alpha)\rho_2 \tag{2.3}$$

is 1 inside fluid 1 with the density ρ_1 and 0 inside fluid 2 with the density ρ_2 . At the interphase between the two fluids α varies between 0 and 1. The surface tension $f_{\sigma i}$, is modelled as continuum surface force (CSF). It is calculated as follows:

$$f_{\sigma i} = \sigma\kappa\frac{\partial\alpha}{\partial x_i} \tag{2.4}$$

σ is the surface tension constant and κ the curvature. The curvature can be approximated as follows:

$$\kappa = -\frac{\partial n_i}{\partial x_i} = -\frac{\partial}{\partial x_i}\left(\frac{\partial\alpha/\partial x_i}{|\partial\alpha/\partial x_i|}\right) \tag{2.5}$$

Interphase Equation

In order to know where the interphase between the two fluids is, an additional equation for α has to be solved.

$$\frac{\partial\alpha}{\partial t} + \frac{\partial(\alpha u_j)}{\partial x_j} = 0 \tag{2.6}$$

The equation can be seen as the conservation of the mixture components along the path of a fluid parcel. [5]

2.3.2 K-Epsilon

K-epsilon (k-) turbulence model is the most common model used in Computational Fluid Dynamics (CFD) to simulate mean flow characteristics for turbulent flow conditions. It is a two equation model that gives a general description of turbulence by means of two transport equations (PDEs). The original impetus for the K-epsilon model was to improve the mixing-length model, as well as to find an alternative to algebraically prescribing turbulent length scales in moderate to high complexity flows. [6]

- The first transported variable is the turbulence kinetic energy (k).
- The second transported variable is the rate of dissipation of turbulence energy (ϵ).

For turbulent kinetic energy k

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right] + 2\mu_t E_{ij}E_{ij} - \rho\varepsilon \tag{2.7}$$

For dissipation ϵ

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho\varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t E_{ij}E_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{2.8}$$

[Rate of change of k or ϵ + Transport of k or ϵ by convection = Transport of k or ϵ by diffusion + Rate of production of k or ϵ - Rate of destruction of k or ϵ]

where,

u_i represents velocity component in corresponding direction

E_{ij} represents component of rate of deformation

μ_t represents eddy viscosity

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{2.9}$$

The equations also consist of some adjustable constants σ_k, σ_ϵ, $C_{1\epsilon}$ and $C_{2\epsilon}$. The values of these constants have been arrived at by numerous iterations of data fitting for a wide range of turbulent flows. These are as follows:

C_μ= 0.09 σ_k= 1.00 σ_ϵ= 1.30 $C_{1\epsilon}$= 1.44 $C_{2\epsilon}$= 1.92

2.4 Case Setup

2.4.1 Boundary Conditions

All patches with type were set with **slip** velocity boundary condition with respective wall functions in files k, nut and epsilon (**kqRWallFunction** in k file, **nutkWallFunction** for nut and **epsilonWallFunction** for epsilon). The values were made dependent on internal field. **topWall** was kept at fixed pressure where walls were made **zeroGradient**. Alpha.water conditions were **zeroGradient** everywhere. **frontAndBack** was made empty in all of the above files.

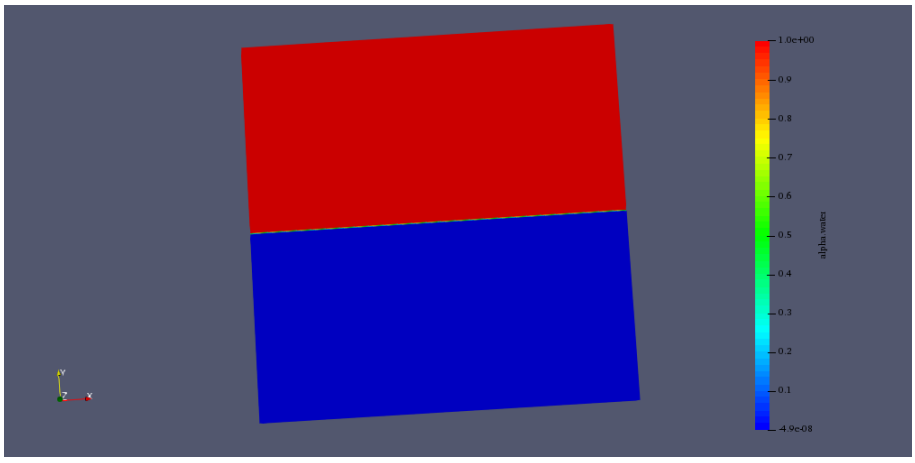

Fig 2: Case setup

2.5 Results

As shown in the following figure, very good results agreeing with experiments were obtain. From the figures shown below, the mushroom formation with heavier liquid penetrating into lighter one can be noticed.

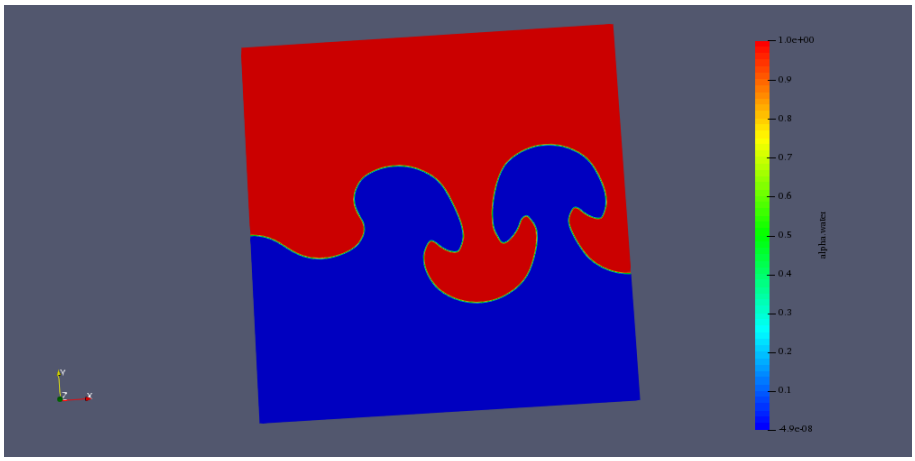

Fig 3a

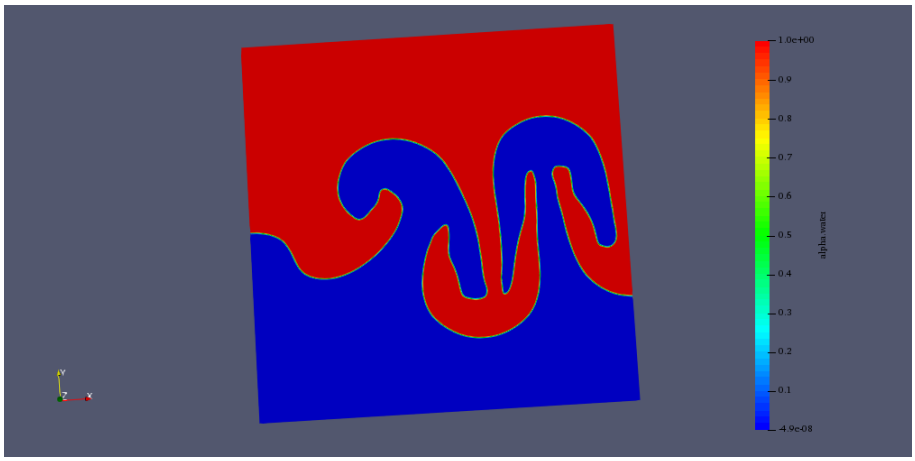

Fig 3b

2.6 Overview

The above case reasonably satisfies the experimental results, but it is not precisely correct. In the actual case, it must locally form the mushrooms and should continue to form until there is no surface present between the liquids (before complete mixing and the final separation).

Reference

- https://en.wikipedia.org/wiki/Rayleigh-Taylor_instability
- Andrs Calvo Luz. Study of bubbly flows in an open-source software. *Aalto University*.
- R.I. Issa. Solution of the implicitly discretised fluid flow equations by operator-splitting.Jpurnal of Computational Physics, 62:4065, January 1986.
- Gerald C.J. Morgan. Application of the interfoam vof code to coastal wave/ structure interaction. Masters thesis, University of Bath Department of Architecture and Civil Engineering, February 2013.
- https://openfoamwiki.net/index.php/InterFoam
- https://en.wikipedia.org/wiki/K-epsilon_turbulence_model