Towards a Moving Mesh Numerical Simulation of a Heated Solid Immersion into Liquid

Alen Cukrov, Željko Tuković and Antun Galović
Faculty of Mechanical Engineering and Naval Architecture, University of Zagreb,
Ivana Lučića 5, 10000 Zagreb, Croatia
e-mail: alen.cukrov@gmail.com

ABSTRACT

This paper presents an update on the project whose goal is to gain insight into heat transfer phenomena during immersion of a heated solid into liquid. The numerical simulation is carried out using a moving mesh finite volume method implemented in OpenFOAM®. The computational domain consists of a fluid and a contour of the solid region. An interface capturing method is used for modelling the two phase fluid flow. The time histories of solid motion are given.

KEY WORDS

Moving mesh, VOF, OpenFOAM®

1. INTRODUCTION

This paper gives update on developing a Computational Fluid Dynamics methodology for prediction of the transient heat conduction during immersion of a heated solid into liquid. The governing equations of such phenomena and related numerical issues are discussed within Chapter 2. Furthermore, Chapter 3 gives the description of the geometry and the applied boundary conditions. Obtained results are shown in Chapter 4. The paper ends with Chapter 5 which summarizes achieved goals and presents next steps.

2. NUMERICAL PROCEDURE

A two-phase Navier-Stokes solver *navalHydro* embedded into OpenFOAM 3.0-ext® which utilizes Volume-of-Fluid (VOF) method is used in this study. The reliability of *navalHydro* solver in solving different two-phase tasks is shown in [1], [2] and [3]. Although only isothermal case is computed within this work, the equations which govern heat transfer phenomena are also presented in the next section.

2.1. Governing Equations

The underlying equations of this multiphase flow problem are the conservation of mass, phase fraction, momentum and energy

$$\nabla \cdot \mathbf{U} = 0 \tag{1}$$

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\mathbf{U}\alpha) + \nabla \cdot [\mathbf{U}_c \alpha (1-\alpha)] = 0 \tag{2}$$

$$\frac{\partial(\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U}\mathbf{U}) = -\nabla p_d - \mathbf{g} \cdot \mathbf{x} \nabla \rho + \nabla \cdot \mathbf{T} + \sigma \kappa \nabla \alpha \tag{3}$$

$$\frac{\partial(\rho c_p T)}{\partial t} + \nabla \cdot (\rho c_p \mathbf{U} T) = \nabla \cdot [\nabla (kT)] \tag{4}$$

Where α denotes the volume fraction field of the liquid. It has values between 0 and 1. A value of α = 0 indicates that the cell contains no liquid, while α = 1 corresponds to a cell full of liquid. The interfacial region between the phases has α values between 0 and 1. The third term on the l.h.s. of Equation 2 is a compression term. The compression term is introduced in order to ensure sharp interface region between the phases. The sharp resolution of the interface while preserving the boundedness and conservation of the phase fraction is the critical issue in numerical simulations of free-surface flows using the VOF model [4]. In order to take into account the transient conduction in a solid, the energy equation for the solid must also be solved

$$\frac{\partial(\rho_s c_s T_s)}{\partial t} = \nabla \cdot [\nabla (k_s T_s)] \tag{5}$$

As stated in the introduction of this chapter, the equations 4 and 5 are not used in this study.

2.2. Physical Properties

As shown in previous section, the model utilizes one-fluid formulation where one set of conservation laws is solved for both phases. The phases are considered immiscible and their density, dynamic viscosity, specific heat capacity and thermal conductivity are defined as follows [4]

$$\rho = \alpha \rho_l + (1 - \alpha) \rho_g \tag{6}$$

$$\mu = \alpha \mu_l + (1 - \alpha) \mu_g \tag{7}$$

$$\rho c_p = \alpha \rho_l c_{p,l} + (1 - \alpha) \rho_g c_{p,g} \tag{8}$$

$$k = \alpha k_l + (1 - \alpha) k_g \tag{9}$$

In the present work, the physical properties of each phase are taken from [5].

2.3. Numerical Implementation

The set of governing equations presented in Section 2.1 is discretized utilizing the Finite Volume Method (FVM). The discretization in OpenFOAM® is based on the FVM with a collocated variable arrangement. The pressure and velocity are coupled using PIMPLE algorithm which is the combination of SIMPLE and PISO algorithms [1]. The application of PIMPLE algorithm enables high Courant numbers, i.e. high time steps. All equations, including the phase fraction equation are treated implicitly, employing bounded discretization schemes [1].

2.4. Moving Mesh

In the present work, the mesh motion algorithm based on Laplace equation

$$\nabla \cdot (\gamma \nabla \mathbf{u}) = 0 \tag{10}$$

is used. The method is implemented in OpenFOAM® by Tuković and Jasak [6]. This vertex-based automatic mesh motion solver calculates the motion of internal points based on the

prescribed motion of interface points by solving the variable diffusivity Laplace equation discretized using the Finite Element Method [7]. In Equation 10, **u** is a displacement vector and γ is a diffusion coefficient. This diffusion coefficient can be optionally set either constant or variable. In this work, the variable diffusion coefficient γ is defined in a file and is given in Listing 1.

Listing 1 Computing the diffusivity for the mesh motion

```
#include "argList.H"
#include "tetPolyMesh.H"
#include "tetFemMatrix.H"
#include "elementFields.H"
#include "volFields.H"

using namespace Foam;

int main(int argc, char *argv[])
{

#   include "setRootCase.H"
#   include "createTime.H"
#   include "createMesh.H"
#   include "createFields.H"

    scalarField centresY =
        mesh.cellCentres().component(vector::Y);

    motionGamma.internalField() +=
        999*pos(centresY - 3.5e-1)*neg(centresY - 4.5e-1);
    motionGamma.write();

    motionGammaFVM.internalField() = motionGamma.internalField();
    motionGammaFVM.correctBoundaryConditions();
    motionGammaFVM.write();

    return(0);
}
```

3. COMPUTATIONAL DOMAIN AND BOUNDARY CONDITIONS

The two dimensional computational domain used in this study is shown on Figure 1. The red cells are occupied by water while the blue cells contain air. The mesh is consisted of 610 cells. At the right side of the domain one can distinguish between *far-field* and *moving far-field* patches. Their separation is necessary for handling the mesh motion. In file *0 / motionU*, *moving far-field* is defined the same as *moving body*. On the other side, *far-field* is a patch defined as *slip*. For the scalar fields, pressure and volume fraction, a zero-gradient (Neumann) boundary condition is specified at *moving body*, *bottom*, *far-field* and *moving far-field* while on *top* an uniform and equal to zero value is employed. Furthermore, for the velocity field *U*, also an uniform and equal to zero value is set on all boundaries except *symmetry plane*. On *symmetry*

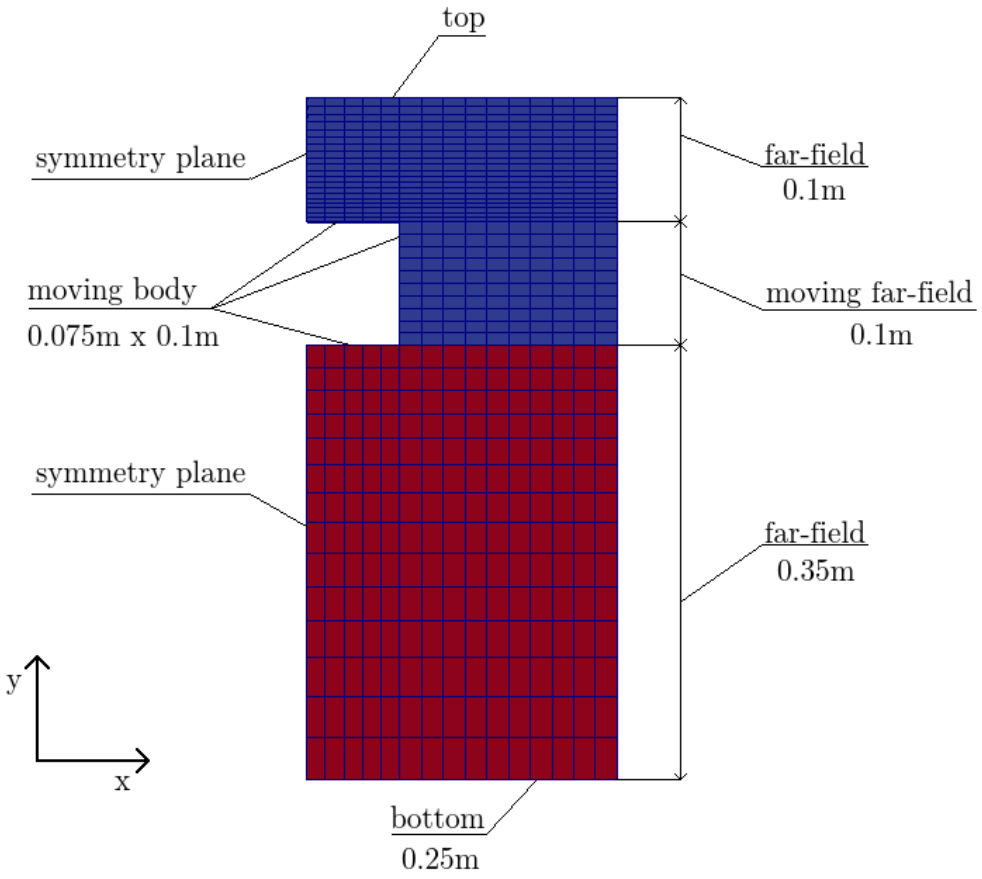

Figure 1 Computational domain.

plane the symmetry boundary condition is applied for all the quantities.

4. RESULTS

The volume fraction field during a solid body immersion is depicted on Figure 2 for different time instances. The body is moving downwards with the constant velocity U = -0.1 m/s (in y direction). The time-step used in simulation is defined based on Courant number in the domain. The maximum Courant number in the calculation is set to 0.25. Even on a coarse mesh, the applied method well resolves the interface between the regions. Furthermore, from Figure 2 the applied mesh deformation technique is visible. The cells below the moving body are compressed, the above expanded while ones under *moving far-field* patch remain unchanged. Since the adaptive time-step is applied, the most time consuming part is when the body is fully immersed as shown on Figure 2.e and Figure 2.f.

5. CONCLUSION AND FUTURE WORK

This paper represents the current steps performed in order to develop a solver for transient heat transfer analysis during immersion of a solid into liquid. The applied method has shown stability without imposing high mesh resolution at the interfacial region. The further step is incorporating the energy equation within the solver. Finally, the goal is to introduce the solid region and observe transient heat transfer during immersion.

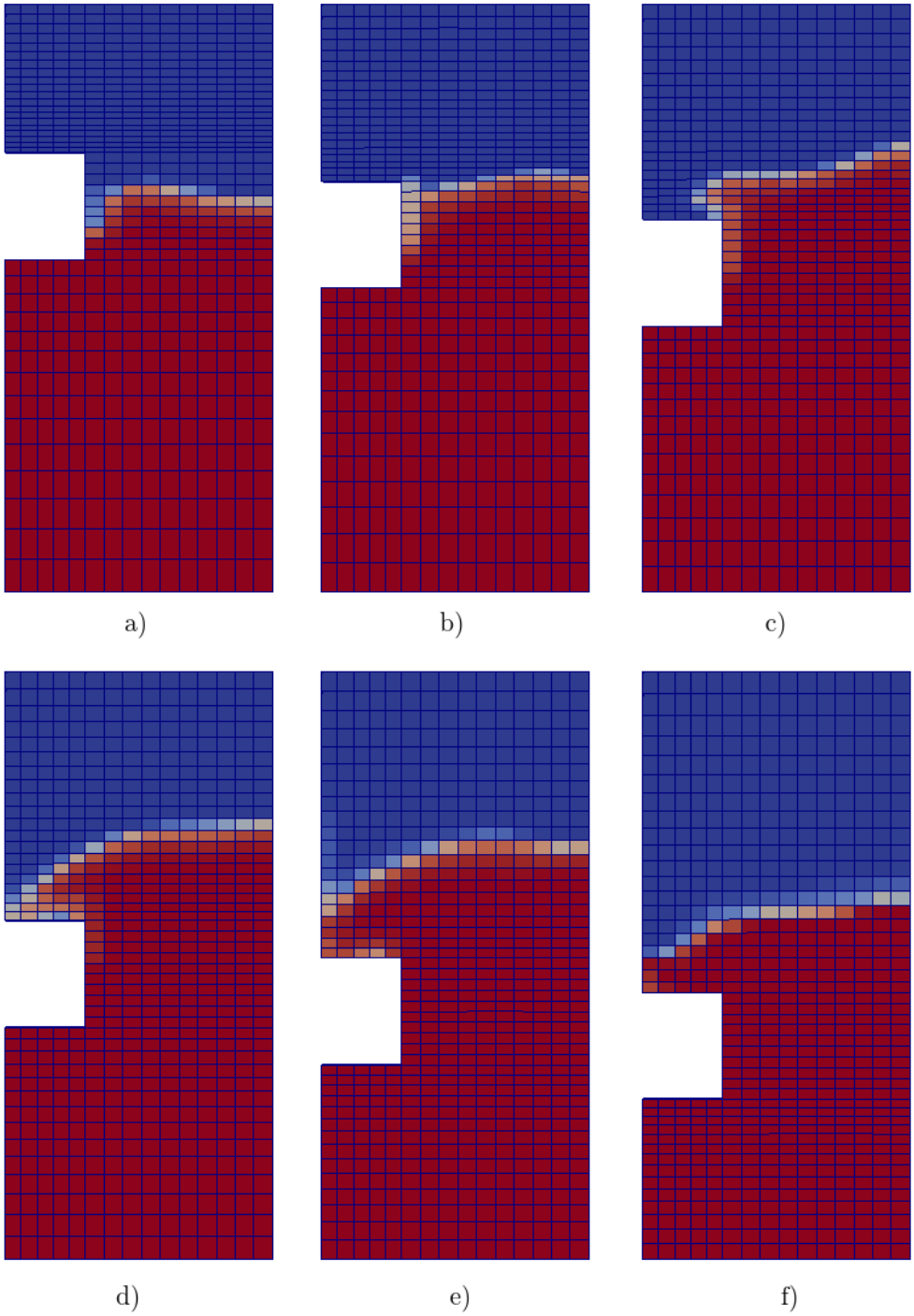

Figure 2 Snapshots of volume fraction field for different time instances: a) t = 0.12 s, b) t = 0.2 s, c) t = 0.3 s, d) t = 0.4 s, e) t = 0.5 s and f) t = 0.6 s.

ACKNOWLEDGEMENT

The authors would like to thank to Professor Hrvoje Jasak for providing *navalHydro* solver.

REFERENCE

[1] Vukčević, V., Jasak, H., Primjena Naval Hydro paketa za dvofazna strujanja, Šesti susret Hrvatskog društva za mehaniku, Rijeka, Croatia, 2014.

[2] Gatin, I., Vukčević, V., Jasak, H., Primjena Naval Hydro paketa za proračun valnih opterećenja, Šesti susret Hrvatskog društva za mehaniku, Rijeka, Croatia, 2014.

[3] Jasak, H., Naval Hydro Pack: Overview of Numerics and Capability, January 2015.

[4] Berberović, E., Investigation of Free-surface Flow Associated with Drop Impact: Numerical Simulations and Theoretical Modeling, Ph.D. thesis, Technische Universität Darmstadt, 2010.

[5] Galović, A., Halasz, B., Boras, I., Toplinske tablice, Fakultet strojarstva i brodogradnje, 2008.

[6] Jasak, H., Tuković, Ž., Automatic mesh motion for the unstructured finite volume method, Transaction of FAMENA 30 (2) (2006), pp. 1–20.

[7] Tuković, Ž., Jasak, H., A moving mesh finite volume interface tracking method for surface tension dominated interfacial fluid flow, Computers and Fluids 55 (2012), pp. 70–84.