Falling of water droplet on the pool of water

Divyesh Variya

Abstract—**This case study demonstrates the simulation of freely falling of a water droplet on the pool of water. The specific case study is designed such that the properties of the system are similar to those of raindrops falling through the air. The study of raindrops is interesting from both an engineering standpoint and from a standpoint of pure curiosity. 2D case model of a water droplet is made with SALOME-9.2.0 mashing tool and setfieldsdict. The whole case study run in OpenFOAMv6. The behavior of the Impact of a water droplet on the water surface and topological changes in droplet itself is observed in results.**

Keywords—**water droplet, free falling, water pool, level set, gravity, interFoam, two-phase flow.**

I. INTRODUCTION

Two phase flow are encountered in a wide range of industrial as well as engineering applications, e.g. bubble and droplet dynamics [1], atomization and spray of liquid jet [2], and other multi-phase flow systems [3]. Due to the importance of droplet dynamics in most of two-phase flow systems, there is an increased attention being given for the prediction of deformation and disintegration of droplets either numerically, analytically or experimentally. It is well known that the combustion efficiency in diesel engines, gas turbine engines, oil burners and liquid rockets is strongly dependent on liquid fuels atomization process [4].

Consequently, two-phase flow remains a challenging topic of research [5] due to the existence of inter-facial surface force such as surface tension and shear stress. Such stresses usually interact with other mechanisms, such as surface instabilities, ligament formation, stretching and fragmentation to transform large scale coherent liquid structures into small scale droplets [6].

The numerical simulations of the two-phase dynamics are scarcely due its computational challenges [7 8]. Although drops seldom occur in isolation, it is essential to understand the behavior of single and binary droplets before a full knowledge on interacting can be achieved. Disturbances, which cause disintegration of drops, include: rapid acceleration and high shear stresses.

Although several papers have recently reported experimental efforts to understand the physics of the droplet deformation, disintegration and its related dynamics in two-phase flow, however, experimental measurements and the observation of dense and small region with high spatial-temporal resolution in such applications have been difficult [9].

In the numerical simulation of droplet dynamics, it has been difficult to predict the physical processes occurred due to the requirement of high resolution, especially for high Weber and Reynolds number. The severe resolution required in such simulation is essentially in order to resolve the important role played by surface tension in ligament and drop formation.

Consequently, in order to obtain an insight in such dynamics, the numerical treatments of such processes are carried out in a number of sequential steps starting from the investigation of the surface instability that leads to droplet deformation followed by ligament formation and drop separation from a single ligament till the secondary break-up of liquid droplets.

More recently, carefully executed simulations in such context can virtually replace experiments. In general, the numerical predictions of droplet dynamics have been limited in accuracy partly by the performance of three key elements, viz.: development of the computational algorithm, interface tracking methods, and turbulence prediction models.

During the last decade, a variety of computational fluid dynamics techniques have been developed to study two-phase flow dynamics. A comprehensive review of the numerical models applied for two-phase flow up to 1996 can be found in [10]. More extended review up to 2010 for the atomization process and its related dynamics can be found in [11].

Usually, in the numerical simulation of two-phase flows, the Navier-Stokes equations are coupled to one of the available tracking methods in order to predict the complex topological changes of the phase interface, see for more details [12, 13]. Given examples for such tracking methods, Volume-Of-Fluid (VOF) method [14] and Level Set Method (LSM) [15] are the most popular interface capturing methods. Although the VOF method has been widely applied for predicting different complex two-phase flows, it suffers from several numerical problems such as interface reconstruction algorithms and the difficult calculation of the interface curvature [16]. These numerical problems can, in particular, limit the accuracy and the stability of the numerical method adopted for calculation of two-phase flows, especially when the surface tension is included. A comprehensive review for the different VOF methods and their numerical constraints can be found in [17].

In contrast to the VOF methods, the level set methods offer highly robust and accurate numerical technique for capturing the complex topological changes of moving interfaces under complex motions. The basic idea of LSM is the use of a continuous, scalar and implicit function defined over the whole computational domain with its zero value is located on the interface. The LSM divides the domain into grid points that contain the value of the scalar function; therefore, there is an entire family of contours. The interface is then described as a signed distance function at any time and, consequently, the geometric properties of the highly complicated interfaces are calculated directly from level set function. Moreover, the complex topological changes of interfaces such as merging and breaking-up are handled automatically in a quite natural way without any additional procedure. In addition, the extension of the LSM to three-dimensional problems is easy and straightforward.

Referring to the previous discussion, the LSMs have seen tremendously in different CFD-applications of diverse areas, e.g. two-phase flows, turbulent atomization, grid generation and turbulent combustion [18]. However, the LSMs suffer from numerical diffusion which may cause a smoothing out of sharp edges of interface. The level set function is usually evolved by a simple Eulerian scheme and, consequently, the final implementation of LSM does not provide full volume conservation, so highly accurate transport schemes are required.

II. GEOMETRY

A 2D square CAD model is created with salome-9.2.0 tool having 40 X 40 mm dimensions. Water pool level of 5 mm and droplet of 2 mm radius is set by LSM (Level set method) available as setFieldsDict in OpenFOAMv6.
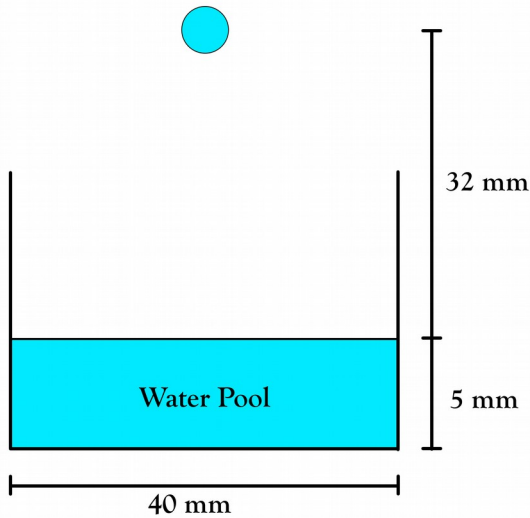
Figure 1. The configuration domain of falling droplet on water pool

III. MESH

Meshing is the crucial part of any simulation and especially in computational fluid dynamics. In this case after making a CAD model meshing is to be done in same software-Salome-9.2.0. Parameter considered for meshing are stated in table given below. After computing mesh in salome-9.2.0, It is exported as .unv file. The .unv file can be easily converted as OpenFOAM mesh. OpenFOAMv6 gives easy access to import mesh from some other tools. A single command called "ideasUnvToFoam" import all boundary patches to the OpenFOAM file name "polyMesh".

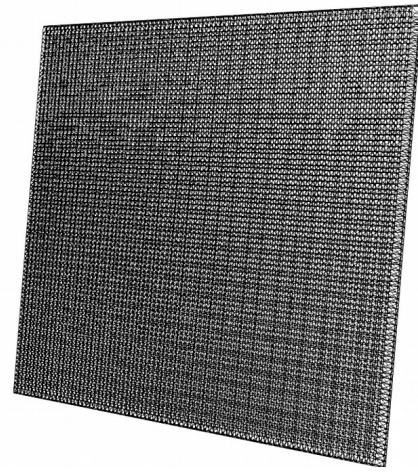| Parameter	Value
Hexahedrons	160000
Max aspect ratio	1
Max skewness	4.16334e-13
Bounding box	(0 0 0) (0.04 0.04 0.0005)

Table 1. CheckMesh results

Figure 2. Mesh after execute command "ideanUnvToFoam"

IV. LEVEL SET

After meshing, it is required to set all water level in this case. So Water droplet and pool of water set after meshing with setfieldsDict located in system directory.

At first, defaultFieldValues set as alpha.water 0.

To set water droplet, region set with cylinderToCell. Which takes two center point and a radius of cylinder.

For water pool, region set with boxToCell. Which takes box minimum and maximum point values.

Here is the code for setFieldsDict:

```
defaultFieldValues
(
    volScalarFieldValue alpha.water 0
    volVectorFieldValue U (0 0 0)
);

regions
(
    cylinderToCell
    {
        p1  (0.02 0.037 0);
        p2  (0.02 0.037 1);
        radius 0.002;

        fieldValues
        (
            volScalarFieldValue alpha.water 1
            volVectorFieldValue U (0 0 0)
        );
    }
    boxToCell
    {
        box (0 0 0) (0.04 0.005 1);
        fieldValues
        (
            volScalarFieldValue alpha.water 1
        );
    }
);
```

V. SIMULATION

Complete analysis done with interFoam solver and in OpenFOAM v6.

A. Boundary Conditions

Water pool defined with setfieldsDict and velocity for both internal air and water is given to the zero value. The case is set as it has symmetrically infinite length of pool. Symmetry plane condition given to both the side, so it acts like water drop falls on very vast length of pool compared to geometry of drop.

The list of abbreviations used in the following table are:

1. SP: Symmetry Plane
2. FV: Fixed Value
3. FFP: Fixed Flux Pressure
4. ZG: Zero Gradient
5. TP: Total Pressure
6. PIOV: Pressure Inlet Outlet Velocity
7. IO: Inlet Outlet

Boundary	U	P_rgh	Alpha
inlet	FV	FFP	ZG
outlet	PIOV	TP	IO
FrontAndBack	Empty	Empty	Empty
symm1	SP	SP	SP
symm2	SP	SP	SP

Table 2. Boundary condition for U, P_rgh & alpha

B. InterFOAM Solver

The official definition for this solver is as follows:

Solver for 2 incompressible, isothermal immiscible fluids using a VOF (Volume of Fluid) phase-fraction interface capturing approach.

Various features of the solver are as follows:

1) Incompressible
2) Transient
3) Laminar and turbulent
4) Multiphase
5) Immiscible
6) Volume of Fluid
7) Isothermal

The VOF model can model two or more immiscible fluids by solving a single set of momentum equations and tracking the volume fraction of each of the fluids throughout the domain. Typical applications include the prediction of jet breakup, the motion of large bubbles in a liquid, the motion of liquid after a dam break, and the steady or transient tracking of any liquid-gas interface.

Solver is mainly integration of these equations:

Continuity equation

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

Momentun Equation

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla.\rho \mathbf{U}\mathbf{U} = -\nabla P + \nabla \rho \gamma [2S] + F_t \tag{2}$$

Volume of Fluid

$$\rho = \alpha \rho_l + (1 - \alpha)\rho_g \tag{3}$$

$$\frac{\partial \alpha}{\partial t} + \nabla \alpha \mathbf{U} + \nabla \alpha (1 - \alpha)\mathbf{U}_r = 0 \tag{4}$$

C. Control simulation

Simulation can be control by modifying controlDict

application	interFoam;
startFrom	startTime;
startTime	0;
stopAt	endTime;
endTime	1.34075;
deltaT	0.00001;
writeControl	adjustableRunTime;
writeInterval	0.00025;
purgeWrite	0;
writeFormat	ascii;
writePrecision	6;
writeCompression	compressed;
timeFormat	general;
timePrecision	6;
runTimeModifiable	yes;

NOTE: The simulation almost converged so it is only runs for 1.34075 seconds.

VI. RESULTS

As shown in below images captured in paraview with different time span, the water drop and its geometry acts exactly same like rain drop falling in atmosphere. The velocity, alpha.water and P_rgh data extracted while snapping. At time 0, formation of water drop occurs with the help of setfieldsDict. So the velocity and pressure contour remains zero at that time. After that second, water drop starts falling on the pool of water because of gravity and the boundary conditions given in the case and to the 0 file folder. Boundary conditions are discussed above and that remain same during simulation. At time 0.001250 second water drop velocity and pressure produce inside the drop can be visualized. As theory says while drop of water fall on surface of earth from cloud, its shape remains spherical because of Pascal's pressure law.

Time: 0.000000

Time: 0.001250

Time: 0.061500

Time: 0.082500

**Time: 0.084000**

Time: 0.093500
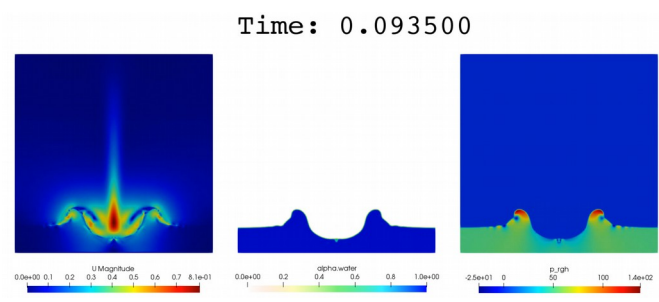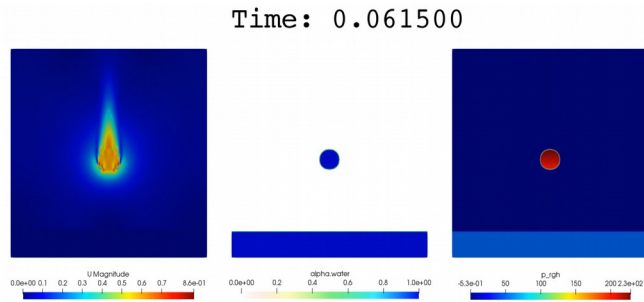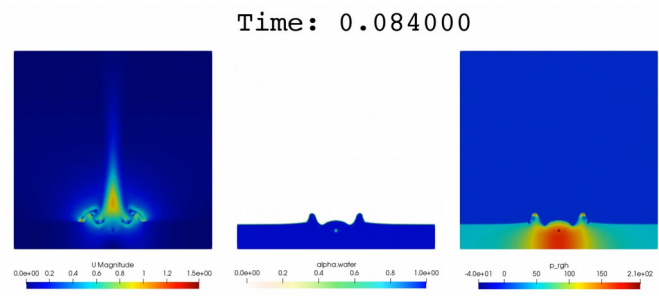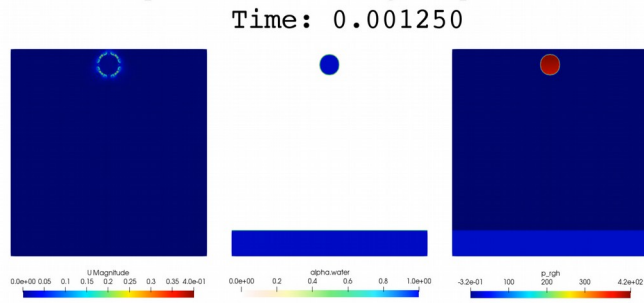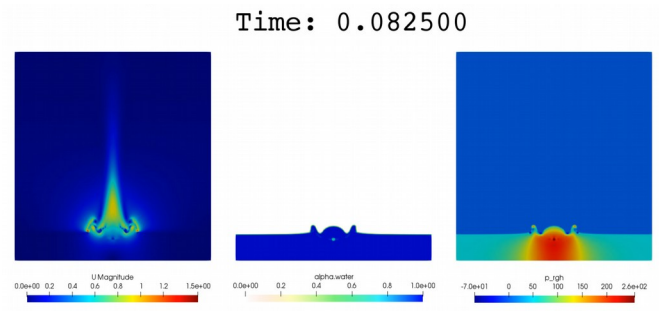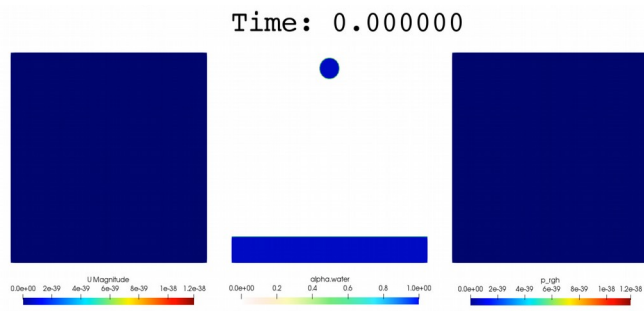
Now, at time 0.61500 seconds, the drop completely set in the movement. Velocity contour shows the effect of gravity on the drop of water and the effect of velocity in air due to movement of water drop.

At the time, while water drop strikes to the pool of water contour of velocity, alpha.water and P_rgh shown below. It shows that, at the time while water drop emerged to the pool of water, due to some air gap, an air bubble appears.

Also, at the time passes the bubble disappear in the atmosphere. Because of the lower height if the water drop this kind of phenomenon occurs. If height of the bubble to the pool makes higher, then after strike of water drop to pool, another water drop release from original drop. Secondary drop may have lesser radius in geometry. But in this case, some waves starts forming due to interference in the pool. That can be visualized by filming an animation from all the values extracted after simulation.
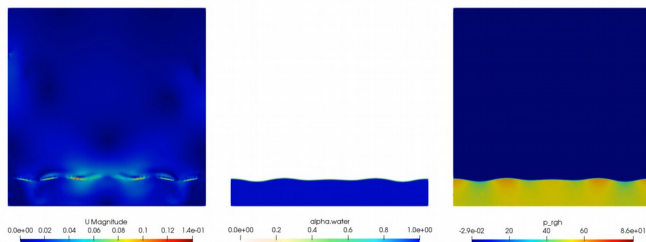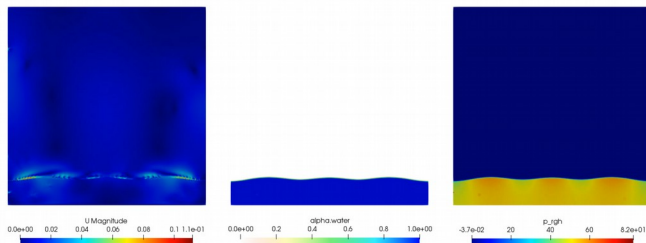
Time: 0.076500

Time: 0.095000

Time: 0.080750

Time: 0.187500
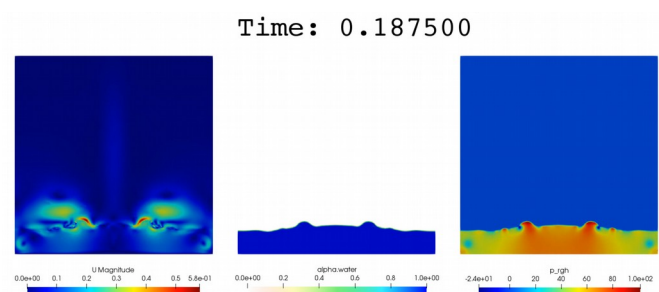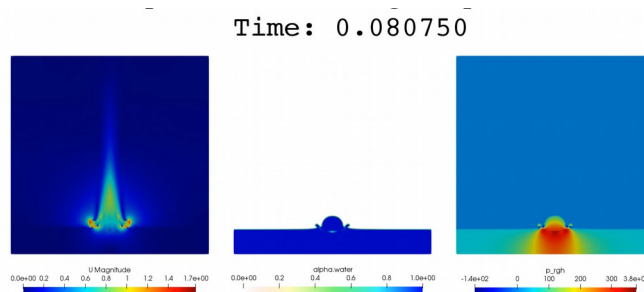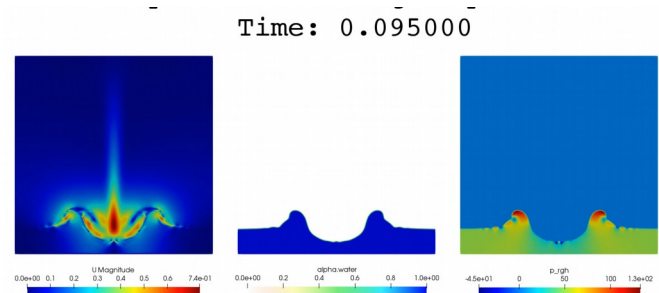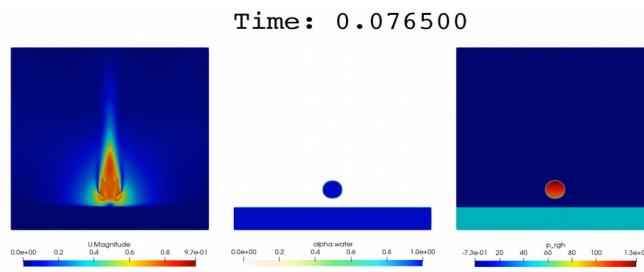
Time: 0.272000
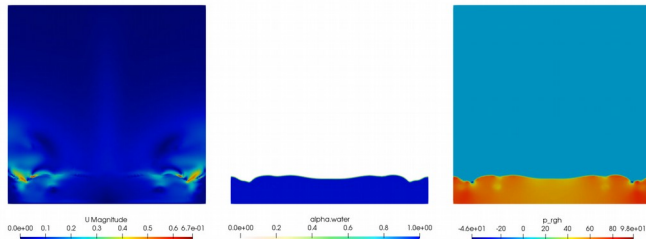
Time: 1.020500
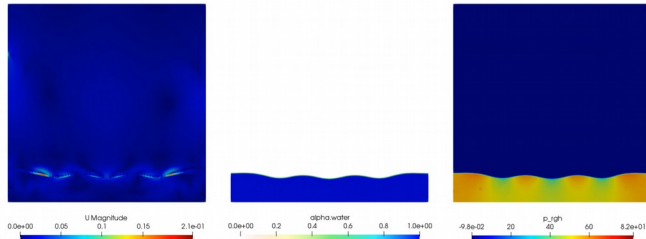
Time: 1.084750

Time: 1.340750

The last three snaps shows the settling down of waves in the pool. After the time passes it will be like as was as starting moment. In the current study, for the time of 1.340750 seconds are enough to understand the effect. It can be run further and can be extract the value of time, when it completely settle down of waves.

CONCLUSIONS

In the case study, one of the most important problems of two-phase flow is numerically simulated; namely, the falling of a droplet in pool under earth gravity conditions. The governing equations are solved by the control volume approach while the moving interfaces are tracked with the level set method. The numerical results showed a remarkable accuracy of the developed numerical method. That can be extended further to include much more complex geometries of two-phase flow applications.

The problem can be further extended by comparing the case with condition of MARS. As scientist looking for water presence on mars.

REFERENCES

[1] Fuster, D., Agbaglah, G., Josserand, C., Popinet, S. and Zaleski, S., Numerical simulation of droplets, bubbles and waves: state of the art, Fluid Dyn. Res. 41: 1-24, 2009.

[2] Lefebvre, A. H., Atomization and sprays, Hemisphere Publishing Corporation, 1989.

[3] Kolev N.I., Multiphase flow dynamics: Thermal and Mechanical Interaction", Springer, 2007.

[4] Yang V., Habiballah M., Hulka J. and Popp M., Liquid Rocket Thrust Chambers: Aspects of Modeling, Analysis, and Design", American Institute of Aeronautics and Astronautics, Inc., 2004.

[5] Linne M., Paciaroni M., Hall T., and Parker T., Ballistic imaging of the near field in a dense spray", Exp. Fluids., 49(4): 911-923, 2006

[6] Menard T., Tanguy S. and Berlemont A., Coupling levelset/VOF/ghost fluid methods: Validation and application to 3Dsimulation of the primary break-up of a liquid jet, Int. J. Multiphase Flow, 33: 510-524, 2007.

[7] Desjardins, O., Moureau, V. and Pitsch, H., An accurate conservative level set/ghost fluid method for simulating turbulent atomization, Journal of Computational Physics, 227: 8395–8416, 2008.

[8] Sussman, M. , Smereka, P. and Osher, S., A level set approach for computing solutions to incompressible two-phase flows, J. Comp. Physics, 114, 146-159, 1994.

[9] Eggers, J., Nonlinear Dynamics and Breakup of Free-Surface Flow, Rews. Modern Phys., 69(3): 865–929, 1997.

[10] Crowe C. T., Troutt T. R. and Chung J. N., Numerical models for two-phase turbulent flows", Annu. Rev. Fluid Mech., 28:11-43, 1996.

[11] Shinjo J. and Umemura A., Simulation of liquid primary breakup: Dynamics of ligament and droplet formation", Int. J. Multiphase Flow, 36(7): 513-532, 2010.

[12] Osher, S. and Fedkiw, R. P.., Level set methods: An overview and some recent results, J. Comp. Phys., 169, 463-502, 2001.

[13] Sethian, J. A. and Smereka, P., Level set methods for fluid interfaces, Annu. Rev. Fluid, Mech., 35, 341-372, 2003.

[14] Nichols B. D. and Hirt C. W., Methods for calculating multi-dimensional, transient free surface flows past bodies, Proc. First Int. Conf. Num. Ship Hydrodynamics Gaithersburg, 20–23, 1975.

[15] Osher S. and Sethian J. A., Fronts propagating with curvature-dependent speed: algorithms based on Hamilton–Jacobi formulations, Journal of Computational Physics, 79: 12–49, 1988.

[16] Li, Z., Jaberi, F. A. and Shih, T., A hybrid Lagrangian–Eulerian particle-level set method for numerical simulations of two-fluid turbulent flows, Int. J. Num. Methods in Fluids, 56: 2271-2300, 2008.

[17] Scardovelli, R. and Zaleski, S., Direct numerical simulation of free-surface and interfacial flow, Annu. Rev. Fluid Mech., 31: 567–603, 1999.

[18] Peters N., Turbulent combustion", Cambridge University Press, Cambridge, UK, 2000.
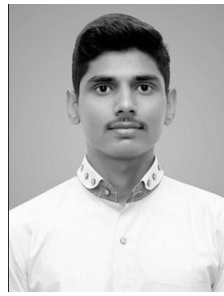
Divyesh Variya (M'97) received the B.E. degree in mechanical engineering from the gujarat technological university, in 2018 and also work as an intern under Prof. Shivasubramanian Gopalakrishnan for FOSSEE (Free and Opensource Software for Education) Project on OpenFOAM in Indian Institute of Technology, Bombay. His research interests include all CAE projects with linear/non-linear structural analysis, computational fluid dynamics, dynamic robotics analysis, failure analysis and prevention, and design development.

Contact:
+91 7777908833
divyeshvariya7@gmail.com