Simulating Multiphase flow over Cylinder and Simple Boat using OpenFOAM

Siddhant Saraswat
Fellow Intern 2018
National Institute of Karnataka, Surathkal
siddhantsaraswat.16me271@nitk.edu.in

ABSTRACT
This report was used to examine the Computational Fluid Dynamics effect using the open source software OpenFOAM, to calculate hydrodynamic values for sea vessels. The solver used was InterFoam. Firstly, it was carried out in a simple 2D geometry, then on a small boat. Later similar simulation was carried on INS Vikramaditya geometry. The vessels were assumed to be operating in deep water conditions.

Author Keywords
Solver details; Geometry; Meshing; Boundary conditons; Postprocessing;

INTRODUCTION

Problem Statement

1. 2D multiphase flow over an obstacle.
2. Multiphase flow over simple geometry such as a small boat.
3. Multiphase flow over complicated geomtry such as INS Vikramaditya.

Solver Details

OpenFOAM Multiphase provides various solvers to work with. InterFOAM solver was chosen as it met the conditions possed by the given problem statement. The official definition of the solver is " **Solver for 2 incompressible, isothermal immiscible fluids using a VOF (Volume of Fluid) phase-fraction interface capturing approach.**"

Various other features of the solver are as follows:

1. Transient
2. Laminar and turbulent

Governing Equations

The law of conservation of mass states that the net trnsport of the mass across the boundaries of a system is zero if the sources are not considered. In general terms, it means that mass can neither be created nor destroyed. **Continuity equation**

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

The momentum equation gives the flow characteristics. **Momentum Equation**

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

Here,

$$\frac{\partial \rho \mathbf{U}}{\partial t} : \textbf{Unsteady Acceleration} \tag{3}$$

$$\nabla.\rho\mathbf{U}\mathbf{U} : \textbf{Convective Acceleration} \tag{4}$$

$$-\nabla P : \textbf{Pressure Gradient} \tag{5}$$

$$\nabla\rho\gamma[2S] : \textbf{Viscosity Forces} \tag{6}$$

$$F_t : \textbf{Turbulent Term} \tag{7}$$

Volume of Fluid

According to ANSYS Fluent, 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.

$$\rho = \alpha\rho_l + (1-\alpha)\rho_g \tag{8}$$

$$\frac{\partial \alpha}{\partial t} + \nabla\alpha\mathbf{U} + \nabla\alpha(1-\alpha)\mathbf{U}_r = 0 \tag{9}$$

α is the phase fraction, ranging from 0 to 1. 0 denoting gas, while 1 denoting the fluid.

SIMPLE 2D CASE

Geometry

To test out the InterFOAM solver for the first time, a simple rectangular two dimensional block with circular obstacle was selected. This geometry was created using a simple **blockMeshDict** file.

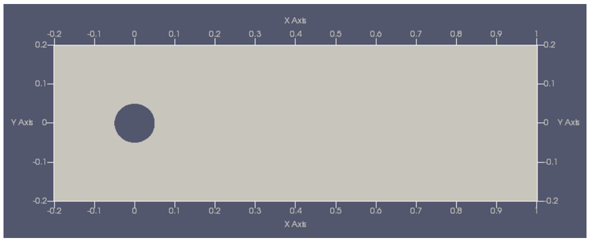

Figure 1. 2D geometry

The boundaries are as follows:

1. **top** type **patch**
2. **bottom** type **symmetryPlane**
3. **inlet** type **patch**
4. **outlet** type **patch**
5. **cylinder** type **wall**
6. **frontAndBack** type **empty**

Meshing

The meshing for this simple geomerty was achieved from the **blockMeshDict** file present in **system** folder. All geometries in OpenFoam are three dimensioanl in nature. In order to simulate a two dimensional case, the number of cells in the z-direction were taken to be one. Also, the front and back faces were taken to be **empty** type.

Various boundary type were mentioned in **section 2.1**

Since it is a simple geometry, the meshing can be done by simply typing the **blockMesh** command in the terminal window.

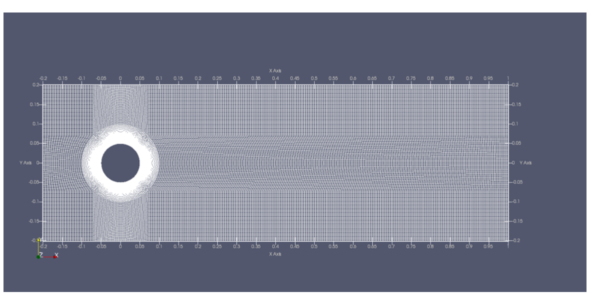

Figure 2. 2D geometry

The quality of the mesh can be check by typing **checkMesh** in the terminal window.

Overall number of cells of each type:

hexahedra: 42400

prisms: 0

wedges: 0

pyramids: 0

tet wedges: 0

tetrahedra: 0

polyhedra: 0

The meshing of the simple 2D case is complete.

Boundary Conditions

The complete table of abbreveations used is at the end of this document

The boundary conditions were achieved by the iterative trials.

Boundary	Type	U	p_rgh	alpha
Inlet	patch	FV(1.668)	FFP	FV(0)
Outlet	patch	OPMV	ZG	VHFR
Bottom	SP	SP	SP	SP
Top	patch	PIOV	TP	IO
Cylinder	wall	MWV	FFP	ZG
frontAndBack	empty	empty	TP	empty

Table 1. Boundary Conditions

Boundary	Type	k	omega	nut
Inlet	patch	FV(0.00015)	FV(2)	FV(5e-07)
Outlet	patch	IO	IO	ZG
Bottom	SP	SP	SP	SP
Top	patch	IO	IO	IO
Cylinder	wall	kqRWF	OWF	nkWF
frontAndBack	empty	empty	empty	empty

Table 2. Boundary Conditions

Formulae used

Kinetic Energy Equation Equation

$$k = \frac{3}{2}.(I|U_{ref}|)^2 \tag{10}$$

Omega Equation

$$\omega = \frac{k^0.5}{C_\mu L} \tag{11}$$

Kinetic ViscosityEquation

$$\gamma_t = 5 * 10^-7 \tag{12}$$

ControlDict File

StartTime 0

endTime 500

deltaT 0.001

postProcessing

Figure 3. 2D geometry at iteration 0

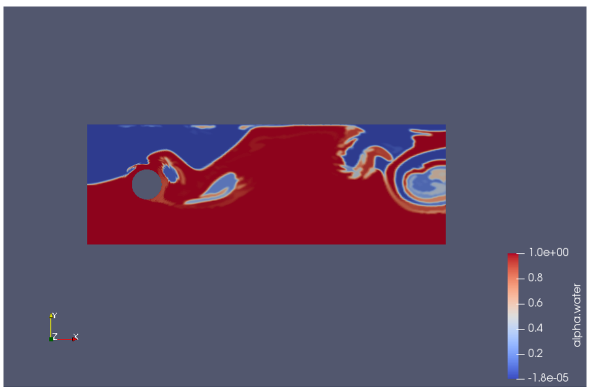

Figure 4. 2D geometry at iteration100

Figure 5. 2D geometry at iteration 299

Validation

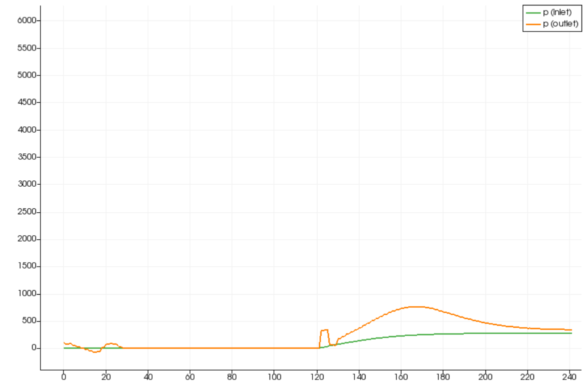

Figure 6. 2D geometry Simulation

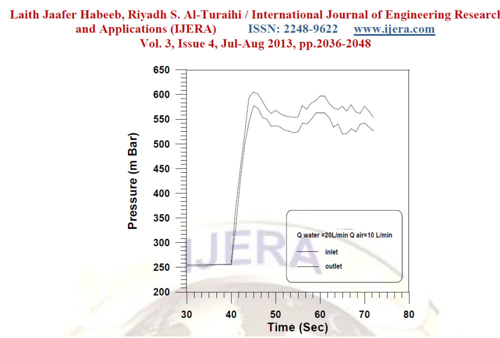

Figure 7. 2D geometry Simulation result

Figure 7 shows established results backed with experimental data, but the obstacle is in the shape of a triagle.

SIMPLE BOAT

A small boat geometry was obtained from the grabCAD website. The downloaded file was a **.step** file. In order to proceed forward with the analysis, the file should either be in **.stl** or **.obj** format. This was done through the CAD package **Salome**.

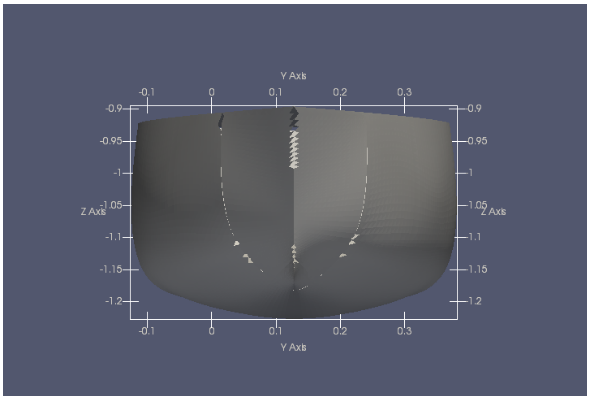

Figure 8. Front View

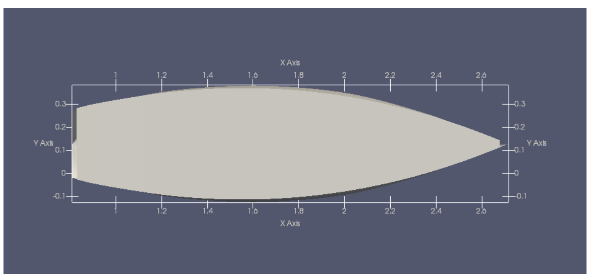

Figure 9. Top View

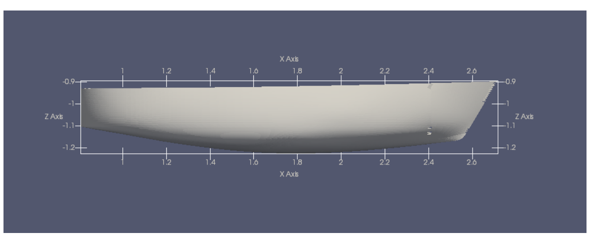

Figure 10. Side View

Meshing

The domain was created using the **blockMesh** utility. The geometry was converted to **.stl** format, and placed in the **triSurface** folder. The meshing for the Small boat case was controlled from the **SnappyHexMeshDict** file present in the **system** folder. After generating the **blockMesh**, it was viewed in the **paraFoam** package. Then a co-ordinate was selected such that it is present in the blockMesh , but outside the geometry.

	Min	Max
X	-10	10
Y	-5	5
Z	-2	4

Table 3. BlockMesh domain specifications

The selected co-ordinate is: x=0 y=15 z=6

Various important sections of the **snappyHexMeshDict** file are shown.

catellatedMesh true;

snap true;

addLayers false;

Expicit feature edge refinement

features

(file "ship.eMesh" level 3;);

---

refinementSurfaces ship //surface-wise min and max refinement level

level (2 3);

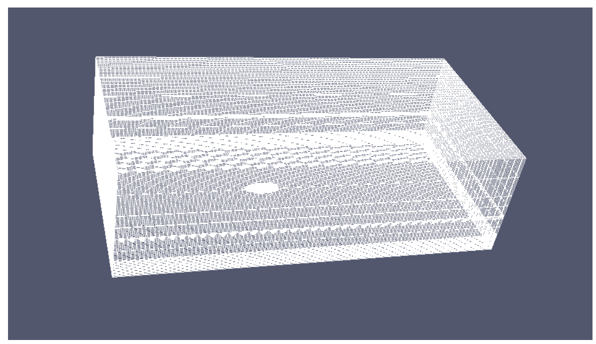

Figure 11. Simple Boat Mesh

Boundary Conditions

The complete table of abbreveations used is at the end of this document

The boundary conditions were achieved by the iterative trials.

Boundary	Type	U	p_rgh	alpha
Inlet	patch	FV(15.443)	FFP	FV(0)
Outlet	patch	OPMV	ZG	VHFR
Side	SP	SP	SP	SP
Atmosphere	patch	PIOV	TP	IO
Ship	wall	MWV	FFP	ZG

Table 4. Boundary Conditions

Boundary	Type	k	omega	nut
textbfInlet	patch	FV(0.04504)	FV(12.403)	FV(5e-07)
Outlet	patch	IO	IO	ZG
Side	SP	SP	SP	SP
Atmosphere	patch	IO	IO	IO
Ship	wall	kqRW	OWF	nkWF

Table 5. Boundary Conditions

Formulae used

Kinetic Energy Equation Equation

$$k = \frac{3}{2}.(I|U_{ref}|)^2 \tag{13}$$

Omega Equation

$$\omega = \frac{k^0.5}{C_\mu L} \tag{14}$$

Kinetic ViscosityEquation

$$\gamma_t = 5 * 10^-7 \tag{15}$$

ControlDict File

StartTime 0

endTime 150

deltaT 0.05

postProcessing

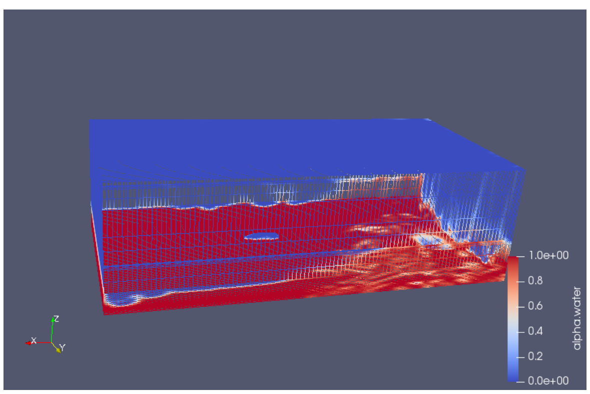

Figure 12. Alpha field for simple boat case

The drag coefficient was found to be **0.067**

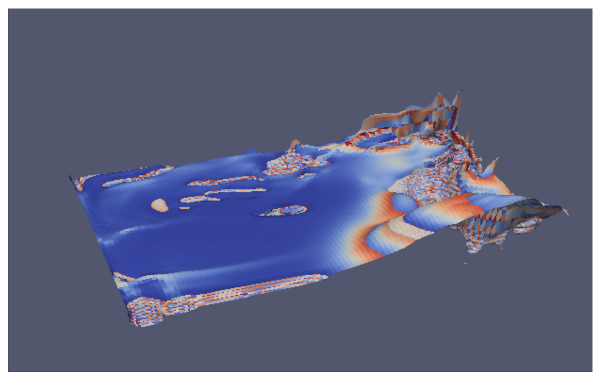

Figure 13. Interface between water and air phase contour