Summer Fellowship Report

On

MultiPhase Flows

Submitted by

Siddhant Saraswat
Variya Divyesh

Under the guidance of

Prof.Shivasubramanian Gopalakrishnan
Mechanical Engineering Department
IIT Bombay

July 30, 2018

Contents

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 and finally on INS Vikramaditya. The vessels were assumed to be operating in deep water conditions.

Chapter 1

Introduction

MultiPhase

Interaction between flow of two or more phases, such that the interface between the phases is influenced by their motion. The term 'MultiPhase' in this report refers to the interaction between the air and water phase.

Solver

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 following equations play a major role:
Continuity equation

$$\nabla.\mathbf{U} = 0 \tag{1.1}$$

Momentun Equation

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla.\rho \mathbf{U}\mathbf{U} = -\nabla P + \nabla \rho \gamma [2S] + F_t \tag{1.2}$$

Volume of Fluid

$$\rho = \alpha \rho_l + (1 - \alpha)\rho_g \tag{1.3}$$

$$\frac{\partial \alpha}{\partial t} + \nabla \alpha \mathbf{U} + \nabla \alpha (1 - \alpha)\mathbf{U}_r = 0 \tag{1.4}$$

Volume of Fluid Method

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.

CFD approach in this project

1. Choosing an appropriate OpenFOAM solver.
2. Convert the geometry into readable format as prescribed by OpenFOAM, i.e., .obj or .stl
3. Create an appropriate blockMeshDict file
4. Modify the snappyHexMeshDict file according to the geometry and need.
5. Modify the controlDict file.
6. Modify the setFieldsDict file.
7. Modify the initial boundary conditions in the 0 folder.
8. Run the interFoam solver, either in serial or parallel computing.
9. postprocess the results.

Chapter 2

Geometry

2.1 Simple 2D Case

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

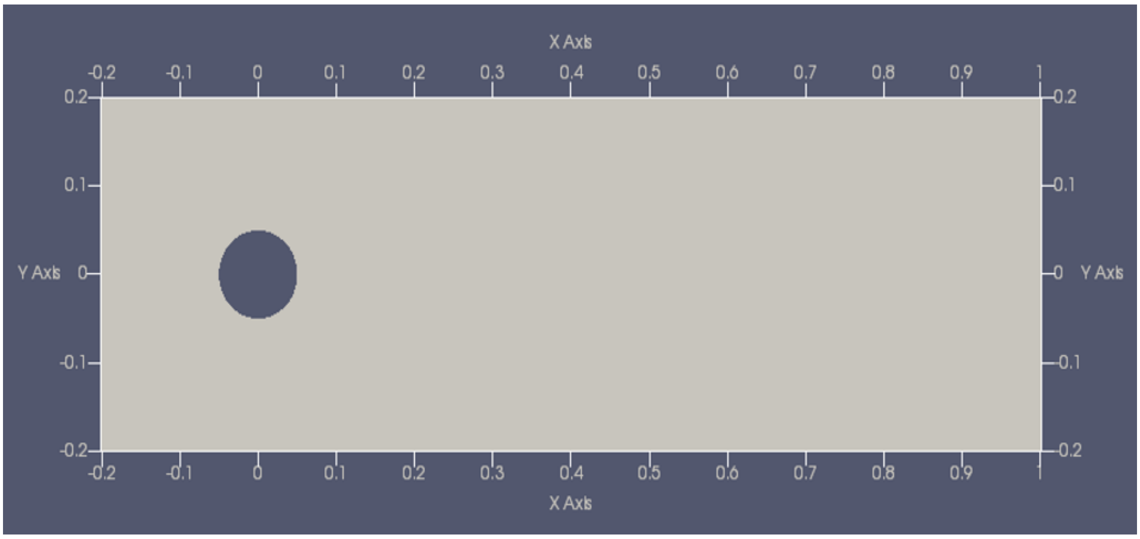

Figure 2.1: 2D geometry

simple **blockMeshDict** file.

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**

2.2 Small Boat Case

Next, 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**.

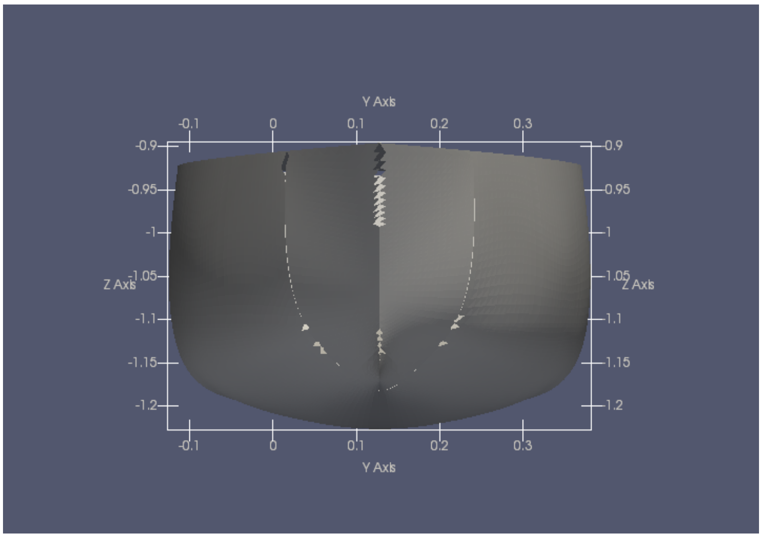

Figure 2.2: Front view of small boat

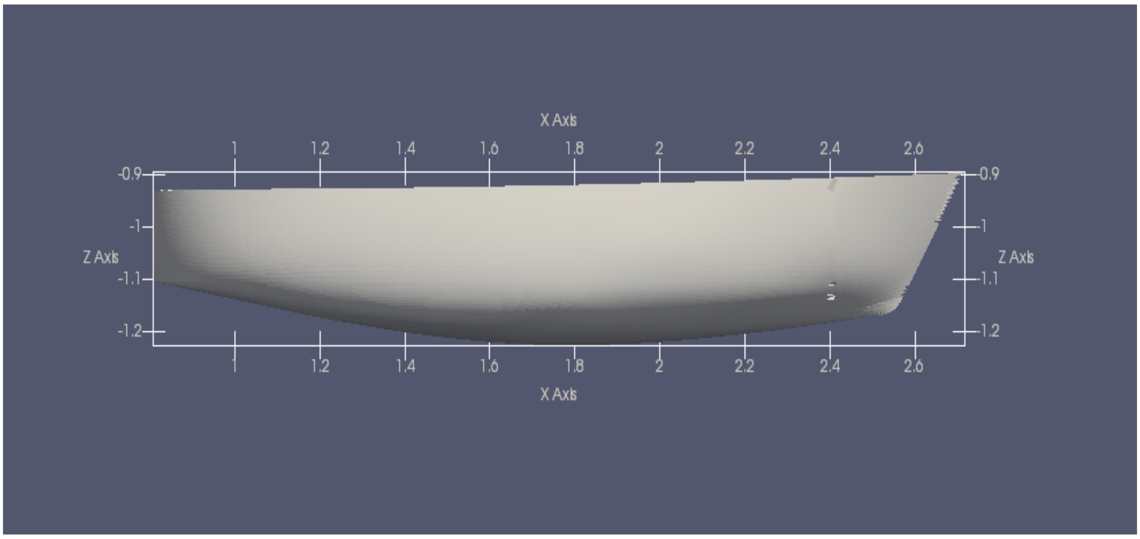

Figure 2.3: Side view of small boat

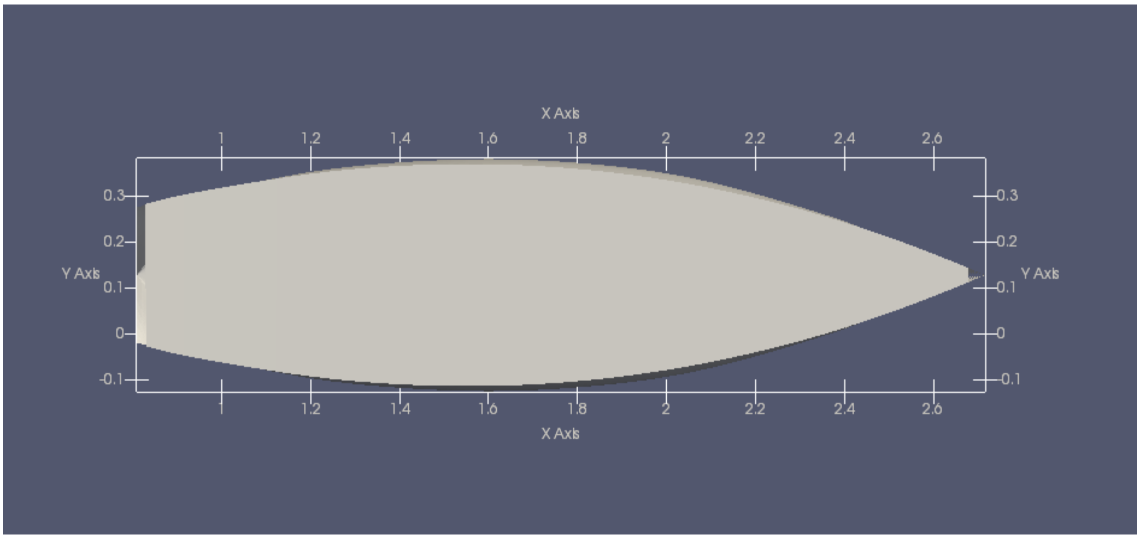

Figure 2.4: Top view of small boat

2.3 INS Vikramaditya Case

Finally, the geometry for INS Vikramaditya was used. Again, it was in **.step** format, and was converted to **.obj** format.

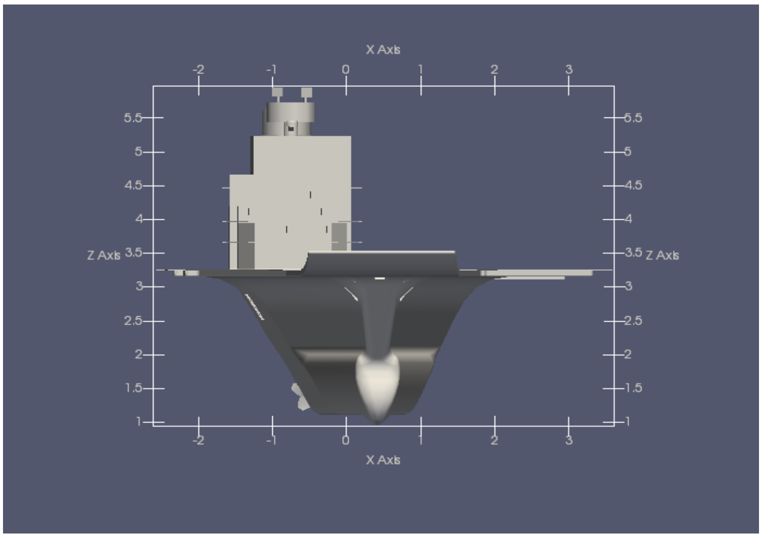

Figure 2.5: Front view of small boat

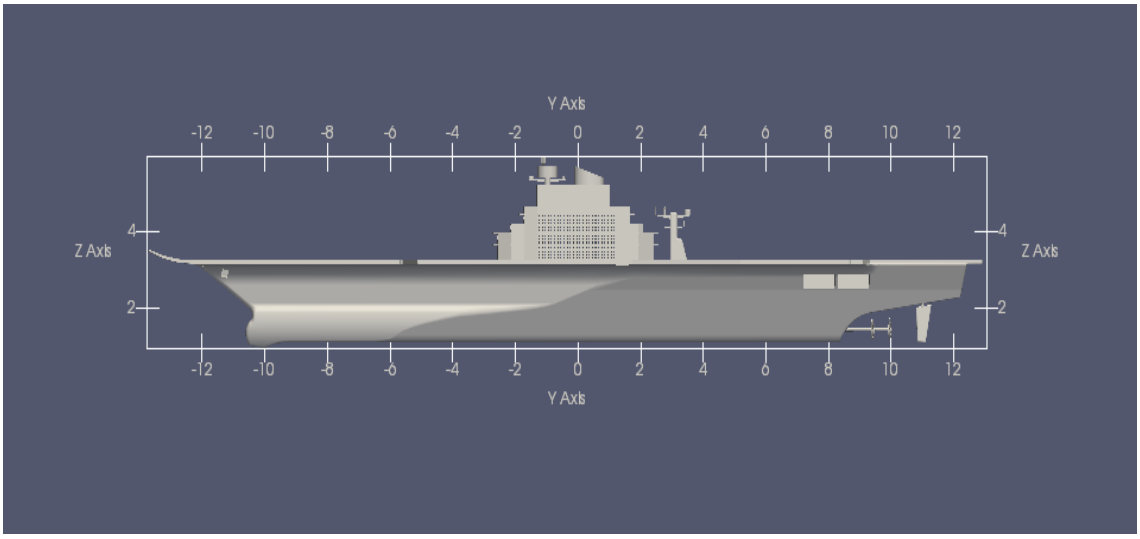

Figure 2.6: Side view of small boat

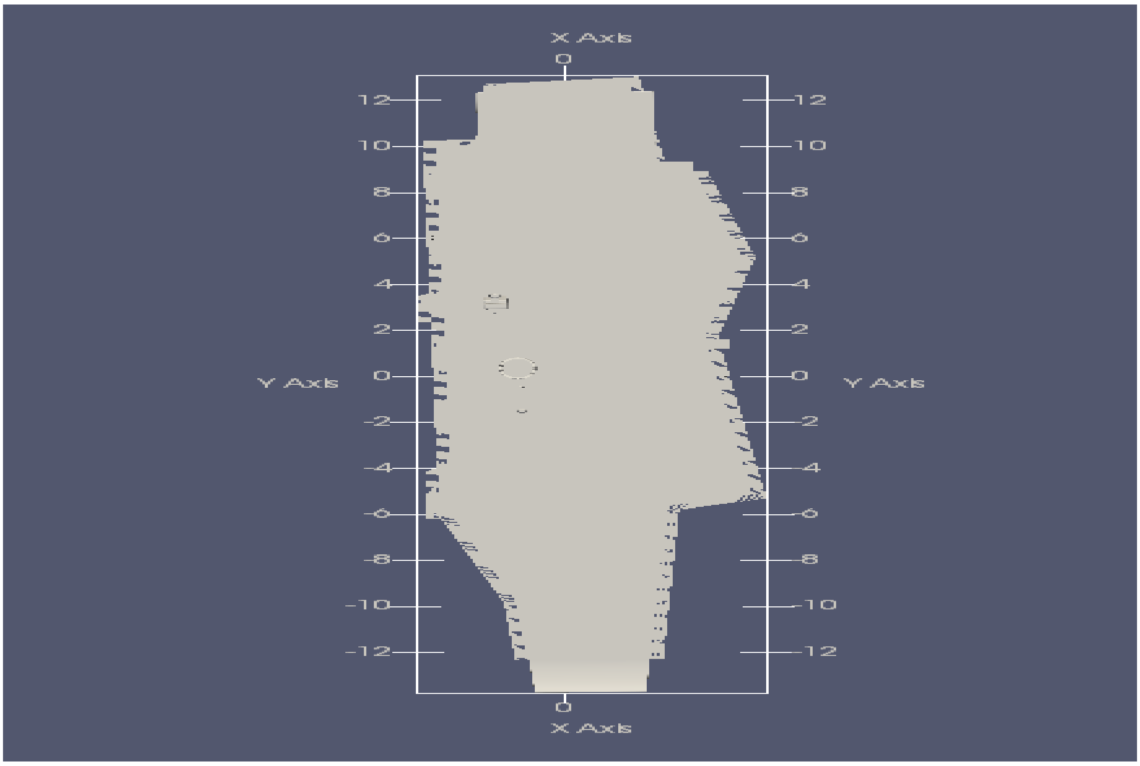

Figure 2.7: Top view of small boat

Chapter 3

Meshing

3.1 2D geometry

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.

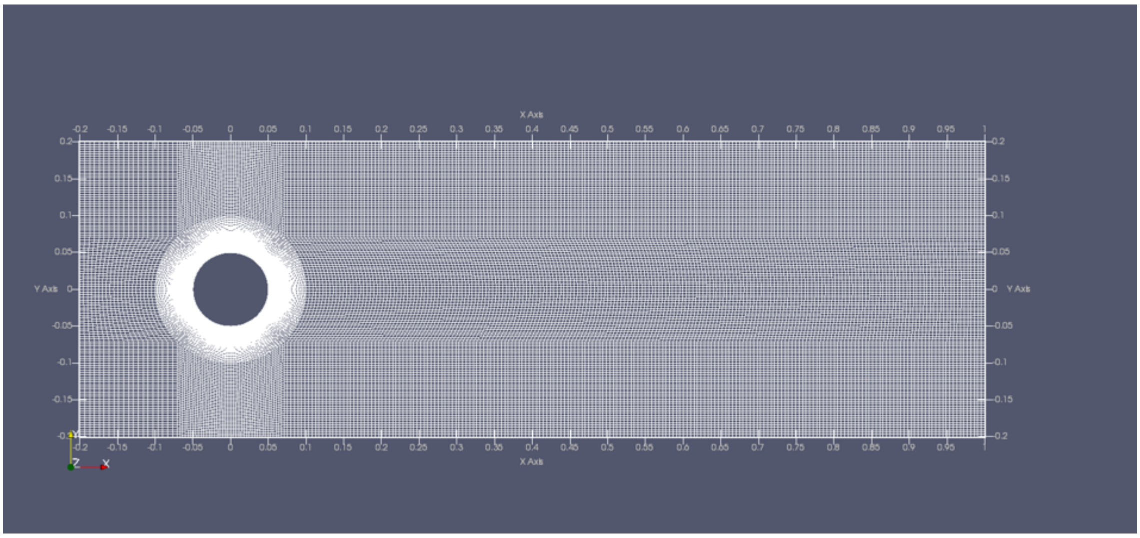

Figure 3.1: 2D mesh

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
```

3.2 Small Boat Case

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.

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);
        }
}
```

After modifying the **snappyHexMeshDict** file, the following commands are run in the terminal window.

```
blockMesh
surfaceFeatureExtract
snappyHexMesh -overwrite
```

The meshing can be viewed in the **paraview** window.

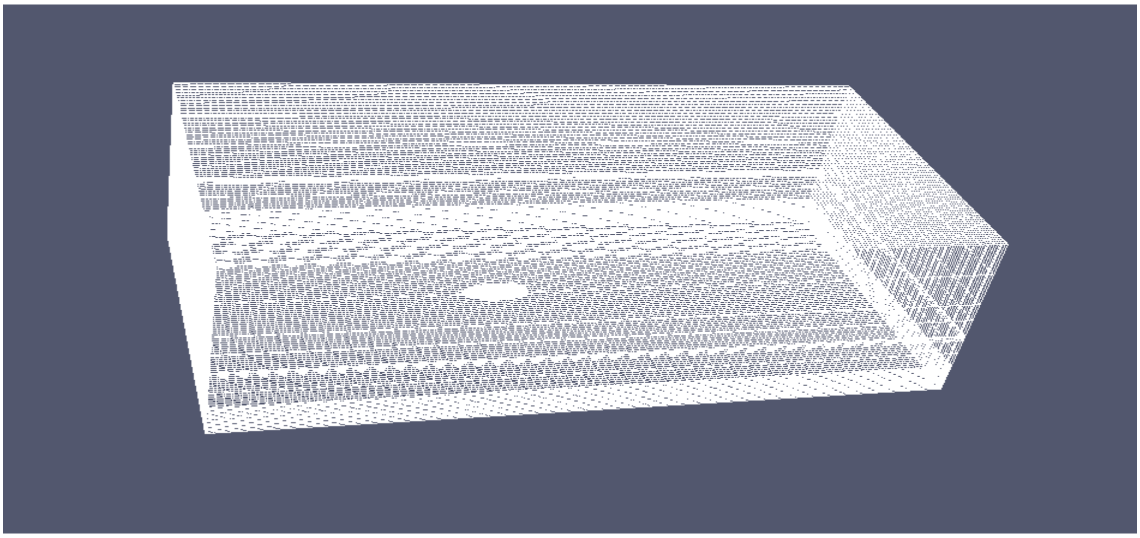

Figure 3.2: Small Boat mesh

3.3 INS Vikramaditya Case

....Di-
vyesh.. ....

Chapter 4

Initial and Boundary Conditions

The initial and bundary conditions are situated in the zero time folder, present in the case directory. The list of abbreveations used in the following table are:

```
SP:      Suymmetry Plane
FV:      Fixed Value
BP:      Buoyant Pressure
ZG:      Zero Gradient
TP:      Total Pressure
WF:      Wall Function
PIOV:    Pressure Inlet Outlet Velocity
IO:      Inlet Outlet
```

4.1 2D geometry

Boundary Conditions				
Boundary	Type	U	p_rgh	alpha
Inlet	patch	FV(1.668)	Fixed Flux Pressure	FV(0)
Outlet	patch	Outlet Phase Mean Velocity	ZG	Variable Height Flow Rate
Bottom	SP	SP	SP	SP
Top	patch	PIOV	TP	IO
Cylinder	wall	Moving Wall Velocity	Fixed Flux Pressure	ZG
frontAndBack	patch	-	TP	empty

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	kqRWallFunction	OmegaWallFunction	nutkWallFunction
frontAndBack	patch	-	-	-

Formulae used

Kinetic Energy Equation Equation

$$k = \frac{3}{2}.(I|U_{ref}|)^2 \tag{4.1}$$

Omega Equation

$$\omega = \frac{k^0.5}{C_\mu L} \tag{4.2}$$

Kinetic ViscosityEquation

$$\gamma_t = 5 * 10^-7 \tag{4.3}$$

4.2 Small Boat Case

Boundary Conditions				
Boundary	Type	U	p_rgh	alpha
Inlet	patch	FV(15.433)	Fixed Flux Pressure	FV(0)
Outlet	patch	Outlet Phase Mean Velocity	ZG	Variable Height Flow Rate
Side	SP	SP	SP	SP
Atmosphere	patch	PIOV	TP	IO
ship	wall	Moving Wall Velocity	Fixed Flux Pressure	ZG

Boundary Conditions				
Boundary	Type	k	omega	nut
Inlet	patch	FV(0.04504)	FV912.403)	FV(5e-07)
Outlet	patch	IO	IO	ZG
Side	SP	SP	SP	SP
Atmosphere	patch	IO	IO	IO
ship	wall	kqRWallFunction	OmegaWallFunction	nutkWallFunction

4.3 INS Vikramaditya Case

Boundary Conditions				
Boundary	Type	U	p_rgh	alpha
Inlet	patch	FV(15.433)	Fixed Flux Pressure	FV(0)
Outlet	patch	Outlet Phase Mean Velocity	ZG	Variable Height Flow Rate
Side	SP	SP	SP	SP
Atmosphere	patch	PIOV	TP	IO
ship	wall	Moving Wall Velocity	Fixed Flux Pressure	ZG

Boundary Conditions				
Boundary	Type	k	omega	nut
Inlet	patch	FV(0.04504)	FV912.403)	FV(5e-07)
Outlet	patch	IO	IO	ZG
Side	SP	SP	SP	SP
Atmosphere	patch	IO	IO	IO
ship	wall	kqRWallFunction	OmegaWallFunction	nutkWallFunction

Chapter 5

Post Processing

5.1 2D geometry

Figure 5.1: Alpha Field at iteration 0

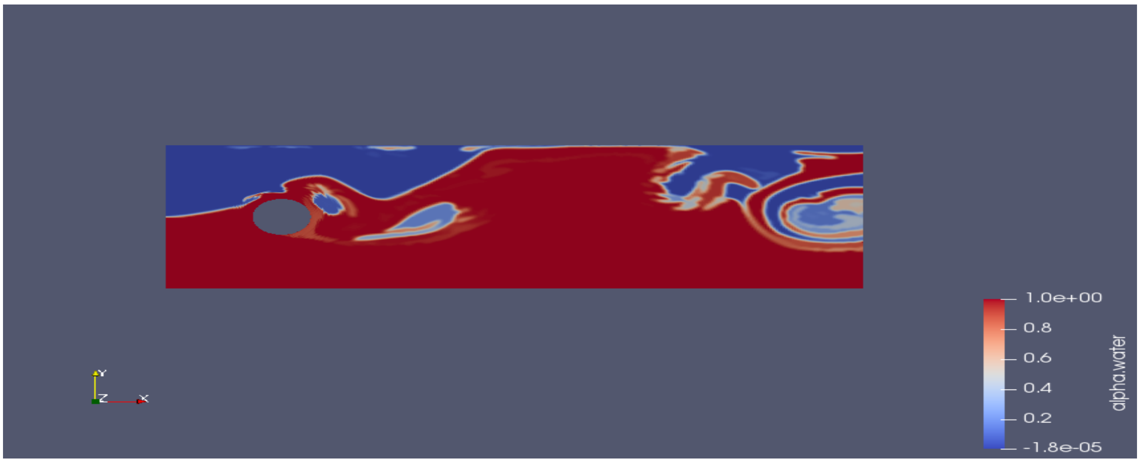

Figure 5.2: Alpha Field at iteration 100

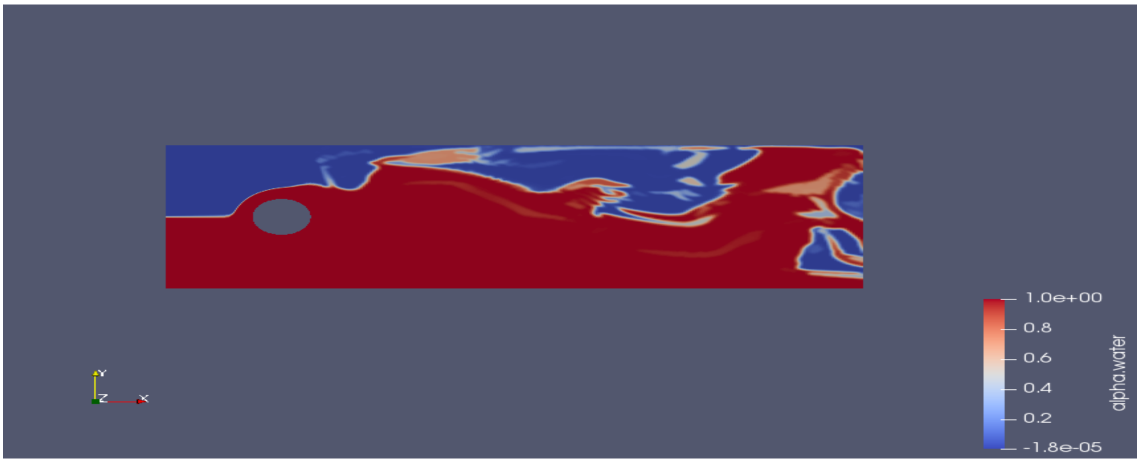

Figure 5.3: Alpha Field at iteration 299

5.2 Small Boat Case

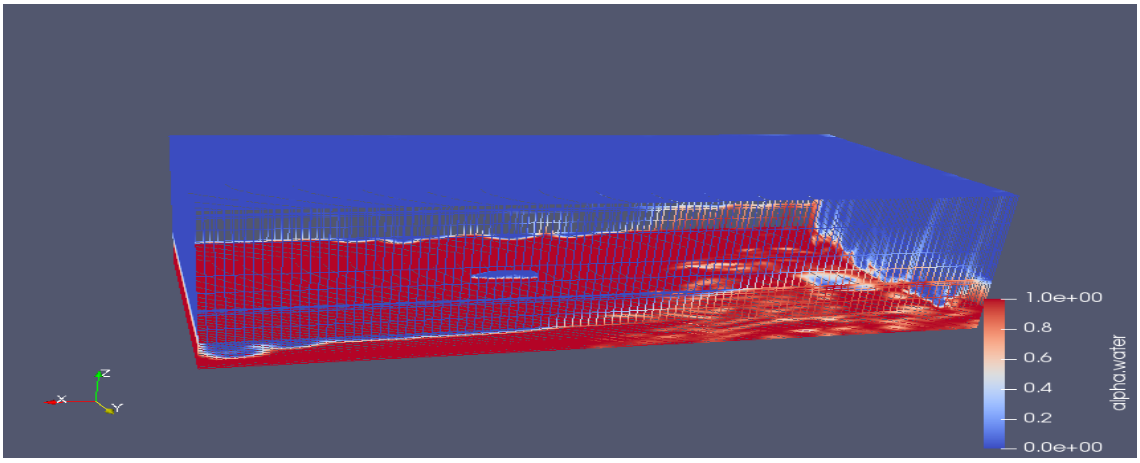

Figure 5.4: Alpha Field

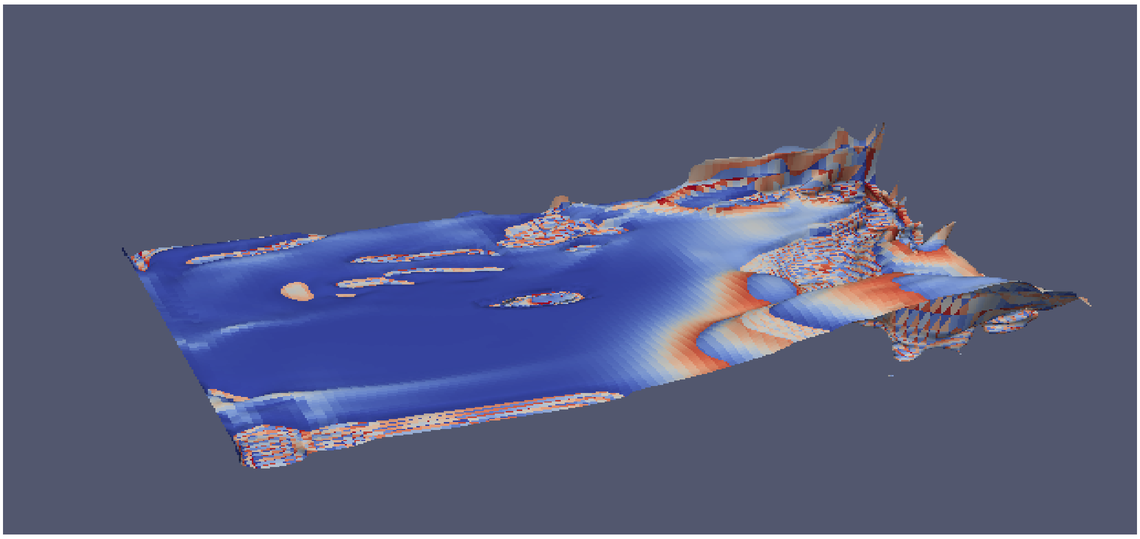

Figure 5.5: Contour of the interface between air and water field

5.3 INS Vikramaditya

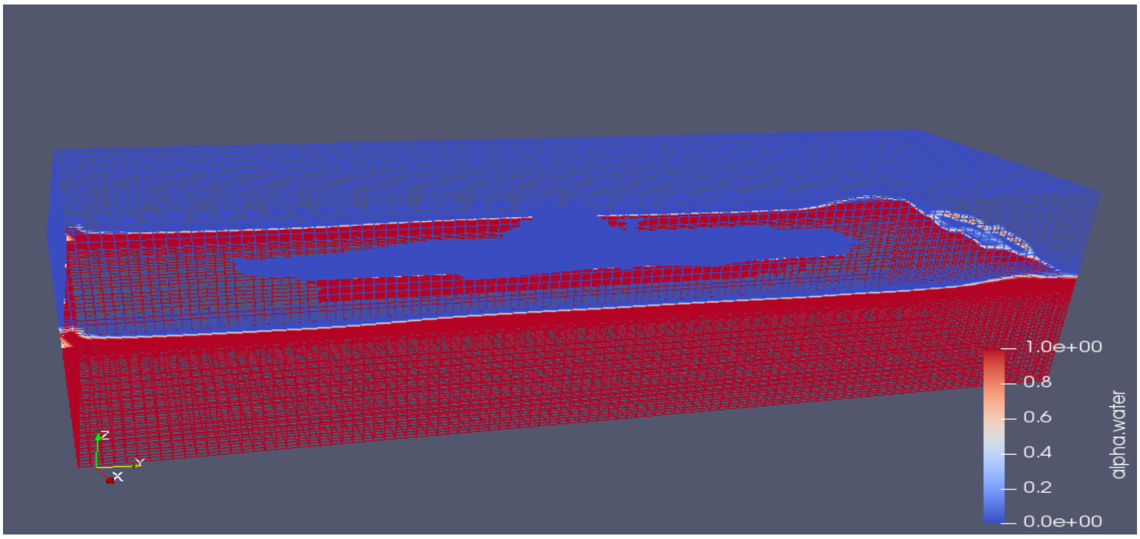

Figure 5.6: Alpha Field

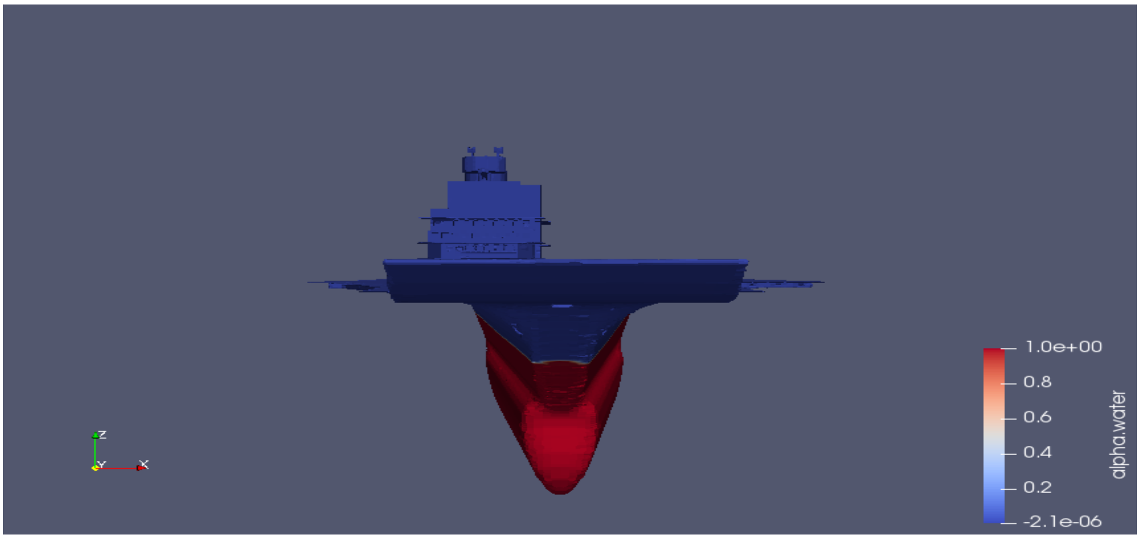

Figure 5.7: Alpha Field

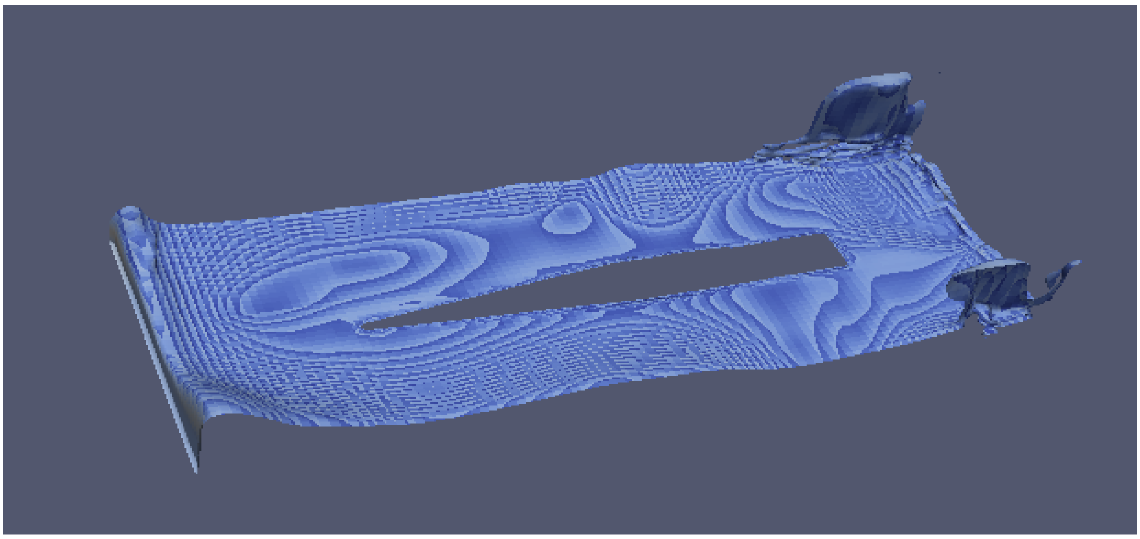

Figure 5.8: Contour of the interface between air and water field