# MultiPhase flow simulation over INS Vikramaditya

Divyesh Variya, OpenFOAM team, FOSSEE Project, IIT Bombay
Government Engineering college, Valsad

***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 on an Indian aircraft carrier INS Vikramaditya. The solver used was InterFoam. The vessels were assumed to be operating in deep water conditions.**

***Keywords*—*INS Vikramaditya, OpenFOAM, interFoam.***

## I. INTRODUCTION

INS Vikramaditya (Sanskrit, Vikramditya meaning ”Brave as the Sun”) is a modified Kiev-class aircraft carrier which entered into service with the Indian Navy in 2013. She has been renamed in honour of Vikramaditya, a legendary emperor of Ujjain, India.

Originally built as Baku and commissioned in 1987, the carrier served with the Soviet Navy and later with the Russian Navy (as Admiral Gorshkov) before being decommissioned in 1996. The carrier was purchased by India on 20 January 2004 after years of negotiations at a final price of $2.35 billion. The ship successfully completed her sea trials in July 2013[20] and aviation trials in September 2013.

She was commissioned on 16 November 2013 at a ceremony held at Severodvinsk, Russia. On 14 June 2014, the Prime Minister of India formally inducted INS Vikramaditya into the Indian Navy and dedicated her to the nation.

**Figure 1: INS Vikramaditya in service**

### A. CFD Approach in this project.

These steps followed in the 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.

### B. Geometry

The Geometry of INS Vikramaditya taken directly from GrabCAD. Dimensions of INS Vikramaditya are 283.5 m (930 ft) Length, 59.8 m (196 ft) and Draught 10.2 m(33 ft). It carries maximum 26 Mikoyan MiG-29K multirole fighters with 10 Kamov Ka-31 AEW&C and Kamov Ka-28 ASW helicopters. It has a 14-degree ski-jump Aviation facility. It is extensively modified of the Kiev-class aircraft carrier.

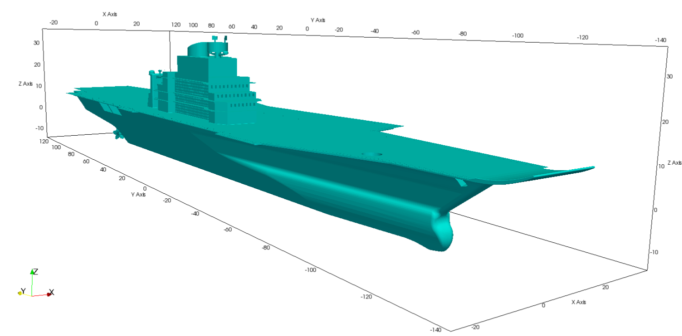

**Figure 2: INS Vikramaditya CAD Model**

### C. Meshing

The meshing for this simulation was done using the OpenFOAM Mesh utilities.BlockMesh and SnappyHexMesh were used to mesh the model. The mesh is little coarse but it can be refined with high refinement levels. Results of checkMesh and Final mesh are:

| | |
|---|---|
| Domain | (-70 -200 -40)(70 200 80) |
| Max Skewness | 11.2556 |
| Max Aspect ratio | 49.1351 |

**Table 1: checkMesh**

| | |
|---|---|
| Edge refinement level | 3 |
| surface Refinement level | (2 3) |
| No. of Layers | 3 |

**Table 2: snappyHexMeshDict**

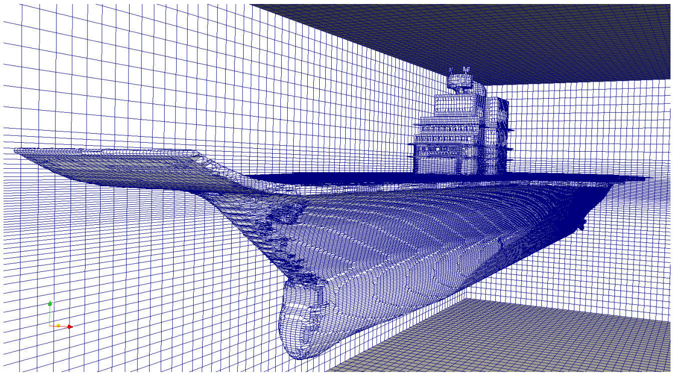

**Figure 3: Cut-section of 3D Mesh**

## II. ANALYSIS

The CFD analysis of the MultiPhase flow over INS Vikramaditya was done using the software OpenFOAM (v-5.0).

### A. Boundary Conditions

Air an water enters the computational domain at a freestream velocity $u_\infty = 30knots$ around $33km/h$ normal to the inlet surface. The reference pressure was taken to be 0 atm. The pressure at inlet was kept fixed and equal to the atmospheric pressure. The deep water or the bottom surface of the domain was defined as symmetry plane and the front and back surfaces of the domain were also defined as symmetry plane.
The list of abbreveations used in the following table are:

1) SP: Suymmetry Plane
2) FV: Fixed Value
3) BP: Buoyant Pressure
4) ZG: Zero Gradient
5) TP: Total Pressure
6) WF: Wall Function
7) PIOV: Pressure Inlet Outlet Velocity
8) IO: Inlet Outlet

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

**Table 3: Boundary conditions for U,p_rgh & alpha**

### B. Turbulence Model

The kOmegaSST turbulence model of OpenFOAM is used for this simulation. This model is a combination of $k-\omega$ and $k-\epsilon$ models. The initial values of k and Omega were calculated to be 0.04504 and 12.403 respectively. Following are the formulae used in the calculations:

**Kinetic Energy Equation Equation**

$$k = \frac{3}{2}.(I|U_{ref}|)^2 \tag{1}$$

**Omega Equation**

$$\omega = \frac{k^0.5}{C_\mu L} \tag{2}$$

**Kinetic ViscosityEquation**

$$\gamma_t = 5 * 10^-7 \tag{3}$$

| **Boundary** | **k** | **omega** | **nut** |
|---|---|---|---|
| **Inlet** | FV(0.04504) | FV(12.403) | FV(5e-07) |
| **Outlet** | IO | IO | ZG |
| **Side** | SP | SP | SP |
| **Atmosphere** | IO | IO | IO |
| **ship** | kqRWall Function | OmegaWall Function | nutkWall Function |

**Table 4: Boundary conditions for k,omega & nut**

### C. Setup fluid levels

INS Vikramaditya is draught at 10.2 m from its extreme bottom surface. To define the specific level of water and air, its setup in the setFieldsDict file.
Initially, its taken that there is only air inside domain then water level specified by bounding box (-999 -999 -999)(999 999 25).

### D. 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**

$$\nabla.\mathbf{U} = 0 \tag{4}$$

**Momentun Equation**

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla.\rho\mathbf{U}\mathbf{U} = -\nabla P + \nabla \rho\gamma[2S] + F_t \tag{5}$$

**Volume of Fluid**

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

$$\frac{\partial \alpha}{\partial t} + \nabla\alpha\mathbf{U} + \nabla\alpha(1-\alpha)\mathbf{U}_r = 0 \tag{7}$$

### E. *Control simulation*

Simulation can be control by modifying controlDict

| | |
|---|---|
| application | interFoam; |
| startFrom | startTime; |
| startTime | 0; |
| stopAt | endTime; |
| endTime | 0.35; |
| deltaT | 0.0005; |
| writeControl | adjustableRunTime; |
| writeInterval | 0.005; |
| purgeWrite | 0; |
| writeFormat | binary; |
| writePrecision | 6; |
| writeCompression | uncompressed; |
| timeFormat | general; |
| timePrecision | 6; |
| runTimeModifiable | yes; |

**NOTE:** The simulation converged so it is only runs for 0.35 seconds.

## III. Results

The results of alpha.water is plotted below for 10 and 20 time steps. Conergence graph is following:

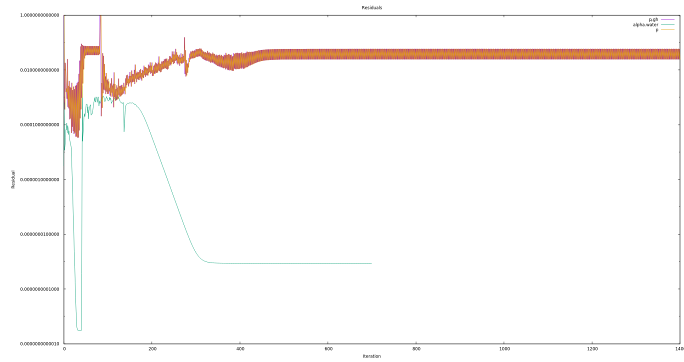

**Figure 4: alpha.water and p_rgh residuals**

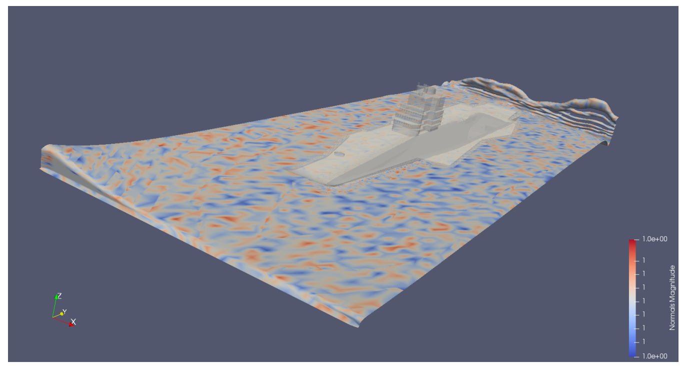

**Figure 5: alpha.water 1**

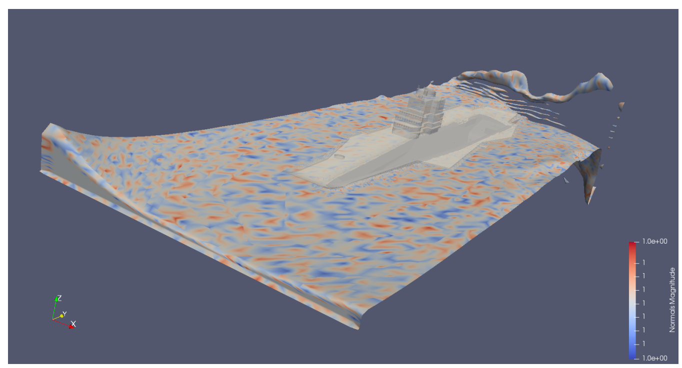

**Figure 6: alpha.water 2**

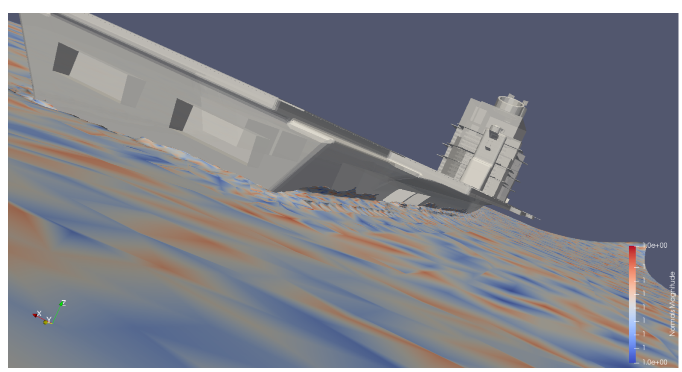

**Figure 7: alpha.water 3**