External Incompressible Flow over hull of ship

Siddhant Saraswat, B. Tech 2nd Year, Mechanical Engineering, National Institute of Technology Karnataka, Surathkal

***Abstract*—This report aims to describe the calculation of steady-state laminar flow field around a hull-like geometry using the software ANSYS and OpenFOAM. This hull-like geometry is assumed to be submerged in water.**

I. Introduction

A hull is the most important body of the ship or any other vessel (such as, a submarine) which includes the bottom and the sides. But a hull does not include the masts, superstructure, rigging, engines and other fittings.

Figure 1: A small rescue boat

***Image source:* dx.doi.org/10.3233/ISP-2011-0068**

Resistance of a ship at a given speed is defined as the force required to tow the ship at the same speed in calm water.

Drag resistance can be divided into two components, they are as follows,

1. The drag exerted by the fluid medium in which the ship is moving. It is also denoted as viscous resistance R_F.
2. The drag generated due to the motion of the ship itself. It is denoted as residuary or wave making resistance R_R.

The total resistance R_T is given by,

$$R_T = R_F + R_R \qquad ...eqn(1)$$

But a more general formula is given by,

$$R_T = R_F + R_R + R_{AA} \qquad ...eqn(2)$$

where, R_{AA} = resistance caused by calm air

It is generally noticed that as the speed of the ship increases, the resistance also increases. This is supported by the diagram presented below.

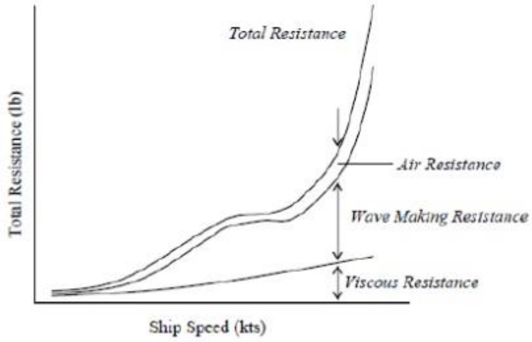


Figure 2: Total resistance versus ship speed

***Image source:* www.usna.edu**

In marine design, many dimensionless coefficients are used. This is done in order to get an idea about the overall performance of the ship.

Some of the most commonly used ones are as follows:

$$C_T = C_V + C_W \qquad ...eqn(3)$$

Where, C_T = coefficient of total hull resistance

C_F = coefficient of viscous resistance

C_R = coefficient of wave making resistance

Here, the coefficient of the total hull resistance is found from the following equation,

$$C_T = \frac{R_T}{\frac{1}{2}\rho V^2 S} \qquad ...eqn(4)$$

Where, R_T = total hull resistance

ρ = water density

V = velocity

S = wetted rea of the underwater hull

Another common dimensionless coefficient 'Froude Number (F_N)' is used.

$$F_N = \frac{V}{\sqrt{gL}} \quad \dots eqn(5)$$

Where, V = velocity

g = acceleration due to gravity

L = length of the ship

A. Geometry

The geometry of the hull of the hip was created using the software Autodesk Inventor 2018 Student Version. Following were the geometrical assumptions take;

Material	Steel (density =7850.00 Kg/m^3)
Total surface area	768.35 m^2
Total volume	1401.724 m^3
Overall length	20 m

Table 1: general material assumptions considered

The file was exported in the STEP format for further processing such as mesh generation.

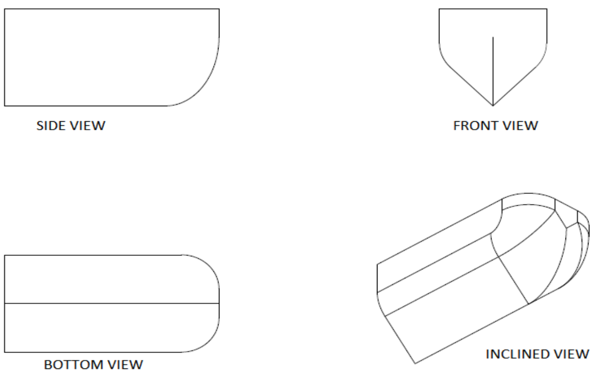


Figure: 2.1

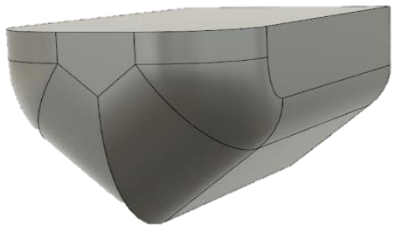


Figure:2.2

Figure 2: 3D view of the geometry

B. Meshing

The meshing for this simulation is done using the software package ANSYS Fluent 16.0 student version.

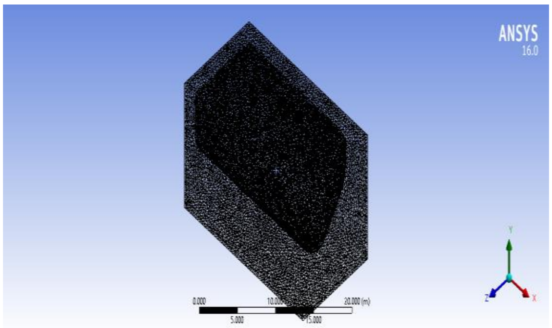


Figure 3: meshing of the geometry

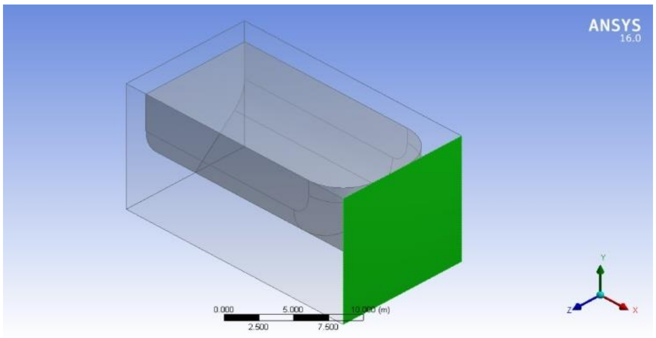
An enclosure was created in ANSYS 16.0 student version so as to define the domain of operation. Then a subtraction Boolean operation was carried out in order to create a hollow void inside the cuboidal enclosure.

II. ANALYSIS

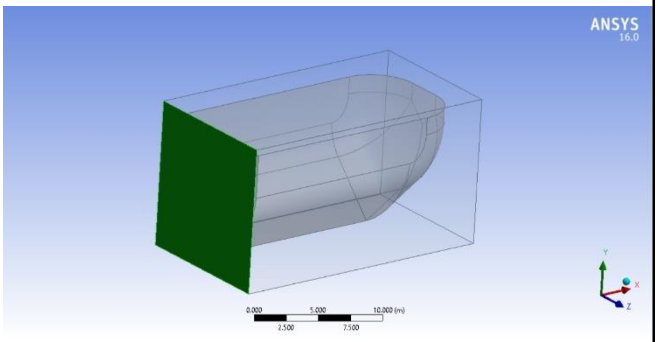
The CFD analysis of the fluid external flow around the hull was done using the software OpenFOAM (v-4.0).

A. *Boundary Conditions*

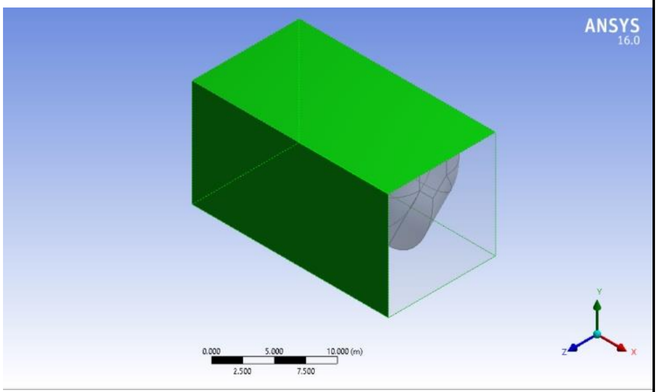
Sea water enters the computational domain at a freestream velocity U_∞ = 10.2889 m/s (20 knots) normal to the inlet surface. The gauge pressure at the outlet was kept fixed.



INLET VIEW(Figure:4.1)



OUTLET VIEW(Figure:4.2)



WALL VIEW(Figure:4.3)

Figure 4: Views of the various boundaries

B. *Laminar Model*

The computational model solves system of Navier-Stokes equations for incompressible fluid flow (implemented in OpenFOAM)

- Mass conservation

$$\nabla.\boldsymbol{u} = 0 \qquad ...eqn(6)$$

(or)

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \qquad ...eqn(7)$$

- Momentum conservation

$$\frac{\partial \boldsymbol{u}}{\partial t} + (\boldsymbol{u}.\nabla)\boldsymbol{u} = \nu\nabla^2 - \nabla p + g \quad ...eqn(8)$$

- Kinematic viscosity

$$\nu = \frac{\mu}{\rho} \qquad ...eqn(9)$$

Where, **u** is the velocity vector

P is the static pressure
μ is the dynamic viscosity
ν is the kinematic viscosity
ρ is the density of the fluid

C. *simpleFoam solver*

The OpenFOAM software provides various solvers. The simpleFoam solver is used. It is a steady-state solver for incompressible flow with turbulence modelling, but the turbulence feature is turned off. Hence, simpleFoam solver can be used for steady-state incompressible laminar flow. The SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm is solving the momentum equation. Following is the SIMPLE algorithm which is the basis of simpleFoam solver:

1) Set the boundary conditions.
2) Solve the discretized momentum equation to compute the intermediate velocity field.
3) Compute the mass fluxes at the cells faces.
4) Solve the pressure equation and apply under-relaxation.
5) Correct the mass fluxes at the cell faces.
6) Correct the velocities on the basis of the new pressure field.
7) Update the boundary conditions.
8) Repeat till convergence.

D. *Force coefficients*

The aim of this study was to calculate the force coefficient, namely the Drag coefficient (C_D). The other coefficient also is calculated but isn't of that importance, those are, Lift Coefficient (C_L) and moment coefficient (C_M). A force coefficient function was called in the controlDict file. It was defined as follows:

```
forces

{
```

```
        functionObjectLibs
    ("libforces.dll );

        patches
        (hull_with_domain_solid
        );

        outputInterval 1;

        liftDir (0.0 1.0 0.0);

        log true;

        pitchAxis (0.0 1.0 0.0);

        dragDir (1.0 0.0 0.0);

        type forceCoeffs;

        Aref 7.4430003E8;

        writeControl timestep;

        p    p;

        lRef 22000.0;

        U    U;

        cofR (0.0 0.0 0.0);

        rho rhoInf;

        rhoInf 1.0;

        magUInf 10.2889;

}
```

III. RESULTS AND DISCUSSION

The C_L, C_D and C_M values are obtained from the analysis. The values obtained were

C_L	1.158236e-006
C_D	0.001903728
C_M	-1.289107e-007

Table 2: Force Coefficients

These values were obtained from the plot of the force co-efficients against the simulated time as follows:

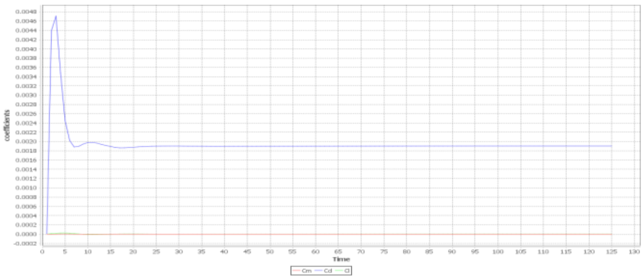


Figure 5: Force coefficients versus simulation time

Alternatively, these values can also be found from the excel sheet exported from the analysis by averaging the values.

The C_D is usually calculated with respect to the L/D ratio and the angle of attack on the rudder.

So, in this case

L/D	4
Angle of attack	0°

Table 3: experimental data consideration

The C_D value obtained is lower than the actual experimental value, which is around 0.003213, as the laminar steady one directional flow was considered and the resistance caused by calm air R_{AA} , equation 2, wasn't considered.

We can visualize the pressure variation over the hull of the ship from the pressure contour. As can be seen, the pressure is maximum on the front of the geometry.

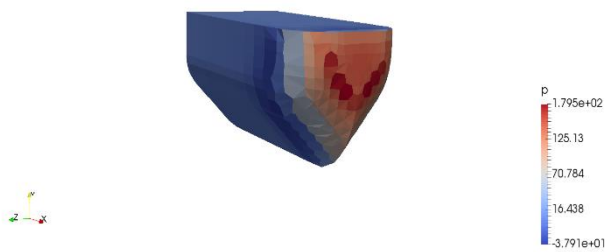


Figure 6: Variation of pressure over the hull

We can also visualize the streamlines due to the flow using the Stream Tracker option in OpenFOAM.

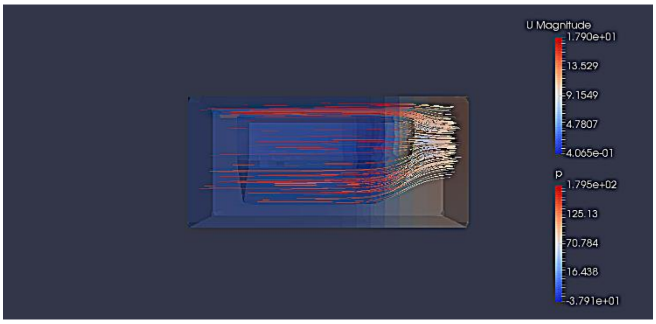


Figure:7.1

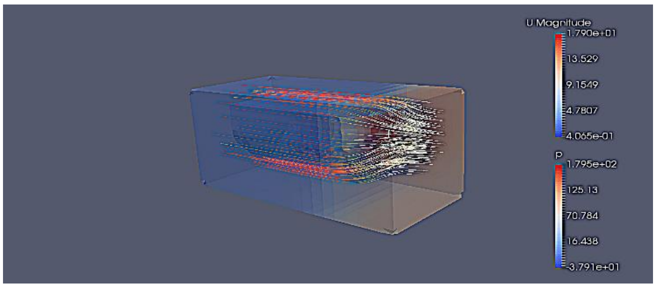


Figure:7.2

Figure 7: Stream Tracker from OpenFOAM 4

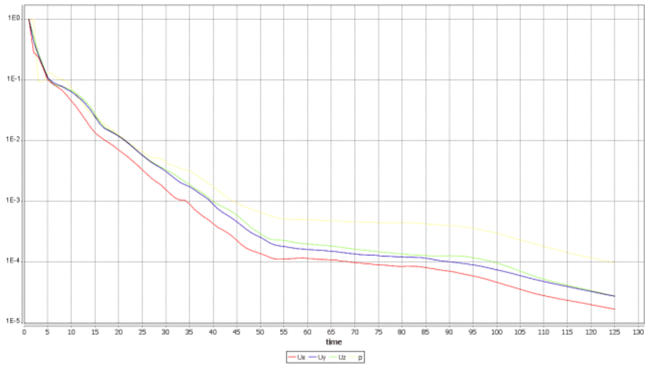


Figure 8: Residual versus time

Figure 8 represents a plot of the initial residuals against the timesteps. As we can see, the residuals are decreasing with the increase in timesteps on an average.

The streamlines for the velocity magnitude are as follows:

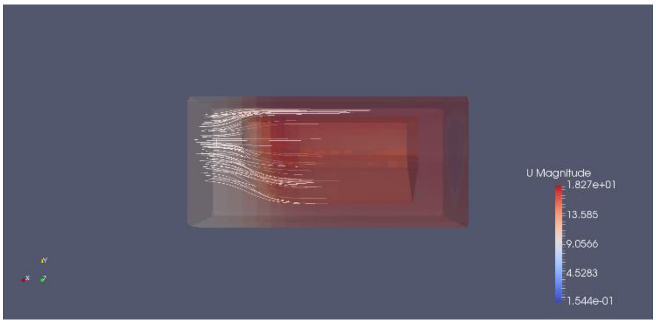


Figure 9: U magnitude in the front of the hull

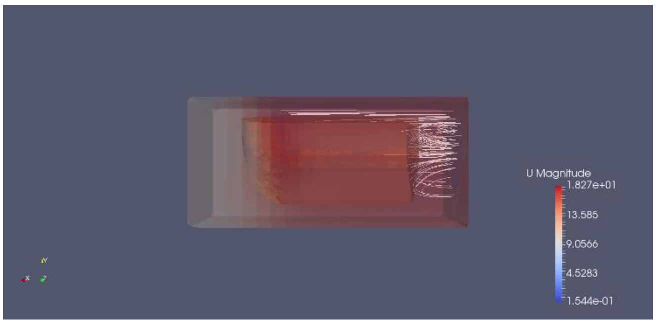


Figure 10: U magnitude at the rear of the hull

<u>A further analysis was taken into consideration</u>. In this analysis, a 2-dimensional hull was used, for simpler calculations. This study tries to improve the overall efficiency and thereby reduce the drag. This in turn helps in the fuel consumption and helps to reduce pollution.

<u>Sub-Case 1:</u>

A. Geometry

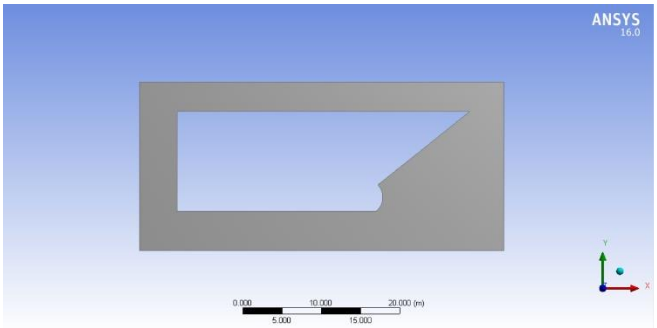
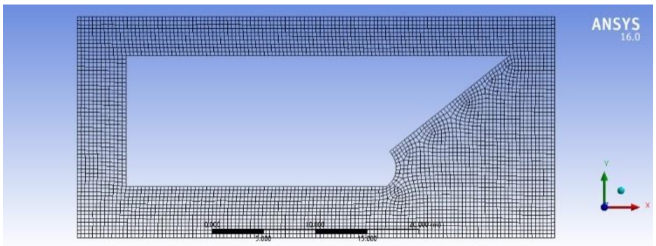


Figure 11: 2D hull design without any modification

This design was made using the DesignModeler package in ANSYS 16.0.



B. Meshing

Figure 12: Meshing of the geometry

The fine meshing is done through Meshing package from ANSYS 16.0

C. simpleFoam solver

This problem was solved using simpleFoam solver provided by OpenFoam.

D. Analysis

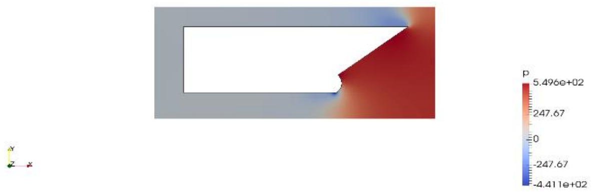


Figure 13: pressure contour for 2D hull without modification

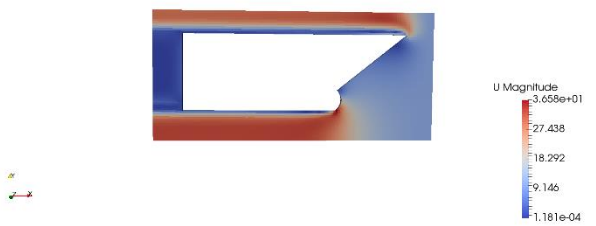


Figure 14: velocity contour for 2D hull without modification

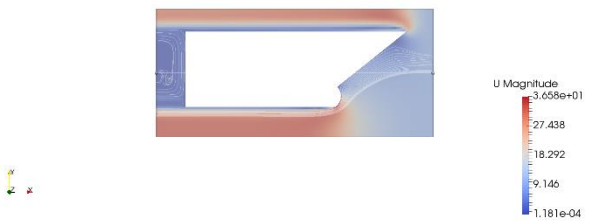


Figure 15: Stream Tracker for 2D hull without modification

<u>Sub-Case 2:</u>

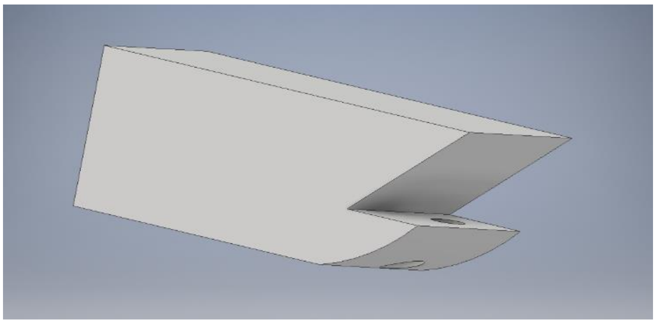


Figure 16: model created in Autodesk Inventor 2018 student version

The figure 16 is given just for imagination purpose so that it becomes easy to imagine what is being discussed here.

A. Geometry

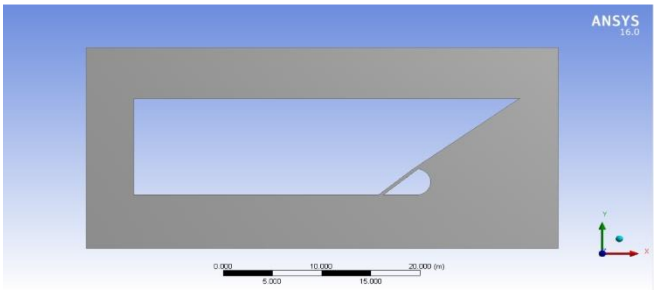


Figure 17: 2D hull with modification

Consider this to be a cross-section about the symmetric plane. The modification is introducing a central hole through the hull so as to reduce the pressure accumulation near the protrusion.

B. Meshing

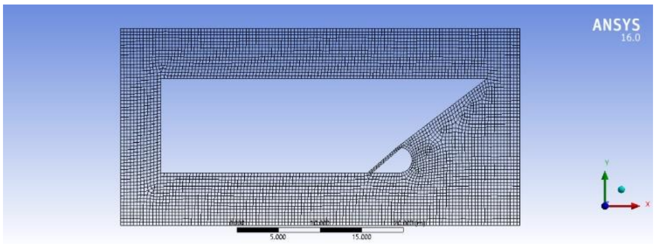


Figure 18: meshing of the modified geometry

The fine meshing is done through Meshing package from ANSYS 16.0

C. simpleFoam solver

This problem was solved using simpleFoam solver provided by OpenFoam.

D. Analysis

Figure 19: pressure contour for 2D hull with modification

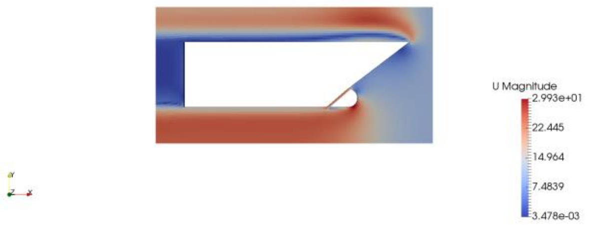


Figure 20: velocity contour for 2D hull with modification

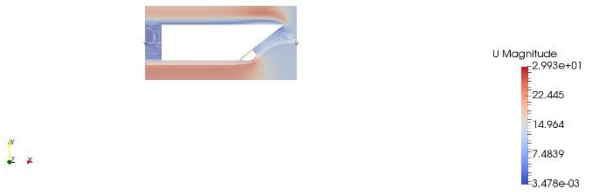


Figure 21: Stream Tracker for 2D hull without modification

E. Discussion

As it can be seen from figure 13 and 19, after introducing a simple modification, the pressure drop is massive. This heavy drop helps in better fuel efficiency. Similarly, many simple modifications can be made to the hull of a ship.

REFERANCES

1. **SHIP RESISTANCE AND PROPULSION,** *DEPARTMENT OF MARINE TECHNOLOGY, UNIVERSITY TEKNOLOGI MALAYSIA.*
2. **Laminar external flow, MathWorks**
3. **OpenFOAM guide/The SIMPLE algorithm in OpenFOAM**
4. **www.openfoam.com/documentation**
5. **viscous drag calculations for ship hull geometry,** *Cheng-Wen Lin, Scott Percival, and Eugene H. Gotimer Design Evaluation Branch, Hydrodynamics Directorate, David Taylor Model Basin, Carderock Division Naval Surface Warfare Centre, Bethesda, MD, USA*
6. **dx.doi.org/**10.3233/ISP-2011-0068

NAME: Siddhant Saraswat

Currently pursuing B. Tech 2nd Year Mechanical Engineering

College: NITK, Surathkal

Email: siddhantsaraswat.16me271@nitk.edu.in

Mobile:00919008818590