IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

NUMERICAL SIMULATION OF SUBMARINE HYDRODYNAMICS AND FLOW FIELD ANALYSIS OF NOSE AND HULL USING ANSYS FLUENT

Subhashis Choudhury[1], Sahul Muhique[2], Dr. Kalyan Kalita[3]

[1,2,3]Department of Mechanical Engineering, Assam Engineering College, Jalukbari, Kamrup Metro, Assam-781014, India.

Abstract

This report describes the submarine hydrodynamics through simulation in ANSYS Fluent. The flow over a Standard Submarine Model with a length-to-maximum diameter (l/d) ratio of 8.75 is studied with the help of pressure contours and velocity contours. The Analysis includes multiple mesh studies required to calculate the Drag Force and their coefficients in a Pressure based Solver. Dependency on the variation of one data on the other is also studied with the help of tables and graphs. Error between the data acquired is also investigated in order to ensure the validity of the project. Agreement with the data calculated is generally agreed as accepted within the error margin of 10% which is quite within the range.

Keywords: Submarine Hydrodynamics, Drag Force, Pressure based Solver.

Introduction:

Flow around submarine hull considering rudder interaction has always been a subject of great concern for naval architects in order to ensure that a submarine can operate efficiently and economically at a desired speed. Although extensive researches concerning the flow around bare submarine hull have been carried out in the past decades, the hull-rudder interaction is very much important for accurate prediction of flow especially at stern region of submarine. In this research, the flow around submarine hull is numerically simulated considering the hull-rudder interaction.

Methodology

Modelling

The standard DRDC-STR standard submarine geometry (hull-sail-tail configuration) reproduced from Mackay M. (2)

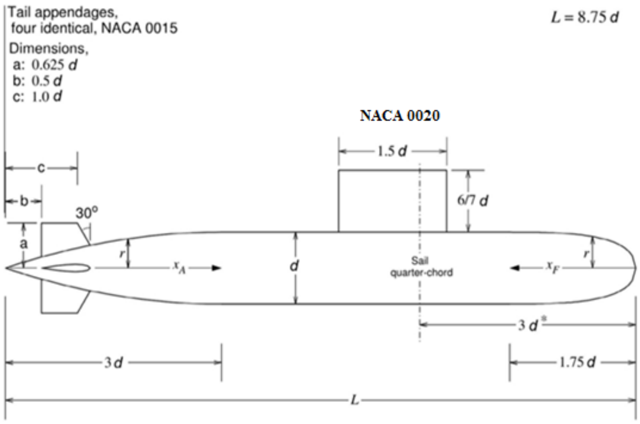


Fig 1. DRDC standard submarine geometry

The hull-sail-tail (HST) configuration of DRDC's generic standard submarine geometry, shown in Fig. 1, is used for all CFD simulations in this work. This submarine has an axisymmetric hull with a length-to-maximum diameter (l/d) ratio of 8.75 and consists of three profile sections: a Riegel's type D2 nose, a constant diameter midsection, and a parabolic tail. A full-scale length of l=30 m is considered for this work. The sail has a NACA0021 profile with a rectangular profile and a flat tip. The tail is a cruciform ('+') configuration of four identical NACA0015 tail planes with flat tips.(1)

Aerofoils

NACA 4 digit aerofoil specification: This NACA aerofoil series is controlled by 4 digits which designate the camber, position of the maximum camber and thickness. If an aerofoil number is NACA MPXX then:

M is the maximum camber divided by 100.
P is the position of the maximum camber divided by 10.
XX is the thickness divided by 100.

All the appendages have NACA four digit aerofoil profiles given by:

$$\frac{y_t}{c} = \pm\left[1.4845\sqrt{\frac{x}{c}} - 0.6300\frac{x}{c} - 1.7580\left(\frac{x}{c}\right)^2 + 1.4215\left(\frac{x}{c}\right)^3 - 0.5075\left(\frac{x}{c}\right)^4\right]$$

Where, c is the chord length and t/c is the maximum thickness to chord ratio. The leading edge is at x=0and the trailing edge, which has non-zero thickness, at $x = c$.

Meshing Results and Discussions

The mesh generation procedure for this project was trial and error complemented by what could be learned with the help of various iterations. The mesh must be fine enough to produce grid independent solutions but coarse enough to have a reasonable computational time.

Completing a mesh sensitivity analysis is how this balance is achieved. In essence, a mesh sensitivity analysis is an iterative process of refining the mesh and checking the results until the results no longer change by an appreciable amount. A mesh sensitivity analysis was done on the various sections of the submarine hull. First, the nose section of the submarine hull was considered. Three different mesh was introduced on with along with three different velocities i.e. 50 m/s, 30 m/s and 20 m/s.

Following the mesh convergence, we obtained three values of C_d (Coefficient of Drag) and Pressure Drag Force. These values are computed with their respective theoretical values and their percentage error computed.

The various types of meshing used for the computation of the submarine hull are stated as follows:

TYPE A: System default mesh with +100 relevance and fine relevance centre.

TYPE B: Vertex sizing mesh with sphere of influence of 1 m and element size 0.01m with patch conforming tetrahedron method.

TYPE C: Vertex sizing mesh with sphere of influence of 1 m and element size 0.01m and edge sizing with size 0.001 m with patch conforming tetrahedron method.

Table 1: Types of Meshing used.

Type	Sphere of Influence		Relevance	Vertex Sizing	Element Shape
	Radius	Element Size		Element Size	
A	-	-	+100	-	Tetrahedron
B	1m	0.01m	0	-	Tetrahedron
C	1m	0.01m	+100	0.001m	Tetrahedron

Boundary Condition

A boundary condition was set at the inlet. The inlet velocity profile was set as 10m/s, 30m/s

And 50 m/s for the three cases respectively.

Density of water: 998.2 kg/m^3

Temperature of water (Operating Temperature): 298.15 K

Dynamic viscosity of water: 0.001003 N-s/m^2

Density of material of construction of submarine: 2719 kg/m^3

The equation used to calculate the theoretical value of drag force is given by

$$F_D = \frac{1}{2} \times C_D \times \rho \times A \times v^2$$

The simulated values of coefficient of drag are calculated by FLUENT using the formula stated below:

$$c_d = \frac{1}{\rho v^2 A}\int_s dA(p - p_0)(\hat{n} \cdot \hat{\imath}) + \frac{1}{\rho v^2 A}\int_s dA(\hat{t} \cdot i)T_w$$

Analysis and Discussions

The flow characteristics over the submarine hull are generally shown in terms of pressure contours and velocity profiles. Firstly, we took into consideration the C_d vs Iteration curve simulated by the software. The number of iterations were done till the solution i.e. C_d value converged and their subsequent values of drag force were calculated by the system. Secondly, the most frequently occurring value of C_d i.e. the mode is found out by the software from which we analytically calculate the value of drag force. Finally, the error between the system generated value and analytically calculated value of drag force is calculated.

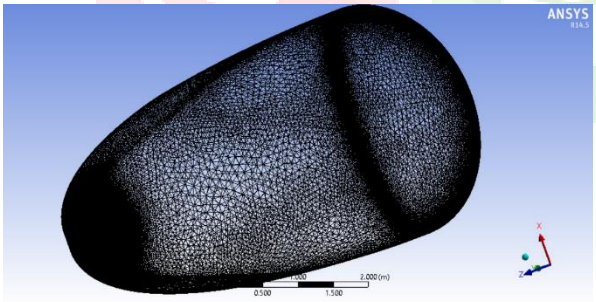
The pressure contour and velocity profiles are also shown such that we can understand the hydrodynamics of each section of the submarine hull. The data is calculated for three different velocities i.e. 10m/s, 30 m/s and 50m/s corresponding to which three different mesh structures are used.

The submarine is divided into three sections, namely the nose, mid-hull and the tail.The simulation is firstly preceded taking into account only the nose section, followed by the combination of nose and mid-hull section which also includes the sail and finally the whole submarine hull.

Results

On selective meshing and iterating the parts of the submarine hull, the pressure drag force and co-efficient of drag at the nose section at 50 m/s was found to be 315420 N and 0.0417 respectively. When compared with the theoretical values (1) which were 8858.69 N and 0.0262 respectively an error of 2.852% was concurred which is well within the acceptable range.

Fig 2: Type C Meshing of the nose section.



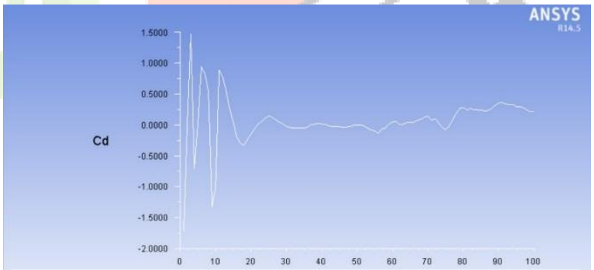
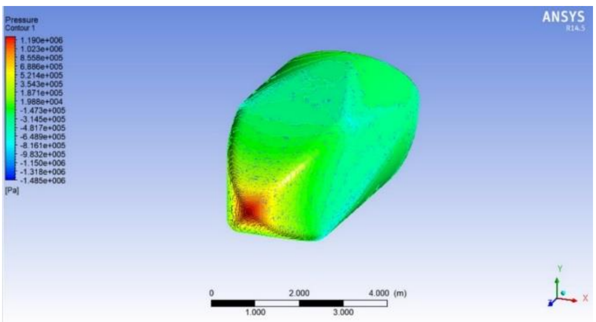


Fig 3: Cd vs Iteration Curve for Type C meshing at 50 m/s for nose section.



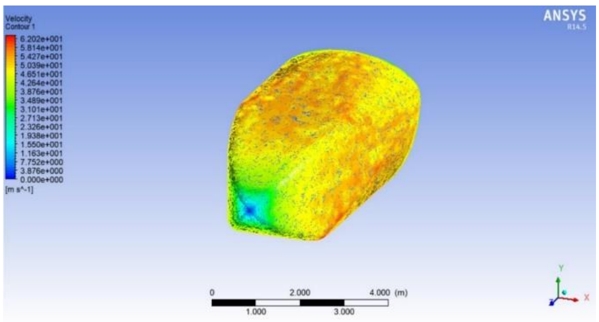


Fig 4: Velocity Contour over nose at velocity of 50 m/s with Type C

Fig 5: Pressure Contour over nose at velocity of 50 m/s with Type C meshing.

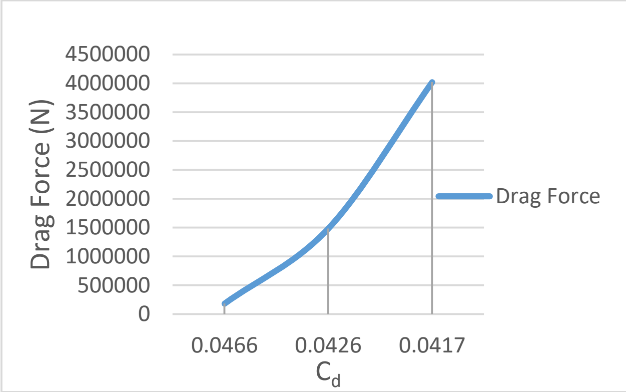


Fig 6: Drag Force vs Cd for Nose and Mid-Hull Section

Pressure Drag 1	C_d 1	Pressure Drag 2	C_d 2	Pressure Drag 3	C_d 3
5598496.6	0.0231	4378947.7	0.0247	5229278.3	0.0224
4419244	0.0433	3125826	0.0498	4212413.2	0.0434
3089839.7	0.0414	3353253	0.0526	3432731.6	0.0456
3392839.7	0.053	3190205.7	0.0472	3223685.1	0.0347
3827102.6	0.0222	2849007.4	0.0343	3099600.6	0.0417
4540970.7	0.022	2321722.2	0.0578	4352786.1	0.0445
4855878.5	0.0343	2280200.8	0.0352	4657202.8	0.0387
4230553.9	0.0595	2067687.2	0.0402	3959563.4	0.0381
3295876.3	0.0236	1874212.7	0.0111	3040675.6	0.0231
2415382.2	0.0213	1664245.8	0.0513	2326961	0.021

Table 2: Combined Datasheet of Nose Section at Velocity 50 m/s.

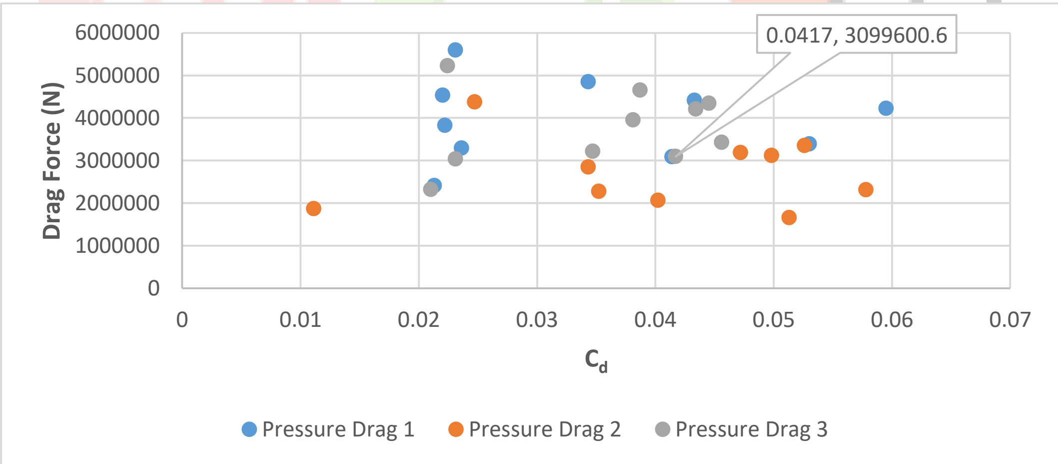


Table 3: Cd vs Drag Force Plot at 50 m/s for nose section.

Conclusion

This experiment was conducted to establish a flow analysis around a Standard Submarine hull while in motion with the help of ANSYS Fluent.

A DRDC-STR Model was simulated to find the coefficient of drag over the submarine hull and thus calculate the drag force. The calculated force was compared with the reference value and a 2.8% error was conquered which is quite under the acceptable range. This proved that the design geometry and meshing can be considered appropriate for further calculations.

The simulation is then carried out with three different types of meshing which allowed us to bring about a linear relation between Velocity vs Cd, Velocity vs Drag force and Velocity vs Pressure and an exponential relationship between Drag force vs Cd.

The velocity and pressure contours over the submarine hull were also studied. It showed that the velocity increases from the nose to the top-hull and then slightly decreases as it approaches the mid-hull. The pressure decreases exponentially from the nose tip. Similar exponential decrease in the results can be seen on the sail in case of both velocity and pressure.

Reference

1. Christopher Baker (2004), "Estimating Drag Forces on Submarine Hulls", Defence R&D Canada-Atlantic CR 2004-125.

2. M. Mackay (2003), "The Standard Submarine Model: A Survey of Static Hydrodynamic Experiments and Semi-Empirical Predictions", Defence R&D Canada-Atlantic TR 2003-079.

3. Abdi Kukner (2016, "Investigation of Flow Distribution around a Submarine", Journal of Naval Science and Engineering 2006, Vol 12, No 2, p. 1-26.