CFD Simulation of Submarine Escape Suit

Sushma. G[1] H. N. Das[2] V. F. Saji[3] Yusuf Ali[4]
[1]PG Student [2]Scientist-G [3]Scientist-E [4]Assistant Professor
[1,4]Department of Mechanical Engineering
[1,4]MVGRCE(A), Vizianagaram, AP India [2,3]DRDO (NSTL), Vishakapatnam, AP, India

Abstract— In case of an unfortunate event of accident of a submarine, it is required to provide some pressurized suit, which will be used by submariners to escape from the sunken submarine. Before releasing from a depth of 180m, the suit is inflated and its internal pressure is adjusted to match the external hydrostatic pressure at that depth .After release, submariner wearing the suit starts ascending up due to positive buoyancy. The speed of ascend is required to be maintained at around 3m/s to suit human physiology. After reaching a depth of 100m (from surface), the speed is required to be limited to 1m/s. Suit is for individual escape .As these devices are critical to save the lives of users, a lot of prudence has to be exercised in designing the equipment. The objective of this project is to estimate added mass and drag coefficient of suit which may be used to simulate the trajectory of the escape suit. The two parameters chosen to be determined here have considerable importance on the motion of submariner. Whenever acceleration is imposed on the fluid flow either by acceleration of submariner or accelerating externally, additional fluid force will act on the body surface in contact with fluid. These inertial forces are characterized as added mass. The work includes meshing of the flow domain, flow solution and post processing of results. The software used are ANSYS ICEM CFD, ANSYS FLUENT and MATLAB.

Key words: Submarine Escape Suit, Escape Suit

I. Introduction

The general interactions between a body and surrounding fluid when both body and the fluid are not moving is only pressure force and net effect is the buoyancy force. When either body or fluid moves at steady rate there is an additional force due to viscous shear. When either body or fluid moves at unsteady rate, the fluid particles around the body generate another force called fluid inertia force

In such case, forces couldn't be described by Newton's 2[nd] law, $F = Ma$.Instead they are given as follows, for an object of mass M ,undergoing acceleration, a, another mass,M_{added}, needed to be "added" in order to better account for the forces,i.e: $F = M + M_{added}$

A. Modeling Methods

- Analytical-By use of formulae exact solution can't be obtained always
- Experimental-Expensive, time consuming
- Computational-capable to handle practical engineering problems
- Therefore in this work Computational Modeling methods are used for Simulation of submarine escape suit.

B. Overall Process

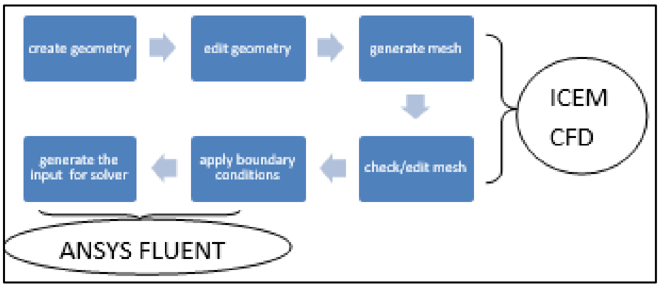

Fig. 1: Overall process

II. Methodology

The basic steps are

A. Problem specification and geometry preparation

1) It involves the specification of the problem, including the geometry, flow conditions and the requirements of the simulation.

B. Selection of governing equations and boundary conditions

1) Boundary conditions

C. Selection of numerical aspects

1) Appropriate numerical method has to be selected depending upon the parameters effecting the model (eg: turbulence or laminar)

D. Interpretation of results

III. Work Flow

- Generation of mesh model for submarine escape suit by using ICEM CFD 14.5
- Solving the problem using ANSYS FLUENT 6.2.16
- Obtain added mass and drag coefficient (c_d)
- Comparing the results obtained from the numerical results with the experimental results done in NSTL.
- Visualizing the results with the help of MATLAB R2007b

IV. Submarine Escape Suit

A. Geometry of Suit

The geometry of the submariner consists of a hydro suit and a breathing apparatus. The hydro suit is a full coverage rubberized suit to protect the submariner. It is made up of air and watertight costume of neoprene coated nylon fabric. The breathing apparatus caters the breathing needs of the submariner until the wearer comes to the surface. This is a self-contained close-circuit breathing apparatus that allows the submariner to pass through various depths till it reaches surface.

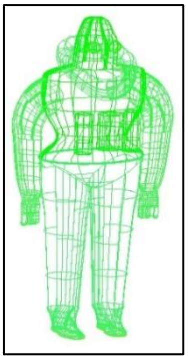

Fig. 1: Geometry of suit

Dimensions	Suit size
Length	200cm
Breadth	100cm
Width	40cm

Table 1: Dimensions of suit

B. Domain of Suit

First, the surface model was generated using the modeling package CATIA as shown in Fig. and then it was imported to ICEM-CFD software for meshing purpose.

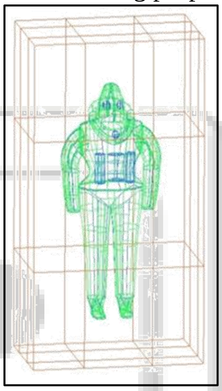

Fig. 2: Sub domain of suit

Dimensions	Sub domain size
Length	280cm
Breadth	160cm
Width	100cm

Table 2: Dimensions of sub domain

C. Mesh Generation

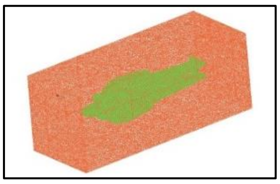

Fig. 3: Mesh generation of sub domain

The accuracy of the results depends upon the meshing strategy used for the work, domain size, number of nodes, type of mesh impacts the result. Different structured and unstructured meshing options are there, here we use the unstructured tetragonal meshing. By analysis the major problem with mesh movement is mesh failure around suit region and remeshing failure in multi dimensions. So this work used a new concept of multiple domain. This concept is achieved by two rectangular domains, inner domain and outer domain. The inner domain had the movement along with the rigid body (suit) and the outer domain are fixed and have only mesh deformations according with the movement of inner domain. Since the inner domain was restricted with mesh deformation the cells near the suit would not be failed under any circumstances. The two domain concept finalized the meshing adaptations through different tests. Here we use tetragonal mesh for sub domain and Hexa mesh for outer domain.

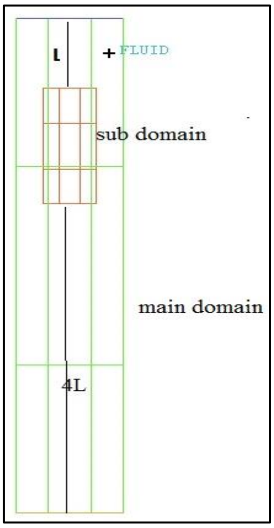

Fig. 3: Domain

Sub domain is kept at a distance of ‘L’ from the inlet and ‘4L’ from the outlet. Because flow characteristics downstream are dominant and also flow gets fully developed at exit.

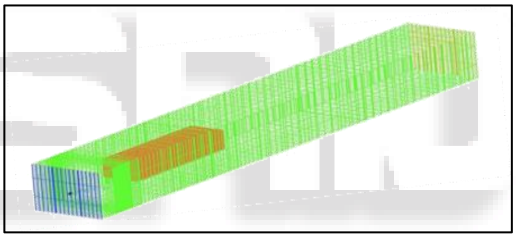

Fig. 4: Hexa mesh generation of Domain

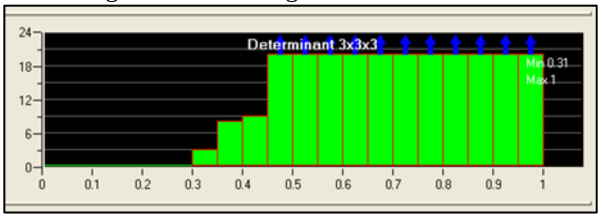

Fig. 5: Mesh Quality

D. Boundary Conditions and Solver Settings

Boundary conditions describe the various boundary values used for this works. The table shows the various boundary values used for this work.

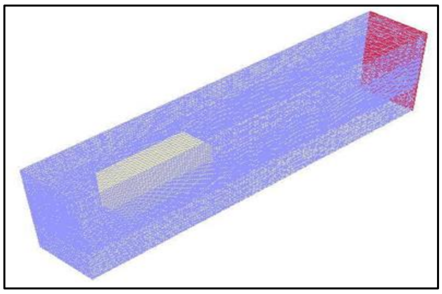

Fig. 6: Mesh after importing to ANSYS FLUENT

ZONE	TYPE
cylinders	wall

far	velocity inlet
inlet	velocity inlet
body	interior
fluid	interior
outlet	outlet-vent
sub domain	interface
suit	wall

Table 3: Boundary conditions

Solver	FLUENT 6.2.1
Pressure Link	SIMPLE
Descritization Method	QUICK
Relaxation Parameter	0.1
Wall function	Standard
Turbulence model	K-ε (2 eqn.)

Table 2: Control parameters

Solution Methods	SIMPLE scheme (Pressure Velocity Coupling) QUICK scheme(Momentum)
Solution Controls	Under Relaxation Factors Pressure-0.1, Momentum-0.1, Turbulent kinetic energy-0.1
Calculation Activities	Time Step size – 0.1(s) Number of time steps-1000

Table 3: Numerical Settings

V. RESULTS

CFD analysis for computations of added mass encountered by a submariner suit is carried out. A mesh with around 2.5 million cells is used for analysis in the FLUENT solver. Standard *k-ε* turbulence model with standard wall function is used to model turbulence. 10000 iterations were required to obtain a converged solution. These iterations are imported to MATLAB for obtaining Drag coefficient C_d. The curve is obtained by using curve fitting tool. The resulting curve is plotted in fig.

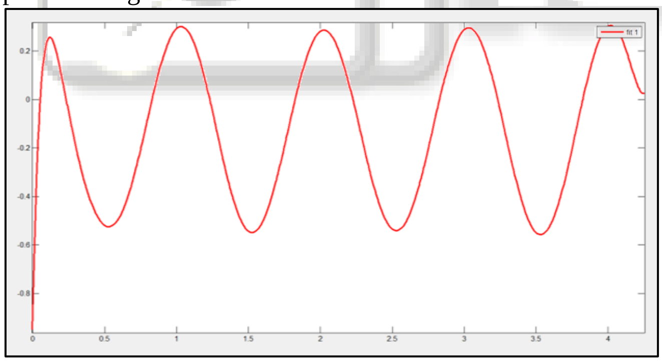

Fig. 7: Curve with frequency 2pi

By inputting the C_d iterations obtained from ANSYS Fluent to Matlab Curve Fitting, we obtain a sinusoidal curve as shown in fig8. The resultant curve is a combination of both added mass and drag force. The movement of suit inside the main domain is considered to be Simple harmonic motion. .So the velocity of a body undergoing SHM is given by $v = A\sin\omega t$

Acceleration (a): $a = A\omega\cos wt$

Consider the body undergoing maximum acceleration is given by $a = A\omega$

Where A is Amplitude (obtained from UDF),ω is frequency

$$A = 0.05$$
$$\omega = 2\pi * 1 \text{ (here n=1)}$$
$$a = 0.05 * 6.28$$
$$\therefore a = 0.314 \qquad M = \rho * volume$$

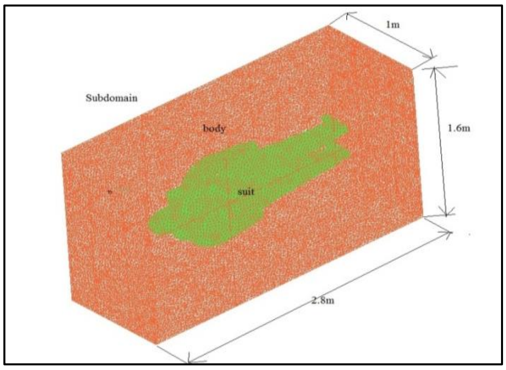

Fig. 8: Sub Domain and Suit

$$(\text{Volume})_{\text{sub domain}} - (\text{Volume})_{\text{body}} = (\text{Volume})_{\text{suit}}$$
$$(\text{Volume})_{\text{sub domain}} = 2.8 * 1.6 * 1 = 4.48$$
$$(\text{Volume})_{\text{body}} = 4.25 \text{ (this is obtained from volume reports)}$$
$$(\text{Volume})_{\text{suit}} = 4.48 - 4.25 = 0.23$$
$$(\text{Mass})_{\text{suit}} = \rho * (\text{Volume})_{\text{suit}} = 230\text{kgs}$$
$$\text{Added mass} \propto \text{acceleration} \propto \cos$$
$$\text{Added mass} = \text{fn}(\cos \theta)$$

Similarly, drag = fn(sin θ)

1) General model Fourier 1:
   f(x) = a0 + a1*Cos(x*w) + b1*Sin(x*w)
2) Coefficients (with 95% confidence bounds):
   - a0 = -0.1243 (-0.1254, -0.1232)
   - a1 = 0.4133 (0.4117 ,0.415)
   - b1 = 0.07323 (0.07, 0.076)
   - w = 6.284 (6.281, 6.287)
3) Goodness of fit
   - SSE: 0.000511;
   - R-square: 0.9948
   - Adjusted R-square: 0.9946;
   - RMSE: 0.001871

The resultant curve obtained as shown in fig 8 is a combination of both added mass and drag force. To obtain the added mass value consider the cos coefficients from the Fourier curve equation.

a_1 is the cos coefficient and it represents C_d

$$\therefore C_d = 0.4124$$
$$F_{\text{inertia}} = \frac{1}{2}\rho A V^2 C_d(\cos.) = \frac{1}{2} * 1000 * 0.3 * 1^2 * 0.4124$$
$$= 61.86\text{N}$$
$$(M + M_{\text{added}})a = F_{\text{inertia}}$$
$$M_{\text{added}} = \left| \frac{F_{\text{inertia}}}{a} - M \right| = 80 \text{ kgs } (\approx 35\%)$$

A. *Submarine Escape Suit Simulation*

Sea trail is straight forward method to obtain trajectory of submariner. However this is not practical considering the risk of life and expenses it need. Simulation offers several significant advantages over sea trail tests for assessing the trajectory of submariner.

In the input block, the code will read the input data such as mass, added mass and constant $(\frac{1}{2}\rho A C_d)$. These input data will then be used in the process block for calculation. Complete model is divided into blocks of force, added mass and mass from which acceleration is obtained. It is integrated once to obtain velocity, double integrated to obtain the translation of motion.

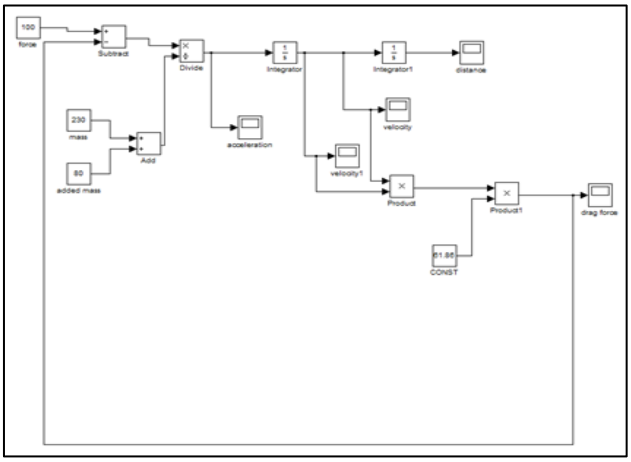

Fig. 9: Simulink model of suit

Fig. 11: Acceleration plot

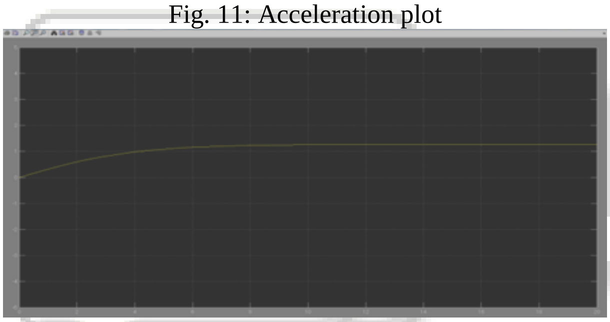

Fig. 12: Velocity Plot

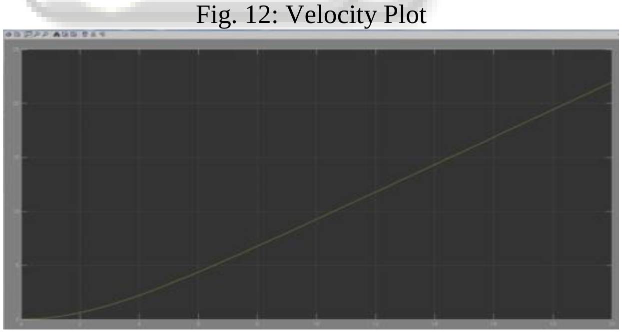

Fig. 13: Displacement plot

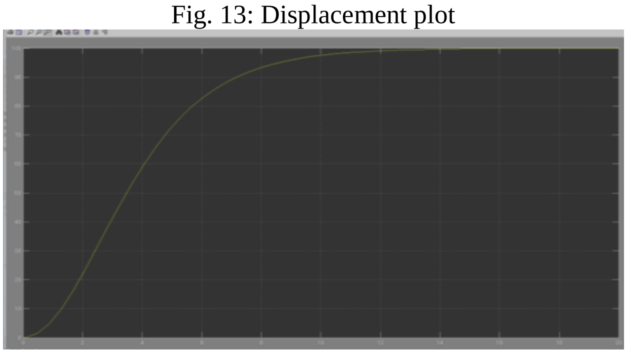

Fig. 14: Drag force plot

VI. Conclusion

CFD analysis has been carried out on a submarine escape suit to estimate its Drag coefficient and added mass coefficient. ANSYS FLUENT software has been used to predict C_d drag coefficient and MATLAB software is used to simulate the trajectory of submariner with fully equipped suit.

Comparison between CFD and experimental results was made and found to be in good agreement. The difference is around 10%.

The study demonstrates capability of CFD for analyzing flow around complex geometries like this suit. This also gives successful prediction of trajectory for one dimensional motion in submerged condition.

A. *Future Work*

Analysis may be carried for the suit with an underwater parachute which will work as speed breaker to suite the physiological requirement of human being.

References

[1] Du Buat, C., Principles D'hydraulique, Paris, 1786.
[2] Brennen, C.E., A Review of Added Mass and Fluid Inertial Forces, Report CR 82.010, Naval Civil Engineering Laboratory, Port Hueneme, California, 1982
[3] Yih, C.S., Fluid Mechanics: A Concise Introduction to the Theory, McGraw-Hill New York, NY, 1969
[4] Lamb, H., Hydrodynamics, 6th Ed., Dover Publications, New York, NY, 1945.
[5] Birkhoff, G., Chapter 6: Added Mass, in Hydrodynamics: A Study in Logic, Fact and Similitude, pp. 148-178, Princeton University Press, Princeton, New Jersey, 1960.
[6] Kennard, E., Irrotational Flow of Frictionless Fluids: Mostly of Invariable Density, Report 2299, David Taylor Model Basin, Washington, D.C., 1967.
[7] Patton, K.T., Tables of Hydrodynamic Mass Factors for Translational Motion, in American Society of Mechanical Engineers - Meeting WA/UNT-2, American Society of Mechanical Engineers (ASME), New York, NY, United States, 1965.
[8] Keulegan, G.H. and L.H. Carpenter, Forces on Cylinders and Plates in Oscillating Fluid, United States Bureau of Standards - Journal of Research, Vol. 60, pp. 423, 1958.
[9] Sarpkaya, T. and M. Isaacson, Mechanics of Wave Forces on Offshore Structures, Van Nostrand Reinhold Company, NY, 1981.
[10] Sarpkaya, T., Lift, Drag, and Added-Mass Coefficients for Circular Cylinder Immersed in Time-Dependent Flow, in Journal of Applied Mechanics Meeting WA-61, pp. 13-15, 1962.
[11] Villaggio, P., The Added Mass of a Deformable Cylinder Moving in a Liquid, Continuum Mechanics and Thermodynamics, Vol. 8, No. 2, pp. 115-120, 1996.
[12] Bird, A.R., Measured Fluid Forces on an Accelerated/Decelerated Circular Cylinder (History Force), PhD Dissertation, Department of Civil Engineering, North western University, Evanston, Illinois, 1984.5