Studying Fluid Mechanics for Better Understanding Simulation Technologies. A Special Approach for Maritime Universities

DUMITRU DINU
Mechanical Engineer Department
Constanta Maritime University
104 Mircea cel Batran str., Constanta, 900663
ROMANIA
E-mail: dinud@imc.ro, dumitru.dinu@cmu-edu.eu

Abstract: - In the computer age, the theoretical study loses its importance. There are simulators for everything and many of their users don't understand the simulated phenomena. Are they good users? Can they realize a simulation without understanding the mathematical model? The students of maritime universities, which have to comply with IMO (*International Maritime Organization)* requirements and academic exigencies, works with many simulators: FLUENT, Bridge Deck, Engine Room, Emergency. All of them are based on the main fluid mechanics equations. In the lecture I explain why the fluid mechanics knowledge is necessary for understanding phenomena and significance of terms, for choosing the material of the fluid (density, viscosity) and of the solid (conduits), for putting boundary conditions and, finally, for analyzing the results. A very important phase is post processing. In some cases, comparison of the simulation results with the experimental ones is also very important. Learning Fluid mechanics adapted to new computational requirements as well as studying simulation technologies together with their physical models is a necessity for maritime students and not only.

Key-Words: - Simulation Technologies, Maritime Universities, Fluid Mechanics, FLUENT, CFD.

1 Introduction

In our e-society the scientists try to solve the problems using computers. For maritime universities students, the simulation is one the main method to study and to train. The marine simulators are based on fluid mechanics equations. International Maritime Organization – IMO – makes the rules for preventing pollution by ships and for maritime safety, standards of training included. This organization imposes the use of simulators. The students of maritime universities, which have to comply with IMO requirements and academic exigencies, works with many simulators: FLUENT, Bridge Deck, Engine Room, Emergency, Liquid Cargo Handling. It is very important to explain why the theoretical knowledge of fluid mechanics is necessary for the correct use of the simulators.

It is very important to emphasise that some equation, like Navier Stokes' equation has not an analytical solution. For the moment this equation is solved only in some particular cases or numerical, in general case.

We consider the numerical presentation of main problems of Fluid Mechanics a modern approach. In this courses the students can find not only the basic of statics and dynamics of fluid but the modality to solve the equations and to illustrate the movements using the new computer programs.

In order to use a numerical program it is recommended to understand in detail the analytical equations that describe the physical phenomena studied and how these equations are implemented in different numerical calculation methods.

2 Fluid mechanics and Bridge deck simulators

The majority of forces participating in the simulation represents hydrodynamic and aerodynamic interactions between ship and other objects or the environment. The other forces are of mechanical nature (Table 2).

The ship is a controllable system. The controlled forces are also of hydrodynamic or mechanical type (Table 3).

We also have so-called basic forces: gravity or displacement, buoyancy or flotation, hydrodynamic force, inertia and static stability (Table 1).

To use correctly the bridge deck simulators you must to know the hydrostatic principles and the stability of floating body. Hydrodynamic force due to the ship movement (vertical component) compensates a part of gravity force. This force is neglected at Froude number small than 0.3 [1].

Table 1 Basic Forces

Forces	Type	Formula
Inertia	Mechanic	
Hydro-dynamic	Hydro-dynamic	
Flotation	Hydro-static	γV
Stability	Hydro-static	
Displace-ment	Mechanic	$G = mg$

Table 2 Environment Forces

Forces	Type	Formula [7]
Wind	Aero-dynamic	$F_w = C_w \frac{\rho v^2}{2} S$
Waves	Hydro-dynamic	
Current	Hydro-dynamic	$F_c = C_c \frac{\rho v^2}{2} S$
Ship-Ship	Hydro-dynamic	$F_{xsh1} = C_{xsh1} f_1 \frac{\rho v_1^2}{2} S$ $F_{ysh1} = C_{ysh1} f_1 \frac{\rho v_1^2}{2} S$ $M_{Zsh1} = C_{mxsh1} f_1 \frac{\rho v_1^2}{2} S L_1$
Ship-Bottom	Hydro-dynamic	$F_{xbot.} = C_{xbot.} \frac{\rho v^2}{2} S$ $F_{zbot.} = C_{zbot.} \frac{\rho v^2}{2} S$
Ship-Wall	Hydro-dynamic	$F_{xwall} = C_{xwal} \frac{\rho v^2}{2} S$ $F_{ywall} = C_{ywal} \frac{\rho v^2}{2} S$
Channel geometry	Hydro-dynamic	$F_{xch.} = C_{xch.} \frac{\rho v^2}{2} S$ $F_{ych.} = C_{ych.} \frac{\rho v^2}{2} S$ $F_{zch.} = C_{zch.} \frac{\rho v^2}{2} S$
Ship collision	Mechanic	$F_x = k_{x1} S + k_{x2} v_x$ $F_y = k_y v_y$

Table 3 Control Forces

Forces	Type	Formula [7]
Propeller	Hydro-dynamic	$F = \rho n^2 D^4 (1 - t) K_F$
Rudder	Hydro-dynamic	$F_y = C_y \frac{\rho v^2}{2} S$ $M_z = C_y \frac{\rho v^2}{2} S \frac{L}{2}$
Thruster	Hydro-dynamic	$F_y = F_{th} f_y$ $M_z = F_{th} f_m x_{th}$
Anchor	Mechanic	$F_a = k_a m_a g$
Moors	Mechanic	$F_m = k_m \epsilon$
Towing Ship	Hydro-dynamic	

Regarding environment actions, wind and current forces depend on the square velocity of the air/water, on the density of the fluid, on the contact surface S and, by the resistance coefficient (C_w, C_c), on the shape of the ship, on the angle of incidence, on the type of the flow (Reynolds number).

Waves determine three forces and three moments, the most important being longitudinal and lateral drifting forces and yawing moment.

Interaction force with the bottom (bottom effect) manifests in the shallow water near the harbors, on the rivers, etc. The influence of the bottom also has a hydrodynamic nature. Similarly, we have the interaction force with wall, differing the direction of second component. Combining wall and bottom interaction forces, will obtain the channel force with three components.

Ship collision of two ships or between the ship and other objects like quay wall, buoy, etc. is characterized by the force applied to the contact point. The objects interaction is partially elastic.

The ship is a controlled body. Let’s review the control forces:

The propeller force depends on its diameter, number of blades, rotation, etc. A lateral force and a torque moment appear. Thruster force is similar but with lateral direction.

Rudder force is a lateral lift force due to the flow around a symmetrical wing (rudder) asymmetrically attacked by the velocity of water. This force creates a rotation moment.

Anchor and moor forces are of mechanical type.

3 CFD for study and train

CFD – Computational Fluid Dynamics became an often used term. CFM – Classic Fluid Mechanics is its mathematical model.

ANSYS FLUENT is a CFD program based on finite volume calculation. The program solves the Navier-Stokes equations for different cases. It develops and supports various engineering software for numerical simulation.

The program Fluent was developed by many engineers during the last decades and now is part of Ansys. It can solve a lot of problems that the industry is facing with.

The program is used on courses to illustrate fluid properties and some important phenomena in hydrodynamics. The students can make a comparison between the results obtained in the laboratory and the results of simulation using CFD. Finally, we emphasize the possibility to use CFD for research. The program can be used as a “virtual stand” [2].

4 Understanding fluid dynamics phenomena

I will explain why the fluid mechanics knowledge is necessary for understanding phenomena. For this we’ll take an example: the flow around a symmetrical and asymmetrical profile [4]:

For this application a hydrodynamic profile was drawn. The section of the profile is a symmetrical one NACA 0015 (Fig. 1). The chord length is 0.2m and the profile span length was imposed to 1m.

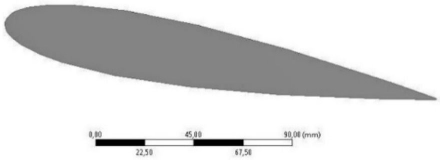

Fig. 1 Geometry of symmetrical hydrodynamic profile

This profile can be a ship, or, better, a rudder of the ship. These have symmetrical profile sections.

To simulate the flow, a paralipedal adjacent domain was created around the 3D wing (Fig.2)[4]. The mesh consists in 617329 tetrahedrons, 46382 wedges, 1000048 hexahedra and 2388 pyramid cells.

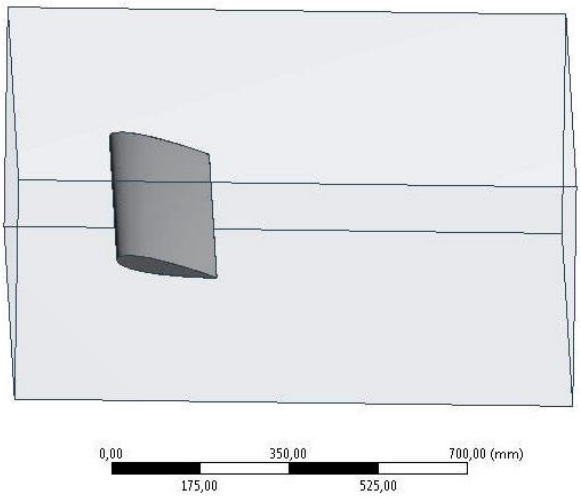

Fig. 2 The adjacent domain of the profile

4.1 Significance of terms

First of all, the students (users) must know the definition of the hydrodynamic profile. Also, the classification and the geometrical characteristics of the profile (chord, thickness, framework, arrow) and of the wing.

Defining the flow type (permanent or non-permanent) and choosing the numerical model: inviscid model (Euler flow), laminar or turbulent model imply to know the motions regime of the fluid, the significance and the values of Reynolds number. For turbulent model we have many choices: e.g. *Spalart-Allmaras, k-epsilon, k-omega, Transition k-kl-omega, Transition SST, Reynolds Stress, Detached Eddy Simulation (DES), Scale adaptive Simulation (SAS), Large Eddy Simulation.* To have the appropriate model, the students must study the theoretical basis of these models.

4.2 Choosing the material of the fluid and of the solid

FLUENT Database will allow us to choose a fluid at our choice or modify the features of the selected fluid. We can import as many fluids as possible.

The next option is named *Cell Zone Conditions.* We will have the possibility to choose what fluid to use: air, water etc. We will choose *water-liquid*.

The students must know the physical properties of fluids and their measuring units.

4.3 Generating network

Regarding mesh, it is very important to know where the great variation of the parameters (velocity, pressure) is expected to be. In this part, the network must be denser.

Let's take an asymmetrical profile, NACA 5412, with a chord length of 0.1m. A rectangular domain was created around the profile to simulate the flow (Fig. 3). The domain is 200 mm wide and 1000 mm long.

The mesh is unstructured and consists in two parts: one region of smaller cells around the profile and behind it and a second region with coarsen cells. The mesh consists in 28068 hexahedra and 20 wedges with 56528 nodes (Fig. 3)[4].

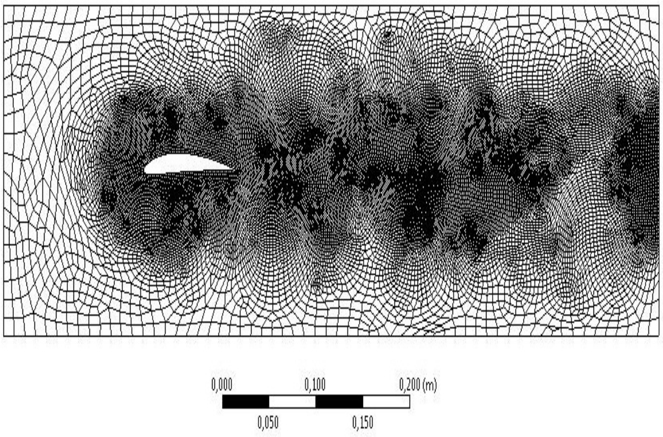

Fig. 3 Unstructured mesh of NACA 5412 hydrodynamic profile

4.4 Boundary conditions

In the *Boundary Conditions* options we run across with the two *Named Selections* mentioned at the meshing stage: *INPUT* and *OUTPUT*. By default these faces are *walls*. But we can convert it into what we need. If we select the item *INPUT*, under the button *Type* we can change the default *wall* to *velocity-inlet*. We can impose here a velocity of 2m/s. Also, if we select the item *OUTPUT* we will change the default *wall* to *pressure-outlet*. Here we will retain the default values.

4.5 Result analysis

The flow was set to be viscous. The fluid used was water with a density of $1000 {kg}/_{m^3}$. The wing is attacked with a velocity of $2 {m}/_{s}$ under an incidence angle of $\alpha = 0^0$. Figures 4 and 5 [4] show pressure and velocity distribution, respectively.

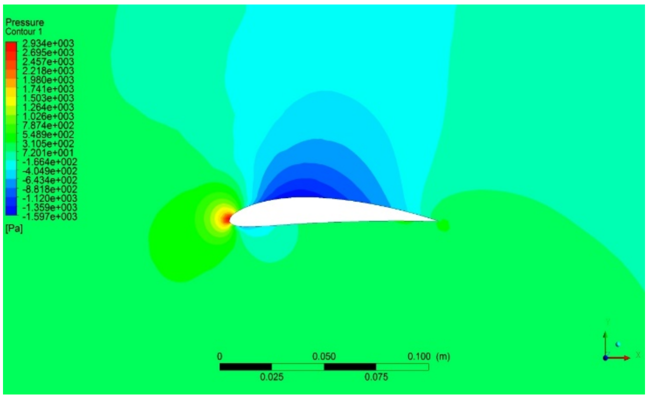

Fig. 4 Pressure distribution around profile

We'll study these results. It can be observed that Bernoulli's law is applying: the velocities asymmetry brings about the static pressures asymmetry. Due to the pressures asymmetry a lift force appears.

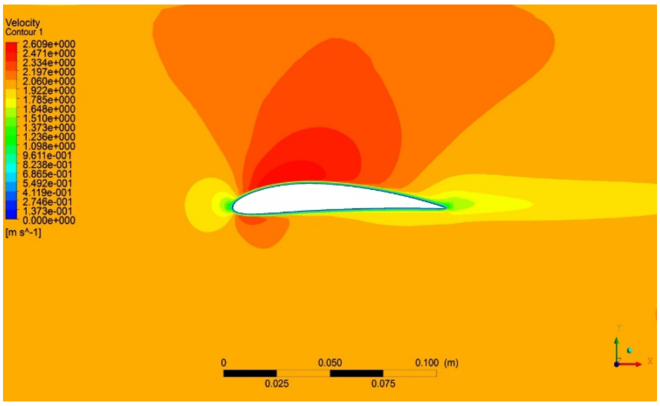

Fig. 5 Velocity distribution around profile

The students must expect the shape of the pressure or velocity distribution. If there is another shape, it is possible to make a mistake.

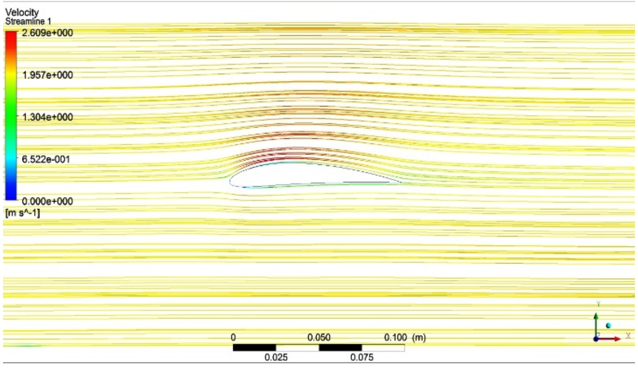

Fig. 6 Streamline distribution around NACA 5412 hydrodynamic profile

Reynolds number calculated is 199201. This implies that we have a turbulent flow. Streamline distribution (Fig. 6) [3] presents the fact that we don't have a boundary layer separation point due to the shape of the profile.

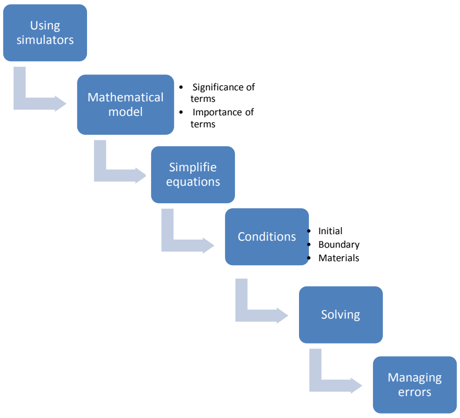

Fig. 7 Using of simulators by understanding fluid mechanics

Fluid mechanics knowledge is necessary for understanding phenomena and significance of terms, for choosing the material of the fluid (density, viscosity) and of the solid (conduits), for putting boundary conditions and, finally, for analysing the results.

In Figure 7 I synthetized the problems presented in the chapter 4. I added a very important problem which demands thorough knowledge of Fluid mechanics: managing errors (source, magnitude, solutions to reduce them, etc.).

5 Conclusion

In the education system, we have to combine CFD – Computational Fluid Mechanics with CFM – Classic Fluid Mechanics, its Mathematical model.

The purpose of this paper was to evidence the importance of Fluid mechanics in different phases of simulators' use, especially regarding maritime education.

Analysing the problem we can conclude:

- Understanding simulation means understanding Mathematical model.
- Mathematical model represents physical phenomena, but not entire, simplified.
- To simplify physical phenomena you must know the significance of terms, which terms are most important in different cases and which can be neglected. It is also important to set correctly initial and boundary conditions.
- General equations of the fluid movement, like Navier-Stokes equation, represents itself a simplified form of the natural phenomena.
- Managing errors and comparing with experimental results.
- Finally the testing method must relive both the simulators abilities and the knowledge of basic fluid mechanics.

Highlighting the contributions of the paper:

- a synthesis of the hydrodynamic forces acting on the ship;
- analysis of hydrodynamic processes on navigation simulators;
- a new approach of maritime education regarding fluid mechanics.

Learning Fluid mechanics adapted to new computational requirements as well as studying simulation technologies together with their physical models is a necessity for maritime students and not only.

References:

[1] Dinu D., Fluid Mechanics for Seafarers, *Nautica Ed.*, 2010.
[2] Dinu D., Using Computer Fluid Dynamics (CFD) for Teaching Hydrodynamics in Maritime Faculties, *10th International Conference on Engineering Education (EDUCATION '13),* Cambridge, 2013.
[3] Dinu D., Modern Methods for Study and Evaluation in Maritime Universities According to MET Requirements, *7th International Conference on Education and Educational Technologies (EET '16),* Istanbul, Turkey, 15-17 April, 2016.
[4] Scupi A., Dinu D., Fluid Mechanics – Numerical Approach, *Nautica Ed.*, 2015.
[5] Scupi A, Avital E J, Dinu D, Williams J J R, Munjiza A Large Eddy Simulation of Flows Around a Kite Used as an Auxiliary Propulsion System *Journal of Fluids Engineering-Transactions of the ASME,* 2015.
[6] *** ANSYS Fluent v. 14.5, Theory Guide, 2012.
[7]***Transas Marine, *NAVI-TRAINER 4000, Mathematical Models, Technical Description*, 2003.