Using Computer Fluid Dynamics (CFD) for Teaching Hydrodynamics in Maritime Faculties

DUMITRU DINU
Marine Engineering Department
Constanta Maritime University
900663, Constanta, 104 Mircea Street
ROMANIA
dinud@imc.ro, dinud@cmu-edu.eu http://www.cmu-edu.eu

Abstract: - ANSYS FLUENT is a CFD program based on finite volume calculation. It solves the Navier-Stokes equations for different cases. To calculate, we need to define geometry, corresponding to our problem and to create the mesh (the volumes in 3D or the squares in 2D). After realizing the geometry and the mesh we proceed to calculate. Post processing is also a very important phase. The program is use 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 Computer Fluid Dynamics. In the paper we give an example: flow measurement using the aperture stop. Mainly, these method is based on the fact that by throttling the flow section, there arises a difference between the pressure upstream and downstream from throttling which depends on the stream velocity and, implicitly, on flow. Finally, we emphasize the possibility to use CFD for research. The program can be used as a “virtual stand” – is one of the interesting conclusions of the paper.

Key-Words: - CFD, Hydrodynamics, Virtual Stand, NACA Profile, Kite, Barrage.

1 Introduction

Our students meet the ANSYS FLUENT program on courses, applications and research (master and Ph.D. studies). We use this program to illustrate fluid properties and some important phenomena in hydrodynamics: Bernoulli’s equations for the relative movement of ideal non-compressible fluid, hydrostatic forces, flow with and without circulation around profiles, velocity distribution in circular conduits, forces on the hydrodynamic profiles, induced resistances in the case of finite span wings, boundary layer, etc. Also the students can make a comparison between the results obtained in the laboratory and the results of simulation using Computer Fluid Dynamics. Vectorial representation of the velocity field is very important to understand the problem. Finally, we emphasize the possibility to use CFD for research, mentioning some of the relevant works in the field. In one of them we studied the flow through circular conduit using distortional similarity by comparing “experimental” results with theoretical results, calculated by application of scale of physical magnitudes. It is difficult to achieve a model for experimenting. Also it is difficult to calculate the phenomena in the nature scale using the FLUENT. So we calculated the physical parameters, using FLUENT on the model and we pass them, using the similitude criteria, in the nature.

2 Illustrate important phenomena in hydrodynamics

We use this program to illustrate fluid properties and some important phenomena in hydrodynamics: Bernoulli’s equations for the relative movement of ideal non-compressible fluid, hydrostatic forces, flow with and without circulation around profiles, velocity distribution in circular conduits, forces on the hydrodynamic profiles, induced resistances in the case of finite span wings, boundary layer, etc. Flow measurement using the aperture stop allows us to show the student a very interesting application of Bernoulli’s and continuity’s equations. Mainly, these method is based on the fact that by throttling the flow section, there arises a difference between the pressure upstream and downstream from throttling which
depends on the stream velocity and, implicitly, on flow. In fact, on the basis of this principle, any local resistance may be used for measuring the flow. To determine the flow in conduits, there are frequently used Venturi tubes and aperture stops. The aperture stop (Fig. 1) is a disk with a central orifice, of a smaller diameter than that of the conduit \((d < D)\), which is coaxial, mounted on the conduit rout, between two flanges. The fall of pressure:

\[
\Delta p = p_2 - p_1 = h, \quad (1)
\]
is measured by means of two piezometric tubes or of a differential manometer.

3. Comparing the results obtained in the laboratory and the results of CFD

The calculation of a hydrodynamic profile for a fluid that flows around it, mainly consists in determining the variation of drag force and bearing force, whose values vary with the shape profile, wing span, incident angle and type of flow (Reynolds number). Thus for NACA 6412 profile, we will calculate and compare the changes of coefficient force values mentioned above for finite span wings (large-span and small-span). The calculation will be done both experimentally in a naval wind tunnel, which will allow us to change the incidence angle and speed of fluid (and hence the Reynolds number), and with a computer fluid dynamics - CFD (ANSYS 14). Thus we can compare the results from the two methods outlined above. We will also study the possibility of performing an experiment on a large-span wing, the transition to the small-span one will be done by the similarity in two scales. These experimental and numerical approaches can be used to study finite span naval profiles (e.g. rudders).

3.1 Working parameters

We considered a NACA 6412 profile with a relative elongation 6 with the following characteristics:
- the length of the chord equal to 0.080 [m];
- the wing span equal to 0.480[m].
The profile is located in an air stream with a velocity of 15 m/s. The Reynolds number has the value of 85,000.

The angles of incidence are:
- \(-10^\circ \div 0^\circ\) step 20;
- \(+0^\circ \div 150^\circ\) step 30;

The forces that are acting upon a hydrodynamic and aerodynamic profile are: the lift force and the friction force or force due to boundary layer detachment. These forces give a resultant force \(R\) which decomposes by the direction of velocity at infinity and by a direction perpendicular to it. \(R_x\) component is called drag force and \(R_y\) component is called lift force.

3.2 Experimental determination of aerodynamic forces

Experiments were made in a naval aerodynamic tunnel. Airflow was uniform on a section of 510 \(\times 580\) mm.

A tensometric balance was used to determine the forces acting upon the wing. In Fig. 4 and Fig. 5 we have represented the graphics results of tests of the function \(C_y(\alpha)\) and \(C_x(\alpha)\), respectively [4].

![Fig. 4 Graphic of \(C_y\) experimentally obtained](image)

3.3 Determination of the aerodynamic forces using CFD

Using Design Modeller v. 13.0, we were able to accurately reproduce the NACA 6412 profile, as represented in Fig. 6. Airflow was uniform on a bigger section 980 \(\times 511\) mm.

![Fig. 6 Geometric representation of the NACA 6412 profile.](image)

We discretized the NACA profile in more than 10 million cells, of which 9 million are hexahedrons, 55,000 are wedges, 35,000 are polyhedral, 1500 are pyramids and only 400 are tetrahedrons. The mesh has also over 30 million faces and 11 million knots. Using Fluent program version 13.0, we set the boundary conditions as follows:
- The profile is attacked with a velocity of 10 m/s, under different angles, namely \(-10^\circ, -8^\circ, -6^\circ, -4^\circ, -2^\circ, 0^\circ, +2^\circ, +4^\circ, +6^\circ, +8^\circ, +10^\circ, +12^\circ, +15^\circ\);
- Behind the profile, atmospheric pressure is equal to 101325 Pa.
The fluid motion is turbulent with a Prandtl number equal to 0.667. The air density is considered constant and it is equal to 1.225 kg/m³; The air dynamic and cinematic viscosity are also considered constant and are equal to $1.7894 \times 10^{-5}$ kg/ms, 0.0001460735 m²/s, respectively; The turbulence viscosity ratio is set to 10.

Process has stabilized after 208 iterations allowing us to visualize the values of drag and lift forces and their coefficients, presented in Fig. 7 and Fig. 8 [4].

Recent Advances in Educational Methods

ISBN: 978-1-61804-163-0

31

In the Fig. 9 is presented the kite having the main characteristics: Length - 26.1 m; Width - 7.0 m; Area - 182.7 m²; Thickness - 0.7 m.

Using Fluent program version 13.0, we put boundary conditions as follows:
- Kite is attacked with a velocity of 10 m/s, under different angles, namely 0°, 30°, 60°, 90°,
- Behind the kite, atmospheric pressure is equal to 101325 Pa.
- The fluid motion is turbulent with a Prandtl number equal to 0.667.
- The air density is considered constant and it is equal to $1.225 \frac{kg}{m^3}$;
- The air dynamic and cinematic viscosity are also considered constant and are equal to $1.7894 \times 10^{-5}$ $\frac{kg}{ms}, 0.0001460735 \frac{m^2}{s}$, respectively;
- The turbulence viscosity ratio is set to 10.


The unconventional naval propulsion system would solve some of the pollution problems posed by marine engines operation. This system also offers substantial fuel savings. We have simulated a kite used as auxiliary propulsion system. This simulation, performed with CFD (Computer Fluid Dynamics), was made for different working conditions by varying the incidence angle and the velocity at infinity. We also calculated total aerodynamic force, which projected on the water surface, gives us the force by which the ship is being towed (the component of the ship direction movement) [2].
For wind velocity $v = 20$ m/s we obtained the aerodynamic forces presented in the table 1 [3].

<table>
<thead>
<tr>
<th>$\phi$ (Incidence angle) $[^\circ]$</th>
<th>D (Drag force) [N]</th>
<th>L (Lift force) [N]</th>
</tr>
</thead>
<tbody>
<tr>
<td>$0^\circ$</td>
<td>141125.17</td>
<td>604.49</td>
</tr>
<tr>
<td>$30^\circ$</td>
<td>103386.95</td>
<td>513.85</td>
</tr>
<tr>
<td>$60^\circ$</td>
<td>52001.92</td>
<td>158.202</td>
</tr>
<tr>
<td>$90^\circ$</td>
<td>22502.57</td>
<td>20.95</td>
</tr>
</tbody>
</table>

Table 1. Aerodynamic forces.

### 5 “Virtual stand” concept

In many cases, it is very difficult (and expensive) to represent the phenomena in the scale, to achieve a model.

Taking into account the physical magnitudes which influence the analyzed phenomena we can establish the model law.

We use the FLUENT program to calculate the physical magnitudes for the model. Afterwards we pass the data in the nature, using the scales.

FLUENT program acts as experimental stand.

FLUENT program allow us to simulate an laboratory experiment. Using the similitude theory, we can pass from the model to the nature.

We apply this to simulate the flow through a broken barrage. The problem of the action of the current is not very simple, especially if we discuss about high velocities (i.e. river in the spring time). Using FLUENT we have obtained the contour of velocities (Fig. 10) and the forces (Table 2) [1].

According to the scale of the force

$$k_F = k_j^3,$$

the force acting on the remaining brakewater is:

$$F = 1714 \times 25^3 = 26781250 \text{ N} =
2678 \times 10^3 \text{ daN}. \quad (3)$$

### 6 Conclusion

The programme is very appropriate to illustrate fluid properties and some important phenomena in hydrodynamics.

The calculation of a hydrodynamic profile allows us to compare the results. Comparing the $C_y$ coefficients values obtained by experiment and using CFD, we can make the observation that they
are very similar in the field of the incidence angles $[-2^\circ, 6^\circ]$.

Also, comparing the $C_x$ graphic, we remark that the graphics are very similar, but between the values there are some differences.

These differences are due to experimental errors (errors of measurement devices), numerical errors (rounding errors), and also discretization errors.

Also, the CFD programme doesn’t take into account the induce resistance in the case of finite span wings. As a consequence an induce angle $\alpha_i$ will appear which thus decreases the incidence angle $\alpha$. The alteration of direction and value of velocity bring about the corresponding alteration of lift force, which is perpendicular on the direction of stream velocity.

As I specified before, it is difficult to achieve a model for experimenting. Also it is difficult to calculate the phenomena in the nature scale using the FLUENT. So we calculated the physical parameters, using FLUENT on the model and we pass them, using the similitude criteria, in the nature.

The results obtained using the single scale similitude method are very close to the nature. The main idea is that FLUENT can be used as an experimental stand.

References:


