# Use of CFD for the prediction of the interaction between ships in restricted waters

Adel A. Banawan and Mohamed S. Ebieda
*Naval Architecture and Marine Eng. Dept., Faculty of Eng., Alexandria University, Alexandria, Egypt*

In this paper, the Computational Fluid Dynamics (CFD) is used to investigate the interaction between two ships without propellers in restricted water in meeting maneuver as well as the bank suction effect on a single ship proceeding on an off-centerline course. The flow around the ship is assumed to be two dimensional, i.e. the flow in the z-direction is neglected, and thus the effect of the free surface will not appear. Also, the motion of the ship is restricted in the y-direction, so she keeps moving at constant heading. This could be accomplished by using a counteraction from the rudder to oppose the hydrodynamic forces and moments generated due to bank suction. The approach used to simulate the meeting maneuver is based on the Artificial Compressibility method. The CFD software used in this study is “MOUSE” which is based on finite volume computations on unstructured grids.

فى هذا البحث تم استخدام ديناميكا الموائع الحسابية فى توقع التأثير المتبادل بين سفينتين تسيران فى ممر مائى ضيق عندما تقابل إحداهما الأخرى التى تسير فى الإتجاه المضاد، كما تم استخدام تلك الطرق فى حساب تأثير جذب ضفة المجرى المائى لسفينة تسير بالقرب منها وبعيداً عن منتصف المجرى المائى. وقد تم افتراض ان السريان حول السفينة ثنائى الأبعاد وبالتالى اهمال السريان فى الأتجاه الرأسى. وتعتمد طريقة الحساب المستعملة على طريقة الأنضغاط الأصطناعى والشبكة الحسابية الغير منتظمة. وقد أستعمل فى الحسابات برنامج للحاسب الآلى متخصص يدعى MOUSE يعتمد على طريقة حسابات الحجم المحدود على الشبكات الحسابية الغير منتظمة.

**Keywords:** Restricted waterways, Ship maneuvering, Interaction between ships, Computational fluid dynamics, bank suction effects

## 1. Introduction

In restricted waters, a ship’s behavior is affected by the presence of the lateral limits of the navigation area, such as banks and quay walls. These restrictions would influence the hydrodynamic forces and moments acting on the ship hull.

When a ship proceeds on an off-centerline course of a channel, the flow around the ship becomes asymmetrical and the speed of the flow between the ship and the near side of the channel increases and thus the pressure decreases. This low pressure acts as an attraction force to pull the ship further to that near side. Also there is an outward moment applied on the ship which tends to turn her toward the channel centerline away from the bank. These forces and moments are known as the “bank suction effect”. The bank suction increases as the ship deviates from the centerline of the channel.

For a given ship, the bank effects depend on several parameters; ship bank distance, ship speed, depth-draft ratio and bank geometry.

Several experimental studies on ship-bank interaction were reported by: Fuehrer and Römisch, Dand [1], Norrbin Ch’ng et al., Vantorre, Li et al. [2] Eda [3], Fujino [4]. Moreover, many other theoretical and numerical studies were also conducted [1- 4].

Another important problem is the interaction between two ships as they meet or pass each other in a narrow canal. In this case, they are affected by forces and moments, which change their intensity and direction as the relative position of two ships changes. It should also be pointed out that it is much more dangerous to pass than to meet each other [5].

There are few published data from experimental research on interaction between ships. Newton [5], Muller [1], Remery, Dand [2]. Comprehensive test series with ship models of both equal and different lengths in overtaking and encountering maneuvers are described by Vantorre et al. [2].

Other authors have developed numerical methods to calculate interaction forces theoretically, e.g. Tuck and Newman, Kijima, Kaplan and Sankaranarayanan [2], Bet et al. [6].

It should be kept in mind that the general pattern of the time histories of the lateral force and of the yawing moment acting on a ship mainly depends on the ships length ratio, and the ships speed ratio.

When a ship meets another ship in a channel there are hydrodynamic interactions between the two ships as follows [7]:

a. When the bow of the first ship meets the bow of the other ship, the two ships are initially repelled, i.e. the two ships are attracted to the sides of the channel. Also there are inward moments applied to the two ships, i.e. the two bows move toward the channel centerline.

b. When the bow of the first ship abreast the amidships of the other ship, the repulsion forces are reduced to zero, after that the ships are then attracted. Also the inward moments changes to outward moments.

c. The attraction forces has a maximum value after the two ships are abreast, and then there are outward moments, i.e. the sterns tend to move toward each other.

d. When the stern of the first ship abreast the stern of the other ship, the forces between the two ships become repulsion, and the moments start to become inward moments, i.e. the bow start to turn toward the channel centerline.

When a ship passes another ship as shown in fig. 1 [8], there are hydrodynamic interactions between the two ships which is nearly the same as the meeting case, Force {Repulsion – Attraction – Repulsion} and Moment {Inward – Outward – Inward}:

The lateral force and yaw moment increase considerably with decrease in water depth, and also with decrease in lateral separation distance, in both meeting and passing maneuvers.

The slower ship of the two ships in the meeting or passing maneuver experiences larger lateral force and moment than the faster ship. The interaction force and moment acting on the slower ship increase drastically if the speed and size of the overtaking ship increase.

Computational Fluid Dynamics (CFD) has recently been recognized as a useful tool at principal and final design stages. CFD is economical in time and cost in comparison to the experimental methods, and allows the designer to estimate and predict some characteristics of the flow pattern around the ship hull form which are not possible to be obtained by model tests.

However, in order to use computational methods, CFD, to predict some characteristics of the flow pattern around the ship, the CFD results must be credible and accurate enough.

This paper investigates the use of CFD for the prediction of forces and moments acting on the ship while meeting another ship in a restricted waterway.

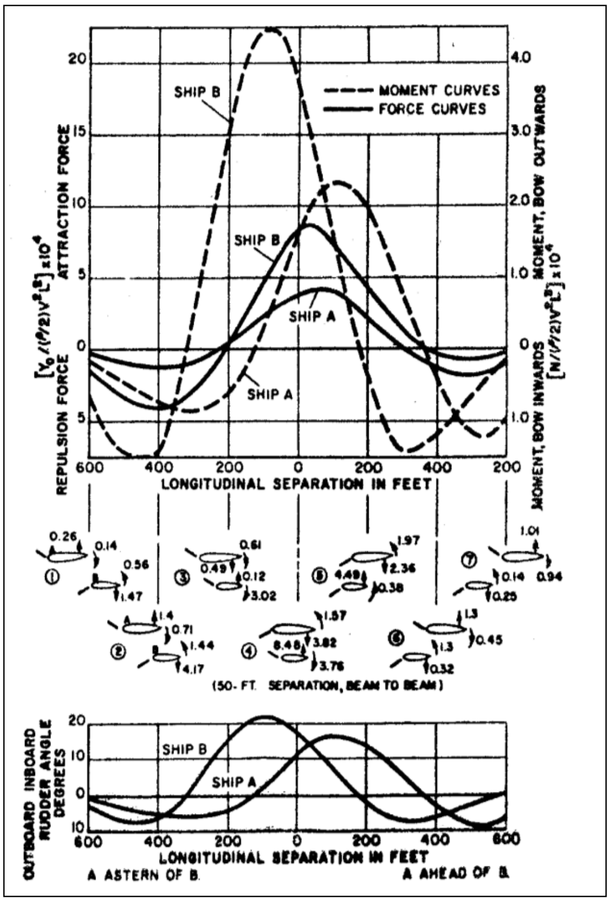

Fig. 1. Interaction forces and moments between two ships.

## 2. Governing equations

The governing equations used in the solution method were the Navier-Stokes equations for an incompressible fluid influenced by gravity [9- 11]. Normalized by the length of the body *L*, the undisturbed flow velocity *U*, the fluid density $\rho$, and viscosity $\nu$, the equations can be written as follows:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 . \qquad (1)$$

$$\frac{\partial u}{\partial t}+\frac{\partial uu}{\partial x}+\frac{\partial vu}{\partial y}+\frac{\partial wu}{\partial z}+\frac{\partial \overline{p}}{\partial x}=\frac{1}{\mathrm{Re}}\theta_x . \qquad (2)$$

$$\frac{\partial v}{\partial t}+\frac{\partial uv}{\partial x}+\frac{\partial vv}{\partial y}+\frac{\partial wv}{\partial z}+\frac{\partial \overline{p}}{\partial y}=\frac{1}{\mathrm{Re}}\theta_y . \qquad (3)$$

$$\frac{\partial w}{\partial t}+\frac{\partial uw}{\partial x}+\frac{\partial vw}{\partial y}+\frac{\partial ww}{\partial z}+\frac{\partial \overline{p}}{\partial z}=\frac{1}{\mathrm{Re}}\theta_z , \qquad (4)$$

where $u$, $v$ and $w$ are velocity components of the flow in $x$, $y$ and $z$ directions, respectively and $\overline{p}$ denoting the total pressure.

The viscous terms $\sigma$ are defined as follows:

$$\theta_x=\frac{\partial}{\partial x}\sigma_{xx}+\frac{\partial}{\partial y}\sigma_{xy}+\frac{\partial}{\partial z}\sigma_{xz} ,$$

$$\theta_y=\frac{\partial}{\partial x}\sigma_{xy}+\frac{\partial}{\partial y}\sigma_{yy}+\frac{\partial}{\partial z}\sigma_{yz} ,$$

$$\theta_z=\frac{\partial}{\partial x}\sigma_{xz}+\frac{\partial}{\partial y}\sigma_{yz}+\frac{\partial}{\partial z}\sigma_{zz} ,$$

with:

$$\sigma_{xx}=2\nu\frac{\partial u}{\partial x} \qquad \sigma_{xy}=\nu(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}) ,$$

$$\sigma_{yy}=2\nu\frac{\partial v}{\partial y} \qquad \sigma_{yz}=\nu(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}) ,$$

$$\sigma_{zz}=2\nu\frac{\partial w}{\partial z} \qquad \sigma_{xz}=\nu(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}) ,$$

where $\nu$ is the kinematic viscosity of the water.

## 3. Concept of artificial compressibility

The pressure in a rigorous incompressible flow acts like a relaxation parameter to satisfy the continuity eq. $\nabla\overline{v}=0$.

There is a possible way to determine the pressure field by the coupling of mass and momentum equations using the concept of artificial compressibility in analogy of the compressible flow.

The original continuity eq. (1) is modified by adding an artificial time derivative of pressure representing the artificial compressibility:

$$\frac{\partial \overline{p}}{\partial t}+\beta^2(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z})=0 . \qquad (5)$$

Where $\beta$ acts as an artificial compressibility parameter. This approach was first proposed by Chorin [12].

Small disturbances in pressure propagate in an incompressible fluid with infinite velocity. The artificial compressibility limits the propagation speed and in contrast to an incompressible fluid, the effects of disturbances will be delayed. The degree of delay depends on the value of $\beta$. For $\beta > 0$ the pseudo-compressible flow is comparable to subsonic flows.

The solution of the system with modified continuity equation is unphysical for transient flows but however, at steady-state the time-derivative $\frac{\partial \overline{p}}{\partial t}$ vanishes and the original continuity equation for incompressible fluids will be remained.

The propagation speed of a pressure wave in a pseudo-compressible flow is influenced considerably by the selection of the parameter $\beta$. An increase of the parameter $\beta$ results in a disturbance spreading faster into the zone of flow, and the solution will approach more closely to the solution of a completely incompressible flow.

Thus the selection of a suitable value for $\beta$ is subject to certain restrictions;

$$\beta^2 = \gamma.\max[(u^2 + v^2 + w^2), \beta_{\min}^2, \qquad (6)$$

where the parameter $\gamma$ is a factor of the order of magnitude 1.

A reasonable lower boundary for $\beta$ can be estimated from the condition that artificial pressure waves propagate faster than the viscous effects.

Suitable values for $\beta$ lay between 0.4 and 2.0 depending on the examined flow problem.

## 4. The software MOUSE

A software called MOUSE [13] was used for the calculations of pressure distribution around ships. MOUSE is an object oriented framework for finite volume computations on unstructured grids. Right now it is mainly targeted at people who want to develop specialized numerical programs. One of the main objectives has been to ease the use of unstructured grids for finite volume codes.

MOUSE is, of course not restricted to CFD problems, it is a library for finite volume computations.

In the current version a node centered control volume is used (see fig. 2).

MOUSE's Control Language (MCL) is a script like language used to describe the structure of the application. MOUSE is a collection of classes which can be used to create CFD or other numerical applications. Objects are arranged in a tree structure. Every command in MCL represents the description of a C++ object, which will be put at this point in the object tree.

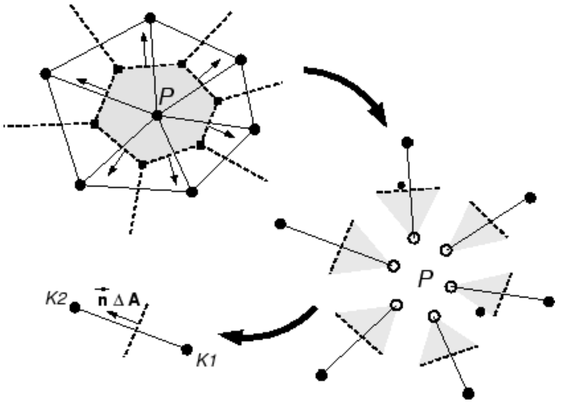

Fig. 2 . Node centered control volume.

## 5. Computation of the bank suction effect acting on a ship

In this case, a single ship is moving in a channel at constant velocity and having an offset from the channel centerline. Due to her position it will be subjected to suction force to the nearer bank side. CFD is used through MOUSE to calculate the pressure distribution along both sides of the ship and hence calculate a round figure for the value of this suction force. Fig. 3 shows the boundary conditions for this problem.

In order to obtain accurate and stable results, a very refined grid is required near the regions where turbulence may occur. This refinement is accomplished by using unstructured grid as shown in fig. 4.

The pressure distribution around the ship obtained from the calculations is shown in fig. 5-a and the pressure distribution along the two sides of the ship and the resultant forces are shown in fig. 5-b.

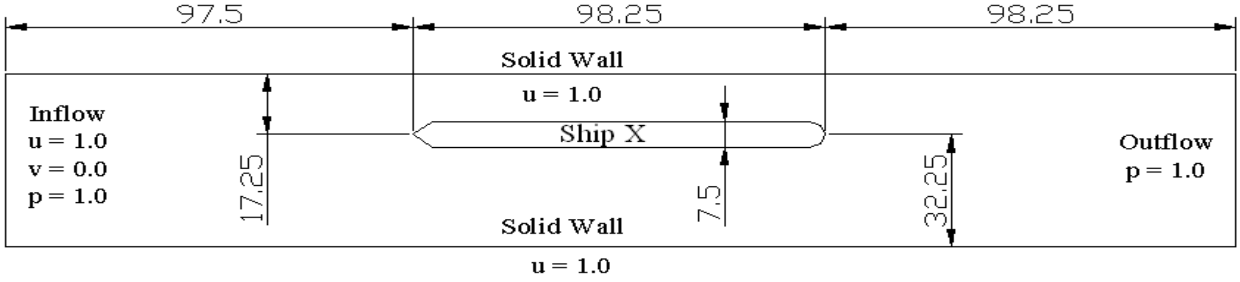

Fig. 3. Boundary conditions for single ship.

## 6. Computation of hydrodynamic forces and moments acting on two ships during meeting maneuver

In this case, ship X is moving in a channel at constant velocity and having an offset from the channel centerline and another smaller ship Y is moving in the opposite direction. Due to meeting, the hydrodynamic force acting on ship X will change from repulsion to attraction and then to repulsion again. Also the hydrodynamic moment acting on ship X will change from inward to outward to inward again as discussed earlier. CFD is used through MOUSE to calculate the pressure distribution along both sides of the ship and hence calculate the hydrodynamic force during this meeting maneuver. Ship Y is moving with the same speed of the flow.

Fig. 6 shows the boundary conditions of the problem.

The pressure distribution around ship X and ship Y is calculated at different relative positions between the two ships (A, B, C, D and E). Figs. 7-11 show the pressure distributions around the two ships and the resultant force for each position.

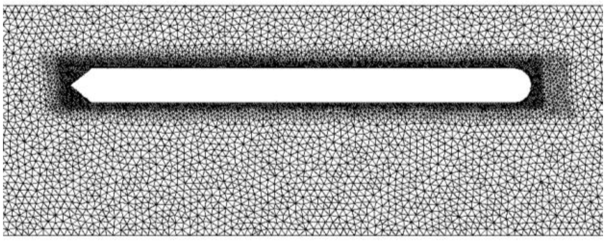

Fig. 4: Unstructured grid

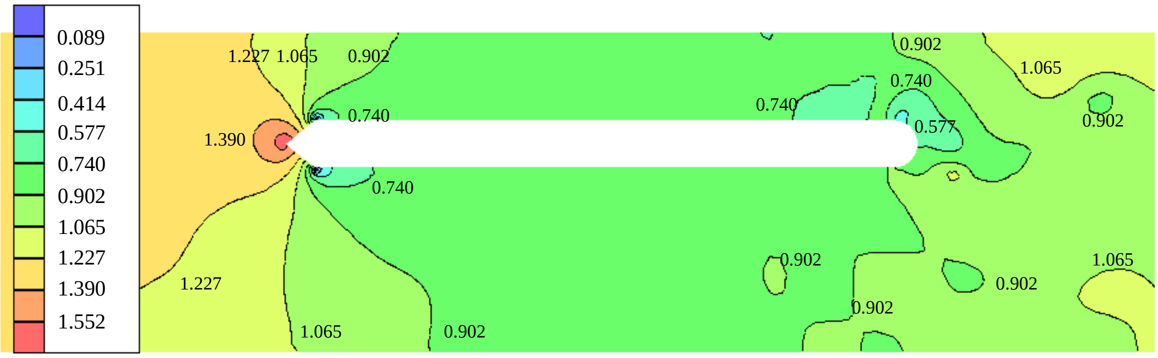

Fig. 5-a. Pressure “p” distribution around a ship.

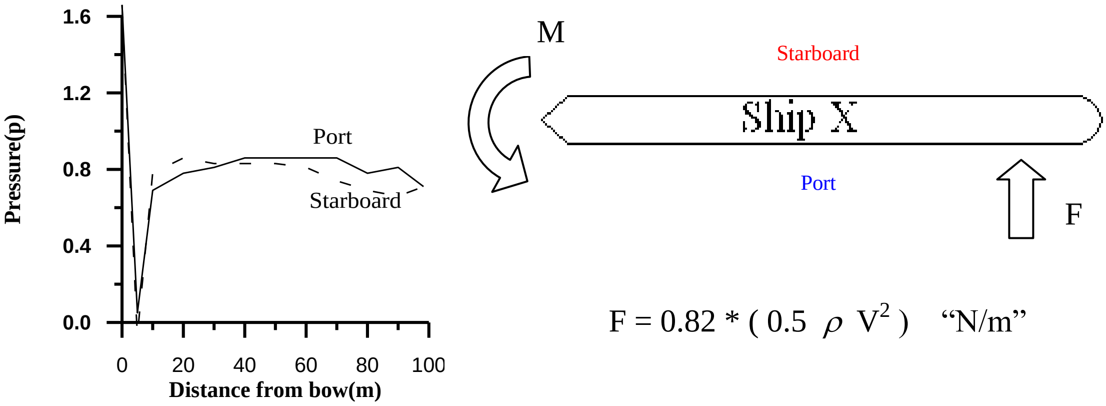

Fig. 5-b. Pressure "p" distribution along the two sides of ship X and resultant force.

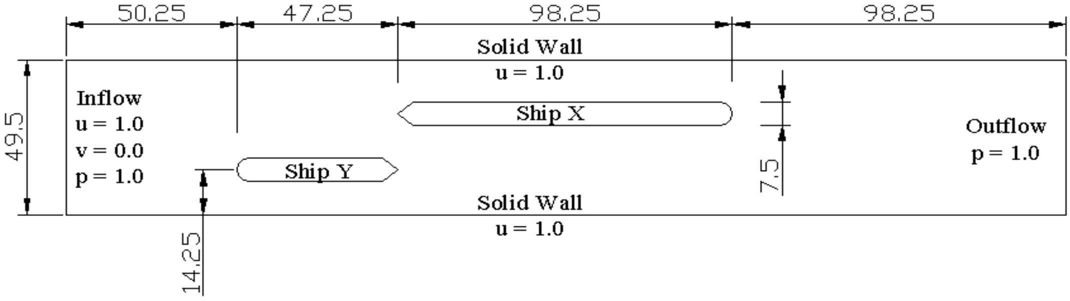

Fig. 6 Boundary conditions for meeting maneuver position A.

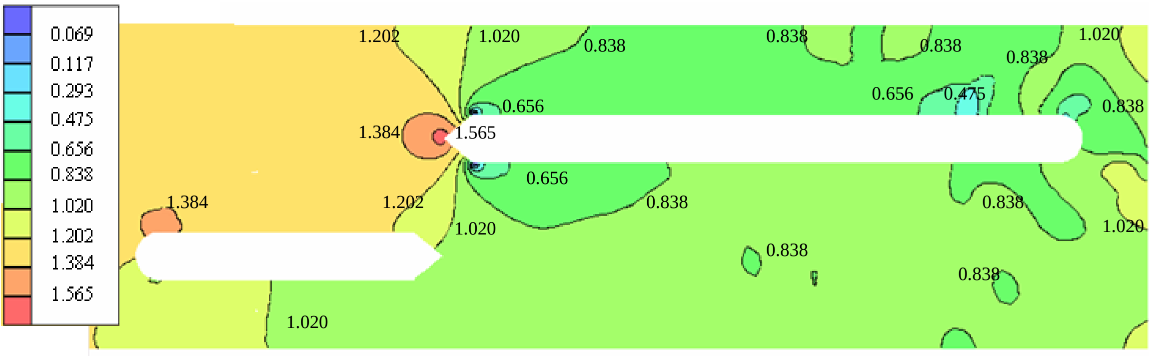

Fig. 7-a. Pressure “p” distribution around ship X and ship Y in position A
(when the bow of ship X meets the bow of ship Y).

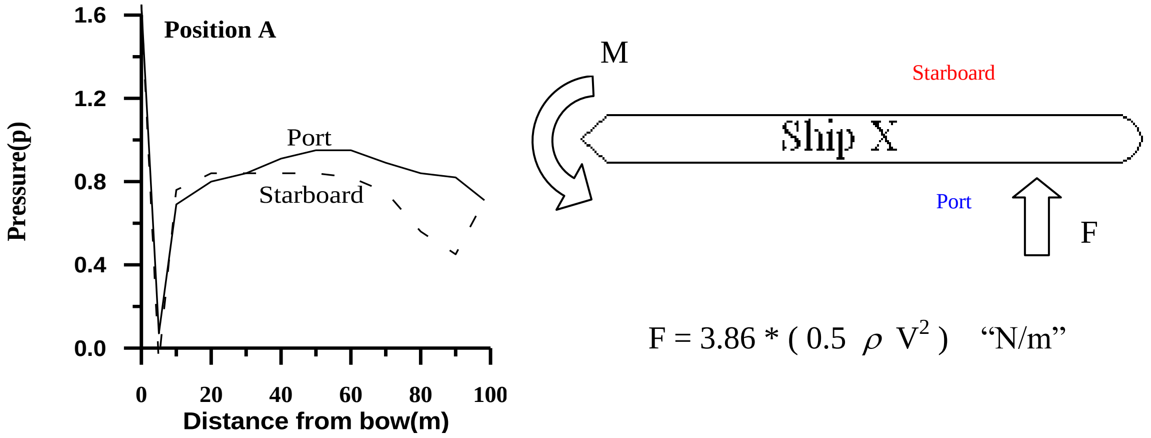

Fig. 7-b. Pressure "p" distribution along the two sides of ship X and resultant force in position A.

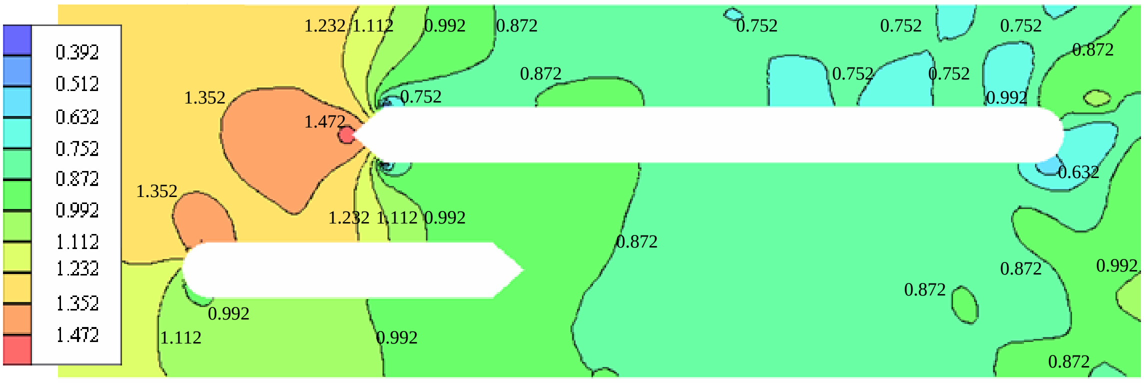

Fig. 8-a. Pressure "p" distribution around ship X and ship Y in position B
(when the bow of ship X abreast the amidships of ship Y).

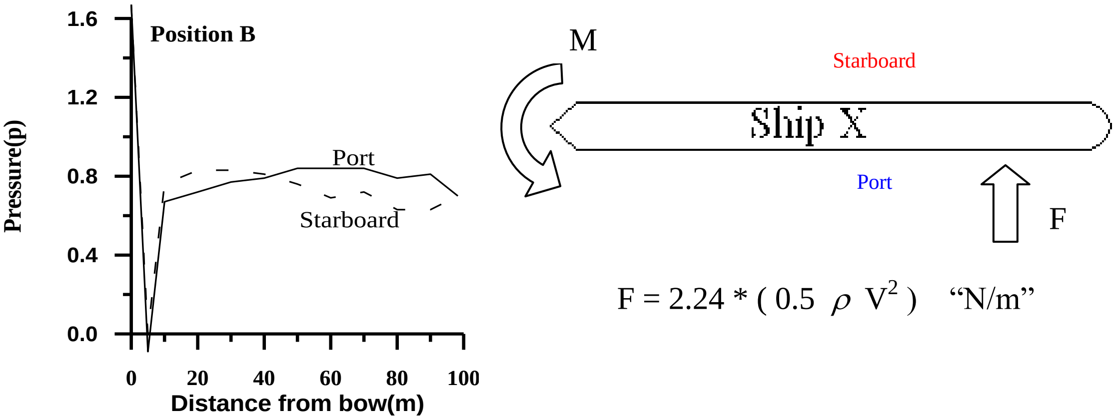

Fig. 8-b. Pressure "p" distribution along the two sides of ship X and resultant force in position B.

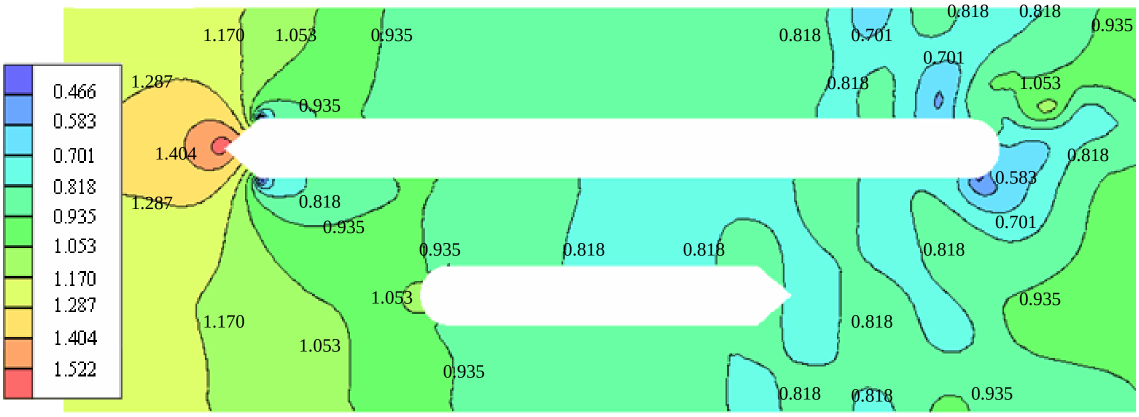

Fig. 9-a. Pressure "p" distribution around ship X, ship Y in position C
(when the two ships are abreast).

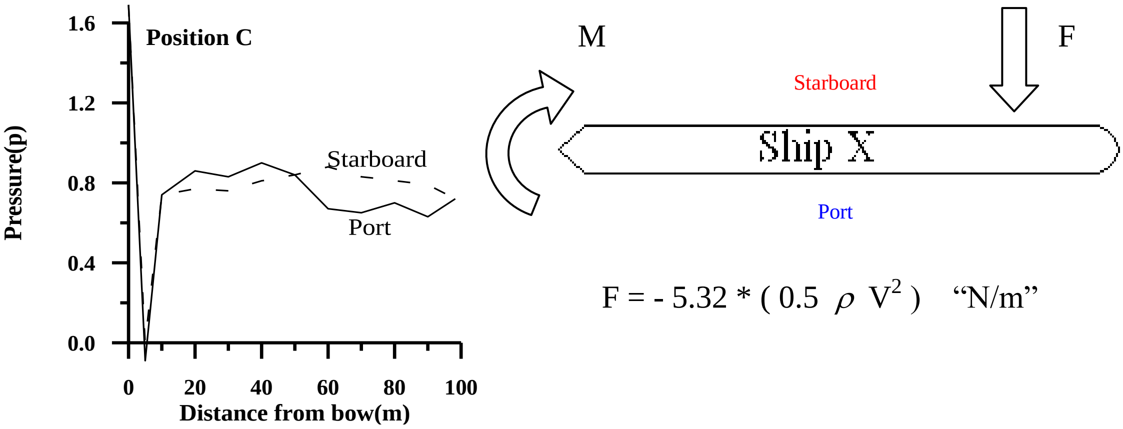

Fig. 9-b. Pressure "p" distribution along the two sides of ship X and resultant force in position C.

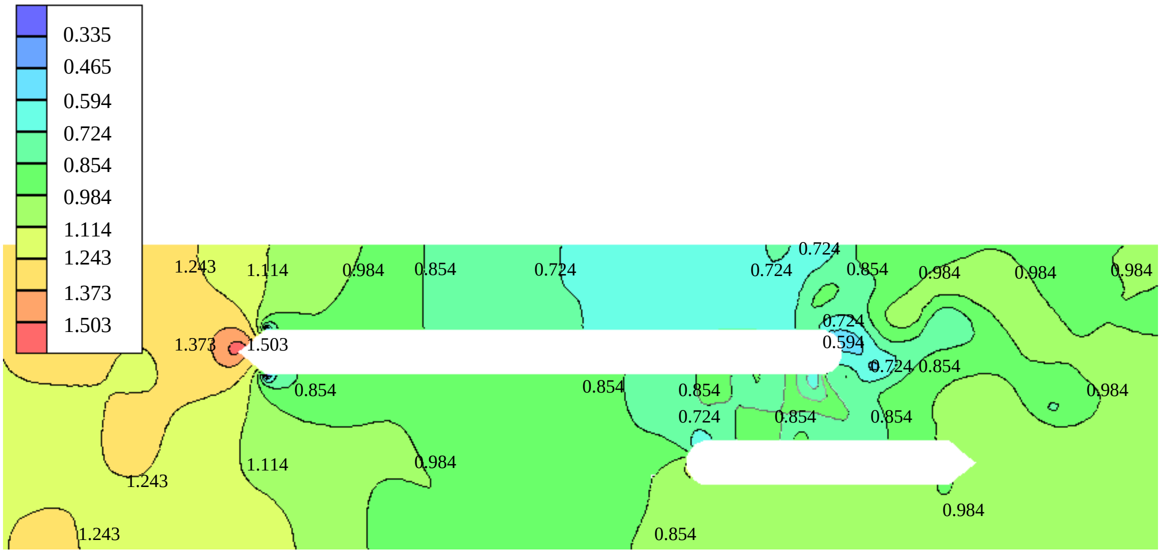

Fig. 10-a. Pressure "p" distribution around ship X, ship Y in position D
(when the stern of ship X abreast the amidships of ship Y).

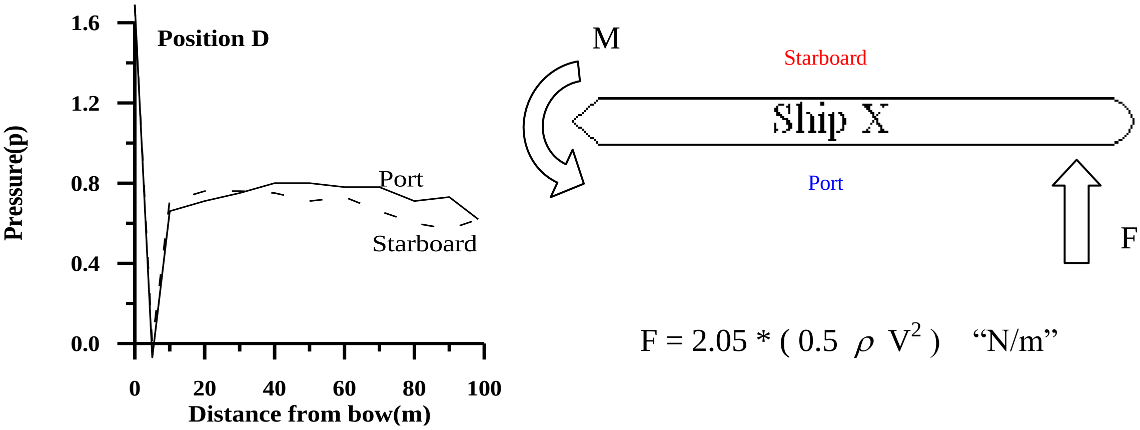

Fig. 10-b. Pressure "p" distribution along the two sides of ship X and resultant force in position D.

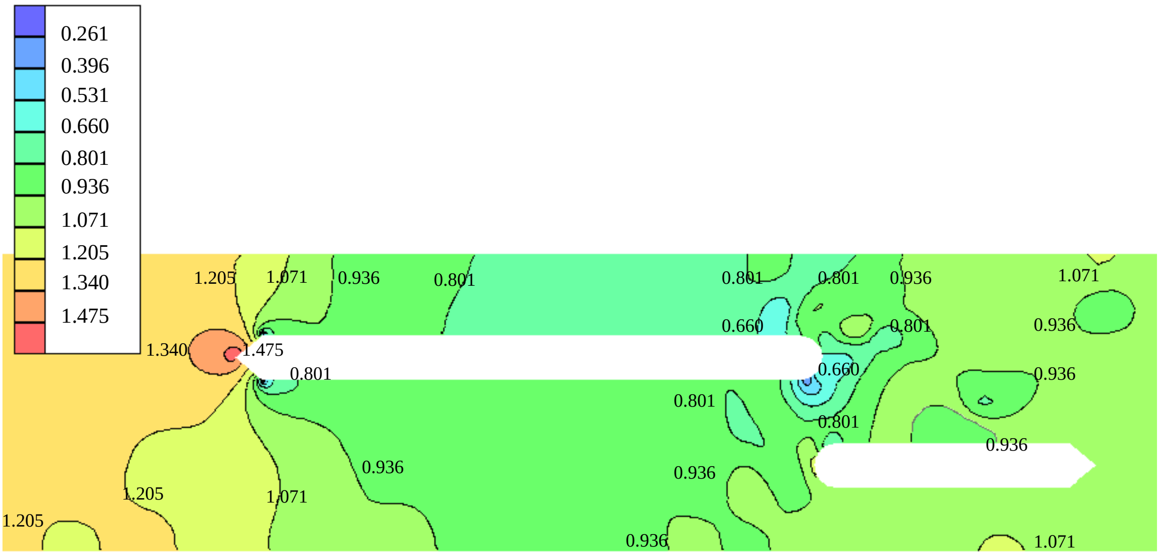

Fig. 11-a. Pressure "p" distribution around ship X, ship Y in position E
(when the stern of ship X abreast the stern of ship Y).

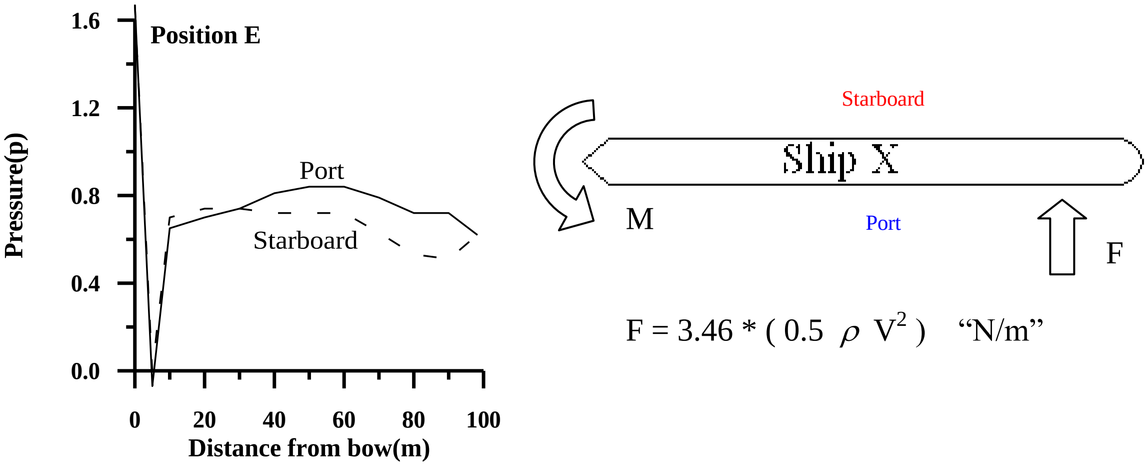

Fig. 11-b. Pressure "p" distribution along the two sides of ship X and resultant force in position E.

Table 1
Summary of results of the two maneuvers

| $L_Y$ (m.) | | Position A | Position B | Position C | Position D | Position E |
|---|---|---|---|---|---|---|
| 47.25 | Force coefficient | - 3.86 | - 2.24 | + 5.32 | - 2.05 | - 3.46 |
| | Moment direction | Inward | Inward | Outward | Inward | Inward |
| 98.25 | Force coefficient | - 8.04 | - 0.24 | + 11.21 | - 0.15 | - 7.86 |
| | Moment direction | Inward | Negligible | Outward | Negligible | Inward |

Another meeting maneuver was simulated using the same software but ship Y was set to be 98.25 m in length, the same as ship X, to show the effect of ship size on the hydrodynamic forces and moments.

The result of the two maneuvers can be summarized in table 1 and fig. 12. Details of case study and its results are given in ref. [14].

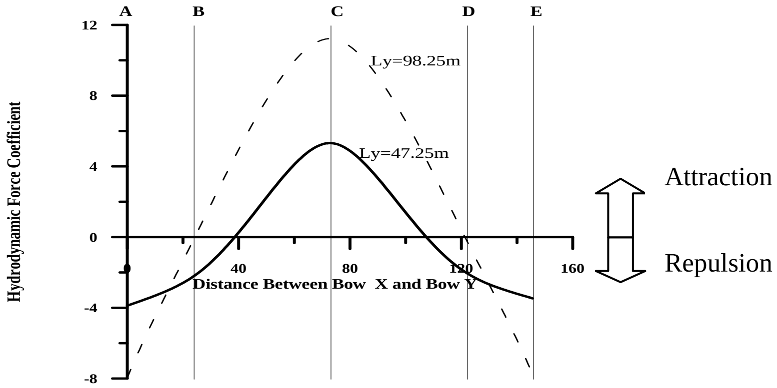

Fig. 12. Resulted hydrodynamic forces acting on ship X due meeting maneuver.

## 7. Conclusions

1. As shown in the above case studies, CFD makes the study of fluid flow easier and more effective, once a model for the problem is established and a suitable technique is applied.
2. The results of the calculated hydrodynamic forces match those shown in fig. 1 from a qualitative point of view.
3. Numerical methods for the hydrodynamic of ships are suitable to evaluate various design alternatives at the early stage of the design process, and with that being faster and cheaper than using model tests for the same purpose but the results are still qualitative rather than quantitative so results validation using model testing in a towing tank must be done.

## References

[1] "Report of the Maneuverability Committee", 16th ITTC, Proceedings, Vol. 1, Leningrad (1981).

[2] "Maneuvering in Shallow and Confined Waters", 23rd ITTC, Vol. I (2002).

[3] H. Eda, "Directional Stability and Control of Ships in Restricted Channels", Transaction SNAME (1971).

[4] M. Fujino, "Experimental Studies on Ship Maneuverability in Restricted Waters Part I", International Ship Building Progress, Vol. 15 (168) (1968).

[5] S. Motora and M. Fujino, "Brief Survey of the Maneuverability in Restricted Waters", 11th ITTC, Tokyo (1966).

[6] F. Bet, N. Stuntz, D. Hänel and S. Sharma, "Numerical Simulation of Ship Flow in Restricted Water", 7th International Conference on Numerical Ship Hydrodynamics, Nantes (1999).

[7] M. Gaafary, "Steering of Ships with Emphasis on Restricted Waterways and Two-Way Traffic, State of the Art", Suez Canal University (2000).

[8] "Principles of Naval Architecture", SNAME, Revised Edition (1976).

[9] H.K. Versteeg and W. Malalasekera, An Introduction to Computational Fluid Dynamics – The Finite Vol. Method, Longman Group Ltd. (1995).

[10] J. Ferziger and M. Peric, Computational Methods for Fluid Dynamics, Springer–Verlag Berlin Heidelberg (1999).

[11] V. Bertram, J. Laudan, "Computation of Viscous and Inviscid Flow Around Ship Hulls", Ship Technology Research, Vol. 40 (3), Aug. (1993).

[12] A.J. Chorine, "A Numerical Method for Solving Incompressible Viscous Flow Problems", J. Computational Physics (1967).

[13] Vug, Using MOUSE for Unix, User Manual, Institute of Gas Dynamics, University of Duisburg, Germany (2004).

[14] M.S. Ebieda, "Application of CFD to Some Flow Problems Associated with Inland Water Transportation", M. Sc. Thesis, Alexandria University (2004).

Received May 3, 2007
Accepted May 23, 2007