Mansoura Engineering Journal, (MEJ), Vol. 26, No. 3, September 2001. M. 1

FLOW AROUND SHIP SPOILER

Nabil H. Mostafa
Mechanical Power Department, College of Engineering
Zagazig University, Zagazig, 44519, Egypt.
E-mail: nmostafa@vt.edu

السريان حول الزائدة الجنيحية لسفينة

نبيل حسن مصطفى
قسم هندسه القوى الميكانيكيه. كليه الهندسه
جامعه الزقازيق- الزقازيق ٤٤٥١٩ - مصر

خلاصه

للتحكم فى الحركة الإهتزازية الرأسية والدوارة وسرعة السفينة تم اقتراح نظام على هيئة زائدة جنيحية تم إلصاقها على السطح السفلى الغاطس للسفينة. الزوائد الجنيحية الموجودة فى المقدمة الغاطسة للسفينة مكونة من عدد زوجى من المقاطع مرتبة على الميسرة و الميمنه إبتداء من مركز كتلة السفينة وإلى الأمام. أما الزوائد الجنيحية الخلفية مكونة أيضا من عدد زوجى من المقاطع مرتبة على الميسرة والميمنة وملصقة على الرافدة الخلفية الغاطسة أو اللوح المستعرض فى المؤخرة الغاطسة للسفينة. إن الفقاعات المتكهفة خلف الزوائد الجنيحيية تستقر بواسطة حقن غاز العادم. إن القوى والعزم التى تعمل على السطح السفلى الغاطس للسفينة تعتمد على درجة ميل الزوائد الجنيحية وطولها ووضعها بالنسبة لمركز كتلة السفينة.

ولتحسين فهم العوامل التى تؤثر فى مجال السريان حول هذة الزوائد الجنيحية قمنا بنمذجة مجال السريان فى أوضاع مختلفة وزاويا ميل مختلفة لهذة الزوائد الجنيحية مع حقن تيار من الهواء. واستخدمنا لذلك برنامج حسابى لحل معادلات نافير استوكس فى ثلاث اتجاهات و المسمى (CFDRC, 2000) لنمذجة مجال السريان ثنائي الطور حول الزوائد الجنيحية للسفينة مع إستخدام طريقة المحاكاة للسطح الحر بطريقه البناء القطعى الخطى الذكى للسطح البينى. يتم تمثيل المعادلات الحاكمة على شبكة منشأة باستخدام نظام فرق الإختلاف المضاد للاتجاة. وفى تلك الظروف المختلفة حسبنا شكل الفقاعة ومجال السريان ثنائي الأبعاد حول جسم الزائده الجنيحية واختلافات الضغط على إمتداد مسار المائع لهذة الزائده الجنيحية. كلما ذادت زاويه ميل الزائده الجنيحيه يذيد فرق الضغط بين المنطقه الاماميه و الخلفيه لها. إن الحقن على مسافه واحد L من الزائده الجنيحيه يذيد فرق الضغط بين المنطقه الاماميه و الخلفيه لها عما إذا كان الحقن ملاصق للزائده. فى حاله الزائده الجنيحيه ذو ميل ٤٥ درجة فإن الفقاعه المتكهفه تكون متزنه و ممتده بشكل قدمى طويل. يعتمد شكل الفقاعة على وضع الزوائد الجنيحية وزاوية ميلها. هذة العوامل تساعد فى التحكم فى الحركة الإهتزازية الرأسية والدوارة وسرعة السفينة.

Accepted August 27, 2001

ABSTRACT

To control the roll, pitch motion and speed of the ship, a proposed system of spoilers is mounted at the bottom of the ship. The bow spoilers consist of an even number of sections arranged port and starboard forward of the center of mass of the ship. The stem spoilers consist of an even number of sections arranged port and starboard attached to the ship's transom or transom plate. Injecting exhaust gas stabilizes cavities behind spoilers. The forces and moments acting on the ship's bottom depend on its inclination, spoiler extensions and positions of the spoilers relative to the ship's center of mass.

To improve the understanding of the parameters affecting the flow field around such spoiler, we computed the flow field with different spoiler position, and inclination angle with air flow injection. We used a three-dimensional Navier-Stokes code (CFDRC, 2000) to model the two-phase flow field around a ship spoiler with the Free Surface simulation in Piecewise Linear Interface Construction (PLIC) method. The governing equations are discretized on a structured grid using an upwind difference scheme. For different conditions, we determine the bubbles shape, the two-dimensional flowfield around the spoiler body and the pressure variation on the wake of the spoiler body. Higher spoiler inclination angle gives higher pressure difference between front and back of the spoiler. Spoiler with injection at a distance L from ship spoiler gives higher pressure difference between front and back of the spoiler than injection just beside the spoiler. In spoiler with 45° inclination angle the cavity has high stability with long bubble foot shape. The bubble shape depends on the spoiler position and inclination. These parameters affect the roll, pitch and speed of the ship.

KEYWORDS

Ship, Spoiler, Cavitation, Hydrofoils, Free surface, Two-Phase Flows.

NOMENCLATURE

d_c	the local cell "dimension" (or length- scale)	
F	liquid volume fraction	
L	spoiler angle	m
P	fluid static pressure	N/m^2
P_t	total pressure	N/m^2
Δt	physical time step	second

Δt_c	the maximum time step that can be taken in cell c	second
u,v,w	velocity in x, y, w respectively	m/s
$\vec{v}_c$	the local velocity vector	
V_{cut}	the volume of the cell truncated by the cutting plane	m^3
V_o	the volume of the whole cell	m^3

GREEK LETTERS

ϕ	the volume- averaged quantity	m^3
ρ	density	kg/m^3
σ	the surface tension between the two fluids	N/m
θ	spoiler inclination angle	degree

SUFFIXES

1	the value of the property fluid one (air)
2	the value of the property fluid two (water)

ABBREVIATION

VOF	Volume- Of- Fluid
SLIC	Single Line Interface Construction Method
PLIC	Piecewise Linear Interface Construction Method

I. INTRODUCTION

A system of spoilers is mounted at the bottom of the ship. They can be classified into Bow spoilers and Stern Spoilers as shown in figure 1. Hull-mounted cavitating spoilers have been shown, by the Krylov Shipbuilding Research Institute (Russia), to be an effective means for motion reduction in high-speed displacement-hull vessels. The specific design details of the Russian system are not available for propriety reasons. There results, quoted by Soper, et al., 1998 show that by appropriately choosing the number, extension, and distribution of the spoilers, one can effect an optimal trim of the ship over the whole speed range and a reduction in the ship motion when moving in waves. Roll reduction by a factor of 2 to 5 depending on the sea state. Pitch reduction by a factor of 1.2 to 1.5 depending on the sea state. An increase in the efficiency and hence the speed are by a factor of 10% to 20%. Under "MURI" research project, a theoretical investigation of the effectiveness and design of such a system has been undertaken by Soper, et al., 1998. They used the panel method by developing a two-dimensional quasi-steady numerical local model for the fixed-cavitation region. The computational-fluid-dynamic boundary-element model is based on a distributed-vorticity potential-flow form. The steady-flow streamlines are determined via an iterative approach that converges to a pre-specified level of accuracy. The cavity shape and quasi-steady hydrodynamic forces have been generated over a wide range of forward-speed/cavity-pressure and spoiler-projection-angle parameter values. Owis, et al., 2000, developed a code of Navier stokes equation with cavitation procedure function of local pressure and density to solve the same problem.

There is no unique definition of the interface between two phases flow. The cavitation phenomenon depends on the local pressure and density, Mostafa (2001). The fundamental of ship spoiler problem is different It is depend on air injection into water and it is not a very high speed phenomenon. The basis of the Free Surface simulation is the Volume- Of- Fluid (VOF) method, as published in an early form by (Hirt and

Nichols, 1981), and as recently extended by (Rider, et al., 1995). In upwind scheme with the Single Line Interface Construction (SLIC) Method, (Noh and Woodward, 1976), the fluid surface is assumed to be parallel to the currently selected cell face, with the relative position of fluid two dependent on the flow direction and the upstream or downstream value of liquid volume fraction, F. In the Piecewise Linear Interface Construction (PLIC) with upwind scheme, (Kothe, et al., 1996), the liquid- gas interface is assumed to be planar and allowed to take any orientation within the cell, and will therefore generally have the shape of an arbitrary polygonal facet. So, PLIC is the most accurate method. The Free Surface simulation computes the mixture of two incompressible, immiscible fluids, including the effects of surface tensions. The relative mixture of the two fluids within the problem domain is tracked in terms of a secondary fluid volume fraction, F, which, by definition, ranges between 0 and 1. Thus, the strength of the Free Surface simulation can model the injection of one fluid into the second fluids with arbitrary immiscible fluid- fluid interfaces, which, includes two fluids with very high density ratio, such as air and water.

The objectives of this paper are to model the two phase flow field around a ship spoiler using three-dimensional Navier-Stokes with the Free Surface simulation in PLIC method. The equations are discretized on a structured grid using an upwind difference scheme. For different conditions, we will determine the bubbles shape, the two-dimensional flowfield around the spoiler body and the pressure variation on the wake of the spoiler body. The effect of the spoiler position and inclination on the bubble shape will be studied which affect the roll, pitch and speed of the ship.

II. THEORY BACKGROUND

The characterizing feature of the VOF methodology is that the distribution of the second fluid (e. g. water) in the computational grid is accounted for using a single scalar field variable, F. Flow field and distribution of F is determined by solving the passive transport equation

$$\frac{\partial F}{\partial t}+\nabla\bullet\vec{v}F=0 \qquad (1)$$

This equation must be solved together with the fundamental equations of conservation of mass and momentum in order to achieve computational coupling between the velocity field solution and the liquid distribution. This requires three related actions: compute mixture properties, reconstruct the fluid- fluid interface in each cell and determine the contribution of the secondary fluid flux. The average value of any volume specific quantity, ϕ, in a computational cell can be computed from the value of F in accordance with

$$\phi=F\phi_2+(1-F)\phi_1 \qquad (2)$$

For an intensive quantity, equation 2 can be extended to include the effect of density, ρ :

$$\phi=\left[F\rho_2\phi_2+(1-F)\rho_1\phi_1\right]\ /\rho_{mix} \qquad (3)$$

The location of the "anchor point" in the PLIC scheme, (Kothe, et al., 1996), is determined by finding the infinite cutting plane perpendicular to the unit normal of the infinite plane that truncates the correct liquid volume from the cell, i. e., that satisfies the condition.

$$V_{cut} = F\,V_c \qquad (4)$$

In the PLIC scheme, each cell has a unique surface normal that can be used to compute the surface curvature from cell to cell. This enables the calculation and addition of surface tension forces for the free surfaces. Within each computational cell, the stability limit is given by the so- called Courant Condition:

$$\Delta t_c = \frac{d_c}{\left|\vec{v}_c\right|} \qquad (5)$$

The net normal force acting on the surface is equal to the summation of all the tangential forces due to surface tension:

$$\int \Delta p ds = \int \tau \left|dx\right| \qquad (6)$$

Since the tangential force is equal to

$$\tau = \sigma\,\vec{n} \times \frac{d\,\vec{x}}{\left|d\,\vec{x}\right|} \qquad (7)$$

this leads to

$$\int \Delta p ds = \int \sigma \vec{n} \times d\,\vec{x}, \qquad \vec{n} = \nabla F \qquad (8)$$

The computing process can be implemented to find the fluid-fluid interface at each cell, find the unit normal of the interface, then apply the above integral to determine an effective cell volume force.

III. RESULTS AND DISCUSSION

The transit flow around a ship spoiler is affected by gas injection, spoiler position and spoiler inclination. The flow is computed for different spoiler position and inclination. The ship spoiler structure is shown in figure 2. The structure grids are divided into four 2-D blocks. This uses the multi-block system. Three of them are under water. Two blocks beside the spoiler have grids 85x25, 75x25 grid points and the third one have 160x50 grid points. The grids are clustered near the spoiler to solve the fluid interaction. One 2-D block in Air represents the injection pipe. The length of the grid in physical domain is about 40L length and 11L depth as shown in figures 3. A primary objective of this paper is to study the two-phase flow field around a ship spoiler using three-dimensional Navier-Stokes with the Free Surface simulation in (PLIC) method. The equations are discretized on a structured grid using an upwind difference scheme. There are three sets of results that are computed air injection beside a spoiler with θ=90°, injection at a distance L from the spoiler with θ=90°, and injection at a distance 0.71L from the spoiler with θ=45°. For the above different conditions, the bubbles shape, the two-dimensional flowfield around the spoiler body and the pressure variation on the wake of the spoiler body are determined. The physical time step is taken to be 1×10^{-3} or 1×10^{-4} second for the unsteady flow computations in order to resolve accurately the transients of the cavity formation.

Figure 4 displays the iso-density contours for cavities formation in a time sequence beside ship spoiler. The injection was just beside ship spoiler with spoiler angle (θ) equals 90°. The ship spoiler has length L. The upstream speed (u) equals 5 m/s. The injection pressure of air equals 5000 N/m^2. The physical time step is taken 1×10^{-3} second. It is demonstrated that the cavity formation has three stages. First, a cavity starts to grow at the wake of the spoiler body as shown in figures 4a-c. At the second stage, the cavity starts to split as shown in figures 4d-f. Finally, the splited cavity runs away to collapse at the third stage as shown in figures 4g-h. The cavity is attached to the ship spoiler from the beginning.

Figure 5 represents the total pressure, pressure and velocities (u and v) in the last condition in figure 4. The iso-total pressure contours are approximately similar to the iso-density contours. There is a reverse flow in the horizontal velocity component at the cavities region near to the spoiler body and in the body wake. The maximum vertical velocity component is concentrated around the injection hole. In this case, the maximum/ minimum pressures around upstream/ downstream of the spoiler are about 17173/-18716 N/m^2.

Figure 6 displays the iso-density contours for cavities formation in a time sequence which injected at a distance L from ship spoiler with spoiler angle (θ) equals 90°. The ship spoiler has length L. The upstream speed (u) equals 5 m/s. The injection pressure of air equals 5000 N/m^2. The physical time step is taken $1x10^{-3}$ second. It is demonstrated that the cavity formation has four stages till it split and run away. First, a cavity starts to grow at downstream of the injection hole as shown in figures 6a-e. At the second stage, the cavity starts to grow at upstream of the injection hole as shown in figures 6g-i due to reverse flow. At the third stage, the cavity attaches the spoiler and splits as shown in figures 6j-k. Finally, the splited cavity runs away to collapse at the fourth stage as shown in figures 6l-n.

Figure 7 represents the total pressure, pressure and velocities (u and v) in the last condition in figure 6. The iso-total pressure contours are approximately similar to the iso-density contours. There is a reverse flow in the horizontal velocity component at the cavities region near to the spoiler body and in the body wake. The maximum vertical velocity component is concentrated around the injection hole. In this case, the maximum/ minimum pressures around upstream/ downstream the spoiler are about 25090/-16378 N/m^2.

Figure 8 displays the iso-density contours for cavities formation in a time sequence which injected at a distance 0.71L from ship spoiler with spoiler angle (θ) equals 45°. The ship spoiler has length L. The upstream speed (u) equals 5 m/s. The injection pressure of air equals 5000 N/m^2. The physical time step is taken $1x10^{-4}$ second. It is demonstrated that the cavity formation has three stages. First, a cavity starts to grow at the wake of the spoiler body as shown in figures 8a-d. At the second stage, the cavity starts to split as shown in figures 8e-f. Finally, the splited cavity starts to run away to collapse at the third stage as shown in figures 4g. The cavity does not completely attach the ship spoiler. The cavity has higher stability with long bubble foot shape.

Figure 9 represents the total pressure, pressure and velocities (u and v) in the condition in figure 8-f. The iso-total pressure contours at the spoiler downstream are approximately similar to the iso-density contours. There is a reverse flow in the horizontal velocity component at the cavities region near to the ship body. The maximum vertical velocity component is concentrated around the injection hole. In this case, the maximum/ minimum pressures around upstream/ downstream the spoiler are about 19178.6 / -13916.1 N/m^2.

IV. SUMMARY AND CONCLUSIONS

The transit flow around ship spoiler with injection of exhaust gases or air forming cavity underwater vehicles is affected by spoiler position and its inclination angle. This ship spoiler has a strong wake effect with gas injection, which can control the ship oscillation.

It is demonstrated that the cavity formation has mainly three stages. First, a cavity starts to grow at the wake of the spoiler. At the second stage, the cavity starts to split. At the third stage, the splited cavity runs away to collapse. Finally, the cavity is attached to the ship spoiler due to reverse flow.

There is a reverse flow in the horizontal velocity component at the cavities region near to the ship body. The maximum vertical velocity component is concentrated around the injection hole. Higher spoiler inclination angle gives higher-pressure difference between front and back of the spoiler. Spoiler with injection at a distance L from ship spoiler gives higher-pressure difference between front and back of the spoiler than injection just beside the spoiler. In spoiler with 45° inclination angle the cavity has higher stability with long bubble foot shape. Thus, changing inclination spoiler angle and position relative to injection angle will gives different forces to control the roll, pitch motion and speed of the ship.

The iso-total pressure contours at the spoiler downstream are approximately similar to the iso-density contours which indicated that modeling the two phase flow field around a ship spoiler using three-dimensional Navier-Stokes with the Free Surface simulation in Piecewise Linear Interface Construction (PLIC) method is successful. In future work, it is aimed to study this work experimentally.

V. REFERENCES

CFDRC, 2000, "CFD-ACE+ Theory and Users' Manuals Ver.6.4" *CFD Research Corporation*, Huntisville, Al., USA.

Hirt and Nichols, "VOF method for the Dynamics of free boundaries," *J. Computational Physics*. Vol. 39, 201 (1981).

Kothe, D.B., Rider, W.J, Mosso, S.J., and Brock, J.S.,, 1996, "Volume tracking of interfaces having surface tension in two and three dimensions," AIAA Paper, 96-0859.

Mostafa, N. H., 2001 "Computed Transient Supercavitating Flow Over a Projectile"*Mansoura Engineering Journal, MEJ*, Vol.26 No.2, PP. M79-M91.

Noh, W.F. and Woodward, P.R., 1976, "SLIC (simple line interface method)," In A.I. van de Vooren and P.J. Zandbergen, editers, *Lecture Notes in Physic,*. Vol 59, pp. 330-340.

Soper, R. R., Nayfeh, A. H. and Mook, D. T., 1998, "Control of Rolling Ships By Hull-Mounted Spoilers" *Fourth Semi-Annual Meeting- MURI, Nonlinear Active control of Dynamical Systems*. Blacksburg, VA.

Rider and Kothe, "Stretching and Tearing Interface Tracking Methods, 1995, " AIAA Papper, 95-1717

Owis, F., Nayfeh, A. H., and Mook, D. T., 2000 "Control of Roll Motion Using a System of Spoilers" *Seventh Semi-Annual Meeting– MURI- Nonlinear Active Control of Dynamical Systems*, Blacksburg, VA, USA.

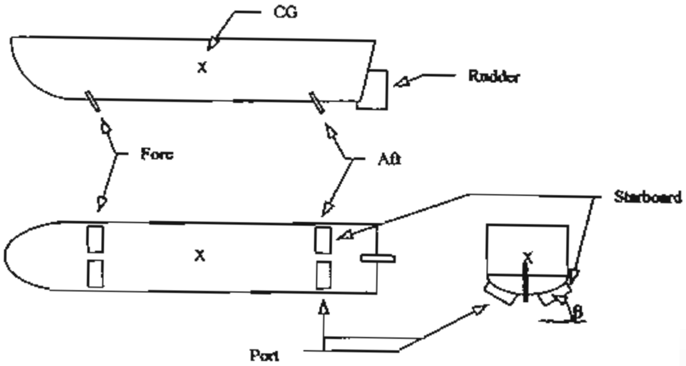

Fig. (1) example of Ship Spoiler Locations, Soper, et al., 1998.

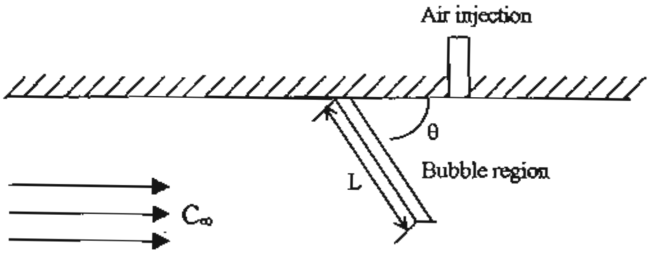

Fig. (2) schematic diagram of the Ship Spoiler.

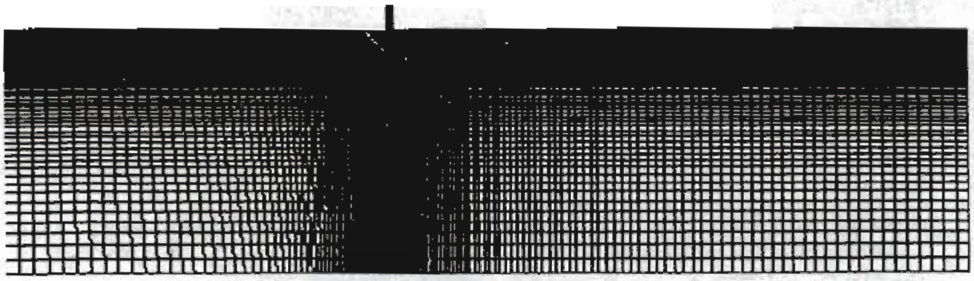

Fig. (3) structured grid of Ship Spoiler.

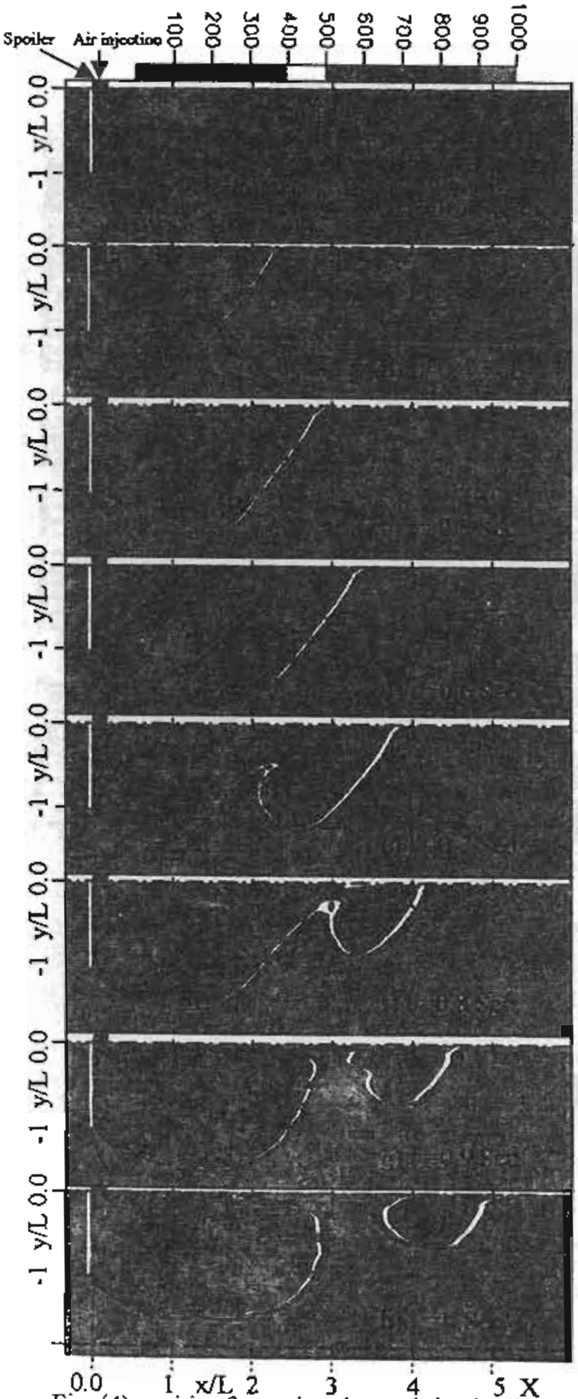

Fig. (4) cavities formation due to injection beside spoiler with θ=90°.

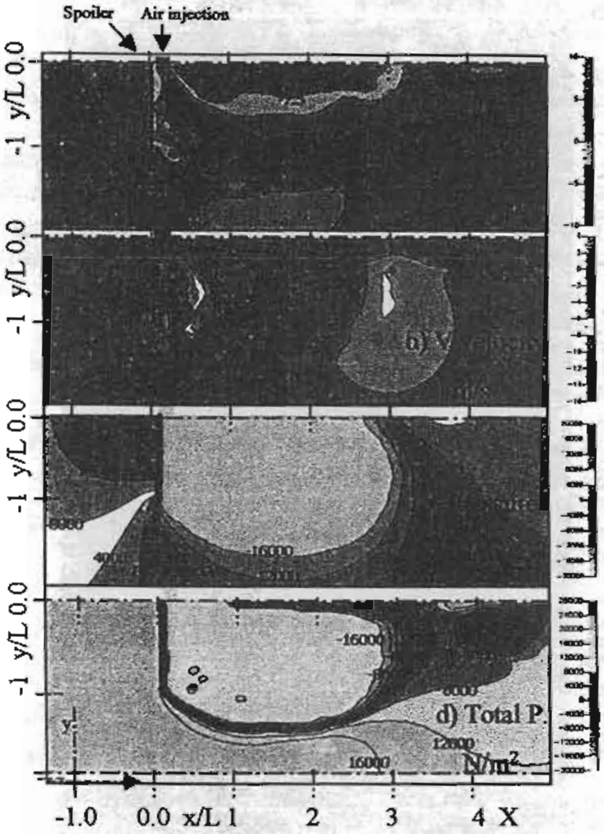

Fig. (5) flow condition around ship spoiler at a condition of injection beside the spoiler with θ =90° and t= 1sec.

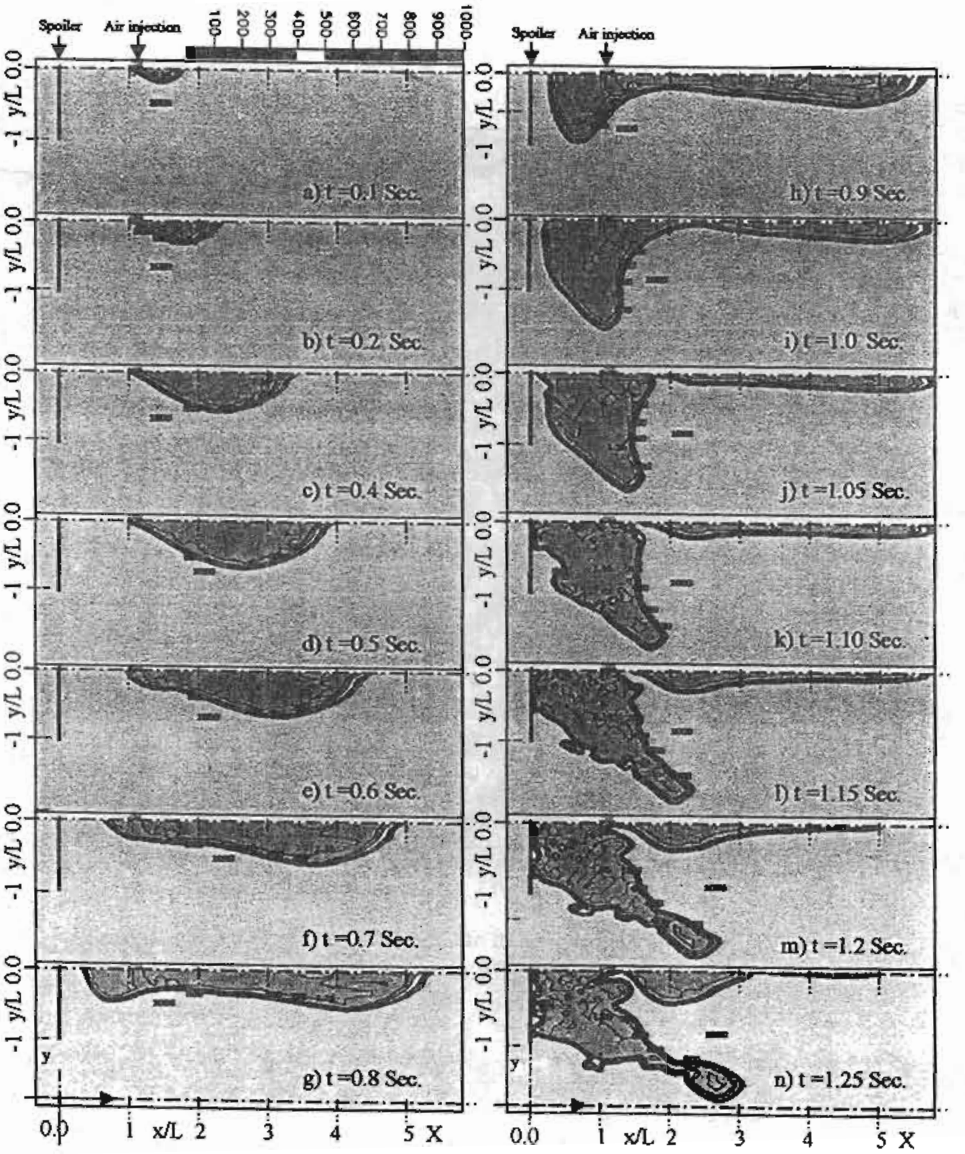

Fig. (6) cavities formation due to injection at a distance L from the spoiler with θ=90°.

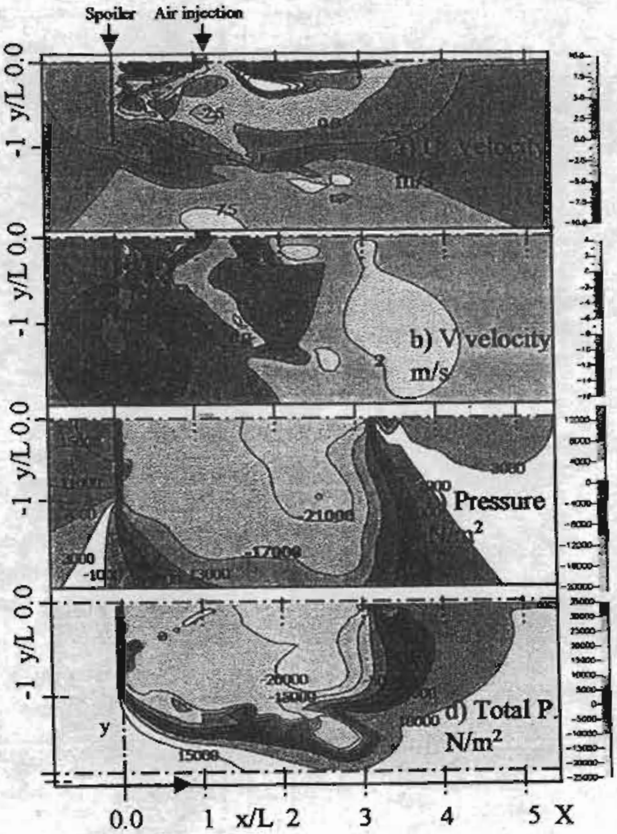

Fig. (7) flow condition around ship spoiler at a condition of injection at a distance L from the spoiler with θ =90° and t= 1.25 sec.

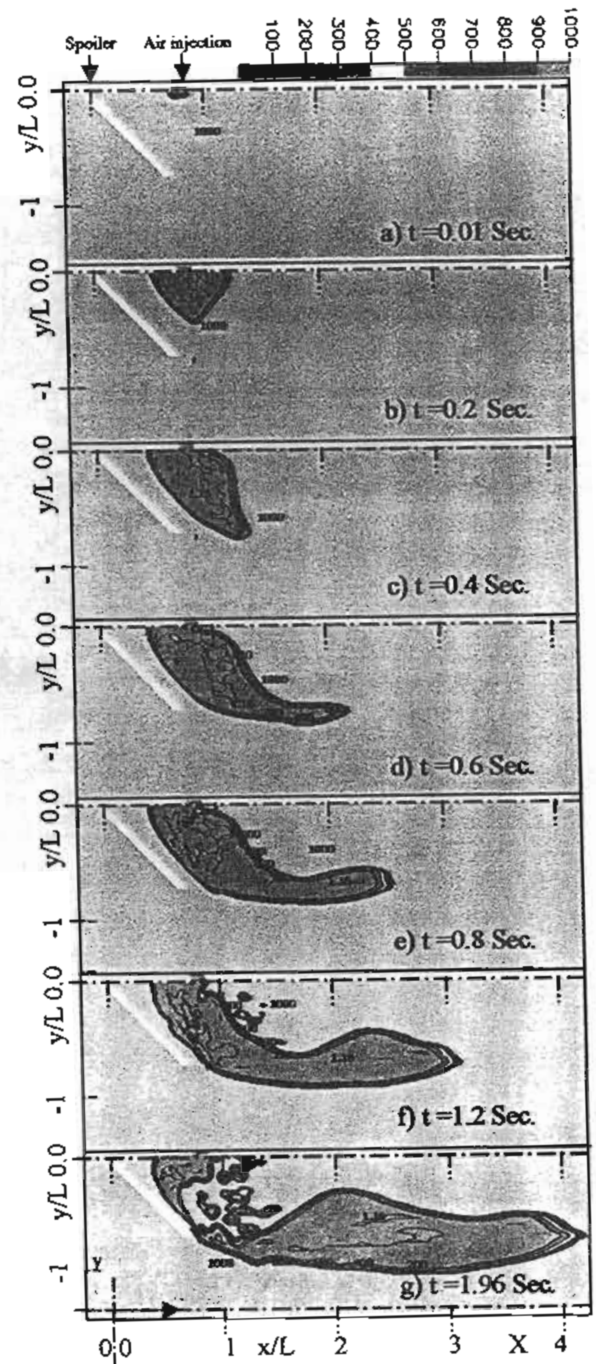

Fig. (8) cavities formation due to injection at a distance 0.71L from the spoiler with θ=45°.

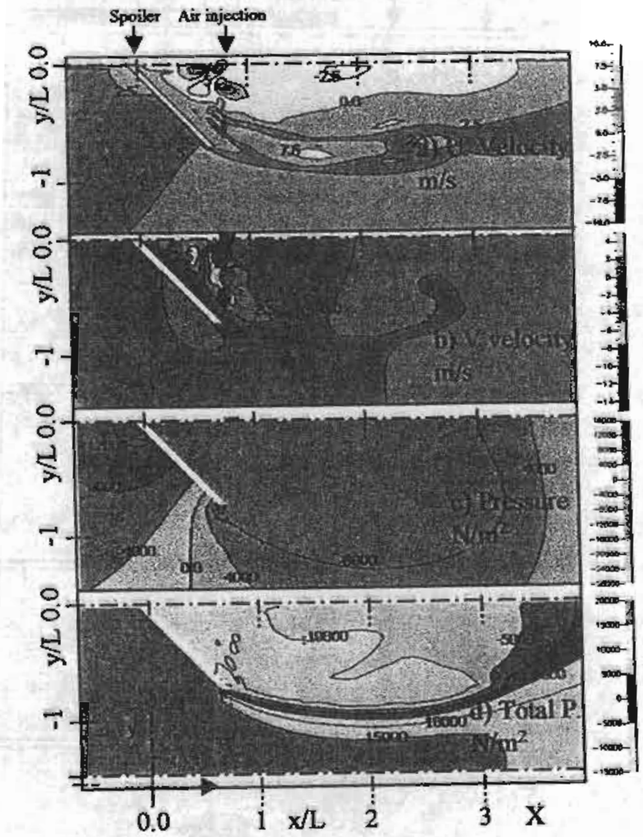

Fig. (9) flow condition around ship spoiler at a condition of injection at a distance 0.71L from the spoiler with θ =45° and t= 1.2 sec.