Preliminary estimates of flow patterns around hulls with yaw angle

Molyneux, W. D.

Publisher's version / Version de l'éditeur:
https://doi.org/10.4224/8895468
Technical Report (National Research Council of Canada. Institute for Ocean Technology); no. TR-2005-03, 2005

DOCUMENTATION PAGE

REPORT NUMBER	NRC REPORT NUMBER	DATE
TR-2005-03		February 2005

REPORT SECURITY CLASSIFICATION	DISTRIBUTION
Unclassified	Unlimited

TITLE

PRELIMINARY ESTIMATES OF FLOW PATTERNS AROUND HULLS WITH YAW ANGLE

AUTHOR(S)

David Molyneux

CORPORATE AUTHOR(S)/PERFORMING AGENCY(S)

Institute for Ocean Technology, National Research Council, St. John’s, NL

PUBLICATION

SPONSORING AGENCY(S)

Institute for Ocean Technology, National Research Council, St. John’s, NL

IOT PROJECT NUMBER	NRC FILE NUMBER
2072	

KEY WORDS	PAGES	FIGS.	TABLES
Escort tugs, Laser Doppler Anemometry	iv, 21	19	1

SUMMARY

Escort tugs use the combination of yaw angle and azimuthing thrusters to generate the hydrodynamic forces that are used to control a disabled tanker. The yaw angles used in escort operations (35-55 degrees) are considered to be extreme ‘off-design’ conditions in normal naval architecture terms but in the case of an escort tug they are expected operating conditions.

ADDRESS

National Research Council
Institute for Ocean Technology
Arctic Avenue, P. O. Box 12093
St. John's, NL A1B 3T5
Tel.: (709) 772-5185, Fax: (709) 772-2462

National Research Council Canada
Institute for Ocean Technology

Conseil national de recherches Canada
Institut des technologies océaniques

PRELIMINARY ESTIMATES OF FLOW PATTERNS AROUND HULLS WITH YAW ANGLE

TR-2005-03

David Molyneux

February 2005

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

PRELIMINARY ESTIMATES OF FLOW PATTERNS AROUND HULLS WITH YAW ANGLE

David Molyneux

Figure 1, Escort tug in operation, viewed from tanker.
(Photograph courtesy of Crowley Maritime Corp.)

1.0 INTRODUCTION

Escort tugs use the combination of yaw angle and azimuthing thrusters to generate the hydrodynamic forces that are used to control a disabled tanker. The yaw angles used in escort operations (35-55 degrees) are considered to be extreme ‘off-design’ conditions in normal naval architecture terms but in the case of an escort tug they are expected operating conditions. An escort tug in a typical operating condition is shown in Figure 1.

NRC’s Institute for Ocean Technology and Memorial University of Newfoundland are collaborating on a project to understand the flow around a typical escort tug and use this information to improve its hydrodynamic performance. The initial proposal is to carry out flow measurements at critical locations around the tug, using Particle Image Velocimetry (PIV) and to predict the flow around the hull using Computational Fluid Dynamics (CFD). The final phase is to validate the predictions against the experiment data. This work forms the basis for the author’s Ph. D. studies.

An important step in planning the PIV experiments was to locate the critical locations for making flow measurements. There is very little data available on the flow patterns and resulting hydrodynamic forces for ship hulls at yaw angles beyond those used in typical manoeuvring conditions. Experiment data for flow around the Series 60, C_B=0.60 hull with two different yaw angles are available. Data for a 1:40 scale model at 10 degrees of yaw was obtained from pitot tubes (Longo & Stern, 1996) and data for a 1:100 scale model at 35 degrees of yaw was obtained using Laser Doppler Anemometry (Di Felice and Mauro, 1999). Several CFD simulations have been validated against the data at 10 degrees of yaw (Alessandri & Delhommeau, 1996; Cura Hochbaum, 1996; Tahara et al, 2002).

The Series 60 experiment data shows that well-defined flow structures occurred on the downstream side of the hull, between the bilge radius and the waterline, but the hull shape is quite different from a typical escort tug and the data with the most detail is for a yaw angle outside the primary interest for escort tugs. As a result, it is necessary to estimate flow patterns from sources other than model experiments. CFD simulations are the most obvious choice, but carrying out full 3-dimensional simulations will be extremely time consuming, and predictions are needed prior to carrying out the first round of experiments, scheduled for February 2005.

Given the short time available to provide the estimates, it was decided to attempt a simplified model of the flow conditions, and compare the results of the simplified simulations against the Series 60 data from experiments. If the simulations showed similar trends to the experiment data, then they should be accurate enough for the planning stages of the PIV experiments.

A flow component across the centreline of the hull is required to simulate data close to the Series 60 model data. In the pure cross-flow condition, the hull is at a yaw angle of 90 degrees. This condition can be simulated using a 2-dimensional flow solver, but the flow will be unrealistic for smaller yaw angles, when the longitudinal flow component becomes strong, relative to the total flow. A more realistic model will be a prismatic section, where the hull geometry does not change, but three-dimensional flow can be simulated. Since the hull shape was simplified to a prismatic geometry, it was not thought to be important to consider the effect of the free surface at this point.

The other 2-dimensional plane considered was a plane parallel to the waterline of the hull. This approach cannot consider a free surface, but this limitation was thought to be less important than predicting the large-scale flow patterns created downstream from the hull.

2.0 SERIES 60, BLOCK COEFFICIENT 0.6

The principal particulars for the Series 60, C_B 0.6 hull are given in Table 1.

Length, BP, m	121.92
Beam, m	16.256
Draft, m	6.502
C_B	0.6
C_M	0.977

Table 1, Principal Dimensions for Series 60, C_B 0.6

2.1 Yaw Angle 10 Degrees, Iowa Data

An extensive flow survey around a model of the Series 60, C_B=0.6 hull was made using five-hole pitot tubes for zero yaw angle (Toda et al., 1992, Longo et al., 1993) and with a 10 degree yaw angle (Longo and Stern, 1996). The experiments were carried out to determine the influence of waves created by a surface-piercing hull on its wake and boundary layer and to provide detailed measurements of the flow field for validating CFD methods. Mean velocity and pressure measurements were made for two Froude numbers (0.160 and 0.316) at multiple sections from the bow to the stern, and into the near wake at the stern. The two speeds were chosen to give the effects of waves on the flow.

A Cartesian measurement grid was used with the origin at the intersection of the forward perpendicular and the static waterline. The x-axis was positive towards the stern, the y-axis was positive to starboard and the z-axis was positive upwards. Velocities in the x, y and z direction were referred to as u, v and w respectively. Results were non-dimensionalized using model length (between perpendiculars) L, carriage velocity U and fluid density ρ. Two models were tested, at scales of 1:40 and 1:66.7.

Data from the experiments was presented as total head and axial (u) velocity contours, crossplane (v, w) velocities and pressures and axial vorticity contours. The y-z planes were at locations of 0, 0.2, 0.6, 0.9 and 1.1L for each of the two Froude numbers. Wave profiles at the hull surface, contours of wave elevation and wave slope were also given. Pressure measurements with the pitot tubes were made at between 200 and 350 data points per section.

Wave profiles at the hull were measured at more locations than the pressures. Wave elevation was measured using an array of wave probes fixed in the tank axis system, referred to in the paper as global elevations. Wave elevation close to the model was measured from a moving wave probe on the towing carriage, and this was referred to as local elevation. For the zero yaw case, the results presented were based on a combination of approximately 4000 carriage runs.

The work at 1:40 scale was expanded to include steady yaw angles up to 10 degrees (Longo & Stern, 1996). Forces and moments were measured for yaw angles from zero to

10 degrees at intervals of 2.5 degrees. Wave profiles at the hull surface and wave elevations were measured at yaw angles of zero, 5 and 10 degrees. Detailed pressure measurements were made at 10 degrees only. The methods used were essentially similar to the ones discussed above, with some minor changes. The biggest difference was that the range of the local wave surface measurements had to be extended, since the projected beam of the ship was wider, due to the yaw angle. Also, measurements were required on both sides of the hull, since the flow was no longer symmetric about the centerline.

The more complex flow around the yawed hull required a more precise spatial definition than the symmetric flow, and so data density for measurements was increased to between 800 and 1500 points per y-z plane.

The results of the experiments for the zero yaw and the yawed case are available from the web site of the Computation Ship Hydrodynamics Laboratory at the University of Iowa (http://www.iihr.uiowa.edu/~towtank/series60bare.htm).

2.2 Yaw Angle 35 Degrees, INSEAN Data

Di Felice & Mauro (1999) measured the flow around a double model of a Series 60 C_B=0.6 hull at a scale of 1:100 in a large cavitation tunnel using Laser Doppler Velocimetry (LDV). In this case, the model hull was symmetrical about the design waterline and the free surface effects were ignored. The yaw angle used was 35 degrees, which was much higher than the 10 degrees used by Longo and Stern (1996) for the same hull form.

The particular LDV used a two-component backscatter method, with estimated velocity resolutions within +/-1%. The flow was seeded with titanium dioxide particles, with a diameter of 1 μm. Measurements were made at two sections, 0.5L and 0.9L. The data density was 600 points for the first section and 800 points for the second. The measurements were made in the axis system of the tunnel, rather than normal to the centerline of the model. The resulting measurement planes were not at a constant location in ship axes, which was the convention used by Toda et al. (1992), but were normal to the direction of the undisturbed flow. This was accepted in order to use the mechanized system for locating the measurement point within the flow, which was fixed in an axis system with the y and z-axes normal to the centerline of the cavitation tunnel.

Results of the experiments were presented as contours of cross flow velocities, vertical and transversal component standard deviation, Reynolds stresses, vorticity and vertical and transverse component skewness for the downstream side of the hull. The results showed distinct vortices at each plane. Di Felice and Mauro state that the advantage of the LDV method was the ability to measure quantities such as turbulence intensity and Reynolds stresses, as well as detailed measurements of the flow in the cross planes. All these results combined to give information on viscous and turbulent aspects of detached flow generated by the yawed hull.

3.0 SIMULATION OF FLOW AROUND A PRISMATIC MODEL USING FLUENT

For a ship with a large amount of parallel middle body, it is possible that the flow around the sections with constant area can be modelled using a constant section prismatic approximation to the geometry. Reducing the hull model to a constant section can simplify the creation of the mesh, since only one face needs to have a detailed mesh, and the third dimension is created from uniform elements in the third dimension. Another simplification was to ignore the effect of the free surface. This was done to simplify the generation of the mesh, and because the data from Di Felice and Mauro was for a double model, where free surface was ignored.

To model the case of a ship with yawed flow, the prismatic section was based on the midship section of the Series 60 C_B=0.6 hull, with sufficient distance upstream and downstream to ensure that the boundaries were not creating an unrealistic effect on the flow around the ship section. The angle between the incoming flow and the hull (yaw angle) was set by adjusting the boundary conditions, so that the velocity at the inlet planes had two components. The pressure outlet planes were set so that the backflow pressure was also in the same direction. The advantage of this approach was that one mesh could be used for all the yaw angles. A summary of the overall mesh dimensions is given in Table 1.

The geometry of the midship section is the same as the 1:40 scale model described by Longo and Stearn (1996).

Element	**X, m**	**Y, m**	**Z, m**
Hull	0.0/2.0	+/- 0.203	0.0/0.163
Inner mesh	0.0/2.0	+/- 0.609	+/-0.163
Outer mesh	0.0/2.0	+0.163/-0.652	-0.163/2.233
Velocity magnitude	1.728 m/s		
	X component	**Y component**	**Z component**
10 degree yaw	0.985	0.174	0.0
35 degree yaw	0.819	0.574	0.0

Table 1, Summary of mesh geometry and velocity components

The mesh was created with *GAMBIT* using two zones. One zone, far away from the hull, consisted of a structured hexahedral mesh with sides of 0.05m. Close to the hull, an unstructured mesh was used of wedge shaped hexahedral elements. The total mesh, consisting of 31,360 elements in the coarse zone and 78,320 elements close to the hull is shown in Figure 2 and a close up of the mesh close to the hull is shown in Figure 3. The velocity contours and vectors were taken at a plane in the mesh, with a uniform z dimension of z =1.5m for the 10 degree yaw, and z = 1.75 m for the 35 degree yaw.

These planes were found to approximate uniform flow over the prismatic section, with no influence caused by the proximity of the boundaries.

All solutions were obtained for a κ−ω turbulence model with the default parameters in *FLUENT*. A boundary layer was used consisting of 3 layers of cells, with the initial layer 0.001 m thick, with a growth factor of 1.02. Turbulence intensity and turbulent viscosity ratios were set at 1% and 1 respectively. Convergence limit was set to 10^{-3} (default values) for all parameters. All solutions converged within these limits.

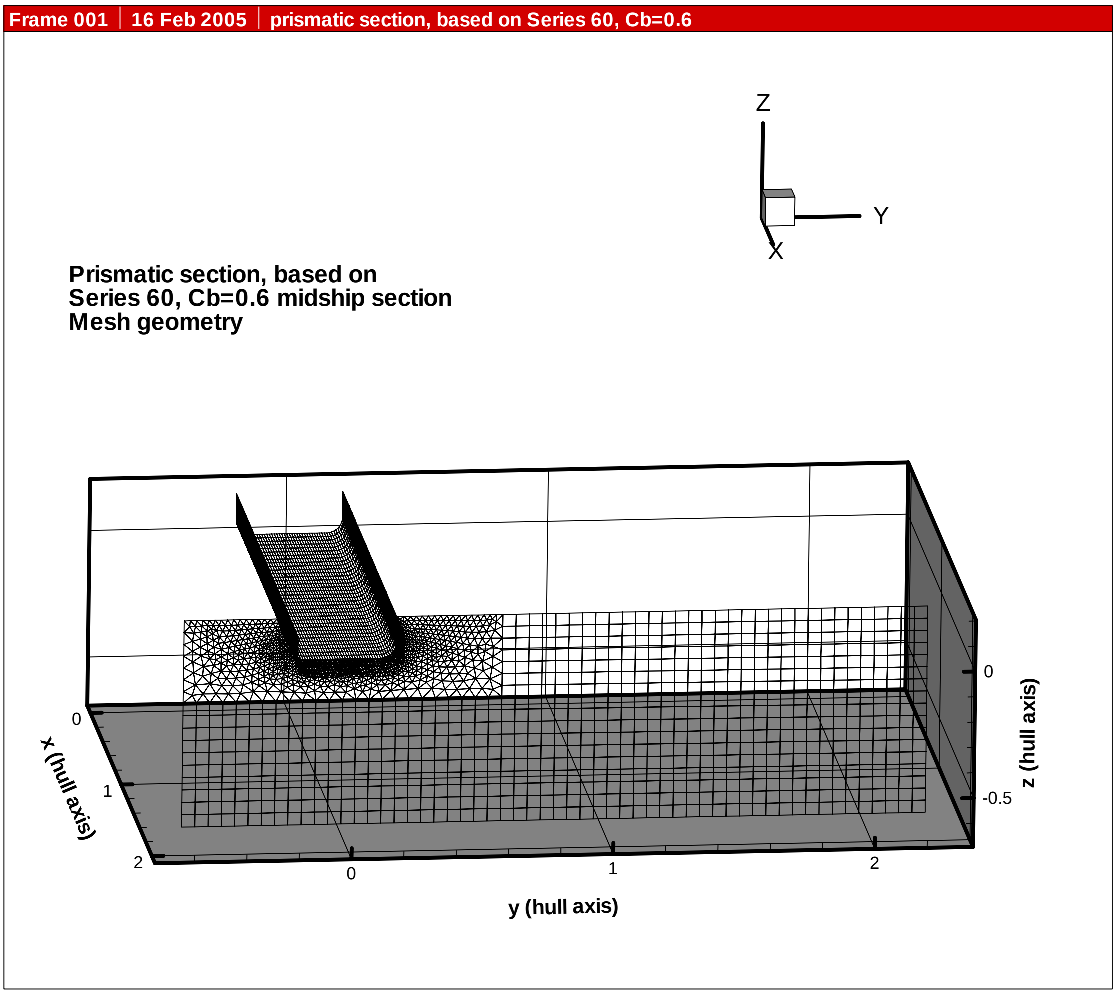

Figure 2, Mesh used for Prismatic simulations of Series 60 C_B=0.6 hull

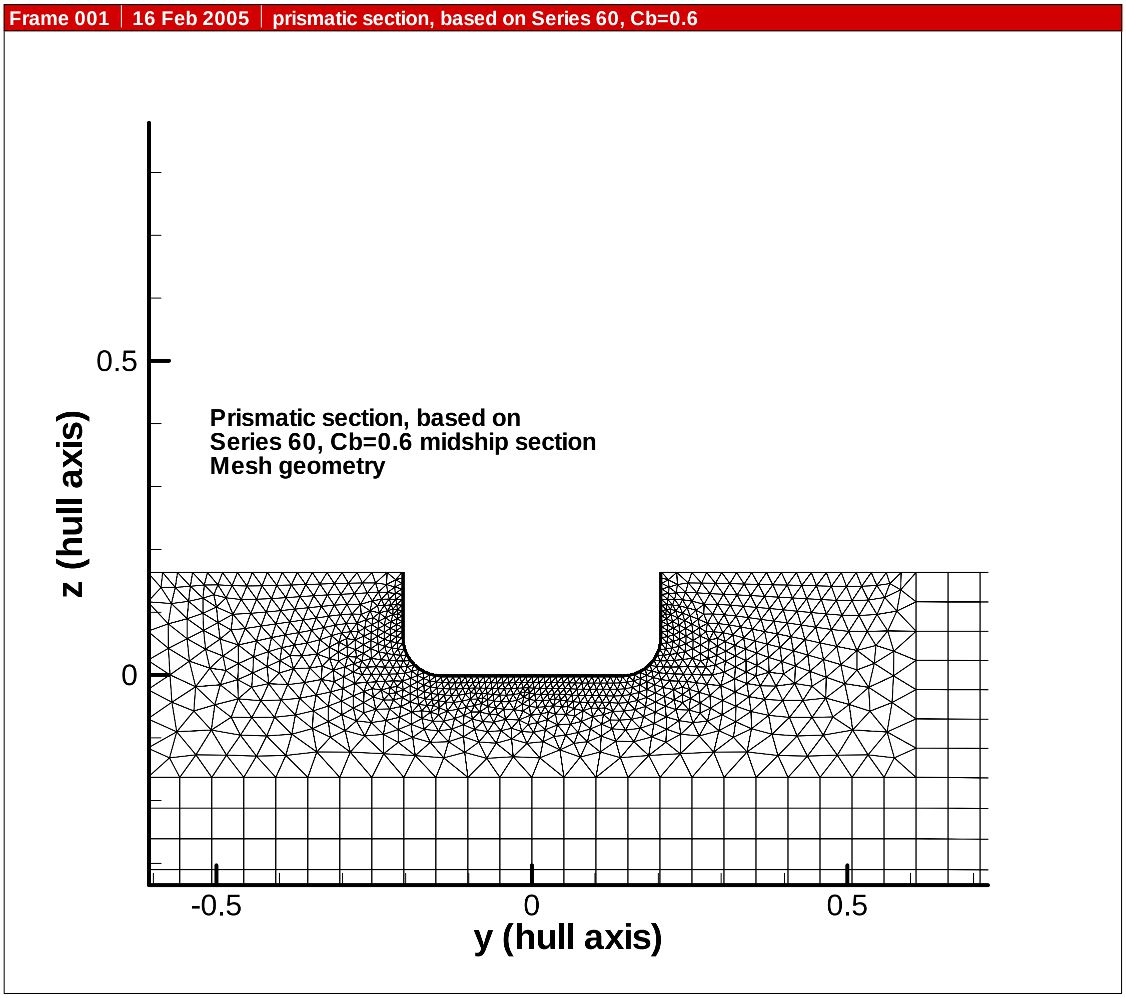

Figure 3, Close-up view of mesh around hull

The axis system for the CFD simulations follows the same system as that used for the experiments, with x along the centreline of the hull, y in the transverse direction and z in the vertical direction. In order to be consistent with the results of the experiments, the CFD simulations were non-dimensionalized as part of the post-processing. Hull geometry was normalized by nominal model length (3.048m). The flow components were normalized by dividing by the free-stream velocity (1.728 m/s). Results are presented as contours of velocity in the x direction (u), and vectors showing the v and w flow components (y and z directions respectively).

4.0 DISCUSSION OF RESULTS

4.1 Yaw Angle 10 Degrees

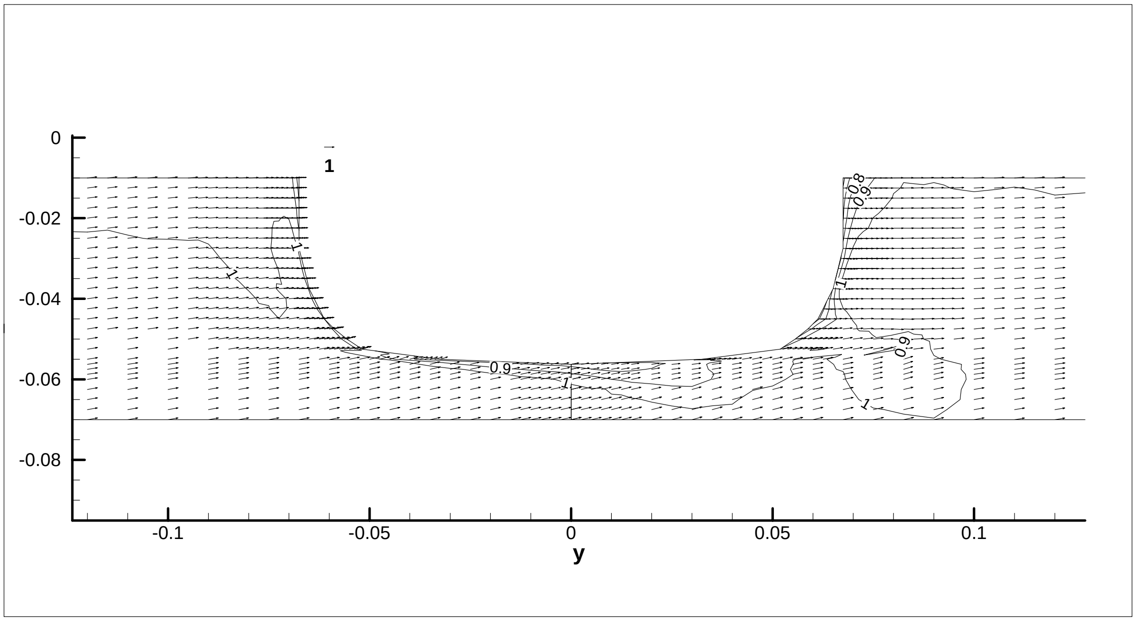

Figure 4a), Measured velocity components around Series 60, Cb=0.6 hull, at 60% L aft of forward end of waterline

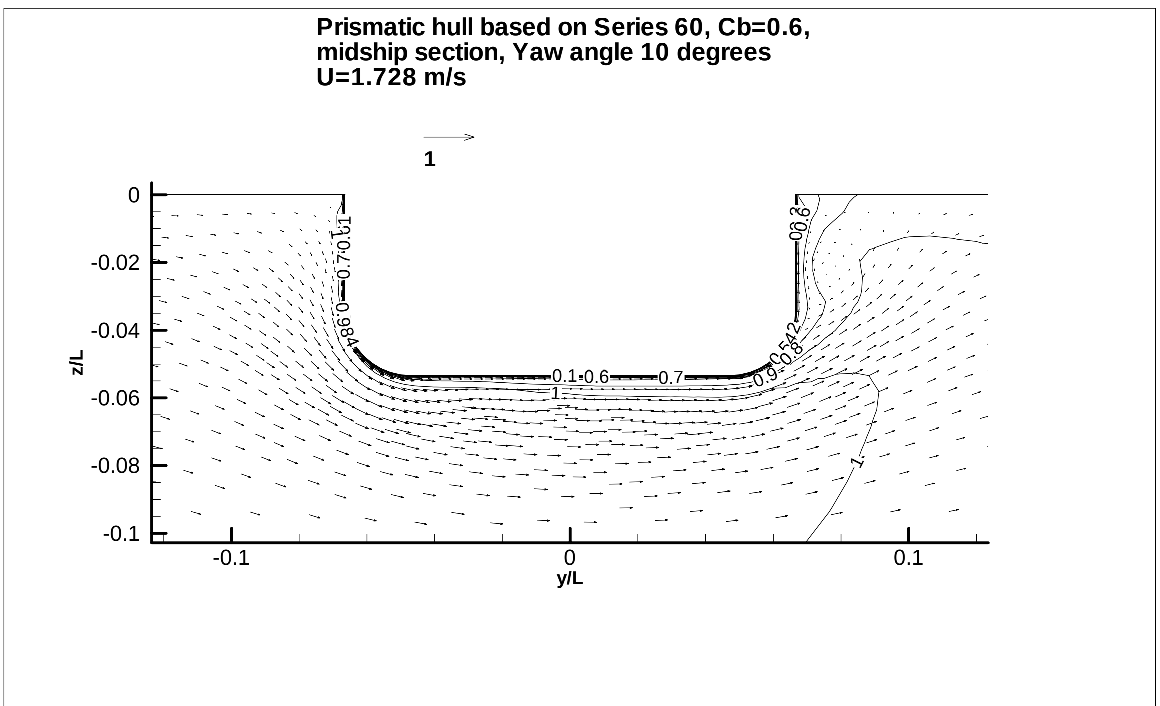

Figure 4b), Fluent predictions for velocity components around prismatic hull with 10 degrees of yaw

The measured experiment results (Longo and Stern, 1996) for a yaw angle of 10 degrees at L=0.6 (which was the closest measured values to the maximum section) are compared with the predicted values for a constant section prism in Figures 4a) and 4b). The experiment results are given in the upper part of the figure and the predictions are given in the lower part.

Overall, for the cases compared, the flow patterns predicted by *FLUENT* are in reasonable agreement with the patterns observed in the experiments. Key features of the flow that are predicted are:

- Low-speed flow on downstream side of hull (u component)
- High-speed flow at upstream bilge radius (v component)
- Zero flow on the downstream side of hull (v component)
- Low-medium speed flow across the bottom of the hull (v component)
- Zero flow on the upstream side, close to the waterline (v component)
- Negative flow on upstream bilge and positive flow on downstream bilge (w component)

There are some areas where the flow is not particularly well predicted and these are as follows:

- v component predictions are much more symmetrical about centreline of hull than experiments.
- Extent of low w components on the upstream side of the hull is under predicted.
- Extent of high w components on the downstream side of the hull is over predicted.

4.2 Yaw Angle 35 Degrees

There is less data available in the open literature for the case with 35 degrees of yaw. Mean cross flow vectors measured at midships are given in Figure 5a) (Di Felice and Mauro, 1999). Also given in the paper are figures showing parameters related to the degree of turbulence in the flow, such as distributions of Reynolds stresses, vorticity and skew, but the basic data was not available at the time of writing.

FLUENT simulations of the velocity vectors on the downstream side of the prismatic hull at a yaw angle of 35 degrees are given in Figure 5b). Comparing the velocity vectors from Figures 5a) and 5b) shows that the closed flow pattern on the downstream side of the hull is well predicted by *FLUENT*. Both figures show the flow at the surface (or the mid-point in the case of the double body) flowing upstream, back towards the centreline of the hull and the flow close to the hull going from the waterline towards the keel. The centre of the vortex is located at about the mid-depth of the hull, and about the same distance downstream from the edge of the hull (given that the edge of the hull infigure 5a) is approximately at the edge of the measured area. Examination of the experiment data and the simulations shows that this vortex was not present when the yaw angle was 10 degrees.

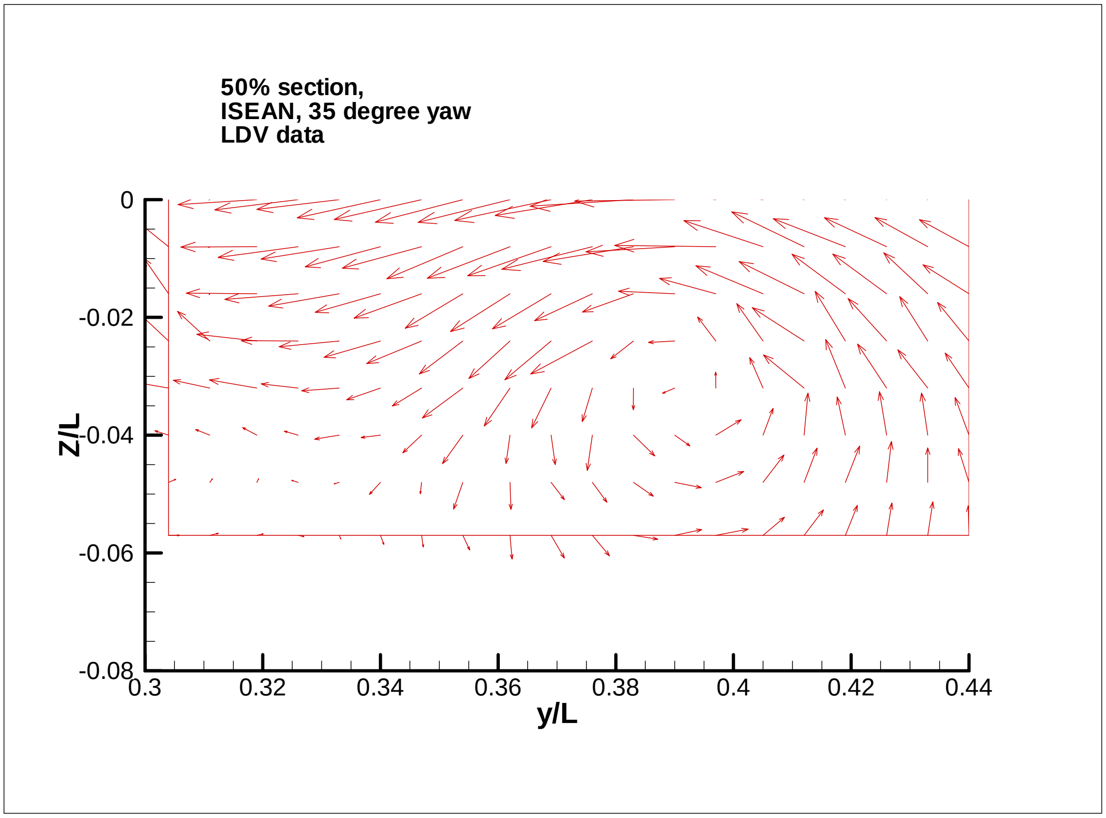

Figure 5a), Cross flow vectors measured with LDV (Di Felice and Mauro, 1999)

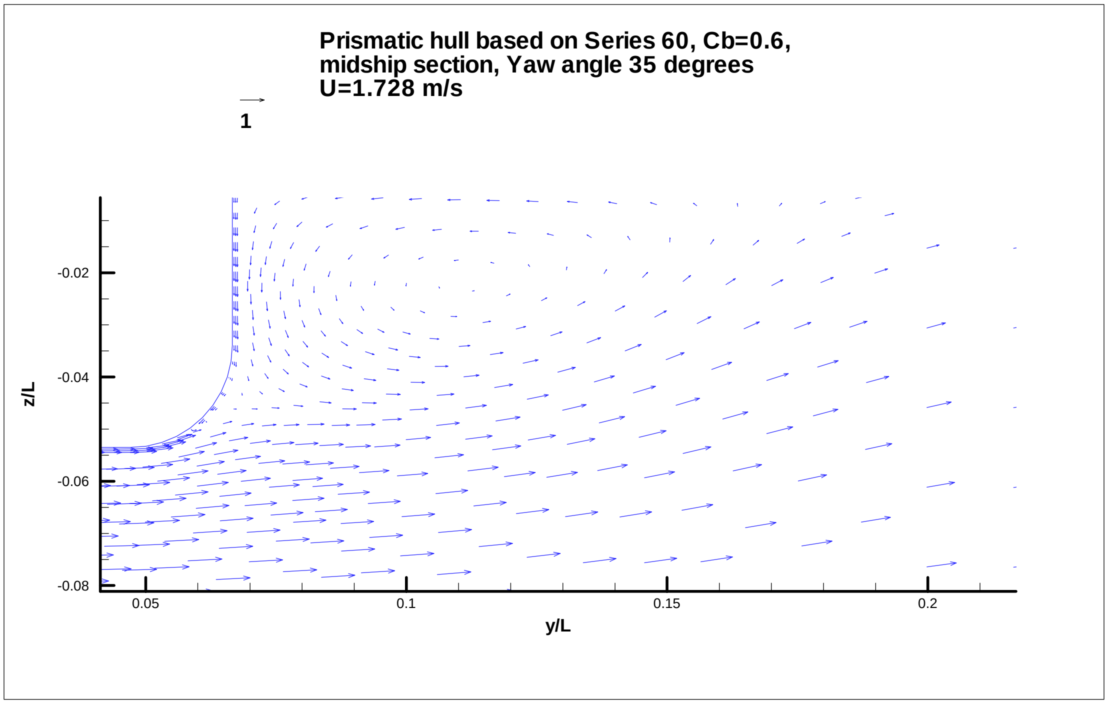

Figure 5b), Cross flow vectors predicted by FLUENT

The velocity contours predicted using *FLUENT* for u, and vectors of v and w at a yaw angle of 35 degrees are shown in Figure 6. This figure shows some flow characteristics that are more representative of the experiment data at 10 degrees than the simulations for the same condition. Figure 6 shows a more asymmetrical flow than for the 10 degree case (Figures 4a) and 4b), and also includes a zone of low speed flow across the bottom of the hull. It also shows less intense high speed and low speed w velocity components, which is a more realistic representation of the experiment results. However, it also shows two regions of high-speed w velocity component on the downstream side of the hull, one of which is separated from the hull.

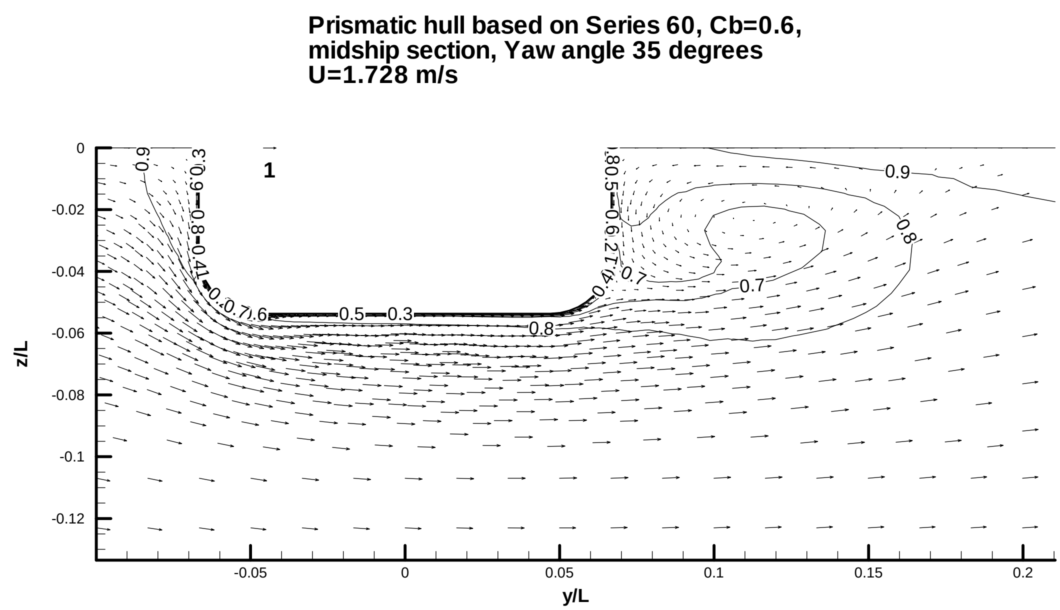

Figure 6, FLUENT prediction of velocity components, 35 degree yaw

4.3 Extreme Yaw Angle, 90 degrees

The most extreme yaw angle is 90 degrees and 2-dimensional model can be used to simulate this flow direction. The mesh used for the simulations is given in Figure 7. Figure 8 shows results plotted as flow vectors of the simulation, with an undisturbed flow speed of 0.3 m/s. This flow speed was the component normal to the model centreline for the speeds used in the Iowa experiments. The main difference in the results is that the closed flow vortex extends further down stream than the case at 35 degrees. In this case the low speed flow on the downstream side of the model extends to the boundary. Although there is no experiment data for comparison with this condition, the result seems to be a reasonable extrapolation of the 35 degree yaw angle case.

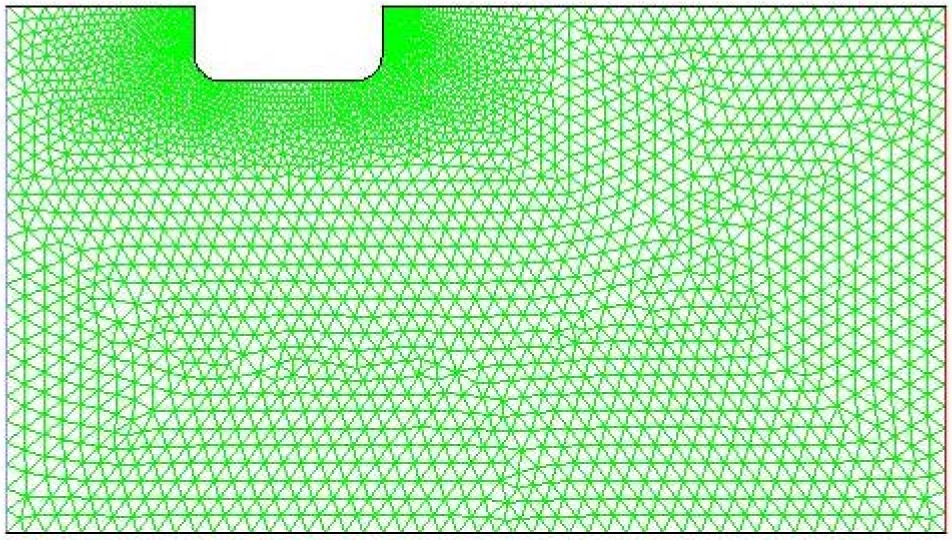

Figure 7, Mesh for 2-dimensional model of Series 60, C_B=0.6 midship section

This case, which is so extreme for conventional ships that it is unlikely to be encountered except at very low speeds during docking, is within the operational envelope of an escort tug. It corresponds to the case of a tug in maximum braking condition, where the tug is directly behind the tanker, with the tug's heading normal to the direction of motion of the tanker.

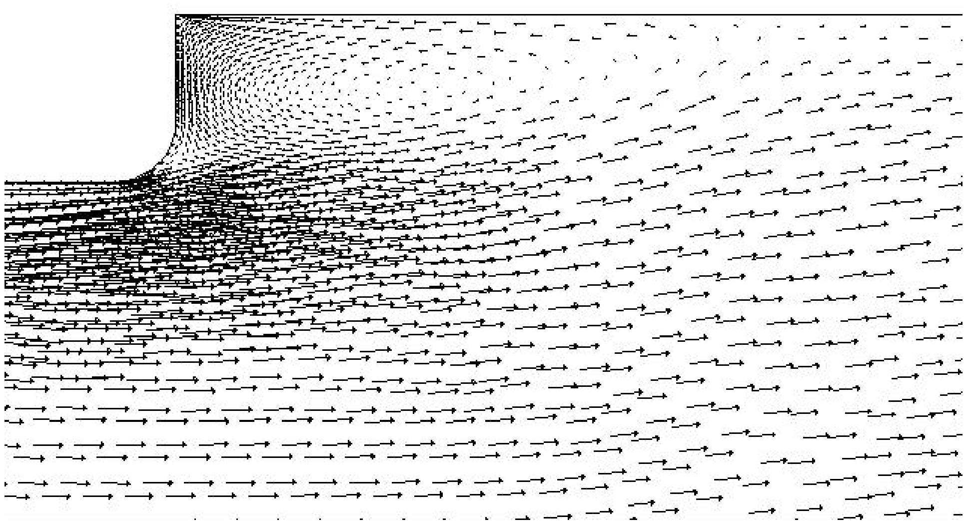

Figure 8, FLUENT predictions of velocity vectors on downstream side of Series 60, C_B=0.6 midship section, 90 degree yaw angle (2-dimensional flow).

4.3 Overall Comments on Simulation of Flow Around Prismatic Hull with Yaw Angle

The research into flow around escort tug hulls will be extended to 3-dimensional CFD simulations, but these are much more complex to complete. The times at which the PIV experiments can be carried out are constrained, and it will be unproductive to complete the 3-dimensional simulations before starting experiments. Based on the results of this study, *FLUENT* can give sufficiently accurate predictions of fluid flow around a ship section at high yaw angles, by approximating the hull as a 2-dimensional prism, or for extreme yaw angles (close to 90 degrees) as a pure 2-dimensional problem.

Overall, *FLUENT* gives reasonable predictions of the flow patterns around the midsection of hull, when the hull is approximated as a 2-dimensional prism, based on validation against model data at 10 degrees and 35 degrees of yaw. *FLUENT* predicts regions of high-speed flow (around the upstream bilge radius) and regions of low speed flow on the downstream side of the hull. The level of prediction appears to be good enough for the purposes of planning PIV experiments.

FLUENT predicts that at some yaw angle between 10 degrees and 35 degrees, a closed flow forms on the downstream side of the hull, and the centre of the vortex forming this closed flow is approximately mid-depth of the hull and about the same distance downstream from the edge of the hull.

5.0 FLOW PATTERNS FOR ESCORT TUG HULL FORMS

5.1 Flow around 2-dimensional hull section shape

A Series 60 hull form is quite different from a typical escort tug form. Relative to the Series 60 model, the escort tug has a lower length to beam ration and a higher beam to draft ratio. Also, the hull used for this study has a midship section with two chines, rather than a bilge radius. As a result, it is important to see the differences in the flow patterns between a Series 60 hull and a typical escort tug hull. The hull selected for the PIV experiments has been tested for force measurements at IOT (Allan & Molyneux, 2004). A 2-dimensional mesh of the maximum section of this tug (at model scale) was created using *GAMBIT*. Flow speeds used were 0.491, 0.735 and 0.976 m/s (which corresponded to 4, 6 and 8 knots for the 1:18 scale model, based on Froude scaling). A view of the mesh, close to the hull is shown in Figure 9.

To simplify the problem, only a yaw angle of 90 degrees will be studied using a 2-dimensional mesh. This is a reasonable simplification for a case that is being planned for testing, since MUN's Mark I PIV system is designed for optimum resolution in a vertical plane. If the hull is aligned for a yaw angle of 90 degrees, by positioning it across the tank, and the laser sheet is directed parallel to the centreline of the tank, then the flow pattern around the maximum section can be obtained.

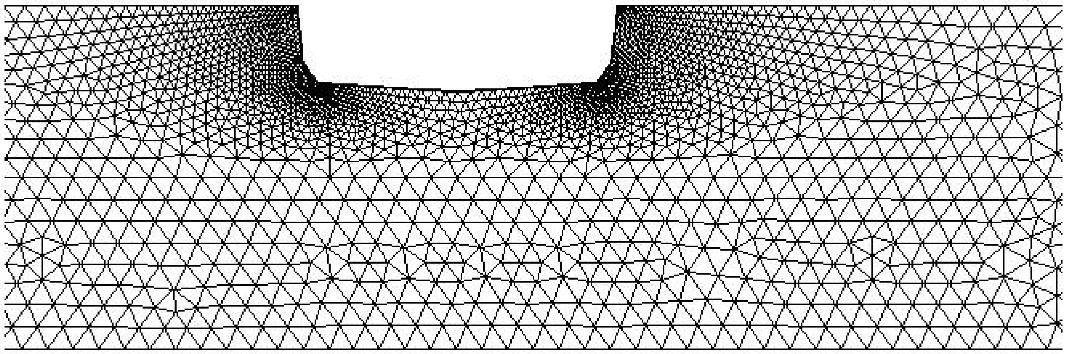

Figure 9, 2-dimensional mesh for escort tug model

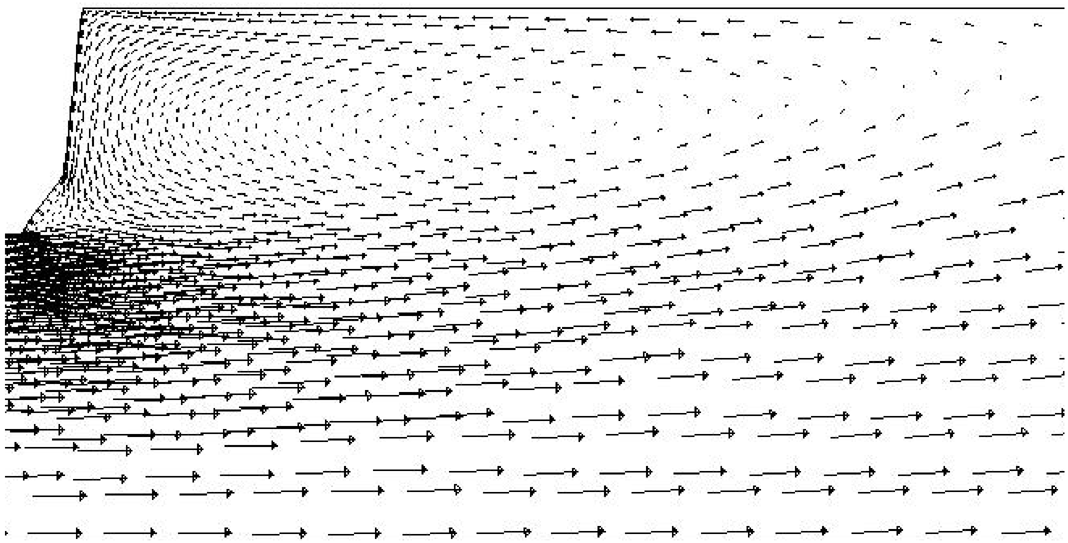

Figure 10, Predictions of velocity vector around escort tug model, undisturbed flow=0.735 m/s.

The simulated flow patters on the downstream side of the tug are shown in Figure 10. The predicted flow patterns show similar structures to the Series 60 data at 90 degrees yaw angle. The three dominant features of the flow are the closed flow on the downstream side of the hull, the high-speed flow at the corner of the up-stream bilge and the low speed flow across the bottom of the hull. These are areas that should be studied during PIV experiments.

The high speed flow at the upstream bilge radius is likely to be over predicted since it shows a high degree of separation at the second chine. It is unlikely that the experiments will show velocities as high as the simulations, but the flow patterns should contain the same structures.

5.2 Flow around 2-dimensional water plane shape

The other logical selection of a two-dimensional plane for studying flow around a hull with a yaw angle is a plane parallel to the waterline. In this orientation, the hull will be equivalent to a wing section, with the fluid flow creating high and low pressure regions on the hull. Depending on the degree of flow separation, the resulting flow patterns and forces can be steady (for small amounts of separation) or unsteady (when periodic vortex shedding occurs). The Iowa data (Longo and Stern, 1996) can be used for regions close to the hull surface for hulls with a small yaw angle, but no experiment data for flow velocities in this plane for ships with large yaw angles has been found while reviewing the literature. Two illustrative examples were found (van Dyke, 1982) which are relevant to the problem being considered. One case was for flow around a flat plate with a 45 degree yaw angle and the second was for a ship leaking oil in a current (with a flow direction assumed to be 45 degrees to the ships heading). These are shown in Figures 11 and 12.

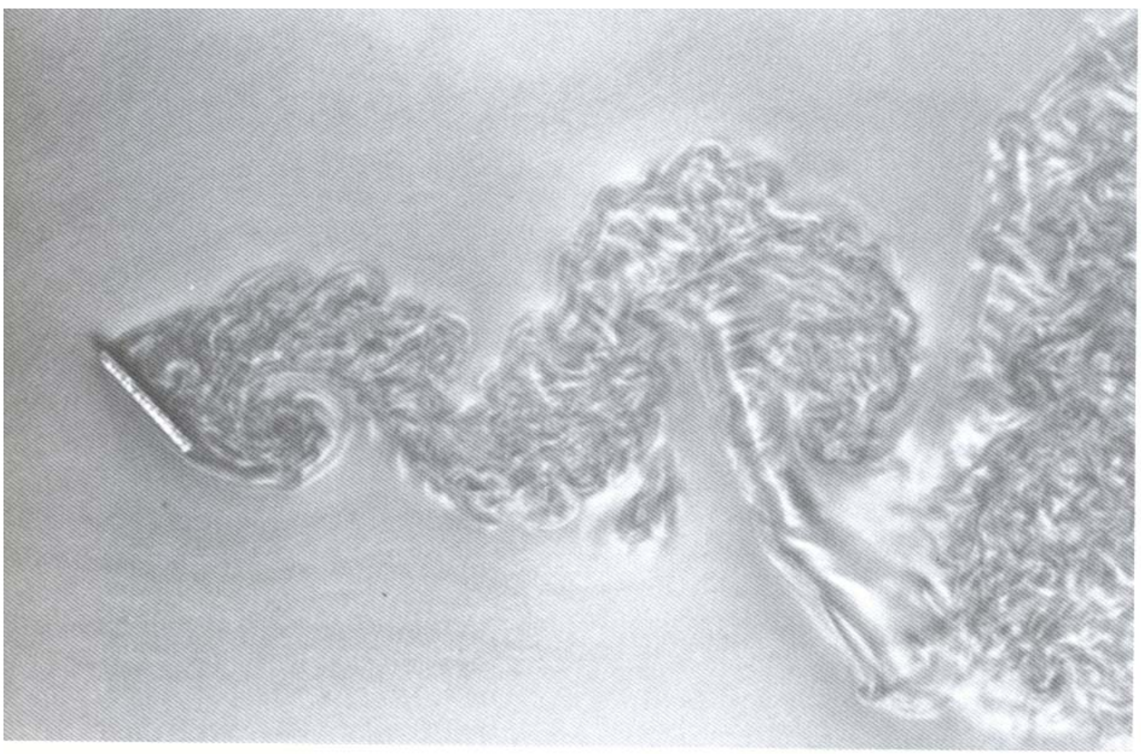

Figure 11, Flow around a plate at 45 degrees to flow (Van Dyke, 1982)

Figure 12, Oil leaking from a damaged ship, in current (Van Dyke, 1982).

Both these figures show that the downstream flow patterns are very large compared to the ship geometry, and that there is a periodicity to the flow patterns. As a result, it will be necessary to run the CFD predictions in the time domain using an unsteady flow solver.

A 2-dimensional mesh was created using *GAMBIT* and based on the waterline of the same escort tug that was used for the maximum section. The mesh was scaled to dimensions of the 1:18 scale model, and the wall boundaries were based on the location of the walls in the OERC towing tank. Three yaw angles were considered, zero, 10 degrees and 35 degrees. The mesh was unstructured, using triangular elements, varying in size, so that they were small close to the hull surface, but grew larger as they approached the wall boundaries. The same number of mesh elements at each boundary was used for each yaw angle.

The mesh for the hull at 35 degrees of yaw is shown in Figure 13. Flow was assumed to be uniform from the left hand boundary, at a speed of 0.735 m/s. The outlet was a pressure boundary, with backflow pressure normal to the surface.

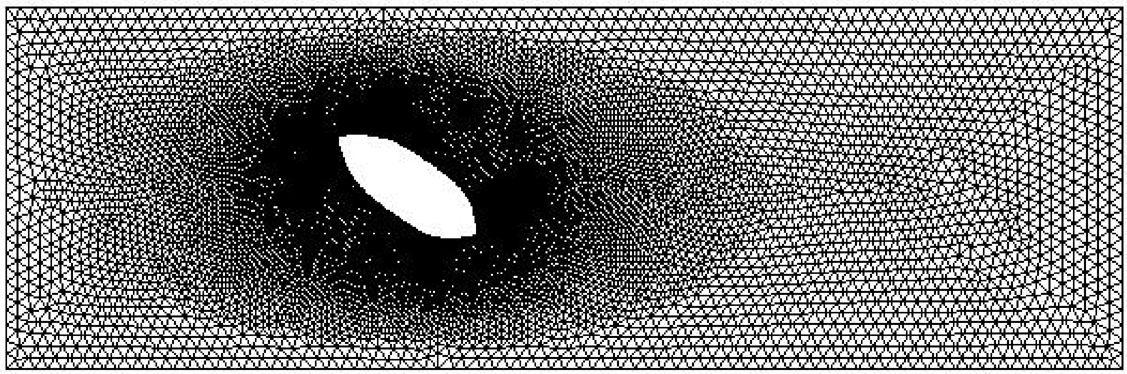

Figure 13, Mesh for 2-dimensional approximation to flow around an escort tug with a yaw angle of 35 degrees.

Contours of velocity magnitude are shown for the whole mesh in Figures 14 to 16, for yaw angles of 0, 10 degrees and 35 degrees. All cases were solved as unsteady flows. In the case of the zero and 10 degree yaw angles the solutions had converged to a steady value after 20 seconds, using time steps of 0.1 seconds. For 35 degrees yaw angle, the solution remained unsteady, and a total of 42.5 seconds of data was simulated.

Figures 14 and 15 show that for zero and 10 degrees of yaw, there is no structure to the wake more than one ship length behind the hull. However, for the case with 35 degrees of yaw, shown in Figure 16, the CFD simulations predict the oscillating structure of the wake that is seen in Figures 11 and 12. Figure 16 shows some very complex flow patterns with regions of high and low speed flow and transitions between them occurring within the length of the hull. Also, down stream from the hull are closed contours of flow velocity. The low speed flow is on the top and the high speed flow is on the bottom.

The simulations for 35 degree yaw angle also indicate that the flow patterns may be influenced by the tank boundaries. In particular, the region of high speed flow on the upper wall is a manifestation of the boundary conditions. If the boundaries are moved further away from the hull, the maximum flow speed is reduced but remains in approximately the same location.

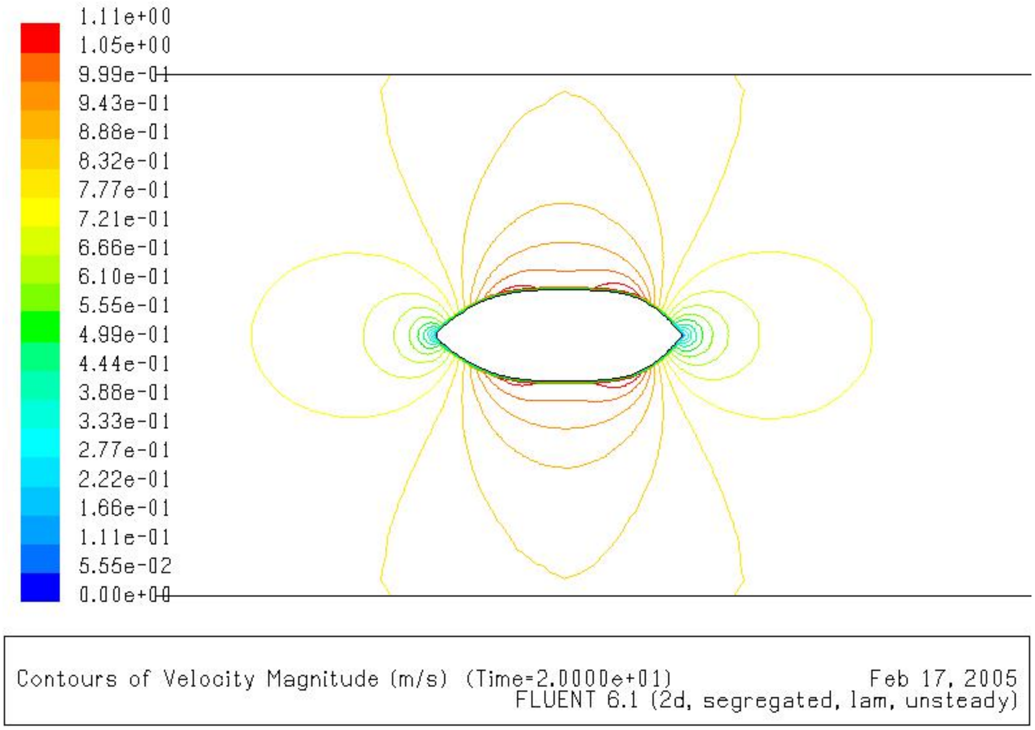

Figure 14, Contours of velocity magnitude for hull at zero yaw angle, time domain solution after 20 seconds of simulation

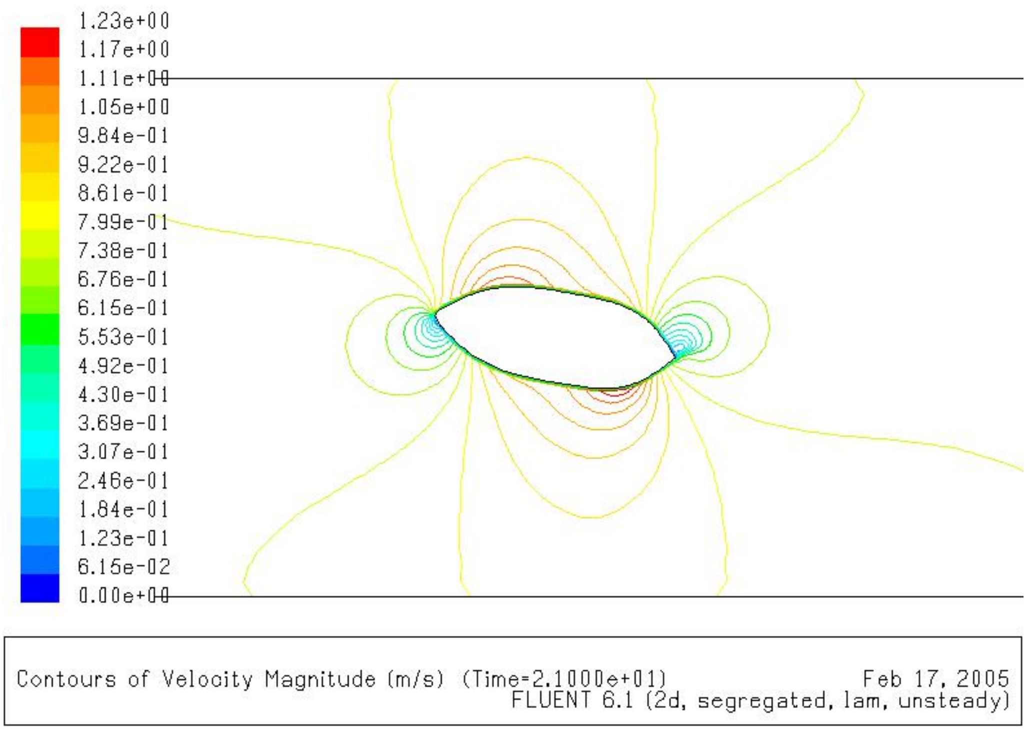

Figure 15, Contours of velocity magnitude for hull at 10 degrees yaw angle, time domain solution after 21 seconds of simulation

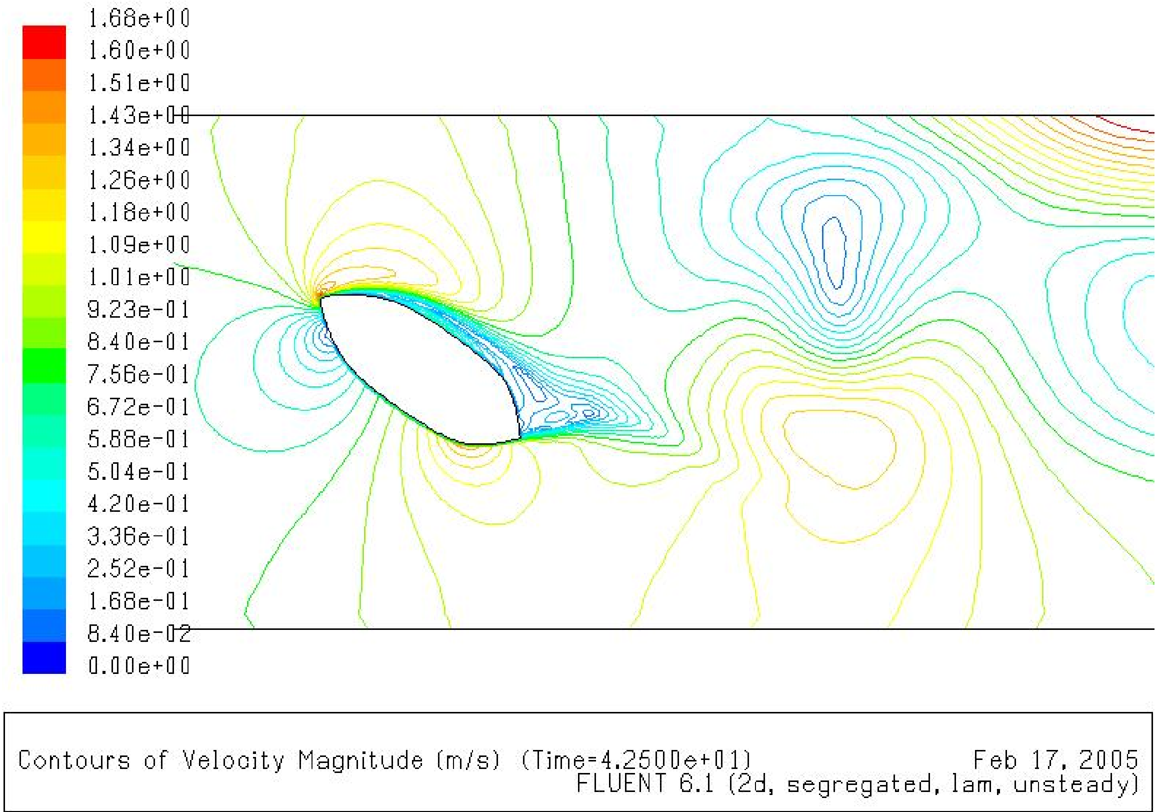

Figure 16, Contours of velocity magnitude for hull with 35 degrees yaw angle, time domain solution after 42.5 seconds of simulation.

6.0 RECOMMENDATIONS FOR PIV EXPERIMENTS AND FURTHER SIMULATIONS

A review of the Series 60 C_B=0.6 data from experiments to measure flow speed around the hull with yaw angle indicates that the most relevant area for escort tugs will be yaw angles of 35 degrees or more. At 10 degrees of yaw, there is no closed vortex on the downstream side of the hull. At 35 degrees, this flow pattern is clearly present in both the experiments and the CFD simulations based on the prismatic hull.

To confirm these flow patterns on escort tugs, PIV experiments should be made around the maximum section of a tug model. The areas of particular interest will be the amount of flow acceleration around the upstream bilge and the size and shape of the vortex on the downstream side of the hull. The size of the vortex should grow as the yaw angle is increased. Two suitable yaw angles for experimental measurements of flow patterns will be 35 degrees (representing typical operating conditions) and 90 degrees (the most extreme condition, and also the closest approximation to 2-dimensional flow).

Based on the results of 2-dimensional CFD simulations, the horizontal plane should result in some very interesting flow patterns at yaw angles of approximately 35 degrees. It will be challenging to make PIV measurements in this region, but high yaw angles are an important feature of escort tug operation, and the flow in these conditions must be understood if escort tug design is to progress based on a sound understanding of hydrodynamic principles.

The CFD simulations should be extended to three dimensions. Introducing additional degrees of freedom to the numerical model will change the flow patterns in both of the planes discussed here. The most obvious hull shapes to consider will be the Series 60 C_B=0.6 and the escort tug. The three dimensional simulations should also include the effect of the free surface, which was ignored in the 2-dimensional simplifications.

7.0 ACKNOWLEDGEMENTS

The author wishes to acknowledge Dr. Fabio di Felice of INSEAN and Dr. Joseph Longo of the Iowa Institute for Hydraulic Research who provided data from their experiments in electronic digital form, which allowed the results to be re-plotted for this report. In each case this required effort on their part to reformat the original test data.

8.0 REFERENCES

Allan, R. and Molyneux, D. 'Escort Tug Design Alternatives and a Review of Their Hydrodynamic Performance', World Maritime Technology Conference, Society of Naval Architects and Marine Engineers, Paper A11, September 2004.

Alessandrini, B. A. and Delhommeau, G., 'Viscous Free Surface Flow Past a Ship in Drift and in Rotating Motion', 22nd Symposium on Naval Hydrodynamics, Washington D.C., 1996 pp. 491-507.

Cura Hochbaum, A. 'Computation of the Turbulent Flow Around a Ship Model in Steady Turn and in Steady Oblique Motion', 22nd Symposium on Naval Hydrodynamics, Washington D.C., 1996, pp. 550-567.

Di Felice, F. and Mauro, S., 'LDV Cross-Flow Survey on a Series 60 Double model at Incidence', Proceedings of 9th International Offshore and Polar Engineering Conference, Brest France, May 30-June 4, 1999, pp. 536-543.

Tahara, Y., Longo, J. & Stern, F., 'Comparison of CFD and EFD for the Series 60 C_B=0.6 in Steady Drift Motion', *Journal of Marine Science and Technology*, Vol. 7, 2002 pp. 17-30.

Toda, Y., Stern, F. & Longo, J., 'Mean-Flow Measurements in the Boundary Layer and Wake and Wave Field of a Series 60 C_B=0.6 Ship Model-Part 1: Froude Numbers 0.16 and 0.316', Journal of Ship Research, Vol. 36, No. 4. Dec. 1992, pp. 360-377.

Longo, J. Stern, F. & Toda, Y., 'Mean-Flow Measurements in the Boundary Layer and Wake and Wave Field of a Series 60 C_B=0.6 Ship Model-Part 2: Scale Effects on Near-Field Wave Patterns and Comparisons with Inviscid Theory', *Journal of Ship Research*, Vol. 37, No. 1. Jan. 1993, pp. 16-24.

Longo, J. & Stern, F., 'Yaw Effects on Model-Scale Ship Flows', Proceedings, 21st Symposium on Naval Hydrodynamics, June 24-28, Trondheim, Norway, 1996, pp. 312-327

Van Dyke, M. '*An Album of Fluid Motion*', The Parabolic Press, 1982.