3rd CFD Techniques in Ship Hydrodynamics, Royal Inst. Naval Architects, London, 2003.

DEVELOPMENTS IN THE USE OF LARGE-EDDY SIMULATION FOR SHIP HYDRODYNAMICS

Richard Pattenden, University of Southampton, UK
Stephen Turnock, University of Southampton, UK
Neil Bressloff, University of Southampton, UK

SUMMARY

A summary is given of the work being carried out at the University of Southampton on the use of large-eddy simulation (LES) for marine hydrodynamics. The case of a truncated cylinder on a ground plane has been used to test an in-house code by comparisons with experimental data. This has shown that the LES model performs well in separated and vortical flow regions, but has limitations in its ability to capture boundary layer flow. Further wind tunnel experiments also used particle image velocimetry (PIV) to measure the instantaneous flow field in the stern region of the KRISO VLCC tanker form. LES simulations of an elliptical body, of similar proportions to the tanker form, illustrate similarities both to the VLCC flow structure and to that of the truncated cylinder. In particular, for all three geometries the instantaneous flow field contains many more large-scale turbulent structures in the wake with a much greater range of vorticity magnitude than in the mean vorticity field. This suggests that LES may give more accurate results than a steady RANS method for ship flows, if the computational cost of resolving the boundary layer can be reduced. Initial simulations of a two-dimensional cylinder flow have been carried out using detached-eddy simulation (DES), a hybrid RANS/LES technique, which is less demanding of grid resolution than full LES, and which could offer such a route.

AUTHORS BIOGRAPHY

Richard Pattenden is a research assistant in the School of Engineering Sciences at the University of Southampton. He completed his MEng in Ship Science at Southampton and ENSTA, Paris, and has been involved with a number of hydrodynamics projects. He is investigating the simulation of turbulent flows in marine hydrodynamics for his PhD.

Stephen Turnock is a senior lecturer in the School of Engineering Sciences at the University of Southampton. His main interests lie in the application of CFD to the analysis and design of marine vehicles.

Neil Bressloff is a senior research fellow in the School of Engineering Sciences at the University of Southampton. He has worked in the field of CFD for over 10 years and been involved in a broad range of projects including a number that are hydrodynamic related. He has written his own parallel three-dimensional, curvilinear CFD code that has been further developed during the course of the work presented here.

NOMENCLATURE

d	Diameter of cylinder / breadth of ship
δ	Thickness of boundary layer
f	Frequency
h	Height of cylinder
ν	Kinematic viscosity
$\boldsymbol{\omega}$	Vorticity vector
Re	Diameter Reynolds number $Re = U_\infty d/\nu$
ρ	Density
Str	Strouhal number $Str = fU_\infty/d$
T	Time
τ_w	Wall shear stress
$\mathbf{u}$	Velocity vector
u,v,w	Velocity components along x,y,z
U_∞	Free-stream velocity
$\boldsymbol{x}$	Coordinate vector
x,y,z	Coordinate axes in streamwise, lateral and vertical directions

1. INTRODUCTION

Many of the flows encountered in ship hydrodynamics are turbulent separated flows or regions containing weak unstable vortices. Examples of such flows can be found in the stern region of full-bodied ships such as crude carriers, flows around appendages to the hull and the flows around superstructures. These flows are currently modelled by solving the Reynolds-averaged Navier-Stokes (RANS) equations using suitable turbulence closures. However, in order to achieve reasonable accuracy careful application and some tuning are required to capture the mean flow characteristics and even then some detail of the turbulent structures is lost[1].

These limitations can be partially overcome by the use of Large-Eddy Simulation (LES)[2]. In this case all of the turbulent structures larger than the grid cell size can be simulated with only the smaller scales being modelled. Advances in computing are now making it possible to carry out LES on real geometries, allowing deeper investigation of the nature of the turbulence in these regions. The remaining issue is the problem of resolving the boundary layer with sufficient accuracy, which for full LES at high Reynolds numbers would require an impractical number of grid cells. Progress in this area includes the development of Detached-Eddy Simulation (DES)[3], which is a modified RANS model that works in RANS mode in the boundary layer while operating as LES in the outer region where the larger scale turbulent structures are found. The use of DES applied to the stern flow of a tanker form is currently being investigated.

The advantage of using LES for the simulation of stern flows, in addition to a potential improvement in accuracy, is the ability to simulate the unsteady behaviour of the flow, in particular the size and frequency of the turbulent structures. This could be of importance with regard to modelling propulsor flows and also for modelling the flow over rudders. Propulsor flows in particular are unsteady flows that could be strongly influenced by any periodicity in the incoming flow. Another area that may benefit is research into acoustic signatures of ships, where the turbulence contributes to the noise generated by a vessel.

The prediction of the flow around a tanker hull was the subject of a workshop in Gothenburg in 2000[1]. Here various RANS models were tested by comparison with a set of data on the KVLCC2 hull form, provided by Van et al.[4]. It was found that simple k-ε models did not reproduce the detail of the flow in the propeller plane, although the k-ω SST model did capture the characteristic hook pattern in the wake contours better.

Much of the research on LES of wake flows has focused on square cylinders and surface mounted cubes, which have sharp edges, fixing the separation point. Rodi [2] performed a comparison between LES and RANS for the flow around these two geometries. He found that the standard k-ε model over predicts the turbulence production in the stagnation region and the length of the separation region was over predicted. LES was able to predict the turbulent fluctuations, and generally predicted the overall flow much better than the RANS.

Murakami [5] also looked at the surface mounted cube, with broadly similar results. He also found that the standard k-ε model over predicted the turbulence production but a modified version corrected this. He also used a Reynolds stress model (RSM), which gave quite poor results. He concluded that this type of model would need some refinement to work well on this type of flow. The LES results were promising, particularly with the use of a dynamic subgrid-scale model.

A number of papers have been published using detached-eddy simulation for various flows, in particular aircraft wings [6], cylinders [7] and a prolate spheroid [8]. This latter case shows that for flows without massive separation DES may not offer any advantage over the RANS model it is based on unless unsteadiness in the region of the detaching boundary layer is properly resolved. However at higher angles of incidence the separation becomes more pronounced and so DES becomes advantageous. This means that predictions of ships at angles of yaw are likely to benefit more from DES than ones at zero incidence.

This paper will present a discussion of the benefits and difficulties associated with the use of LES for ship hydrodynamics. A number of results will be presented starting with a study of the flow around a cylinder with a free-end mounted on a ground plane, initial details of which were presented in [9]. This shows that LES gives good predictions of the recirculation region but cannot resolve the boundary layer flows. Initial results using the detached-eddy simulation approach on a two-dimensional cylinder are then presented giving encouraging results on a coarse grid. Measurements of the flow around the KVLCC2 tanker hull are presented to demonstrate the unsteadiness of the wake flow and to draw comparisons with the truncated cylinder flow. Finally an LES simulation of an elliptical body created by stretching the truncated cylinder grid in the x-direction is presented. This had the same L/B ratio as the KVLCC2 tanker but with the same cross-section as the cylinder. The flow here is very similar to that of the ship, in that the longitudinal tip vortices are dominant. The region of separated flow is reduced compared to the cylinder, due to the weaker adverse pressure gradient.

2. COMPUTATIONAL APPROACH

The computational simulations were carried out using a code developed at the University of Southampton [10], which is a fully parallel finite volume solver with a range of turbulence models and numerical schemes available. It is a structured grid solver and is parallelised according to the block structure. The code was originally developed for unsteady RANS methods but has since been extended to allow large-eddy simulations and now also detached-eddy simulations. Second order differencing was used and the SIMPLEC pressure-correction method was employed. The solver uses the MPI libraries to allow parallel computation on a PC cluster. The truncated cylinder computations were carried out on 16 nodes each with 2 1GHz processors and 512Mb of RAM. Mesh generation is also handled by a code developed at the University [12].

An LES model using the structure function subgrid scale model, as proposed by Métais and Lesieur [12], has been implemented in the code. Here the eddy viscosity is given by,

$$\upsilon_t = 0.105 C_K^{-3/2} \Delta x F_2^{1/2}$$

where,

$$F_2 = \left\langle \left\| \mathbf{u}(\mathbf{x},t) - \mathbf{u}(\mathbf{x}+\mathbf{r},t) \right\|^2 \right\rangle_{\|\mathbf{r}\|=\Delta x}$$

C_K is the Kolmogorov constant and Δx is the cell size.

Detached-eddy simulation (DES) is a method proposed by Spalart et al[3], in order to overcome the considerable computational cost involved with full LES simulations. The basis of this method is that RANS models are quite capable of modelling boundary layer flows, but generally fail to predict the larger scale eddies in regions of separated flow. In these regions LES is far better at capturing the flow physics. DES is therefore a hybrid of RANS and LES, where the model operates in RANS mode close to the walls and in LES mode away from the walls. The original DES method of Spalart is based on the Spalart-Allmaras one equation turbulence model [13]. The standard S-A model uses the distance to the nearest wall as the length scale, *d*. In DES the length scale is modified so that it depends on the grid spacing, Δ as follows:

$$\tilde{d} \equiv \min(d, C_{DES}\Delta)$$

where Δ is the largest dimension of the cell, and C_{DES} is a constant normally taken as 0.65.

The detached-eddy simulation model was implemented in the same code as before, by modifying an existing implementation of the Spalart-Allmaras model to use the new length scale. The numerical schemes are otherwise the same as for the RANS model.

3. FLOW AROUND A TRUNCATED CYLINDER

3.1 DESCRIPTION OF THE FLOW

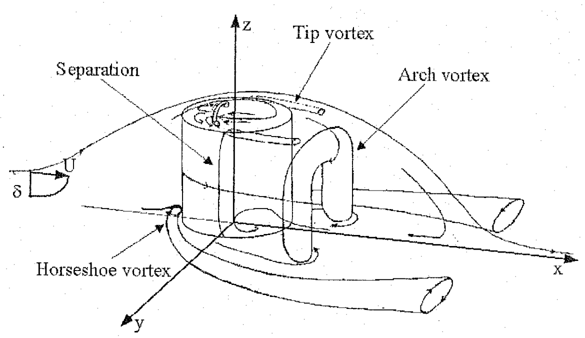

Figure 1: Diagram showing mean flow topology around truncated cylinder

The flow around a two-dimensional or infinitely long circular cylinder is a widely researched topic and is commonly used as a test case for unsteady bluff body flow calculations. The effects of truncating a cylinder to form a free-end are less well documented, particularly for low-aspect ratios of around 1. In this case the flow becomes dominated by the three-dimensional effects of the free-end. This gives rise to a highly turbulent region of separated flow. This is difficult for RANS methods to model accurately due to the dominance of large-scale structures that are anisotropic and strongly influenced by the boundary conditions.

Experiments were carried out on a cylinder mounted on a ground plane in a wind tunnel. The cylinder had a height/diameter ratio = 1 with a free end, giving rise to a highly three-dimensional wake structure. The von-Karman type vortex shedding, seen on longer cylinders is suppressed at this aspect ratio leaving a symmetric arch vortex which is shed with weak periodicity [14]. The flow topology is illustrated in Figure 1. There are also four main longitudinal vortex structures arising from the free end and the juncture with the ground plane. These latter are the so-called horseshoe or necklace vortices, while the former are formed by the flow wrapping around the free end. A complex pattern of vortices is also seen in the separation bubble on the top of the cylinder.

At the chosen Reynolds number of $2x10^5$ the attached flow on the front of the cylinder is laminar with laminar separation, while the flow on the ground plane is turbulent. Transition occurs in the shear layers immediately after separation. This presents problems for a RANS method, which will not predict the separation point correctly. An LES model without wall-functions will need very fine grid resolution on the walls to simulate the boundary layer flows fully.

The experiments were carried out in a 0.9x0.6m wind tunnel. The model had a diameter and height of 0.15m and most of the tests were conducted at a flow speed of 20m/s. The model was mounted on a ground plane with a boundary layer thickness of 15mm or 10% of the height. Further details of this experiment and the initial LES simulations were presented in [9]. This paper also contained results from a k-ε RANS model, which were compared to the LES results. Some of the main findings are repeated here along with the latest LES results.

3.2 COMPUTATIONAL SETUP

The computational grids were designed to match the geometry of the model in the wind tunnel working section, including the walls. They were composed of 32

Table 1: Comparison of results from different models

Model	No. cells	Separation (degrees)	Fwd. Sep. (x/d)	Reattachment (x/d)
k-e	2052480	107.2	-0.79	2.14
LES	2052480	82.8	-1.48	1.55
LES	2592640	90.0	-1.73	1.53
Exp		70		1.56

structured blocks in an O-grid format. The cells were clustered towards the walls of the cylinder and the floor of the tunnel as shown in Figure 2. On the finest grid the cell size on the floor of the tunnel was *0.0005d* and on the cylinder walls *0.0001d*, leading to a non-dimensionalised first cell size, y^+, less than 1, where $y^+ = u_\tau y/\upsilon$ and $u_\tau = \sqrt{\tau_w/\rho}$. The domain size was *3.0h* upstream of the cylinder, *6.7h* downstream, *6.0h* laterally and *4.0h* vertically. No-slip conditions were applied to the walls of the tunnel, although the grid was not fine enough here to resolve the wall flow. A uniform velocity was applied to the inlet and a zero-gradient condition to the outlet. The LES simulations were carried out on two different grids, the finer one having smaller cells near the wall. The time-step was set at 0.005 units, normalised with velocity and diameter, which corresponds to a CFL number of less than 1 at the side of cylinder in the high speed flow. The statistical quantities were averaged over 200000 time steps.

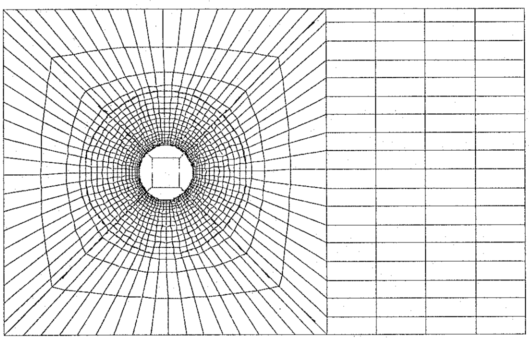

Figure 2: Diagram showing arrangement of the computational grid (z-plane)

3.3 ANALYSIS OF RESULTS

Table 1 compares the positions of separation on the cylinder side and on the ground plane upstream, as well as the reattachment point on the ground. Both LES solutions give good agreement with the reattachment point behind the cylinder. This is supported by Figs. 3, 4 and 5 which show profiles of the velocity components in the wake of the cylinder, comparing the PIV experimental data with the LES results on the fine grid. It can be seen that the agreement is quite good in this region.

The problems occur, however when it comes to predicting separation from a wall. The forward separation point and the form of the horseshoe vortex are not captured correctly. Also the separation on the cylinder side is not correct, although closer than the RANS model as the boundary layer here is laminar. These weaknesses are probably due to the lack of resolution in the wall region, particularly in the wall parallel directions, which are required to be almost as small as the wall normal size for a full LES solution. In fact the results show that the finer grid is actually worse

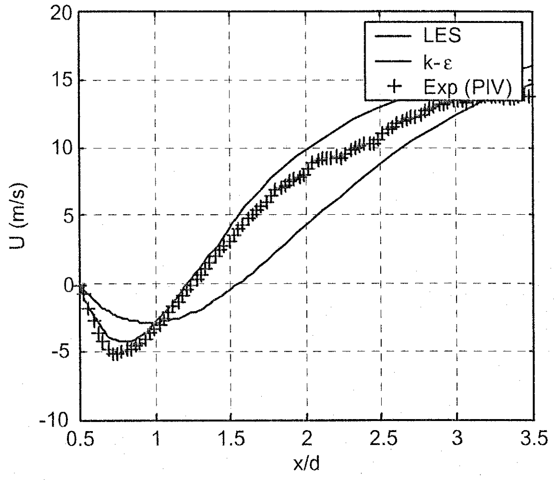

Figure 3: Plot of U velocity at $y/d=0$, $z/d=0.25$

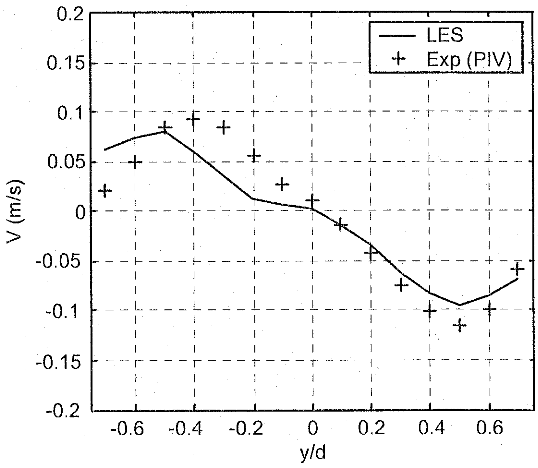

Figure 4: Plot of V velocity at $x/d=1.5$, $z/d=0.25$

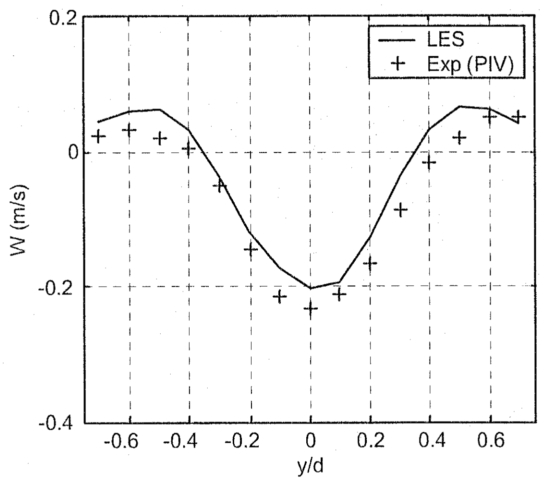

Figure 5: Plot of W velocity at $x/d=1.5$, $z/d=0.25$

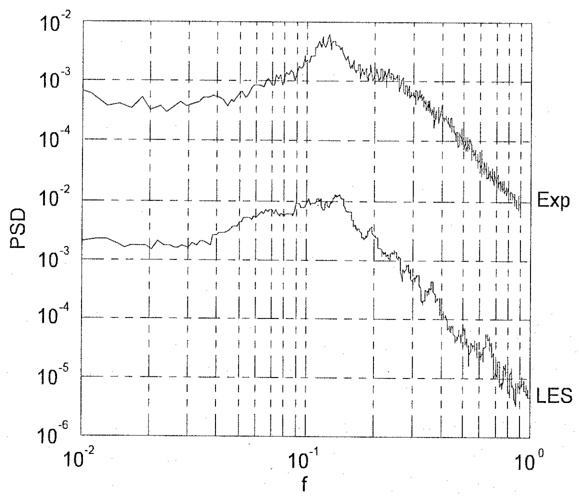

Figure 6: Power spectral density function of U velocity for LES and experiment (Curves offset for clarity)

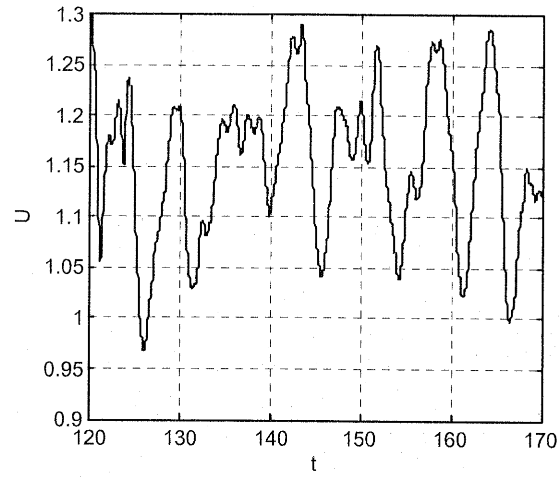

Figure 8: Time history of U velocity in wake of 2D cylinder from DES simulation

than the coarser grid but this probably indicates a lack of grid convergence. The use of a wall function such as that proposed by Werner and Wengle [15] could be used to try to overcome this although there are uncertainties about the validity of wall functions in an LES simulation [2].

As was shown in [9] the pressure distributions and local drag coefficients are dependent on the accuracy of the prediction of separation from the cylinder side and so they are poorly predicted here. Figure 3 present,s a plot of U-velocity along the centreline of the tunnel at a height of $z/d=0.25$ downstream of the cylinder. This highlights the difference in the length of the recirculation region between the k-ε models and the LES and experimental results. The LES curve is much closer to the experimental data.

A plot of the power spectral density function of the U velocity, measured outside the wake near the top of the cylinder, from the LES results is given in Fig. 6. This shows that the LES predicts the shedding frequencies quite well compared to the hot-wire measurements from the experiments. The vortex shedding frequency predicted by the LES is in good agreement with the hot-wire measurements from the experiments, with a Strouhal number of 0.135 for the LES compared to 0.125 for the experiments. This could also be associated with the late separation, creating a narrower wake.

The results presented here show that the LES model performs well at simulating the large-scale turbulent structures in the recirculation region. Its weakness however is in the prediction of boundary layer flows, where a very fine grid resolution is required.

3.4 TWO-DIMENSIONAL CYLINDER FLOW

One of the standard test cases for bluff body flows is the "two-dimensional" or infinitely long circular cylinder flow which exhibits von-Karman vortex shedding at Reynolds numbers greater than 100. This case was used to test the implementation of the DES model. Travin et al.[7] carried out a series of tests using DES to model the flow around a circular cylinder which generally gave good results although there is some uncertainty in the separation point. This appears to have more to do with the complexity of the flow in this Reynolds number range than the model itself.
A purely two-dimensional simulation was performed although this is not strictly realistic when carrying out large-eddy simulations because there are still significant three-dimensional effects in this flow. A relatively coarse grid was used with a minimum y+ of 5. A Reynolds number of 10^6 was chosen in order to be in the fully turbulent regime. The time history of the U-velocity at x/d=1.5 and y/d=1.5 is shown in Figure 8. A Strouhal number of 0.2 was found from the velocity fluctuation in the wake. The separation point was predicted at 90 degrees, which is perhaps too far forward for a turbulent boundary layer, which should be nearer 110 degrees[16]. These results would no doubt be improved by using a three-dimensional grid and refining it further to properly resolve the three-dimensional effects.

Interestingly an LES simulation was run on the same grid but this didn't predict separation until around 155 degrees, presumably due to the lack of the third dimension and grid resolution.

4. SHIP-LIKE FLOWS

4.1 DISCUSSION OF SHIP FLOWS

Flows around modern full-bodied ships such as tankers and bulk carriers are typified by the flow around the KVLCC2 hull form which was developed at the Korean Institute of Ships and Ocean Engineering [4] as a test case for numerical hydrodynamics. The flow in the propeller plane is of particular interest due to the interaction of the bilge vortex with the propeller. It was used in the Gothenburg 2000 workshop, where it was found that some RANS models were able to produce reasonable results. However, the anisotropic nature of the turbulence here means that two-equation RANS models have difficulty modelling the turbulent kinetic energy in the wake.

While many of the features found in the truncated cylinder flow have similarities with the flows found around the sterns of ships, there are certain differences. In particular the Reynolds number around large tankers is typically 10^9, four orders of magnitude greater than that considered here. There is a long length of attached flow along the hull with a turbulent boundary layer. This boundary layer interacts with the bilge vortex towards the stern to produce a region of unsteady turbulent flow.

4.2 PIV EXPERIMENTS ON THE KVLCC2 HULL

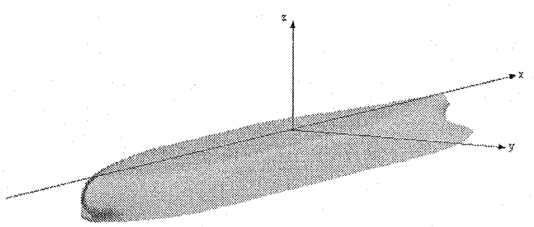

Figure 9: Axis system of KVLCC model

Wind tunnel experiments have been carried out at Southampton on a segmented model of the KVLCC2 hull form (Fig. 9). This model is divided longitudinally into three sections so that the parallel mid-body can be removed, and also allows for an extra piece to be added above the waterline to increase the depth of the model. The model was mounted on a ground plate in the wind tunnel in the same way as the cylinder described before. While this is not a true representation of the real ship due to the presence of a boundary layer on the ground plate, a CFD model can be set to model this. Particle image velocimetry (PIV) was used to obtain data on the flow in the propeller plane. The laser sheet was oriented in the y-z plane so that vectors of V and W were measured. The data was averaged over 500 samples. Figure 10 shows plots of velocity vectors with contours of vorticity, $\boldsymbol{\omega} = \nabla \times \mathbf{u}$, normalised by the maximum instantaneous vorticity magnitude. Only half of the plane was measured to enable greater resolution. The plots are presented with the ship inverted (keel upwards) so that they can be compared with those from the cylinder in Figure 7.

In the mean flow there are the longitudinal bilge vortices close to the hub of the propeller as found in [4]. However, as in the truncated cylinder flow these mean vortices appear very unsteady in nature revealing a chaotic instantaneous flow with many large-scale coherent structures. Only in the mean flow do these appear as the two counter-rotating vortices. This is similar to the flow found by Bearman [17], for the flow around a car. It is questionable whether the mean flow produced by a RANS calculation or even measured by time-averaging techniques is a valid picture of the flow for many applications. For example the flow over the propeller blades could be more influenced by this short time-scale flow pattern than by the mean flow. While this may not matter for the mean thrust characteristics it may be a factor in the simulation of vibrations and pressure fluctuations.

4.3 LES OVER AN ELLIPTICAL BODY

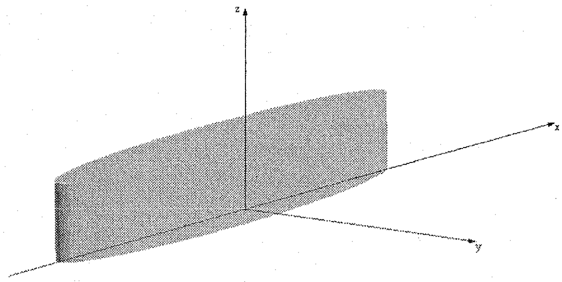

Figure 11: Axis system of elliptical body

As an intermediate step before simulating a full ship geometry, the truncated cylinder grid was stretched in the x-direction to give an elliptical shape with the same length/breadth ratio as the KVLCC2 tanker (Fig. 11). The computational method was identical to that in the case of the LES for the truncated cylinder with the time step doubled to account for the longer x-dimension. The solution was restarted from the existing cylinder results but the averaging process was reinitialised. Another 40000 time steps were run although this was still not enough for a reliable average as can be seen in Fig. 10c.

The flow in this case is more like the ship flow in that there is no massive separation at the back of the body, but due to the sharp corner at the tip, a pair of longitudinal vortices is formed, equivalent to the bilge vortices. Figure 10c and 10d show the mean and instantaneous flows in the yz-plane. It can be seen that the instantaneous flow is composed of a number of large-scale vortex structures indicating that the longitudinal vortices are unsteady. While the mean flow is not properly averaged due to insufficient time steps, this shows that a similar flow pattern to that which occurs in the wake of the cylinder and the KVLCC.

5. CONCLUSIONS

1) The results from the truncated cylinder simulations have shown that LES performs well at predicting the region of separated flow in the wake of the cylinder. The longitudinal vortex structures are well predicted as well.
2) The LES was found to be weak at predicting the boundary layer flow on the floor and the side of the cylinder. Much greater resolution would be needed in this region for a full LES solution. The alternative is to use a hybrid RANS/LES technique such as detached eddy simulation.
3) The initial tests of DES on a two-dimensional cylinder flow were encouraging with the vortex shedding being correctly predicted as well as the separation point.
4) The PIV experiments on the KVLCC hull have shown that the flow in the propeller plane is highly unsteady with large-scale vortical structures being formed, similar to those behind the cylinder.
5) An LES simulation of a ship-like body, a stretched cylinder, was carried out, which shows that it is possible to resolve the wake flow although the boundary layer is still a problem.

The results presented above have shown that the large-eddy simulation method is well suited to some of the flows found in ship hydrodynamics. The truncated cylinder flow was found to be a useful test case for LES as it exhibits many characteristics of the stern flows found on tanker hulls and showed that the LES is capable of simulating this type of flow.

The problem with using LES for these flows is the grid resolution required to model the boundary layer sufficiently. This can be overcome by using the detached-eddy simulation method, which combines RANS and LES. This has been shown to perform well in LES mode by predicting the two-dimensional cylinder flow but the computational cost of this method is considerably less than for full LES as it operates in RANS mode near the wall.

REFERENCES

1. LARSSON, L., STERN, F. AND BERTRAM, V., 'Gothenburg 2000: A workshop on numerical ship hydrodynamics', Chalmers University of Technology, 2000.
2. W. RODI, 'Comparison of LES and RANS calculations of the flow around bluff bodies', Journal of Wind Engineering and Industrial Aerodynamics, 69-71:55--75, 1997.
3. SPALART, P.R., JOU, W-H., STRELETS, M. AND ALLMARAS, S.R., 'Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach', 1st AFOSR Int. Conf. on DNS/LES, 1997.
4. VAN, S.H., KIM, W.J., KIM, G.T., KIM, D.H. AND LEE, C.J., 'Experimental investigation of the flow characteristics around practical hull forms', Proc. 3rd Osaka colloquium on advanced CFD applications to ship flow and hull form design, 1998.
5. S. MURAKAMI, 'Overview of turbulence models applied in CWE-1997', Journal of Wind Engineering and Industrial Aerodynamics, 74-76:1--24, 1998.
6. MORTON, S., FORSYTHE, J., MITCHELL, A. AND HAJEK, D., 'DES and RANS simulations of delta wing vortical flows', AIAA Paper No. 2002-0587, 2002.
7. A. TRAVIN, M. SHUR, M. STRELETS, AND P. SPALART, 'Detached-eddy simulations past a circular cylinder', Flow, Turbulence and Combustion, 63:293--313, 2000.
8. CONSTANTINESCU, G.S., PASINATO, H., WANG, Y-Q., FORSYTHE, J.R. AND SQUIRES, K.D., 'Numerical investigation of flow past a prolate spheroid', Trans. ASME, Journal of Fluids Engineering, 124:904-910, December 2002.
9. PATTENDEN, R.J., TURNOCK, S.R. AND BRESSLOFF, N.W., 'An experimental and computational study of three-dimensional unsteady flow features found behind a truncated cylinder', 24th Symposium on Naval Hydrodynamics, Fukuoka, Japan, 2002.
10. N.W. BRESSLOFF, 'A parallel pressure implicit splitting of operators algorithm applied to flow at all speeds', Int. Jnl of Numerical Methods in Fluids, 36:497-518, 2001.
11. N.C. RYCROFT, 'Three-dimensional multi-block grid generation – Fleximesh', Ship Science Report 101, University of Southampton, November 1997.
12. O. MÉTAIS AND M. LESIEUR, 'Spectral large-eddy simulations of isotropic and stably-stratified turbulence', Journal of Fluid Mechanics, 239:157--94, 1992.
13. P.R. SPALART AND S.R. ALLMARAS, 'A one-equation turbulence model for aerodynamic flows', La Recherche Aérospatiale, (1):5--21, 1994.
14. OKAMOTO, S. AND SUNABASHIRI, Y., 'Vortex shedding from a circular cylinder of finite length placed on a ground plane', Trans. ASME, Journal of Fluids Engineering, 114:512--521, December 1992.
15. WERNER, H. AND WENGLE, H., 'Large-eddy simulation of turbulent flow over and around a cube in a plane channel', in Durst, F., Friedrich, R., Launder, B., Schmidt, F., Schumann, U. and Whitelaw, J. eds, 'Selected papers from the 8th Symposium on Turbulent Shear Flows', Springer, 155-168, 1993.
16. ROSHKO, A., 'Experiments on the flow past a circular cylinder at very high Reynolds number', Journal of Fluid Mechanics, 10(3):345-356, 1961.
17. BEARMAN, P.W., 'Near wake flows behind two- and three-dimensional bluff bodies', Journal of Wind Engineering and Industrial Aerodynamics, 69-71:33-54, 1997.

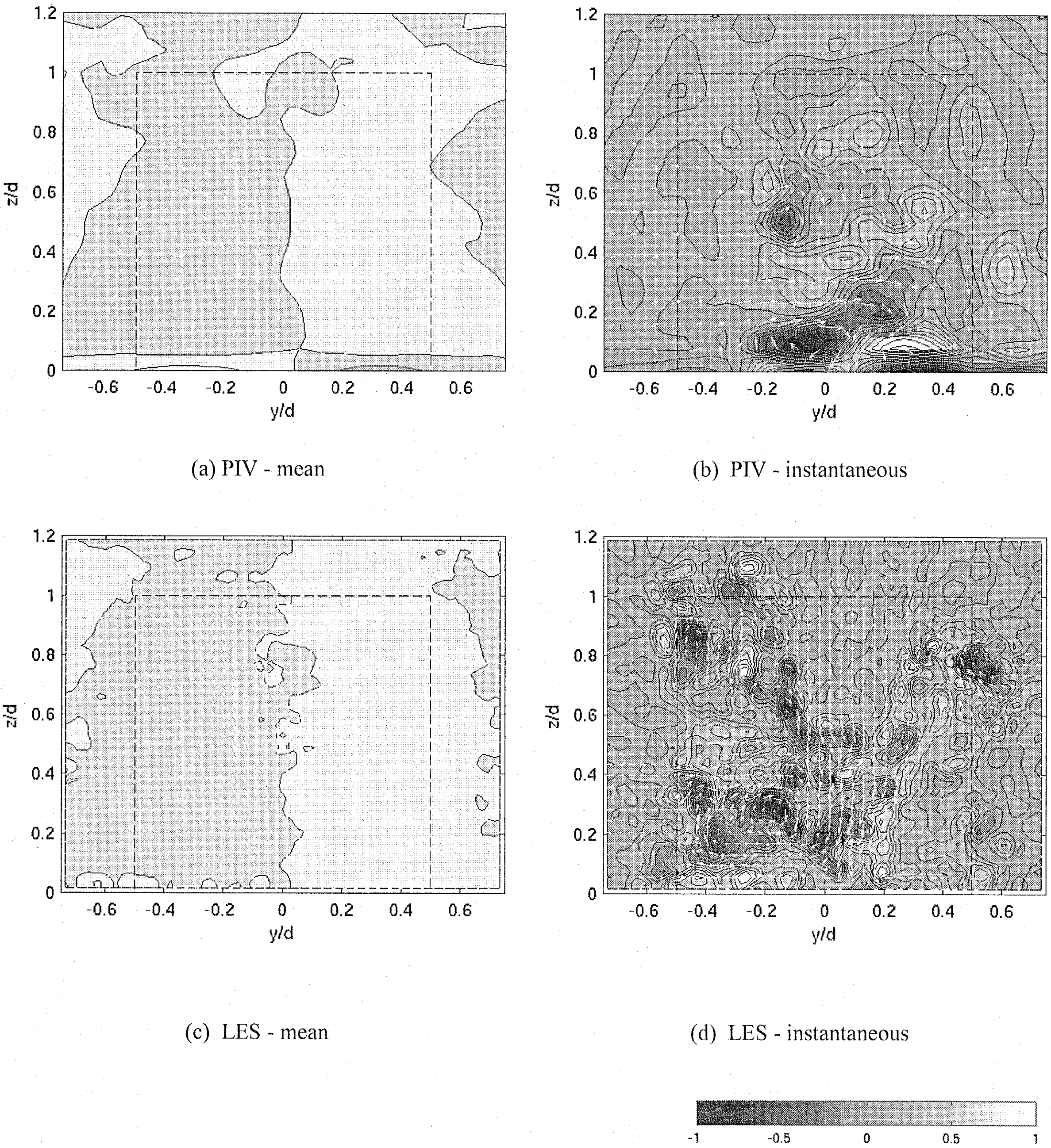

Figure 7: Velocity vectors and contours of normalised vorticity of the PIV measurements and LES simulations of the truncated cylinder flow at x/d=1.5.

Maximum values of vorticity used for normalisation were: (a) and (b) 12.16, (c) and (d) 8.88

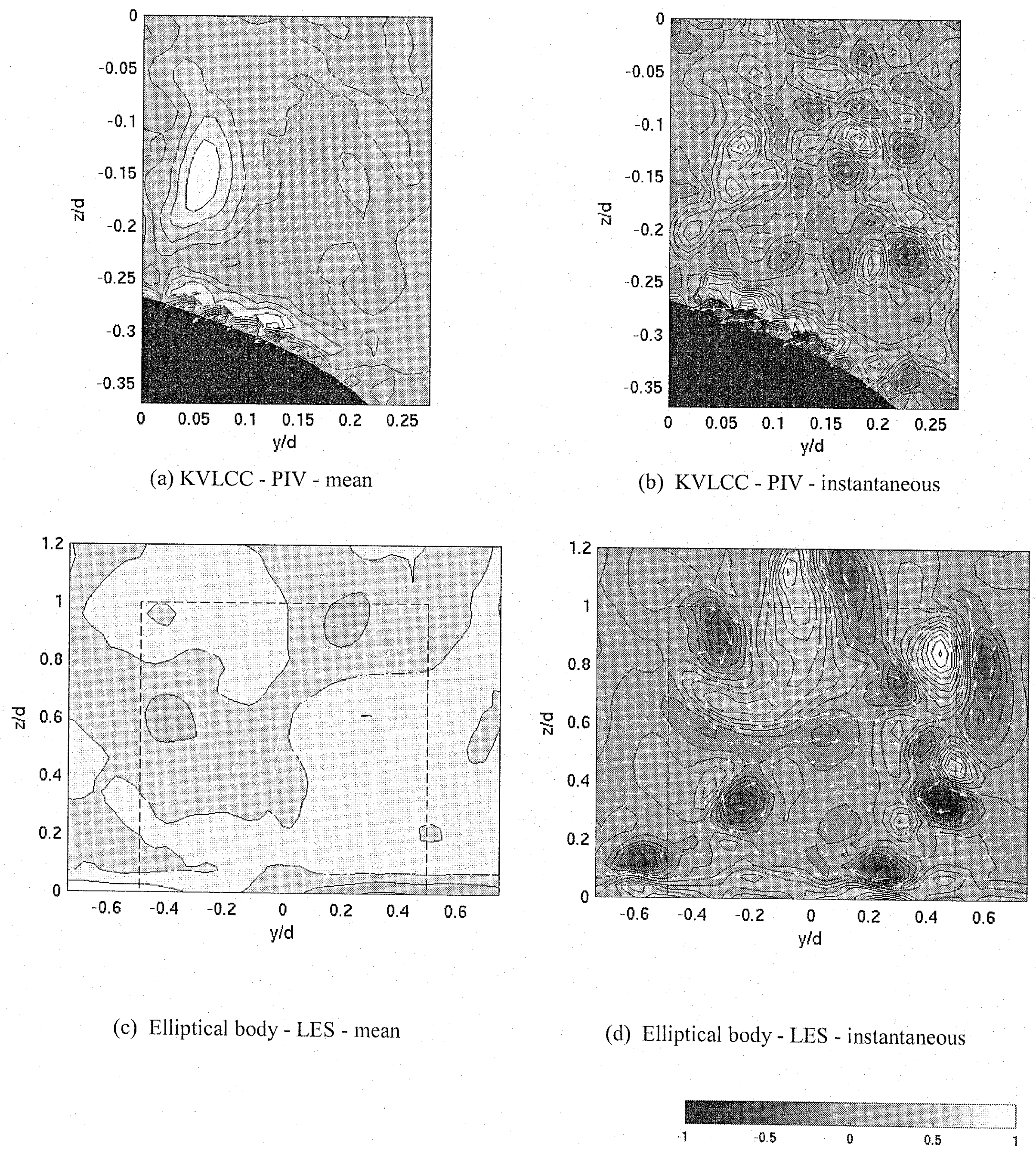

Figure 10: Velocity vectors and contours of normalised vorticity of the PIV measurements of the KVLCC hull at the propeller plane and the LES simulations of the elliptical body at x/L=1.5

Maximum values of vorticity used for normalisation were: (a) and (b) 57.70, (c) and (d) 1.70