Adaptive Grid Refinement
Applied to RANS Ship Flow Computation

J. Wackers, K. Ait Said, G.B. Deng, P. Queutey, M. Visonneau
(Laboratoire de Mécanique des Fluides, CNRS / Ecole Centrale de Nantes, France),
I. Mizine (CSC Advanced Marine Center, Washington, DC USA)

ABSTRACT
An adaptive grid refinement method has been developed for the ISIS-CFD flow solver. The method is fully integrated in the flow solver and is meant to be used for all the different fields of application of this RANS solver. Therefore, the method is made to be as general, flexible and robust as possible, to enable directional refinement, unstructured grids, and fully parallelised computation. The evaluation of the refinement criterion is made such that refinement criteria can be easily exchanged.

In this paper, the refinement method is applied to two highly different test cases, to show its versatility. In the first test, the double-model flow around the KVLCC2 tanker, pressure-based refinement criteria are used to create a locally refined grid from a very coarse initial grid. Results in the propeller plane indicate that good accuracy is obtained with significantly less cells than on fine grids made by hand. The second test case is the HALSS fast trimaran concept. For this case, parameter studies are first performed on non-refined grids, in order to determine the influence of the outer hull positions and the centre hull skeg design. Grid refinement around the position of the water surface is then applied to selective cases in order to get a clearer view of the wave field generated; these results allow us to confirm the conclusions from the parameter study.

INTRODUCTION
Automatic adaptive grid refinement is a technique for optimising the grid in the simulation of fluid flow, by adapting the grid to the flow as it develops during the simulation. This is done by locally dividing cells into smaller cells, or if necessary, by merging small cells back into larger cells in order to undo earlier refinement. Grid refinement is effective for flows that have localised structures; fine cells can be used to keep the error near these structures low and to get good resolution, while coarse cells are used in the rest of the domain to reduce the total costs for the simulation.

Grid refinement is ideal for ship flow simulation. The flow around a ship contains several different flow phenomena that are highly local in nature. The first of these is the water surface; if this surface is modelled with any type of capturing technique, then good grid resolution at the surface is essential for the accurate computation of the ship’s wave pattern. Also the orbital velocity fields of waves are local phenomena. And lastly, many types of vorticity, like bilge or keel vortices interacting with the propeller or the rudder, consist of local very strong gradients.

Thus, adaptive refinement is an interesting technique for simulating these flows. It can however be seen from this example that the local flow phenomena associated with a ship are fundamentally different. To be successful for ship flow, the adaptive refinement technique must work for all these different types of flows.

An automatic mesh adaptation method has been developed for ISIS-CFD, the flow solver entirely developed by the CFD group of the Fluid Mechanics Laboratory. The goal of the development was to produce a method that can be used in daily practice for all the different applications of this code. Also, it should be easily combined with the existing capabilities of ISIS-CFD, notably complex geometries, steady and unsteady flows, free-surface capturing, and grid deformation for the imposed or resolved motion of ship hulls. Furthermore, the method needs to be easily maintainable as the code develops over the years. For all these reasons, the mesh adaptation method has been made as general as possible: it allows unstructured grids, directional refinement to keep the size of 3D refined grids low, and derefinement of refined grids to enable unsteady flow simulation. The method is flexible to allow the easy changing of refinement criteria (that indicate where the grid is to be refined) and, like the flow solver, it is completely parallel. The refinement method is integrated in the flow solver.

This paper describes the application of the grid adaptation method to two very different test cases, in order to indicate its versatility and to show various mesh refinement
strategies. First, the technique underlying the grid adaptation method is outlined. Then, the double-body flow around the KVLCC2 tanker is computed. In this test case, a locally adapted fine grid is created entirely using automatic refinement, in order to compute accurately the flow in the propeller plane. The goal of this test is to show, that accurate solutions can be obtained on grid with less cells than fine grids created by hand, and that these grids can be created without requiring expert user knowledge.

Finally, the HALSS fast trimaran model test is used to compute the resistance in calm water at both model and full-scale for various trimaran configurations. The objective of this last case is to automatically optimise the grids around the water surface and to confirm that favourable interference can offset the side hull drag as shown from towing tank tests and from computations based on non-refined grids.

THE ISIS-CFD FLOW SOLVER

ISIS-CFD, developed by the CFD group of the Fluid Mechanics Laboratory and available as a part of the FINE™/Marine computing suite, is an incompressible unsteady Reynolds-averaged Navier-Stokes (URANS) method. The solver is based on the finite volume method to build the spatial discretisation of the transport equations. The unstructured discretisation is face-based, which means that cells with an arbitrary number of arbitrarily shaped faces are accepted. A detailed description of the solver is given by (Duvigneau and Visonneau, 2003) and (Queutey and Visonneau, 2007).

The velocity field is obtained from the momentum conservation equations and the pressure field is extracted from the mass conservation constraint, or continuity equation, transformed into a pressure equation. In the case of turbulent flows, transport equations for the variables in the turbulence model are added to the discretisation. Free-surface flow is simulated with a multi-phase flow approach: the water surface is captured with a conservation equation for the volume fraction of water, discretised with specific compressive discretisation schemes, see (Queutey and Visonneau, 2007).

The method features sophisticated turbulence models: apart from the classical two-equation $k-\varepsilon$ and $k-\omega$ models, the anisotropic two-equation Explicit Algebraic Stress Model (EASM), as well as Reynolds Stress Transport Models, are available, see (Deng and Visonneau, 1999) and (Duvigneau and Visonneau, 2003). The technique included for the 6 degree of freedom simulation of ship motion is described by (Leroyer and Visonneau, 2005). Time-integration of Newton’s laws for the ship motion is combined with analytical weighted or elastic analogy grid deformation to adapt the fluid mesh to the moving ship. Furthermore, the code has the possibility to model more than two phases. For brevity, these options are not further described here.

REFINEMENT TECHNIQUE

During a flow calculation with adaptive grid refinement, the refinement procedure is called repeatedly to keep the grid permanently adapted to the developing solution. Globally, the method works as follows: the flow solver is run on an initial grid for a limited number of time steps. Then the refinement procedure is called. If a refinement criterion, based on the current flow solution, indicates that parts of the grid are not fine enough, these cells are refined and the solution is copied to the refined grid. On this new grid, the flow solver is restarted. Then the refinement procedure is called again, to further refine or to derefine the mesh. This cycle is repeated several times. When computing steady flow, the procedure eventually converges: once the flow starts to approach a steady state and the grid is correctly adapted to this state according to the refinement criterion, then the refinement procedure keeps being called, but it no longer changes the grids.

Data structure

The refinement is cell-based. At this moment, it is limited to unstructured hexahedral cells, but the code is written such, that other cell types can be easily included. The grid data structure of ISIS-CFD, that uses node coordinates and pointers between cells, faces, and nodes, is used as the basis for the grid refinement. The number of extra pointers is limited, the main addition is a system of cell family ties. These store the history of the refinement, so refined cells can be derefined again to recover the original grid. When a cell is refined, all the new small cells get a ‘mother’ pointer to the old big cell and ‘sister’ pointers to each other. Thus, the group can be found again later and derefined back into the single large cell. The large cell is saved as a ‘dead’ cell, that has no faces nor a state vector, only family ties. Thus, it conserves its own sisters and mother (probably ‘dead’ as well), in case it has to be derefined itself, after being restored. Family ties for other grid elements (faces, nodes) are not used, as their refinement history can be reconstructed from that of the cells.

Refinement decision

For maximum flexibility and to reduce the chance of errors, the code is divided in three separate parts, that exchange only minimal information between them: the calculation of the refinement criterion, the refinement decision, and the actual (de)refinement. The refinement cri-
The refinement criterion indicates where the grid is to be refined; it can be based on any desired aspect of the flow solution. To permit the user choice of refinement criteria and the easy incorporation of new refinement criteria in the code, the criterion is computed as a field variable (comparable to the velocity or the pressure). It does not depend explicitly on the type or the orientation of the cells. A more detailed description of the refinement criterion computation is given in a following section.

In the second step, this refinement criterion is transformed into the decision of which cells to refine or to derefine. While this decision may depend on the type of the cells, it does not depend on the specific way in which the refinement criterion is calculated. It remains the same for any refinement criterion.

During the refinement decision step, the decision in each cell is adapted to its neighbour cells. To guarantee the quality of the mesh, extra cells may need to be refined, or derefinement of cells may be prevented. The most important quality criteria are given in Figure 1. A face of a cell may not be divided two times, which would cause too great differences in the sizes of its neighbour cells. And the angle between face normals and lines cell centre - face centre may not be too great: this reduces the quality of the state reconstruction at the faces, as indicated by (Queutey and Visonneau, 2007).

![Figure 1: Grid quality criteria. Forbidden are: (a) faces that are divided twice, (b) too large angles between face normals and lines cell centre - face centre. The images represent 2D examples.](image)

Finally, specific measures are taken in order to protect the wall-aligned boundary layer meshes, if these are present. In general, to keep a good grid quality for these high-aspect ratio cells, refinement parallel to the wall is not permitted. Also, the refinement in each column of cells is made the same, so that the boundary layer grid conserves its column structure.

At the end of the decision step, before a single cell is refined, the refinement of the whole mesh is known; this makes the actual refinement much easier.

**Refinement**

The actual (de)refinement is done cell by cell. Care is taken, after the treatment of each individual cell, to leave a valid mesh with all the pointers between cells, faces, and nodes in place, even if one knows that certain pointers will be changed again when a neighbour cell is refined later on. This guarantees that, when a cell is refined, it does not have to distinguish between neighbour cells that are refined, that will be refined, or that remain unchanged. The added flexibility and robustness of the code are well worth the extra work this represents.

Furthermore, in the code, the refinement of cells and faces is completely decoupled. These parts exchange only minimal information; a face does not need to know all the details of the refinement of its neighbour cells. This greatly facilitates unstructured-grid and directional refinement. Another advantage comes from the parallelisation of the algorithm: as the grid is divided in several blocks, the faces between these blocks exist in two blocks at once. In these two blocks, the face must be refined in exactly the same way. This becomes easier when the refinement of the face is based on the properties of the face itself and not strongly related with the refinement of a neighbour cell (that is not present in the two blocks).

Finally, the derefinement and refinement procedures are separated, although they are very similar. Derefinement is performed first.

**Parallel redistribution**

For efficient parallel flow computation, automatic redistribution of the cells over the processes is included in the grid adaptation procedure. This redistribution is completely integrated, it is performed between the derefinement and refinement steps; at that moment, the grid is at its smallest. Due to the preceding calculation of the refinement decision, the final topology of the grid is already known then. Using this information, the grid is first repartitioned in parallel with the ParMeTiS library, see (Karypis and Kumar, 1999).

The next step is the actual displacement of cells between the processes. This step is complicated by the strictly local numbering used in each block. To solve this problem, each block is domain-decomposed itself into sub-blocks to be dealt with by specific processes. The data structure for each sub-block is the same as the structure of the blocks and each individual sub-block gets its own local numbering. These sub-blocks are exchanged between the processes by MPI (Message Passing Interface) and then concatenated to produce the new blocks.
REFINEMENT CRITERIA

The refinement criterion is an important part of the algorithm, as it controls the location of the refinement and the shape of the refined cells. The refinement criterion has to be carefully chosen depending on the flow problem that is simulated. In order to offer a flexible framework for the specification of anisotropic grid refinement, we developed a metric-based method of criterion evaluation, in which the refinement criteria are specified as tensors. In this section, this method is explained first, then three different refinement criteria for ship flow are introduced: a criterion that refines at the water surface, a criterion based on solution gradients, and one based on Hessian matrices.

Metric-based criteria

Our refinement criterion computation is based on metrics. This technique has been used often for the generation, and the adaptive refinement, of unstructured tetrahedral meshes (see e.g. (George and Borouchaki, 1998) and (Alauzet and Loseille, 2010)). For the unstructured hexahedral meshes that we use, it is the most flexible way of specifying any type of anisotropic refinement.

In the method, the refinement criterion is computed as a field of $3 \times 3$ SPD tensors. The metric tensor $C_i \in \mathbb{R}^{3 \times 3}$ in each cell is considered as a geometric operator that transforms each cell $\Omega_i$ in the physical space into a deformed cell $\tilde{\Omega}_i$ in a modified space. Then, in each cell, directional refinement is applied to produce a grid that is as nearly uniform as possible in the modified space (see Figure 2). Using suitable tensors $C_i$, desired cell sizes in any given direction can be specified.

Free-surface criterion

Our first criterion refines in the neighbourhood of the water surface. Directional refinement is employed to refine the grid in the direction normal to the surface only. Where the free surface is diagonal to the grid directions, isotropic refinement is used, but where the surface is horizontal, directional refinement is chosen; the resulting zone of directional refinement includes the undisturbed water surface, as well as smooth wave crests and troughs. This is essential to keep the number of grid cells low, as the water surface is often nearly undisturbed in most of the domain. Figure 3 gives an illustration of this refinement principle.
explained above, this poses no problems from a numerical point of view.

**Gradient criteria**

A second group of refinement criteria is based on the absolute values of the gradients of solution quantities in each cell. These criteria detect the regions where the flow field changes rapidly; they react to most features of a flow and are thus more general than the free-surface criterion. Initially, these criteria have been applied to single-fluid flows.

Three criteria are chosen, one based on the gradient of the pressure:

\[ \text{Crit}_p = \sqrt{\left(\frac{\partial p}{\partial x}\right)^2 + \left(\frac{\partial p}{\partial y}\right)^2 + \left(\frac{\partial p}{\partial z}\right)^2}, \]  

(1)

one on the gradients of the three velocity components:

\[ \text{Crit}_v = \sqrt{\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial u}{\partial y}\right)^2 + \left(\frac{\partial u}{\partial z}\right)^2 + \left(\frac{\partial v}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial v}{\partial z}\right)^2 + \left(\frac{\partial w}{\partial x}\right)^2 + \left(\frac{\partial w}{\partial y}\right)^2 + \left(\frac{\partial w}{\partial z}\right)^2}, \]  

(2)

and one on the vorticity:

\[ \text{Crit}_\omega = \sqrt{\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right)^2 + \left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)^2}. \]  

(3)

Effectively, the vorticity can also be seen as a norm of the velocity gradients. These three are all scalar criteria: in each cell, the criterion gives only one cell size. In the context of metric-based refinement, this means that the refinement target is, to have the same cell size in all directions. Therefore, the refinement tensors \( C_i \) in each cell are the identity matrix \( I \), multiplied by the value of Crit.

The main difference between the pressure gradient criterion and the two velocity-gradient based criteria is their effect on boundary layers. There, the pressure varies little, while the velocity gradients are very strong. Thus, the velocity-gradient based criteria refine everywhere in boundary layers, but the pressure gradient criterion only there where the external flow creates a pressure gradient. In those cases where the flow outside the boundary layers is the main interest of the computation, costly and unnecessary refinement in all boundary layer cells can be prevented by using the pressure gradient criterion. An example will be shown in the next section.

**Hessian-based refinement**

The Hessian matrix of second spatial derivatives of the solution is a common choice as a refinement criterion. Using the same reasoning as above, we base this criterion on the pressure field, so it reacts to wave fields and to pressure-induced flow features at the hull, but not directly to the presence of boundary layers.

\[ C_i = \begin{bmatrix} p_{xx} & p_{xy} & p_{xz} \\ p_{yx} & p_{yy} & p_{yz} \\ p_{zx} & p_{zy} & p_{zz} \end{bmatrix}. \]  

(4)

The major difficulty in using the Hessian tensor as a refinement criterion is the accurate evaluation of the second derivatives, independent of the mesh. If the criterion computation is perturbed by local grid refinement, it may react more to existing grid refinement than to the pressure field. To prevent this undesired effect, numerical errors in the computed second derivatives must be significantly smaller than the derivatives themselves in all cells.

A particular problem associated with unstructured hexahedral meshes, is that the grid remains irregular when it is refined. For structured grids, and even for most unstructured tetrahedral meshes, when the grid is refined the cells get more and more the same shape and size as their neighbours. On unstructured hex meshes however, there will always be cells that are two times smaller than their direct neighbours. This means, that numerical schemes which rely on mesh regularity to get good accuracy are not suited for these meshes; a useful scheme must give sufficient accuracy for arbitrary cell configurations.

For the computation of second derivatives, we use a least-squares method based on third-order polynomials. In each cell, the polynomial is computed that best fits the pressure in the cell, its neighbours and its neighbours’ neighbours, in the least-squares sense. The approximated Hessian is constructed from the second derivatives of this polynomial. The least-squares procedure guarantees that the difference between the approximating polynomial and the real pressure is not in the space of third-order polynomials; therefore, it is at least fourth-order. Hence, the approximated second derivatives are second-order accurate, independent of the mesh geometry. For simpler methods like the well-known Gauss integration, this cannot be guaranteed.

**KVLCC2 TANKER**

The first test that is computed here, is the double-model flow around the KVLCC2 (KRISO Very Large Crude Carrier) tanker. The goal of the test using this well-known test case, is to study the gradient and pressure Hessian criteria and their capacity to create efficient fine meshes when starting from a very coarse initial mesh.

The study is concentrated on the flow in the propeller plane. The flow in this plane is strongly influenced by the vorticity that is generated on the ship hull and shed.
into the fluid at the rear of the ship, where the hull tapers into the stern. This vorticity generates a non-uniform flow field, that is an essential input for good propeller design and a key flow feature in hull shape optimisation for propeller efficiency. It is known that, while the resistance of the KVLCC2 tanker is accurately predicted by RANS methods on coarse grids, a fine grid of at least 1M cells is needed to resolve correctly the propeller plane flow. Moreover, this grid must contain local refinement near the stern; the result is sensitive to the correct placement of this fine zone. Thus, if the grid is to be created by hand using standard grid generator software, expert knowledge is required from the person generating the grid.

We shall compute the propeller plane flow with automatic grid refinement, by starting from a coarse grid without any local refinement. The ultimate target is to prove that similar accuracy can be obtained using less cells than grids refined by hand, using in particular the possibility of directional refinement to reduce the number of refined cells there, where the flow is aligned with a grid direction. Alternatively, we strive to demonstrate that similar accuracy can be obtained with the same number of cells on grids refined by hand, but without requiring the experience needed to generate these grids.

**Test case, setup, and grids**

The test case is the KVLCC2 hull at model scale, $Re = 4.6 \cdot 10^6$, in double-body configuration and without propeller; the computations are performed with a single fluid (water) and a symmetry boundary condition is imposed at the position of the undisturbed water surface. This test is one of the cases from the Gothenburg 2000 workshop, see (Larsson et al., 2000). Only half of the hull is meshed, so a symmetry boundary condition is imposed also on the hull $Y$ symmetry plane. Concerning the far field, the velocity is imposed on the inflow boundary, the pressure on the sides and the outflow boundary.

The initial coarse grid is an unstructured hexahedral mesh, generated with the HEXPRESS(TM) grid generator. The cells on the hull have sizes parallel to the wall of $L_{pp}/125$, which is at least twice as coarse as usual. In the direction normal to the wall however, given our boundary layer refinement strategy (see above), cells have the small sizes required for accuracy. For the wall-law boundary condition used in ISIS-CFD, the first cells are chosen to give $y^+ = 30$. The original grid has 58k cells.

Turbulence is modelled both with the two-equation $k-\omega$ SST model of (Menter, 1993) and with the anisotropic two-equation EASM model (Deng and Visonneau, 1999). We have shown in the past, that anisotropic turbulence modelling is required in order to correctly model vortex roll-up in ship aft-body flows. Here, we shall compare the two turbulence models in order to confirm this conclusion for automatically refined grids.

**Gradient criteria**

Initially, the flow is computed using the gradient-based refinement criteria introduced in the previous section. These criteria can be easily computed and give smooth, gradual refinement. However, due to their nature they can only be used with isotropic refinement.

A first comparison is made between the three criteria, using the $k-\omega$ SST turbulence model. To obtain a reasonable comparison, the computation for each criterion is started from the converged solution on the original 58k cells grid. Then, the grid after one refinement step is studied. The refinement threshold, which indicates the lowest value of the criterion that gives refinement, is adjusted for each criterion until the number of cells per grid after one refinement step are equal. Once these three thresholds are found, the computations and refinement steps are continued until the solution and the grid converge. In order to prevent excessive refinement in small regions with very high gradients, the number of times that an original cell can be refined is limited to two. The resulting grid sizes are 630k cells for the velocity gradient criterion, 620k cells for the vorticity criterion and 803k cells for the pressure gradient criterion.

Results in the propeller plane are shown in Figure 4. The ship hull can be seen at the top of these figures, the propeller would be placed below the hull. The most striking feature of the flow, as seen in the experimental result (Figure 5), is the hook-shaped region of low axial velocity caused by the roll-up of vorticity created on the aft hull. In the computation on the original coarse grid, this hook shape is completely absent. On the refined grids, a tendency towards a hook shape can be seen, which is most pronounced for the pressure gradient criterion.

Looking at the grids in the cross-section, it can be seen that all three criteria detect the gradients that appear in the propeller disk area. However, the pressure criterion refines much more than the other two. This is because the velocity and vorticity criterion react to the strong velocity gradients in the boundary layer and thus refine the boundary layer grid over the entire ship length, even in the middle section where the flow is nearly uniform in $X$-direction and where refinement is thus not necessary. (It is interesting to see that these two criteria, while computed in a different way from the velocity gradients, give nearly identical results.) The pressure gradient in the middle part of the boundary layer is close to zero, so for a given number of cells the pressure gradient criterion refines more in the region around the bow and the stern of the ship. As a result, the grid in the propeller disk is finer and the
flow there is resolved better: it shows the beginning of a hook shape and also in the region away from the symmetry plane, the solution is smoother and closer to the experimental results.

Figure 4: KVLCC2 tanker, cuts in the propeller plane at $x/L = 0.0175$. Grid cross-sections and axial velocity $u/U_\infty$ isolines are shown on the original coarse grid (58k cells) and the refined grids for the velocity gradient criterion (630k cells), the vorticity criterion (620k cells) and the pressure gradient criterion (803k cells).

Figure 5: KVLCC2 tanker, measurements of the axial velocity $u/U_\infty$ in the propeller plane at $x/L = 0.0175$, from (Larsson et al., 2000).

Figure 6: KVLCC2 tanker with the EASM turbulence model. Grid cross-sections and axial velocity $u/U_\infty$ isolines are shown on the original coarse grid (58k cells) and the refined grid for the pressure gradient criterion (1.07M cells).

The remaining differences between the computation and the experiment are mostly due to the turbulence model. To study the effect of this model, the $k-\omega$ SST model is replaced by EASM. Computations with this turbulence model are performed for the original coarse grid and for the pressure gradient criterion, using the same threshold as before. The result is given in Figure 6. The refined grid is similar to the grid for the pressure gradient criterion and the $k-\omega$ SST model. However, due to the more pronounced roll-up, the pressure gradients are slightly stronger so the refined grid has more cells: 1.07M as opposed to 803k. Concerning the flow solution, on the original grid the result has changed very little. But on the refined grid, the hook shape is now clearly visible and in good agreement with the experiment. Overall, the quality of this solution is excellent.

**Hessian criterion**

A first computation has been made with the criterion based on the Hessian of the pressure, retaining the EASM turbulence model. With respect to the pressure gradient criterion, this criterion has two advantages. First, the second spatial derivatives of the solution are generally considered to be a better indicator of the local error than the solution gradients. And second, the Hessian is a tensor criterion so it allows directional refinement, which means that elongated cells can be produced in those re-
regions where the flow is uniform in one direction. This reduces the overall number of cells.

Figure 7: KVLCC2 tanker, cross-sections of the grid based on the Hessian criterion at $Y/L = 0.02$ at the front of the ship (top) and at the back (bottom), showing isotropic and directional refinement.

Two cuts through the refined grid can be seen in Figure 7. As the Hessian criterion is based on the pressure, it does not react to the presence of the boundary layer on the middle part of the hull. In the zones of strong pressure variation near the bow and stern the flow is diagonal to the grid directions and the pressure varies in all directions, so isotropic refinement is predominant. Closer to the middle of the ship, the flow no longer varies strongly in $X$-direction, so the refinement is mainly directional. The same can be seen in the wake flow behind the ship.

We can also see that the Hessian-based criterion is more sensitive to small perturbations than the pressure gradient criterion (despite the evaluation using third-order polynomials): the grid is slightly less smooth. Overall however, the grid quality is good.

Figure 8: KVLCC2 tanker, grid cross-sections and axial velocity $u/U_\infty$ isolines for the refined grid based on the Hessian criterion.

In the propeller plane, Figure 8, the grid refinement is concentrated in the zone of the propeller disk, where the pressure changes most strongly. To the exterior, less refinement is applied than for the pressure gradient, so the solution is not as well resolved there. However, the hook shape is perfectly resolved.

The size of this grid is 600k cells, compared to the 1.07M for the pressure gradient criterion, but the solution quality in the propeller disk is equivalent. This indicates the effectiveness of the directional refinement procedure.

HALSS TRIMARAN

The second case studied concerns a large trimaran ship, the Heavy Air Lift Support Ship (HALSS) tested at NSWCCD, where interference between the hulls has a major impact on the wave making resistance. The HALSS concept design (Mizine and Amromin, 1999) is a 35-knot ship with the following main dimensions: flight deck length 335m, flight deck width / docking hull beam 83.5m / 54.9m, draft 11.5 m. A general view of the HALSS is shown in Figure 9.

Figure 9: General view of the HALSS concept.

The hydrodynamic research and model testing showed large changes in resistance due to moderate shifts in the longitudinal position of the side hulls. Also, a strong influence of the skegs on the stern flow was observed. In order to understand the factors leading to the interference effects, several earlier CFD studies have been performed (Mizine and Karafiath, 2008), (Mizine et al, 2009).

It is a common point of view (Begovic et al, 2005), (Doctors and Scrace, 2003) that successful design of the trimaran hulls is accomplished when the interference drag between hulls is zero. This means that the resistance of the trimaran is equal to the resistance of the centre hull plus resistance of the side hulls if each is operating alone. The model tests of the HALSS design showed however that favourable interference can offset the side hull drag, leading to a total resistance that is lower than the summation of the individual hull resistances.

In this section, the experimental model test is presented first, then results of computations on non-refined grids suitable for the prediction of the forces and main characteristics of the flow field will be presented. Some selected configurations are used to detail the influence of
Table 1: Limited list of experiments.

<table>
<thead>
<tr>
<th>Number</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Centre Hull only No Skegs</td>
</tr>
<tr>
<td>3</td>
<td>Centre Hull only Hull &amp; Skegs</td>
</tr>
<tr>
<td>8</td>
<td>Centre Hull and Side Hulls Hull &amp; Skegs, SH. Fwd.</td>
</tr>
<tr>
<td>5</td>
<td>Centre Hull and Side Hulls Hull &amp; Skegs, SH. Mid.</td>
</tr>
<tr>
<td>9</td>
<td>Centre Hull and Side Hulls Hull &amp; Skegs, SH. Aft</td>
</tr>
</tbody>
</table>

Table 1 lists the experimental conditions used in this article; for the trimaran cases, the side hulls are in the innermost transverse position. A complete description of the Model 5651 tests is detailed in (Mizine et al, 2009).

A photograph of the HALSS model experiment at speed 35 kts is shown in Figure 10 for the case referenced as Experiment 9. In this photograph it is clearly seen that the free surface can have strong deformations with breaking transom waves. This requires adequate RANS CFD methods because wave breaking is a real physical phenomenon that influences the optimization of multihulls.

Simulations on non-refined grids

The computed cases studied have a ship waterline length (LWL) of 289.8m and a ship draft of 11.5 m for the centre hull and a draft of 7.5 m for the side hulls. The corresponding model scale has a ship waterline length of 5.367m for the centre hull and 3.271m for the side hulls. The pitch and heave are prescribed from the values obtained during the experimental campaign. Due to symmetry considerations, only the Y>0 part of the model is meshed with the help of the HEXPRESS(TM) automatic grid generator. Figure 11 shows the computational domain around the half-trimaran with the free surface at rest represented by and internal refinement surface on which specific target cell sizes are prescribed.

In the following, FS and MS indicate full scale simulations and model scale simulations, respectively. For normalised quantities, lengths are normalised by LWL and speeds are normalised by the speed of the considered case.

While the experimental speed range is from 10 knots to 45 knots, only five speeds between 25 knots and 40 knots were selected for the numerical study. The model scale speed for the 25 knots case is 1.7485 m/s and 2.8006 m/s for the 40 knots case. Considering the Reynolds numbers, Table 2, the 35 knots design speed was selected to build a unique mesh for a given trimaran configuration at all speeds. The y+ = 30 constraint on wall functions requires meshes of about 2.8M cells (hull+skeg+side hulls) for model scale simulations with 8 viscous layers inserted in the Euler mesh. For the full scale simulation, the wall functions method requires a y+ = 300 corresponding to a...
Table 2: Characteristics of the computed ship speeds.

<table>
<thead>
<tr>
<th>Speed</th>
<th>Fr</th>
<th>Re(MS)</th>
<th>Re(FS)</th>
</tr>
</thead>
<tbody>
<tr>
<td>25 kts</td>
<td>0.241</td>
<td>9.78 × 10⁶</td>
<td>11.03 × 10⁶</td>
</tr>
<tr>
<td>30 kts</td>
<td>0.289</td>
<td>11.73 × 10⁶</td>
<td>13.24 × 10⁶</td>
</tr>
<tr>
<td>32 kts</td>
<td>0.309</td>
<td>12.54 × 10⁶</td>
<td>14.12 × 10⁶</td>
</tr>
<tr>
<td>35 kts</td>
<td>0.337</td>
<td>13.69 × 10⁶</td>
<td>15.44 × 10⁶</td>
</tr>
<tr>
<td>40 kts</td>
<td>0.386</td>
<td>15.66 × 10⁶</td>
<td>17.65 × 10⁶</td>
</tr>
</tbody>
</table>

Forces will be presented in terms of normalised coefficients following the classical definitions:

\[ C_r = \frac{F}{0.5 \rho U^2 Sw}, \quad PE = F_X U, \quad (5) \]

\[ C_r = C_T - C_{f0}, \quad \text{with} \quad C_T = C_P + C_F, \quad (6) \]

in which \( C_r \) represents the normalised force coefficient \( F \). The axis \( X \) is aligned with the ship advancing direction and \( PE \) represents the power required to maintain the prescribed advancing velocity \( U \). \( Sw \) is the wetted surface, \( \rho \) is the specific mass of the water. The frictional resistance coefficient \( C_{f0} \) from the ITTC’57 correlation line will be used to compute and compare the residuary resistance coefficient, Equation 6, as reported experimentally. It will be derived from the computed total resistance coefficient \( CT \), from the computed pressure resistance coefficient \( CP \), and from the computed frictional resistance coefficient \( CF \).

The major objective of the computations is to see if the power efficiency due to resistance, \( PE \) in Equation 5, is in agreement with the value obtained from towing tank measurements on the model-scale ship and extrapolated to the full-scale trimaran. When full-scale simulations are performed, the power efficiency is directly derived from the computed total resistance force at the corresponding speed without any other approximations than those contained in the physical modelling. When model-scale simulations are considered, the extrapolation to full-scale of the power efficiency will be obtained by computing the resistance \( FX \) force given by the reverse Equation 5 using the full-scale wetted surface, velocity, and the computed residuary resistance coefficient which is supposed to be the same at full scale as at model scale.

**Influence of the skegs**

Together with the side hulls, the sternward extension on each side of the centre hull that supports the propeller shaft is a crucial element to investigate, since it implies severe modifications and constraints on the bare hull in the aft region. The flow field in front of the propeller area and the changes in resistance must be carefully evaluated for further possible improved designs.

The influence of the skegs has been investigated on the centre hull only from Experiment 1 (without skegs) and Experiment 3 (with skegs). The presence of the skegs increases the wetted surface by a factor 1.11. Table 3 compares the computed frictional, pressure, and total resistance coefficients for the two configurations at some selected speeds. First of all, one can notice that the viscous resistance is much larger than the pressure resistance for both configurations. However, for the design speed of 35 knots for instance, the pressure resistance represents 28% of the total resistance without skeg and 40% for the centre hull with skegs. Although the viscous resistance is not strongly affected by the presence of skegs, the pressure resistance may be multiplied by a factor 2 when skegs are included. At the design speed of 35 knots, the pressure resistance is much larger than the pressure resistance for both configurations. However, for the design speed of 35 knots, the pressure resistance is multiplied by a factor 1.7. One can also notice the relatively good agreement between the predicted viscous resistance and the values provided by the ITTC-57 correlation.

The corresponding residuary resistance \( Cr \) is presented in Figure 12 where one can notice a very satisfactory agreement between measurements and computations, except for the lowest speed of Experiment 3. The reliability of the computations is confirmed with the spectacular influence of the skegs especially for speeds ranging from 30 to 40 knots where the computed residuary resistance coefficient behavior is in agreement with the measured one.

Therefore, the main origin of the increased resistance is the modification of the pressure field. The comparison of the computed free-surface elevations in normalised form is given in Figure 13. The large difference in terms of resistance between Experiments 1 and 3 comes from a completely different behaviour of the free surface on the...
Table 3: Computed resistance coefficients (x 1000) at model scale for Experiments 1 and 3 at various speeds.

<table>
<thead>
<tr>
<th>Exp.</th>
<th>Speed</th>
</tr>
</thead>
<tbody>
<tr>
<td>1:</td>
<td>25</td>
</tr>
<tr>
<td>3:</td>
<td>25</td>
</tr>
<tr>
<td>1:</td>
<td>30</td>
</tr>
<tr>
<td>3:</td>
<td>30</td>
</tr>
<tr>
<td>1:</td>
<td>32</td>
</tr>
<tr>
<td>3:</td>
<td>32</td>
</tr>
<tr>
<td>1:</td>
<td>35</td>
</tr>
<tr>
<td>3:</td>
<td>35</td>
</tr>
<tr>
<td>1:</td>
<td>40</td>
</tr>
<tr>
<td>3:</td>
<td>40</td>
</tr>
</tbody>
</table>

Figure 12: Evolution of the residuary resistance with ship speed for Experiments 1 and 3. Measured - lines with small symbols; computations - large empty symbols.

The drastic change in pressure distribution due to the presence of the skegs can also be clearly detected from a section-by-section analysis of the forces. The forces by section distribution is obtained from the integration of the forces (frictional, pressure, and total) on the trimaran surface, on a prescribed number of sections. For all the presented forces by section in this paper, 100 equi-distributed sections of width 0.01LWL are used. The distributions in Figure 14 correspond to Experiment 1 at 35 knots and contain the plots of the resistance coefficients, together with the intersection of the free surface with the trimaran configuration in the background. Figure 15 plots the same information for Experiment 3.

Up to X/L=0.6, the pressure distribution is similar between the two configurations, starting with an unfavourable pressure gradient until x/L=0.2 and the same unfavourable gradient until x/L=0.6 approximately. Further back, until x/L=0.85, the Experiment 3 configuration produces a less efficient pressure gradient than Experiment 1 does. Then a small pressure plateau occurs between 0.85L and 0.95L where the minimum of the water elevation appears on the hull (-0.009L), as can be seen from the narrow shape of the geometry on the same figure and from Figure 13. Past x.L=0.95L, the wave breaking takes place. Thus, the effect of the skegs is to considerably reduce the extent of the region of favorable pressure gradients, with a reduction starting early at x/L=0.65 approximately. This could have been observed from the wave elevation distribution considering the iso-0.0 as a reference on Figure 13, but not in a quantitative way as can be done from the longitudinal distribution of the pressure force.

While not illustrated, it was observed that the skeg strongly modifies the wall pressure distribution, by modifying the curvature of the hull, leading to the development of a longitudinal vortex at the extremity of this appendage, although no local flow reversal can be detected. Figure 16...
Figure 15: Forces by section: centre hull only with skeg - 35 kts.

shows the computed and measured velocity field at the starboard shaft at 30 knots immediately downstream of the end of the shaft. In the isowake distribution, one can clearly see the wake of the skeg and two contrarotative longitudinal vortices in the computations, which are hardly visible in the measurements due to the measurement accuracy. However, the good agreement of this local flow field between the measurements and the viscous simulations is very reassuring. Secondly, the modification of the pressure field has an impact on the free-surface elevation by creating, just behind the skeg, a deep trough followed by the strong breaking wave already mentioned.

Figure 16: Computed (left) and measured (right) isowake distribution and secondary velocities at the starboard shaft - Experiment 3, 30 kts.

Influence of longitudinal location of side hulls

The influence of the longitudinal location for the side hulls is presented using Experiment 8 for the forward longitudinal location, Experiment 5 for the middle longitudinal location, and Experiment 9 for the aft longitudinal location.

Excellent agreement between the measurements and the computations is observed in the trends and differences on the residuary resistance, which are remarkably captured by the computations, see Figure 17. When the speed is higher than 25 knots, the configurations of Experiments 8 and 9 which correspond to the forward and aft longitudinal locations, respectively, are characterized by an increase of about 300% of the residuary resistance. To analyse the origins of this dramatic increase, the two extreme computed speeds of 25 and 40 knots have been chosen to compare the characteristics of the waves. The free-surface elevations for the three longitudinal locations are shown in Figure 18 for a speed of 25 knots and in Figure 19 for a speed of 40 knots.

At 25 knots, for Experiments 5 and 9 corresponding to the middle and aft side hull longitudinal locations, there is no significant interaction between centre and side hull bow waves, contrary to the forward longitudinal location (Experiment 8) where this interaction is very strong even for this moderate speed. In this case, the strong interaction leads to more complex wave trains between the hulls, which contribute to increase the resistance. However, the forward longitudinal location has no impact on the stern breaking wave developing along the centre hull contrary to Experiment 9, for which one notices a very strong interaction between waves emanating from the side and centre hulls.

At 40 knots, the situation is completely different because of the magnitude and spatial extension of the bow wave created by the centre hull. A strong interaction between the bow waves of both hulls can be observed for the middle and forward longitudinal locations (Experiments 5 and 8), with a very spectacular internal wave train in the last case. In Case 8, the reflected wave on the centre hull is so intense that it leads to a rooster tail breaking wave behind the side hull, which may explain the higher residuary resistance observed both in the experiments and the...
Figure 18: Influence of the longitudinal location of side hulls at 25 knots - wave elevations for Case 5 (top of top and bottom figures), Case 8 (bottom of top figure) and Case 9 (bottom of bottom figure).

computations for this side hull position.

Figure 19: Influence of the longitudinal location of side hulls at 40 knots - wave elevations for Case 5 (top of top and bottom figures), Case 8 (bottom of top figure) and Case 9 (bottom of bottom figure).

Table 4 shows frictional, pressure, and total resistance coefficients at the design speed of 35 knots for the trimaran with side hulls in the three different longitudinal locations and also for the case without side hulls (Experiment 3). As computed for the cases without side hulls, the viscous resistance dominates the pressure resistance for all configurations. The only exception is Experiment 5 where the pressure resistance is only 20% of the total resistance and with a slightly lower frictional resistance than computed with the other longitudinal configurations.

The corresponding free-surface elevations at the design speed, compared with the result obtained for the centre hull only (Experiment 3) are given in Figures 20, 21, and 22. A common contour range is used based on Experiment 8 for the lowest elevation (-0.023LWL) detected between the side hull and the centre hull and the highest computed elevation (0.02LWL) detected behind the transom. The corresponding values for the computed Experiment 5 are approximately 1.5 times lower with -0.015LWL for the minimum and +0.0135LWL for the maximum. This gives a first explanation for the low pressure resistance of Experiment 5.

Table 4: Computed resistance coefficients (x 1000) at model scale for Experiments 1, 3, 8, 5, 9 - 35 kts.

<table>
<thead>
<tr>
<th></th>
<th>Simulation</th>
<th>CF[% of CT]</th>
<th>CP[% of CT]</th>
<th>CT</th>
</tr>
</thead>
<tbody>
<tr>
<td>EXP 3</td>
<td>2.913 [60%]</td>
<td>1.906 [40%]</td>
<td>4.820</td>
<td></td>
</tr>
<tr>
<td>EXP 8</td>
<td>3.205 [56%]</td>
<td>2.540 [44%]</td>
<td>5.745</td>
<td></td>
</tr>
<tr>
<td>EXP 5</td>
<td>2.982 [79%]</td>
<td>0.784 [21%]</td>
<td>3.765</td>
<td></td>
</tr>
<tr>
<td>EXP 9</td>
<td>3.050 [53%]</td>
<td>2.682 [47%]</td>
<td>5.733</td>
<td></td>
</tr>
</tbody>
</table>

Figure 20: Free-surface elevation: centre hull with skeg (top) and side hulls in forward longitudinal position (bottom) - 35 kts.

Figure 21: Free-surface elevation: centre hull with skeg (top) and with side hulls in middle longitudinal position (bottom) - 35 kts.

Favorable interference effects are effectively produced with the side hull in the middle location since less wave interference between the side and the centre hulls is found in that case. Moreover, the wave system outside the side
hull changes so that one wave length corresponds to about one ship length, compared to about 0.8 ship length for Experiment 3, Figure 21. Also, the amplitude of this divergent wave system is lower in Experiment 5 even when compared to the amplitude from Experiment 3 or 1.

To continue the design speed discussion, the longitudinal distribution of the forces is investigated following the approach already outlined for the skeg influence. Figures 23, 24, and 25 give the forces by section distribution for the three configurations. Since the computed resistance coefficient results from the integration over the surface of the ship, a high positive region observed in such a longitudinal distribution is not enough to conclude about its influence on the integrated coefficients since it is the result of the balance between positive and negative regions. This is not the case with the CF frictional coefficient when reverse flow is marginal.

For Experiment 8, Figure 23, the most significant observation is the considerable increase of the unfavourable pressure gradient between the fore region of the centre hull and part of the front region of the side hulls. This is related with the strong wave interference observed in the same region in Figure 20. The large reduction of the favourable pressure gradient starting at \( x/L = 0.4 \) corresponds to the presence of a deep trough (-0.020L) near the centre hull, centered at \( x/L = 0.45 \). Later, past the end of the side hull location, the favourable pressure gradient is still reduced. Concerning Experiment 9, Figure 25, since the side hull is shifted aft, the favorable pressure gradient starting at \( x/L = 0.2 \) from the centre hull becomes unfavourable at the beginning of the side hull so that the region of positive CP is higher in this case. For Experiment 5, Figure 24, the main characteristic of the longitudinal pressure coefficient is a quasi-linear behaviour in most of the region corresponding to the side hull location. This remarkable behaviour indicates that very little interaction takes place in that region, as already observed from the wave elevations.

Simulations on refined grids

An important point to underline for the trimaran study is, that the original grids used are not “coarse” original grids as for the KVLCC2 case. Instead, the non-refined grids of the earlier study are used. The experiment number 5 was selected at 35 knots to check the free-surface criterion for both MS and FS simulations. The refinement target is a grid spacing of about 0.0005LWL.

The solution is restarted from the original-grid solution. Figure 26 plots the resulting evolution of the number of cells during to the refinement process, related to the number of cells in the original grid. The refinement algorithm is applied every 25 time steps. While not fully completed due to resource limitations, the refinement process converges for both MS and FS. The refined grid contains about 4.5M cells for the MS simulation and about 7.5M cells for the wave elevations.
Free-surface elevations for Experiment number 5 at the 35 knots design speed are presented in Figure 27 for the model scale case with the original grid solution on top and the refined grid solution at the bottom. The same kind of comparison is presented in Figure 28 but for the full scale simulations. The most visible effect of grid refinement is in the improvement of the details of the free surface near the rising wave behind the transom. Grid refinement also confirms the very slight increase of the transom wave between model and full scale.

The effect of automatic grid refinement is clearly seen in Figure 29 showing the grid density on the free surface for the model scale case. The most evident observation is the increase of the grid density at and behind the skeg region where breaking wave occurs. The grid resolution is also improved near the waves emanating from the bow region and from the sharp ends of the side hulls.

Moreover, as seen in the enlarged view of Figure 30, the grid density is now improved to confirm that the divergent wave system outside the side hulls has a shorter wave length with lower amplitude compared to the result of Experiment 3, Figure 13 or Figure 21. As a consequence, less wave breaking induced by the skegs, longer wave lengths, and lower amplitudes mean that less energy is spent to deform the water surface with the side hulls in middle location.

Figure 32 shows the cut plane $X/L_{pp}=1.03$ where the wave breaking phenomenon close to the centre hull has been captured on the refined grid at model scale. Also improved is the capture of the wave generated by the side hull. This cut plane result is compared with the result obtained on the original grid, Figure 31. The free-surface criterion used is perfectly validated with respect to the required threshold. In some regions, for quality preserving reasons, the threshold value retained (0.0005$L_{WL}$) can even produce one step smaller isotropic cell sizes (0.00025$L_{WL}$); this is typically the case in wave breaking regions, see Figure 32. If an explicit refinement box (by hand) with isotropic target cell size of 0.00025$L_{WL}$ was used on the original grid near the free surface, restricted to the region where wave breaking occurs close to the centre hull, the additional grid size is roughly estimated as 20M cells. This has to be compared with the 1.6 multiplication...
factor produced by the grid refinement.

The effect of automatic grid refinement on the forces is given in Table 5 at both model and full scale. The full-scale coefficients are expressed as percent changes of the corresponding model-scale coefficients. The expected decrease of the frictional coefficient from model to full-scale is the same with the same values (less than 0.01%) on the refined grid. Only small adjustments are produced between the original and the refined grid for the pressure coefficient. The slightly equivalent effect of grid refinement on pressure resistance between model and full-scale (-2%) is diminished at model scale on the total coefficient since the pressure coefficient represents 35% of the total coefficient at model scale but only 21% at full scale.

Table 6 summarizes the predictions and direct computations of the power efficiency (PE) for the design speed of 35 knots with the wetted surface that depends on the considered case. When full-scale simulation (FS) is considered, the power efficiency is directly obtained from the integrated forces without any extrapolation. When model-scale simulation is considered (MS), the predicted power efficiency is extrapolated as for the towing tank test. Experiment 1 is used to quantify the effect of the skegs and Experiment 3 is used to quantify the influence of the location of the side hulls on the effective power.

Although Experiment 1 has not been computed at full scale.

<table>
<thead>
<tr>
<th>Simulation</th>
<th>CF</th>
<th>CP</th>
<th>CT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original: MS</td>
<td>2.982</td>
<td>0.784</td>
<td>3.765</td>
</tr>
<tr>
<td>Refined: MS</td>
<td>0%</td>
<td>-2.2%</td>
<td>-0.5%</td>
</tr>
<tr>
<td>Original: FS</td>
<td>1.390</td>
<td>0.744</td>
<td>2.134</td>
</tr>
<tr>
<td>Refined: FS</td>
<td>0%</td>
<td>-2.0%</td>
<td>-0.7%</td>
</tr>
</tbody>
</table>

Table 5: Computed resistance coefficients (x 1000) on original and refined grids for Experiment 5 - 35 kts.
Table 6: Power efficiency - 35 kts.

<table>
<thead>
<tr>
<th>Case</th>
<th>SW ($m^2$)</th>
<th>PE (kW)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EXP 1:Extrap.(MS)</td>
<td>9448</td>
<td>67085</td>
</tr>
<tr>
<td>EXP 1:Towing Tank</td>
<td></td>
<td>63805</td>
</tr>
<tr>
<td>EXP 3:Extrap.(MS)</td>
<td>10490</td>
<td>98614</td>
</tr>
<tr>
<td>EXP 3:Towing Tank</td>
<td></td>
<td>99521</td>
</tr>
<tr>
<td>EXP 8:Extrap.(MS)</td>
<td>16038</td>
<td>194165</td>
</tr>
<tr>
<td>EXP 8:Comput.(FS)</td>
<td></td>
<td>191599</td>
</tr>
<tr>
<td>EXP 8:Towing Tank</td>
<td></td>
<td>204035</td>
</tr>
<tr>
<td>EXP 5:Extrap.(MS)</td>
<td>16038</td>
<td>101554</td>
</tr>
<tr>
<td>EXP 5:Comput.(FS)</td>
<td></td>
<td>99977</td>
</tr>
<tr>
<td>EXP 5:Towing Tank</td>
<td></td>
<td>100681</td>
</tr>
<tr>
<td>EXP 9:Extrap.(MS)</td>
<td>16038</td>
<td>193567</td>
</tr>
<tr>
<td>EXP 9:Comput.(FS)</td>
<td></td>
<td>192229</td>
</tr>
<tr>
<td>EXP 9:Towing Tank</td>
<td></td>
<td>209406</td>
</tr>
</tbody>
</table>

In this paper, an automatic adaptive grid refinement method for ship flow computation has been applied to two very different test cases.

From the results of the KVLCC2 tanker case, we have seen that for the simulation of viscous double-model ship flow, pressure-based gradient or Hessian criteria are to be preferred due to their insensitivity to boundary layers. As in computations without grid refinement, the anisotropic EASM turbulence model gives much better results than the $k - \omega$ SST model, confirming that anisotropic turbulence modelling is required for this type of flow.

Simple gradient criteria are able to detect the relevant flow features. In general, the results are equivalent to those obtained on manually generated locally refined grids (good resolution requires about 1M cells). Yet, with the automatic grid adaptation procedure, the expert knowledge needed to manually create these grids is not needed. Hessian-based criteria, while sensitive to small perturbations, show a real potential of being able to generate equivalent results with less cells, thus reducing both the effort needed in grid generation and in computation time.

Concerning the HALSS model test and simulation results, it has been demonstrated that a trimaran can be designed such that favourable hydrodynamic interactions offset almost all of the side hull drag over a practical range of speeds. It was found that in case of unfavourable interference we are dealing in particular with stern/transom wave breaking. For the highest computed speeds, nonlinear effects are extremely large, which justifies having recourse to an accurate free-surface capturing viscous simulation. Due to the large deformation of the free-surface and the occurrence of local breaking waves, automatic grid refinement has proved to be an excellent tool to produce reliable simulations with quantitative agreement, by optimising the grids around the water surface in order to account for these important physical phenomena.

**CONCLUSION**

**ACKNOWLEDGMENTS**

The initial HALSS concept evaluation and test program work have been sponsored by Center for Commercial Deployment of Transportation Technologies (CCDOTT) and the Office of Naval Research (ONR). The FINE™/Marine calculations and analysis have been performed based on Cooperative Research Agreement between ECN/CNRS, CSC Advanced Marine Center, CCDOTT and ONR.

The French authors gratefully acknowledge the scientific committee of IDRIS (Institut du Développement et de Ressources en Informatique Scientifique du CNRS, project 000129) for the attribution of CPU time.
REFERENCES


