THE ROYAL INSTITUTION OF NAVAL ARCHITECTS

APPLICATION OF OpenFOAM® TO HULL FORM OPTIMISATION AT STX France

S Cordier, L Morand, J-M Roux, STX France S.A., Fr, and **E Cortey,** CFD-Numerics, Fr

SUMMARY

After over a decade of using potential flow solvers for hull form development and optimisation, STX France has sought to enhance its Computational Fluid Dynamics (CFD) capability to include a free surface, viscous flow solver. The approach developed at STX France SA with CFD-Numerics uses the OpenFOAM® software package which has been adapted to solve the problem of ship resistance. The hull form optimisation process also relies on the ability to efficiently make changes to the design. This function is provided through a CAD software which can be programmed for parametric variations. This was approach initially tested on a hull form with a high bloc coefficient, triple skeg hull form with a deep transom. It was then used to develop 2 very close hull forms, each optimised for a different speed. The computed resistance calculated is compared with model test resistance data for the 2 forms as well as potential flow results in terms of free surface elevation. The relative ranking of the 2 forms as well as the absolute value of resistance at model scale are well predicted.

1. INTRODUCTION

Computational Fluid Dynamics has been used in the development of ship hull forms since the early 1990's with the first panel methods which are still widely used today [1, 2, 3] to compute the flow field and wave system around a ship advancing at speed. The main limitation in modelling capability of these methods is the fact that viscosity is ignored which implies no viscous resistance, no vorticity generation except where prescribed (keels, rudders) when this option is available, and finally no natural separation on the hull surface or at the transom unless it is also explicitly prescribed. Often, the wave resistance value can only be obtained by analysis of the wave system which cannot be used as an estimate of the wave resistance unless experimental data exists for a very similar hull form. This limitation has prevented the use of these methods to produce explicit and reliable resistance predictions.

In this case, the optimisation process can only be performed by comparing the computed wave fields and/or the values of wave resistance of different hull forms. These codes (SHIPFLOW at STX France) are currently used in the ship hull optimisation process, in particular for bulb and stern shapes. They also have the advantage of that they require a very short computational time. However, for cruise vessels where the wave resistance is a small portion of the total resistance (10 to 20% at the speed used for optimisation), it is however important to have estimates of both viscous and wave resistance in order to be able to choose the right trade offs between the two resistance components.

Given the limitations of potential flow panel methods, the development of free-surface viscous flow solvers in the context of increasing computer capabilities and resources has become a priority. Different conferences such as the CFD workshops [4] and the Numerical Ship Hydrodynamics International Conference which have been organised regularly over the last decade have demonstrated the progress of these techniques in universities and specialised institutes.

In this context, STX France has sought a solution to compute free surface viscous flows in house for the purpose of developing its hull forms. Further advantages expected from the use of such a code are the possibility to compute the resistance and optimise the position and shape of appendages such as fins, bilge keels, struts and rudders, thrusters' openings, etc... This paper explains how the software used was selected and how it has been used to address the problem of the flow around a ship considering the issue of mesh generation.

In a hull form optimisation process, the flow solver is one component of a computational procedure which will also require a procedure to modify the hull form. Several methods have been described and implemented in order to generate a large number of hull forms for computation in an optimisation process [5, 6, 7, 8]. The use of a CAD software in conjunction with a numerical tool capable of programming parametric modifications of the hull form within a CAD environment is described.

The combined use of CFD with parametric CAD hull form deformation software along with a fast and robust meshing algorithm was used to develop 2 hull forms for the same ship, each one optimized for a different speed. The test results of the two hull forms allows us to verify the relative ranking ability of the solver as well as its ability to predict the ship resistance accurately.

2. CFD SOFTWARE

2.1 CHOICE of OpenFOAM®

OpenFOAM® is produced by OpenCFD Ltd and is freely available and open source, licensed under the GNU General Public Licence. The choice to adopt OpenFOAM® for the computation of the flow around a ship was influenced by cost concerns and also technical and strategic reasons.

Concerning the cost, running a open source code provides the capability to compute a large number of hull shapes simultaneously which is required in an optimisation process. The optimisation process becomes limited only by the computer resources available and not by the number of licences available. On the hardware side, the use of open source also enables easy access to outside computing capacity. Indeed, licenses are typically tied to the computer system which runs the application. Expansion of the computing capability would then require an investment in the hardware and maintenance expertise which are costly. In the case of open source, the code can be compiled on very powerful outside computer resources (in the order of 1000 nodes) which can be readily accessed through internet. A reasonably small in-house capability is nevertheless required to perform tests on new configurations.

However, it should be noted that although the license is free, there are neither detailed user's manual nor hotlines available to set up a problem. It is therefore the responsibility of the user to spend the necessary resources to find within the OpenFOAM® CFD environment the best solution for the problem at hand. Technically, numerous publications have been made which showed good results with this solver in the field of naval hydrodynamics [9].

Strategically, besides the guarantee of always being able to run the calculation procedure developed, STX France is a member of the CRS cooperative ship research group which has recently adopted OpenFOAM® as a CFD development environment for several hydrodynamic problems. It then becomes a necessity to be able to run in this environment.

2.2 COMPUTING STRATEGY

The simulation of free surface flow is a challenging goal for all marine applications. This requires some accurate numerical schemes to describe the interface by limiting as much as possible the numerical diffusion. For hull forms optimisation, an additional challenge comes from the need to move the hull and its mesh during the analysis to properly compute the equilibrium position, the forces on the hull and the waves system around the ship.

Free Surface Model:

There are two main methods to compute the free surface: the interface tracking method and the front capturing method. The former treats the free surface as a sharp interface whose motion is followed by moving the grid and the free surface. The latter, commonly used, is performed on a fixed grid and the shape of the interface is determined by the fraction of each near-interface cell that is partially filled. In this paper, the front capturing method has been used as a volume of fluid method ("VOF"). The "VOF" method is a two phase surface compression method that solves the Navier-Stokes equations and an advection transport equation for the volume fraction:

$$\frac{\partial c}{\partial t} + \nabla \cdot (c\vec{u}) = 0 \quad (1)$$

Where c is the volume fraction:
c = 1 in pure water
c = 0 in pure air.

The challenge is to solve the equation (1) by limiting the numerical diffusion and keeping a bounded solution. Several schemes have been developed for this purpose: the HRIC method from Peric [10] or the CICSAM method from Ubbink [11]. In OpenFOAM®, the convective-only equation (1) is solved using the dedicated scheme MULES (for "Multidimensional Universal Limiter for Explicit Solution"). The equations are solved using a PISO ("Pressure Implicit with Splitting of Operators") algorithm. One can get a sharp description of the interface as illustrated on Figure 1.

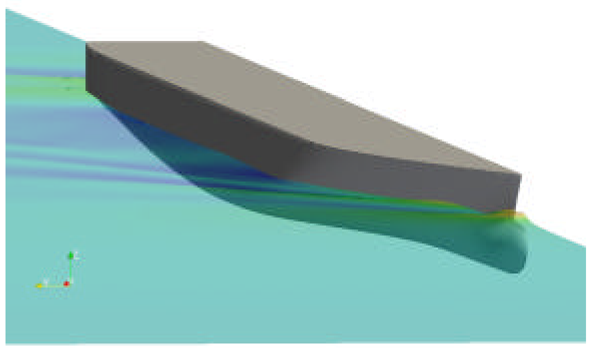

Figure 1: VOF method: sharp water/air interface

Sinkage and Trim:

OpenFOAM® contains a general 6DOF ("6 degree of Freedom") solver as illustrated on Figure .

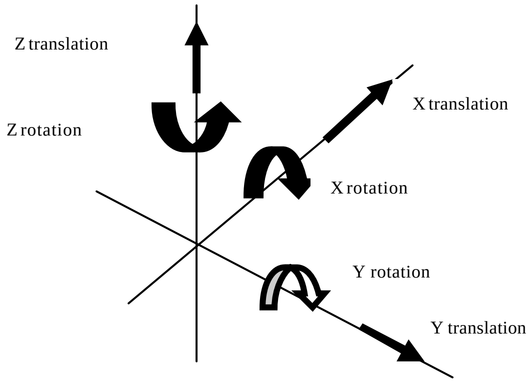

Figure 2: 6 DOF motions

In this paper we only consider two movements: trim and sinkage. To derive the displacement, we solve the dynamic equation by integrating over the body the forces:

$$F = \sum_S F_{ext} + F_{flow} \qquad (2)$$

$$M = \sum_S M_{ext} + M_{flow} \qquad (3)$$

Where F_{flow} are the pressure and viscous forces from the flow, F_{ext} is the gravity force since no other external efforts are taken in account. M is the momentum around the gravity centre (whose position has to be specified by the user as well as the inertial momentum).

3. MESH GENERATION AND MOTION

The mesh has been built using the embedded OpenFOAM® mesh generation application: SnappyHexMesh. The basic principle of this tool is to build a custom mesh with several refinements regions. This custom mesh is then snapped around the CAD geometry to get a body fitted mesh as illustrated on Figure .

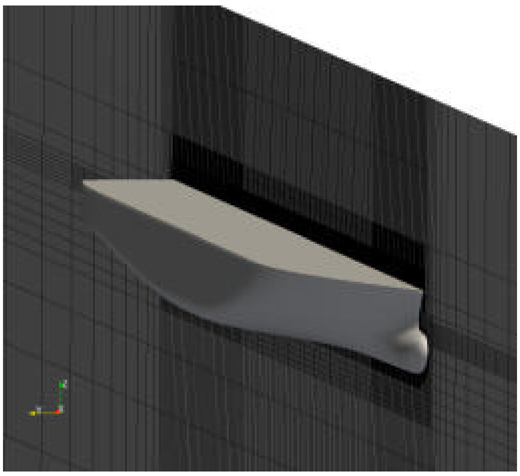

Figure 3-a: Hexahedral hull mesh

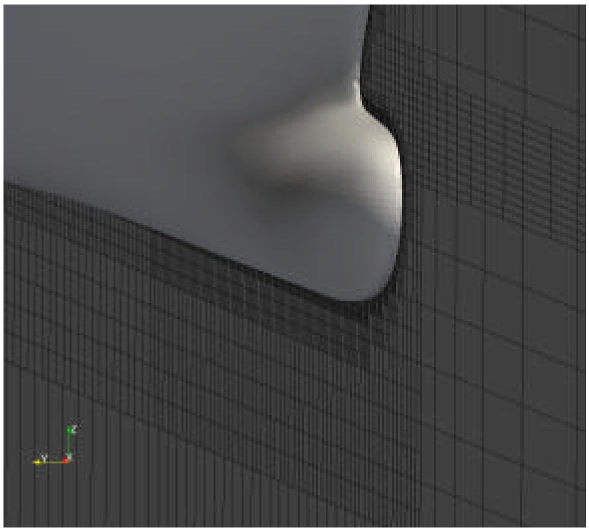

Figure 3-b: Close up of mesh around the bulb

Once the displacement of the hull is computed using the method described previously, the mesh around the hull is moved and a morphing algorithm is used to adapt the mesh. An example of mesh deformation algorithm is given in reference [12].

This meshing strategy has the advantage of being extremely robust and capable of generating meshes for complex hull forms such as in the example below which shows how the code is able to compute flows around complex after-bodies, here a triple skeg vessel with two gondolas and a centre-plane skeg (figure 4). This hull form also sails with a large zone of flow recirculation at the stern. These features are typically difficult or impossible to compute with potential flow methods or RANS methods using structured grids.

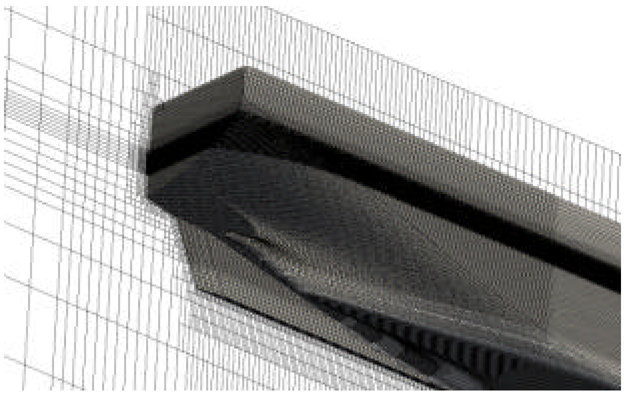

Figure 4: Mesh of a twin gondola hull form with centre skeg

4. HULL MODELLING STRATEGY

The process of hull optimisation requires the capability of modifying the hull form under the different constraints imposed by the architecture of the vessel such as main dimensions, displacement, LCB, stability, propulsion motor and propeller integration, etc... These modifications can be performed either using a CAD software, or directly by deformation of the computational grid using specifically developed functions.

The first solution was pursued for two principal reasons:

? The CAD environment can more easily be used to enforce the design constraints

? At the end of the optimisation process a CAD definition of the hull form is directly available

The use of CAD deformations can be made more efficient if they can be parameterized so that complex geometrical changes may be performed with a reduced number of parameters, preferably chosen to correspond to those used by naval architects.
For this purpose, the Rhino modelling software was selected in conjunction with RhinoScript and Grasshopper which are two complementary development environments tightly integrated with Rhino's 3-D modeling tools.

4.1 RHINO

Rhino is a very well established CAD package which has found widespread use for the description of complex shapes such as hull forms. Furthermore, different "plug-ins" have been developed which tailor to specific needs such as those of naval architects with products such as ORCA which computes the hydrostatics of the hull forms.

4.2 RHINOSCRIPT

RhinoScript is Rhino's embedded scripting language that make possible the development of complex sequences of commands to apply incremental modifications to the hull shape in different ways, export files and then generate all specific outputs for CFD solvers.

4.3 GRASSHOPPER

Another product developed for Rhino is the Grasshopper application which allows the development of very sophisticated functions which can be programmed through a graphical interface.
This promising young environment is actively developed and allows us the real-time generation of full parameterized geometric objects such as hull surfaces or appendages (Figure 5).

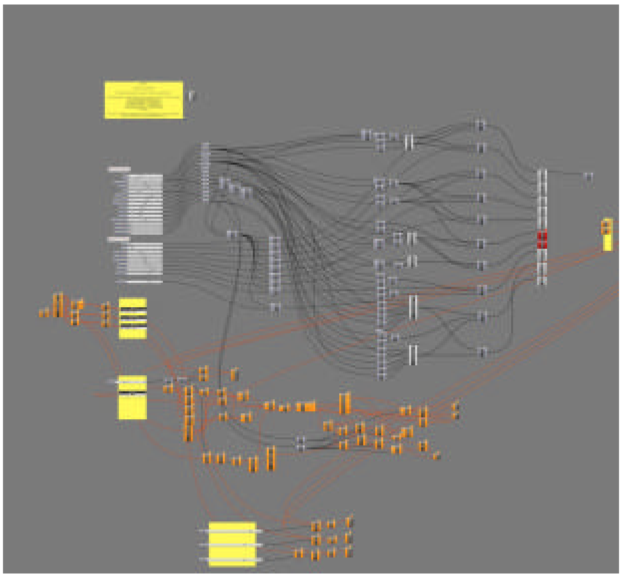

Figure 5: Grasshopper script for hull form generation

4.4 EXAMPLES OF DEFORMATION

Using the combination of Rhino and Grasshopper it is possible to generate variations of different features of an initial hull forms in order to perform parametric studies based on a limited set of parameters and used for identify optimum hull shapes. One of the deformation scripts is illustrated below (figure 6) for the bulb of the ship where the waterline length, the height or the width of the bulb can be systematically varied.

Figure 6-a

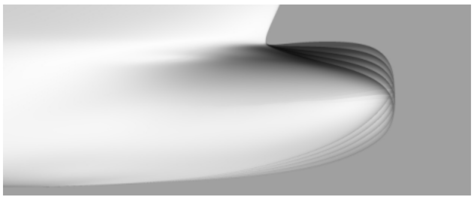

Figure 6-b

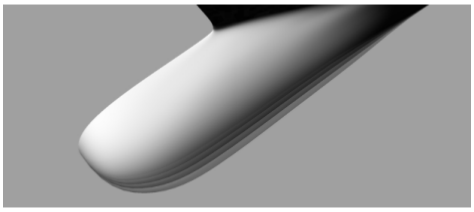

Figure 6-c

Figure 6: Systematic variations of bulb shapes

4. TESTS ON A DREDGER HULL FORM

The approach presented in the previous paragraphs was tested on a dredger hull form in order to test its ability to compute and optimise a form with challenging features such as complex geometries and heavily separated free surface flows.

The mesh generated by the SnappyHexMesh application for the after-body is shown on figure 4. The mesh comprises about 1 million elements.

The history of the pressure and friction resistance during the convergence of the computation of the flow around the hull form which is free to sink and trim is shown on figure 7.

The resulting flow is presented on figure 8 where the free surface deformation is shown with wave breaking on the blunt bow, separation behind the deep transom, and the streamlines on the optimised bulb and gondola shapes.

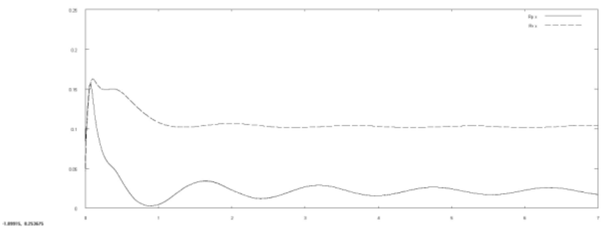

Figure 7: History of pressure and friction resistance during convergence

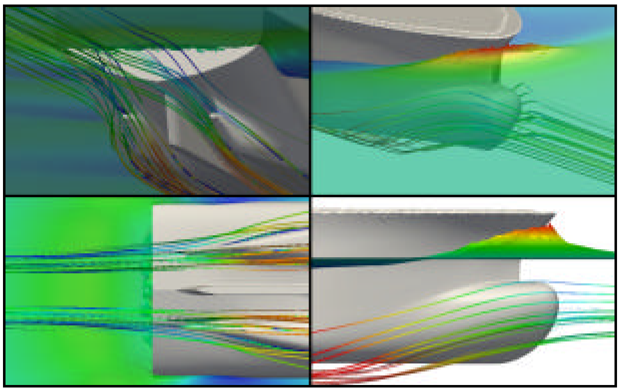

Figure 8: Flow features of the computed domain

5. TEST ON A CRUISE VESSEL HULL FORM

5.1 OPTIMISATION OF 2 HULL FORMS

In the context of a 250 m cruise vessel project, a hull form optimisation process was conducted where, for the same global ship dimensions and displacement, two hull forms were optimised in order to minimize their resistance at 16 and 21 knots respectively.

The objective of this study was to identify the potential fuel consumption gain which may be achieved with a hull form optimised for intermediate speeds. Indeed, depending on the cruise profiles, the ship speed may vary considerably on the different legs of the journey and maximum speed is seldom used. Also of interest is the penalty on installed power or maximum speed of selecting a hull form not optimised for maximum speed.

This exercise led two the 2 very similar hull shapes (figure 9 and 10), both able to be used for the construction of the vessel in terms of all the constraints such as displacement, stability, LCB, propulsion integration, etc… These two hull forms differ mostly in their forward and aft sections which were adapted to the different flow conditions. They are further referred to the Low Speed Hull Form (LSHF) and the High Speed Hull Form (HSHF).

Figure 9: LSHF - Hull form optimised for the lower speed

Figure 10: HSHF - Hull form optimised for the higher speed

5.2 OpenFOAM® RESULTS

Both hull forms were calculated with both OpenFOAM and another free surface RANS code at several speeds including their respective design speeds of 16 and 21 knots. The corresponding wave elevations at 16 and 21 knots computed with OpenFOAM are shown on figures 11 to 14.

Free Surface elevations at 16 knots

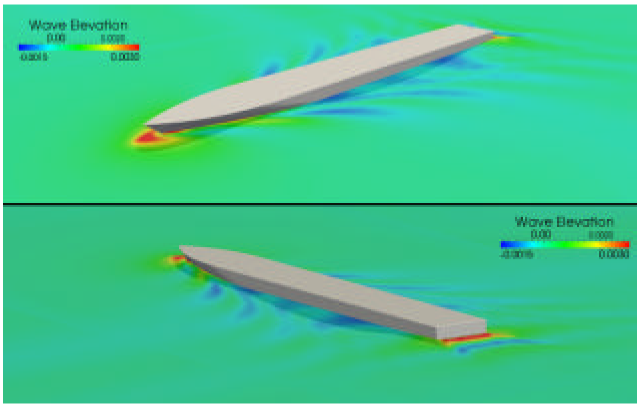

Figure 11: LSHF at 16 knot

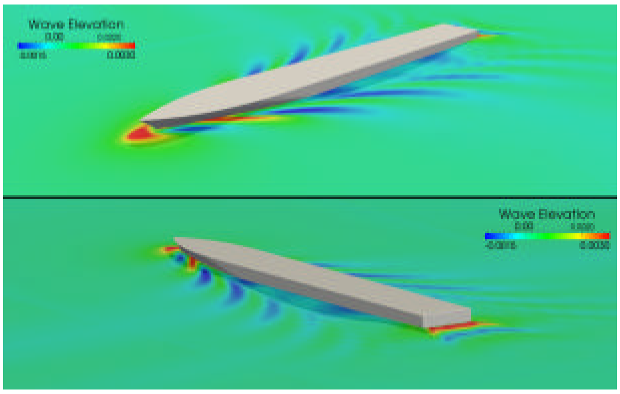

Figure 12: HSHF at 16 knots

Free Surface elevations at 21 knots

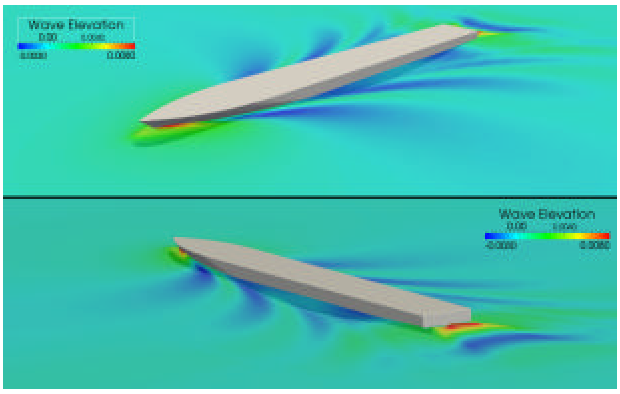

Figure 13: LSHF at 21 knots

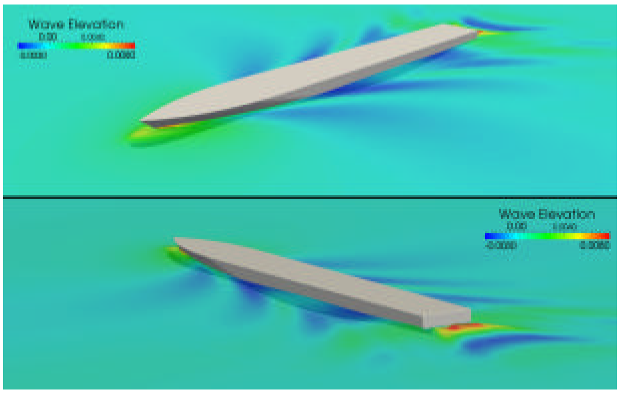

Figure 14: HSHF at 21 knots

5.3 COMPARISON OF CFD WITH TOWING TANK RESISTANCE

Models of both hull forms were built and tested for resistance as well as for self propulsion with a pair of pods. Only the bare hull resistance tests are referred to here.

Concerning the towing tank results, we consider only the full scale resistance values given by the towing tank institute based on the resistance tests and taking into account the extrapolation procedure and correlation coefficient based on a 3D method. This choice is based on the fact that only the full scale value is of practical interest for the shipyard and the extrapolation methods are adapted to give the best possible estimates of full scale resistance.

Concerning CFD, OpenFOAM was used as presented earlier in the paper. Another reference free surface RANS CFD tool was also used to check its ability to estimate the full scale resistance. Both CFD codes are used in the same manner where only the pressure resistance from the computation is used while the frictional resistance is based on the ITTC 78 model to full scale correlation line. Although it may appear as a paradoxical in a context where viscous flow simulations are used, this method which does not use the CFD calculated frictional resistance has been chosen because it is very difficult to compute a frictional resistance at full scale. Indeed, the requirements on grid density and topology, the associated computational time, and the simulation of roughness make this type of simulation unpractical.

However, it should be kept in mind that free surface viscous codes offer many advantages in terms of flow simulation compared to potential flow codes, including vortex generation, separated flows, wave breaking, complex geometries…

The comparisons of the two CFD codes with model test results for the two hull forms (LSHF and HSHF) are presented in figures 15 and 16.

The difference between model tests and the resistance computed based on OpenFOAM for both hull form at 3 different ship speeds (figure 15) is very small at the lower speed and increases to about 2 % at the higher speed.

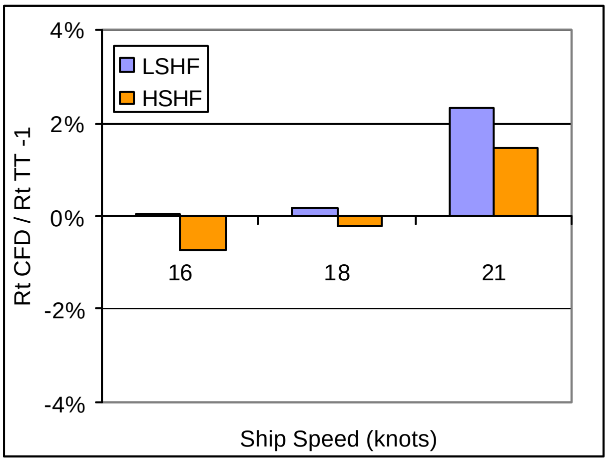

Figure 15: Comparison between OpenFOAM and Towing Tank estimates of bare hull resistance

The difference between model tests and the resistance computed based the other CFD code for both hull form at 3 different ship speeds (figure 16) is rather large at the lower speeds and reduces with increasing speed where it reaches about 1% at 21 knot although for each form the trend is opposite.

Figure 16: Comparison between other CFD code and Towing Tank estimates of bare hull resistance

5.4 COMPARISON OF HULL FORM RANKINGS

Finally, the same information can be used to check the ability of the two CFD codes to rank the hull forms at different speeds compared to the towing tank estimate. This comparison is shown on Figure 17 where the ratios of the resistances estimated by the 3 methods are plotted for three ship speeds.

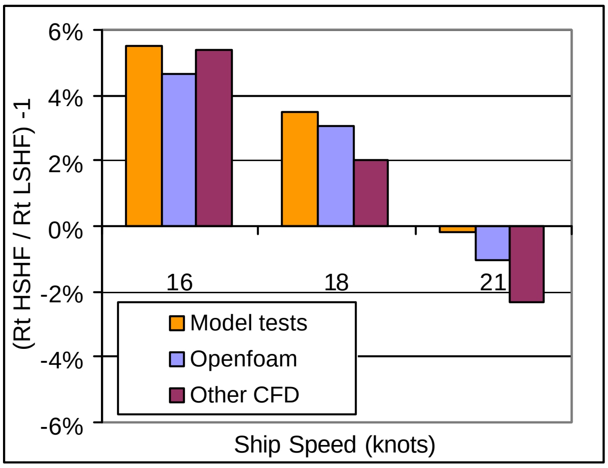

Figure 17: Resistance difference between the two forms based on CFD and Towing Tank estimates

From this graph, it is clear that all methods predict that the ship optimised for the lower speed will have a smaller resistance at lower speed (5% at 16 knots and about 3% at 18 knots. At 21 knots, the model tests predict a very small advantage for the higher speed hull form compared to the lower speed hull form, whereas the gain expected by CFD during the optimisation process was about 1%

.

Although some further investigations are needed to understand how better agreement may be obtained, the absolute and relative accuracy of the CFD predictions appear to be of the same order of magnitude as that expected from model tests and hence, may be used for optimisation purposes for this type of vessel.

6. CONCLUSIONS

The solution developed at STX France for the numerical optimisation of hull forms is based on the OpenFOAM open source package for the computation of the ship resistance has been presented and used in combination with parametric description of the hull form based on Rhino, Rhinoscript, and the Grasshopper application for the systematic generation of hull form modifications.

The practical advantages of such a solution were discussed and a practical imp lementation of the process on a real case study has demonstrated its usefulness for the selection of a hull shape best adapted to a customer's requirements.

Further developments of this solution include the optimisation of the computational time, the presence of appendages and the effect of the propulsor.

7. REFERENCES

1. Shipflow : *www.flowtech.se*

2. H. Raven, 'Inviscid calculations of ship wavemaking capabilities, limitations and prospects' 22nd Symp. on Naval Hydrodynamics, Washington DC, 1998.

3. G. Delhommeau, JJ Maisonneuve "Calculation of Flow around several Ship Hulls by a Rankine Source Method", Proceedings of the 2th International Conference Hydrosoft, Southampton, 1986

4. B. Alessandrini, E. Jacquin, R. Luquet, P.-E. Guillerm, "RANSE with free surface computations around fixed and free DTMB 5415 model, in still water and waves", CFD Workshop, Tokyo, Japan, March 2005

5. E. Jacquin, Q. Derbanne, D. Bellevre, S. Cordier, B. Alessandrini, Y. Roux, "Hull Form Optimization Using A Free Surface RANSE Solver", 25th Symposium on Naval Hydrodynamics, St John's, Newfoundland, Canada, August 2004

6. Van der Ploeg, A. and Hoekstra, M. "Multi-objective Optimization of a Tanker Afterbody using PARNASSOS", NuTTS 12th Numerical Towing Tank Symposium, 2009

7. Jochen Marzi "Use of CFD methods for hullform optimisation in a model basin"
MARNET – CFD Workshop, Haslar 20/21-03-2003

8. Hendrix, Dane, Percival, Scott and Noblesse, Francis (2001). "Practical Hydrodynamic Optimization of a Monohull," SNAME Transactions, Vol. 109, pp 173-183.

9. E. Paterson, K. Smith, and S. Ford, "Simulation Of Wakes, Wave Impact Loads, and Seakeeping Using OpenFOAM", 4th OpenFOAM® Workshop, Montreal, Canada, June 2009

10. M. Peric & J.H. Ferziger, "Computational method for fluid dynamics", 3rd edition, Springer, 2002.

11. O. Ubbink, "Numerical prediction of two fluids system with sharp interface", Imperial College, 1997.

12. H. Jasak, Z. Tukovic, "Automatic Mesh Motion For The Unstructured Volume Method", Transactions of FAMENA, 2007.

8. AUTHORS BIOGRAPHY

Stephane Cordier: In charge of the Ship Performance Division comprising hydrodynamics, noise and vibration as well weights and stability from preliminary design to customer acceptance. His previous experiences include Technical Director of the hydrodynamic expertise centre in Val de Reuil as well as naval architect for the Fr defense procurement agency (DGA).

Loïc Morand : In charge of Hydrodynamic Dept. He is responsible for speed power performance and pressure pulse level, seakeeping and manoeuvring performance from the pre-design project up to the sea trial. His experience in hydrodynamics covers naval vessels, cruise ship, off shore projects, and yachts.

Jean-Marie Roux : In charge of CFD calculations and hull form design and optimisation in the Hydrodynamic Dept.

Edgar Cortey: Co-founder CFD-Numerics Company. The company delivers analyses, training and methodologies development for fluid flows, heat exchange, combustion and chemical reactions applications.