Validation of the CFD code Flow-3D for the free surface flow around the ships' hulls

Ema Muk-Pavic[1], Shin Chin [2] and Don Spencer[1]

[1]*Oceanic Consulting Corporation Ltd., #401-95 Bonaventure Ave., St. John's, NL A1B 2X5, Canada*

[2] *National Research Council of Canada, Institute for Ocean Technology, Arctic Avenue, P.O. Box 12093, Station A, St. John's, NL A1B 3T5, Canada*

Email: *ema_muk@oceaniccorp.com*

ABSTRACT

This paper describes the Computational Fluid Dynamics (CFD) calculations that were completed to model the free surface flow around the ships' hulls. Published experimental data for the DTRC 5415 combatant model is commonly used for validation of numerical codes.

Simulations were performed using the software Flow-3D, a Reynold's Averaged Navier-Stokes (RANS) solver with structured orthogonal mesh.

The verification was based on the examination of the flow around the hull for range of speeds and by comparison of the results for resistance obtained by CFD simulations and by experiments. Additional analysis has been conducted to investigate mesh sensitivity and the implementation of different advection schemes. The second order advection scheme with monotonicity preserving was optimal for the qualitative analysis of the problem under consideration.

This study shows that CFD code Flow-3D has a limited capability to resolve the physics of the flow around the hull. The shape of the free surface and wave distributions around the hull corresponds approximately to the experimental observations.

For quantitative analysis of ship total resistance, Flow-3D shows a lack of accuracy. It appears that the code does not have the capability to properly resolve boundary layer on the hull and properly predict frictional resistance. It can be improved by using only dynamic pressure results and by using some established empirical/experimental approach for estimating frictional resistance.

The multi-block grids and the different turbulent models are being used to obtain valid numerical results that are crucial for making sound design decision.

1. INTRODUCTION

The simulation of free surface flows around the ship hulls on the higher Froude numbers is a major challenge for any CFD code. The validation of this problem using experimental data in the simulation of bow and stern waves and the overall flow resistance has been determined with a limited success.

The numerical simulation has been conducted on the US Navy Combatant model in scale 1:24.8, DTRC 5415, for the range of Froude numbers 0.17 – 0.4. This hull is streamlined with transom stern and sonar dome at the bow. The experimental data for this model is published and is often used for validation of numerical codes.

Flow-3D was chosen for its simplicity, versatility, and volume of fluid (VOF) method used for free surface interface tracking. Keepng in mind the inaccuracy of using Cartesian fractional area/volume method (FAVOR) used for geometry definition, special attention has been devoted to obtain appropriate mesh for the streamline body.

In this study we performed a series of numerical simulations for the sole purpose of validating available experimental data. The goal is to be able to perform the majority of analysis at the design stage with numerical simulations. This would leave only the minimum scope of the cases for expensive experimental analysis.

2. FLOW–3D CODE OVERIEW

FLOW-3D is a general purpose CFD software capable of simulating a wide range of fluid flows. The equations solved are RANS equations given, in non-dimensional form, by the conservation of mass and momentum.

The code features of interest for this project are:

- Fractional areas/volumes (FAVOR) for geometry definition,
- Structured finite difference multi-block grid,
- Volume-of-Fluid (VOF) method for fluid interfaces tracking, and
- Implicit numerical modeling.

FLOW-3D is an all-inclusive package; graphical user interface ties together problem setup, pre-processing, solver and post-processor.

3. EXPERIMENT

US Navy Model 5415 represents a modern naval combatant and is widely used for validation of CFD codes. The hull is of the semi-displacement type with sonar dome and transom stern as presented in Figure 1.

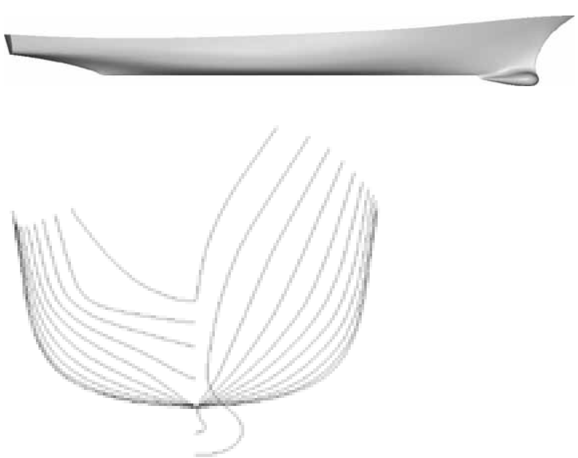

Figure 1: US Navy Model 5415 (L = 5.72 m)

The resistance data and wave profiles along the hull were taken in the bare hull condition (without appendages or propellers) in 1982 and 1997 at David Taylor Towing Tank. During these experiments the model was free to sink and trim as it was towed by the carriage. The wave profile data was obtained at Froude numbers 0.28 and 0.41.

4. COMPUTATIONAL DETAILS

The numerical simulation problem is symmetrical, hence only one side of the hull was modeled. A computational domain was created in the rectangular shape, with a semi-hull solid on the symmetry plane (Figure 2). The geometry file of the model was imported in Flow-3D as stereolithographic file.

The model was simulated at a static vs. dynamic waterline in experiments. In the experiments at the higher Froude numbers model had a sinkage and trim. Unfortunately, the dynamic waterline data is available only for two speeds. Running simulations with the hull positioned at the static waterline presents a pre-imposed source of error.

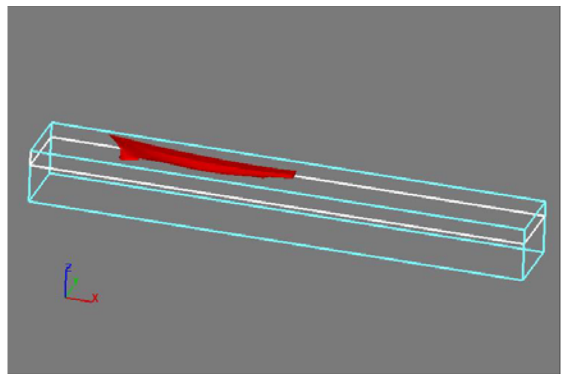

Figure 2: Simulation domain

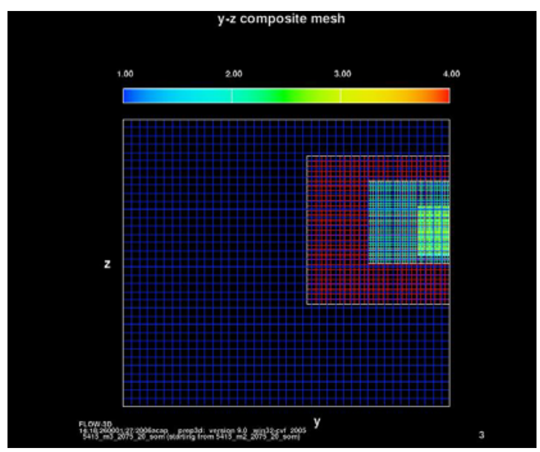

Figure 3: Mesh presentation (1,507,184 mesh elements)

A general background mesh was refined in the area closer to the hull by using multiple mesh blocks and gradually reducing the size of the mesh elements, as shown in Figure 3. For a mesh sensitivity study, three meshes with different mesh refinements were created (Table 1).

Mesh type	No. of Froude numbers	The smalest element size (m)
m1	509,748	0.030 (0.5%L)
m2	1,046,760	0.020 (0.35%L)
m3	1,507,184	0.015 (0.25%L)

Table 1: Mesh characteristics

A physical model is defined as a uniform viscous flow around the hull with specified velocity in the X direction and a hydrostatic pressure field. We deemed that Renormalized group theory (RNG) turbulent model was appropriate for this simulation.

The boundary conditions were as follows:

- Specified velocity on inlet;
- Outflow boundary that minimizes wave reflections;
- Hull surface - obstacle with no slip walls; and
- Symmetry plane, side, bottom and top side of domain as free slip walls.

Flow-3D has various numerical options for a solving process. By default, the upwind implicit advection scheme for solving momentum equations is used. The whole resistance curve (a range of Froude numbers flows) has been obtained with the upwind advection scheme, which is robust and fast to resolve. Additionally, for Froude number 0.28, the second order advection scheme and the second order advection scheme with monotonicity preserving were also applied.

5. Results and Discussion

The simulation matrix that has been resolved to date is presented in Table 2.

As shown on Figure 4, the numerical prediction of the total resistance curve, obtained with a 509,748 element mesh and the upwind advection scheme, corresponds to experimental curve but is significantly overestimated. The free surface shape and wave pattern corresponds well to experiments. Total resistance results are overestimated for almost factor 3. A closer correlation is obtained using only numerical results for wave resistance and approximating frictional resistance using the ITTC-57 method. When comparing them with experiments, the total resistance curve looks better but still over predicted by factor 2.

No. of mesh refinements	Ragne of Froude numbers	Advection scheme
1	6 (0.17 – 0.4)	1st order
1	1 (0.28)	2nd order
3	1 (0.28)	2nd order with monotonicity preserving

Table 2: Simulation Matrix

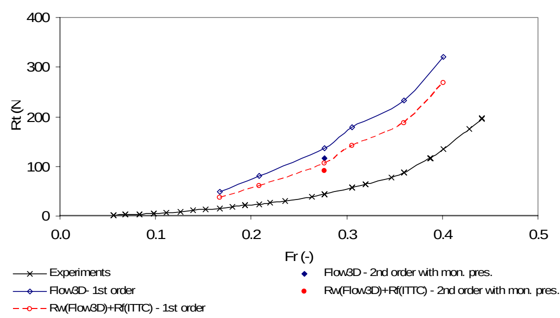

Figure 4: Total resistance curve (509,748 mesh elements)

It appears that Flow-3D does not have the capability to accurately enough resolve boundary layer around the hull and properly predict frictional resistance. This drawback was expected having in mind limitations of the FAVOR method used in Flow-3D for geometry definition. Numerical simulations should still take viscosity into account in the turbulence model so that the flow field and the free surface shape can be properly resolved.

The results obtained with the upwind advection scheme were improved using second order and second order with monotonicity preserving advection schemes (for the case Fr=0.28). The second order advection scheme with monotonicity preserving is considered to be the optimal, even though calculation time has been prolonged significantly. The obtained resistance and wave pattern are qualitatively compared with experiments (see Figure 5). The waves are located approximately on the same locations along the hull.

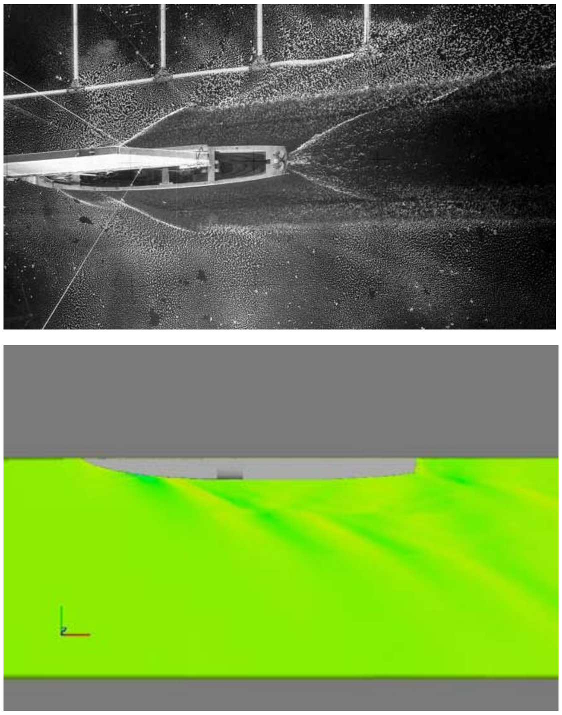

Figure 5: Flow-3D’s and experimental wave pattern comparison (Fr=0.28)

For this case, a mesh sensitivity study was conducted (see Figure 6). It showed that the frictional component of the resistance reached mesh independence. The wave resistance, and consequently total resistance, continues to change with mesh refinement. Achieved improvement is still not satisfactory for the given computational effort (error of 65% of total resistance). It would be beneficial to continue with the mesh sensitivity study until the full mesh independence has been reached.

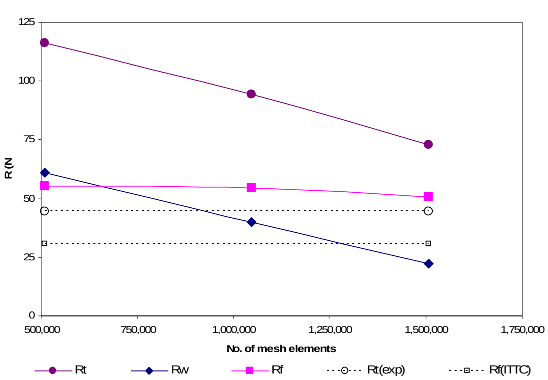

Figure 6: Mesh sensitivity study (Fr=0.28)

The intention is to obtain the whole resistance curve using the second order with monotonicity preserving advection scheme and the mesh for which mesh independence is reached and, hopefully, include it in the full paper.

6. Conclusion

This numerical study indicated that Flow-3D is an appropriate tool for qualitative analysis of the free surface flow around the ship’s hulls. However, for the range of Froude numbers, total resistance is over estimated by factor 3.

The second order advection scheme with monotonicity preserved is considered as the most suitable giving comparative results of 60%. The mesh refinement and multi-blocks schemes improved the results. Different turbulence techniques might give further improvement of the results.

User skill and experience are important in making proper engineering judgment based on the simulation results of such a complicated problem as the free surface flow around the ship’s hulls.

Acknowledgements

We acknowledge with thanks to NRC-CNRC Institute for Ocean Technology in St. John’s, Canada, and to Professor Don Bass from Memorial University of Newfoundland in St. John’s, Canada.

References

[1] Barkhudarov, M., “*Multi-Block Gridding Technique for FLOW-3D*", Technical Note #59-R2, FSI-00-TN59-R2, Flow Science Inc., 2004

[2] Ferziger, H. J., and Peric, M., “Computational Methods for Fluid Dynamics”, 2001, Springer-Verlag

[3] Lewis, E. V., “Principles of Naval Architecture” SNAME, USA, 1989

[4] Lin, A. C., "Bare Hull Effective Power Predictions and Bilge Keel Orientation for DDG51 Hull Represented by Model 5415," DTNSRDC/SPD-0200-03, 1982 (http://conan.dt.navy.mil/5415/)

[5] Ratcliffe, T. J., Muntick, I., Rice, J., “Stern Wave Topography and Longitudinal Wave Cuts obtained on Model 5415, With and Without Propulsion”, DTM, USA, 2001

[6] Yao, G. F., “Development of New Pressure-Velocity Solvers in FLOW-3D”, Flow Science, Inc., USA, 2004