

# META Tool Suite Fluid Analysis Validation with the Wigley Hull

Ozgur Yapar <sup>1</sup>, Michael R. Myers <sup>1</sup>, Mahyar Varasteh <sup>1</sup>, Sandeep Neema <sup>1</sup>, Ted Bapty <sup>1</sup>

<sup>1</sup> *Vanderbilt University, Institute for Software Integrated Systems (ISIS)  
1025 16th Ave S, Suite 102, Nashville, TN 37212*

## Abstract

The existence of empirical and theoretical results for the Wigley hull make it a good test case to validate simulation software operation and results. The Wigley hull is being used for validation of META tools in the Defense Advanced Research Projects Agency's Fast, Adaptable, Next-Generation Ground Vehicle program. This program's purpose is to produce tools to help design and develop a new heavy, amphibious infantry fighting vehicle. The program leverages META design tools and the VehicleFORGE collaboration environment to significantly change the design experience and widen the aperture for design innovation. The META tool suite consists of tools for compositional design synthesis at multiple levels of abstraction, design trade space exploration, metrics assessment, and probabilistic verification of system correctness. This work details fluid dynamics simulations of the Wigley hull in calm water with the free surface incorporated into the simulations. Simulations are conducted using the META tool suite which leverages the open-source fluid dynamics toolbox OpenFOAM<sup>®</sup> to perform the fluid dynamics simulation. Simulation results show very good agreement with published empirical data. The results of this study provide the necessary validation of the META tool suite for estimating vehicle drag.

## Nomenclature

$B$	Beam at midship (m)
$C_t$	Total resistance coefficient
$D$	Draft at midship (m)
$D_\omega$	Cross-diffusion term
$F_d$	Drag force (N)
$Fr$	Froude number
$G_k$	Generation of $k$
$G_\omega$	Generation of $\omega$
$k$	Turbulence kinetic energy (joule)

$L$	Waterline length (m)
$Re$	Reynolds number
$S$	Wetted surface area (m <sup>2</sup> )
$S_k$	User-defined source term of $k$
$S_\omega$	User-defined source term of $\omega$
$t$	Time (s)
$u$	Velocity (m/sec)
$Y_k$	Dissipation rate of $k$
$Y_\omega$	Dissipation rate of $\omega$

#### Greek Letters

$\alpha$	Volume fraction function
$\Gamma_k$	Effective diffusivity of $k$
$\Gamma_\omega$	Effective diffusivity of $\omega$
$\varepsilon$	Turbulence dissipation rate (1/s)
$\mu$	Kinematic viscosity (m <sup>2</sup> /s)
$\rho$	mass density (kg/m <sup>3</sup> )
$\omega$	Turbulence dissipation rate (1/s)

## 1 Introduction

Some of the most important performance attributes of an amphibious vehicle are the combined sea and land range and the sustained sea forward cruising speed. Estimating these performance attributes requires accurate drag force predictions. Empirical drag correlations are useful for low-fidelity estimates, however higher fidelity drag estimates demand Computational Fluid Dynamics (CFD) simulations. In a previous study, an amphibious vehicle's speed vs. resistance relation is estimated using FLUENT<sup>®</sup> and validated using experimental data [Guo et al., 2012]. Additionally, the simulation data is compared with 1957 ITTC formula calculations. CFD simulation results are in good agreement with the empirical data as well as the formula calculations. Another study details multiphase flow and buoyancy FLUENT<sup>®</sup> simulations of a tracked amphibious military vehicle [Kotowski et al., 2012]. The process of amphibious vehicle water entry is studied in the literature with STAR-CCM+<sup>®</sup> [Burciu et al., 2012]. During the simulations, Reynolds Averaged Navier-Stokes (RANS) equations are solved using the finite volume method. The study also presents experimental results of a scaled vehicle model in a towing tank at the Gdynia Maritime University Ship Design and Research Center.

This work explores the accuracy of CFD simulations as part of the Defense Advanced Research Projects Agency's (DARPA) Fast, Adaptable, Next-Generation Ground Vehicle (FANG) program. This program's purpose is to produce tools to help design and develop a new heavy, amphibious infantry fighting vehicle. The program leverages META design tools, an open-source design tool suite for creation, testing and verification of vehicle designs. The META tool suite consists of tools for compositional design synthesis at multiple levels of abstraction, design trade space exploration, metrics assessment, and probabilistic verification of system correctness.

Vehicle drag validation of the META tool suite is accomplished in this work with the Wigley hull [Wigley, 1934]. CFD analysis of the Wigley hull is completed using META tools which

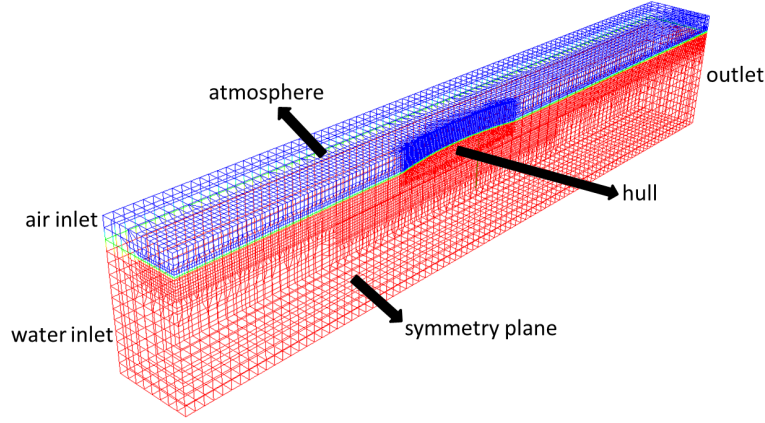


Figure 1: Computational domain, boundary conditions and mesh of the Wigley hull model with symmetry plane.

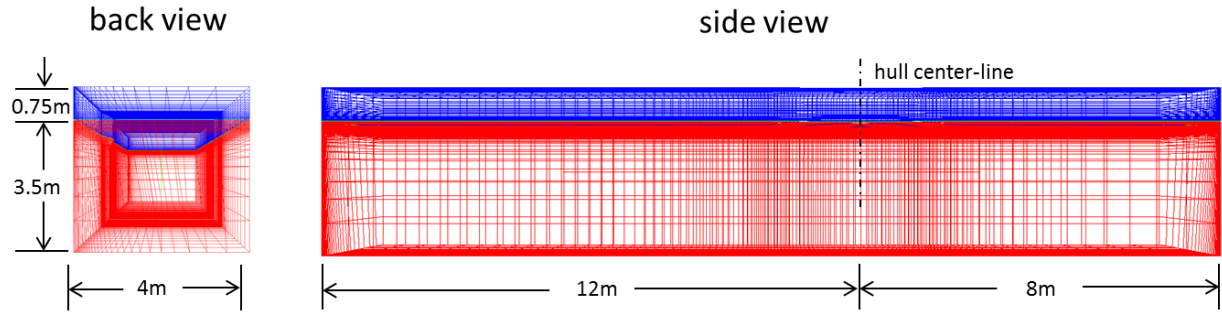


Figure 2: Dimensions of the Computational domain.

automatically perform OpenFOAM<sup>®</sup> [OpenFOAM Development Team, 2013] simulations. Steady resistance 3D CFD simulation results of the Wigley hull, in calm water with constant speed, are presented. The Wigley hull case presented in the literature [Maki, 2011] is modeled and simulated using the META tool suite with seven different hull speed values. The results are validated using published experimental data [Kajitani et al., 1983].

## 2 Model Description

For validating the vehicle drag predictions in the META tools, a general model has been used. For that purpose, a Wigley hull has been chosen as the test case. A specific case of the Wigley hull presented in the literature is analyzed in this work [Maki, 2011]. The drag force of the hull is calculated for a 3D domain. The dimensions of the computational domain, mesh details and boundary conditions are shown in Figures 1 and 2. The geometric details of the hull are presented in Table 1.

Table 1: Geometric Parameters of the Hull

Parameter Name	Variable	Value
Waterline length	$L$	4.0 m
Beam at midship	$B$	0.4 m
Draft at midship	$D$	0.25 m
Wetted Surface	$S$	2.3796 m <sup>2</sup>

Table 2: Summary of Simulations

Simulation Number	Froude Number ( $Fr$ )	Reynolds Number ( $Re$ )
1	0.09	$2.25 \times 10^6$
2	0.15	$3.76 \times 10^6$
3	0.21	$5.26 \times 10^6$
4	0.26	$6.51 \times 10^6$
5	0.32	$8.02 \times 10^6$
6	0.36	$9.02 \times 10^6$
7	0.40	$1.00 \times 10^7$

For calm water resistance CFD simulations of the Wigley hull, the interFoam solver of OpenFOAM® version 2.2.0 is used. Simulations of the hull is completed with fixed trim and sinkage which results in 0 degrees of freedom. Total simulation time is set at 40 seconds. Force values obtained between 10 and 40 seconds are averaged to obtain the drag force. For the purpose of saving computational effort, only half of the domain is modeled and near the hull, refined mesh is used. The computational domain used for CFD simulations is 3D and is meshed with hexahedral, polyhedral and prism type of elements. The mesh consists of a total of 1,471,076 cells with 1,408,244 hexahedral, 62,198 polyhedral and 626 prism elements.

The simulation boundary conditions include setting the free-stream and the internal field initial velocities to match Froude numbers detailed in Table 2. The initial velocity at the outlet is set to zero gradient since the velocity in the downstream does not affect the drag force on the hull. The boundary condition on the hull surface is set to no-slip. The outlet and top boundaries are set to zero dynamic pressure. The air inlet and water inlet are set to zero gradient. Initially  $\alpha_1$ , which represents volume fraction, is set to zero at the cells over the water-line and set to one at the cells which are below the water-line.

Froude number ( $Fr$ ), a dimensionless number named after William Froude, is a way of determining if flow in a channel is sub or supercritical. Like Mach number which indicates subsonic if it is less than 1 and supersonic if it is greater than 1, a Froude number less than 1 indicates a subcritical flow and greater than 1 indicates supercritical flow. Mathematical representation of the Froude number used for ship hydrodynamics applications is shown in equation 1 [Kay, 2007].

$$Fr = \frac{v}{\sqrt{gL}}. \quad (1)$$

The main use of the Froude number is to compare the resistance of partly submerged ob-

jects, with different sizes, moving through water which makes it an important parameter for ship hydrodynamics. It may be defined as the ratio of a body's inertia to gravitational forces. When the flow is critical ( $Fr = 1$ ), the speed of the surface wave and the velocity of the object is the same. When the flow is subcritical ( $Fr < 1$ ), velocity of the object is smaller than the speed of the disturbance waves traveling on the surface which means gravitational forces are dominant. When the flow is supercritical ( $Fr > 1$ ), inertial forces are dominant [Kay, 2007].

For capturing the turbulence behavior, the shear-stress transport (SST)  $k$ - $\omega$  turbulence model is used [Menter, 1993]. The model can be described as a two-equation eddy-viscosity model [Larsson, 1997]. The main idea of the SST  $k$ - $\omega$  turbulence model is blending the  $k$ - $\omega$  formulation, which gives accurate results in near-wall regions, with the  $k$ - $\varepsilon$  model, which works well in the far-field. The switching between the two turbulence models is accomplished by the SST formulation. SST  $k$ - $\omega$  turbulence model can be described by using equations 2 and 3 [Ren and Ou, 2009].

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}(\Gamma_k \frac{\partial k}{\partial x_j}) + \tilde{G}_k - Y_k + S_k \quad (2)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j}(\Gamma_\omega \frac{\partial \omega}{\partial x_j}) + G_\omega - Y_\omega + D_\omega + S_\omega. \quad (3)$$

With the addition of a blending function to ensure that the model gives accurate results both at the near-wall regions and far-field zones and a cross-diffusion term in the  $\omega$  equation, the SST  $k$ - $\omega$  turbulence model behaves appropriately for a wider class of flows including ship hydrodynamics applications.

The multi-dimensional universal limiter with explicit solution volume of fluid (MULES-VoF) method with interface compression is used for interface capturing. Two-phase liquid flow presents complexities due to the presence of an interphase surface which contains non-uniform physical properties such as pressure, density, and viscosity. Modeling can be performed using moving boundary with special boundary conditions [Marek et al., 2008]. The main principal of the VoF method is keeping track of the free surface which is also a fluid-fluid (two-phase) interface. The VoF method is used along with the Navier-Stokes equations, but they are solved separately. Although other methods exist for approximating free boundaries in numerical simulations, VoF has been proven to be much more efficient and flexible [Hirt and Nichols, 1981]. The method can be described as follows:

The scalar volume fraction function  $\alpha$  that has a value one at any point where there is a fluid and zero elsewhere. If  $\alpha$  is between 0 and 1, an interface is present (see Figure 3).

First, the transport equation needs to be solved for the volume fraction function.

$$\frac{\partial \alpha}{\partial t} + \frac{\partial u_i \alpha}{\partial x_i} = 0. \quad (4)$$

0	0	0	0	0	0
0	0	0	0	0	0
0	0	0.15	0.35	0.25	0
0.80	0.75	0.95	1.0	1.0	0.85
1.0	1.0	1.0	1.0	1.0	1.0

Figure 3: Approximated interface by using volume fraction  $\alpha$ .

The density and kinematic viscosity of the fluid can be calculated by using the linear dependence of the volume fraction function.

$$\alpha(x, t) = \alpha, \quad (5)$$

$$\mu(x, t) = \mu_{water}\alpha + \mu_{air}(1 - \alpha), \quad (6)$$

$$\rho(x, t) = \rho_{water}\alpha + \rho_{air}(1 - \alpha). \quad (7)$$

The passive advection of void fraction can be represented by equation 8, since the fluid type remains constant along particle paths. [Gerlach et al., 2006]

$$\frac{\delta \alpha}{\delta t} + u \cdot \nabla \alpha = 0. \quad (8)$$

Equation 8 can be reformulated into a conservative form shown in equation 9 by using the continuity equation. [Gerlach et al., 2006]

$$\frac{\delta \alpha}{\delta t} + \nabla \cdot u \alpha = 0. \quad (9)$$

By solving equation 9 numerically, volume fraction  $\alpha$  can be calculated. After that, by using the updated volume fraction, fluid properties is calculated using equations 6 and 7 and finally the interface is reconstructed.

The direction in which  $\alpha$  changes the fastest defines the normal direction to the boundary.  $\alpha$  is a step function so its derivatives can be used to determine the boundary normal in the mesh. Ultimately, after the value of  $\alpha$  and the normal direction of a cell are found, they can be used to

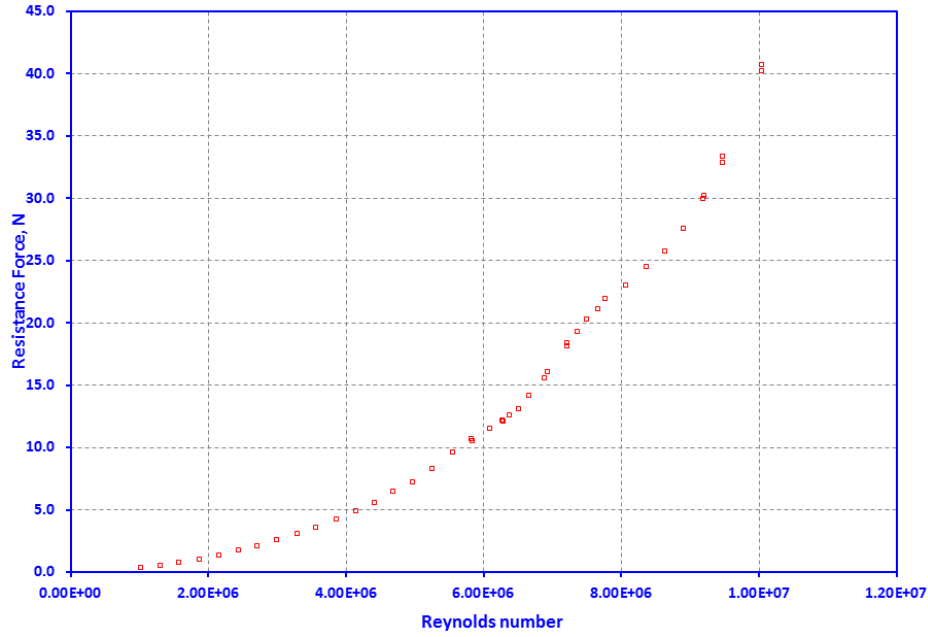


Figure 4: Empirical resistance force with respect to Reynolds number for the SRI hull [Kajitani et al., 1983].

approximate the interface at that location [Hirt and Nichols, 1981].

During the simulations, unsteady Reynolds Averaged Navier-Stokes Equations (URANS) is the flow model of the simulation. The fluid solver interFoam also includes the PIMPLE algorithm which is a merged Pressure-Implicit with Splitting of Operators (PISO) and Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) procedure [OpenFOAM Development Team, 2013]. By coupling a SIMPLE outer-corrector loop with a PISO inner-corrector loop, the PIMPLE algorithm enables a more robust pressure-velocity coupling in stiff differential equations [Rodrigues et al., 2011].

### 3 Experimental Studies

Validation of the META tool suite with respect to fluid analysis is accomplished by comparing META tools simulation results with published experimental data [Kajitani et al., 1983]. Among the experimental work presented in the study, the data of the SRI hull is used for the validation purposes. The results of the resistance tests with fixed sink and trim is used. The empirical total resistance coefficient ( $C_t$ ) vs. Froude number ( $Fr$ ) and empirical resistance force ( $F_d$ ) vs. Reynolds number ( $Re$ ) used for validation purposes are presented in Figures 4 and 5 respectively. The formula for total resistance coefficient is presented in equation 10.

$$C_t = \frac{F_d}{1/2 \rho u^2 S}. \quad (10)$$

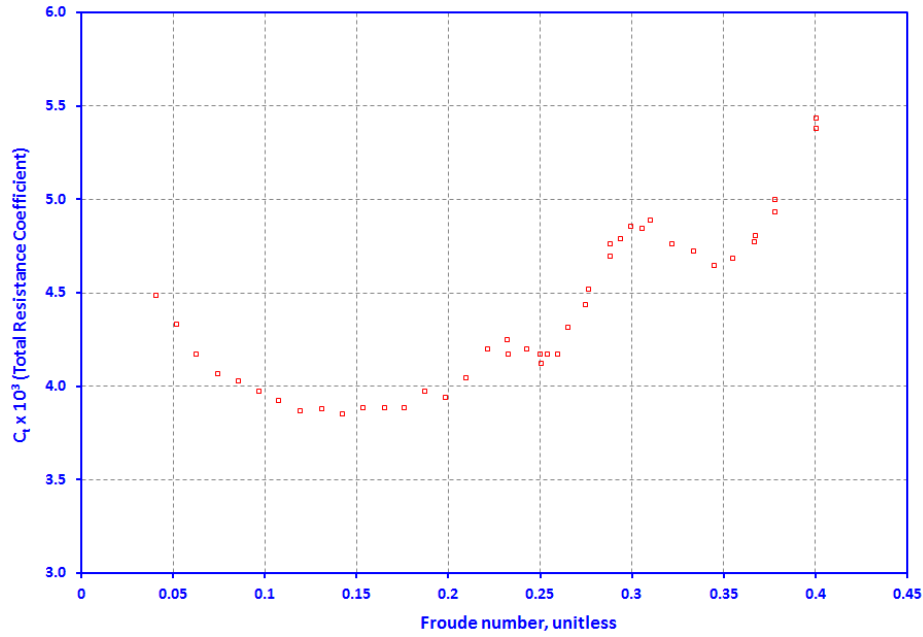


Figure 5: Empirical total resistance coefficient with respect to Froude number for the SRI hull [Kajitani et al., 1983].

Table 3: Summary of CFD Simulation Results

Froude Number ( $Fr$ )	Reynolds Number ( $Re$ )	$C_t \times 1000$	Resistance Force ( $F_d$ )
0.09	$2.25 \times 10^6$	4.005	1.51 N
0.15	$3.76 \times 10^6$	4.062	4.27 N
0.21	$5.26 \times 10^6$	4.315	8.89 N
0.26	$6.51 \times 10^6$	4.480	14.14 N
0.32	$8.02 \times 10^6$	4.902	23.44 N
0.36	$9.02 \times 10^6$	4.628	28.00 N
0.40	$1.00 \times 10^7$	4.879	36.45 N

## 4 Results

All simulations converge after the first few of seconds of simulation time (Figure 6). The simulation data between the 10th second and end of the simulation (40th second) is averaged to estimate the steady-state resistance force ( $F_d$ ). Wave contours obtained from the CFD model is illustrated in Figure 7.

The total resistance coefficient values obtained from the META tools for seven different Froude numbers is presented in Table 3. Comparisons of the META tools simulation results and the published empirical data are illustrated in Figure 8 and 9. Agreement between the META tools simulations and the empirical data is very good.



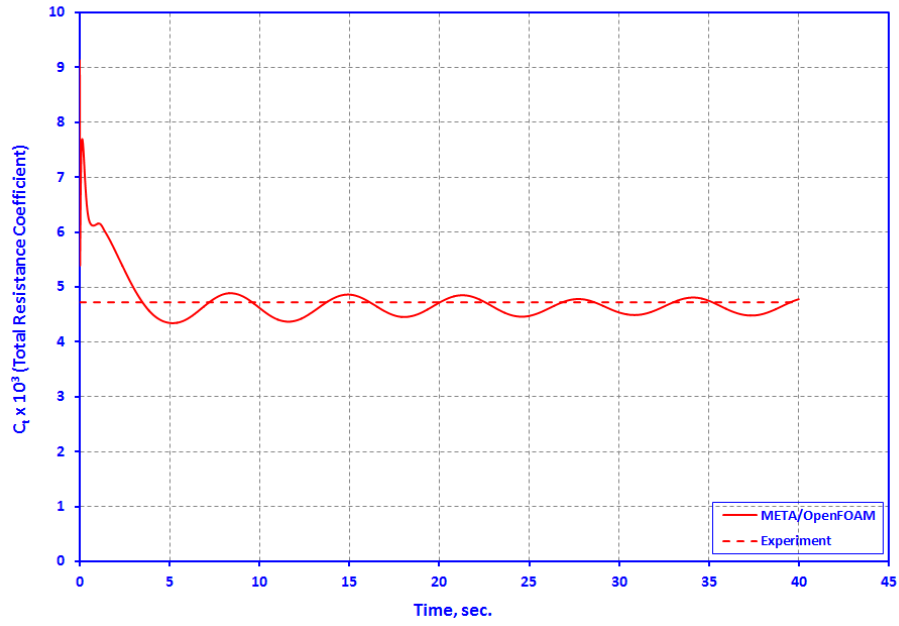


Figure 6: Total resistance coefficient ( $C_t$ ) vs. time for Froude number  $Fr = 0.36$ .

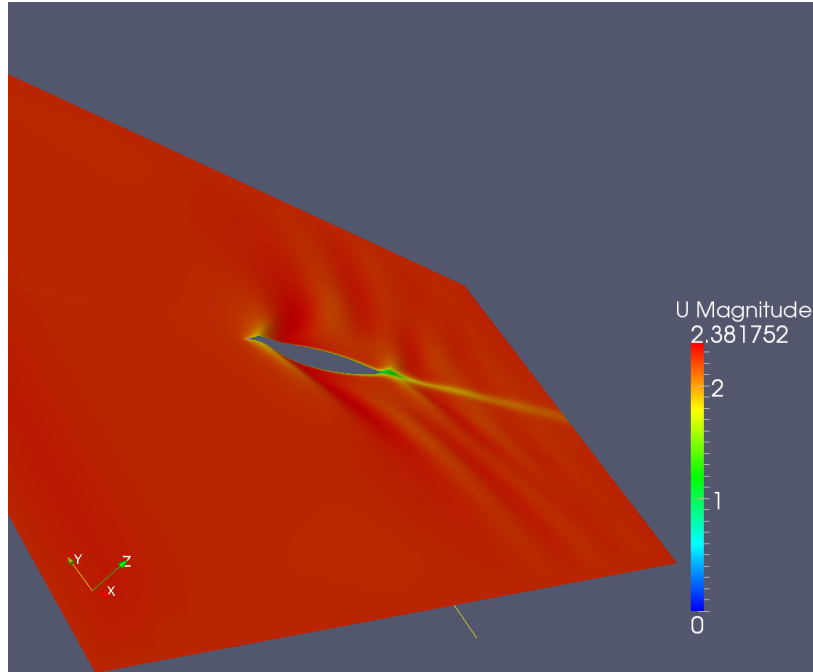


Figure 7: Wave contour for Froude number  $Fr = 0.36$  with velocity color legend.

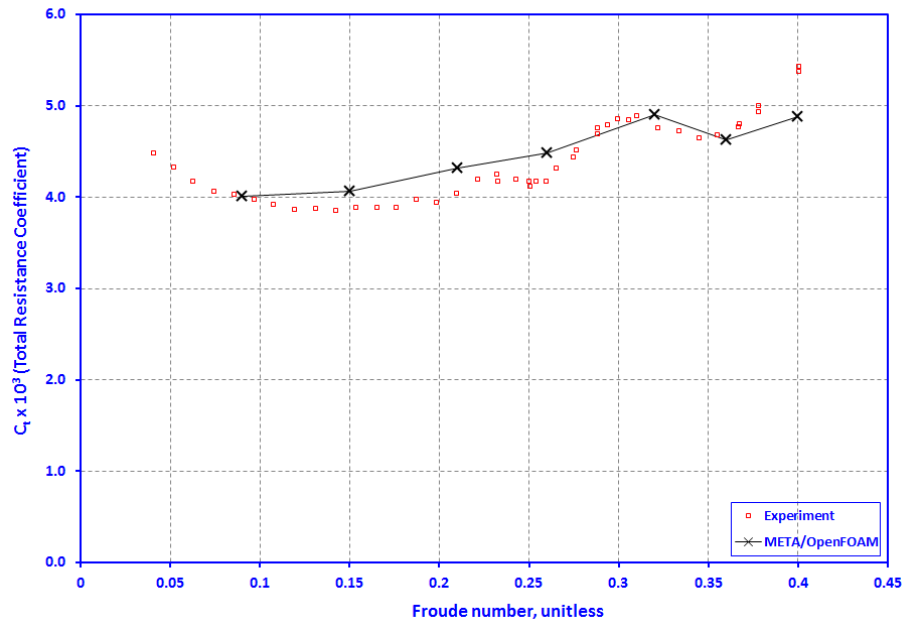


Figure 8: Comparison of META tools simulations and empirical data [Kajitani et al., 1983] for total resistance coefficient vs. Froude number.

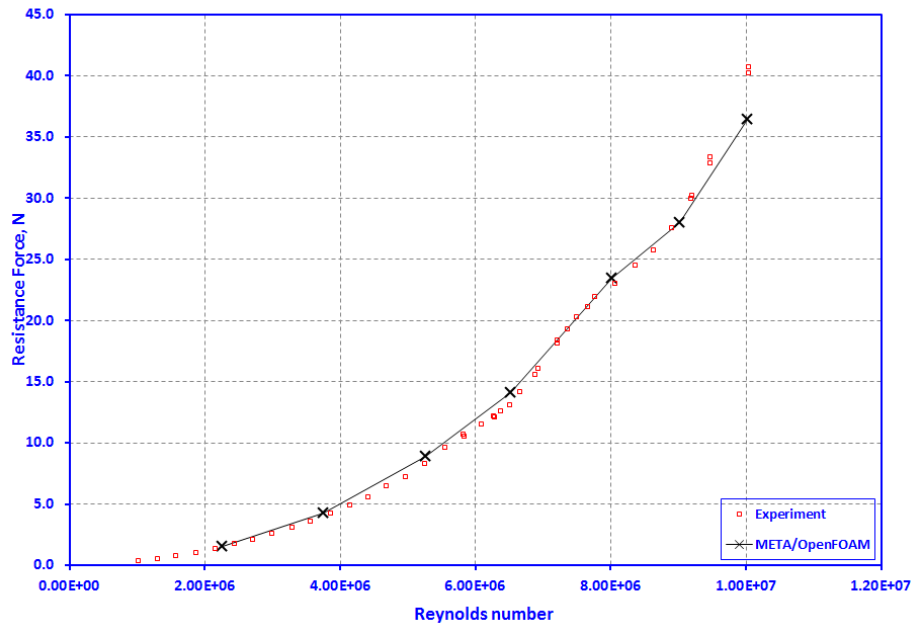


Figure 9: Comparison of META tools simulations and empirical data [Kajitani et al., 1983] for resistance force vs. Reynolds number.

## 5 Conclusions

A Wigley hull is modeled and simulated under calm water conditions with fixed trim and sinkage in a 3D computational domain with Froude numbers  $0.09 \leq Fr \leq 0.40$  and Reynolds numbers  $2.25 \times 10^6 \leq Re \leq 1.00 \times 10^7$ . The META tool suite which leverages OpenFOAM® is used for fluid analyses. Published empirical data [Kajitani et al., 1983] is used to validate the META tool suite. Agreement between the simulations and the empirical data is very good leading to the conclusion that the META tools are validated to provide high fidelity drag force estimates for hulls of similar geometry and similar flow conditions.

## References

- [Burciu et al., 2012] Burciu, Z., Kraskowski, M., and Gerigk, M. (2012). Analysis of the process of water entry of an amphibious vehicle.
- [Gerlach et al., 2006] Gerlach, D., Tomar, G., Biswas, G., and Durst, F. (2006). Comparison of volume-of-fluid methods for surface tension-dominant two-phase flows. *International Journal of Heat and Mass Transfer*, 49(3):740–754.
- [Guo et al., 2012] Guo, Z., Pan, Y., Fu, Y., and Pei, C. (2012). Numerical simulation of resistance of a amphibious vehicle. In *World Automation Congress (WAC), 2012*, pages 1–3. IEEE.
- [Hirt and Nichols, 1981] Hirt, C. W. and Nichols, B. D. (1981). Volume of fluid (vof) method for the dynamics of free boundaries. *Journal of computational physics*, 39(1):201–225.
- [Kajitani et al., 1983] Kajitani, H., Miyata, H., Ikehata, M., Tanaka, H., Adachi, H., Nami-matsu, M., and Ogiwara, S. (1983). The summary of the cooperative experiment on Wigley parabolic model in Japan. Technical report, DTIC Document.
- [Kay, 2007] Kay, M. (2007). *Practical hydraulics*. Taylor & Francis.
- [Kotowski et al., 2012] Kotowski, M., Barnat, W., Grygorowicz, M., Panowicz, R., and Dybcio, P. (2012). Experimental and numerical buoyancy analysis of tracked military vehicle. *Journal of KONES*, 19:321–324.
- [Larsson, 1997] Larsson, J. (1997). *CFD online*. Jonas Larsson.
- [Maki, 2011] Maki, K. (2011). Ship resistance simulations with OpenFOAM. In *6th Open-FOAM Workshop*.
- [Marek et al., 2008] Marek, M., Aniszewski, W., and Boguslawski, A. (2008). Simplified volume of fluid method (svof) for two-phase flows. *TASK Quarterly*. v12, pages 255–265.
- [Menter, 1993] Menter, F. R. (1993). Zonal two equation kappa-omega turbulence models for aerodynamic flows. *c1993*, 1.
- [OpenFOAM Development Team, 2013] OpenFOAM Development Team (2013). OpenFOAM User Guide. [Online; accessed 25-Sep-2013, <http://www.openfoam.org/archive/2.2.0/docs/>].

- [Ren and Ou, 2009] Ren, N. and Ou, J. (2009). Numerical simulation of surface roughness effect on wind turbine thick airfoils. In *Power and Energy Engineering Conference, 2009. APPEEC 2009. Asia-Pacific*, pages 1–4. IEEE.
- [Rodrigues et al., 2011] Rodrigues, M. A., Padrela, L., Geraldés, V., Santos, J., Matos, H. A., and Azevedo, E. G. (2011). Theophylline polymorphs by atomization of supercritical anti-solvent induced suspensions. *The Journal of Supercritical Fluids*, 58(2):303–312.
- [Wigley, 1934] Wigley, W. C. S. (1934). A comparison of experiment and calculated wave-profiles and wave-resistances for a form having parabolic waterlines. *Proceedings of the Royal Society of London. Series A, Containing Papers of a Mathematical and Physical Character*, 144(851):pp. 144–159.