

# META Fluid Analysis Tool

Vanderbilt University, Institute for Software Integrated Systems

February 4, 2014

## Abstract

The META fluid analysis tool predicts the drag coefficient of a given geometry traveling through a stated fluid. For the Defense Advanced Research Project Agency's (DARPA's) Advanced Vehicle Make (AVM) program, the META fluid analysis tool is tuned for amphibious vehicles. Three aquatic tiers are used to differentiate the fidelity of the prediction and the sea state for the vehicle. The tool leverages empirical correlations from existing military amphibious vehicles for the first aquatic tier. computational fluid dynamics is used for the higher tiers. Empirical correlations yield a simple, closed-form solution which produces the predicted drag for a given design within a few minutes. The computational fluid dynamics tiers require significantly more computing time having run times ranging from a day to a few days. This document presents the theory of operation of the META fluid analysis tool and covers the input variables and data output.

## Nomenclature

$c_d$	Drag coefficient
$B$	Beam at midship (m)
$D$	Draft at midship (m)
$D_\omega$	Cross-diffusion term
$F_d$	Drag force (N)
$G_k$	Generation of $k$
$G_\omega$	Generation of $\omega$
$k$	Turbulence kinetic energy (joule)
$L$	Waterline length (mm)
$m$	Mass (kg)
$S$	Wetted surface area (mm <sup>2</sup> )
$S_k$	User-defined source term of $k$
$S_\omega$	User-defined source term of $\omega$
$t$	Time (s)
$u, V$	Velocity (m/sec)
$v$	Volume (mm <sup>3</sup> )

$W$	Weight (N)
$Y_k$	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

The META fluid analysis tool predicts the drag coefficient of a given geometry traveling through a stated fluid. For the Defense Advanced Research Project Agency's (DARPA's) Advanced Vehicle Make (AVM) program, the META fluid analysis tool is tuned for amphibious vehicles. The tool is designed to be run by non-specialist engineers with minimal input. While drag coefficient could be a design metric, the AVM program has metrics that depend upon the drag coefficient predicted by the META fluid analysis tool. For example, maximum speed in up to sea state three is an AVM metric that depends upon drag prediction.

## 2 Theory

As an amphibious vehicle transits the bodies of fresh or salt water, part of the vehicle is below water and part of the vehicle is above water. This presents a two-phase fluid flow problem with a liquid phase for the water and a gaseous phase for the air [Yeoh and Tu, 2009, Wrobel and Brebbia, 1993]. As the vehicle moves through the water, a bow wave is formed which contributes significantly to the total vehicle drag. Modeling the two fluid phases requires addressing the bow wave, the water free surface, and gravity effects.

Drag force is described as

$$F_d = \frac{1}{2} c_d \rho V^2 S \quad (1)$$

where  $F_d$  is the total resistance force imparted by the fluid on the object,  $c_d$  is the drag coefficient of the object,  $\rho$  is the fluid density,  $V$  is the object's velocity, and  $S$  is the object's wetted surface area. The META fluid analysis tool predicts the drag force  $F_d$  for a specified velocity  $V$  whereas the drag coefficient  $c_d$  is unknown. Thus, Equation 1 can be rewritten as

$$c_d = \frac{2F_d}{\rho V^2 S} \quad (2)$$

to recover the unknown drag coefficient  $c_d$ . Equation 1 can then be used to estimate drag force  $F_d$  for a velocity range in a dynamics simulation. In general, the drag coefficient  $c_d$  is not constant for

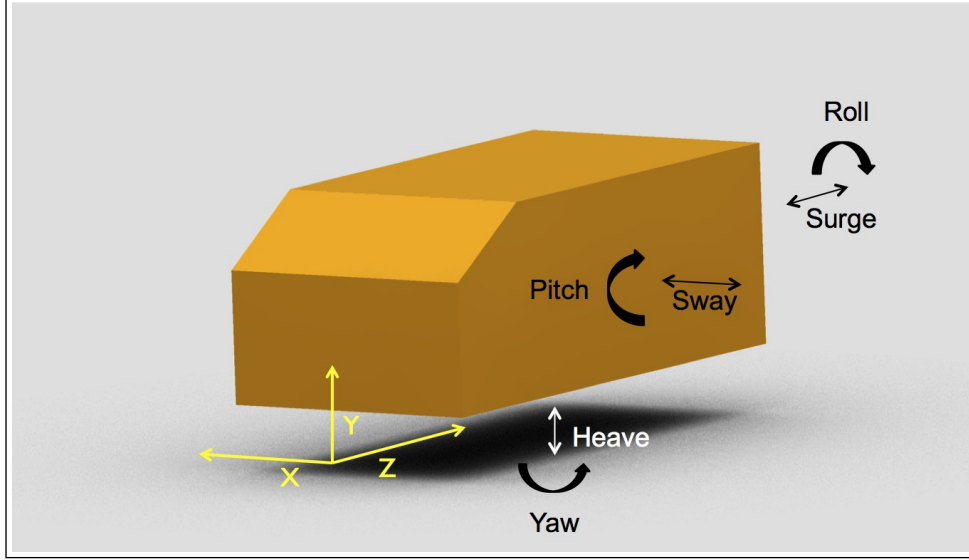


Figure 1: Aquatic vehicle degrees of freedom.

a given body shape for a large velocity range but can be assumed constant for a limited velocity range.

## 2.1 Tiers

Three aquatics tiers are used to differentiate the fidelity of the prediction and the sea state. Figure 1 illustrates the degrees of freedom for a vehicle in water while Figure 2 illustrates all three tiers.

The tool leverages empirical correlations from existing military amphibious vehicles for the first aquatics tier [Hoyt, 1994, Helvacioğlu et al., 2011]. Computational Fluid Dynamics (CFD) is used for the higher tiers [Hoffmann, 1989]. Empirical correlations yield a simple, closed-form solution which produces the predicted drag for a given design within a few minutes. This computation time includes creating the assembly, computing the mass, and computing the drag coefficient. The computation of the drag coefficient itself requires less than a second. The CFD tiers require significantly more computing time with simulations having run times ranging from a day to a few days.

For Tier 1 and for  $V < 10$  knots (5.144 m/s), the drag force is predicted using

$$\frac{F_d}{W} = (-0.00004V^3 + 0.0025V^2 - 0.0029V) \quad (3)$$

where  $F_d$  is the drag force,  $W$  is the vehicle weight, and  $V$  is velocity. The quantity  $F_d/W$  is a non-dimensional number, therefore when computing the drag force  $F_d$ , the units for  $F_d$  will be the same as the units for the weight  $W$  used. For Tier 1 and  $V \geq 10$  knots (5.155 m/s), the drag force uses the relation

$$\frac{F_d}{W} = (0.0000145V^3 - 0.00785V^2 + 0.1343V - 0.5210). \quad (4)$$

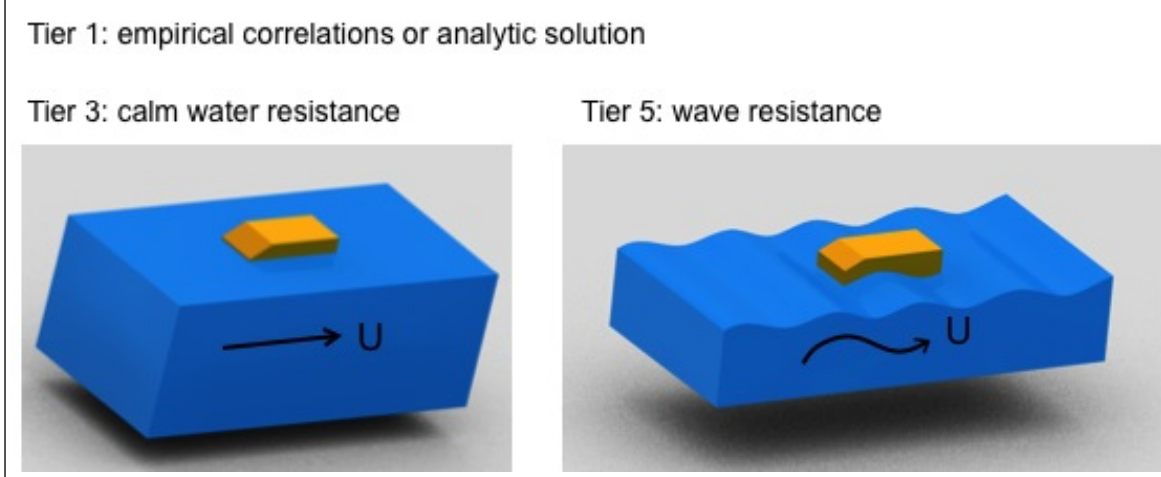


Figure 2: Aquatic tiers 1 to 5.

Equations 3 and 4 are derived from towing-tank experiments conducted on existing military amphibious vehicles.

CFD, used in Tiers 3 and 5, leverages OpenFOAM<sup>®</sup>, a free and open source set of tools released under the GNU General Public License. OpenFOAM<sup>®</sup> employs the finite-volume method, a method for representing and evaluating partial differential equations. A computational domain of fluid surrounding the object being analyzed is used to bound the problem and that domain is discretized into elements.

## 2.2 OpenFOAM<sup>®</sup> Details

Meshing is performed by using blockMesh and snappyHexMesh tools. Tier 3 uses the interFoam solver while Tier 5 uses the waveFoam solver.

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  $k-\epsilon$  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 5 and 6 [Ren and Ou, 2009].

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}(\Gamma_k \frac{\partial k}{\partial x_j}) + \tilde{G}_k - Y_k + S_k \quad (5)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j}(\Gamma_\omega \frac{\partial \omega}{\partial x_j}) + G_\omega - Y_\omega + D_\omega + S_\omega \quad (6)$$

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, the SST  $k-\omega$  turbulence model behaves appropriately for

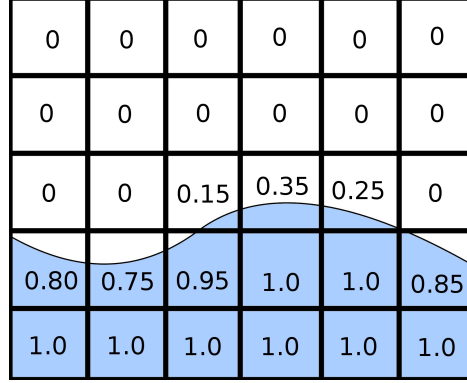


Figure 3: Approximated interface by using volume fraction  $\alpha$ .

a wider class of flows including ship hydrodynamics applications [Ren and Ou, 2009].

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.

$$\frac{\partial \alpha}{\partial t} + \frac{\partial u_i \alpha}{\partial x_i} = 0 \quad (7)$$

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

$$\alpha(x, t) = \alpha \quad (8)$$

$$\mu(x, t) = \mu_{water}\alpha + \mu_{air}(1 - \alpha) \quad (9)$$

$$\rho(x, t) = \rho_{water}\alpha + \rho_{air}(1 - \alpha) \quad (10)$$

The passive advection of void fraction can be represented by equation 11, since the fluid type remains constant along particle paths. [Gerlach et al., 2006]

$$\frac{\delta\alpha}{\delta t} + u.\nabla\alpha = 0 \quad (11)$$

Equation 11 can be reformulated into a conservative form shown in equation 12 by using the continuity equation. [Gerlach et al., 2006]

$$\frac{\delta\alpha}{\delta t} + \nabla.u\alpha = 0 \quad (12)$$

By solving equation 12 numerically, volume fraction  $\alpha$  can be calculated. After that, by using the updated volume fraction, fluid properties is calculated by using equations 9 and 10 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 approximate the interface at that location [Hirt and Nichols, 1981].

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]. The algorithm of the PISO and SIMPLE procedures are as follows:

PISO Algorithm:

1. Momentum predictor stage in which the momentum equation is solved that gives an approximation of the new velocity field.
2. Pressure solution stage in which the pressure equation is formulated by using predicted velocities.

3. Explicit velocity correction stage in which the velocity field is corrected explicitly by using the pressure distribution.

SIMPLE Algorithm:

1. By solving the momentum equation, an approximation of the velocity field is obtained. The calculations in this step are under-relaxed in an implicit manner.
2. In order to obtain the new pressure distribution, the pressure equation is formulated and solved.
3. A new set of conservative fluxes is calculated and the pressure equation is solved again in order to obtain a better approximation of the “correct” pressure field. In this step, in order to eliminate the pressure error, under-relaxation is implemented into the pressure solution.

Courant number can be represented in simple terms by using

$$C_O = U \frac{\Delta t}{\Delta x}. \quad (13)$$

For arbitrary polyhedral finite volume, Equation 13 becomes

$$C_O = \frac{U \cdot S_f}{d \cdot S_f} \Delta t \quad (14)$$

where  $d$  is the vector from pole center to neighbor center.

### 3 Operation

The META fluid analysis tool is operated through the META tools via the Generic Modeling Environment (GME) and CyPhy. CFD test benches are built and configured with the desired fluid analysis options. Consult the Generic Modeling Environment (GME) documentation and other META documentation for details on GME, CyPhy, and building assemblies, design spaces, and test benches.

When creating a new CFD test bench, the user right clicks the desired test bench folder, selects Insert Model, then selects CFD Test Bench. The minimum required test bench objects for a CFD test bench are: Computer Aided Design (CAD) workflow, System Under Test, CFDComputation, metrics, HydroStaticsSolverSettings, and hydrodynamic solver settings (CorrelationSettings, CalmWaterSolverSettings, or WaveResistanceSolverSettings). Optional objects include Parameters, PostProcessing, and Properties.

When a CFD test bench is executed, the CAD assembly is created and exported to stl

The fluid domain size is set according to the bounding box of the assembly. The fluid domain is set to 10 times the assembly length and 10 times the assembly width. The fluid domain height is set at 10 times the assembly height with the lower half composed of liquid and the upper half composed of air.

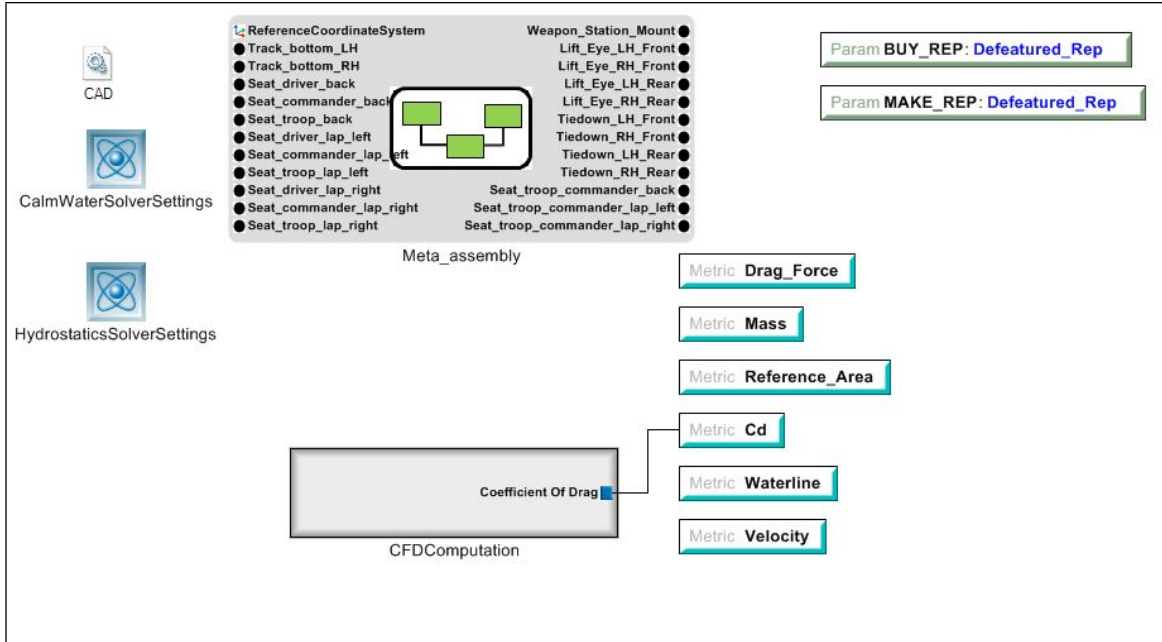


Figure 4: Example CFD Test Bench.

Figure 4 illustrates a CFD test bench in CyPhy. Orientation of the assembly in the fluid domain can be controlled through the ReferenceCoordinateSystem object. Placing this object in a component in the CyPhy design and exposing that ReferenceCoordinateSystem up to the top-level assembly instructs the META tools to use that component to orient the assembly. Figure 4 contains the ReferenceCoordinateSystem (top left) at the top assembly level. This ReferenceCoordinateSystem is connected to another ReferenceCoordinateSystem inside the sub-assembly which in turn is connected to the ReferenceCoordinateSystem in the hull component (Figure 5). The component used to orient the assembly needs the default datums to be properly configured. The ASM\_FRONT, ASM\_RIGHT, and ASM.TOP datums in a .asm component or FRONT, RIGHT, TOP datums for a .prt component must be set to their proper orientation. For example, viewing the component normal to ASM\_FRONT must result in a view of the component's front.

Test bench attributes allow the user to control some aspects of the CFD simulation:

- **CopyResults.** The default value (False) instructs the tools to copy some, but not all of the results from VehicleForge to the local machine when the test bench is finished. Selecting True instructs the tools to copy all results, including CFD databases, from VehicleForge to the local machine. The size of all results can be many gigabytes of data and therefore the True option should be used sparingly.

### 3.1 CAD Representations

The META tools support multiple representations of the geometry. Multiple representations allow component designers to include featured representations and defeatured representations of the geometry in the component package. CFD test benches may benefit from using defeatured



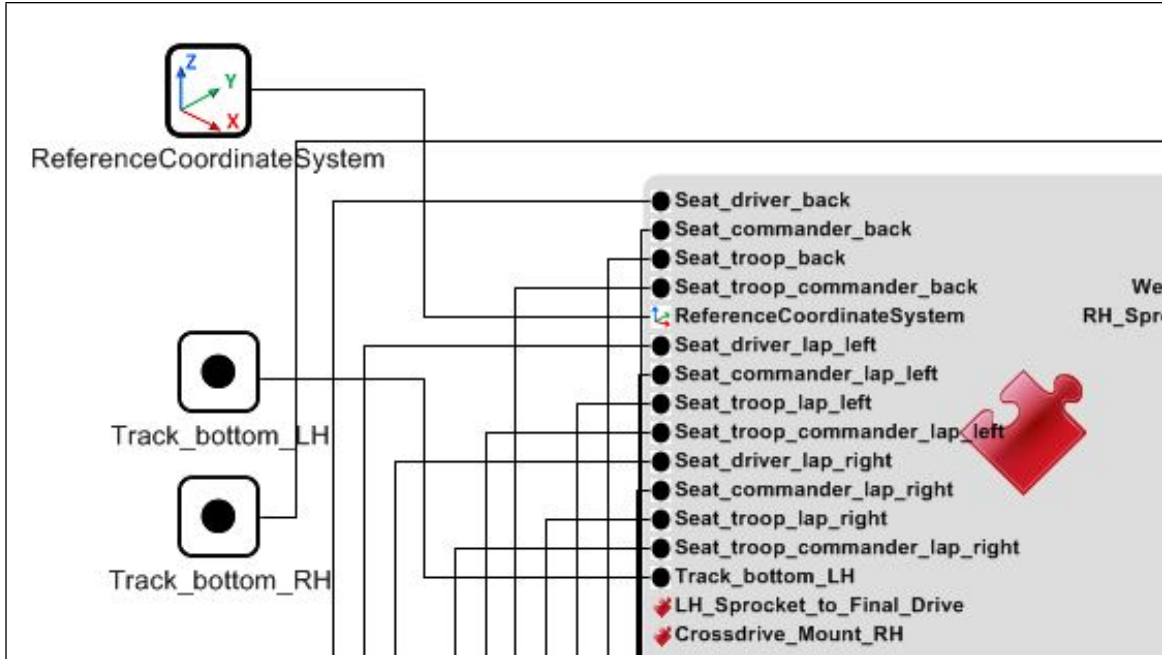


Figure 5: Connect ReferenceCoordinateSystem objects to designate a component for assembly orientation in the fluid domain.

Table 1: Test Bench CAD Representation Parameters.

Description	Parameter Name
Buy component representation	BUY_REP
Make component representation	MAKE_REP
Default component representation	DEFAULT_REP

geometry in the analysis through shorter mesh times and reduced resources. The META tools allow any test bench that uses CAD to specify the CAD representation to be used for the analysis. A separate specification can be made for buy components and for make components. The buy or make distinction is made according to the value in the component's manufacturing model "procurement\_make\_or\_buy" parameter. Table 1 details the Parameter objects used to specify the representations and Table 2 details the parameter values needed to specify a particular representation. If the requested representation is available in the CAD file, that representation is used otherwise the specified DEFAULT\_REP is used if that representation is available. If the specified DEFAULT\_REP is not available, Creo's default representation, Master Rep, is used.

Table 2: Test Bench CAD Representation Parameter Values.

CAD Representation	Parameter Name
Featured representation	Featured_Rep
Defeatured representation	Defeatured_Rep
Default representation	Master Rep (or other desired default representation)

Table 3: Standard CFD Metrics.

Description	Metric Text
Drag Coefficient	Cd
Drag Force	Drag_Force
Mass (kg)	Mass
Reference Area (wetted surface area in mm <sup>2</sup> )	Reference_Area
Velocity (m/s)	Velocity
Waterline (distance in mm from assembly bottom to waterline)	Waterline

## 3.2 Metrics

CFD test benches may contain any or all standard CFD metrics along with any custom metrics the user wants to define. The standard CFD metrics are contained in Table 3. These metrics are written to the `testbench_manifest.json` file in the results folder. Custom metrics may be added to support scoring or other purposes. For example, a scoring metric could be `Cd_Sea_State_3_Acceptable` that is either true or false depending upon the estimated drag coefficient for the design in sea state three. The scoring metric could be true if the drag coefficient is less than a certain value and false if the drag coefficient is greater than that value. A post processing python script, specified by the `Post-Processing` object, would need to be authored by the user to examine the `testbench_manifest.json` results to determine and write the value for `Cd_Sea_State_3_Acceptable`.

## 3.3 HydrostaticsSolverSettings

The hydrostatics solver is used to compute the waterline of the assembly. It uses the assembly total mass, computes displaced volume according to the formula

$$v = \frac{m}{\rho} \quad (15)$$

where  $v$  is displaced fluid volume,  $m$  is the total assembly mass, and  $\rho$  is the density of the fluid. The fluid type and temperature are specified in the `ObjectInspector`. The hydrostatics solver computes the waterline and reference area (wetted surface area) based on the displaced volume.

If `HydrostaticsSolverSettings` is omitted from the test bench, the waterline is assumed to be 1 m (i.e., the bottom-most point on the assembly is 1 m below the fluid surface) and the reference area is assumed to be 47 m<sup>2</sup>. If the test bench name is "Wigley\_hull" and `HydrostaticsSolverSettings` is omitted, the waterline is assumed to be 0.25 m and the reference area to be 2.3796 m<sup>2</sup> [Yapar et al., 2013].

## 3.4 Hydrodynamic Solver Settings

One of three hydrodynamics solvers may be specified for a CFD test bench.

### 3.4.1 CorrelationSettings

`CorrelationSettings` represents CFD Tier 1 which leverage empirical correlations discussed above. The input to the solver, specified in the `ObjectInspector` window, is velocity in m/s.

### 3.4.2 CalmWaterSolverSettings

CalmWaterSolverSettings represents CFD Tier 3 which is zero degrees of freedom with 3D flow and calm water. Inputs to the solver include velocity in m/s, fluid material (fresh water or salt water), fluid temperature in Celsius.

### 3.4.3 WaveResistanceSolverSettings

WaveResistanceSolverSettings represents CFD Tier 5 which is zero degrees of freedom with 3D flow and wave conditions. Inputs to the solver include velocity in m/s, fluid material (fresh water or salt water), fluid temperature in Celsius, cyclic wave frequency in Hz, wave direction in degrees, wave number in radians/m, and wave height in m.

Wave direction is specified in degrees with  $0^\circ$  representing head waves and  $90^\circ$  representing wave direction abeam the vehicle's starboard side.

## 3.5 Data output

The fluid analysis tool provides estimated drag force  $F_d$ , estimated drag coefficient  $c_d$ , and reference area  $S$  as data output for the given vehicle design and input variables. This data can be used in dynamics simulations for estimating performance such as aquatic speed and aquatic range.

## 4 Validation

Validation models are distributed with the tools and are installed in the META Documents folder.

## 5 Known Limitations

The META CFD tool currently requires the assembly to be oriented with x+ right, y+ up, and z+ back. This contrasts with the META tools default of x+ back, y+ right, and z+ up. The transformation is handled within the tools but the ASM\_FRONT, ASM\_RIGHT, and ASM.TOP datums in a .asm component or FRONT, RIGHT, TOP datums for a .prt component must be set to their proper orientation. Additionally, the ReferenceCoordinateSystem must be present in the component in CyPhy and propagated up to the top-level assembly.

The META CFD tool currently supports zero degrees of freedom and three-dimensional analysis. Additional tiers will be added in the future to handle multiple degrees of freedom and two-dimensional analysis.

Wave direction other than zero degrees is currently not supported therefore only head seas are currently supported.

Air drag is not considered with the two-phase flow. The liquid fluid dominates the solution rendering air drag an insignificant portion of the overall drag value.

The following CFD test bench objects are currently not supported: ComplexityMetric, ComponentAssemblyTopLevelSystemUnderTest, ComponentRefTopLevelSystemUnderTest, ComponentTopLevelSystemUnderTest, Constant, CustomFormula, MetricConstraint, Property, Random-

ParameterDriver, Requirement Link, SimpleFormula, Test Component, TestComponentTopLevel-SystemUnderTest, and ValueFlowTypeSpecification. Additionally, CFD test benches currently cannot be included in a Suite of Test benches (SOTs) and value flow is not supported (i.e., values cannot flow from a component, computation, or another test bench into or out of a CFD test bench).

The hydrostatics tool currently cannot compute waterline for non-zero roll and pitch angles. The waterline and reference area computation currently uses surrogate equations in lieu of a solution based on geometry. The tool does not compute center of buoyancy. When this feature is supported, the hydrostatics tool will be used to determine roll stability at specified heel angles and reserve buoyancy.

## References

- [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.
- [Helvacioğlu et al., 2011] Helvacioğlu, S., Helvacioğlu, I. H., and Tuncer, B. (2011). Improving the river crossing capability of an amphibious vehicle. *Ocean Engineering*, pages 2201–2207.
- [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.
- [Hoffmann, 1989] Hoffmann, K. A. (1989). Computational fluid dynamics for engineers(book). Austin, TX: *Engineering Education System*, 1989.
- [Hoyt, 1994] Hoyt, J. G. (1994). Test and evaluation of the Propulsion System Demonstrator. Technical report, Naval Warfare Center. Report CRDKNSWC/HD-1452-01.
- [Larsson, 1997] Larsson, J. (1997). *CFD online*. Jonas Larsson.
- [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 antisolvent induced suspensions. *The Journal of Supercritical Fluids*, 58(2):303–312.

- [Wrobel and Brebbia, 1993] Wrobel, L. C. and Brebbia, C. (1993). *Computational Modelling of Free and Moving Boundary Problems II: Second International Conference on Computational Modelling of Free and Moving Boundary Problems 93*. WIT Press.
- [Yapar et al., 2013] Yapar, O., Myers, M. R., Varasteh, M., Neema, S., and Bapty, T. (2013). META tool suite fluid analysis validation with the Wigley hull. In *Open Source CFD International Conference*. ICON.
- [Yeoh and Tu, 2009] Yeoh, G. H. and Tu, J. (2009). *Computational techniques for multiphase flows*. Butterworth-Heinemann.