Computational Fluid Dynamics (CFD)

User Tutorial for the Computational Fluid Dynamics (CFD) Test Bench and Tool

May 2, 2014

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

VANDERBILT
UNIVERSITY

1.0 Purpose

The purpose of the Computational Fluid Dynamics (CFD) Test Bench and Tool is to provide the designer with aquatic performance of the design. This test bench calculates the coefficient of drag of the design and the drag force acting on it accurately under different aquatic conditions.

2.0 Procedures

The instructions in this manual assume that the user has installed the latest version of GME and has access to Creo, either locally or via the remote server.

2.1 Installation

Initial installation of this test bench will be provided with the installation of the CyPhy tool suite. Future editions of the tool may be packaged as a standalone or combined test bench installation package.

2.2 Tool

The CFD Test Bench is the test bench in GME that the designer uses to interface the OpenFoam software, which is an open-source C++ based CFD toolbox. OpenFoam software is running in the VehicleFORGE servers which accepts a design, simulates it by using numerical CFD procedures, and returns an accurate simulation result. CFD test bench performs a numerical simulation procedure to determine the performance of the design under aquatic conditions. The simulation will determine the coefficient of drag and the drag force acting on the design by taking into account the input given by the user.

3.0 Requirements Tested

- **Coefficient of Drag (unitless):** The drag coefficient is a number which is used to model all of the complex dependencies of drag on a shape, inclination, and some flow conditions. The drag coefficient is equal to the drag force divided by the quantity: density times wetted surface area times one half of the velocity squared.
- **Drag Force (N):** The drag force is a force that opposes the motion of an object traveling in/on fluids. This is a mechanical force that results from the

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

VANDERBILT
UNIVERSITY

interaction of solid bodies with fluids such as water and air. Drag is as a result of difference in velocity between the object and fluid.

- **Center of Buoyancy (x,y,z) (mm):** Comparing the center of gravity and the center of buoyancy yields a measure of hydrostatic stability, the ability of the waterborne craft to right itself when disturbed from level trim and heel.
- **Reserve Buoyancy (%):** Reserve buoyancy is an indicator of the ability to negotiate various sea states and the ability to carry additional cargo.

4.0 Operation

The design is assembled into a 3D CAD representation, including the customization / generation of any parameterized components. Input data taken from the user is also assembled together for a CFD simulation by OpenFoam CFD toolbox. The whole information is packaged up and sent via remote server for the CFD simulation.

5.0 Test Bench Structure

This test bench contains a system under test that is to be assembled and analyzed for its aquatic performance.

Assumptions: Without the HydrostaticsSolverSettings, the CFD test bench assumes the bottom point of the object is 1m (25cm for Wigley hull example) below the water line. If the dimension of the object in the z-axis direction is less than 1m (25cm for Wigley hull example) then the object is fully submerged. If it is higher than 1m, then the bottom 1m of the object is submerged the rest is above the water. CFD test bench also assumes that the object moves in the negative x-direction.

To begin, we will import our components.

Step 1

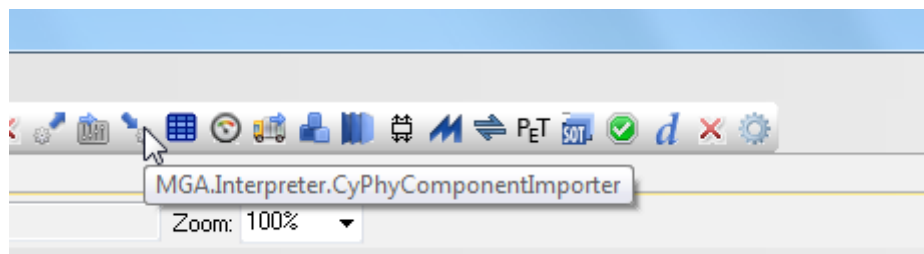In GME, on top of the screen, click "MGA.Interpreter.CyPhyComponentImporter". (Fig. 1)

Figure 1

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

VANDERBILT
UNIVERSITY

Step 2

Select the correct acm file corresponding to your components and click Open. (Fig. 2)

Figure 2

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

VANDERBILT
UNIVERSITY
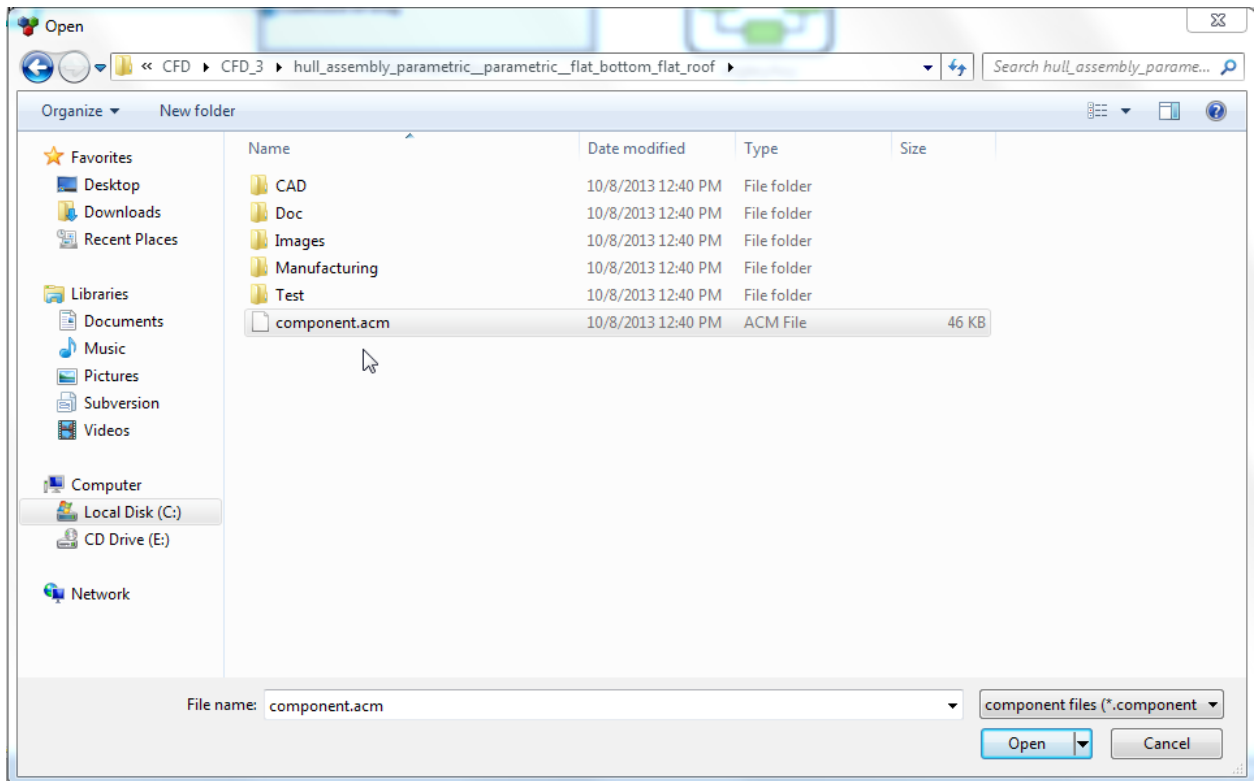
Step 3

CyPhy will automatically import your components into the GME. Now you should be able to see your components under the "Imported_Components" folder (Fig. 3). CAD parts should have the same material definitions with CyPhy. If that is not the case, modify CAD parts the same material definitions and make it the same with CyPhy.

Figure 3

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

VANDERBILT
UNIVERSITY
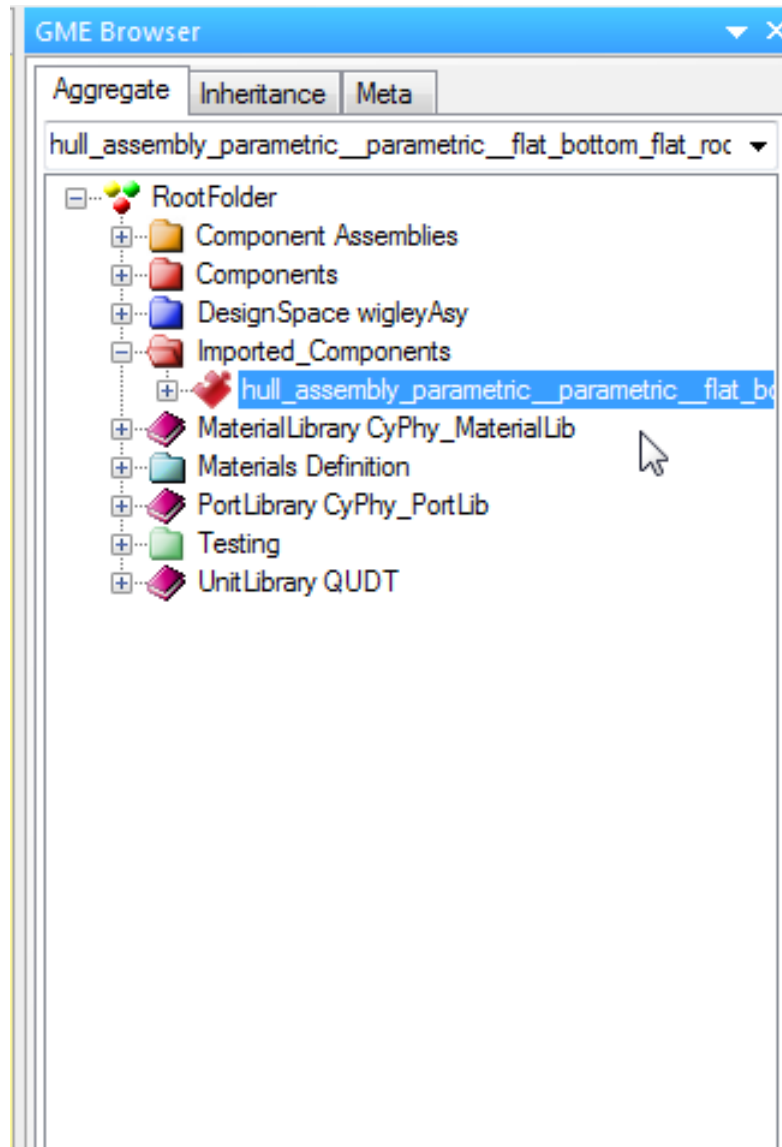
Step 4

Now we will create the test bench. In the GME Browser, right click on the Testing folder, insert a new CFDTestBench, and name it. (Fig.4)
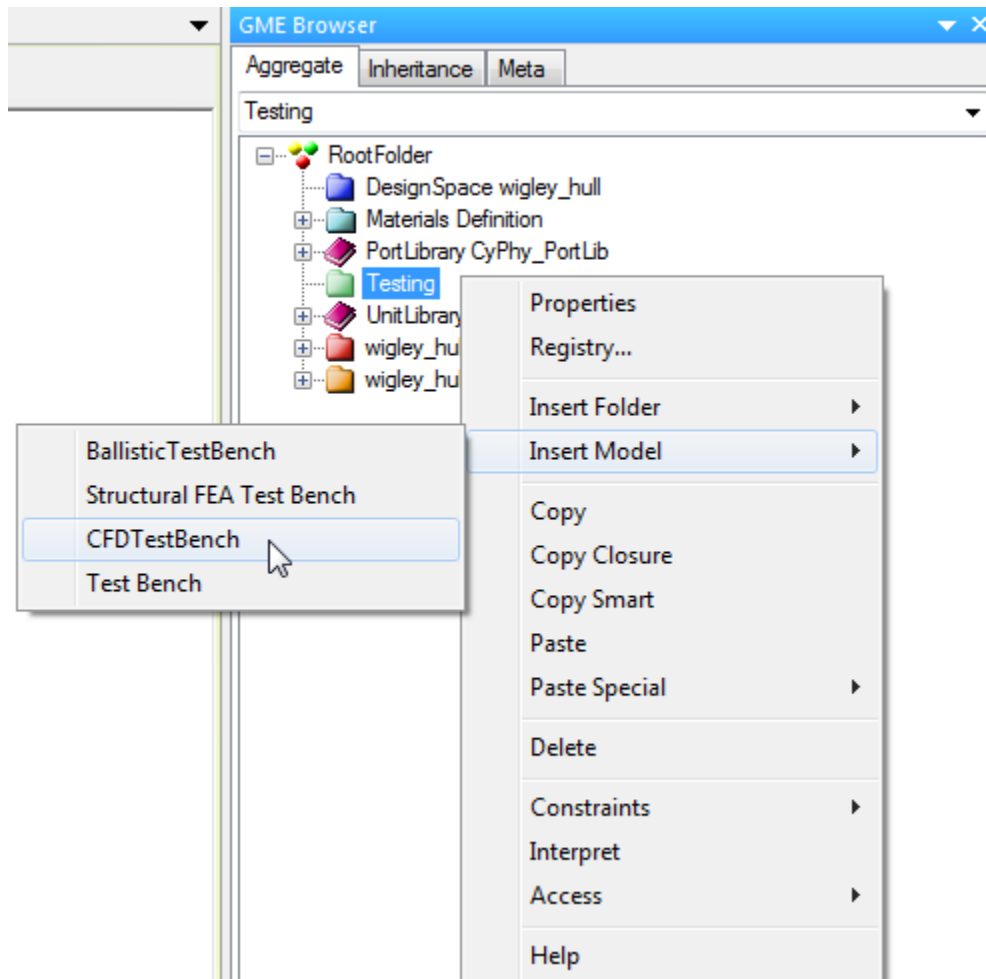
Figure 4

Step 5

Double click on the created test bench. GME should now look like this. (Fig.5)
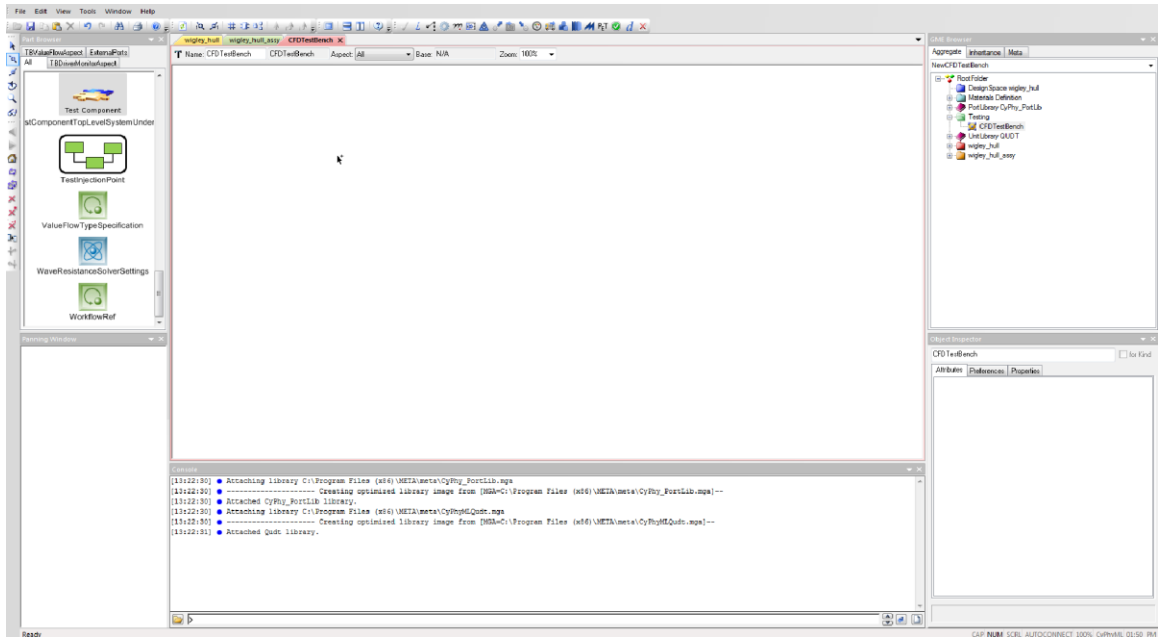

Figure 5

Step 6

In the Part Browser, find the CFDComputation part. This part will execute all of the numerical calculations needed for CFD Analysis. (Fig. 6)
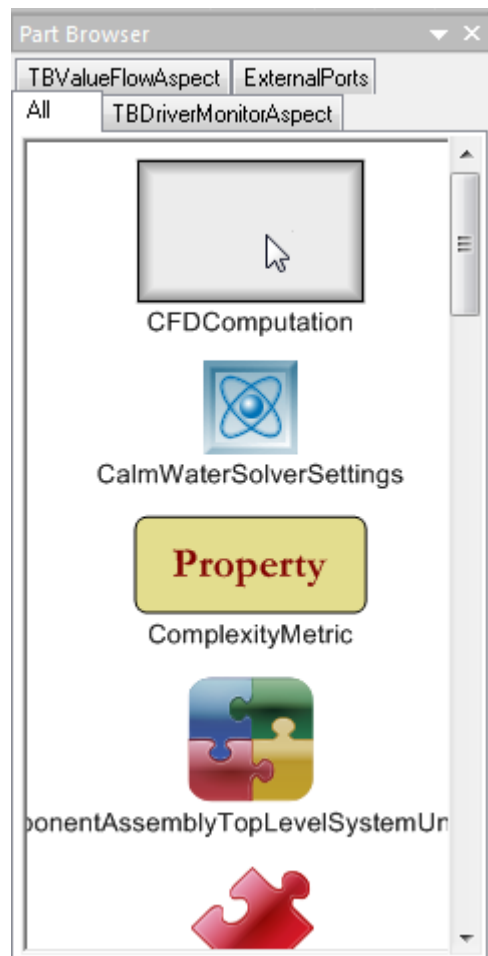

Figure 6

Step 7

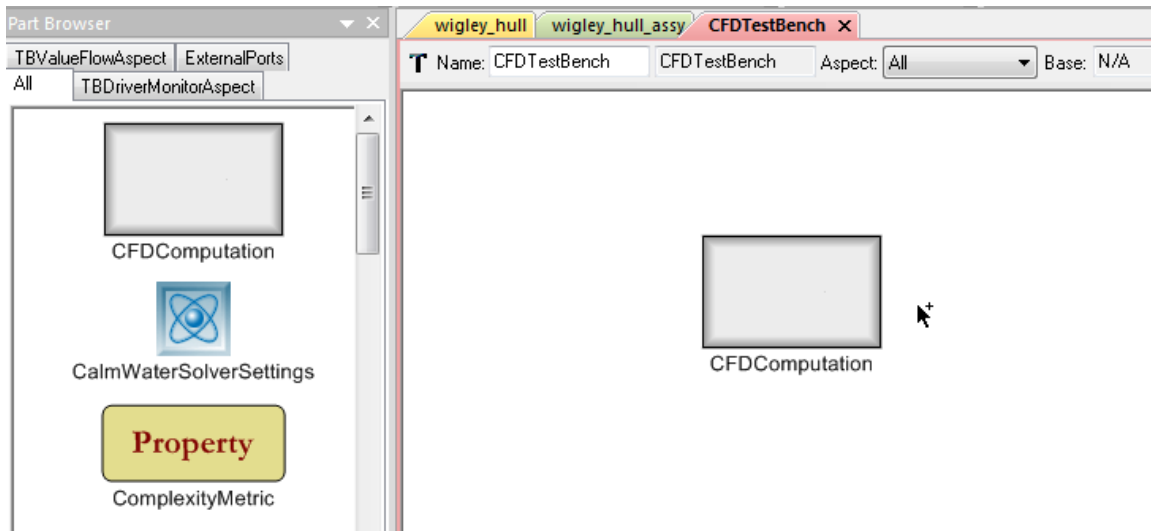Drag the CFDComputation part into the test bench. (Fig. 7)

Figure 7

Solver Settings

We will now define the solver settings. Three options are available here: CorrelationSettings, CalmWaterSolverSettings and WaveResistanceSolverSettings. Only one of these should be used at a time, depending on the purpose of the CFD simulation. CorrelationSettings is for predicting the performance of the vehicle by correlating present experimental data in the database, which have created from towing-tank tests of real-life size amphibious vehicle models. CalmWaterSolverSettings is for CFD simulations undertaken to compute performance of the vehicle for calm water conditions (vehicle traveling steadily in calm water) and WaveResistanceSolverSettings is for CFD simulations undertaken to compute performance of the vehicle in wave environments (vehicle traveling through the waves).

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

All three of these can be found in the Part Browser window. On the object inspector of the CorrelationSettings, CalmWaterSolverSettings and WaveResistanceSolverSettings items, under attributes, settings of the solvers can be found. Description of each setting is presented in the below subsections. Also the graphical representation of each CFD aquatic tier is shown in (Fig.8) and (Fig.9).
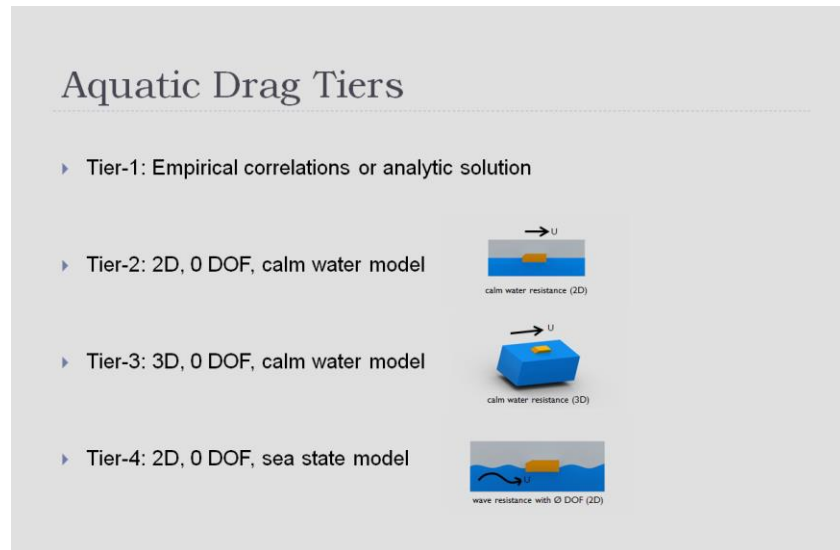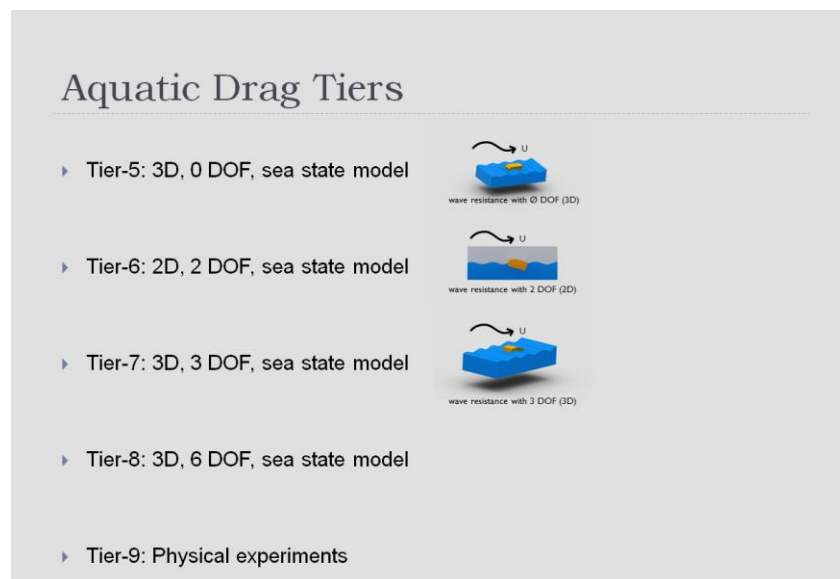

Figure 8

Figure 9

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

Correlation Settings

- **Velocity:** Velocity of the vehicle in (m/s).

Calm Water Solver Settings

- **Velocity:** Velocity of the vehicle in (m/s).
- **FluidMaterial:** Material of the fluid that the vehicle will be exposed to during testing. Available options are Fresh water and Salt water.
- **Fluid Temp (Celsius):** Temperature of the fluid that the vehicle will be exposed to (Celsius).
- **Tier:** Changes the analysis type. Options are 2D (2-dimensional) and 3D (3-dimensional). Usually, 3D simulation setting will give more accurate results with longer computational time. On the other hand, 2D simulation setting completes simulation faster but the accuracy is lower.

Wave Resistance Solver Settings

- **Velocity:** Velocity of the vehicle in (m/s).
- **FluidMaterial:** Material of the fluid that the vehicle will be exposed to during testing. Available options are Fresh water and Salt water.
- **Fluid Temp (Celsius):** Temperature of the fluid that the vehicle will be exposed to (Celsius).
- **Tier:** Changes the analysis type. Options are 2D with 0DOF (2-dimensional with 0 degrees of freedom), 3D with 0DOF (3-dimensional with 0 degrees of freedom), 2D with 2DOF (2-dimensional with 2 degrees of freedom - heave and yaw) and 3D with 3DOF (3-dimensional with 3 degrees of freedom - heave, yaw and roll). For the definitions of previously mentioned degrees of freedom, please see (Fig. 10). Usually, 3D tiers give more accurate results with greater computational time. 2D tiers complete simulation faster than 3D, but the accuracy is lower. 0 DOF tiers give faster and accurate results for smaller wave heights. On the other hand, 2 and 3 DOF tiers give more realistic results for larger wave heights with longer computational time. *Picking 2 and 3 DOF tiers will cause the simulation to crash since these features haven't been implemented yet.*
- **Cyclic Wave Frequency:** Cyclic wave frequency is the number of waves that passes a given point per second (Hz). Frequency and Wave Number are related with each other with the Wave speed (Frequency / Wave number = Wave Speed). *Using unrealistic values for that parameter may cause simulations to crash (Recommended value = 1).*
- **Wave Direction:** Direction of the waves (Degrees). *Has no effect since this feature hasn't been implemented yet.*

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

VANDERBILT
UNIVERSITY

- **Wave Number:** Wave number is the number of waves that exist over a specified distance (1/m). Wave Number and Frequency are related with each other with the Wave speed (Frequency / Wave number = Wave Speed). *Using unrealistic values for this parameter may cause simulations to crash (Recommended value = 0.25).*
- **Wave Height:** Height of the generated waves for the simulation (m). *Using unrealistic values for this parameter may cause simulations to crash (Recommended value < 1).*
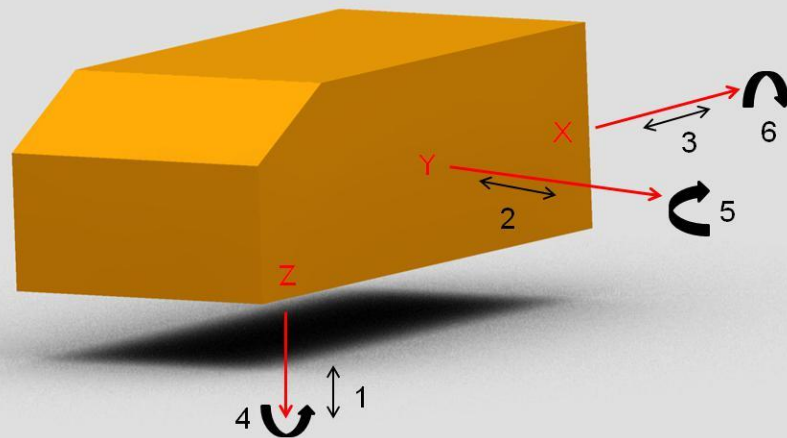
Figure 10

Figure 11

Figure 12

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

VANDERBILT
UNIVERSITY
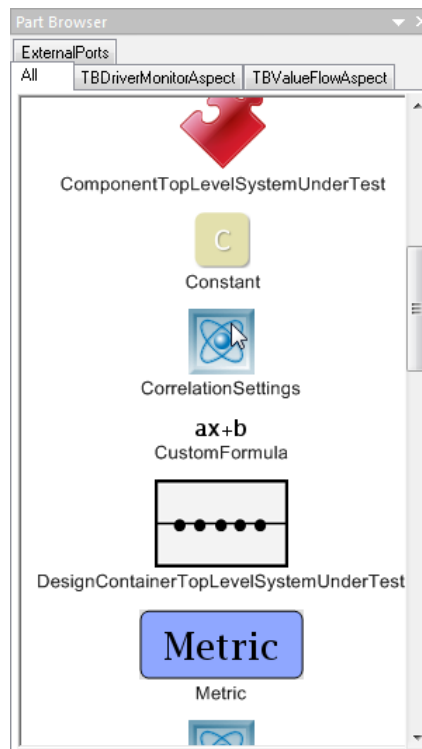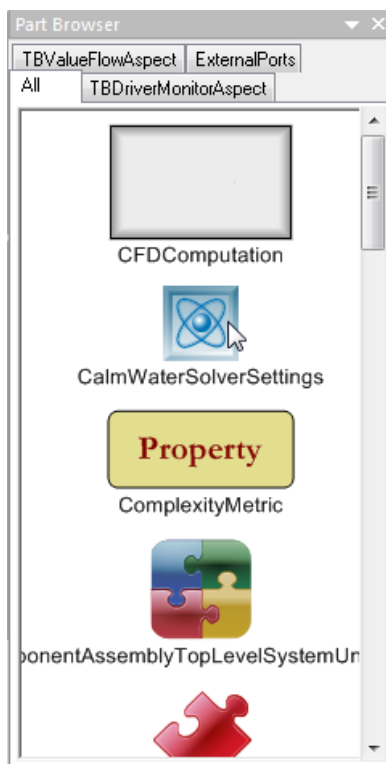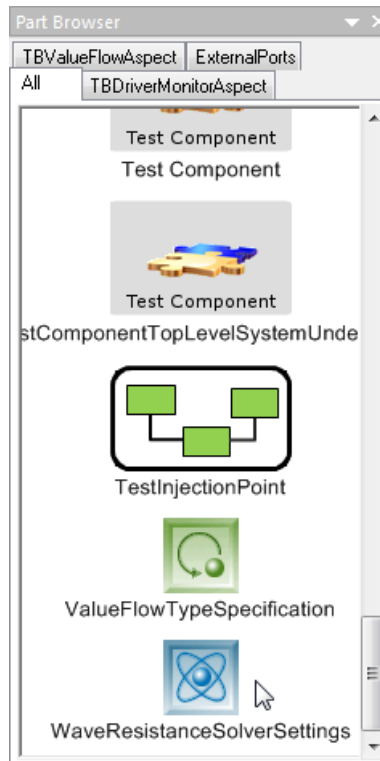
Figure 13

Step 8

Drag the CalmWaterSolverSettings part into the test bench. (Fig.14).

Figure 14

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.
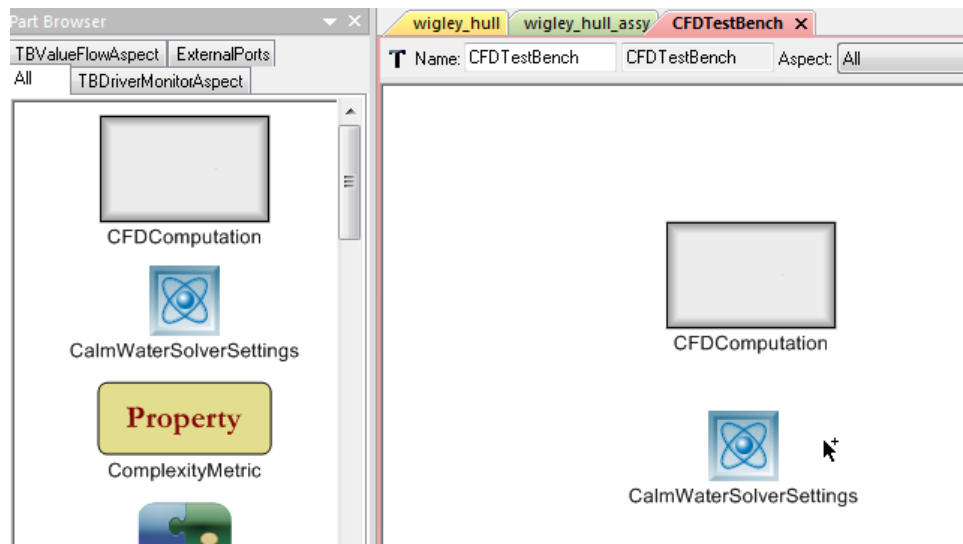
Step 9

Double click on the CFDComputation part that has been placed in the test bench window. The screen should now look similar. (Fig. 15)
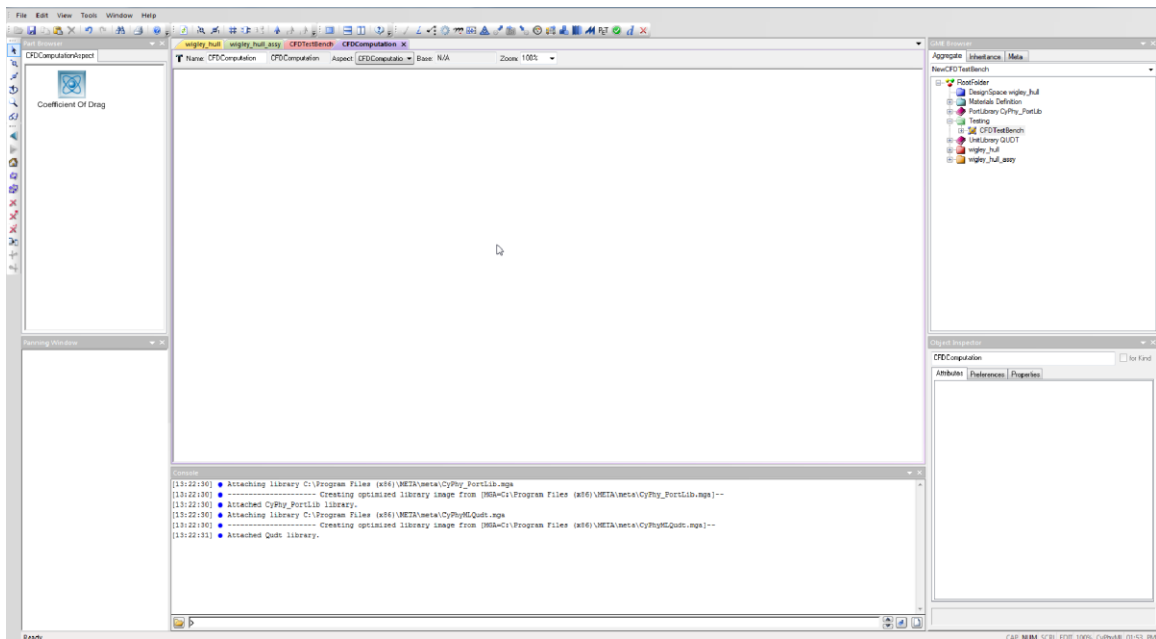

Figure 15

Step 10

Drag the Coefficient of Drag from the Part Browser to the CFDComputation window. This lets GME know that we want the Coefficient of Drag from the results. (Fig. 16)
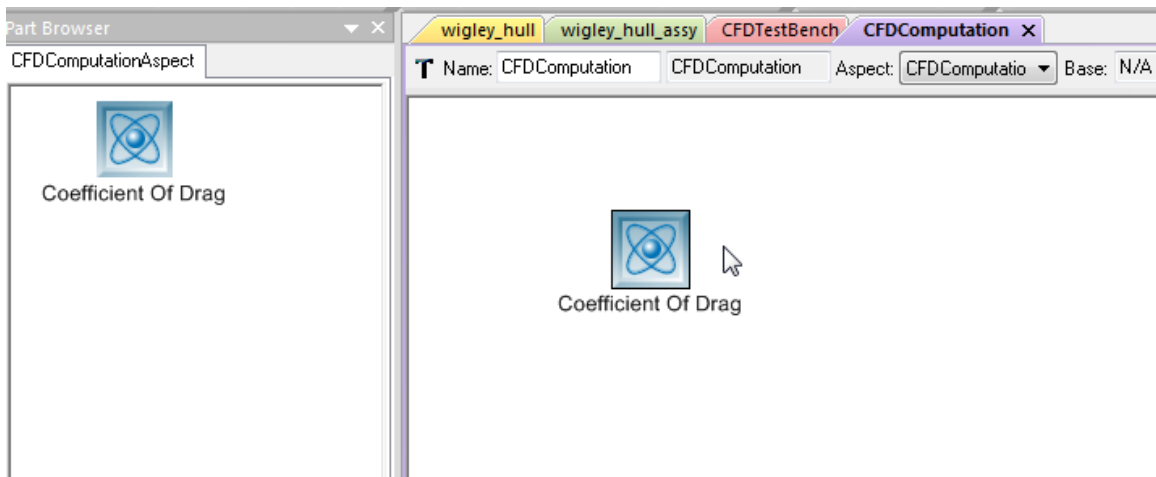

Figure 16

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

VANDERBILT
UNIVERSITY

Step 11

Click on the CFDTestBench on the top. The test bench should now look similar to Fig. 17.
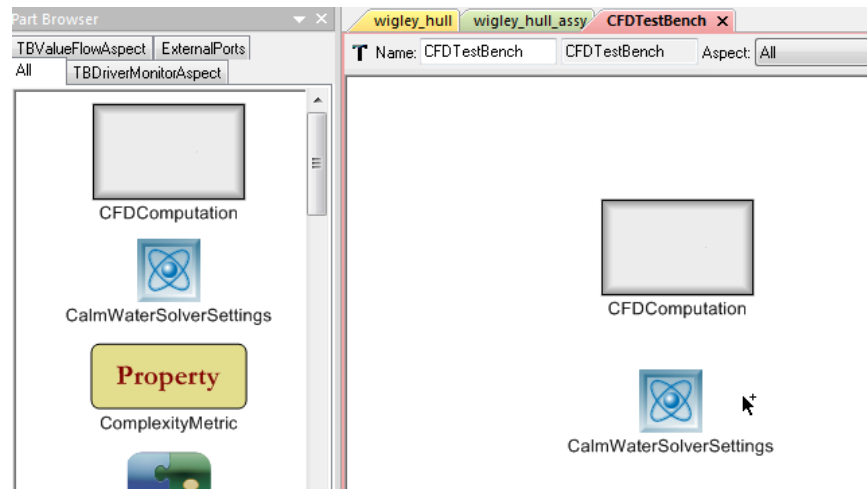
Figure 17

Step 12

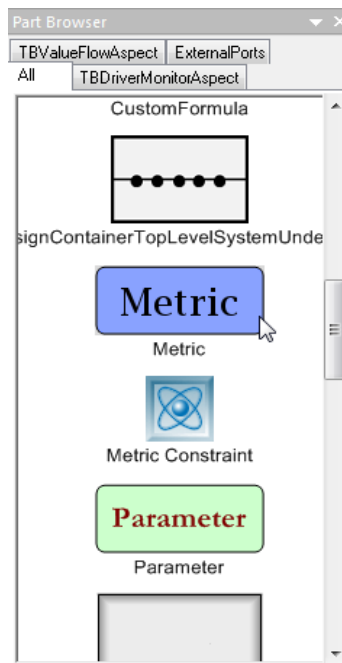In the Part Browser, find the Metric part. (Fig. 18)

Figure 18

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

VANDERBILT
UNIVERSITY

Step 13

Drag the metric object into the test bench and connect the metric to the Coefficient of Drag object in the CFDComputation block. (Fig. 19)
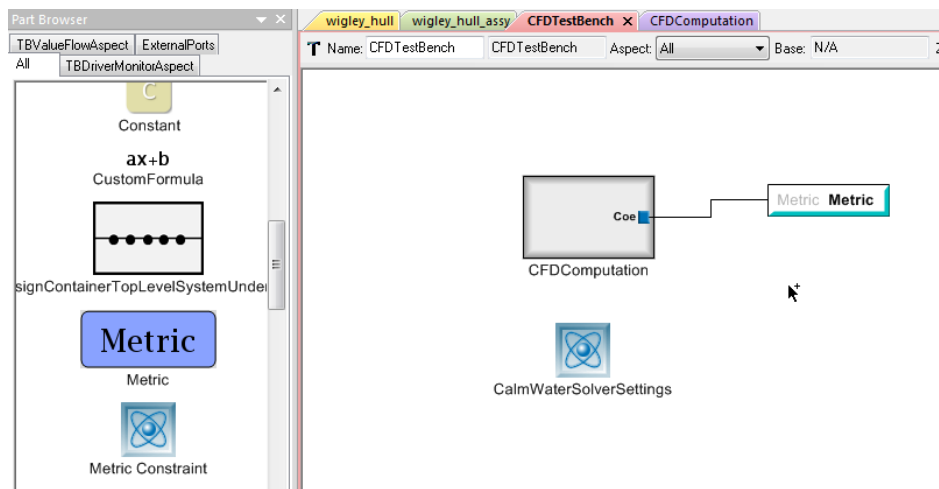

Figure 19

Step 14

Now, we will add the desired assembly to the test bench. This assembly is sent to the CFD solver. In the GME Browser window, copy the desired assembly. (Fig. 20)
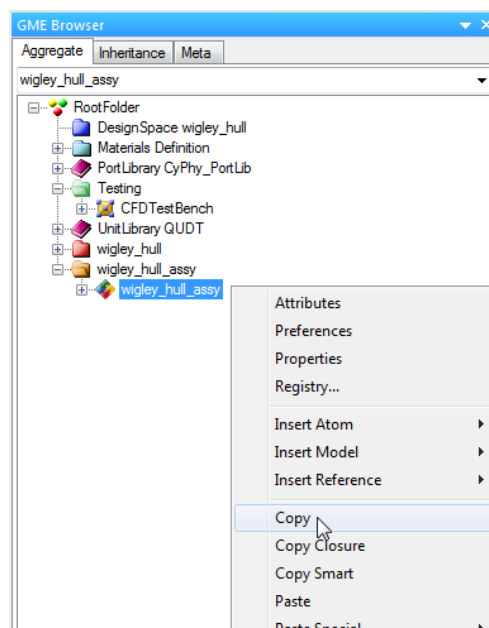

Figure 20

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

Step 15

Paste the assembly as reference in the test bench. Pasting as reference is crucial so that another copy is not made. Select TopLevelSystemUnderTest when prompted for the Reference Role Type. (Fig. 21)
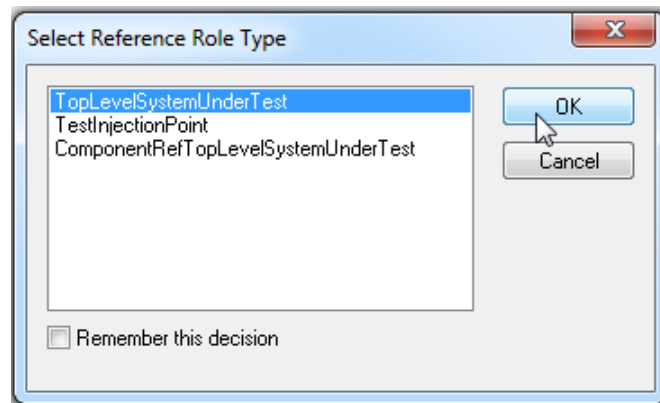

Figure 21

Step 16

On the GME Browser, right click on Testing, go to Insert Folder, and create a Workflow Definition. (Fig. 22)
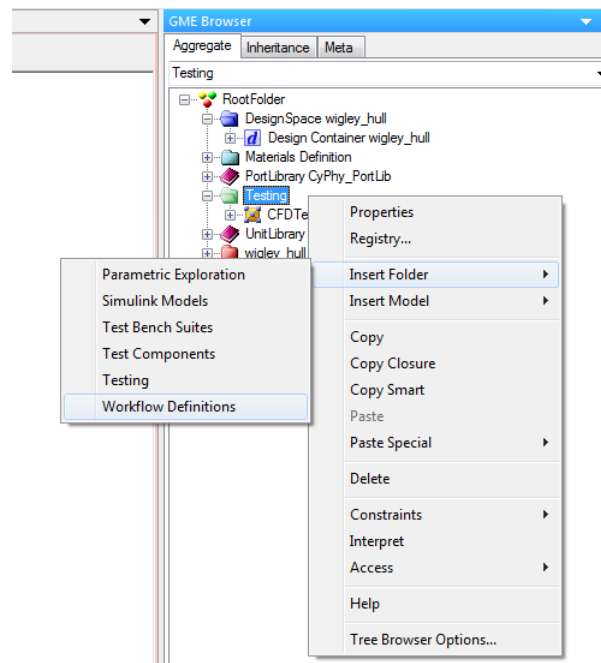

Figure 22

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

VANDERBILT
UNIVERSITY

Step 17

Click on the newly created Workflow Definition folder and insert a new Work flow.
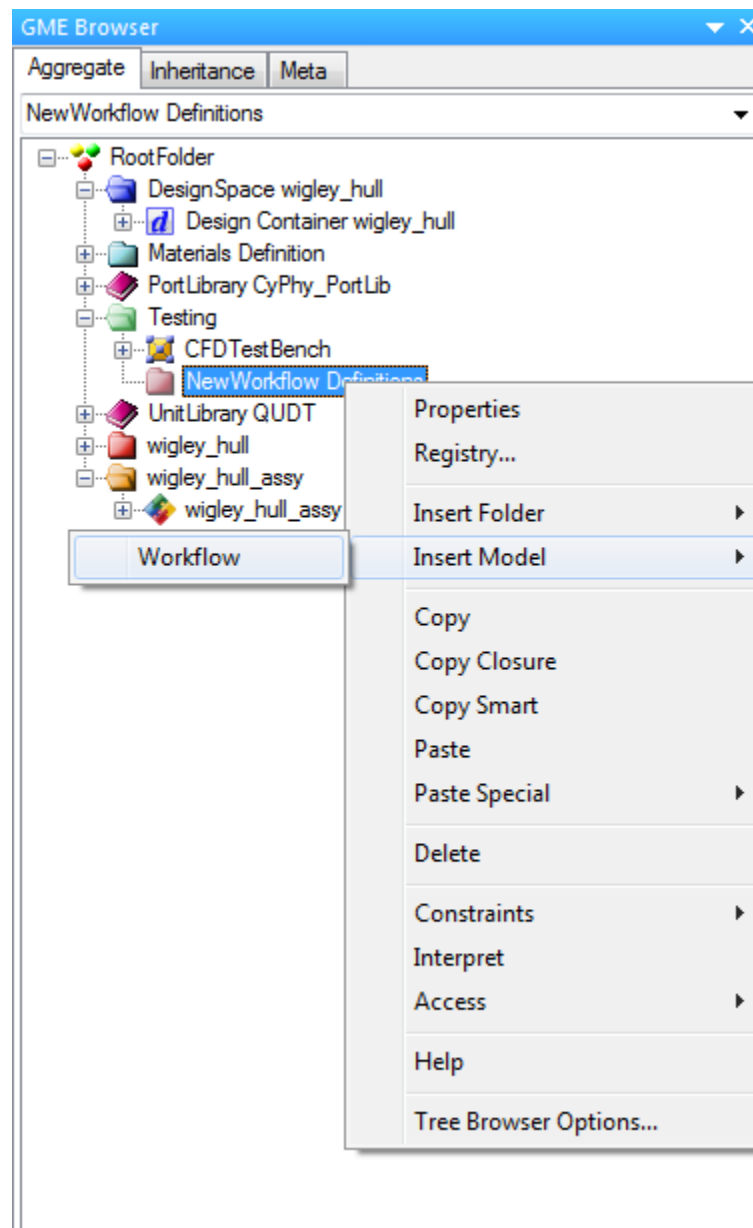Name it CAD. (Fig. 23)

Figure 23

Step 18

Double click on the CAD definition. GME should now look like this. (Fig. 24)

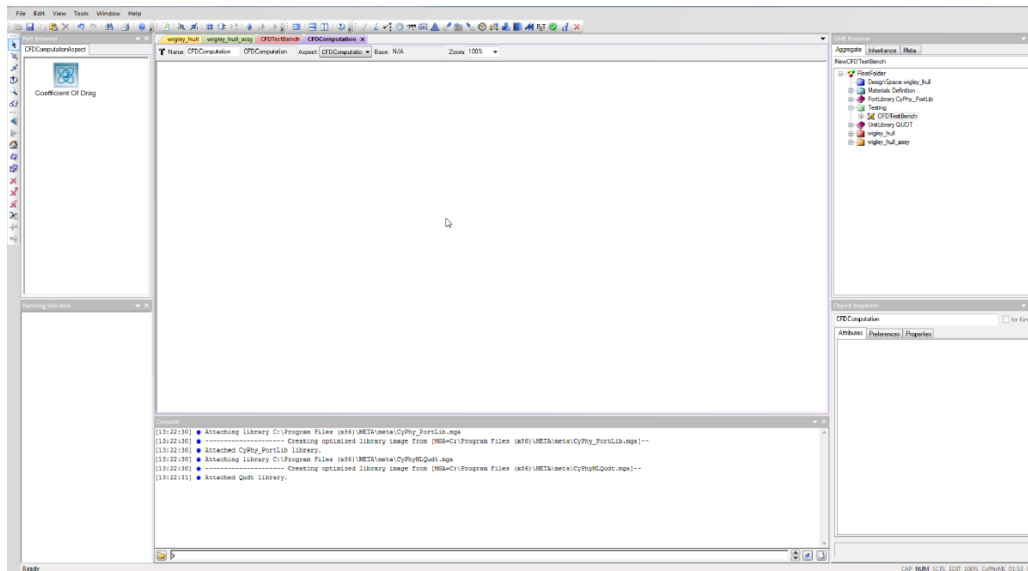

Figure 24

Step 19

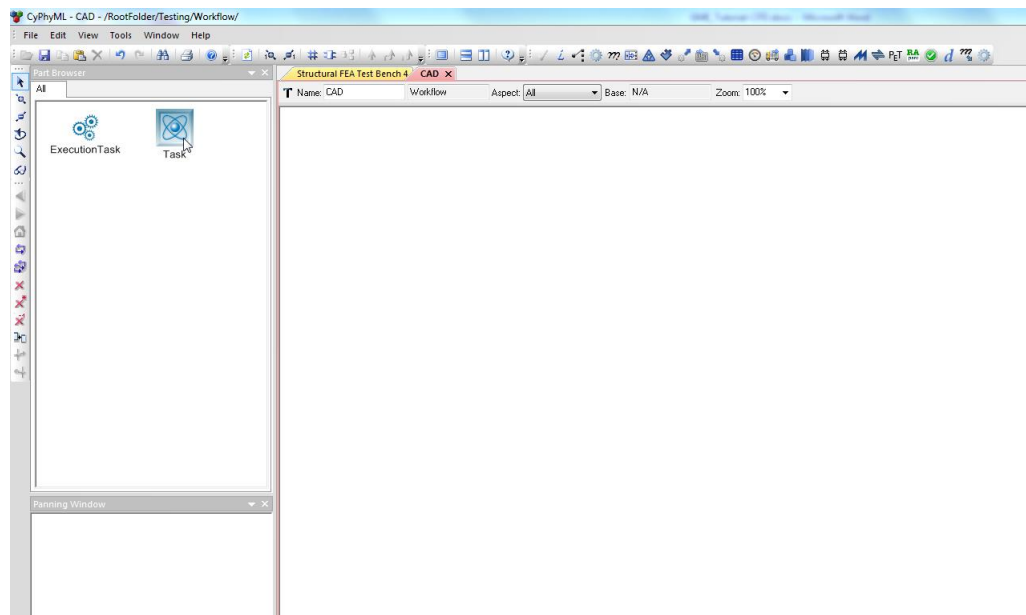Drag the Task object from Part Browser into the CAD definition. (Fig. 25)

Figure 25

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

VANDERBILT
UNIVERSITY

Step 20

The Interpreter Selection window will pop-up. Select ChyPhy2CAD_CSharp from the list. (Fig. 26)
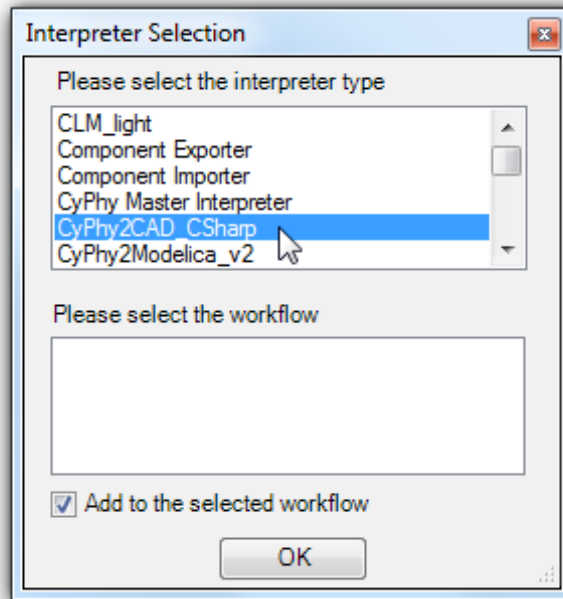
Figure 26

Step 21

GME should now look like this. (Fig. 27)

Figure 27

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.
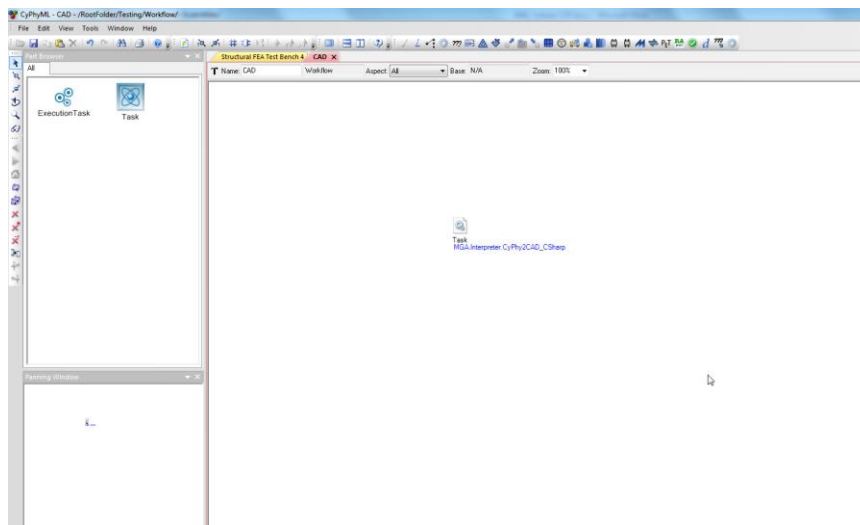
Step 22

Right click on the CAD work flow definition and copy it. Paste it as reference in the Test Bench. (Fig. 28)
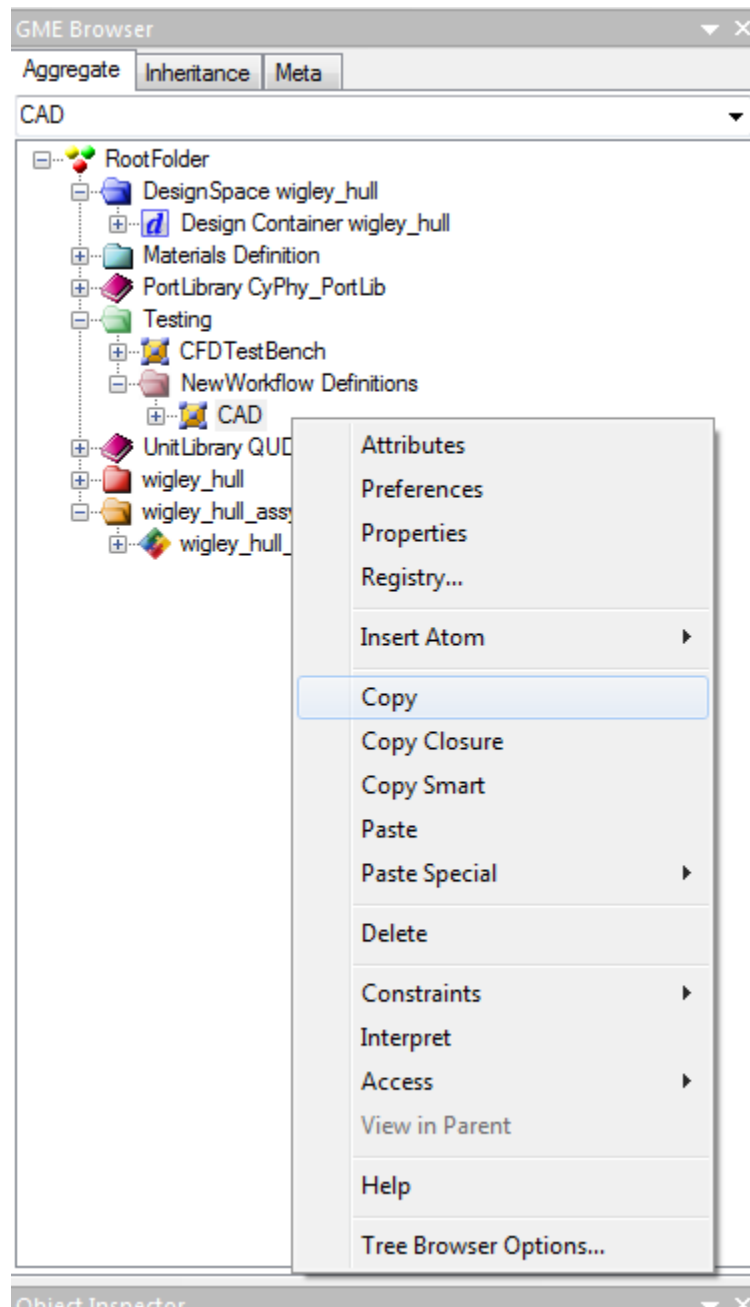
Figure 28

Step 23

The Test Bench should now look like this. (Fig. 29)

Figure 29

Step 24

Set the CFD simulation options in the test bench attributes (Fig. 30). Mesh fineness controls the level of detail in the fluid domain mesh. A coarse mesh will solve in significantly less time than a fine mesh however the solution accuracy will be better with a fine mesh.

Figure 30

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.
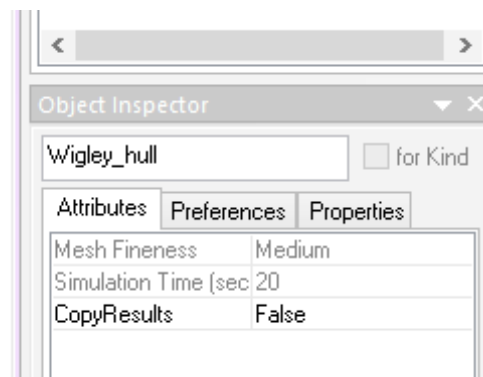
Step 25

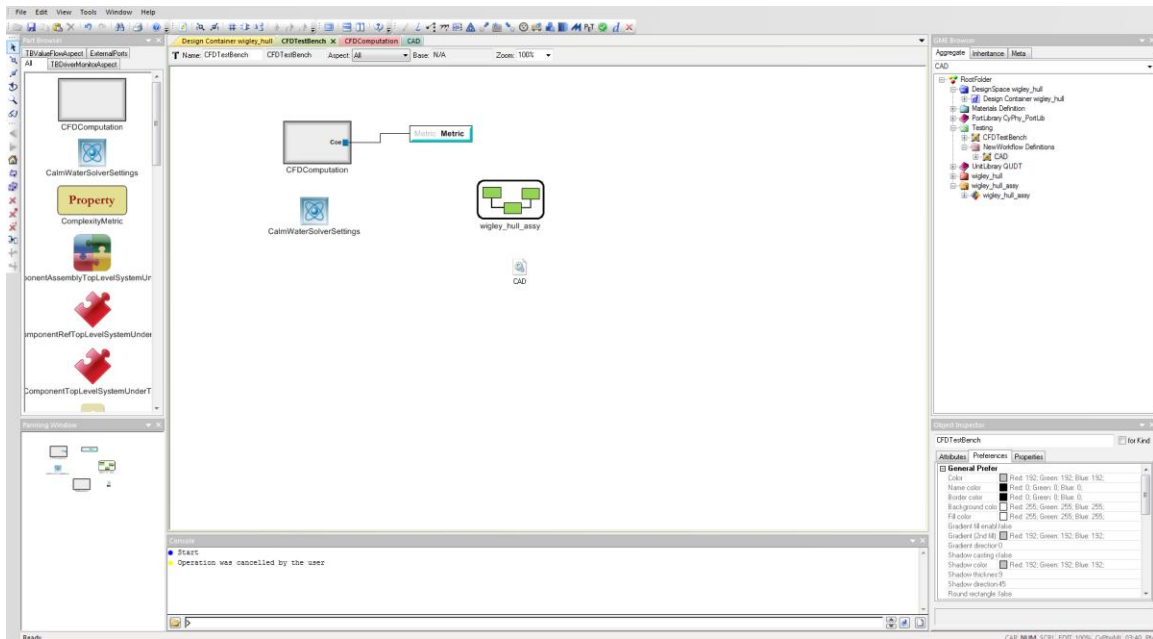Once the test bench creation is complete, now we can run the test bench. On the top, click on the Master Interpreter. The Master Interpreter will determine how the test bench is going to be run. (Fig. 31)
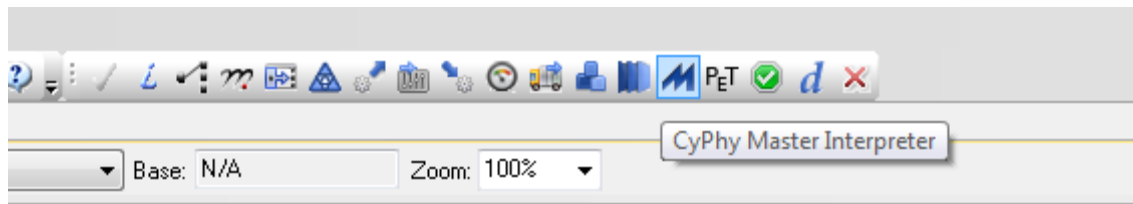
Figure 31

Step 26

When the Master Interpreter dialogue comes up, click on the configuration. Also, make sure "Post to META Job Manager" is checked. Then click OK. (Fig. 32)

Figure 32

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.
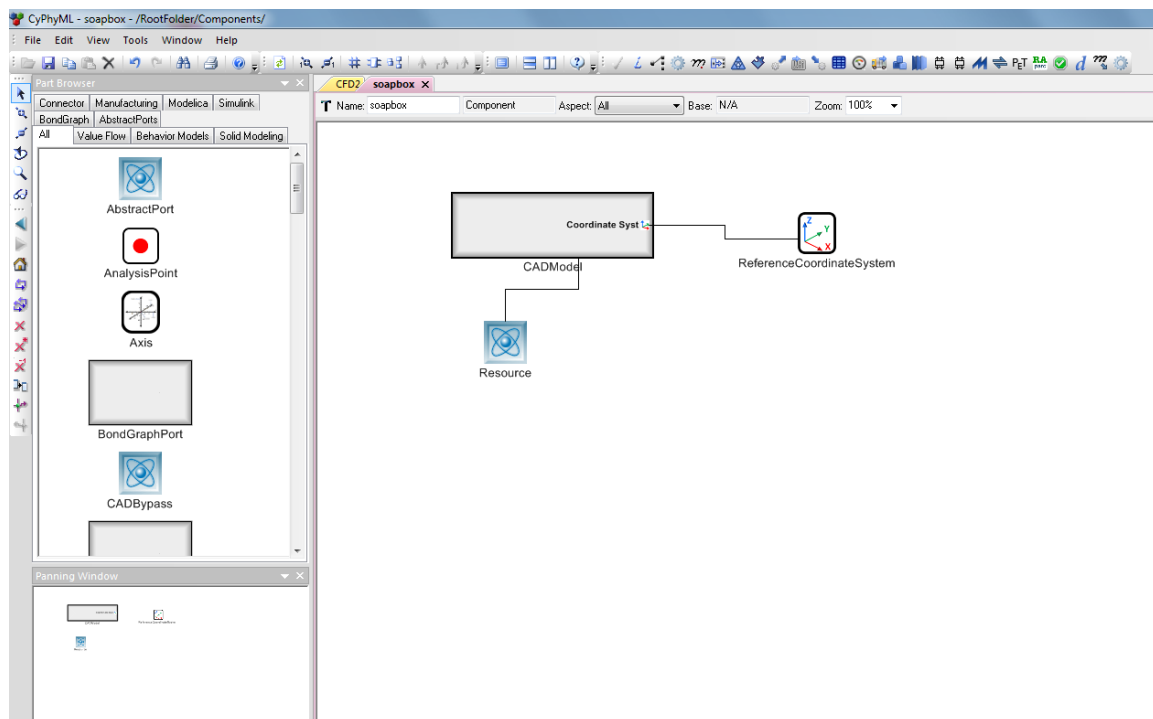
Step 27

The CAD Options window will come up. Click OK. (Fig. 33)

Figure 33

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.
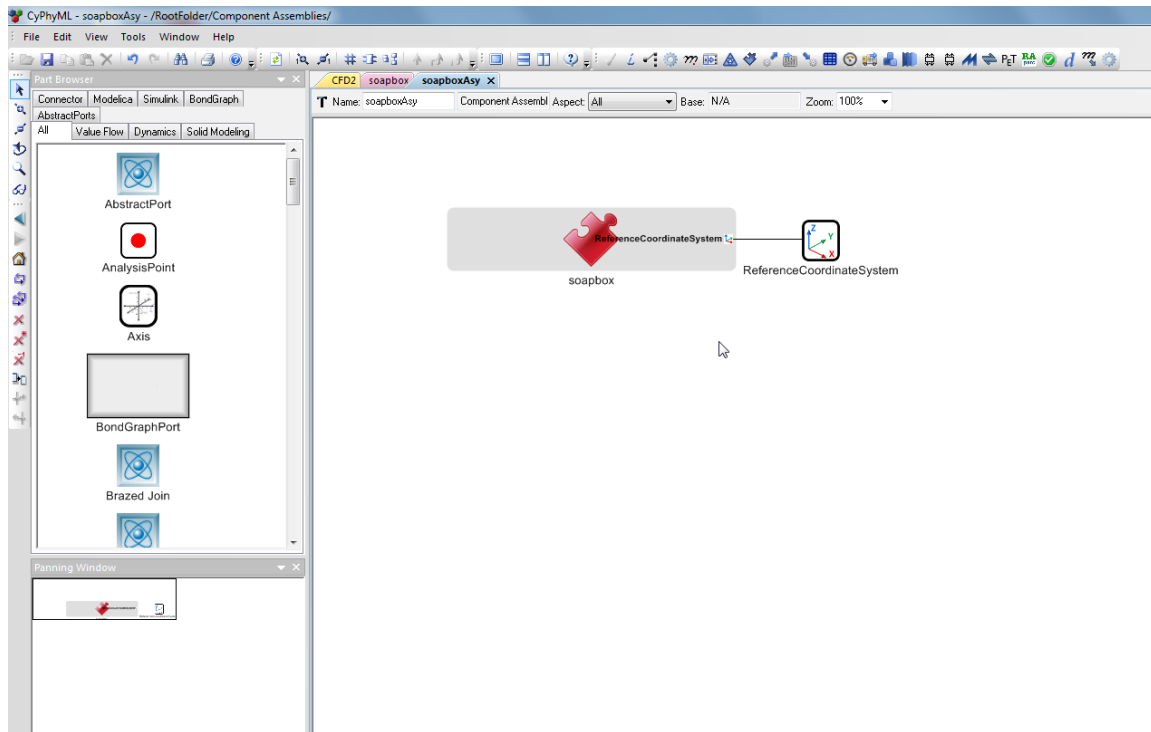
Step 28

The JobManager Configuration window will now open. (Fig. 34)
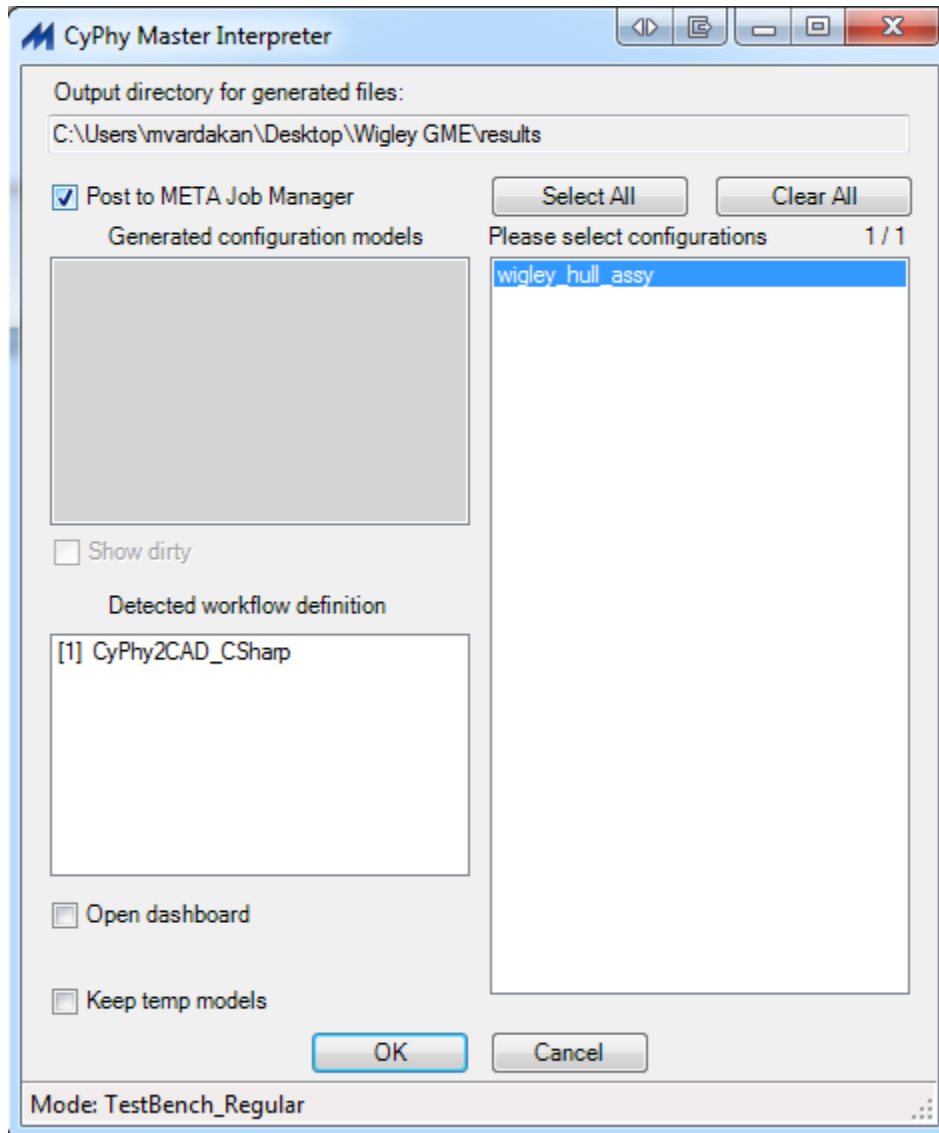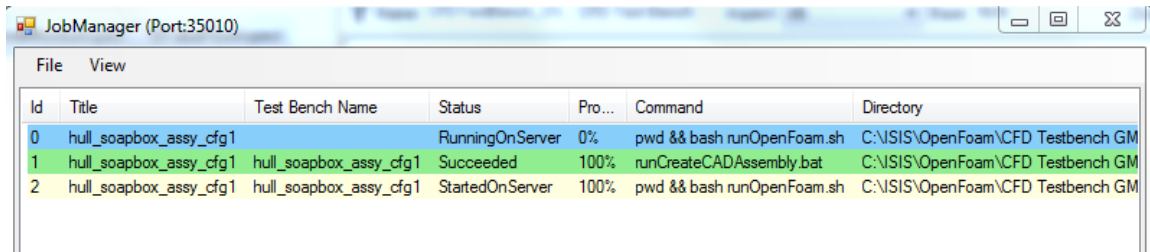
Figure 34

Step 29

Input your VehicleForge credentials and click Save.

Step 30

Job Manager will start the CFD computations. (Fig. 35)

Figure 35

Step 31

After the job is completed, to see the results, open the main folder for the GME. (Fig. 36)

Figure 36

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

Step 32

Click on the correct folder. (Fig. 37)

Figure 37

Step 37

The forceOutput.dat file, which gives the "Average Drag Force", can be found in the path shown in Figure 38.

Figure 38

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

Step 38

The summary.testresults.json file is located in the location shown in Figure 39.
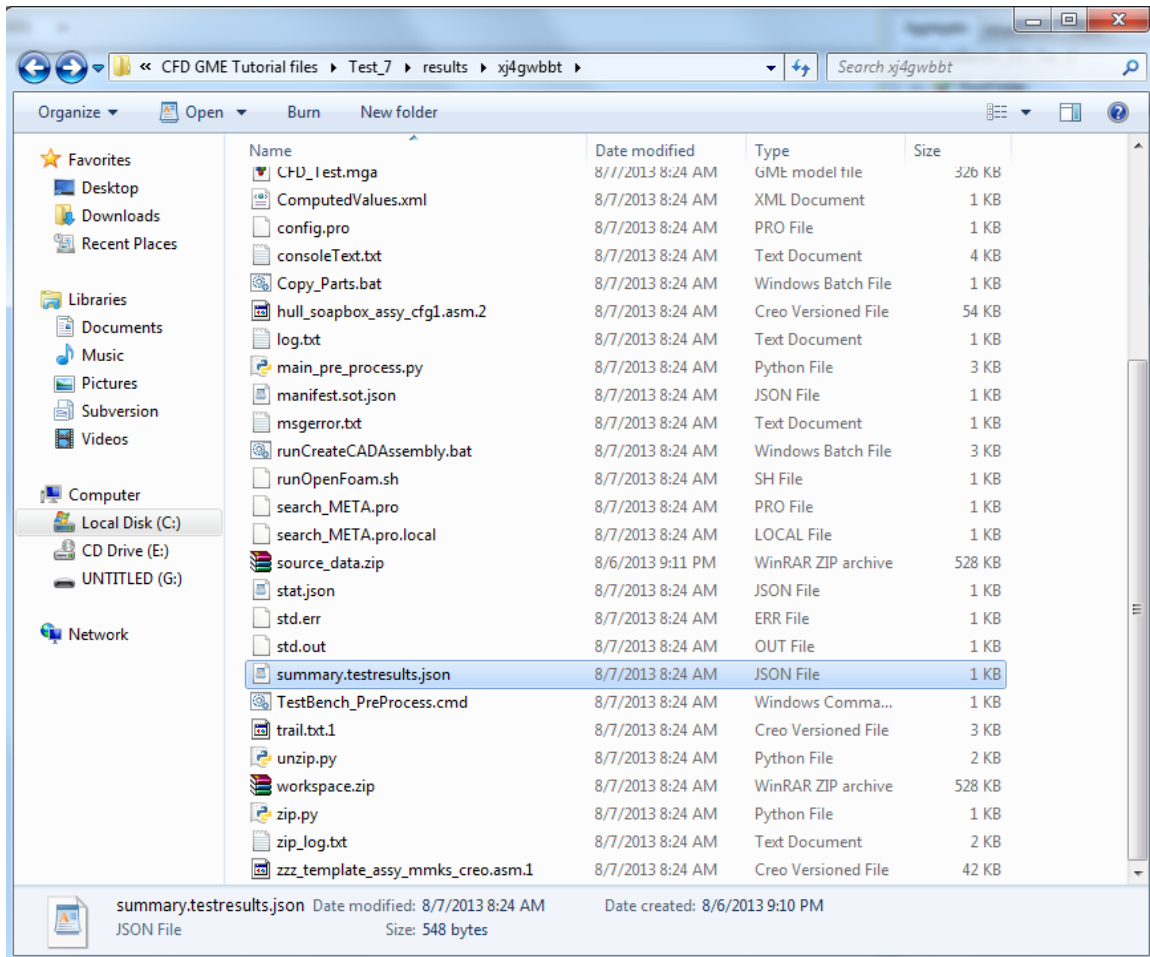
Figure 39

Step 39

Opening the summary.testresults.json file will allow you to view the coefficient of drag. (Fig. 40)

```
{
  "Parameters": [],
  "Artifacts": [],
  "AnalysisStatus": "OK",
  "TestBench": "CFDTestBench_DS_Tier_1",
  "DesignName": "hull_soapbox_assy_cfg1",
  "Metrics": [
    {
      "Name": "Metric",
      "DisplayedName": null,
      "GMEID": "id-0067-000002fa",
      "Value": "0.00863132041617",
      "ID": "fa791643-90e3-4c1d-b11d-f6354cb8e24f",
      "Unit": ""
    }
  ],
  "DesignID": "{428cb75a-99db-450d-bcb5-9a4fa2ff63d4}",
  "Design": "hull_soapbox_assy_cfg1.metadesign.json",
  "Details": "",
  "Time": "2013-08-06 21-11-55"
}
```
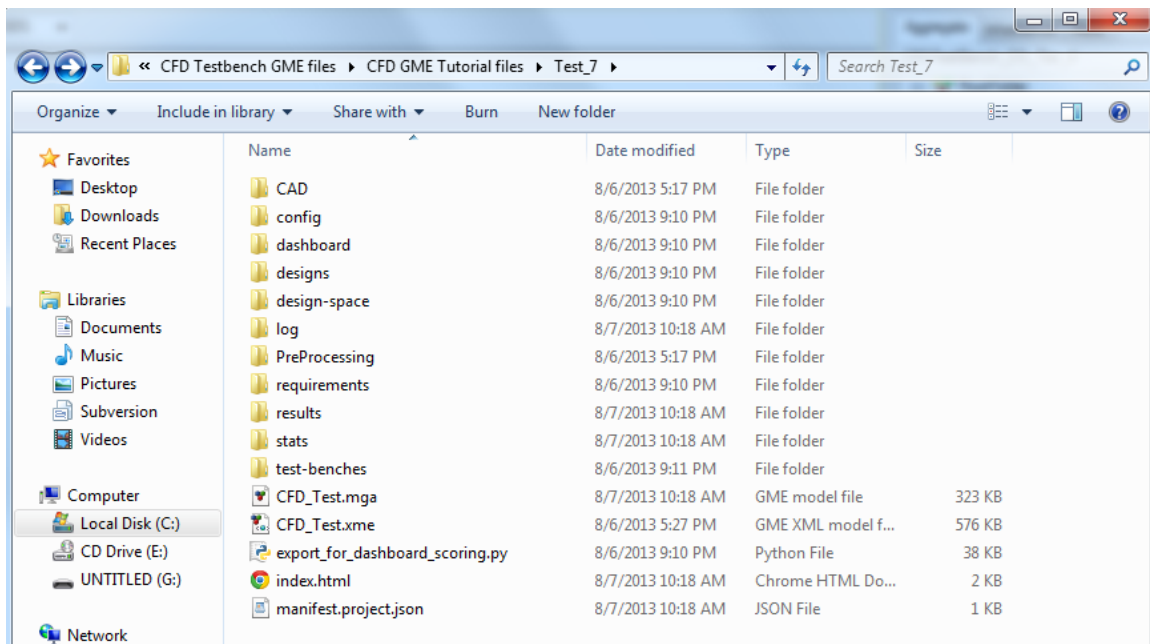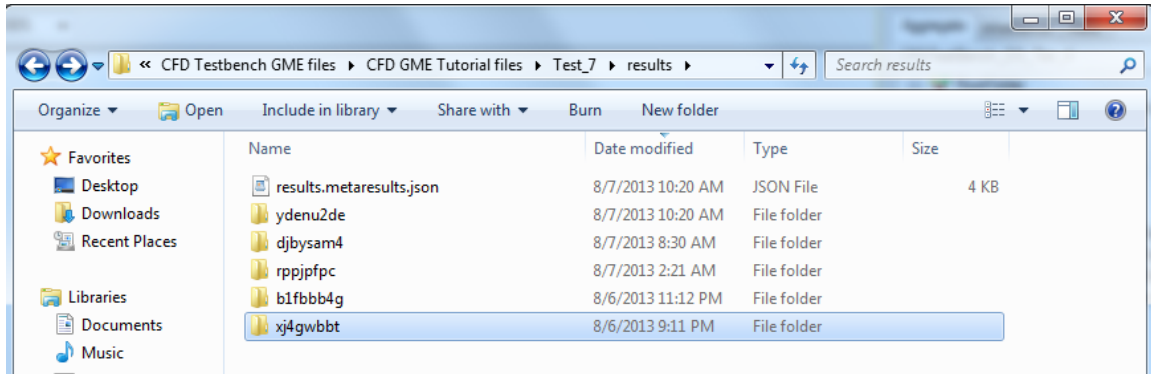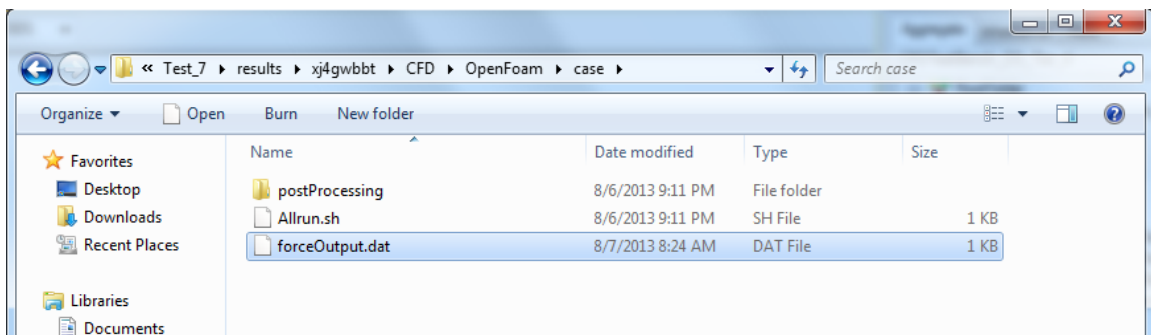
Figure 40

Typical analysis times are between 1 and 2 days depending on the conditions simulated.
The CFD test bench is a server only test bench and cannot be run on the designer's machine because the analysis tool is instantiated on a server.

6.0 Description

CFD simulation is a numerical procedure to determine the aquatic performance of the design. The test bench will compute the drag force and coefficient of drag accurately under the conditions provided by the user. Open source CFD toolbox, OpenFoam is used for CFD simulations.
The System under Test is assembled in CREO and then each component making up the system is saved as an individual step file. Input from the user is also gathered.
This information is packaged and sent (via remote server) to the VehicleFORGE servers for CFD simulations.
The fluid domain size is set according to the bounding box of the assembly. The fluid domain is set to five times the assembly length and 10 times the assembly width.

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

VANDERBILT
UNIVERSITY

The fluid domain height is set at five times the assembly height with the lower three-fifths composed of liquid and the upper two-fifths composed of liquid composed of air.

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 41 contains the ReferenceCoordinateSystem (top left) at the top assembly level.
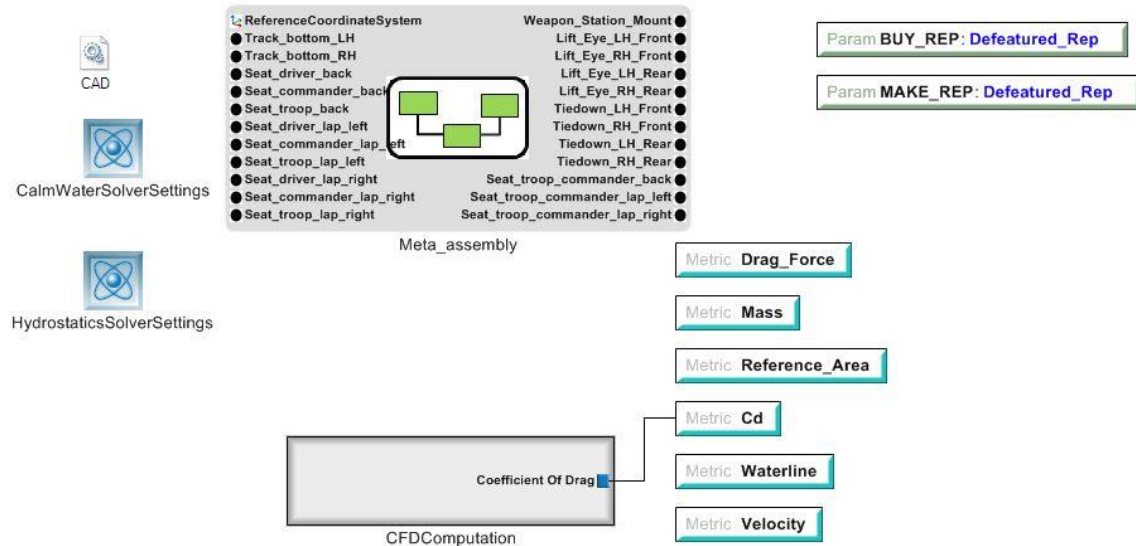
Figure 41

This ReferenceCoordinateSystem is connected to another ReferenceCoordinateSystem inside the sub-assembly that in turn is connected to the ReferenceCoordinateSystem in the hull component (Figure 42). 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 an .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.

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

Figure 42

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.

Results are returned in two files "summary.testresults.json" and "forceOutput.dat" containing coefficient of drag and drag force, respectively. The results can be visualized by opening up the files mentioned before with a simple text editor.

HydrostaticsSolverSettings

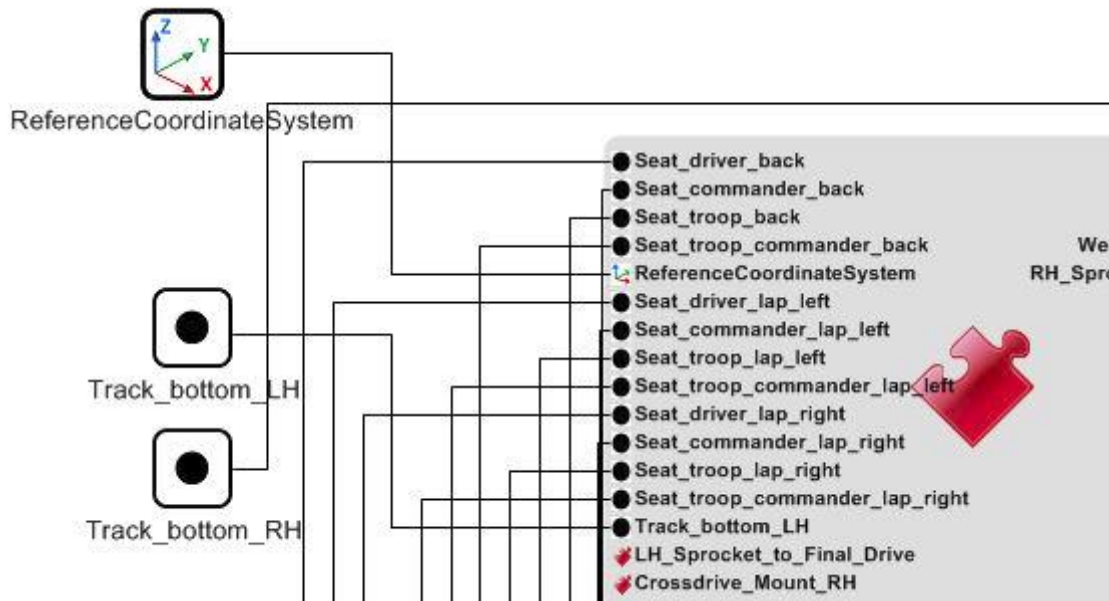The hydrostatics solver is used to compute the waterline of the assembly. It uses the assembly total mass, computes displaced volume according to the formula $v = m/rho$ where v is displaced fluid volume, m is the total assembly mass, and rho is the density of the fluid. The fluid type and temperature are specified in the ObjectInspector. The hydrostatics solver computes the waterline and reference area (wetted surface area) based on the displaced volume. Input is FluidMaterial, Fluid Temp (Celsius), Roll Angle Start (deg), Roll Angle End (deg), Roll Angle Step Size (deg), Pitch Angle Start (deg), Pitch Angle End (deg), and Pitch Angle Step Size (deg). Output is center of buoyancy (cb_x, cb_y. cb_z) (mm), center of gravity (cg_x, cg_y,

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

VANDERBILT
UNIVERSITY

cg_z) (mm), righting_moment_arm (mm), displaced_volume (mm3), roll (deg), pitch (deg), yaw (deg), mass (kg), waterline (mm), wetted surface area (mm2), and hydrostatic volume (mm3). The waterline is the vertical distance from the assembly reference coordinate system to the fluid surface.

If HydrostaticsSolverSettings is omitted from the test bench, the waterline is assumed to be 1 m and the reference area is assumed to be 47 m2. 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 m2.

Hydrodynamic Solver Settings

One of three hydrodynamics solvers may be specified for a CFD test bench. CorrelationSettings represents CFD Tier 1 that leverages empirical correlations. The input to the solver, specified in the ObjectInspector window, is velocity in m/s. 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. Simulation time with this solver is set at 20 seconds. If the test bench name is "Wigley_hull_quick", the simulation time is set at 0.1 seconds. This mode produces invalid numerical results and is intended only to verify tool operation.

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 0O representing head waves and 90O representing wave direction abeam the vehicle's starboard side. Simulation time with this solver is set at 7 seconds. If the test bench name is "Wigley_hull_quick", the simulation time is set at 0.1 seconds. This mode produces invalid numerical results and is intended only to verify tool operation.

7.0 Test Bench Tiers

Attribute	Tier 1	Tier 3	Tier 5
Test Bench Name	CFD with correlation solver. Results feed into other test benches such as Sustained Forward Cruise Speed.	CFD with calm water settings. Results feed into other test benches such as Sustained Forward Cruise Speed.	CFD with wave resistance. Results feed into other test benches such as Sustained Forward Cruise Speed.

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

VANDERBILT
UNIVERSITY

Attribute	Tier 1	Tier 3	Tier 5
Description	Uses a correlation formula to estimate drag based on empirical data.	Uses CFD to estimate drag in calm water	Uses CFD to estimate average drag in wave conditions
Estimate Run Time	3 minutes	8-16 hours	12-24 hours
Error Margin	Designs similar to those vehicles used in creating the correlation should produce a reasonable approximation.	Small error margin for calm water conditions with small to significant error margin for wave conditions.	Small error margin for wave conditions and for calm water conditions. Tier 3 produces better results for calm water conditions.
Results Provided	Drag coefficient	Drag coefficient	Drag coefficient
Local/Remote	Remote	Remote	Remote
Tool Used	OpenFOAM	OpenFOAM	OpenFOAM
How to Interpret Results	n/a	Movie, use ParaView to examine mesh and results	Movie, use ParaView to examine mesh and results
Model Requirements	Vehicle mass, velocity	STL geometry, vehicle mass, velocity, fluid, fluid temperature	STL geometry, vehicle mass, velocity, fluid, fluid temperature, wave frequency, wave direction, wave number, wave height.

8.0 Metrics

Standard CFD metrics are itemized in Table 1. These may be specified in a CFD test bench for inclusion in the testbench_manifest.json.

Description	Metric Text
Drag Coefficient	Cd
Drag Force	Drag_Force
Mass (kg)	Mass
Reference Area (mm2) wetted surface area	Reference_Area
Velocity (m/s)	Velocity
Waterline (mm) distance from assembly bottom to waterline	Waterline

Table 1: Standard CFD Metrics

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

VANDERBILT
UNIVERSITY

| Test Bench #	Metric	Description
2	Sustained_Fwd_Speed_in_up_to_SS3	Top forward speed in sea conditions up to sea state 3. Requires a Suite of Tests (SOT) with drag computation and dynamics cruise speed.
8	Reserve_Buoyancy	Percentage of buoyancy remaining when vehicle is in water.
116	Stability_Rough_Seas	True if vehicle can negotiate rough seas without capsizing.
117	Roll_Correction	True if vehicle can right itself from a 90 degree roll.

Table 2

9.0 Required Connections to System Under Test

None.

10.0 Outputs

The output of this test bench is two results files "summary.testresults.json" and "forceOutput.dat". The summary results presents the coefficient of drag and forceOutput presents the average drag force in Newtons. Both files can be viewed with a simple text editor.

The forceOutput.dat file, which gives the "Average Drag Force", can be found in the path shown in (Fig. 43).
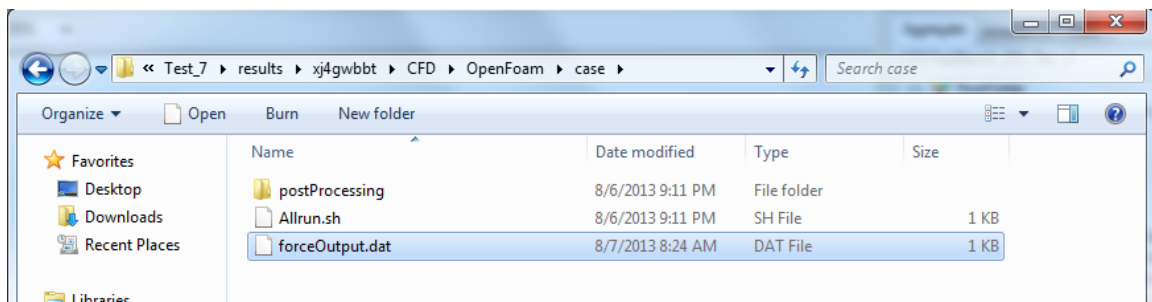
Figure 43

The summary.testresults.json is located in the location shown in (Fig. 44).
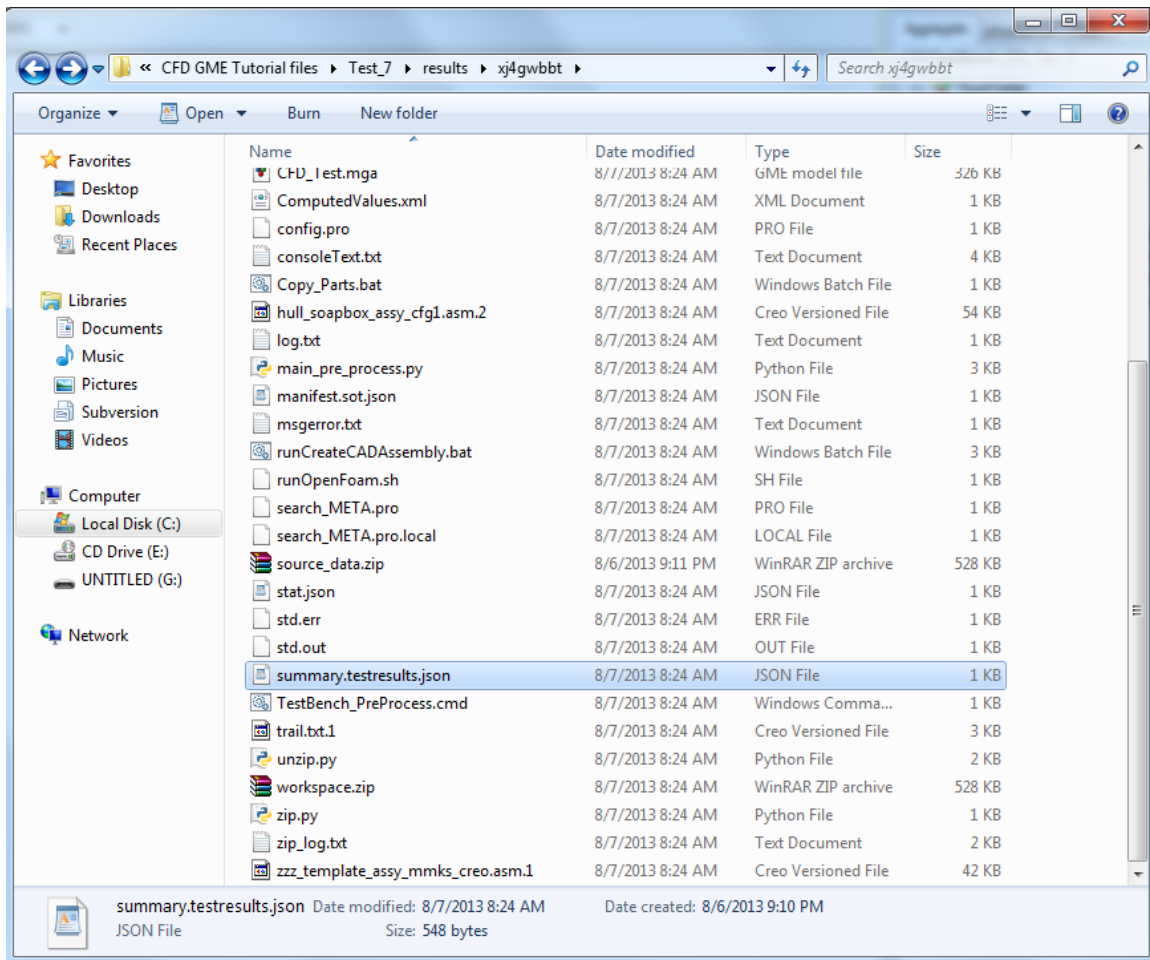

Figure 44

Opening the summary.testresults.json file will allow you to view the coefficient of drag. (Fig. 45)

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

VANDERBILT
UNIVERSITY

```
{
    "Parameters": [],
    "Artifacts": [],
    "AnalysisStatus": "OK",
    "TestBench": "CFDTestBench_DS_Tier_1",
    "DesignName": "hull_soapbox_assy_cfg1",
    "Metrics": [
        {
            "Name": "Metric",
            "DisplayedName": null,
            "GMEID": "id-0067-000002fa",
            "Value": "0.00863132041617",
            "ID": "fa791643-90e3-4c1d-b11d-f6354cb8e24f",
            "Unit": ""
        }
    ],
    "DesignID": "{428cb75a-99db-450d-bcb5-9a4fa2ff63d4}",
    "Design": "hull_soapbox_assy_cfg1.metadesign.json",
    "Details": "",
    "Time": "2013-08-06 21-11-55"
}
```

Figure 45

The CFD simulation results of the Wigley hull example can be compared with the experimental values presented in the plots below (Fig. 46 and 47).
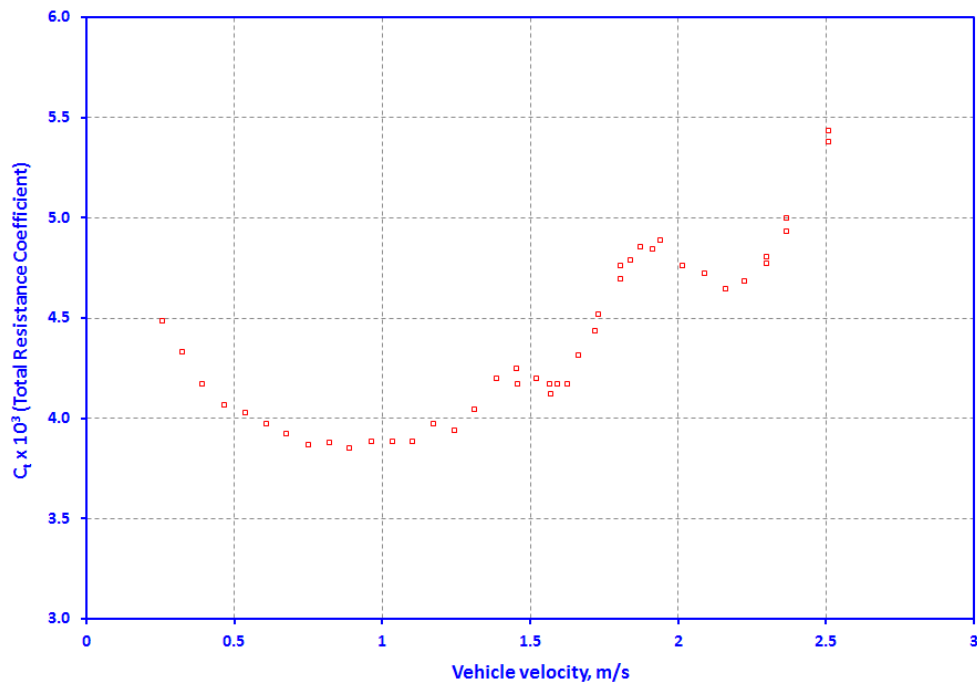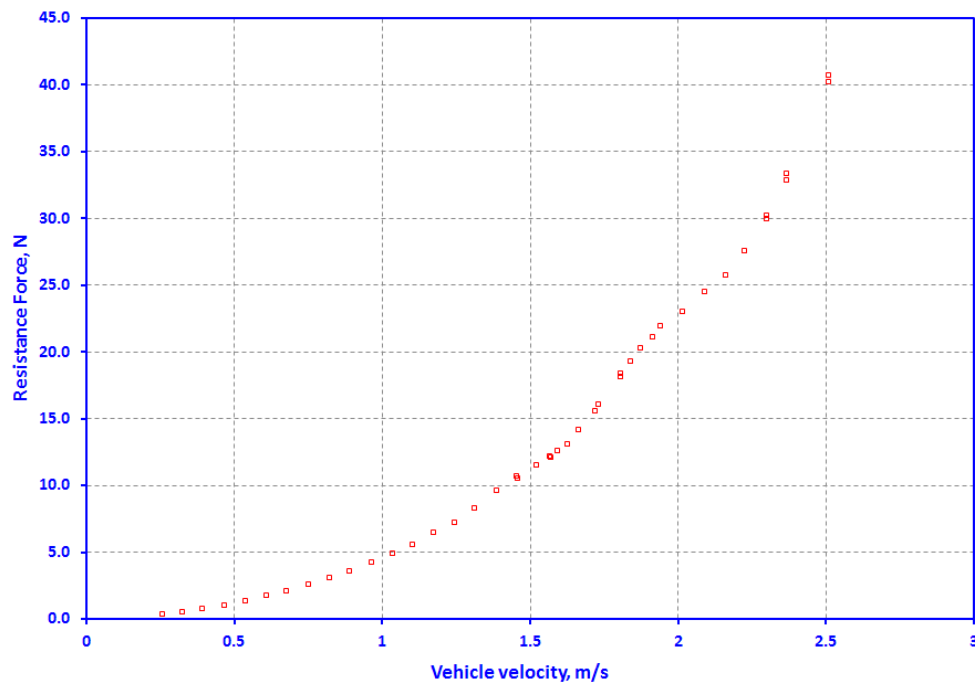

Figure 46

Figure 47

11.0 Known Limitations

- Test bench names cannot include the hyphen (-) character.
- 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 Tiers 1, 3, and 5. The recommended simulation time in Tier 3 (calm water solver) is 20 seconds which is needed to reach steady state. The recommended simulation time in Tier 5 (wave conditions) is 7 seconds.
- 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.

Institute for Software Integrated Systems
World-class, interdisciplinary research with global impact.

VANDERBILT
UNIVERSITY

- 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, RandomParameterDriver, Requirement Link, SimpleFormula, Test Component, TestComponentTopLevelSystemUnderTest, 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. Fluid temperature currently has no effect on the waterline computation.
- CFD test benches cannot be included in a suite of tests at this point. Therefore, test benches, like Sustained_Fwd_Speed_in_up_to_SS3, must be run in a manual fashion where the CFD test bench is run and the drag results manually entered into the dynamics test bench to compute maximum aquatic speed.
- Stability_Rough_Seas is not yet supported. CFD Tier 7 (3 DOF) needs to be implemented to support this test bench.