

Computational Fluid Dynamics (CFD) Test Bench and Tool

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. The conditions that the test bench can simulate are calm water, wave resistance with 0 degrees-of-freedom (DOF), wave resistance with 2/3DOF (not implemented yet) and wave resistance with 6DOF (not implemented yet). The CFD Test Bench can accomplish those simulations for both 2 Dimensional (2D) and 3D domains.

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.

Test Bench: CFD

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 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.

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.

Test Bench Structure

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

Assumptions

Table of Contents

- Computational Fluid Dynamics (CFD) Test Bench and Tool
 - 1.0 Purpose
 - 2.0 Procedures
 - 2.1 Installation
 - 2.2 Tool
- Test Bench: CFD
 - Requirements Tested
 - Operation
 - Test Bench Structure
 - Assumptions
 - Step 1
 - Step 2
 - Step 3
 - Correlation Settings
 - Calm Water Solver Settings
 - Wave Resistance Solver Settings
 - Step 4
 - Step 5
 - Step 6
 - Description
 - Metrics
 - Required Connections to System Under Test
 - Outputs

Right now, CFD test bench assumes the bottom point of the object is 1m (25cm for Wigley hull example) below the water line. So if the dimension of the object in the z-axis direction is less than 1m (25cm for Wigley hull example) than the object is fully submerged. If it is higher than 1m, than the bottom 1m of the object is submerged the rest is on the water. And also CFD test bench assumes that the object moves in the negative x-direction.

Step 1

First, we will import our components. We have two options to do that:

1st Option (if an acm file is present):

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

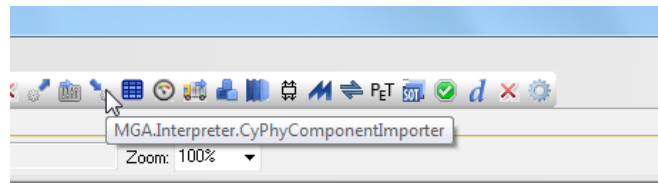


Figure 1

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

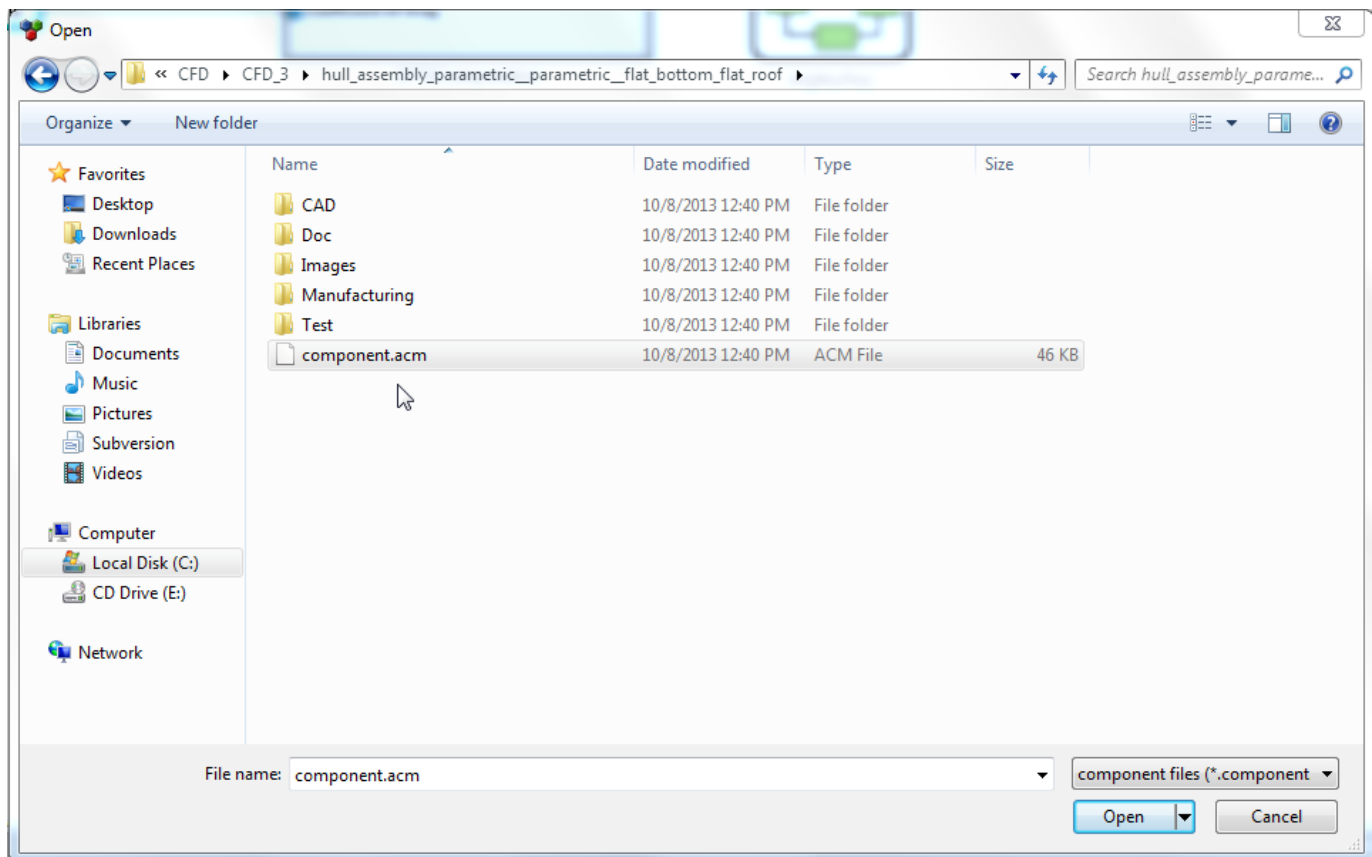


Figure 2

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.

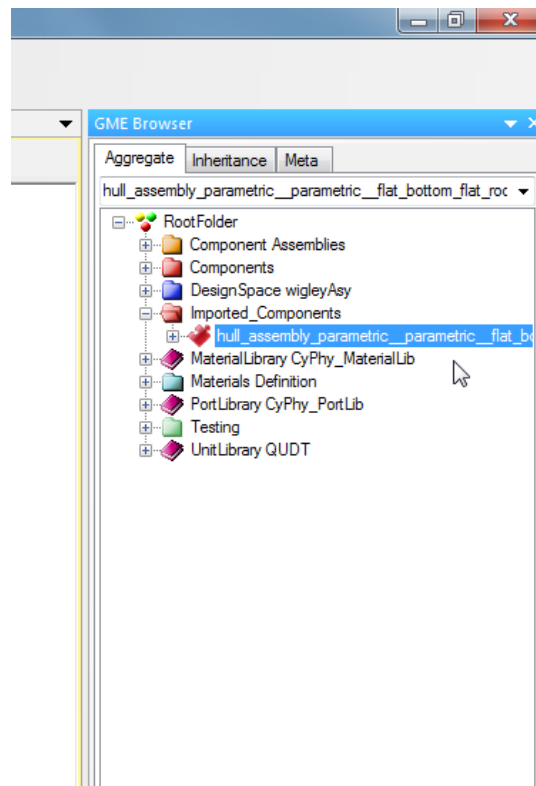


Figure 3

2nd Option (if there is no acm file):

In the GME Browser, expand "MaterialLibrary CyPhy MaterialLib" object and then expand "MaterialDefinitions" folder. (Fig. 4)

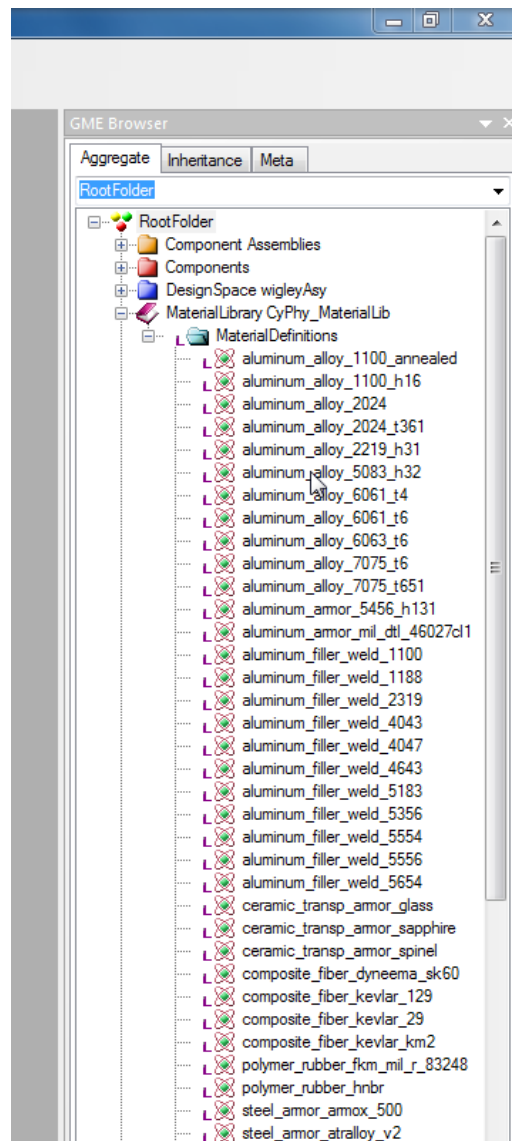


Figure 4

In this tutorial, "steel_armor_cast_mil_a_11356cl1" material is used as an example. Right-click on the "steel_armor_cast_mil_a_11356cl1" material and select Copy. (Fig. 5)

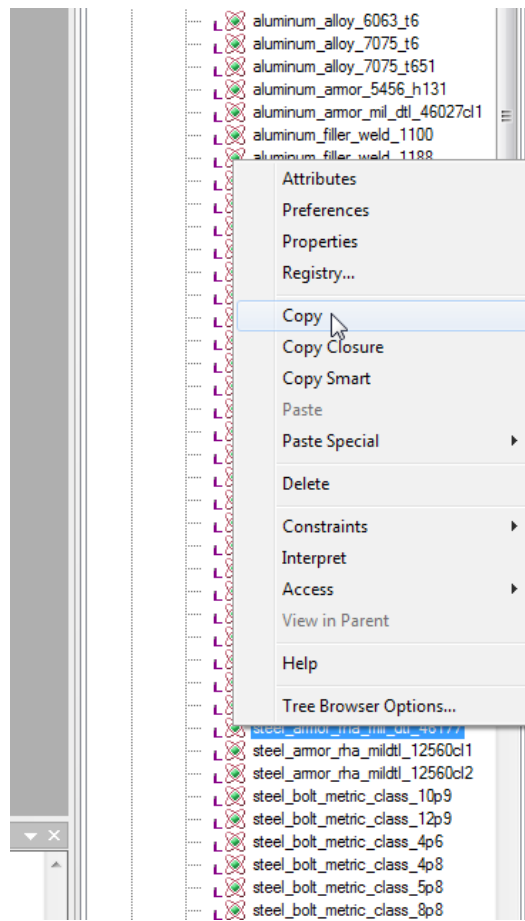


Figure 5

Then Right-click on the materials definition folder and select Paste. (Fig. 6)

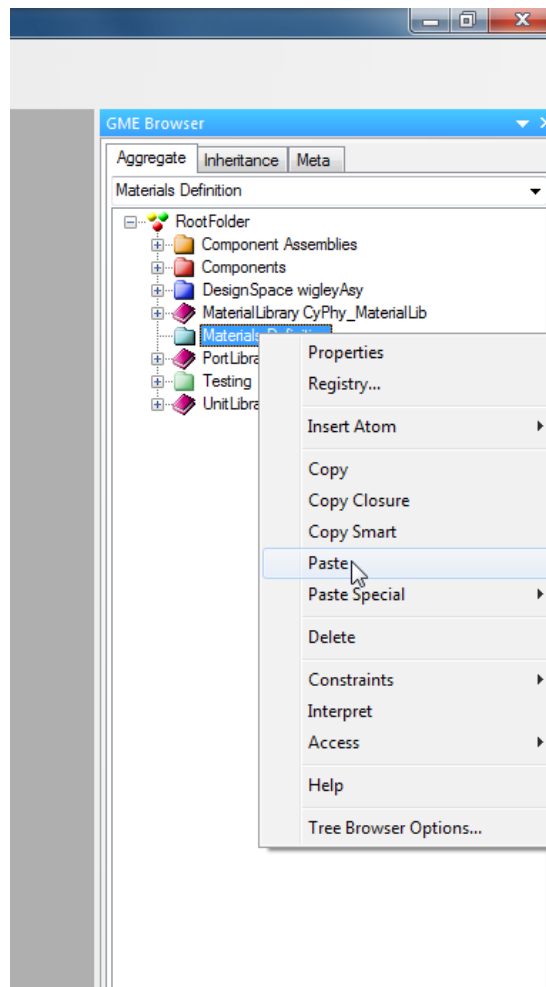


Figure 6

After the Materials Definition is created, it will be added to the component. On the upper left corner in the Part Browser, click on the SolidModeling aspect. (Fig. 7)

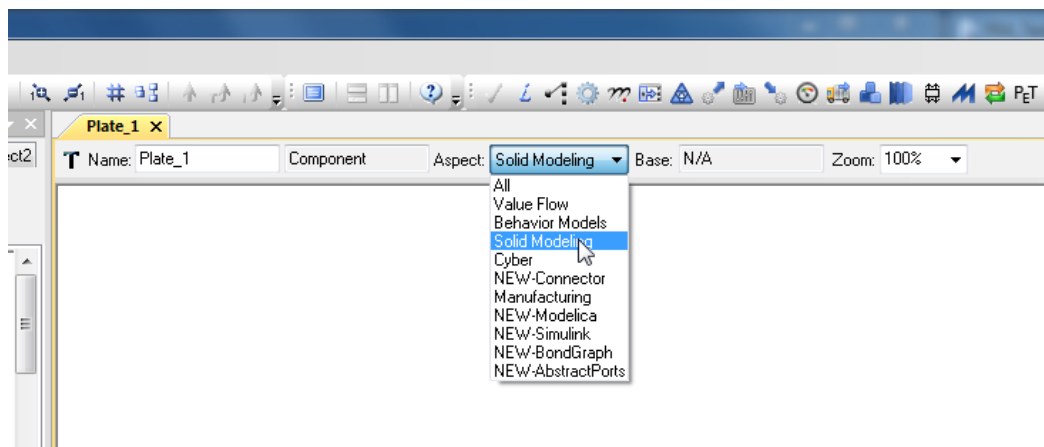


Figure 7

In the GME Browser, right click on the material and copy. (Fig. 8)

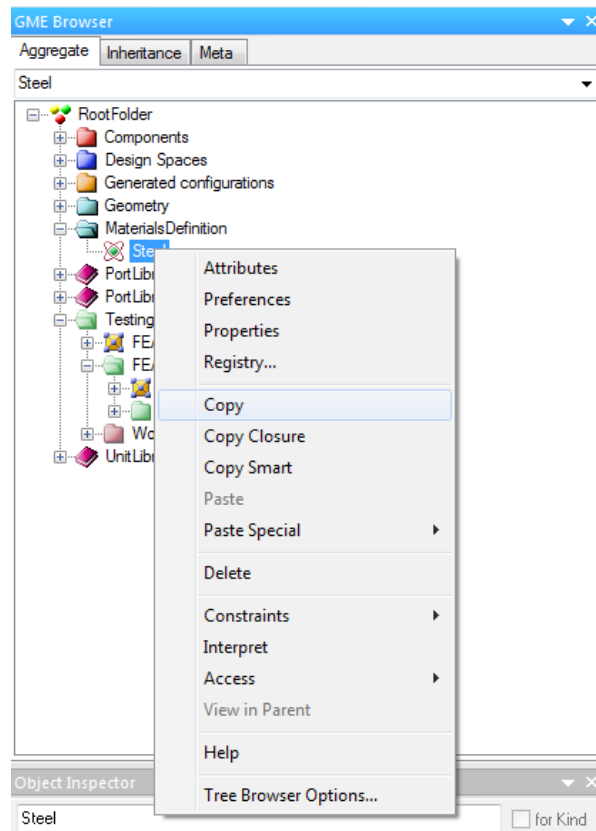


Figure 8

Right click in the part, and paste the material as reference. Pasting as reference is critical (Fig. 9). CAD parts should have the same material definitions with CyPhy. If that is not the case, modify CAD parts material definitions and make it the same with CyPhy.

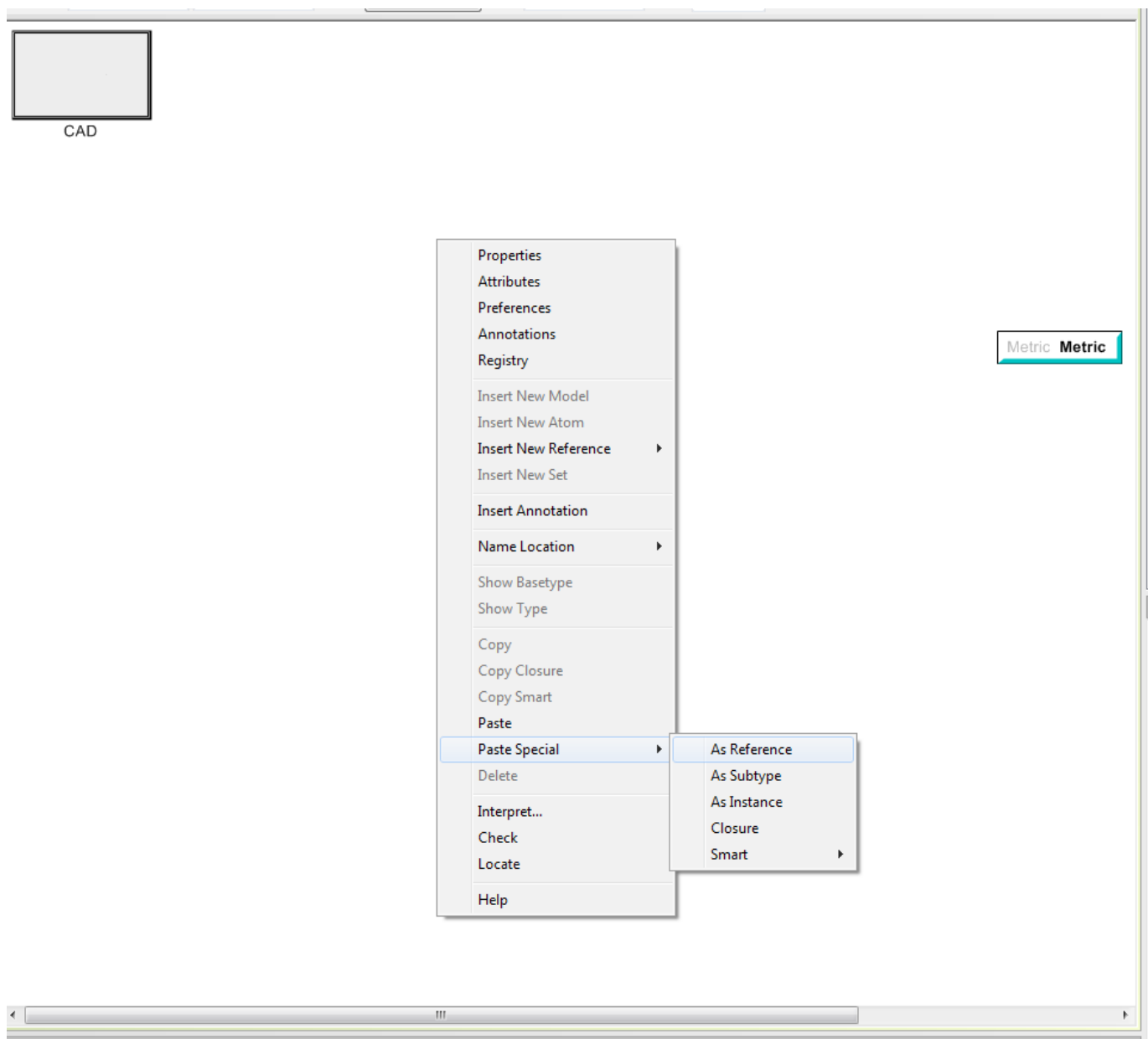


Figure 9

Step 2

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. 10)

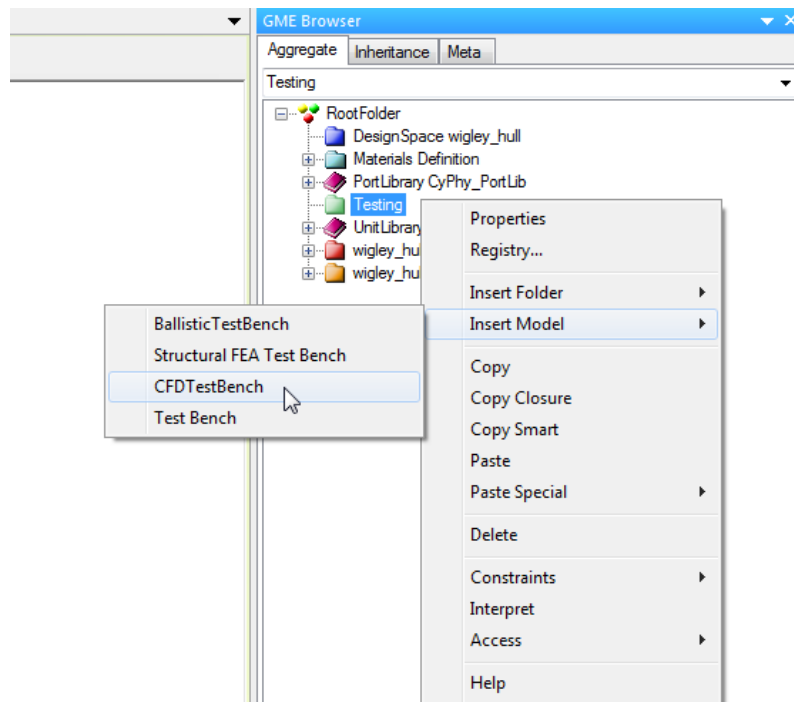


Figure 10

Double click on the created test bench. GME should now look like this. (Fig. 11)

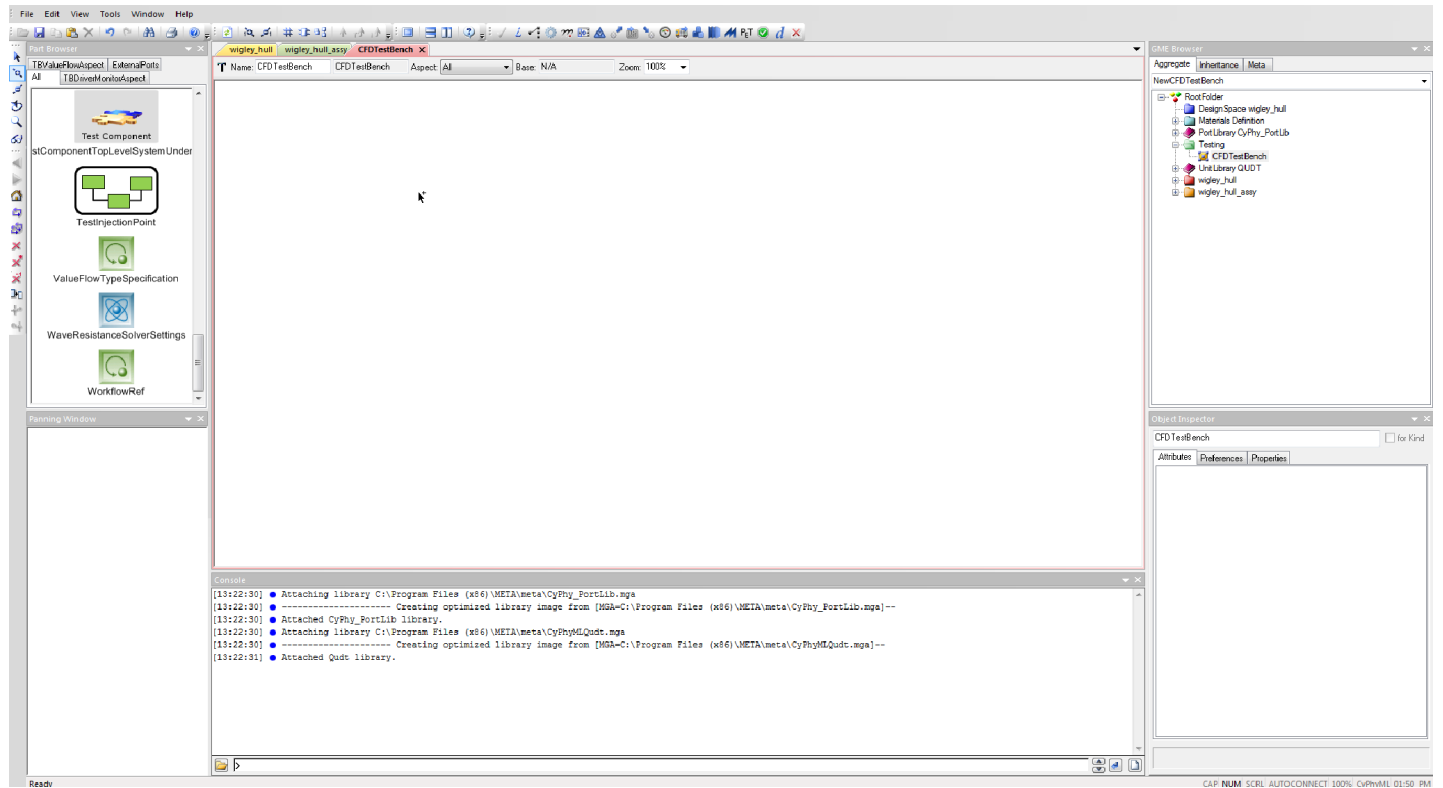


Figure 11

In the Part Browser, and the CFDComputation part. This part will execute all of the numerical calculations needed for CFD Analysis. (Fig. 12)

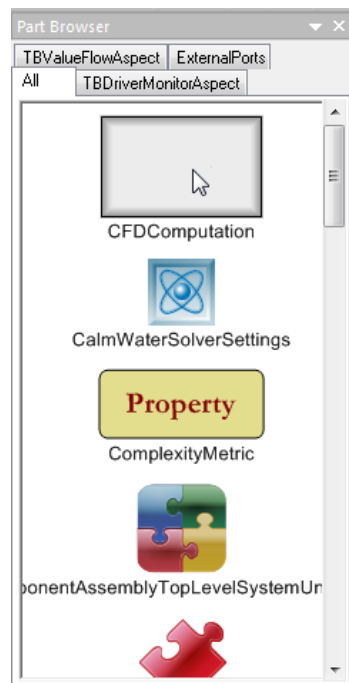


Figure 12

Drag the CFDComputation part into the test bench. (Fig. 13)



Figure 13

Step 3

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.

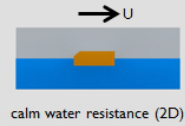
CalmWaterSolverSettings is for CFD simulations undertaken to compute performance of the vehicle for calm water conditions (vehicle traveling steadily in calm water) and WaveResistanceSolverSettings is for CFD simulations undertaken to compute performance of the vehicle in wave environments (vehicle traveling through the waves).

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. A description of each setting is presented in the below subsections. The graphical representation of each CFD aquatic tier is shown in (Fig. 14) and (Fig. 15).

Aquatic Drag Tiers

- ▶ Tier-1: Empirical correlations or analytic solution

- ▶ Tier-2: 2D, 0 DOF, calm water model



- ▶ Tier-3: 3D, 0 DOF, calm water model



- ▶ Tier-4: 2D, 0 DOF, sea state model

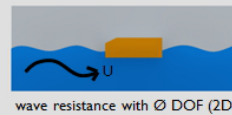
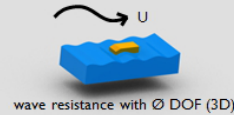


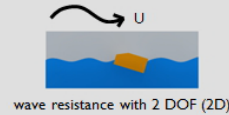
Figure 14

Aquatic Drag Tiers

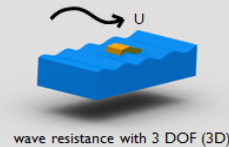
- ▶ Tier-5: 3D, 0 DOF, sea state model



- ▶ Tier-6: 2D, 2 DOF, sea state model



- ▶ Tier-7: 3D, 3 DOF, sea state model



- ▶ Tier-8: 3D, 6 DOF, sea state model

- ▶ Tier-9: Physical experiments

Figure 15

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 (Celcius):** Temperature of the fluid that the vehicle will be exposed to (Celcius).
- **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 (Celcius):** Temperature of the fluid that the vehicle will be exposed to (Celcius).
- **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. 16). Usually, 3D tiers give more accurate results with greater computational time. On the other hand, 2D tiers complete simulation faster 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 gives 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 feature 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.*
- **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 that 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 that parameter may cause simulations to crash (Recommended value < 1).

CorrelationSettings option, CalmWaterSolverSettings option and WaveResistanceSolverSettings option is presented in Figures (Fig. 17), (Fig. 18) and (Fig. 19) respectively.

6DOF

- ▶ 1: Heave
- ▶ 2: Sway
- ▶ 3: Surge
- ▶ 4: Yaw
- ▶ 5: Pitch
- ▶ 6: Roll

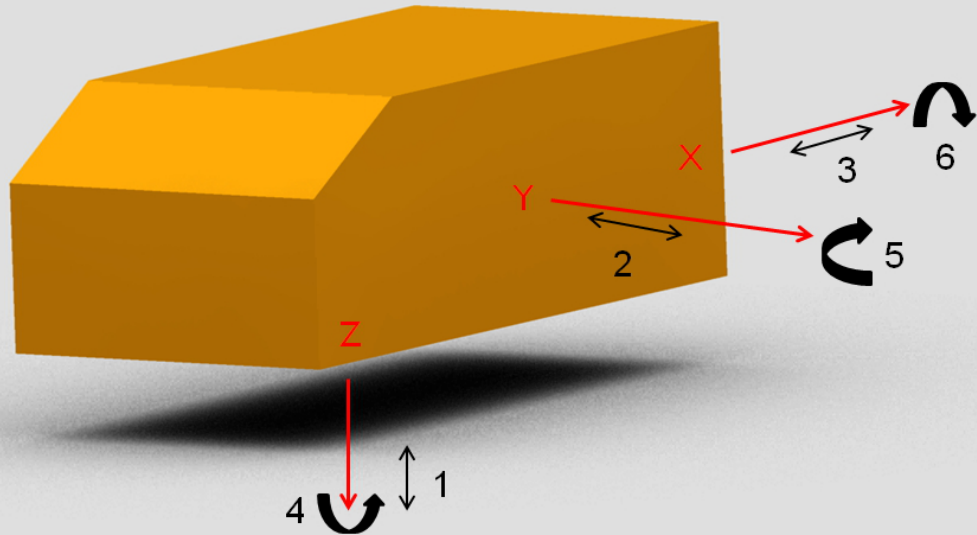


Figure 16

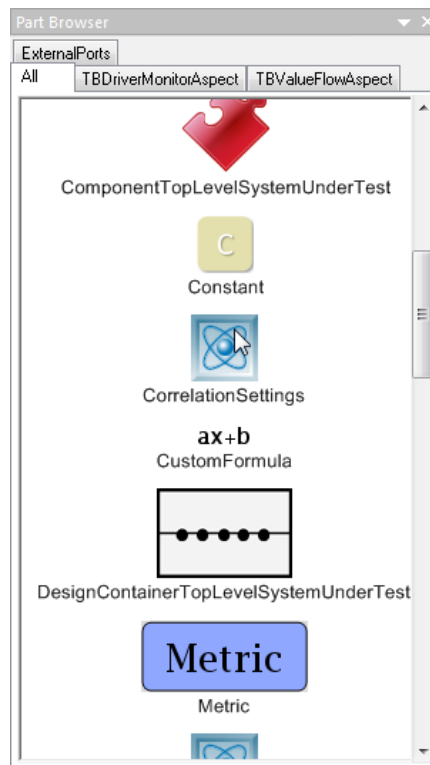


Figure 17

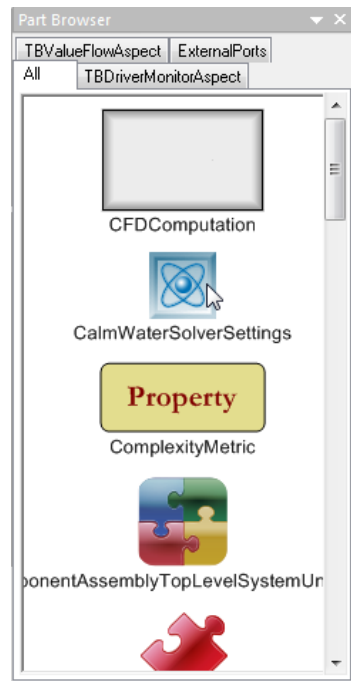


Figure 18

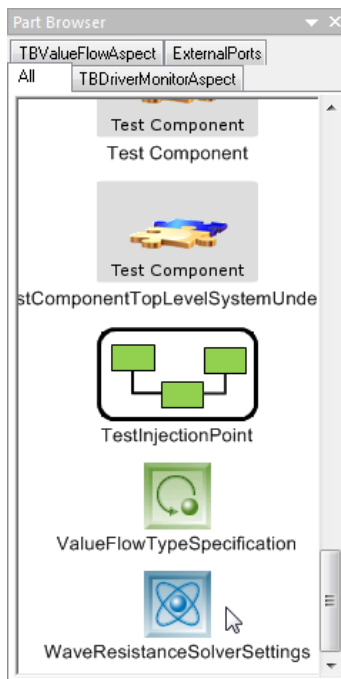


Figure 19

Drag the CFDComputation part into the test bench. (Fig. 20)

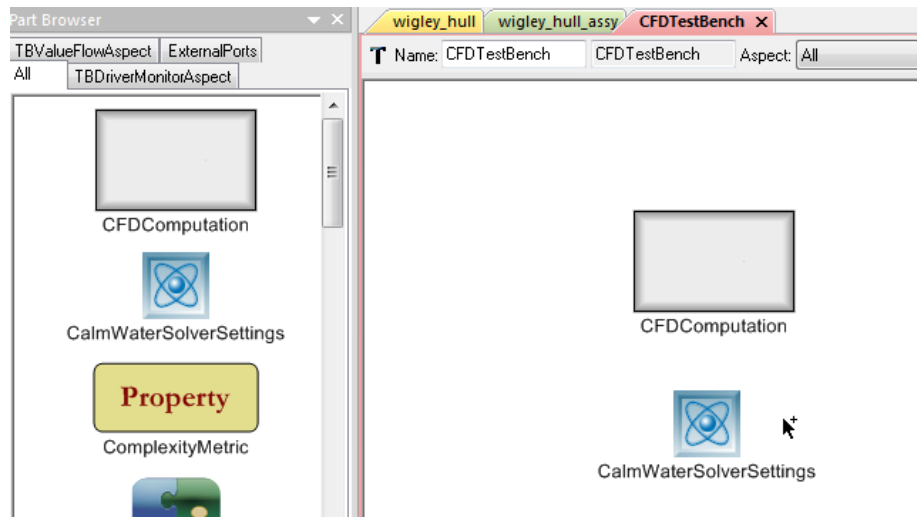


Figure 20

Double click on the CFDComputation part that has been placed in the test bench window. The screen should now look similar. (Fig. 21)

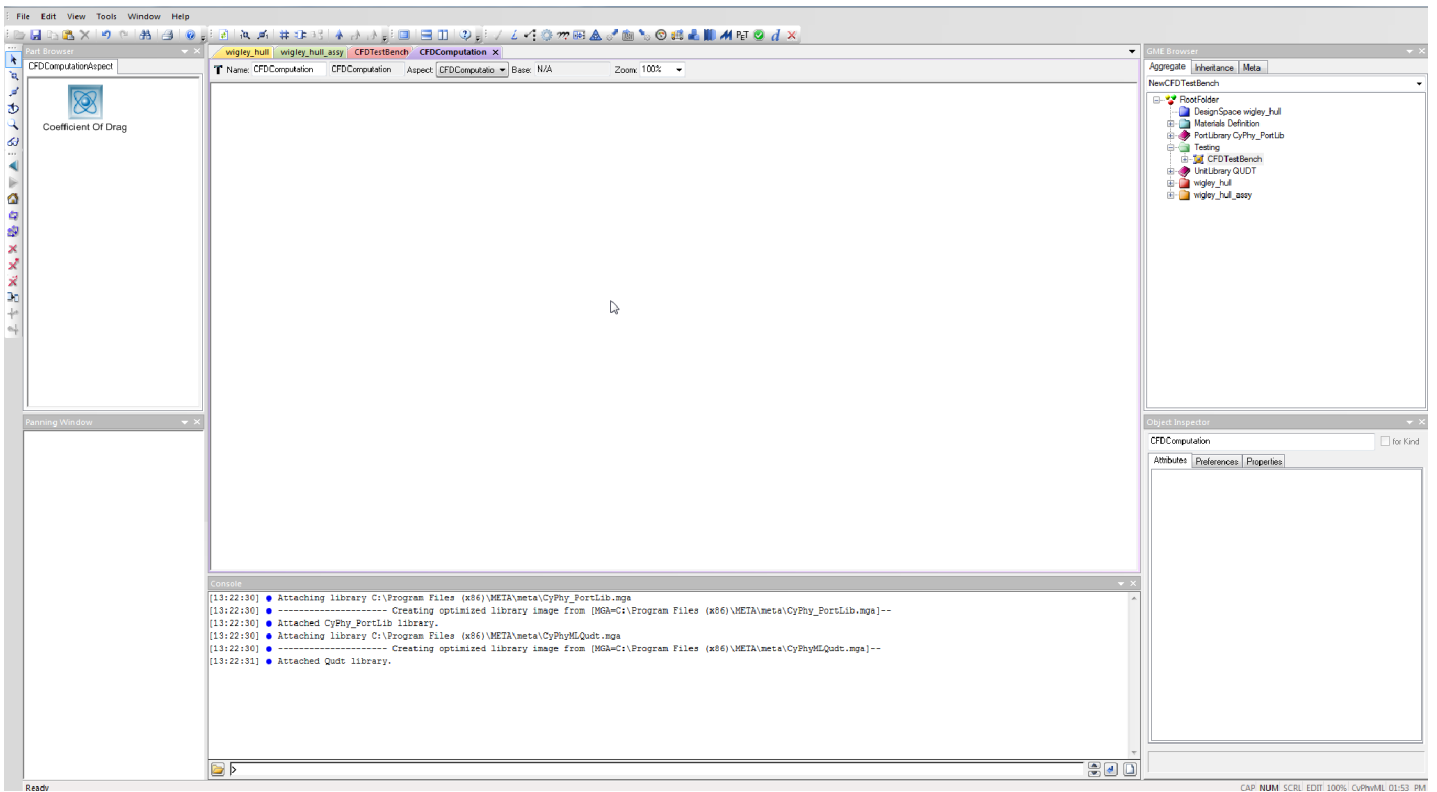


Figure 21

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. 22)

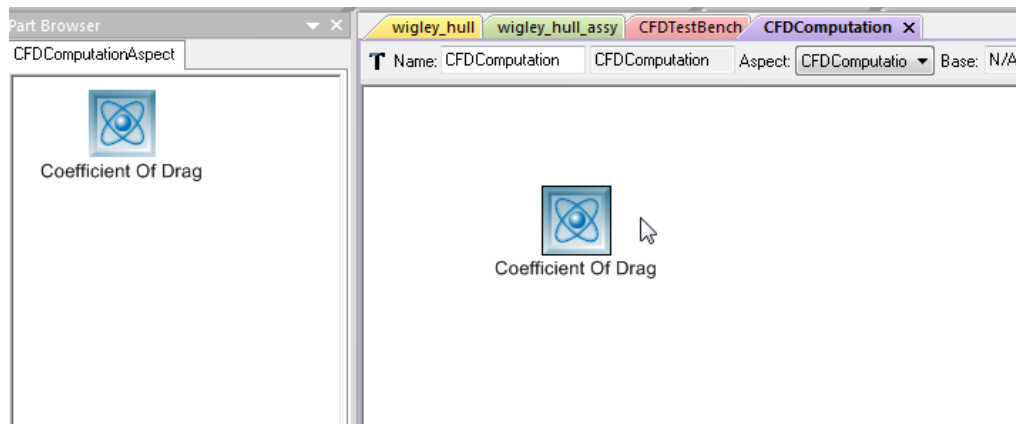


Figure 22

Click on the CFDTestBench on the top. The test bench should now look similar. (Fig. 23)

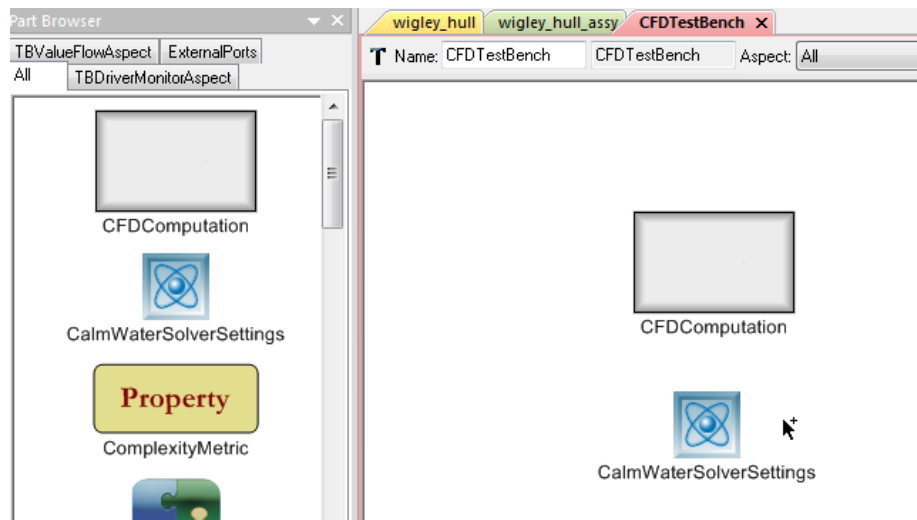


Figure 23

In the Part Browser, find the Metric part. (Fig. 24)

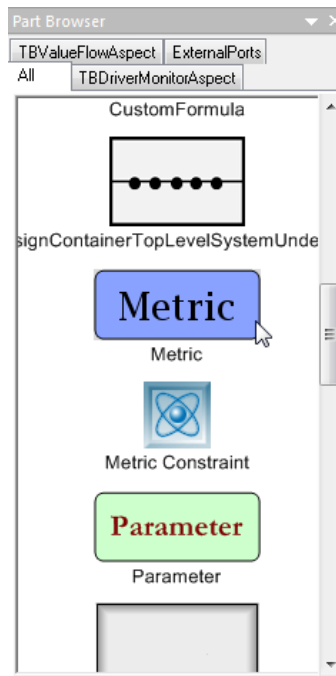


Figure 24

Right click in the part, and paste the material as reference. Pasting as reference is critical. (Fig. 25)

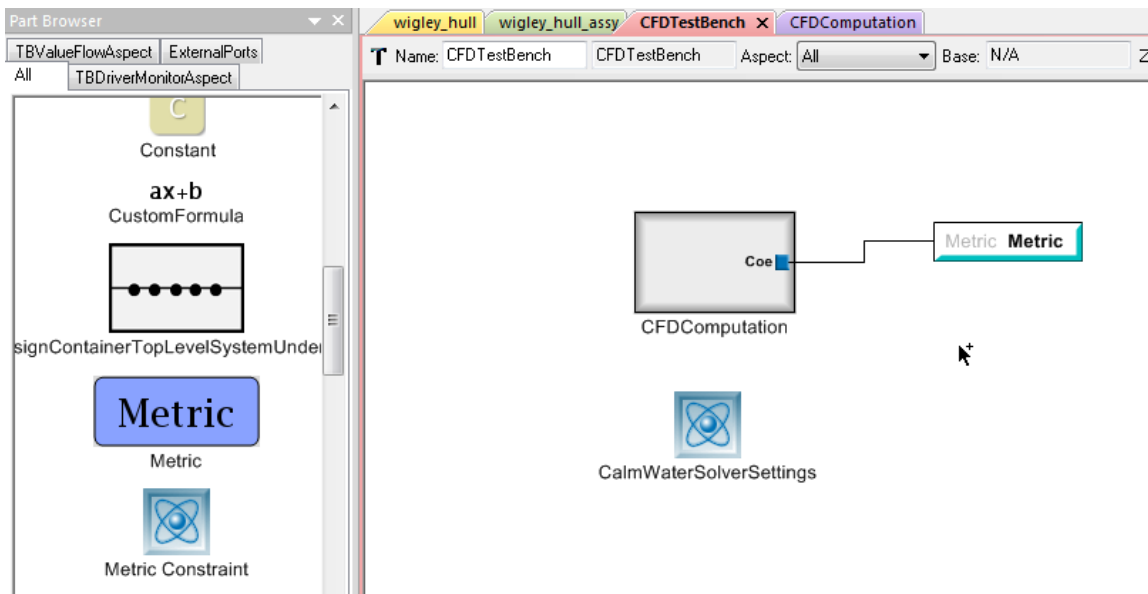


Figure 25

Step 4

Now, we will add the assembly to the test bench so that GME knows what assembly to run the test bench on. In the GME Browser window, copy the assembly. (Fig. 26)

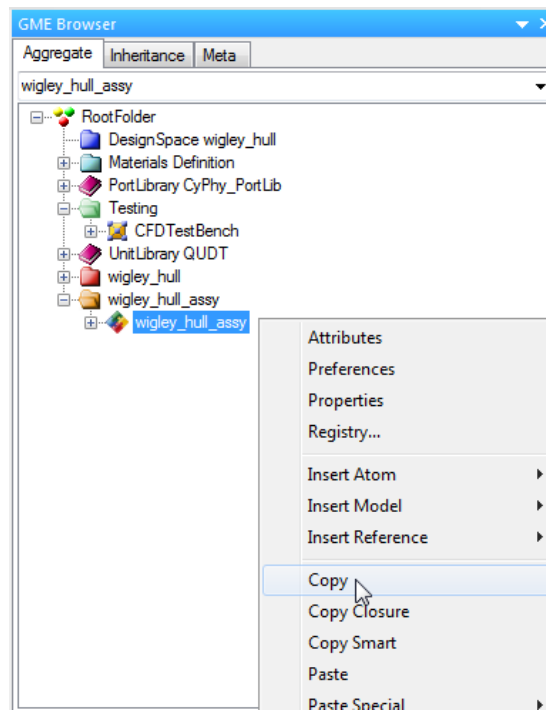


Figure 26

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. 27)

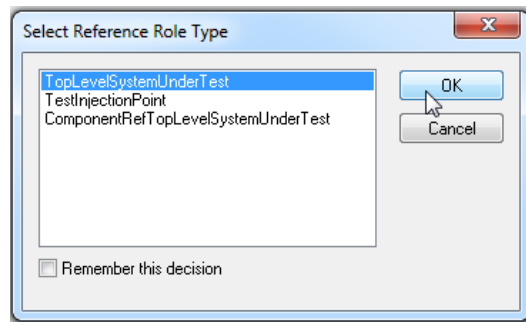


Figure 27

Step 5

On the GME Browser, right click on Testing, go to Insert Folder, and create a Workflow Definition. (Fig. 28)

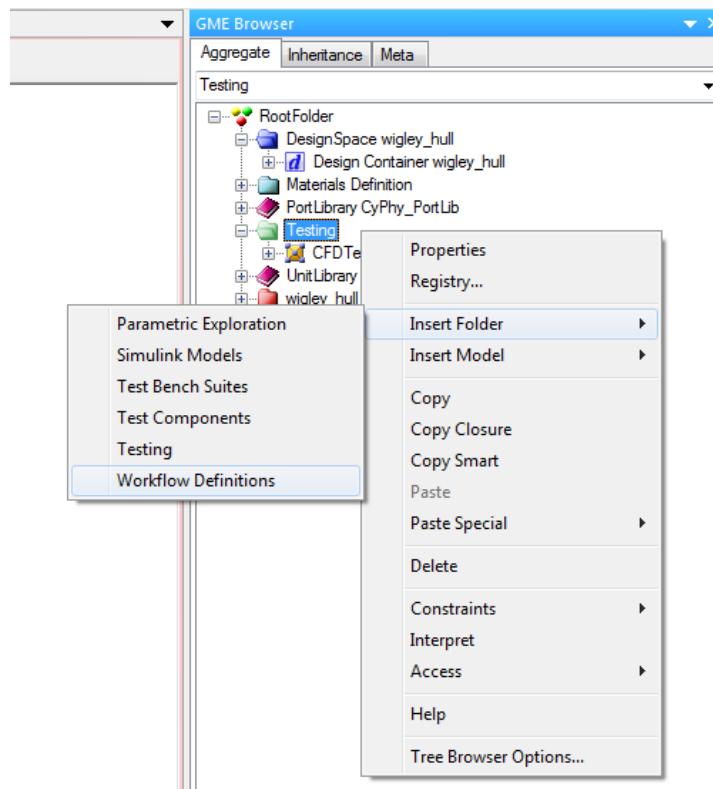


Figure 28

Click on the newly created Workflow Definition folder and insert a new Work flow. Name it CAD. (Fig. 29)

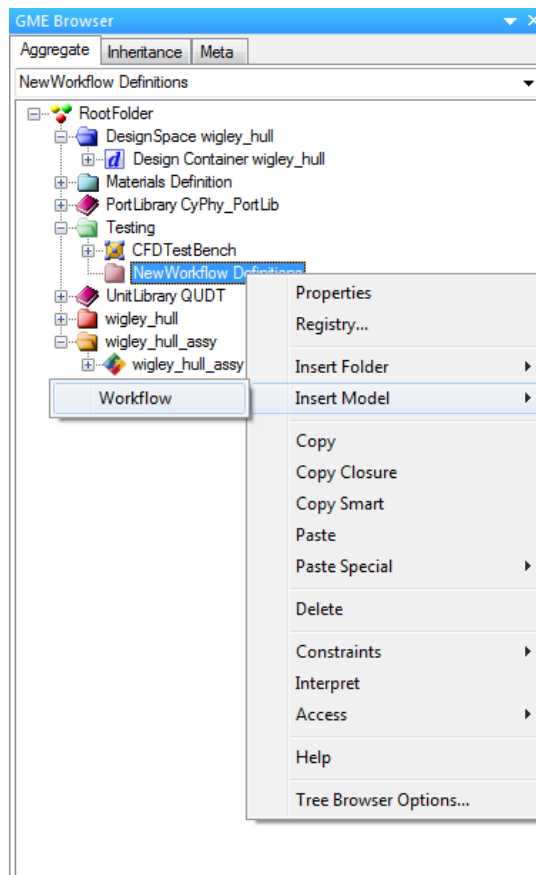


Figure 29

Double click on the CAD definition. GME should now look like this. (Fig. 30)

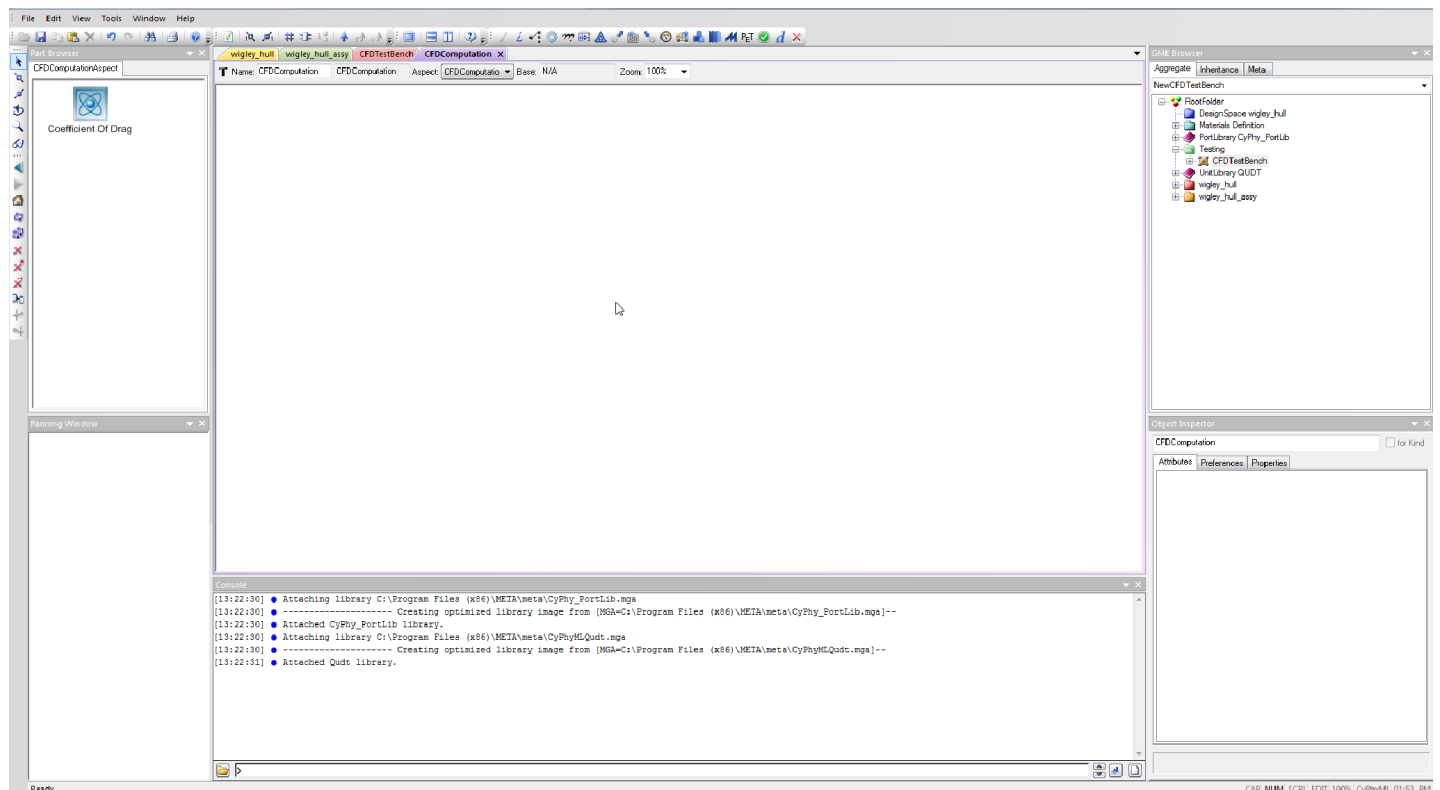


Figure 30

Right click on the CAD work flow definition and copy it. Paste it as reference in the Test Bench. (Fig. 31)

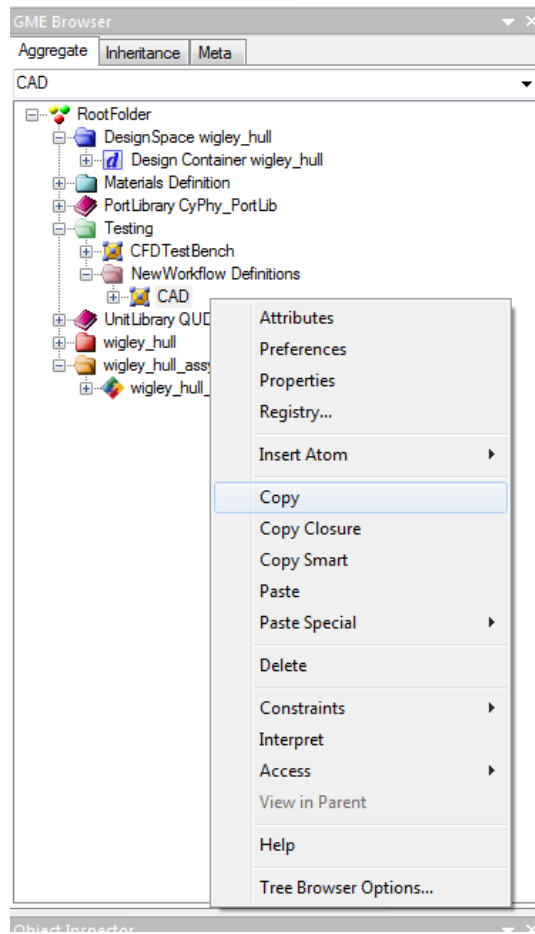


Figure 31

The Test Bench should now look like this. (Fig. 32)

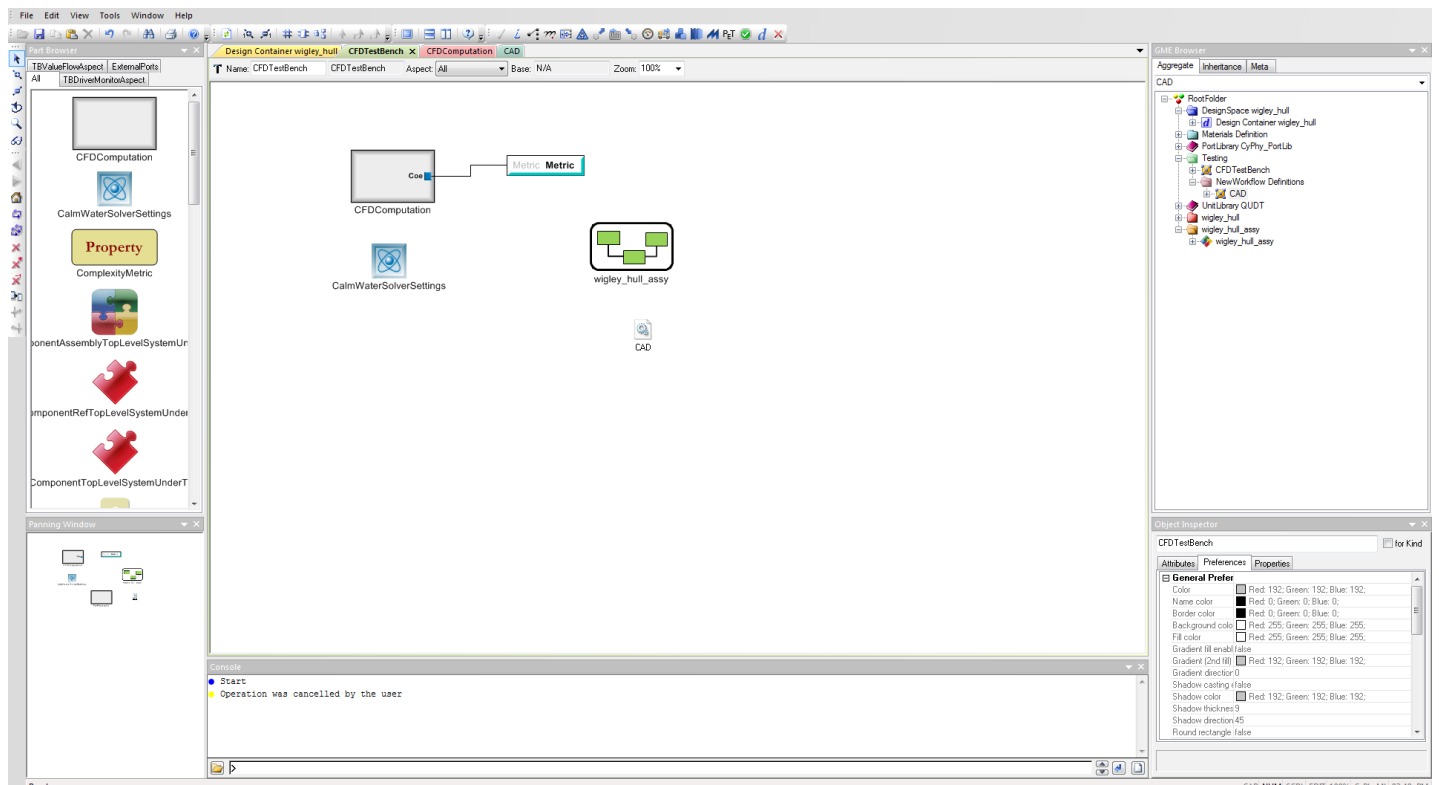


Figure 32

Step 6

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. 33)

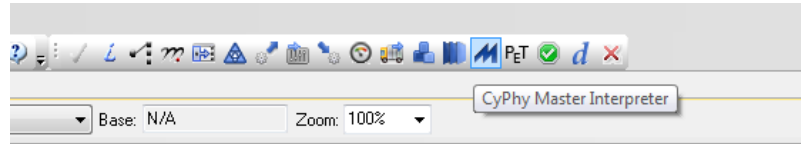


Figure 33

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. 34)

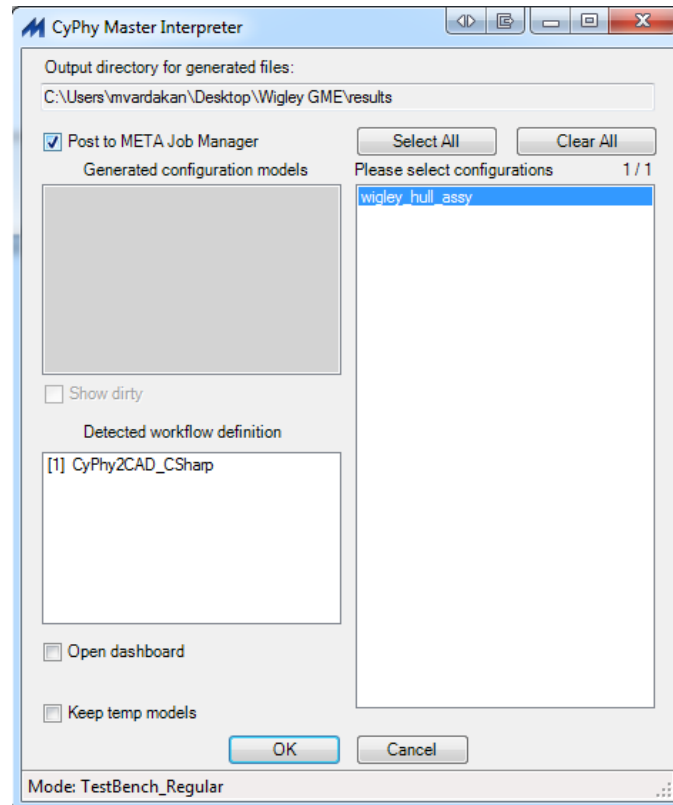


Figure 34

The CAD Options window will come up. (Fig. 35)

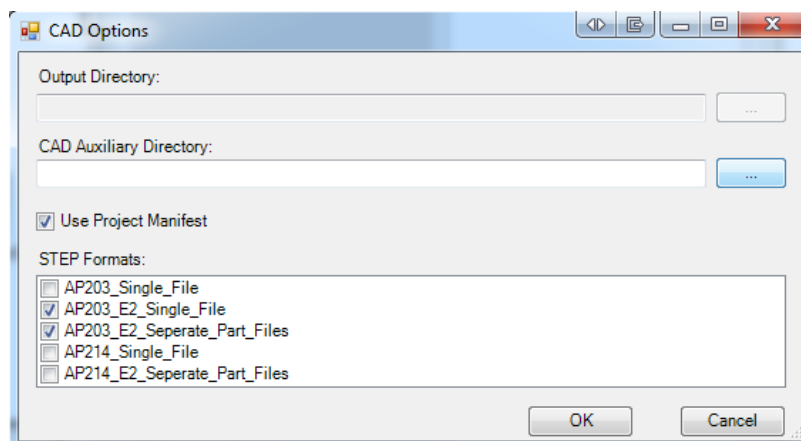


Figure 35

Select the CAD Auxiliary directory and click OK. (Fig. 36)

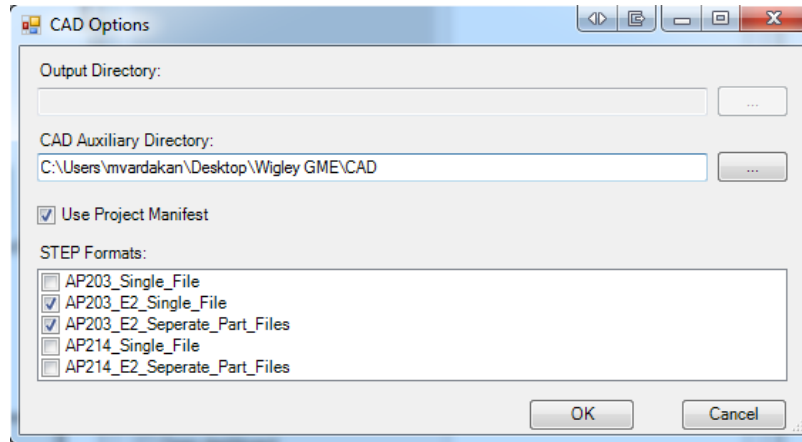


Figure 36

The JobManager Configuration window will now open. (Fig. 37)

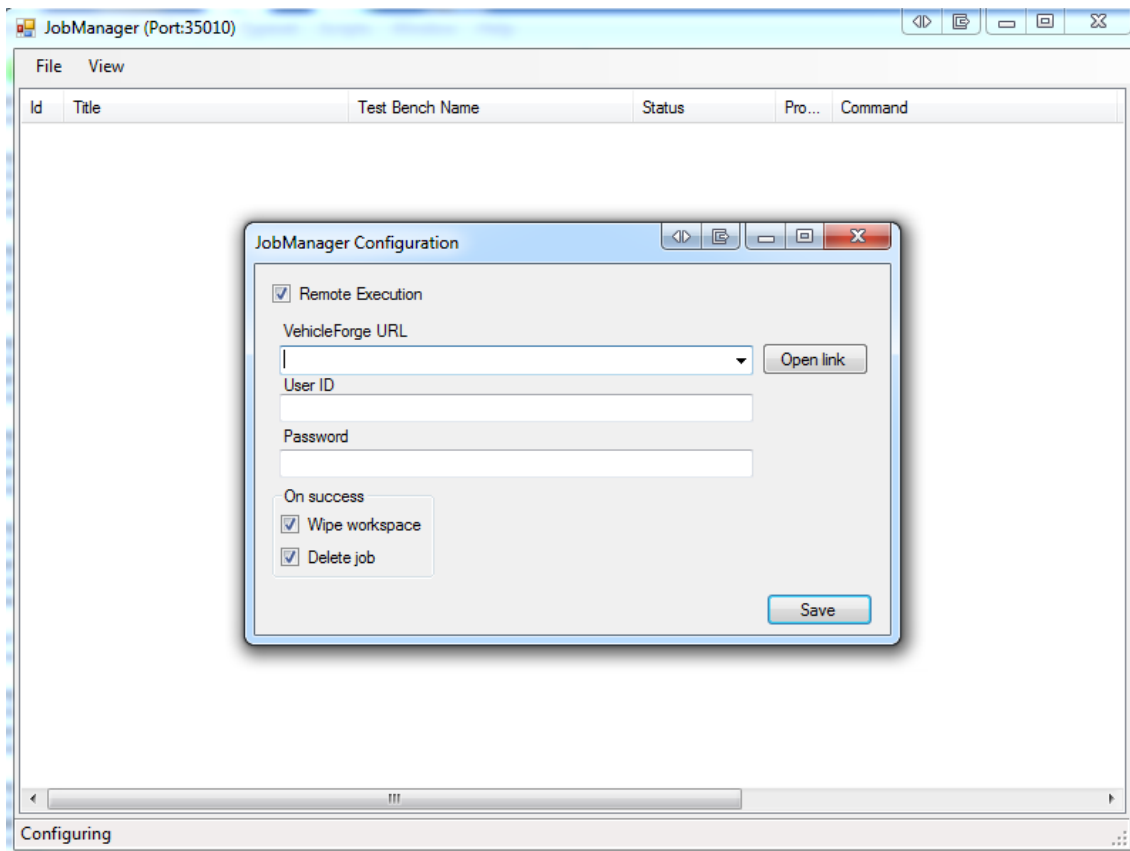


Figure 37

Input your VF information and click Save. (Fig. 38)

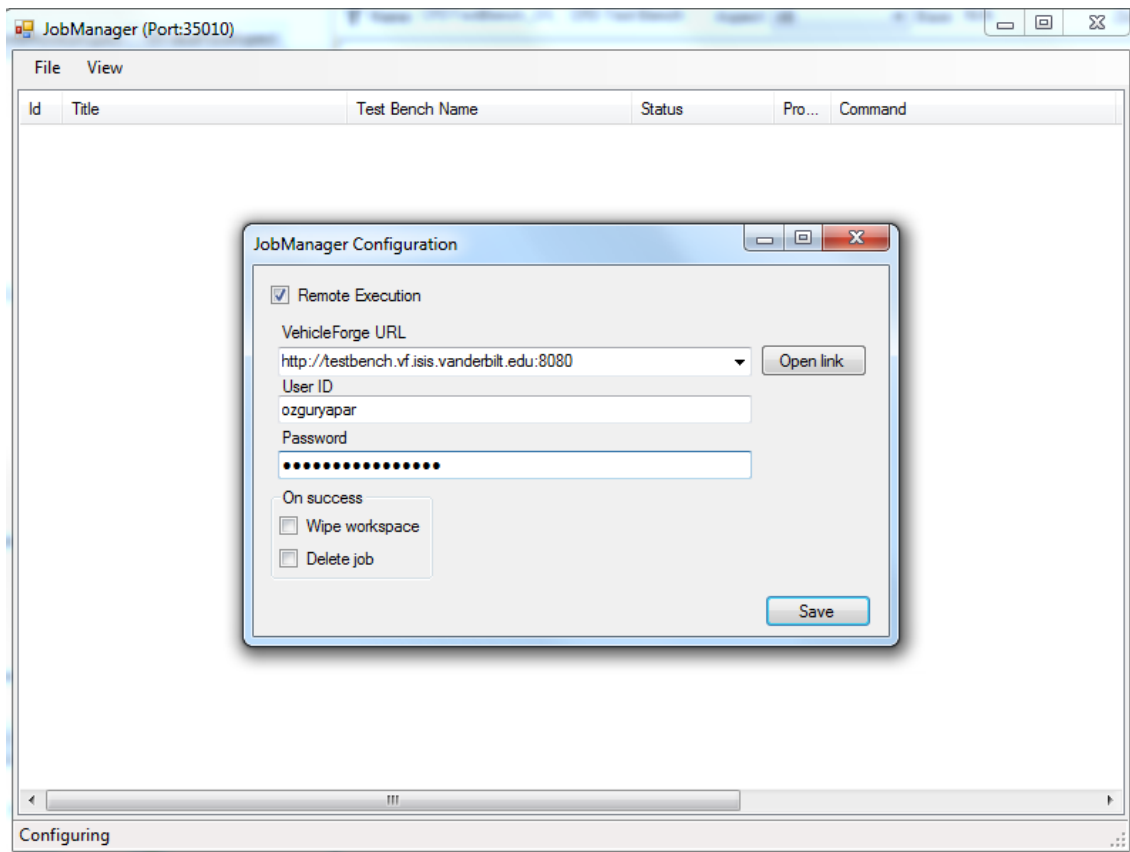


Figure 38

Job Manager will start the CFD computations. (Fig. 39)

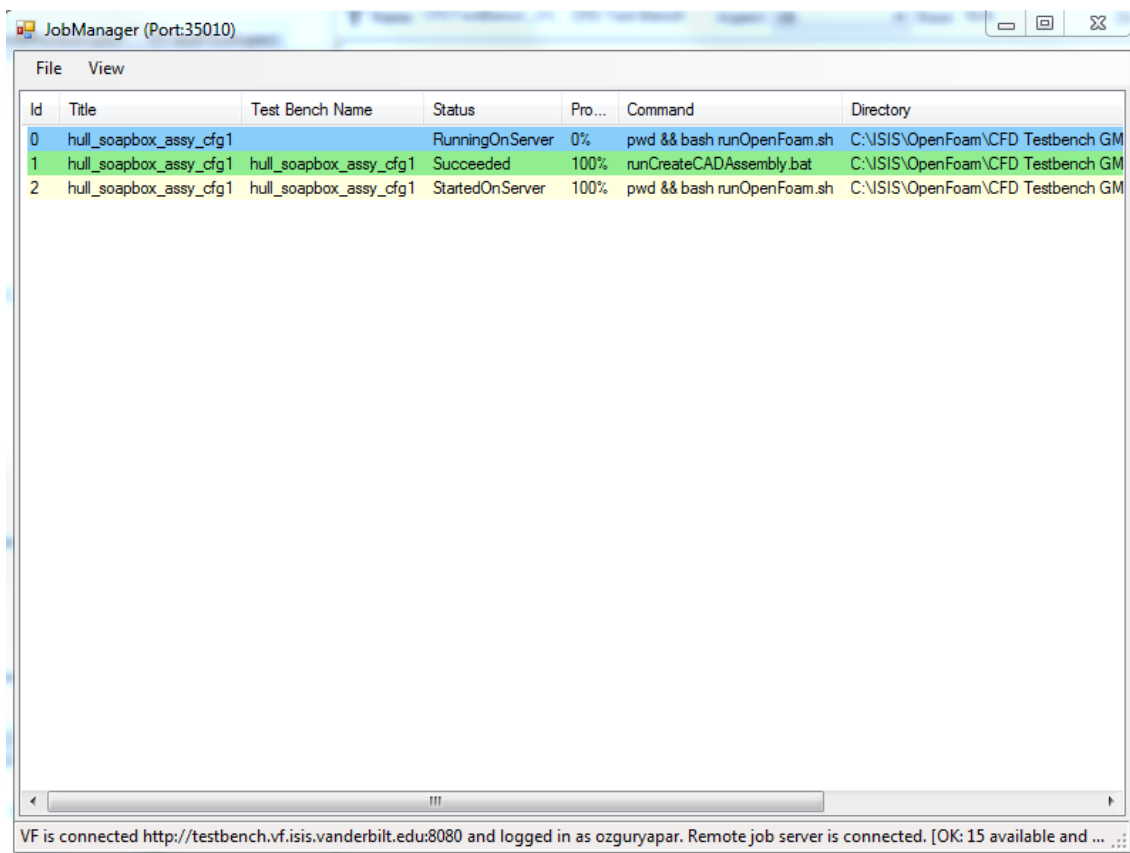


Figure 39

After the job is completed, to see the results, open the main folder for the GME. (Fig. 40)

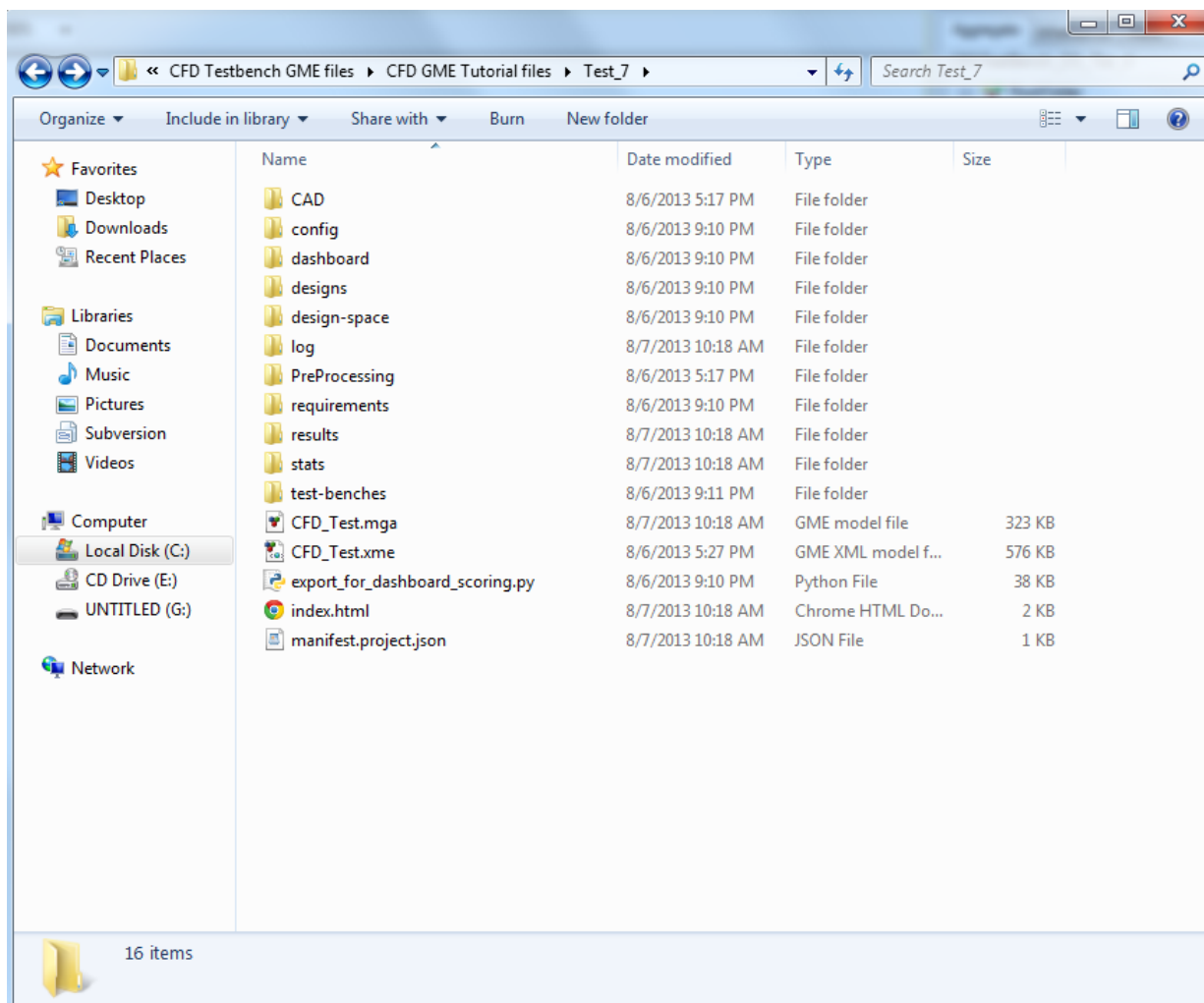


Figure 40

Click on the correct folder. (Fig. 41)

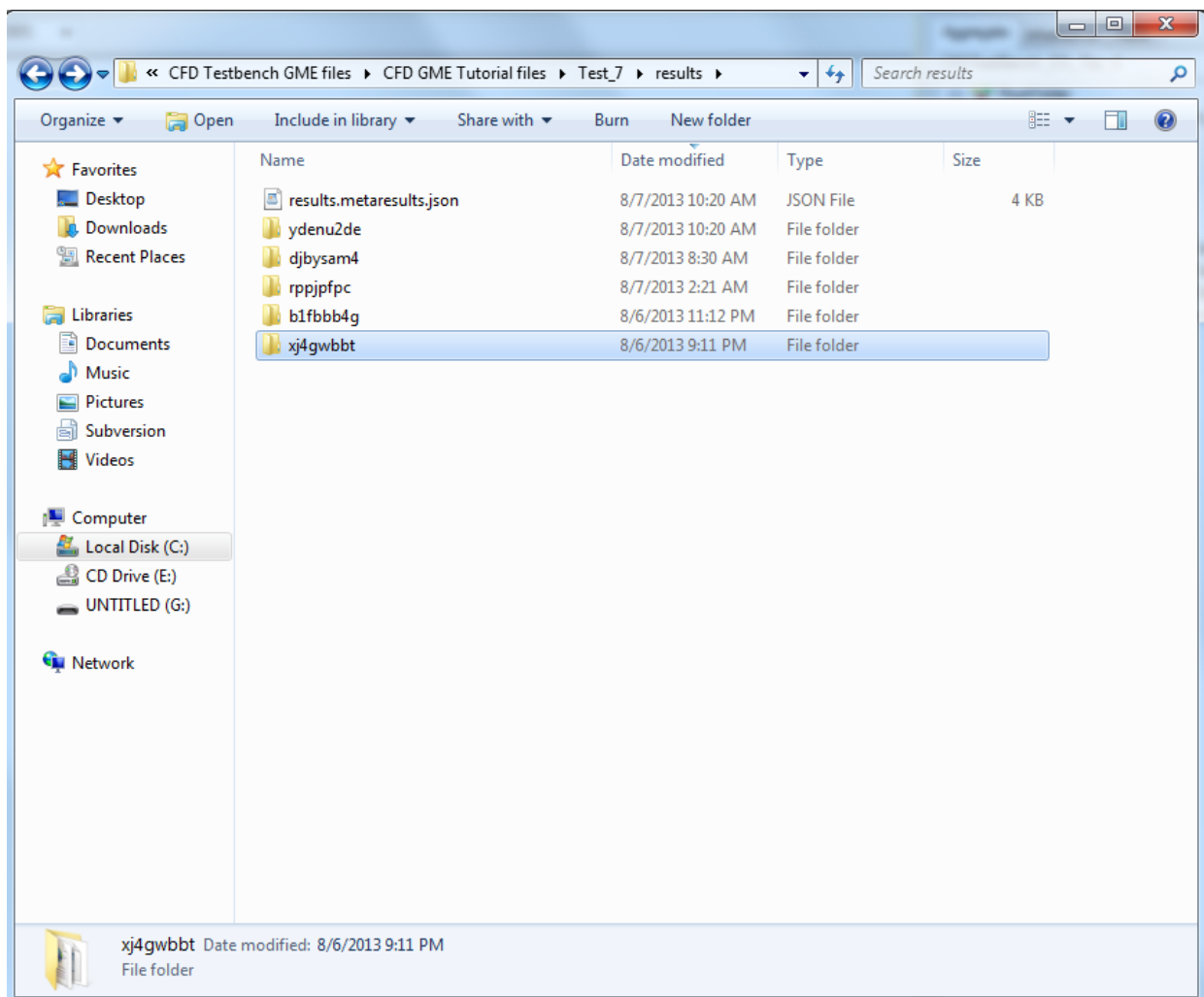


Figure 41

The forceOutput.dat file, which gives the "Average Drag Force", can be found in the path shown in (Fig. 42).

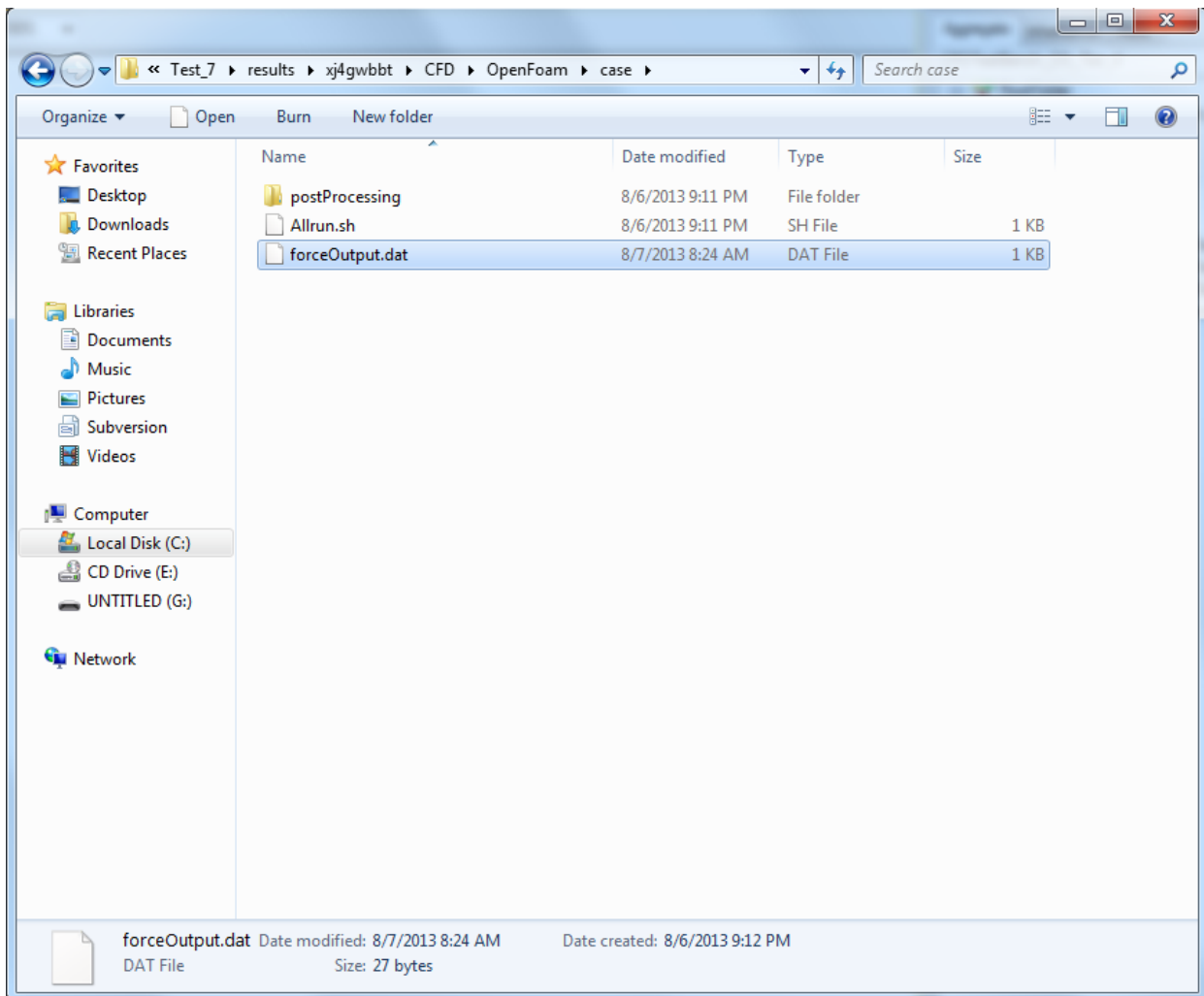


Figure 42

The summary.testresults.json is located in the location shown in (Fig. 43).

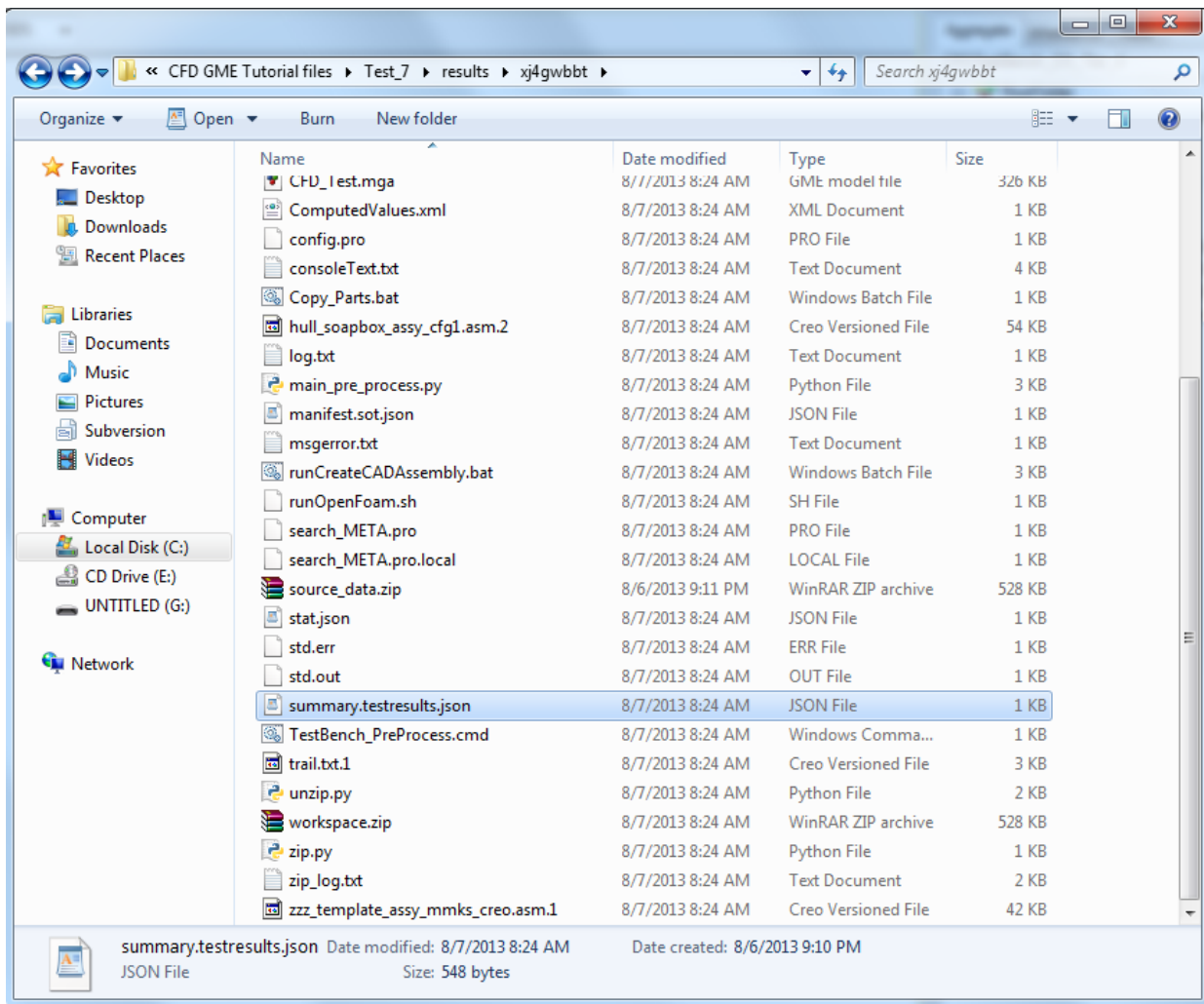


Figure 43

Opening the summary.testresults.json file will allow you to view the coefficient of drag. (Fig. 44)

```
{
  "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 44

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.

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.

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.

Metrics

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 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.

Required Connections to System Under Test

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. 45).

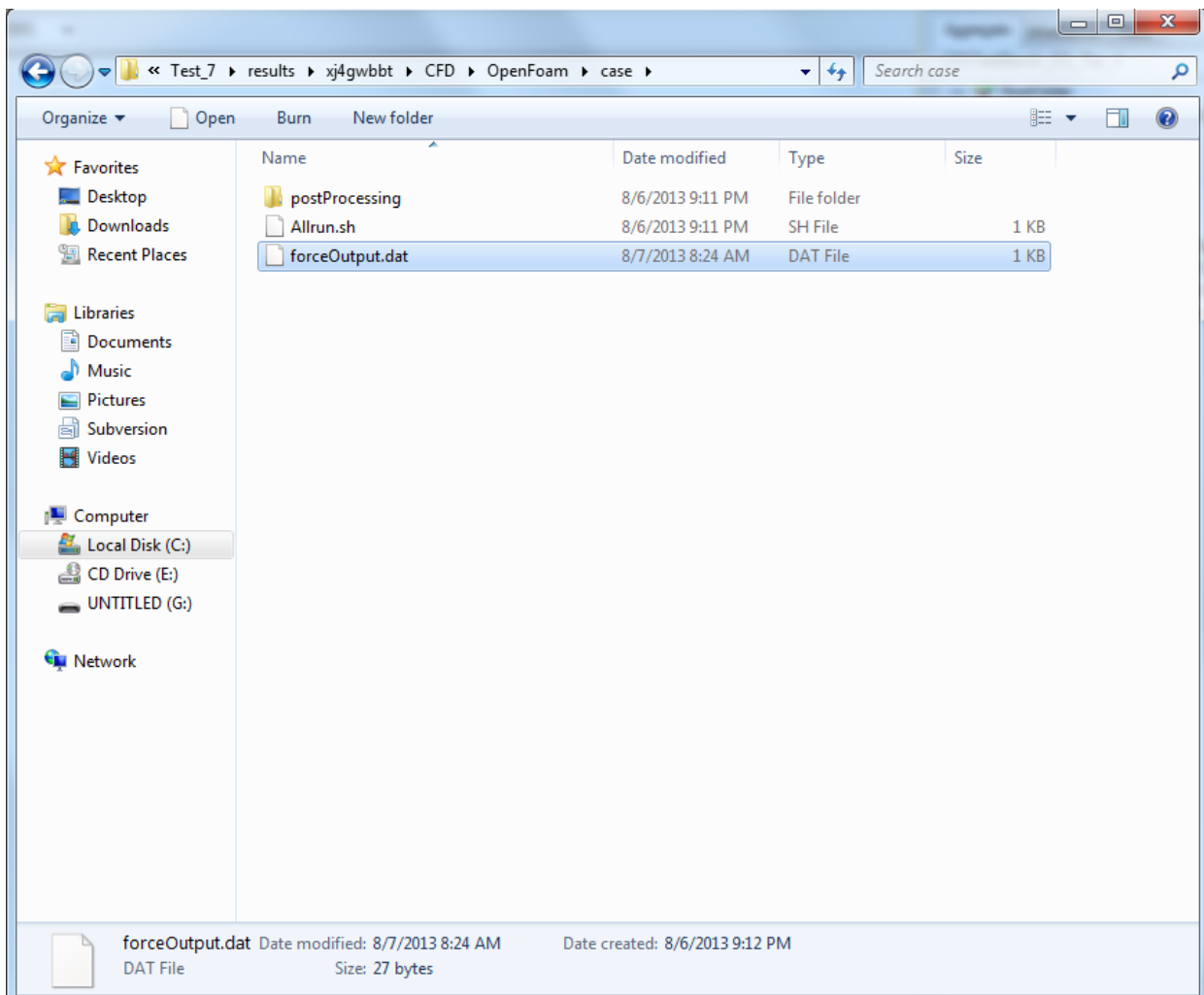


Figure 45

The summary.testresults.json is found in the location shown in (Fig. 46).

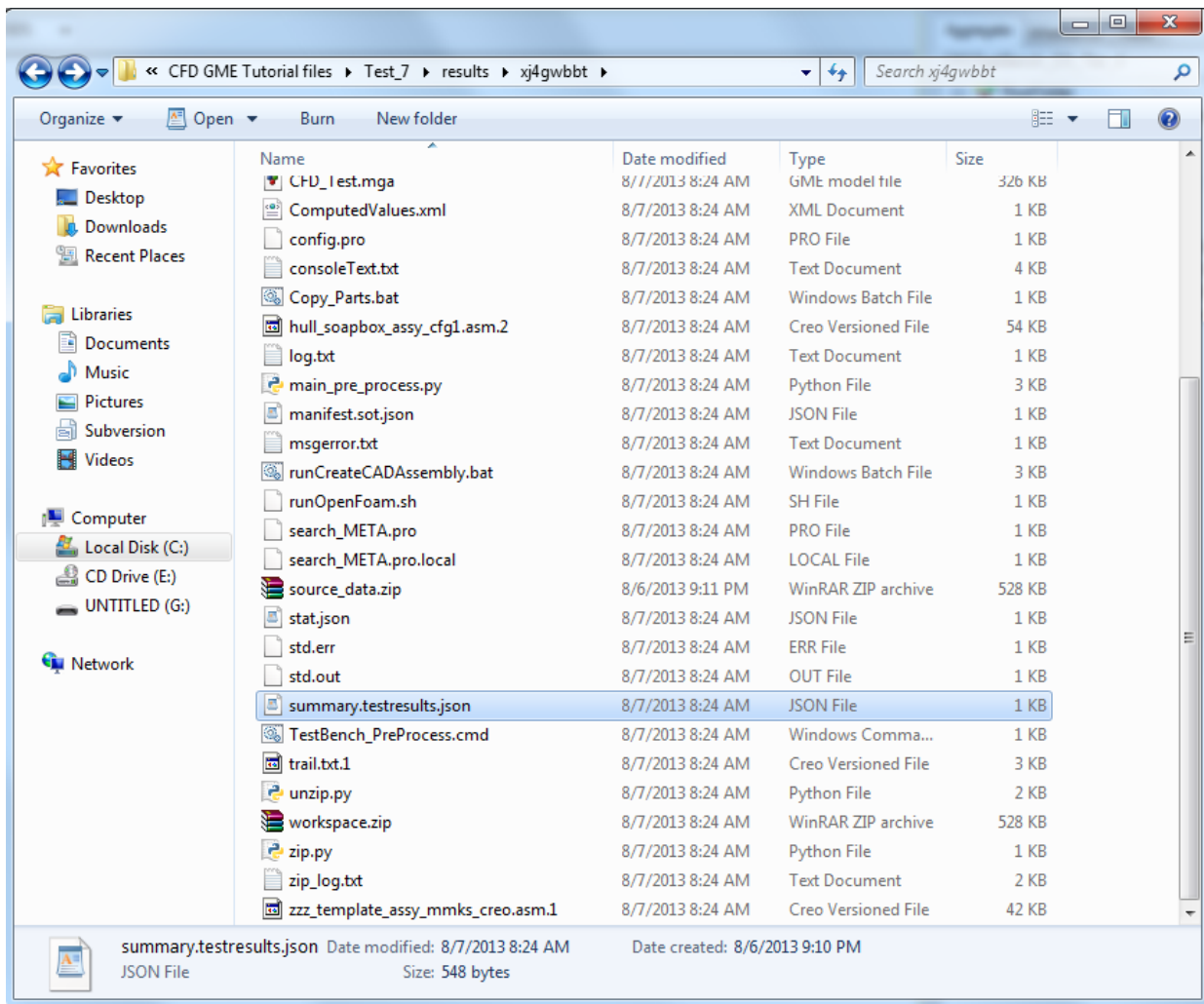


Figure 46

Opening the summary.testresults.json file will allow you to view the coefficient of drag. (Fig. 47)

```
{
  "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 47

The CFD simulation results of the Wigley hull example can be compared with the experimental values presented in the plots below (Fig. 48 and 49).

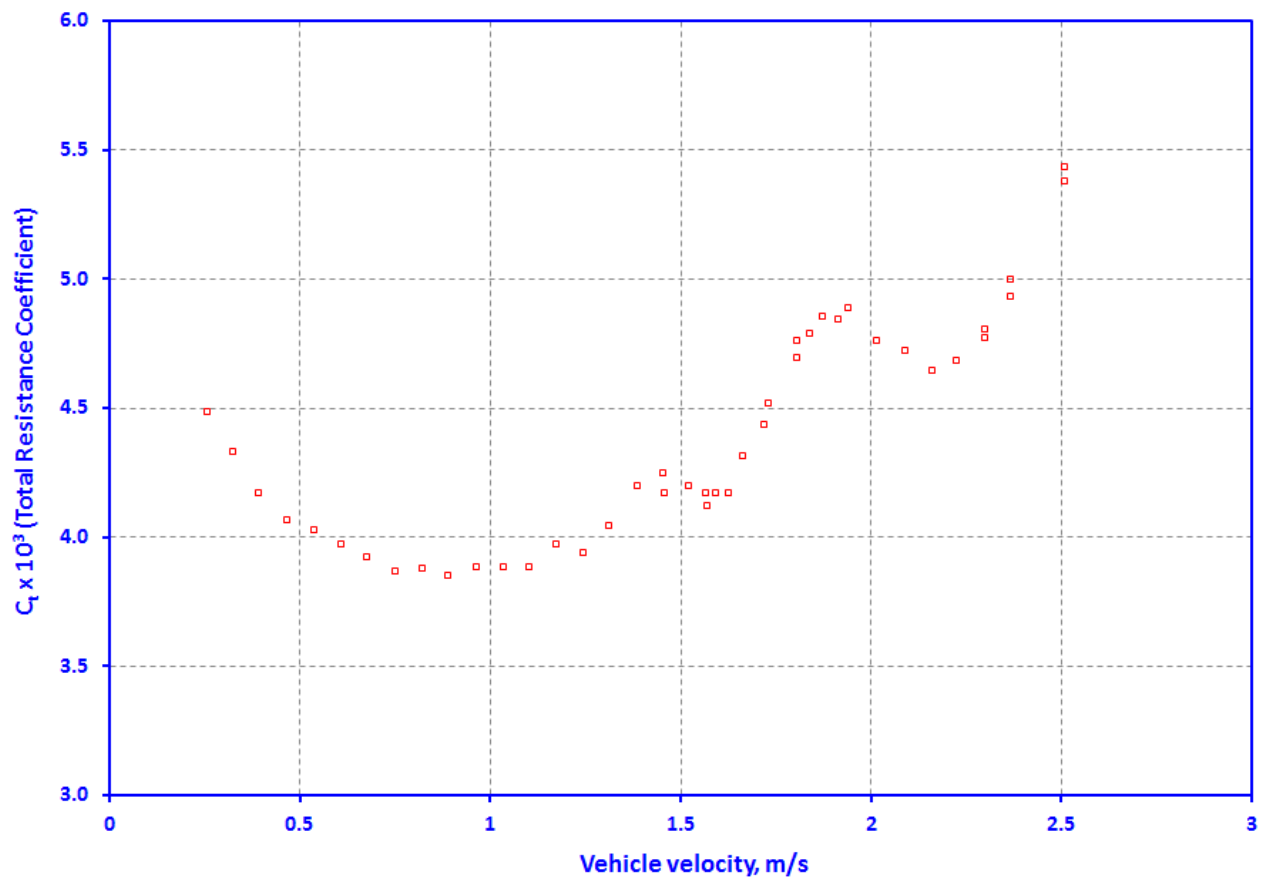


Figure 48

