User's Guide to using OrcaFlexInterface in FAST v8

Introduction

This document briefly describes how to use OrcaFlex coupled to FAST v8 for modeling floating offshore wind turbines. OrcaFlex is a commercial software package developed by Orcina for the design and analysis of marine systems. When the OrcaFlexInterface module is used in FAST, all hydrodynamic and mooring loads will be computed using OrcaFlex, while the turbine, tower, and floating platform structural dynamics; aerodynamics; and control and electrical-drive dynamics will be computed by FAST. To use this module with FAST, a valid OrcaFlex license is needed and a special DLL compiled by Orcina is required (*FASTlinkDLL.dll*). This DLL is called by FAST during the FAST simulation to compute the loads on the platform by OrcaFlex. A 32- and 64-bit version of *FASTlinkDLL.dll* compatible with the 32- and 64-bit Windows executable version of FAST is available at http://orcina.com/Support/FASTlink.zip. One should choose the version that matches the addressing scheme of the version of OrcaFlex that is installed, otherwise the simulation will not run. A 32- and 64-bit Windows executable version of FAST v8 is available at https://nwtc.nrel.gov/FAST8.

At this time, FAST's ElastoDyn structural-dynamics module assumes for a floating platform that the substructure (floating platform) is a six degree-of-freedom (DOF) rigid body. At each time step (the FAST glue code and OrcaFlex must both use the same time step), OrcaFlex receives the six-DOF position, orientation, and velocities of the platform, computes the hydrodynamic and mooring loads, and returns them back to FAST in the form of a six-by-six added-mass matrix and six-by-one load vector (three forces and three moments). FAST will take these OrcaFlex outputs, along with the six-DOF platform acceleration, and compute the total six-component hydrodynamic and mooring loads to be applied to the platform in ElastoDyn, including hydrodynamic added-mass effects.

This document was written when the interface between OrcaFlex 9.8d and FAST v8.12 was first developed, but may apply to newer releases. It may also work with earlier versions of OrcaFlex, starting with 9.7b, though compatibility has not been tested.

Modifying the FAST Input File

When the OrcaFlexInterface module is used in FAST, all hydrodynamic and mooring loads will be computed using OrcaFlex. This replaces the HydroDyn module and alternative mooring modules. In the FAST input file, this is done by setting the following options in the FAST primary input file:

- CompHydro = 0
- CompMooring = 4
- MooringFile = "*<OrcaIptFileName>*"

where *<OrcaIptFileName>* is the name of the input file for the OrcaFlexInterface module. The format of this file is as given in Appendix A. This file specifies the name of the OrcaFlex input file without the *.dat* file extension (OrcaIptFileName) and the name of the Windows DLL supplied by Orcina with the *.dll* file extension (DLL_FileName).

As OrcaFlex and the FAST glue code must both use the same time step, an appropriate time step suitable to both software should be selected. Additionally, the six platform DOFs in FAST's ElastoDyn module should also be enabled to properly transfer loads and motions between FAST and OrcaFlex. Also, the same platform initial conditions should be set in both ElastoDyn and OrcaFlex.

No other changes are required or limitations imposed on FAST.

FAST Output Variables

When FAST is executed with the OrcaFlexInterface module enabled, an additional 18 output channels are automatically added to the FAST output file (see Table 1). Additional results calculated internally within OrcaFlex are written to the OrcaFlex simulation file, *<OrcaIptFileName>.sim*, accessible through OrcaFlex.

<div align="center">Table 1: Output Parameters from OrcaFlexInterface</div>

Name	Description	Convention	Units
OrcaFxi	Total platform horizontal surge force (including added mass)	Directed along x_i-axis	(kN)
OrcaFyi	Total platform horizontal sway force (including added mass)	Directed along y_i-axis	(kN)
OrcaFzi	Total platform vertical heave force (including added mass)	Directed along z_i-axis	(kN)
OrcaMxi	Total platform roll tilt moment (including added mass)	About the x_i-axis	(kN·m)
OrcaMyi	Total platform pitch tilt moment (including added mass)	About the y_i-axis	(kN·m)
OrcaMzi	Total platform yaw moment (including added mass)	About the z_i-axis	(kN·m)
OrcaHMFxi	Platform horizontal surge force calculated by OrcaFlex (hydrodynamic + mooring)	Directed along x_i-axis	(kN)
OrcaHMFyi	Platform horizontal sway force calculated by OrcaFlex (hydrodynamic + mooring)	Directed along y_i-axis	(kN)
OrcaHMFzi	Platform vertical heave force calculated by OrcaFlex (hydrodynamic + mooring)	Directed along z_i-axis	(kN)
OrcaHMMxi	Platform horizontal roll tilt moment calculated by OrcaFlex (hydrodynamic + mooring)	About the x_i-axis	(kN·m)
OrcaHMMyi	Platform horizontal pitch tilt moment calculated by OrcaFlex (hydrodynamic + mooring)	About the y_i-axis	(kN·m)
OrcaHMMzi	Platform horizontal yaw moment calculated by OrcaFlex (hydrodynamic + mooring)	About the z_i-axis	(kN·m)
OrcaAMFxi	Platform horizontal surge force due to added mass effects	Directed along x_i-axis	(kN)
OrcaAMFyi	Platform horizontal sway force due to added mass effects	Directed along y_i-axis	(kN)
OrcaAMFzi	Platform vertical heave force due to added mass effects	Directed along z_i-axis	(kN)
OrcaAMMxi	Platform horizontal pitch roll moment due to added	About the x_i-axis	(kN·m)

	mass effects		
OrcaAMMyi	Platform horizontal pitch tilt moment due to added mass effects	About the y_i-axis	(kN·m)
OrcaAMMzi	Platform horizontal yaw moment due to added mass effects	About the z_i-axis	(kN·m)

Constructing the OrcaFlex Model

The steps required to develop an OrcaFlex model compatible with FAST are as follows:

1. In OrcaFlex, open a **New Model (Ctrl+N)**.
2. In the model browser, double click on **Variable Data**. In the **Externally Calculated Data** panel, select **External Functions**. Add one **Number of data sources**, and rename **ExternalFunction1** to **ExtFn**. Browse for the *FASTlinkDLL.dll* file. For the **Function Name**, select **ExtFn**.
3. In the model browser, go to **General** -> **Integration and Time Step** -> **Integration Method**, and make sure this value is set to **Implicit**.
4. Insert a new vessel. In the model browser, go to **Vessel1**:
   a. Rename **Vessel1** to **Platform**.
   b. Set **Initial Position** and **Orientation** both to (0,0,0). Note: If there is to be an initial platform offset, the initial platform displacement must be set in the FAST ElastoDyn input file as well.
   c. Under **Calculation**:
      i. Set **Primary Motion** to **Externally Calculated**.
      ii. Set **Superimposed Motion** to **None**.
      iii. Under **Included Effects**, check **Applied Loads**, **Wave Load (1st order)** and **Added Mass and Damping**. Other effects may be included if the WAMIT input file contains the necessary information. Note that the 2nd-order wave-drift loads are not included in the WAMIT input files we provide, and the **Applied Loads** value will be defined in a later section.
   d. Under the **Primary Motion** tab:
      i. Set **Externally calculated primary motion** to **ExtFn**.
5. Change the properties of the vessel by opening **Vessel Type1** from the model browser.
   a. In the bottom of the screen, press **Import**. Locate your WAMIT *.out file, and press open.
   b. Under **Structure**, set **Mass and Moment of Inertia Tensor** to a small non-zero values (because these properties are accounted for in FAST).
   c. Optional: At this stage, we suggest changing the vessel geometry to reflect an appearance representative of the system being modeled. This will not influence the simulation, but it will help with visualizing the vessel motions and mooring/vessel connections after the simulation is completed. To do so, go to the **Drawing** tab and add/remove lines and vertices of the vessel.
6. In the model browser, go to **Environment**.
   a. Under **Seabed** change the water depth.
   b. If desired, the wave properties can be given under **Waves**.

7. The mooring can now be presented in the model.
   a. Click on the **New Line** icon on the toolbar and add a mooring line to the model. In the model browser, open **Line1**.
   b. Under **Connect to Object**: Fix one end to **Platform** and one end to **Anchored**.
   c. Under **Object Relative Position**, set the desired positions of the ends of the mooring cable.
   d. If needed, change the **Length** and **Target Segment Length** under **Structure**.
   e. Repeat 7 a-d for each mooring line.
8. Change the properties and attributes of the mooring by opening **Line Type1** in the Model Browser.
9. The next step in the assembly of the model is to include bodies that account for the quadratic drag force the platform.
   a. Click on **New 6D Buoy** on the toolbar and add the buoy to the model. In the model browser, open **6D Buoy1**.
   b. Click on **Give Buoy Negligible Properties** in the bottom of the window.
   c. Set **Type** to **Spar Buoy**.
   d. Set **Connection** to **Platform**.
   e. Set **Initial Position and Attitude** of the origin of the element.
   f. Under **Geometry**:
      i. Discretize the body into smaller sub-bodies of desired length by increasing the number of **Cylinders**. This is done to ensure sufficient resolution of the drag along the body. Note: Remember to extend the body above the MWL.
      ii. All cylinders must have the **Outer Diameter** set to a small non-zero value. This is done to make sure there is no extra submerged volume and hence no extra hydrostatic stiffness, because this is already accounted for under **Vessel Type1**.
   g. Under **Drag and Slam**:
      i. Define the **Areas** and **Coefficients** under **Drag Forces**.
   h. Repeat 9 a-g if multiple drag members are desired.
10. The final step needed is to prescribe an **Applied Load** on the Platform. This step is crucial to ensure the simulation begins at equilibrium.
    a. Define the total mass of your system. This can be found in the **.ED.sum* file on line 64 under **Mass Incl. Platform**.
    b. Multiply the mass by *-Gravity* to get the total force in Z (here named *FzFAST*). Note: *Gravity* is by default 9.80665 m/s^2 in OrcaFlex and set within the FAST ElastoDyn input file.
    c. In OrcaFlex, run **Single Statics (F9)**, open **Select Results (F5)**. Select **Summary Results** and double click **Platform**. Under **Loads in Global Axes Directions**, the forces in **Z** are summed up. The **Total** force has to be equal and opposite to *FzFAST*. If this is not the case:
       i. Open the **Platform** from the model browser.
       ii. Go to the **Applied Loads** tab.
       iii. Set **Global Loads** to **1**.

iv. Under **Applied Force (relative to global axes)** put in the force correction under **Z**.

v. Close the window and repeat 10c to verify equilibrium.

Provided Examples

Three different example models (all using the NREL 5-MW turbine) and eight different test cases are provided with the FASTlink archive, as summarized in Table 2.

Table 2: Content of the Example Models Provided with the FASTlink Archive

Name	Description
MIT-NREL_TLP	MIT/NREL tension-leg platform (TLP) exposed to stochastic wind and waves
OC3_Spar	OC3-Hywind spar exposed to stochastic wind and waves
OC4_LC_1.3a	Free surge decay of the OC4-DeepCwind semisubmersible
OC4_LC_1.3b	Free heave decay of the OC4-DeepCwind semisubmersible
OC4_LC_2.1	OC4-DeepCwind semisubmersible exposed to deterministic waves
OC4_LC_2.2	OC4-DeepCwind semi-submersible exposed to stochastic waves
OC4_LC_3.1	OC4-DeepCwind semisubmersible exposed to deterministic wind and waves
OC4_LC_3.2	OC4-DeepCwind semisubmersible exposed to stochastic wind and waves

Simulate

The simulation is started from FAST and FAST is run normally. E.g., for a 64-bit simulation, open a Windows command prompt to a folder where all the simulation files are housed and type:

FAST_x64.exe NRELOffshrBsline5MW_Floating_OC3Hywind.fst

The simulation will run with FAST coupled to OrcaFlex. Once it is completed, the FAST-related outputs are written to the normal FAST output files and the OrcaFlex-related outputs are written to the OrcaFlex simulation file, *<OrcaIptFileName>.sim*, accessible through OrcaFlex.

Appendix A: OrcaFlexInterface Module Input File

This example input file is for the OC3-Hywind spar offshore turbine. Another set of example input files using the MIT/NREL TLP and the OC4-DeepCwind semisubmersible are also available.

```
------- OrcaFlex Interface v1.00.* INPUT FILE ---------------------------------
Test for OrcaFlex interface to FAST v8
--------------------- SIMULATION DATA ------------------------------------
   False          Echo             - Echo input data to "<RootName>.ech" (flag)
   NRELOffshrBsline5MW_Floating_OC3Hywind_Orca         OrcaIptFileName  - name of OrcaFlex input file without the .dat
file extension (relative, or including the full path)
   FASTLinkDLL.dll   DLL_FileName      - name of dll to use for OrcaFlex including the .dll file extension (relative, or
including the full path)
```