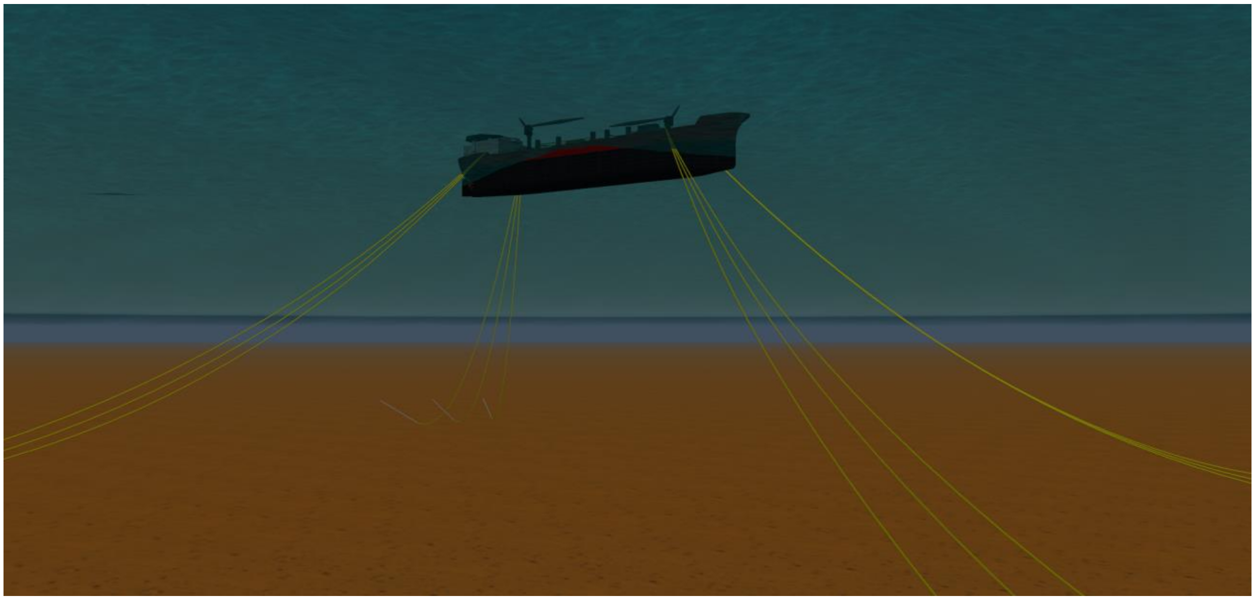
C10 Multiple statics



Introduction

In OrcaFlex version 10.0, a feature called 'Multiple Statics' was removed. This was done because the automation tools now available with OrcaFlex allow the user to create the same set of actions, but with the flexibility to customise the parameters and output to suit their exact needs.

The *multiple statics* calculation performs a series of static analyses for a grid of different positions of a vessel, and is mainly used in mooring analyses. The output would typically be tables and graphs of load-offset curves. This example provides a Python script that replicates the multiple statics calculation that was previously built into OrcaFlex.

The calculation requires as input an OrcaFlex model, specified by file name. The model must contain at least one vessel object. In case multiple vessel objects are present, a single vessel must be identified. This vessel is then placed at a series of offsets about the assumed mean position. For each offset a static analysis is performed, and key results extracted. Finally, the results are output in an Excel workbook.

The Python script is supplied with a simple example to demonstrate its operation. The following files are provided:

File	**Description**
C10 Multiple statics.dat	Example OrcaFlex data file
MultipleStatics.bat	A command script to execute the Python script
MultipleStatics.py	The Python script
MultipleStatics.yml	Specification of the data file, the vessel, and the offsets

Requirements

In order to use the script you will need a suitable Python installation, see the [Python interface | Installation] of the API help file for guidance.

You will also need the OrcFxAPI module. This is installed as part of the OrcaFlex installation process, as long as Python is present at the time of installation. If not, you will need to install this module manually, as described in the above help link.

In addition, this particular script requires the following third party modules to be installed:

- `numpy`
- `xlsxwriter`
- `yaml`

On a standard Python installation this should be possible at the command line using pip:

```
C:\Users\ExampleUser> pip install numpy xlsxwriter pyyaml
```

Input data

The input data for a multiple statics analysis is specified in a YAML (human readable structured data) file. For example, *MultipleStatics.yml* contains the following data:

```
Model: C10 Multiple statics.dat
VesselName: Vessel1
AzimuthFrom: 0
AzimuthTo: 90
AzimuthStep: 15
OffsetFrom: 0
OffsetTo: 30
OffsetStep: 2
```

- *Model* specifies the OrcaFlex data file that defines the model to be analysed.
- *VesselName* specifies which vessel is to be analysed. If the model contains only a single vessel, this entry can be omitted.
- *AzimuthFrom*, *AzimuthTo* and *AzimuthStep* define the offset directions. In the example above the analysis is performed on azimuth directions 0°, 15°, 30°, 45°, 60°, 75° and 90°. These directions are relative to the global axis system. So, 0° is along the global X axis, 90° is along the global Y axis, and so on.
- *OffsetFrom*, *OffsetTo* and *OffsetStep* define the offset steps. In the example above the analysis is performed at offsets of 0, 2, 4, ..., 28 and 30. These values use the same length units as the OrcaFlex data file.

The YAML file can be edited using a text editor e.g. Notepad++.

Running the script

For convenience a .bat command script, *MultipleStatics.bat*, has been included. This can be double-clicked from a file explorer to run the script. For most purposes, this method of running the script will suffice. However, in case more control is required, the full command line options are now listed:

```
C:\Users\ExampleUser> python MultipleStatics.py --help
usage: MultipleStatics.py [-h] [-e] [spec]

OrcaFlex multiple static analysis.

positional arguments:
spec          name of the specification file

optional arguments:
-h, --help     show this help message and exit
-e, --echo     echo specification
```

The specification file name can be passed as an argument, e.g.

```
C:\Users\ExampleUser> python MultipleStatics.py example.yml
```

If the specification file name is omitted then the script file name (with the .py extension replaced by .yml) is assumed. The `--echo` option (or abbreviated to `-e`) can be used to echo the specification to the console:

```
C:\Users\ExampleUser> python MultipleStatics.py --echo
Model file name: example.dat
Vessel name: Vessel1
Azimuths (from, to, step): 0, 90, 15
Offsets (from, to, step): 0, 30, 2

Case 1/112: azimuth 0 offset 0
Case 2/112: azimuth 0 offset 2
Case 3/112: azimuth 0 offset 4
...
```

Results

The results are saved to an Excel workbook. This has the same file name stem as the OrcaFlex model data file, but with `.MultStat.xlsx` added. So, for the example input data, the output file name is *C10 Multiple statics.MultStat.xlsx*.

The workbook contains a sheet named *Input* that echoes the input specification:

	A	B	C	D
1	**Multiple statics input data**			
2				
3	Model file name	C10 Multiple Statics.dat		
4	Vessel name	Vessel1		
5	Azimuths (from, to, step)	0°, 90°, 15°		
6	Offsets (from, to, step)	0m, 30m, 2m		

The remaining sheets contain the results for each offset direction, for example:

	A	B	C	D	E	F	G
1	**Azimuth 60.0°**						
2							
3		**Restoring**	**Restoring**	**Vertical**	**Yaw**	**Worst**	**Worst tension**
4	**Offset (m)**	**force (kN)**	**direction (°)**	**force (kN)**	**moment (kN.m)**	**tension (kN)**	**line name**
5	0	5.53462E-10	0	5665.315093	-1.34532E-08	711.2847998	Line3
6	2	423.5340431	235.8574781	5677.003367	497.6134812	795.2143965	Line9
7	4	864.7786125	235.6368763	5712.567146	2033.650911	898.6060154	Line9
8	6	1342.476789	235.279081	5772.804399	4725.558378	1027.539683	Line9
9	8	1881.346299	234.7835984	5860.672884	8816.705432	1191.175308	Line9
10	10	2509.7781	234.1485426	5978.978863	14692.78965	1401.748002	Line9
11	12	3272.131029	233.4067879	6133.251107	22899.27562	1677.526089	Line9
12	14	4270.550442	232.5347893	6342.9994	35120.46546	2092.716443	Line9
13	16	5962.073857	231.3356337	6725.688628	59163.58704	3011.703655	Line9
14	18	9697.901622	229.6860691	7621.821408	117466.9203	5347.107149	Line9
15	20	16767.01292	228.0553922	9355.528134	229049.7791	9178.007506	Line9
16	22	26952.29026	226.7498376	11858.83756	386105.4498	13466.49535	Line9
17	24	38885.61671	225.9170839	14786.62472	567621.3571	17863.36454	Line9
18	26	51342.07541	225.4599335	17817.5925	756920.6804	22295.78967	Line9
19	28	63950.19861	225.2091897	20867.64931	949158.2087	26743.80564	Line9
20	30	76623.50618	225.0710112	23912.4489	1143079.822	31200.32689	Line9

The results variables are defined as follows:

Restoring force

The magnitude of the horizontal component of the total force applied to the vessel. Note that this force is not necessarily in the offset direction.

Restoring direction

The direction of the restoring force.

Vertical force

The vertically downwards component of the total force applied to the vessel.

Yaw moment

The total moment, about the vertical, applied to the vessel.

Worst tension and *Worst tension line name*

The largest tension in any segment of any line connected to the vessel, and the name of the line in which that tension occurs.

The workbook also generates plots of these results, for each offset direction. The graphs below show the results for the 60° offset direction for the spread moored vessel considered for this example: