Memo

To
Bo Terp Paulsen

Date
15 October 2015

Number of pages
4

From
Niels Jacobsen

Direct line
+31(0)88335 8359

E-mail
niels.jacobsen@deltares.nl

Subject
Coupling between OceanWave3D and waves2Foam

Dear Bo,

This memo is to inform you that I have successfully coupled OceanWave3D ([1]) in a streamlined fashion into waves2Foam ([2]). I had access to the original coupling ([3]), but the code has now been improved, such that it works for all supported versions of OpenFoam and with and without permeable layers. No special attention needs to be given to the coupling, if waves2Foam should be used in a new solver for e.g. geotechnical simulations or evaluation of the hydraulic loading on floating structures.

This memo will shortly describe the functionality and for now the memo will also serve as the documentation.

Kind regards

Niels

Purpose of the coupling

The purpose of the coupling is to be able to provide a fully nonlinear wave kinematics in the CFD-model to improve on the evaluation of the hydraulic loading on coastal, offshore and maritime structures. OceanWave3D solves the Laplace problem for propagating waves together with the fully nonlinear free surface boundary conditions. The high-order discretisation of the numerical model means that it is possible to transform waves over a long distance with only limited numerical errors ([1]), which makes OceanWave3D suitable for the transformation of a linear super-imposed irregular wave field from deep/intermediate to shallower water.

The benefit is that while there is no stable perturbation theory beyond second order ([4]) for irregular waves, the numerical model can provide a fully nonlinear irregular wave solution over an arbitrary bathymetry. This numerical solution is similar to the stream function theory for regular waves over a flat bottom.

Functionality

The implementation is such that the expensive CFD-calculation (OpenFoam) does not have to begin at the same time as the less expensive potential flow solver (OceanWave3D). This means that a starting time for OpenFoam larger than 0 s can be specified and OceanWave3D will perform time steps until this starting time is reached. As this point in time, the solution from OceanWave3D is mapped to OpenFoam and the two solvers will run side-by-side.

The coupling is built into the externalWaveForcing class in waves2Foam and the implementation does support parallel execution. There will be started one instance of OceanWave3D for each processor used by OpenFoam.

It is likewise possible to use the results from OceanWave3D in multiple relaxation zones, but this does not increase the number of instances of OceanWave3D.

Note that the vertical direction in OceanWave3D is always z, but this does not restrict OpenFoam, where e.g. y can still be used as a vertical axis. This possible discrepancy is handled inside the coupling on the waves2Foam-side.

Dependencies and source files

OceanWave3D depends on Lapack and SPARSKIT2. Furthermore, the installation procedure depends on git for the access of OceanWave3D from the git-repository. The dependencies Lapack and SPARSKIT2 will be compiled as part of the compilation procedure for waves2Foam.

The coupling is placed in the source files oceanWave3D.[C,H] in source code for waves2Foam. There is also a set of files related to the pre-processing, such that the utility setWaveParameters recognises this new “wave theory”.

Control – OceanWave3D

The control of OceanWave3D will not be described in this memo, but there are example scripts available in the git-repository. These example scripts are downloaded as part of the installation process. The coupling only works, if the input file ‘OceanWave3D.inp’ is available in the root of the OpenFoam case.

There is being worked on a GUI for OceanWave3D and this is planned to be made available open-source. This will ease the set-up of the OceanWave3D input files.

Control – waves2Foam

The following has to be added to waveProperties.input in order to activate the coupling between OceanWave3D and waves2Foam:

```
// External forcing definition
externalForcing oceanWave3D;

// Coefficients needed for the external forcing
externalForcingCoeffs
{
    waveType oceanWave3D;

    // Define the intervals for the OpenFoam calculations
```

```
    nIntervals 1;
    startTimes nonuniform List<scalar> 1(5);
    endTimes nonuniform List<scalar> 1(20);

    // Should the interval be ramped?
    rampInterval off;

    // Name of the sub-dictionary (without Coeffs), where the externalSource
    // definition is given. Is needed, when the mapping OceanWave3D to penFoam
    // is performed.
    mappingZone inlet;
};
```

Furthermore, any relaxation zone or boundary condition requires the specification of the following wave type:

```
<name>Coeffs
{
    waveType externalSource;

    // Remaining definition of e.g. relaxation zones
}
```

It is possible to define multiple intervals for the OpenFoam simulation (see the text box), but this option is currently de-activated, because the restart on a second interval was not always stable. Consequently, it is recommended to perform multiple simulations in side-by-side, if more intervals are required.

Finally, note that the beginning of the computational domain in OceanWave3D is always at $x = 0$ m and the waves propagate from left to right. The mapping from OceanWave3D to OpenFoam requires that the computational domain for OpenFoam is within the horizontal plane of the OceanWave3D domain; preferably by at least 3-4 cell sizes. The computational domain for OpenFoam may not extend below the bottom of the computational domain for OceanWave3D.

Suggested referencing

This coupling brings together a number of independent works and all deserve proper credit, thus a formulation similar to the following is suggested:

"The waves are computed with the fully nonlinear potential wave solver OceanWave3D ([1]) and these are coupled to OpenFoam/waves2Foam ([2]) through the interface described in [3]."

References

[1]: Engsig-Karup, A.P., Bingham, H.B. and Lindberg, O. (2009). An efficient flexible-order

model for 3D nonlinear water waves. *Journal of Computational Physics*, **228**, 2100-2118.

[2]: Jacobsen, N.G., Fuhrman, D.R. and Fredsøe, J. (2012), A wave generation toolbox for the open-source CFD library: OpenFoam®. *International Journal for Numerical Methods in Fluids*, **70**(9), 1073-1088.

[3]: Paulsen, B.T., Bredmose, H. and Bingham, H.B. (2014). An efficient domain decomposition strategy for wave loads on surface piercing circular cylinders. *Coastal Engineering*, **86**, 57-76.

[4]: Madsen, P.A. and Fuhrman, D.R. (2012). Third-order theory for multi-directional irregular waves. *Journal of Fluid Mechanics*, **698**, 304-334.