TUTORIALS

for SHIPFLOW MOTIONS

Case 1 – Setting up a new configuration for calm water

This is a minimum case with the KCS hull is sailing in calm water. The only required input from the user is parameters for the geometry, speed and reference length.

```
xflow
  titl( titl="Case 1" )
  prog( xmtd, xptd )
  hull( mono, fsflow, td, fine )
  offs( iges="kcs_g2010.igs", zori=10.8, lpp=230 )
  vshi( vknot = [24], reflen = 230 )
end
xpan
  para( nthr = 6 )
end
```

The the program will run until convergence of the resistance.

In the OUTPUT file we can find relevant data for this kind of simulation, such as sinkage, trim and wave resistance.

Sinkage and trim, as it is commonly called in a calm water context, is in a simulation like this the same thing as “heave” and “pitch” in the converged state.

Hints for the GUI

Case 1. Calm water. Minimum input

1. Import the IGES file
2. Select all patches and create a Surface Group
3. Import the configuration file case1
4. Check that the commands
5. Export the surface group grp1 in the XFLOW:OFFSET:IGES command:
   grp1.exportIGES_deprecated("kcs.igs")
6. Adjust the number of threads to your computer
7. Save the project
8. Run the case1

Case 2 – Calm water with user defined mass properties and initial position of the hull

Simulations in MOTIONS are dynamic and the program requires the mass properties of the hull, i.e. mass, center of gravity and radius of gyration. In Case 1 we did not specify any of this so the program did it for us.

The mass and longitudinal center of gravity was computed based on the hydrostatics of the initial floating condition but the vertical center of gravity and pitch radius of gyration can not be computed from statics and was therefore approximated by setting it to ZCG = 0.0 and KYY = 0.25*LPP.

However, note that since we are simulating a hull sailing in calm water we expect it to eventually reach a steady state where it is not moving and therefore the radius of gyration will not be so important for the end result.

In this case we add two commands to the command file,as follows:

```
xflow
  titl( titl="Case 2" )
  prog( xmtd, xptd )
  hull( mono, fsflow, td, fine )
  offs( iges="kcs_g2010.igs", zori=10.8, lpp=230 )
  vshi( vknot = [24], reflen = 230 )
  mass( xcg = 0.5148, zcg = -0.0153, kyy = 0.25, mass = 52030000 )
  fixe( surge )
  ipos( surge = 0.0, heave = 0.0, pitch = 0.0 )
end
xpan
  para( nthr = 6 )
end
```

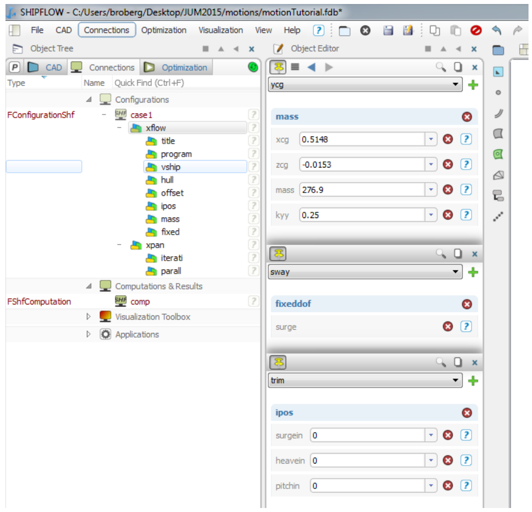
When manually specifying mass properties, such as mass and center of gravity, it is important to be careful that the mass properties corresponds to the desired floating condition, e.g. design or ballast condition.

For example, if the given mass corresponds to ballast loading condition and the initial position corresponds to design loading condition, the hull will be too light for that draft and rather quickly float upwards when the simulation begins. This may result in a very strong transient which will make the simulation take longer time before converging. In the worst case the effect will be so strong that the solution breaks down and the simulation stops.

Hints for the GUI

Case 2. Calm water. Mass and motion properties

- Create a new design and rename it to case2
- Add the additional commands to specify mass properties, fix the surge and initial position



- Run case2

Case 3 – Ship in regular waves

In cases 1 and 2 we have simulated the ship sailing in calm water. However, SHIPFLOW MOTIONS is especially suited for analysis of ships sailing in waves. One of the more common types of problem is the computation of response amplitude operators for various wave lengths. Also added resistance due to waves is typically of interest.

In order to accomplish this we need to tell the program to generate and use regular incoming waves rather than a calm free surface. This is done by modifying the command file slightly:

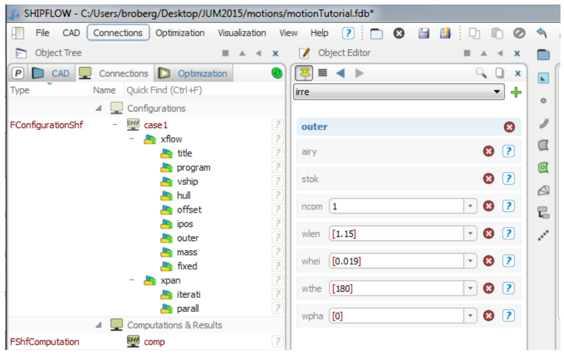
```
xflow
  titl( titl="Case 3" )
  prog( xmtd, xptd )
  hull( mono, fsflow, td, fine )
  offs( iges="kcs_g2010.igs", zori=10.8, lpp=230 )
  vshi( vknot = [24], reflen = 230 )
  mass( xcg = 0.5148, zcg = -0.0153, kyy = 0.25, mass = 52030000 )
  fixe( surge )
  oute( stokes, ncom=1, wlen=[1.15], whei=[0.019], wthe=[180], wpha=[0] )
end
xpan
  para( nthr = 6 )
end
```

The program will now use one (`ncom=1`) regular long-crested wave based on 5th-order Stokes' wave theory (`stokes`) with a wave length equal to 115% of LPP (`wlen=[1.15]`) and a wave height equal to 1.9% of LPP (`whei=[0.019]`). The phase of the wave is 0 degrees (`wpha=[0]`), meaning that a wave peak coincides with LPP/2 at time zero. The wave direction is 180 degrees (`wthe=[180]`) meaning that the wave propagates in the opposite direction of the ship heading. The simulation will continue until convergence. Normally 10-15 wave encounters are required.

Hints for the GUI

Case 3. Regular waves

- Create a new design from case2
- Add the commands for the external waves



- Run case3

Case 4 – Ship in irregular waves

Sometimes it is of interest to simulate the ship sailing in irregular sea. SHIPFLOW MOTIONS can generate irregular waves by means of superposition of wave components defined by an ITTC-spectrum.

This is done by telling the program to use irregular waves and choosing either a pre-defined sea state, e.g. SS4, or by defining significat wave height Hs and zero-crossing period Tz.

We will change the command file slightly in order to simulate with sea state 4 instead of a regular wave.

```
xflow
  titl( titl="Case 2" )
  prog( xmtd, xptd )
  hull( mono, fsflow, td, fine )
  offs( iges="kcs_g2010.igs", zori=10.8, lpp=230 )
  vshi( vknot = [24], reflen = 230 )
  mass( xcg = 0.5148, zcg = -0.0153, kyy = 0.25, mass = 52030000 )
  fixe( surge )
  oute( irreg, ss = 4, wthe=[180] )
end
xpan
  para( nthr = 6 )
end
```

Results and output from the program

During a simulation, the program saves time series of all data to an ASCII-file called TSERIES. When a simulation is done, the program computes a number of secondary data such as harmonic amplitudes and phases of the motions and resistance as well as response amplitude operators, RAOs, and write them to the OUTPUT file.

The generated report contains data similar to the output file in form of tables. A video from the simulation is also created. The remaining figures in the report are similar to the XPAN results and shows data from the last time step. Thus it is the stationary solution in calm water, but just a snap shot for the cases with ambient waves.

The time steps are treated as iterations in the GUI and can be accessed one by one for visualization. The time series of forces, velocities and motions can be displayed. Results from the output file are imported in to a table of results.