CAESIM Software

Demonstration Report

Wave Impact Modeling

CONTENTS

1.0 Introduction

The purpose of this report is to demonstrate the capability of the CAESIM software system to solve transient wave impact applications, specifically on moving floats. A wave force impact analysis has also been conducted for 3 CFD simulations presented.

The developed CFD models utilized the body-fitted coordinate system to accurately model the geometry of the application.
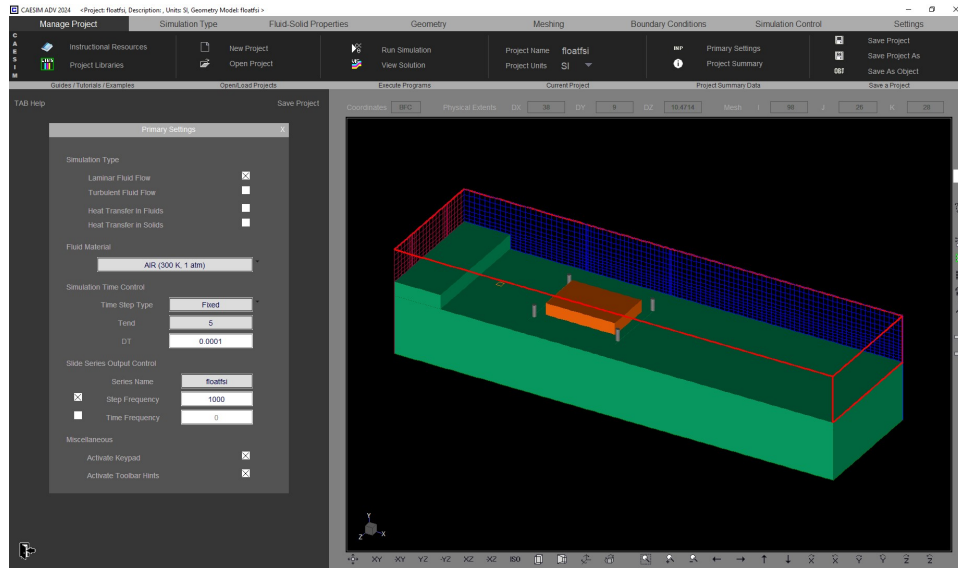
Screen shot of the CFD model "floatc"

The CFD models utilize the free-surface module, and the fluid-structure interaction (FSI) module to accomplish modeling the physics of the application. The float motion has been limited to the vertical (Y) direction for the CFD simulations presented in this report.
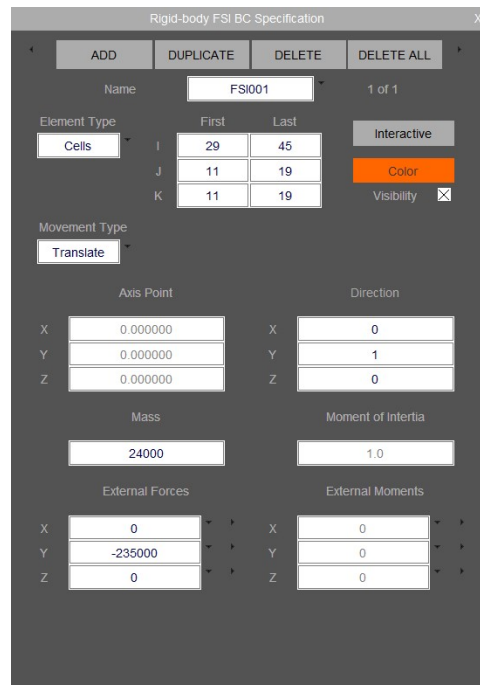
2.0 CFD Model #1 – Free Surface Only

The first CFD model was executed without the fluid-structure interaction module activated. The purpose of the first CFD simulation is to demostrate the transient wave behavior as it impacts a stationary float. The definition of the CFD model is as follows:

Float dimensions: 6m x 4m x 2m

Mesh distribution: 68 I x 40 J x 48 K

5 blockage BCs defining float and pylons
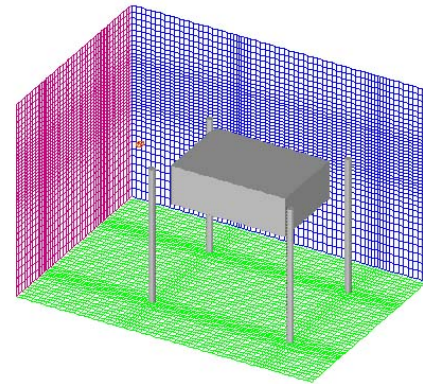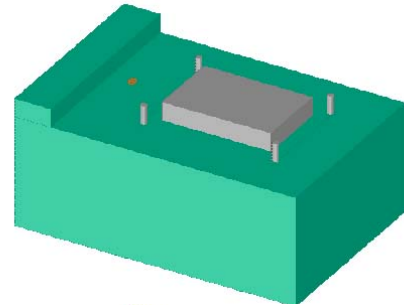
1 gravity BC for domain

Initialization field defined to generate initial wave

Fluid I set to air (rho=1.177 kg/m^3)

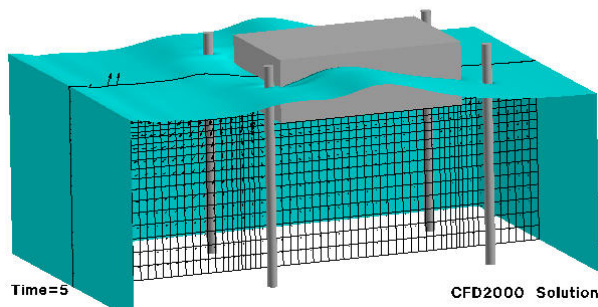Fluid II set to water (rho=996.56 kg/m^3)

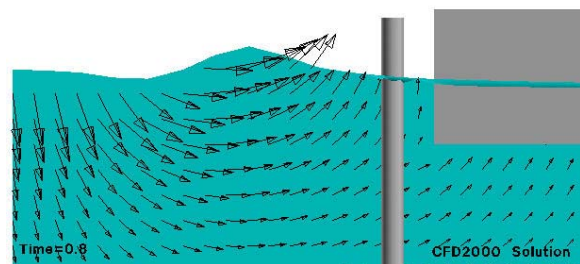Simulation timestep: 0.0001 seconds

Simulation time: 10 seconds

Simulation Results

The simulation results show a generated wave of about 0.6m in height.

3.0 CFD Model #2

The second CFD model was executed with the fluid-structure interaction module activated. The purpose of the second CFD simulation is to demostrate the transient wave behavior as it impacts the float, and observe the vertical mostion due to float interacting with the wave motion. The definition of the CFD model is as follows:

Float dimensions: 6m x 4m x 2m

Mesh distribution: 42 I x 26 J x 28 K

5 blockage BCs defining float and pylons

1 gravity BC for domain

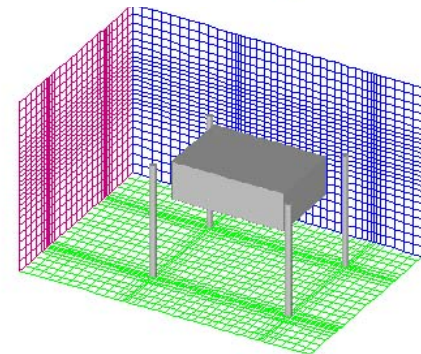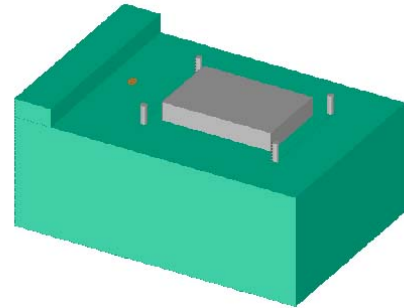Initialization field defined to generate initial wave

Fluid I set to air (rho=1.177 kg/m^3)

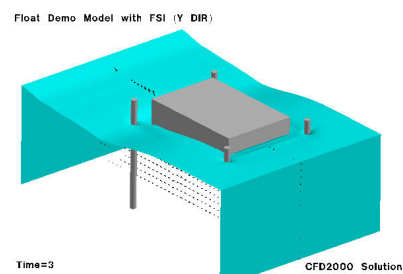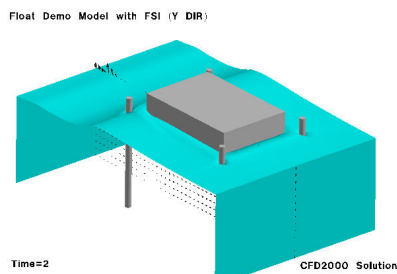Fluid II set to water (rho=996.56 kg/m^3)
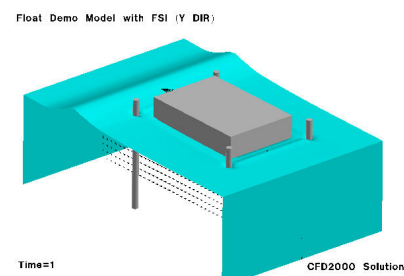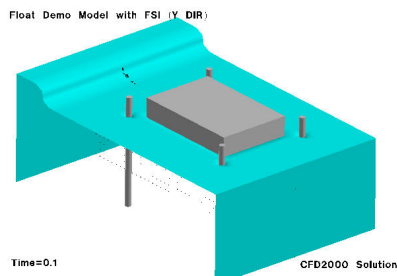
Mass of float set to 24000 kg

Simulation timestep: 0.0001 seconds
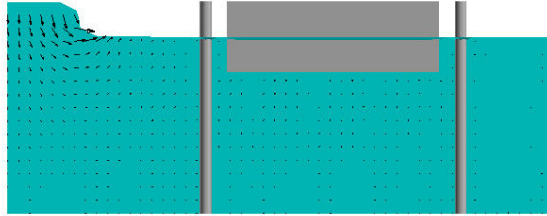
Simulation time: 10 seconds

Simulation Results

The figures below show an ISO view of 4 transient times (0.1, 1.0, 2.0, and 3.0 seconds respectively) during the simulation.

The figures below show a side view of 4 transient times (0.1, 1.0, 2.0, and 3.0 seconds respectively) during the simulation.

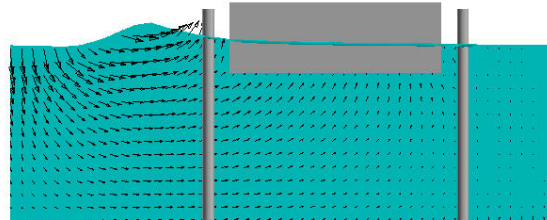Float Demo Model with FSI (Y DIR)

Time=0.1   CFD2000 Solution

Float Demo Model with FSI (Y DIR)
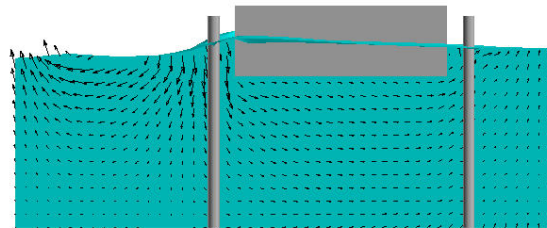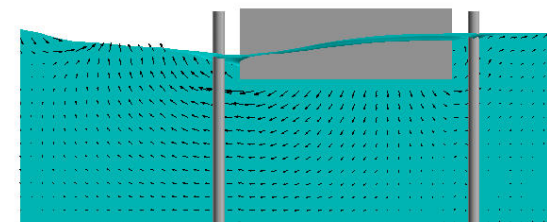

Time=1   CFD2000 Solution

Float Demo Model with FSI (Y DIR)

Time=2   CFD2000 Solution
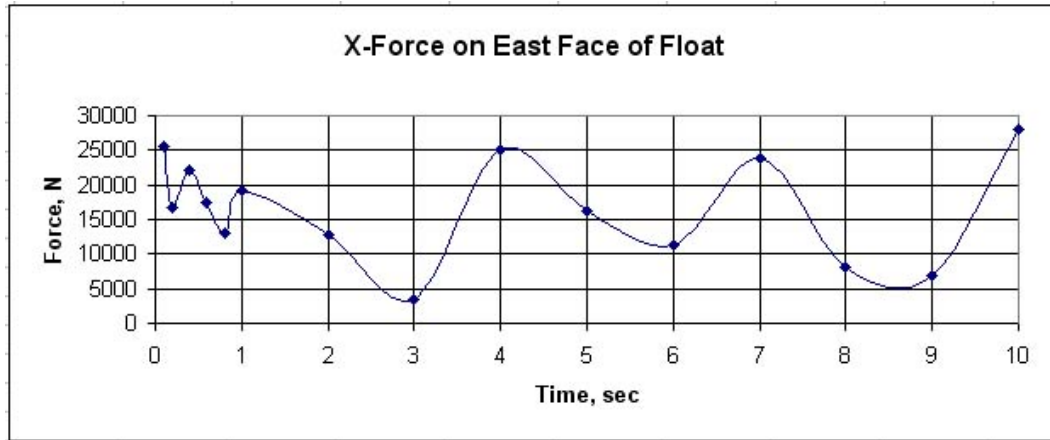
Float Demo Model with FSI (Y DIR)

Time=3   CFD2000 Solution

The second CFD model also had force computation activated, whereby a force data acting on all sides of solids within the computational domain is reported. Force information was extracted for the east/wave side of the float and is shown in the following graph.

To verify the reported forces, pressure acting on the east face of the float was analyzed (see figures below).
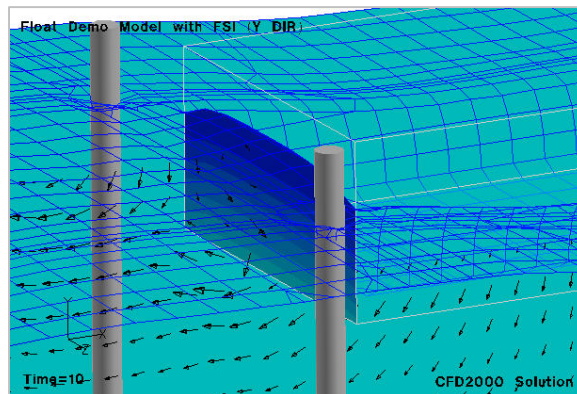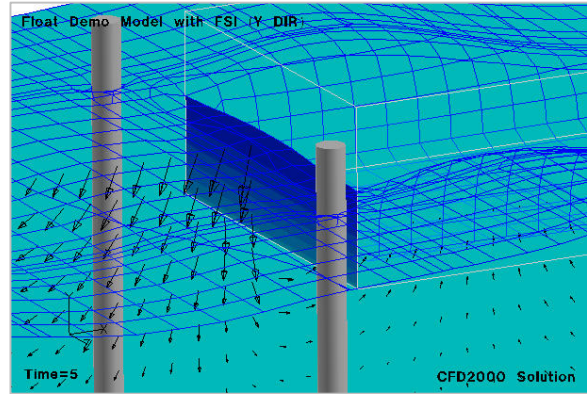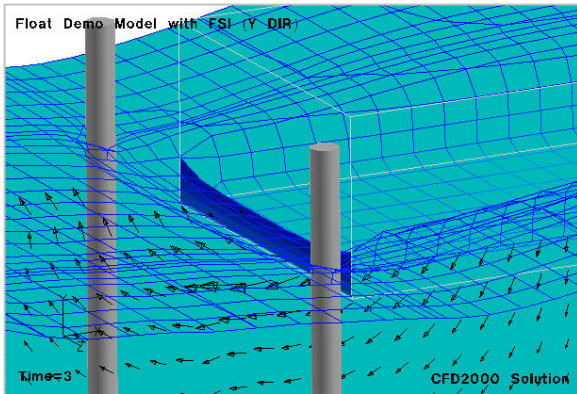
4.0 CFD Model #3

The third CFD model was also executed with the fluid-structure interaction module activated. The third CFD model's physical domain was extended before and after the float, and the initial wave condition was modified to achieve a wave height of ~0.8m and period of ~6m. The definition of the CFD model is as follows:

Float dimensions: 6m x 4m x 2m

Mesh distribution: 66 I x 26 J x 28 K

5 blockage BCs defining float and pylons

1 gravity BC for domain

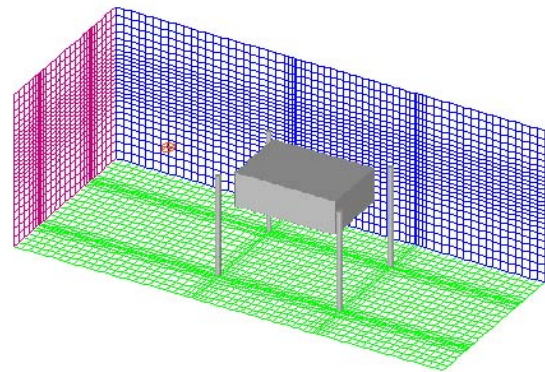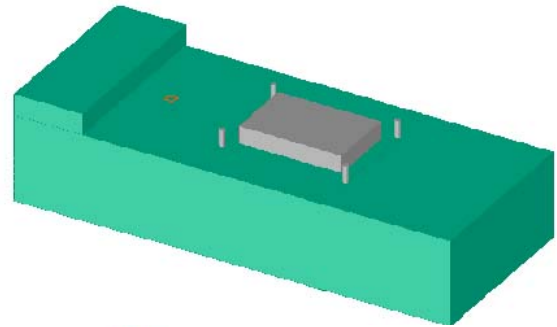Initialization field defined to generate initial wave

Fluid I set to air (rho=1.177 kg/m^3)

Fluid II set to water (rho=996.56 kg/m^3)

Mass of float set to 24000 kg
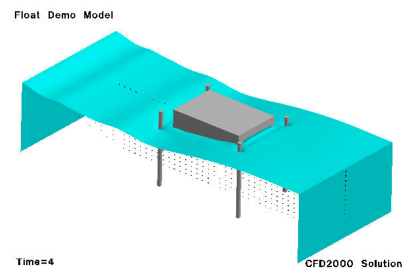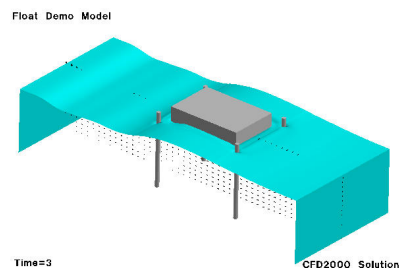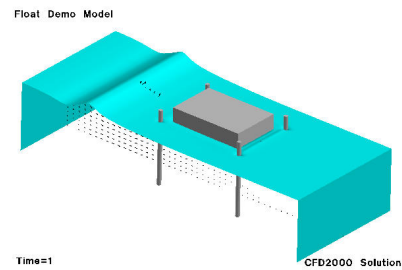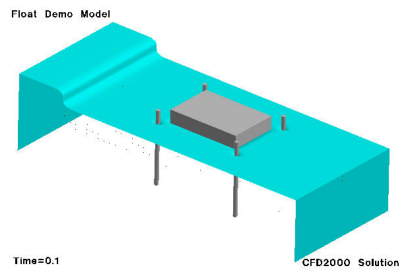
Simulation timestep: 0.0001 seconds
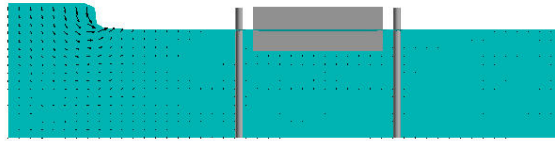
Simulation time: 5 seconds

Simulation Results

The figures below show an ISO view of 4 transient times (0.1, 1.0, 3.0, and 4.0 seconds respectively) during the simulation.

The figures below show a side view of 4 transient times (0.1, 1.0, 3.0, and 4.0 seconds respectively) during the simulation.

Float Demo Model

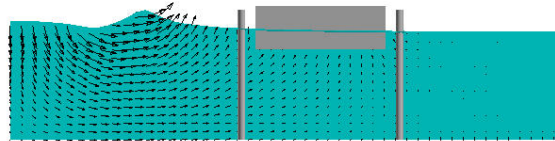Time=0.1   CFD2000 Solution

Float Demo Model

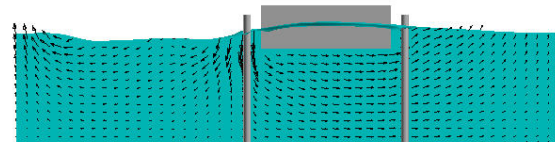Time=1   CFD2000 Solution

Float Demo Model

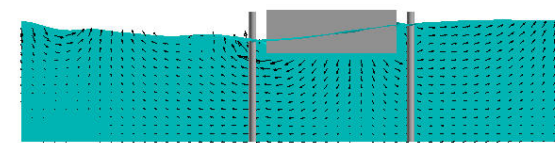Time=3   CFD2000 Solution
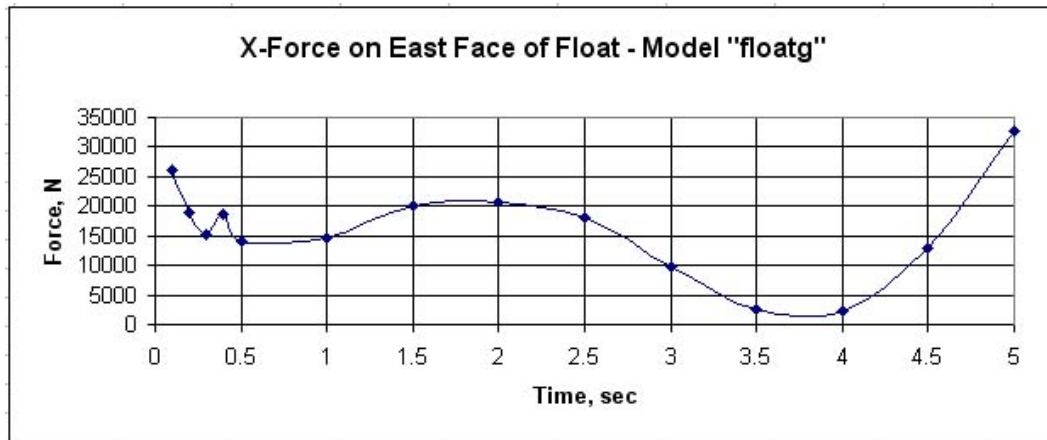
Float Demo Model

Time=4   CFD2000 Solution

The third CFD model also had force computation activated, whereby a force data acting on all sides of solids within the computational domain is reported. Force information was extracted for the east/wave side of the float and is shown in the following graph.

5.0 CFD Model #4

The fourth CFD model was also executed with the fluid-structure interaction module activated. The fourth CFD model's physical domain was extended after the float, and an outlet was added to minimize wave reflection. The definition of the CFD model is as follows:

Float dimensions: 6m x 4m x 2m

Mesh distribution: 98 I x 26 J x 28 K

5 blockage BCs defining float and pylons

1 gravity BC for domain

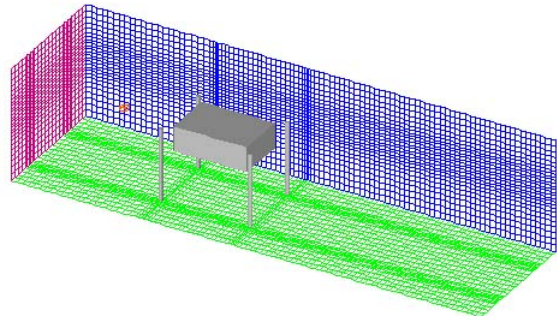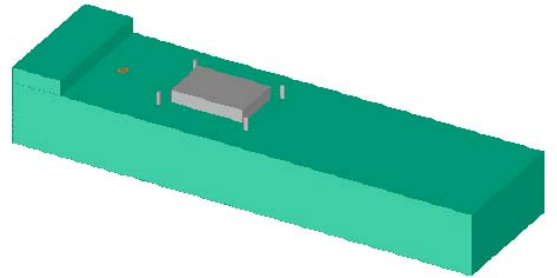Initialization field defined to generate initial wave

Fluid I set to air (rho=1.177 kg/m^3)

Fluid II set to water (rho=996.56 kg/m^3)
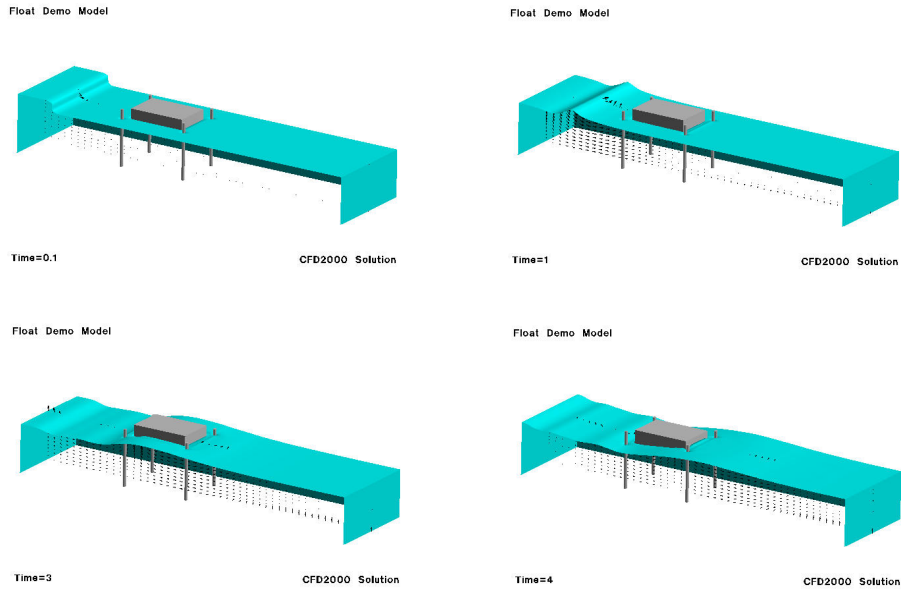
Mass of float set to 24000 kg

Simulation timestep: 0.0001 seconds
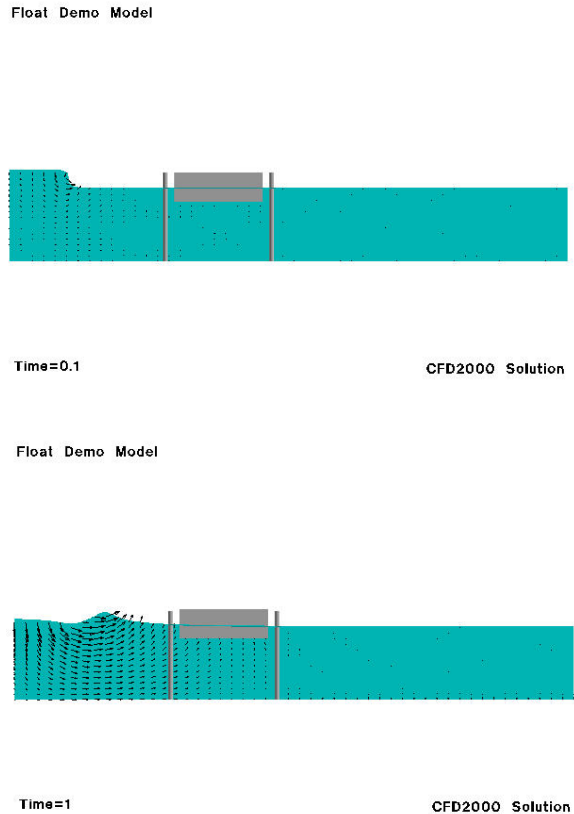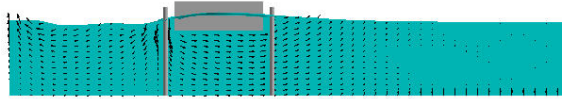
Simulation time: 5 seconds

Simulation Results

The figures below show an ISO view of 4 transient times (0.1, 1.0, 3.0, and 4.0 seconds respectively) during the simulation.

The figures below show a side view of 4 transient times (0.1, 1.0, 3.0, and 4.0 seconds respectively) during the simulation.

Float Demo Model

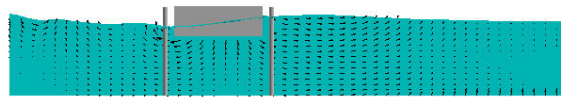Time=3   CFD2000 Solution
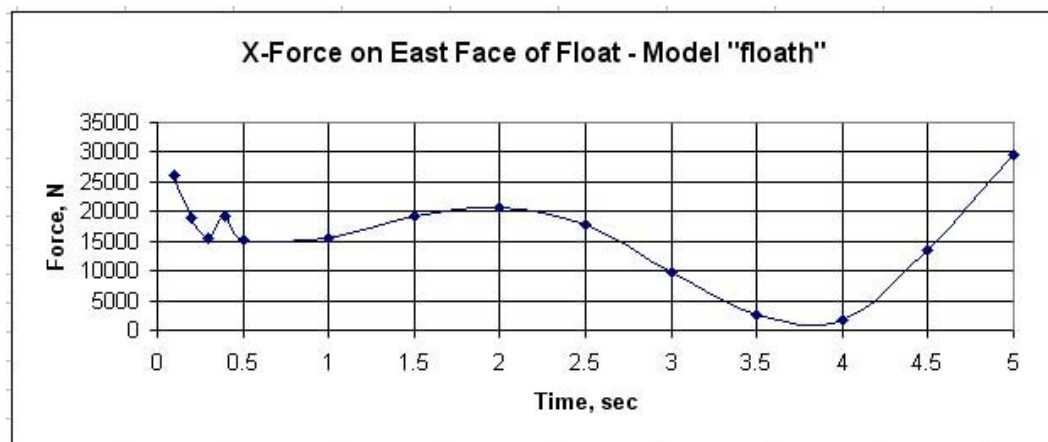
Float Demo Model

Time=4   CFD2000 Solution

The fourth CFD model also had force computation activated, whereby force data acting on all sides of solids within the computational domain is reported. Force information was extracted for the east/wave side of the float and is shown in the following graph.

6.0 Summary

Four CFD models were developed and executed. Three models implemented an FSI boundary condition for the defined float (to allow for vertical movement). The final two CFD models implemented a tranisent wave condition of ~0.8 m wave height and ~6.0 m wave period.

Float Demo Model — floatg
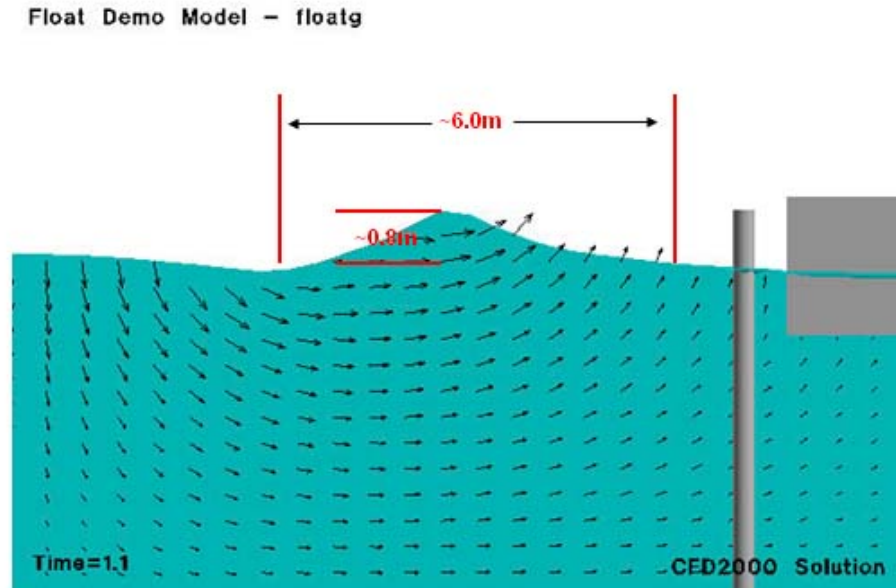
Transient force computations were reported for all of the FSI CFD models. The range is comparable for all cases, reporting a maximum force of ~30KN. The oscillatory nature of the temporal force curves is due to the "bounding" defintions of the computational domains, and the up/down movement of the float.

CAESIM was able to model this type of free-surface CFD application without any customization.