



Practical Session with DualSPHysics code and DesignSPHysics



Objectives: to understand some of the basic behaviour of the SPH-based mesh-less *DualSPHysics* code.

You will be using the open-source SPH code called DualSPHysics (www.dual.sphysics.org) to study fluid flows following various simulations, including *dam break*, *wave generation and absorption*, *wave-structure interaction*, *periodic boundary conditions*, *sloshing tanks and fluid-driven objects*. The course is based on the use of the new Graphical User Interface called DesignSPHysics (www.design.sphysics.org). FreeCAD is chosen as the host 3D modelling software for the plug-in due to its multi-platform capability. The implementation was carried out using Python as the default scripting language and QT as its GUI framework.

Full package of the code available at <http://dual.sphysics.org/index.php/downloads/>

List of references at <http://dual.sphysics.org/index.php/references/>

FIRST:

1. Install Paraview v5.5.0 <http://www.paraview.org/>
2. Install Notepad++ <http://notepad-plus-plus.org/>
3. Install FreeCad v0.17 <https://www.freecadweb.org/>
4. Download the course from http://dual.sphysics.org/sphcourse/CPU_Labima_2018/

User: training

Password: pomodoro

- **DesignSPHysics_September2018.zip:**
package of the GUI application and executables of DualSPHysics package v4.2
- **Material_September2018.zip:**
material needed for the course

CASES OF EXAMPLE:

CaseDambreak3D: 3-D dam break flow impacting on a structure (STL)

POST: plot particles, surface, inflow

CaseDambreak2D: 2-D dam break created starting from CaseDambreak3D

POST: plot particles

CaseFloatingSphere: 2-D sinking floating sphere with density half the water density

POST: plot particles, compute center.z position and buoyant force of sphere

CaseWaves: 2-D regular waves with piston and comparison with 2nd order wave theory

POST: plot particles, slice, compute surface elevation and orbital velocities

CaseWavesWall: 2-D regular waves with piston impacting on vertical wall without AWAS

POST: plot particles, compute force against wall

CaseWavesWallAWAS: 2-D regular waves with piston impacting on vertical wall with AWAS

POST: plot particles, compute force against wall

CaseWavesFloating: 2-D floating box under the action of regular waves validated with experiment

POST: plot particles, compute box motions and rotations

CaseWaveTankFile: 2-D tank with piston motion from file and impacting on a structure (STL)

POST: plot particles, compute surface elevation and force against structure

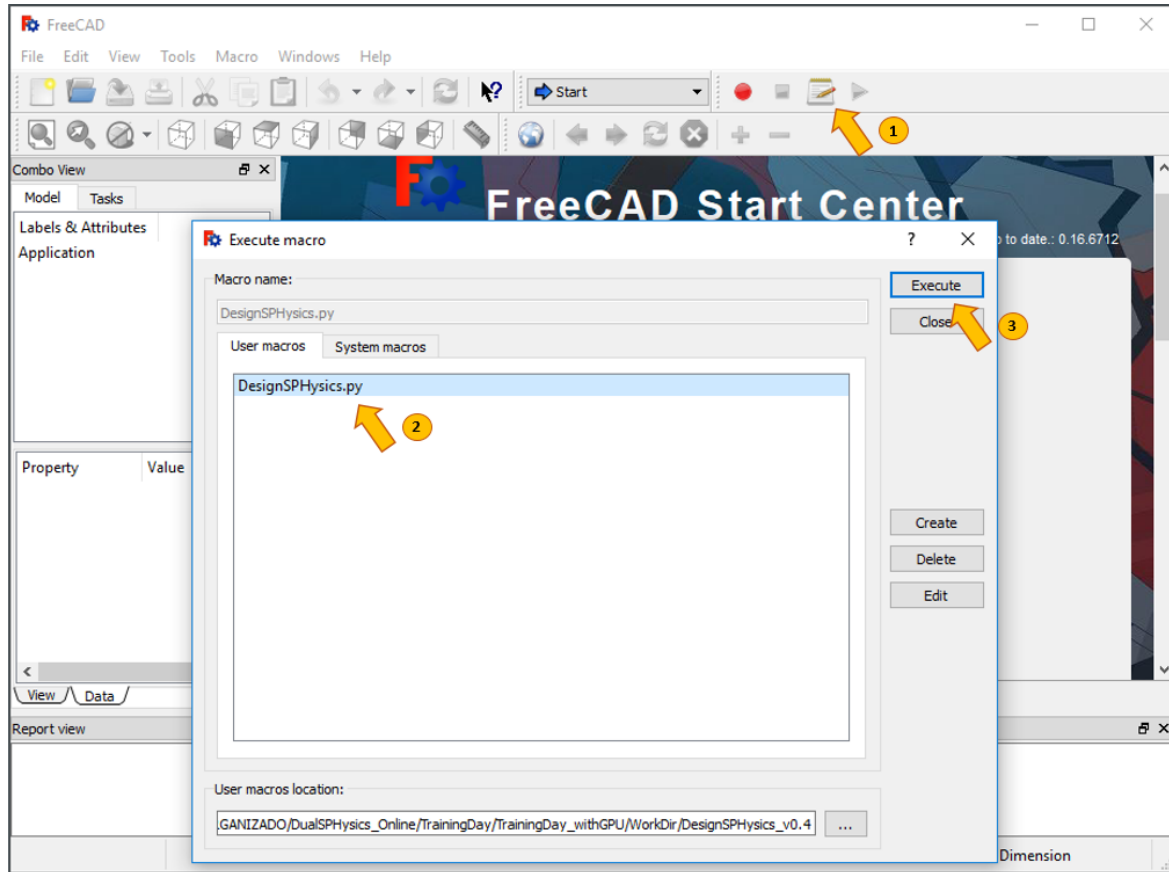
CaseSloshingTank: 2-D sloshing tank with sinusoidal rotational movement

POST: plot particles

Name	dp (m)	Number of particles	Physical time (s)	Output files	Runtime (min) <i>Intel 3 GHz (2 threads)</i>	Runtime (min) <i>GeForce GTX 1080 Ti</i>
CaseDambreak3D	0.015	34,236	0.5	100	5.9	0.1
CaseDambreak2D	0.005	5,281	2.0	200	3.6	0.7
CaseFloatingSphere	0.025	4,941	6.0	300	7.8	2.0
CaseWaves	0.015	8,417	8.0	320	13.6	2.8
CaseWavesWall	0.015	3,741	10.2	408	7.8	2.6
CaseWavesWallAWAS	0.015	3,741	10.2	408	7.7	2.6
CaseWavesFloating	0.015	8,531	8.0	320	15.5	2.7
CaseWaveTankFile	0.010	13,649	6.0	300	30.1	1.8
CaseSloshingTank	0.002	23,850	2.5	250	56.0	3.2

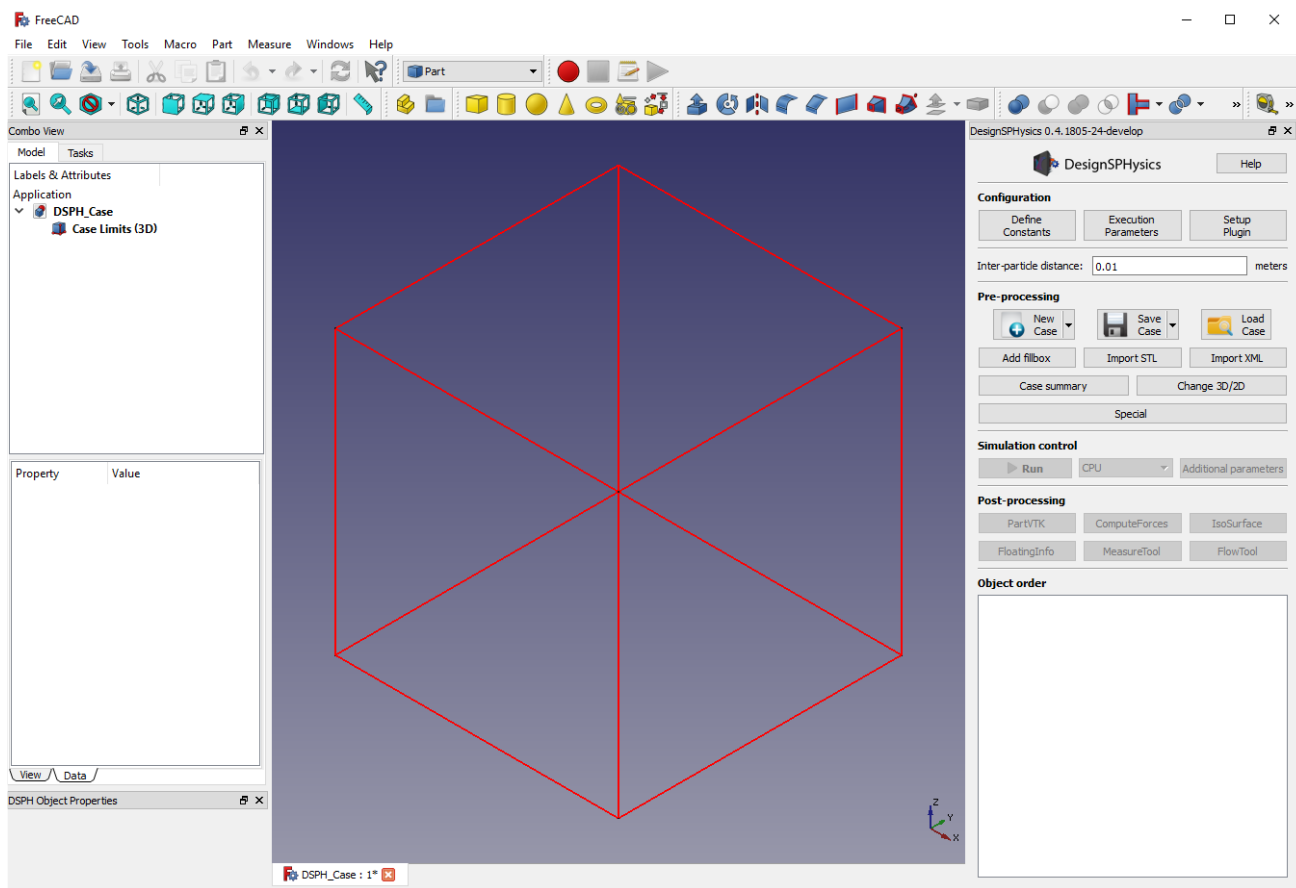
TO START:


- Unzip **DesignSPHysics_June2018.zip** and **Material_June2018.zip**
- Open FreeCad
- Navigate to User Macros location and select folder: **DesignSPHysics_June2018**
“DesignSPHysics.py” and click Execute



- Right panel: Configuration section: Click on **Setup Plugin**: DualSPHysics programs are automatically configured, but the **ParaView** executable must be configured manually.
 - o GenCase Path: DesignSPHysics_June2018/dualsphysics/EXECS/GenCase4_win64.exe
 - o DualSPHysics Path: DesignSPHysics_June2018/dualsphysics/EXECS/DualSPHysics4_win64.exe
 - o PartVTK Path: DesignSPHysics_June2018/dualsphysics/EXECS/PartVTK4_win64.exe
 - o ComputeForces Path: DesignSPHysics_June2018/dualsphysics/EXECS/ComputeForces4_win64.exe
 - o FloatingInfo Path: DesignSPHysics_June2018/dualsphysics/EXECS/FloatingInfo4_win64.exe
 - o MeasureTool Path: DesignSPHysics_June2018/dualsphysics/EXECS/MeasureTool4_win64.exe
 - o IsoSurface Path: DesignSPHysics_June2018/dualsphysics/EXECS/IsoSurface4_win64.exe
 - o BoundaryVTK Path: DesignSPHysics_June2018/dualsphysics/EXECS/BoundaryVTK4_win64.exe
 - o FlowTool Path: DesignSPHysics_June2018/dualsphysics/EXECS/FlowTool4_win64.exe
 - o ParaView Path: *C:/Program Files/ParaView 5.5.0-Qt5-Windows-64bit/bin/paraview.exe*

- Right panel: Pre-processing section: Click on **New Case**



Rotate using *SHIFT+MOUSE-Right*
 Move using *MOUSE-middle*
 Hide object *SELECT + SPACE*
 Adjust view  **Fit all**

- Note that **Button** indicates a button to click

CASEDAMBREAK3D

1.1. Open FreeCad and load macro

1.2. Right panel: Pre-processing section: **New Case**

1.3. Left panel: Application/DSPH_Case: **Case Limits (3D)**

Base/Placement/Position: x=-50 mm, y=-50 mm, z=-50 mm

Box: Length=1700 mm, Width=700 mm, Height=600 mm

1.4. Right panel: Pre-processing section: **Import STL**


Import STL options:

STL File: Material/CaseDambreak3D_structure.stl

Scaling factor X:1; Y:1; Z:1 (STL already defined in mm)


Import object name: **Tank**

Property	Value
Attachment	
Map Mode	Deactivated
Base	
Placement	[(0.00 0.00 1.00); 0.00 °; (-5...
Angle	0.00 °
Axis	[0.00 0.00 1.00]
Position	[-50.00 mm -50.00 mm -5...
x	-50.00 mm
y	-50.00 mm
z	-50.00 mm
Label	Case Limits (3D)
Box	
Length	1700.00 mm
Width	700.00 mm
Height	600.00 mm

1.5. “Create a cube solid” : Rename to **Building**

Base/Placement/Position: x=900 mm, y=200 mm, z=0 mm

Box: Length=100 mm, Width=100 mm, Height=450 mm

1.6. “Create a cube solid” : Rename to **Water**

Base/Placement/Position: x=0 mm, y=0 mm, z=0 mm

Box: Length=400 mm, Width=600 mm, Height=300 mm

1.7. Left panel: Select object + **Add to DSPH Simulation**

Tank: Type of object=Bound, MKBound=0, Fill mode=Face

Building: Type of object=Bound, MKBound=1, Fill mode=Face

Water: Type of object=Fluid, MKFluid=0, Fill mode=Full

1.10. Right panel: **Object order**: Define the order to create the SPH particles:

F0 Water	▲ ▼
B0 Tank	▲ ▼
B1 Building	▲ ▼

In this way, fluid SPH particles are first created and then the boundaries of the tank and the building that will overwrite the fluid particles if needed.

1.11. Right panel: Configuration: **Define Constants**: as it is (all by default)

1.12. Right panel: Configuration: **Execution Parameters**:

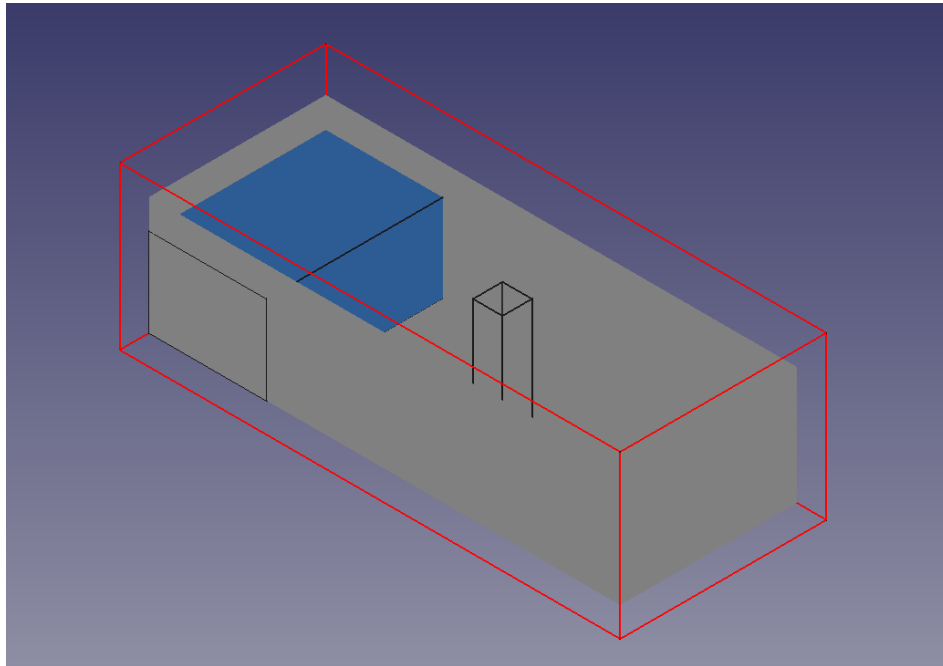
Viscosity value (alpha): 0.05

Enable DeltaSPH: Yes; DeltaSPH value: 0.1

Time of simulation: 0.5 s; Time out data: 0.005 s

- 1.13. Right panel: Configuration: **Inter-particle distance: 0.015 meters**
(Note that the resolution is very low to minimize runtime on CPU)

The case is now ready to be **generated**



- 1.14. Right panel: Pre-processing section: **Save Case** → **Save and run GenCase**
Name of the case: **CaseDambreak3D**

Save and run GenCase means that the FreeCAD software:

- Creates folder automatically for you for execution with that name **CaseDambreak3D**
- Creates geometries and files for FreeCad (DSPH_Case.FCStd & casedata.dsphdata)
- Creates initial XML input file (**CaseDambreak3D_Def.xml**)
- Creates a second output folder inside (**CaseDambreak3_out**)
- Exports STL if needed (**Tank.stl**)
- Generate particles (Executes **GenCase** code)

- 1.15. In the window “**Save & GenCase**”

You can read: *GenCase exported 34,236 particles.*

Show **Details** gives information about the execution of **GenCase**

Open with Paraview: CaseDambreak3D_All.vtk

Click **Apply** on the Properties Tab under Pipeline Browser field (left-hand side)

Click on the **+Y** button on the toolbar 

- 1.16. Check the content of **Case summary** (right panel)

- 1.17. Open the folder **CaseDambreak3D_out** and check the content:
- | | |
|--------------------------|--------------------------|
| CaseDambreak3D.bi4 | CaseDambreak3D.xml |
| CaseDambreak3D_All.vtk | CaseDambreak3D__Dp.vtk |
| CaseDambreak3D_Bound.vtk | CaseDambreak3D_Fluid.vtk |

The case is now ready to be **simulated**

1.18. Right panel: Simulation control:

Case will be executed using **CPU** (CPU by default)

▶ **Run** to start execution of the SPH solver (indicates estimated time to finish)

Show **Details** gives information and shows the log file of the execution (with runtimes, execution warnings and description of output files)

1.19. When **simulation is complete**:

Open again the folder **CaseDambreak3D/CaseDambreak3D_out** and check the content:

PartInfo.ibi4, Part_Head.ibi4, PartOut_000.obj4

Part_XXXX.bi4 (100 output files)

Run.out (log file)

1.20. Right panel: Post-processing section: **PartVTK**

This tool generates files for visualisation

Output format: VTK

Types to export: Fluid

File name: PartFluid

Export

1.21. Visualise the simulation

Open Paraview (Start → All Programs → Paraview)

File → Open “CaseDambreak3D_Dp.vtk”

Click Apply on the Properties Tab under Pipeline Browser field (left-hand side)

Change **Coloring** *Mk* to *Solid Color*

In **Styling** Opacity:0.5

File → Open “PartFluid_ .vtk”

Click Apply on the Properties Tab under Pipeline Browser field (left-hand side)

Change **Coloring** *Idp* to *Vel.* Click on *Edit*

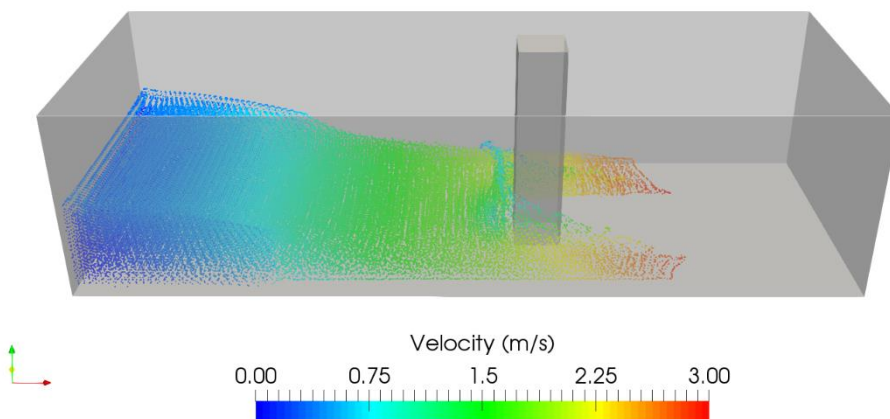
In Color Map Editor (right-hand side)

Click on *Choose Preset*  button and select “Blue to Red Rainbow” and click Apply

Click on *Rescale to custom range*  button and change the data range to 0.0 to 3.0

Play ▶ to visualise the simulation

Time: 0.400 s



1.22. Right panel: Post-processing section: **IsoSurface**

This tool generates files for visualisation

Save: Surface

File name: Iso

Export

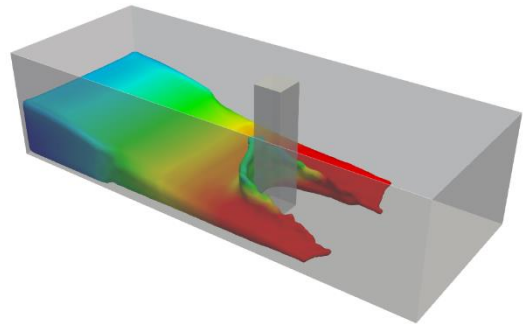
1.23. Open Paraview (Start → All Programs → Paraview)

File → Open “CaseDambreak3D_Dp.vtk”.

File → Open “Iso_..vtk”

Play ► to visualise the simulation

Surface only looks ok if high resolution is used



1.24. Right panel: Post-processing section: **FlowTool**

This tool computes inflow and outflow

List of boxes: **New Box**

BOX: Edit

1.0	0.67	0.0
1.0	0.0	0.0
1.6	0.0	0.0
1.6	0.67	0.0
1.0	0.67	0.4
1.0	0.0	0.4
1.6	0.0	0.4
1.6	0.67	0.4

CSV file name: _ResultsFlow

VTK file name: PartInFlow

Export

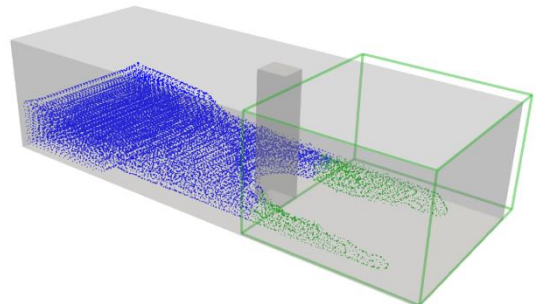
1.25. Open Paraview (Start → All Programs → Paraview)

File → Open “CaseDambreak3D_Dp.vtk”.

File → Open “_ResultFlow_boxes.vtk” (**BOX**)

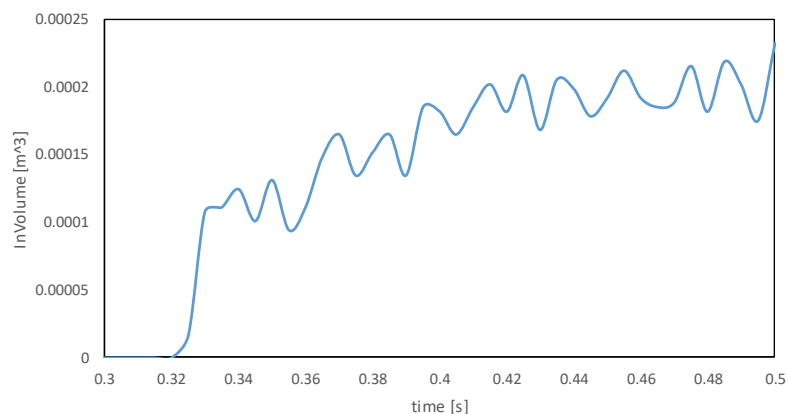
File → Open “PartInFlow.vtk”

Play ► to visualise the simulation



1.26. Open “_ResultFlow.csv” (from **CaseDambreak3D_out**) with EXCEL

Plot *Time[s]* versus *InVolume_BOX [m^3]* time series of the inflow in **BOX**



CASEDAMBREAK2D

2.1. Do not close FreeCad... starting from CaseDambrek3D (**Load Case** if needed)

Right panel: **Change 3D/2D** → New Y position (mm) = 100 mm

AUTOMATIC: *Left panel: Application/DSPH_Case: Case Limits (2D)*

Base/Placement/Position: x=-50 mm, y=100 mm, z=-50 mm

*Box: Length=1700 mm, **Width=1 μ m**, Height=500 mm*

2.2. Right panel: Configuration: **Execution Parameters:**

Time of simulation: 2.0 s; Time out data: 0.01 s

2.3. Right panel: Configuration: **Inter-particle distance: 0.005 meters**

2.4. Right panel: Pre-processing section: **Save as...**

Name of the case: **CaseDambreak2D**

2.5. Right panel: Pre-processing section: **Save and run GenCase**

In the window “Save & GenCase” you can read: *GenCase exported 5,281 particles*

Show **Details** gives information about the execution of **GenCase**

Open with Paraview: CaseDambreak2D_All.vtk

Click **Apply** on the Properties Tab under Object Inspector field (left-hand side)

Click on the **+Y** button on the toolbar

2.6. Right panel: Simulation control:

Case will be executed using **CPU**

► **Run** to start execution of the SPH solver (indicates estimated time to finish)

Show **Details** gives information and shows the log file of the execution (with runtimes!)

2.7. Right panel: Post-processing section: **PartVTK**

This tool generates files for visualisation

Output format: VTK

Types to export: All

File name: PartAll

Export

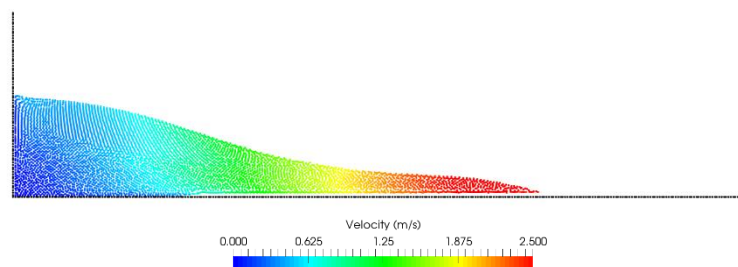
2.8. Visualise the simulation

Open Paraview (Start → All Programs → Paraview)

File → Open “PartAll_..vtk”

Play ► to visualise the simulation

Time: 0.350 s



CASEFLOATINGSPPHERE


3.1. Open FreeCad and load macro

3.2. Right panel: Pre-processing section: **New Case**

3.3. Left panel: Application/DSPH_Case: **Case Limits (3D)**


Base/Placement/Position: $x=-1500$ mm, $y=-1500$ mm, $z=-50$ mm

Box: Length=3000 mm, Width=3000 mm, Height=3000 mm

3.4. “Create a cube solid” : Rename to **Bottom**


Base/Placement/Position: $x=-1000$ mm, $y=-1000$ mm, $z=0$ mm

Box: Length=2000 mm, Width=2000 mm, Height=1 mm

3.5. “Create a sphere solid” : Rename to **Sphere**

Base/Placement/Position: $x=0$ mm, $y=0$ mm, $z=500$ mm

Sphere: Radius=400 mm

3.6. “Create a cube solid” : Rename to **Water**

Base/Placement/Position: $x=-1000$ mm, $y=-1000$ mm, $z=0$ mm

Box: Length=2000 mm, Width=2000 mm, Height=1500 mm

3.7. Left panel: Select object + **Add to DSPH Simulation**

Bottom: Type of object=Bound, MKBound=0, Fill mode=Full

Sphere: Type of object=Bound, MKBound=1, Fill mode=Full

Float state: Configure. Set floating=True Mass/density: $\rho_{\text{body}}=500$ kg/m³

Water: Type of object=Fluid, MKFluid=0, Fill mode=Full

3.8. Right panel: **Object order**: Define the following order:



3.9. Right panel: **Change 3D/2D** → New Y position (mm) = 0 mm

AUTOMATIC: Left panel: Application/DSPH_Case: **Case Limits (2D)**

Base/Placement/Position: $x=-1500$ mm, **$y=0$ mm**, $z=-50$ mm

Box: Length=3000 mm, **Width=1 μ m**, Height=4000 mm

3.10. Right panel: Configuration: **Define Constants**: CoefH: 1.2

3.11. Right panel: Configuration: **Execution Parameters**:

Precision: Double Step Algorithm: Symplectic

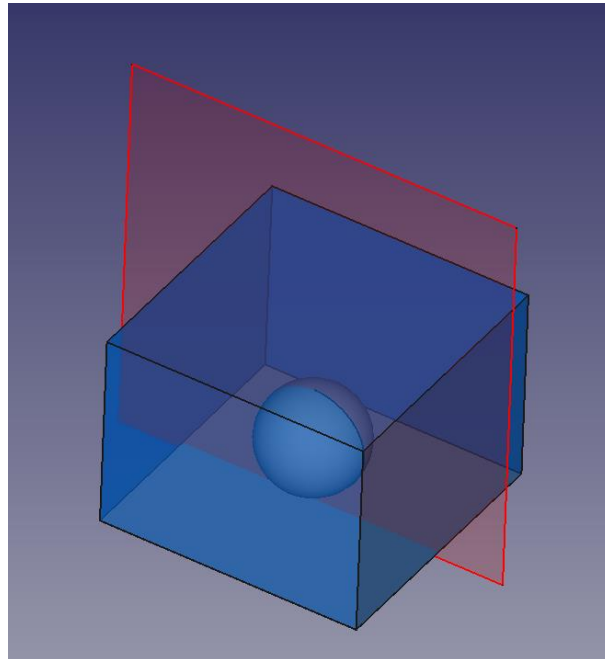
Enable DeltaSPH: Yes; DeltaSPH value: 0.1

Time of simulation: 6 s; Time out data: 0.02 s

X Periodicity: Y & Z increment: 0.0

3.12. Right panel: Configuration: **Inter-particle distance: 0.025 meters**

The case is now ready to be **generated**



3.13. Right panel: Pre-processing section: **Save Case: Save and run GenCase**
Name of the case: **CaseFloatingSphere**

3.14. In the window “Save & GenCase” you can read: *GenCase exported 4,941 particles*

Show **Details** gives information about the execution of **GenCase**

Open with Paraview: CaseFloatingSphere_All.vtk

Click **Apply** on the Properties Tab under Object Inspector field (left-hand side)

Click on the **+Y** button on the toolbar

3.15. Open the folder **CaseFloatingSphere_out** and check the content:

CaseFloatingSphere.bi4 CaseFloatingSphere.xml

CaseFloatingSphere_All.vtk CaseFloatingSphere__Dp.vtk

CaseFloatingSphere_Bound.vtk CaseFloatingSphere_Fluid.vtk

Note that *theoretical value of mass* of the sphere (circle in 2D) should be:

$$\text{mass(theory)} = \text{density} * \text{area} = 500 * \pi * \text{radius}^2 = 251.327 \text{ kg}$$

however, the *mass computed in SPH* (as summation of masses of discrete particles) is:

$$\text{mass(SPH)} = 271.563 \text{ kg (in CaseFloatingSphere.xml)}$$

with higher resolution (more particles): mass(SPH) will converge to mass(theory)

The case now is ready to be **simulated**

3.16. Right panel: Simulation control:

Case will be executed using **CPU**

► **Run** to start execution of the SPH solver (indicates estimated time to finish)

Show **Details** gives information and shows the log file of the execution (with runtimes!)

3.17. When **simulation is complete**:

Open again the folder **CaseFloatingSphere/CaseFloatingSphere_out** and check the content:

PartInfo.ibi4, Part_Head.ibi4, PartOut_000.obj4

Part_XXXX.bi4 (100 output files)

Run.out (log file)

3.18. Post-processing section: **PartVTK**

This tool generates files for visualisation

Output format: VTK

Types to export: Fluid

File name: PartFluid

Export

3.19. Post-processing section: **PartVTK**

This tool generates files for visualisation

Output format: VTK

Types to export: Floating

File name: PartFloating

Export

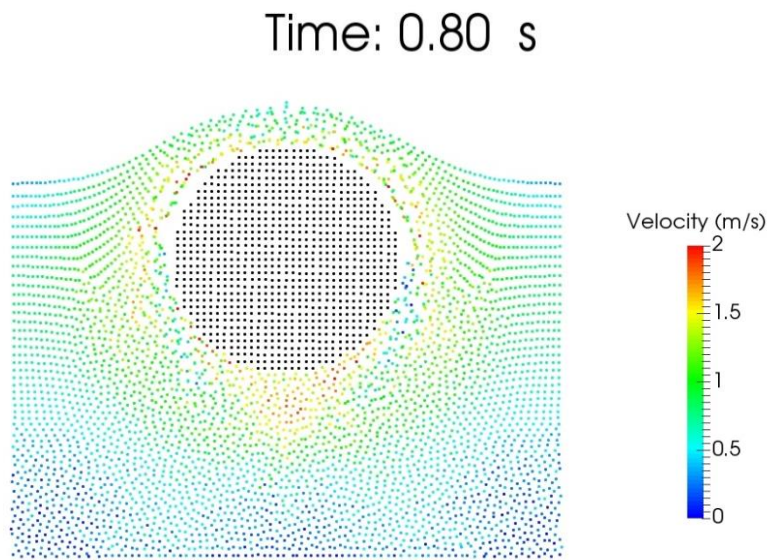
3.20. Visualise the simulation

Open Paraview (Start → All Programs → Paraview)

File → Open “PartFluid_..vtk”

File → Open “PartFloating_..vtk”

Play ► to visualise the simulation



3.21. Right panel: Post-processing section: **FloatingInfo**

This tool creates a CSV file with different data of the floating objects such as linear velocity, angular velocity, displacement of the centre, motions and angles of rotation.

MK to process: 12 (check the value of the sphere in Pre-processing section: **Case summary**)

File Name: FloatingMotion

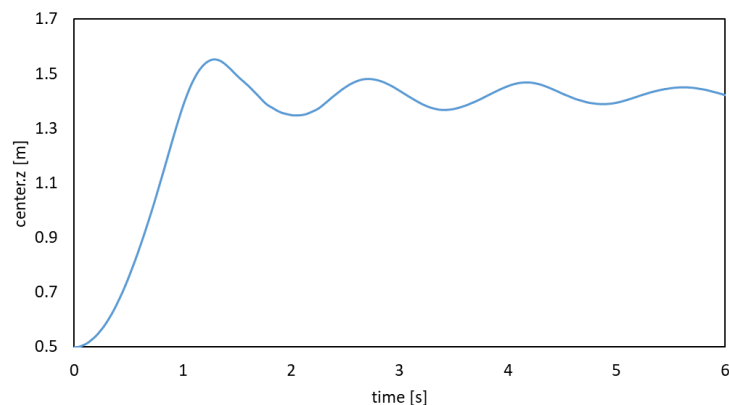
Export

3.22. Open “FloatingMotion_mk12.csv” (from **CaseFloatingSphere_out**) with EXCEL

Plot *Time[s]* versus *center.z [m]*: time series of the z-position of the center of the sphere

When the sphere reaches the equilibrium, center.z should be at final free surface level.

Note that theoretical value of the final free surface should be around 1.4 m.



3.23. Right panel: Post-processing section: **ComputeForces**

This tool computes the force exerted by the fluid onto a boundary object

Output format: CSV

MK to process: 12 (check the value of the sphere in Pre-processing section: **Case summary**)

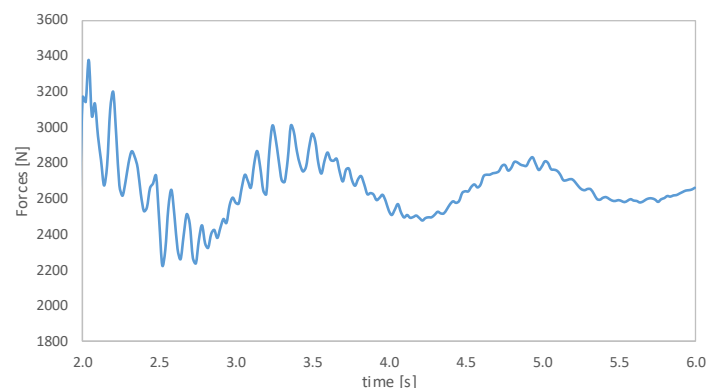
File Name: FloatingForce

Export

3.24. Open “FloatingForce.csv” (from **CaseFloatingSphere_out**) with EXCEL

Plot *Time[s]* versus *Forces[N]*: time series of the buoyant force exerted on the sphere

When the sphere reaches the equilibrium, the buoyant force should be the weight of the fluid that the body displaces, which theoretically is around 2465.5 N



Better results (and less noisy signal) are obtained using higher resolution (more particles)

CASEWAVES

4.1. Open FreeCad and load macro


4.2. Right panel: Pre-processing section: **New Case**

4.3. Right panel: **Change 3D/2D** → New Y position (mm) = 0 mm

Left panel: Application/DSPH_Case: Case Limits (**2D**)


Base/Placement/Position: x=-100 mm, y=**0 mm**, z=0 mm

Box: Length=11500 mm, **Width=1 μm**, Height=1000 mm

4.4. Left panel: “Create a cube solid” : Rename to **Bottom**

Base/Placement/Position: x=-100 mm, y=-500 mm, z=0 mm

Box: Length=2600 mm, Width=1000 mm, Height=1 mm


4.5. “Create a cube solid” : Rename to **Beach**

Base/Placement/Angle= 5.7°

Base/Placement/Axis: x=0 mm, y=-1 mm, z=0 mm

Base/Placement/Position: x=2500 mm, y=-500 mm, z=0 mm

Box: Length=7035 mm, Width=1000 mm, Height=1 mm

4.6. Left panel: “Create a cube solid” : Rename to **Piston**

Base/Placement/Position: x=0 mm, y=-500 mm, z=0 mm

Box: Length=10 mm, Width=1000 mm, Height=700 mm

4.7. Right panel: Pre-processing section: **Add fillbox**: Rename to **Water**

FillLimit: Base/Placement/Position: x=0 mm, y=-500 mm, z=0 mm

Box: Length=10000 mm, Width=1000 mm, Height=400 mm

FillPoint: Base/Placement/Position: x=1000 mm, y=0 mm, z=100 mm

Sphere: Radius=100 mm (for visualisation purposes in the GUI)

4.8. Left panel: Select object + **Add to DSPH Simulation**

Bottom: Type of object=Bound, MKBound=0, Fill mode=Full

Beach: Type of object=Bound, MKBound=1, Fill mode=Full

Water: Type of object=Fluid, MKFluid=0, Fill mode=Solid

Piston: Type of object=Bound, MKBound=2, Fill mode=Full

Motion: Configure. Set motion=True

Global Movements: Create New: → **Regular wave generator (Piston)**

Select “Regular wave generator (Piston)”

Duration: 0 s (zero is the end of simulation)

Wave Order: 2nd Order Depth: 0.4 m

Piston direction: (1,0,0)

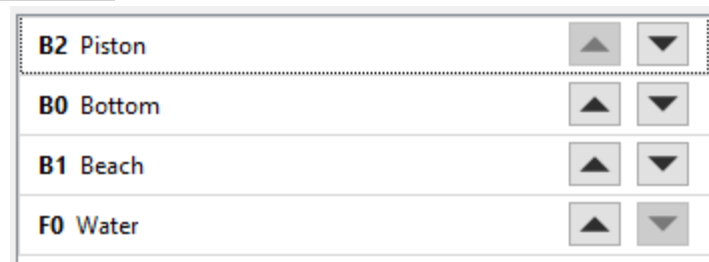
Wave height: 0.1 m, Wave period: 1.2 s

Phase:0 rad, Ramp: 0 (at least 3 periods should be used)

Save theoretical values: 10 periods, 20 period steps, xpos:2.0, zpos:-0.15

Global Movements: Select **USE**

4.9. Right panel: **Object order**: Define the following order:



4.10. Right panel: Configuration: **Define Constants**: CoefH: 1.2

4.11. Right panel: Configuration: **Execution Parameters**:

Step Algorithm: Symplectic

Viscosity value (alpha): 0.01

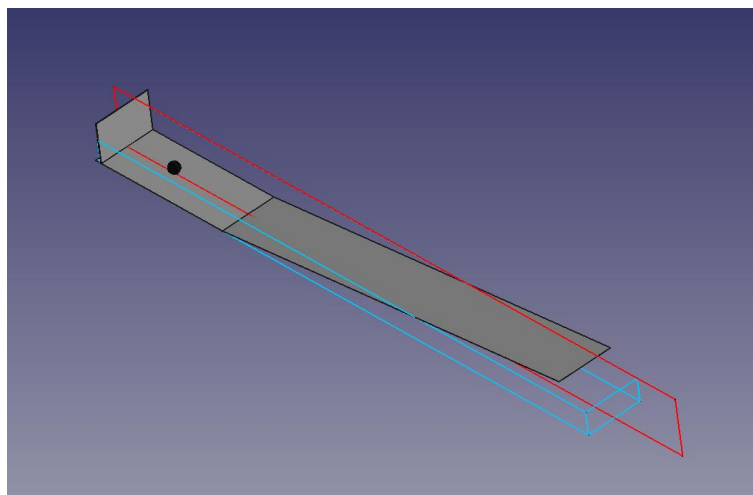
Viscosity factor with boundary: 0

Time of simulation: 8 s

Time out data: 0.025 s

4.12. Right panel: Configuration: **Inter-particle distance: 0.015 meters**

The case is ready now to be **generated**



4.13. Right panel: Pre-processing section: Save Case: **Save and run GenCase**

Name of the case: **CaseWaves**

4.14. In the window “**Save & GenCase**”

You can read: *GenCase exported 8,417 particles.*

Show **Details** gives information about the execution of **GenCase**

Open with Paraview: CaseWaves_All.vtk

Click **Apply** on the Properties Tab under Object Inspector field (left-hand side)

Click on the **+Y** button on the toolbar

4.15. Open the folder **CaseWaves_out** and check the content

The case is ready now to be **simulated**

4.16. Right panel: Simulation control:

Case will be executed using **CPU**

► **Run** to start execution of the SPH solver (indicates estimated time to finish)

Show **Details** gives information and shows the log file of the execution (with runtimes!)

4.17. When **simulation is complete**:

Open again the folder **CaseWaves_out** and check the content

4.18. Post-processing section: **PartVTK**

This tool generates files for visualisation

Output format: VTK

Types to export: Fluid

File name: PartFluid

Export

4.19. Post-processing section: **PartVTK**

This tool generates files for visualisation

Output format: VTK

Types to export: Moving

File name: PartPiston

Export

4.20. Right panel: Post-processing section: **IsoSurface**

This tool generates files for visualisation

Save: Slice

File name: Slice

Export

4.21. Visualise the simulation

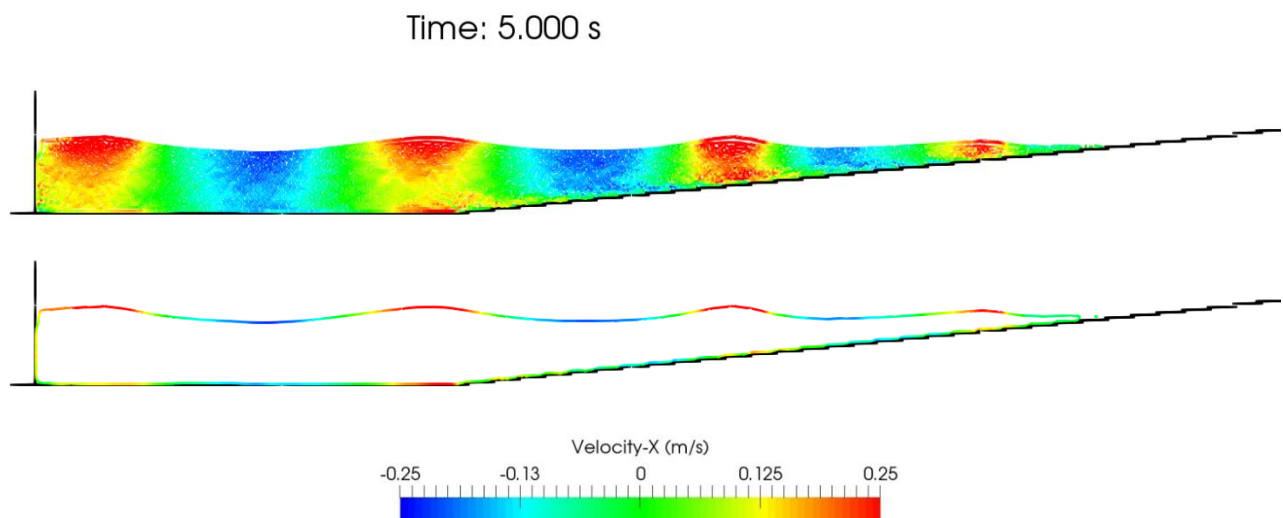
Open Paraview (Start → All Programs → Paraview)

File → Open “CaseWaves_Bound.vtk”

File → Open “PartFluid_..vtk” & File → Open “PartPiston_..vtk”

File → Open “Slice_..vtk”

Play ► to visualise the simulation



4.22. Right panel: Post-processing section: **MeasureTool**

This tool allows to compute different physical quantities (velocity, density, pressure, water elevation, etc.) at a set of given points.

Output format: CSV

Variables to export: Mass

Select “Calculate water elevation”

Grid of points:

BeginX	BeginY	BeginZ	StepX	StepY	StepZ	CountX	CountY	CountZ	FinalX	FinalY	FinalZ
2.0	0.0	0.0	1	1	0.001	1.0	1.0	601	2.0	0.0	0.6

File name: SPH_WG

Export

4.23. Right panel: Post-processing section: **MeasureTool**

Output format: CSV

Variables to export: Velocity

List of points:

X	Y	Z
2.0	0.0	0.25

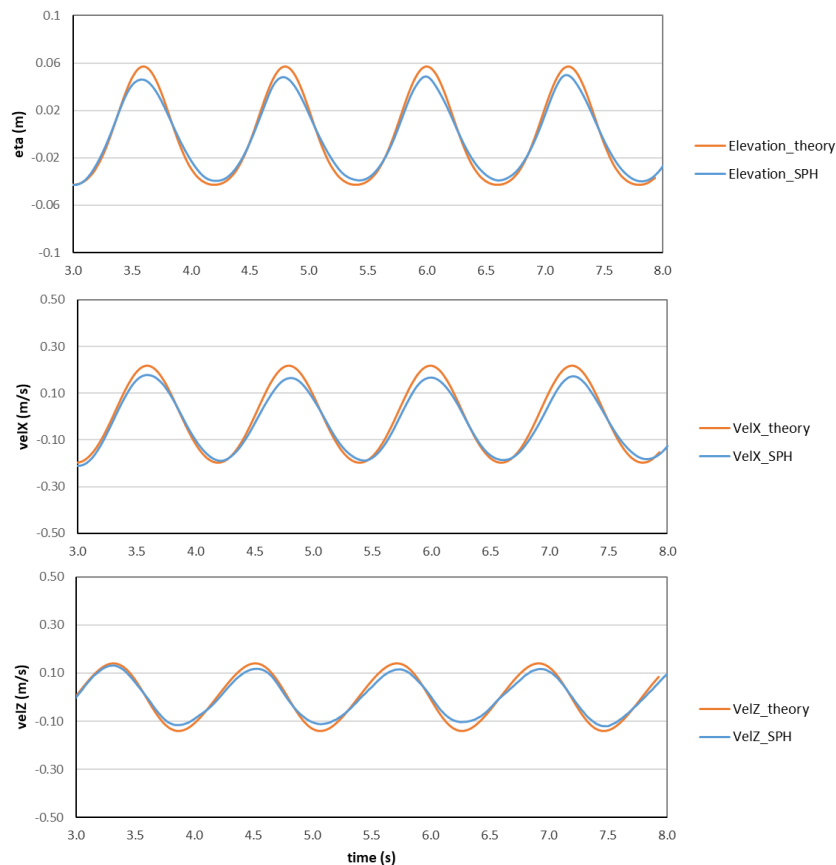
File name: SPH_VG

Export

4.24. Open Material/ CaseWaves_Validation_Theory.xls with EXCEL

Paste SPH results of SPH_WG_Height.csv and SPH_VG_Vel.csv (**CaseWaves_out**).

(Theoretical results were taken from: WavePaddle_mkb0002.csv)



CASEWAVES_WALL

5.1. Open FreeCad and load macro


5.2. Right panel: Pre-processing section: **New Case**

5.3. Right panel: **Change 3D/2D** → New Y position (mm) = 0 mm

Left panel: Application/DSPH_Case: Case Limits (**2D**)


Base/Placement/Position: x=-500 mm, y=**0 mm**, z=0 mm

Box: Length=3000 mm, **Width=1 μm**, Height=1000 mm

5.4. Left panel: “Create a cube solid” : Rename to **Bottom**


Base/Placement/Position: x=-500 mm, y=-500 mm, z=0 mm

Box: Length=2500 mm, Width=1000 mm, Height=1 mm

5.5. “Create a cube solid” : Rename to **Wall**

Base/Placement/Position: x=2000 mm, y=-500 mm, z=0 mm

Box: Length=10 mm, Width=1000 mm, Height=700 mm

5.6. Left panel: “Create a cube solid” : Rename to **Piston**

Base/Placement/Position: x=0 mm, y=-500 mm, z=0 mm

Box: Length=10 mm, Width=1000 mm, Height=700 mm

5.7. Right panel: Pre-processing section: **Add fillbox**: Rename to **Water**

FillLimit: Base/Placement/Position: x=-500 mm, y=-500 mm, z=0 mm

Box: Length=3000 mm, Width=1000 mm, Height=400 mm

FillPoint: Base/Placement/Position: x=1000 mm, y=0 mm, z=100 mm

Sphere: Radius=100 mm

5.8. Left panel: Select object + **Add to DSPH Simulation**

Bottom: Type of object=Bound, MKBound=0, Fill mode=Full

Wall: Type of object=Bound, MKBound=1, Fill mode=Full

Water: Type of object=Fluid, MKFluid=0, Fill mode=Solid

Piston: Type of object=Bound, MKBound=2, Fill mode=Full

Motion: Configure. Set motion=True

Global Movements: Create New: → **Regular wave generator (Piston)**

Select “Regular wave generator (Piston)”

Duration: 0 s (zero is the end of simulation)

Wave Order: 2nd Order Depth: 0.4 m

Piston direction: (1,0,0)

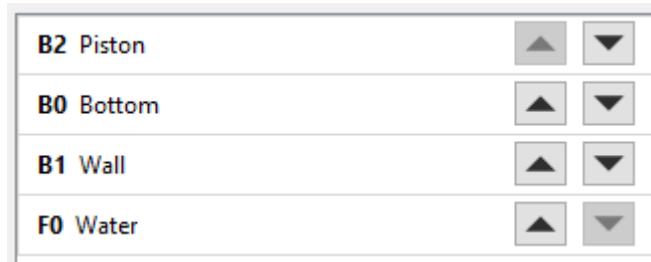
Wave height: 0.1 m, Wave period: 1.2 s

Phase:0 rad, Ramp: 0 (at least 3 periods should be used)

Save theoretical values: 10 periods, 20 period steps, xpos:2.0, zpos:-0.15

Global Movements: Select **USE**

5.9. Right panel: **Object order**: Define the following order:



5.10. Right panel: Configuration: **Define Constants**: CoefH: 1.2

5.11. Right panel: Configuration: **Execution Parameters**:

Step Algorithm: Symplectic

Viscosity value (alpha): 0.01

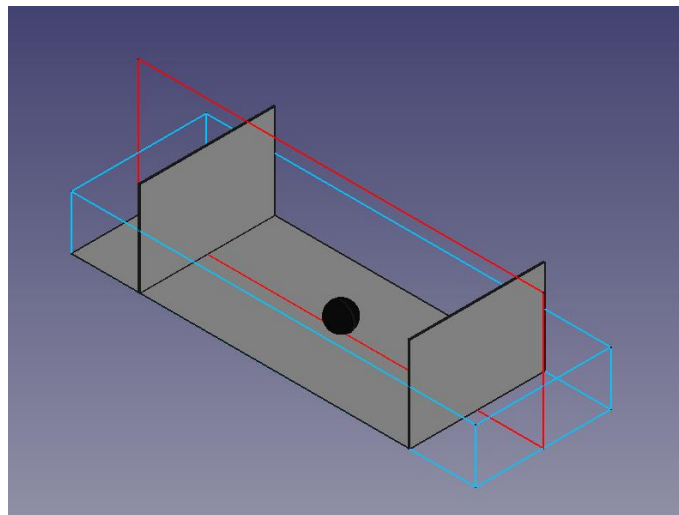
Time of simulation: 10.2 s

Viscosity factor with boundary: 0

Time out data: 0.025 s

5.12. Right panel: Configuration: **Inter-particle distance: 0.015 meters**

The case is ready now to be **generated**



5.13. Right panel: Pre-processing section: Save Case: **Save and run GenCase**
Name of the case: **CaseWavesWall**

5.14. In the window “Save & GenCase”

You can read: *GenCase exported 3,741 particles.*

Show **Details** gives information about the execution of **GenCase**

Open with Paraview: CaseWavesWall_All.vtk

Click **Apply** on the Properties Tab under Object Inspector field (left-hand side)

Click on the **+Y** button on the toolbar

5.15. Open the folder **CaseWavesWall_out** and check the content

The case is ready now to be **simulated**

5.16. Right panel: Simulation control:

Case will be executed using **CPU**

► **Run** to start execution of the SPH solver (indicates estimated time to finish)

Show **Details** gives information and shows the log file of the execution (with runtimes!)

5.17. When **simulation is complete**:

Open again the folder **CaseWavesWall_out** and check the content

5.18. Post-processing section: **PartVTK**

This tool generates files for visualisation

Output format: VTK

Types to export: Fluid

File name: PartFluid

Export

5.19. Post-processing section: **PartVTK**

This tool generates files for visualisation

Output format: VTK

Types to export: Moving

File name: PartPiston

Export

5.20. Visualise the simulation

Open Paraview (Start → All Programs → Paraview)

File → Open “CaseWavesWall_Bound.vtk”

File → Open “PartFluid_..vtk” & File → Open “PartPiston_..vtk”

Play ► to visualise the simulation

5.21. Right panel: Post-processing section: **ComputeForces**

This tool computes the force exerted by the fluid onto a boundary object

Output format: CSV

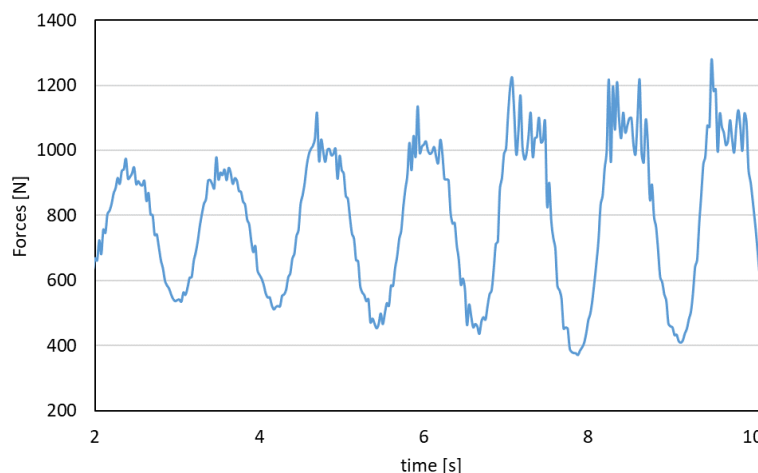
MK to process: 12 (check the value of the sphere in Pre-processing section: **Case summary**)

File Name: WallForce

Export

5.22. Open “WallForce.csv” (from **CaseWavesWall_out**) with EXCEL

Plot *Time[s]* versus *Forces[N]*: time series of the force exerted by the waves onto the wall



NOTE that force includes the initial hydrostatic one!!

CASEWAVES_WALL_AWAS

6.1. Do not close FreeCad... starting from CaseWavesWall

Right panel: Pre-processing section: Save Case: **Save As**

Name of the case: **CaseWavesWallAWAS**

6.2. Left panel: Select **Piston** object and edit **DSPH Object Properties**

Piston: Type of object=Bound, MKBound=2, Fill mode=Full

Motion: Configure. Set motion=True

Global Movements: Create New: → **Regular wave generator (Piston)**

Select “Regular wave generator (Piston)”

Duration: 0 s (zero is the end of simulation)

Wave Order: 2nd Order Depth: 0.4 m

Piston direction: (1,0,0)

Wave height: 0.1 m, Wave period: 1.2 s

Phase:0 rad, Ramp: 0 (at least 3 periods should be used)

Save theoretical values: 10 periods, 20 period steps, xpos:2.0, zpos:-0.15

NOW: AWAS configuration ENABLED

Start AWAS: 1 s Still water level: 0.4 m

Gauge X (coef*h): 4 Gauge Y: 0 m

Gauge Z Min: 0.15 m Gauge Z Max: 0.65 m

Gauge dp: 0.25 Coef. mass limit: 0.4

Limit acceleration: 2

Global Movements: Select **USE**

6.3. Right panel: Configuration: **Inter-particle distance: 0.015 meters (the same as before)**

The case is ready now to be **generated**

6.4. Right panel: Pre-processing section: Save Case: **Save and run GenCase**

6.5. In the window “**Save & GenCase**”

You can read: *GenCase exported 3,741 particles.*

Show **Details** gives information about the execution of **GenCase**

Open with Paraview: CaseWavesWall_All.vtk

Click **Apply** on the Properties Tab under Object Inspector field (left-hand side)

Click on the **+Y** button on the toolbar

The case is ready now to be **simulated**

6.6. Right panel: Simulation control:

Case will be executed using **CPU**

► **Run** to start execution of the SPH solver (indicates estimated time to finish)

Show **Details** gives information and shows the log file of the execution (with runtimes!)

6.7. Post-processing section: **PartVTK**

This tool generates files for visualisation

Output format: VTK

Types to export: Fluid

File name: PartFluid

Export

6.8. Post-processing section: **PartVTK**

This tool generates files for visualisation

Output format: VTK

Types to export: Moving

File name: PartPiston

Export

6.9. Visualise the simulation

Open Paraview (Start → All Programs → Paraview)

File → Open “CaseWavesWall_Bound.vtk”

File → Open “PartFluid_..vtk” & File → Open “PartPiston_..vtk”

Play ► to visualise the simulation

6.10. Right panel: Post-processing section: **ComputeForces**

This tool computes the force exerted by the fluid onto a boundary object

Output format: CSV

MK to process: 12 (check the value of the sphere in Pre-processing section: **Case summary**)

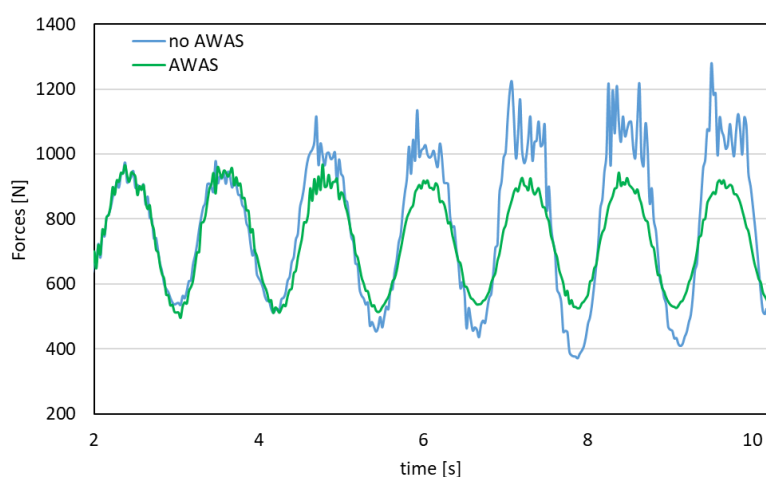
File Name: WallForce_AWAS

Export

6.11. Open “WallForce_AWAS.csv” (from **CaseWavesWallAWAS_out**) with EXCEL

Plot *Time[s]* versus *Forces[N]*: time series of the force exerted by the waves onto the wall

Compares with and without AWAS



When AWAS is disable, extra energy is introduced into the system due to wave re-reflection, as noticeable also by the time series of the forces exerted on the vertical wall. Instead, when AWAS is employed, the force time series keeps a periodic regular behaviour in time which proves that only regular incident waves hit the wall and that reflected waves are absorbed by the piston.


CASEWAVES_FLOATING (slow in low range CPU)

7.1. Open FreeCad and load macro

7.2. Right panel: Pre-processing section: **Load Case** → CaseWaves\casedata.dsphdata

Right panel: Pre-processing section: Save Case: **Save As**

Name of the case: **CaseWavesFloating**

7.3. Left panel: “Create a cube solid”  : Rename to **Floater**

Base/Placement/Position: x=1850 mm, y=-500 mm, z=300 mm

Box: Length=300 mm, Width=1000 mm, Height=200 mm

7.4. Left panel: Select object + **Add to DSPH Simulation**

Floater: Type of object=Bound, MKBound=3, Fill mode=Solid

Float state: Configure. Set floating=True

Mass/density: massbody=30 kg Gravity center: 2, 0, 0.405

7.5. Right panel: Update **Object order**: Define the following order:

B2 Piston	▲	▼
B0 Bottom	▲	▼
B1 Beach	▲	▼
B3 Floater	▲	▼
F0 Water	▲	▼

7.6. Right panel: Configuration: **Define Constants**: CoefH: 1.2

7.7. Right panel: Configuration: **Execution Parameters**:

Step Algorithm: Symplectic

Viscosity value (alpha): 0.01

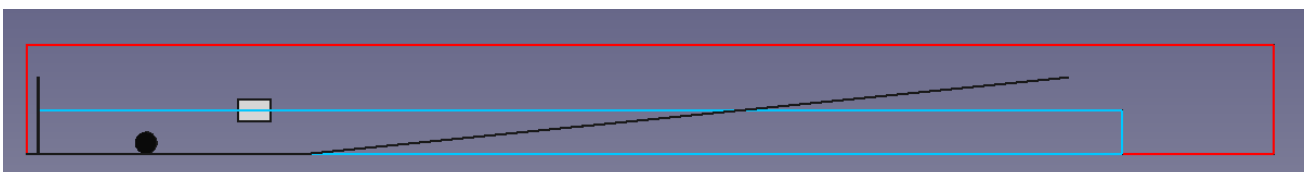
Viscosity factor with boundary: 0

Time of simulation: 8 s

Time out data: 0.025 s

7.8. Right panel: Configuration: **Inter-particle distance: 0.015 meters**

The case is ready now to be **generated**



7.9. Right panel: Pre-processing section: Save Case: **Save and run GenCase**

7.10. In the window “Save & GenCase”

You can read: *GenCase exported 8,531 particles.*

Show **Details** gives information about the execution of **GenCase**

Open with Paraview: CaseWavesFloating_All.vtk

Click **Apply** on the Properties Tab under Object Inspector field (left-hand side)

Click on the **+Y** button on the toolbar

7.11. Open the folder **CaseWavesFloating_out** and check the content

The case is ready now to be **simulated**

7.12. Right panel: Simulation control:

Case will be executed using **CPU**

▶Run to start execution of the SPH solver (indicates estimated time to finish)

Show **Details** gives information and shows the log file of the execution (with runtimes!)

7.13. When **simulation is complete**:

Open again the folder **CaseWavesFloating_out** and check the content

7.14. Right panel: Post-processing section: **PartVTK**

Output format: VTK

Types to export: Fluid

File name: PartFluid

Export

7.15. Right panel: Post-processing section: **PartVTK**

Output format: VTK

Types to export: Moving

File name: PartPiston

Export

7.16. Right panel: Post-processing section: **PartVTK**

Output format: VTK

Types to export: Floating

File name: PartFloating

Export

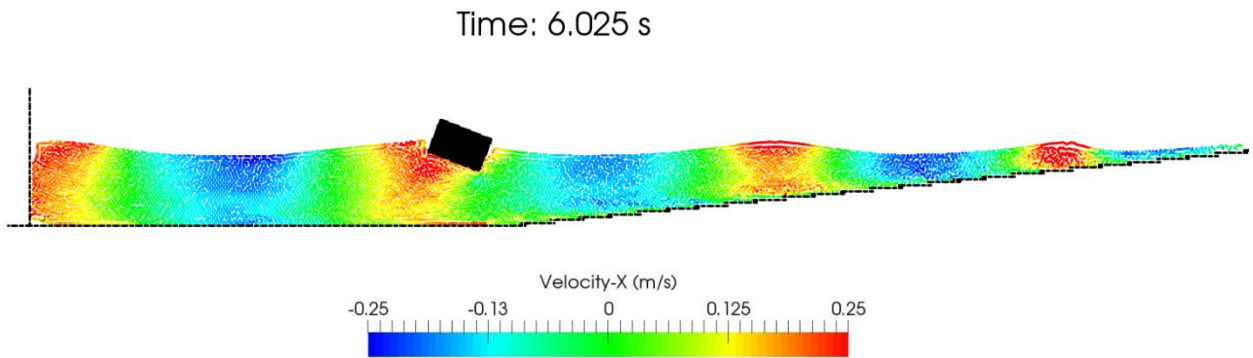
7.17. Visualise the simulation

Open Paraview (Start → All Programs → Paraview)

File → Open “CaseWavesFloating_Bound.vtk”

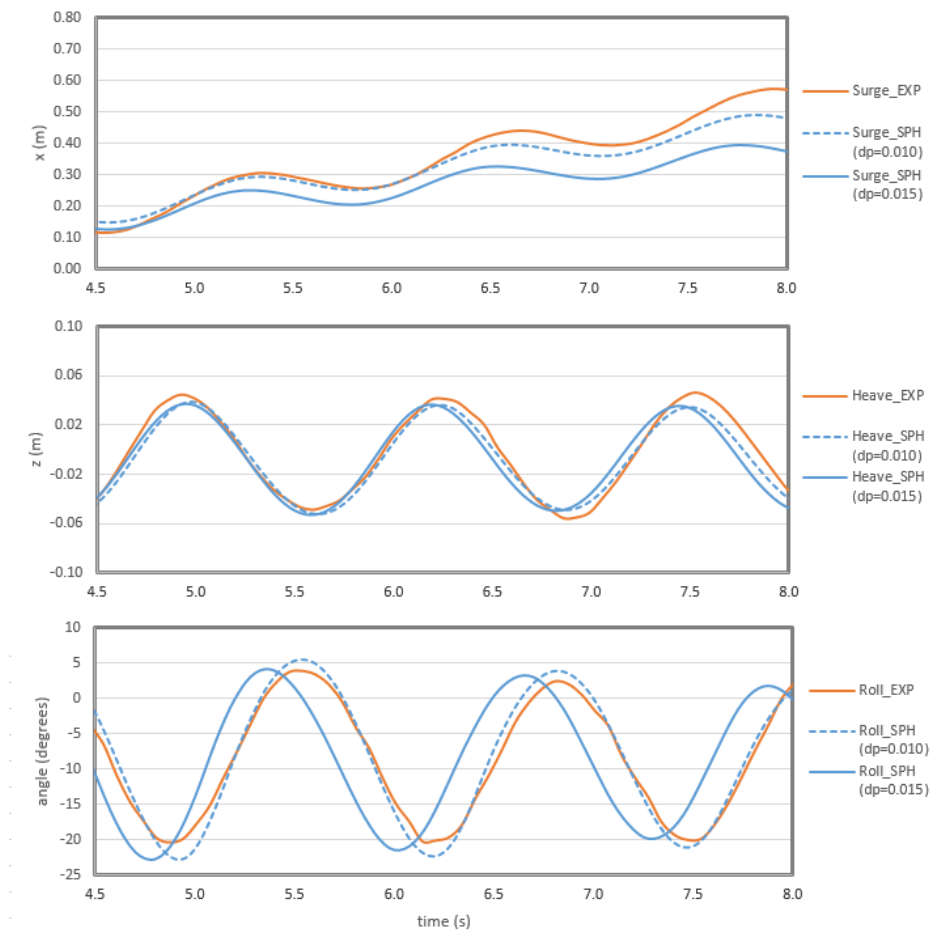
File → Open “PartFluid_ .vtk” & “PartPiston_ .vtk” & “PartFloating_ .vtk”

Play ▶ to visualise the simulation



- 7.18. Right panel: Post-processing section: **FloatingInfo**
MK to process: 14 (check the value of the sphere in Pre-processing section: **Case summary**)
 File Name: FloatingMotion
Export

- 7.19. Open Material/CaseWavesFloating_Validation_EXP.xls with EXCEL
 Paste SPH results (surge, heave and roll) of FloatingMotion_mk14.csv.



In this case numerical results only agree with experimental data if higher resolution is used.

CASEWAVETANKFILE (slow in low range CPU)

8.1. Open FreeCad and load macro

8.2. Right panel: Pre-processing section: **New Case**

8.3. Left panel: Application/DSPH_Case: Case Limits (**3D**)

Base/Placement/Position: x=-500 mm, y=0 mm, z=0 mm

Box: Length=6500 mm, Width=370 mm, Height=700 mm

8.4. Right panel: Pre-processing section: **Import STL**


Import STL options:

STL File: Material/CaseWaveTankFile_structure.stl

Scaling factor X:1000; Y:1000; Z:1000 (STL is defined in “meters”, but FreeCad uses mm)


Import object name: **Structure**

Base/Placement/Position: x=3000 mm, y=0 mm, z=0 mm

8.5. Left panel: “Create a cube solid” : Rename to **Bottom**

Base/Placement/Position: x=-500 mm, y=0 mm, z=0 mm

Box: Length=6500 mm, Width=370 mm, Height=1 mm

8.6. “Create a cube solid” : Rename to **Piston**

Base/Placement/Position: x=-30 mm, y=0 mm, z=0 mm

Box: Length=30 mm, Width=370 mm, Height=550 mm

8.7. Right panel: Pre-processing section: **Add fillbox**: Rename to **Water**

FillLimit: Base/Placement/Position: x=0 mm, y=0 mm, z=0 mm

Box: Length=6000 mm, Width=370 mm, Height=310 mm

FillPoint: Base/Placement/Position: x=3000 mm, y=200 mm, z=100 mm

Sphere: Radius=100 mm

8.8. Left panel: Select object + **Add to DSPH Simulation**

Structure: Type of object=Bound, MKBound=0, Fill mode=Face

Bottom: Type of object=Bound, MKBound=1, Fill mode=Full

Water: Type of object=Fluid, MKFluid=0, Fill mode=Solid

Piston: Type of object=Bound, MKBound=2, Fill mode=Full

Motion: Configure. Set motion=True

Global Movements: Create New → **Linear motion from a file**

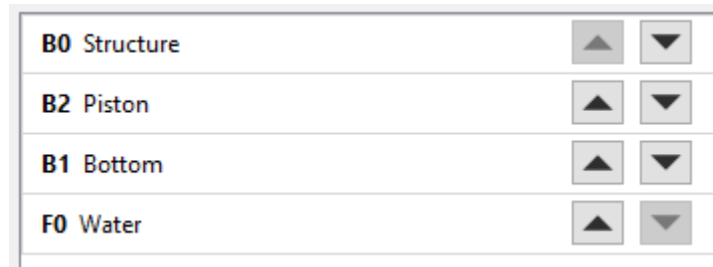
Select “Linear motion from a file”

Duration: 25 s

File name: Material/CaseWaveTankFile_movement.dat

Number of fields: 3; Column with time: 0; X position column: 1

8.9. Right panel: **Object order**: Define the following order:



8.10. Right panel: Configuration: **Define Constants**: CoefH: 1.5

8.11. Right panel: Configuration: **Execution Parameters**:

Step Algorithm: Symplectic

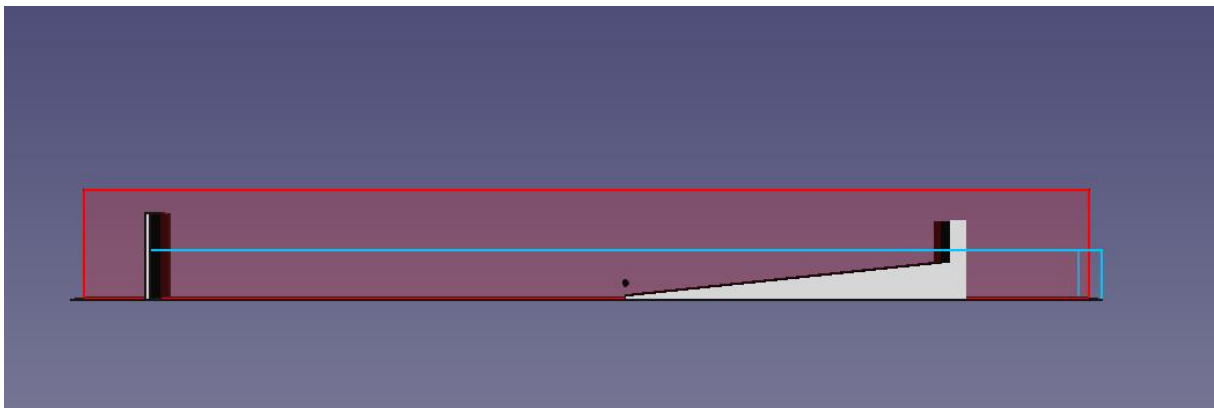
Viscosity value (alpha): 0.02

Time of simulation: 6 (try 11.5 s for better visualisation of force onto the structure)

Time out data: 0.02 s

8.12. Right panel: Configuration: **Inter-particle distance: 0.01 meters**

The case is ready now to be **generated**



8.13. Right panel: **Change 3D/2D** → New Y position (mm) = 200 mm

AUTOMATIC: Left panel: Application/DSPH_Case: Case Limits (2D)

Base/Placement/Position: $x=-500$ mm, $y=200$ mm, $z=0$ mm

Box: Length=6500 mm, **Width=1 μ m**, Height=700 mm

8.14. Right panel: Pre-processing section: Save Case: **Save and run GenCase**

Name of the case: **CaseWaveTank**

8.15. In the window “Save & GenCase”

You can read: *GenCase exported 13,649 particles.*

Show **Details** gives information about the execution of GenCase

Open with Paraview: CaseWaveTank_All.vtk

Click **Apply** on the Properties Tab under Object Inspector field (left-hand side)

Click on the **+Y** button on the toolbar

8.16. Open the folder **CaseWaveTank_out** and check the content

The case is ready now to be **simulated**

8.17. Right panel: Simulation control:

Case will be executed using **CPU**

► **Run** to start execution of the SPH solver (indicates estimated time to finish)

Show **Details** gives information and shows the log file of the execution (with runtimes!)

8.18. When **simulation is complete**:

Open again the folder **CaseWaveTank_out** and check the content

8.19. Post-processing section: **PartVTK**

Output format: VTK

Types to export: Fluid

File name: PartFluid

Export

8.20. Post-processing section: **PartVTK**

Output format: VTK

Types to export: Moving

File name: PartPiston

Export

8.21. Visualise the simulation

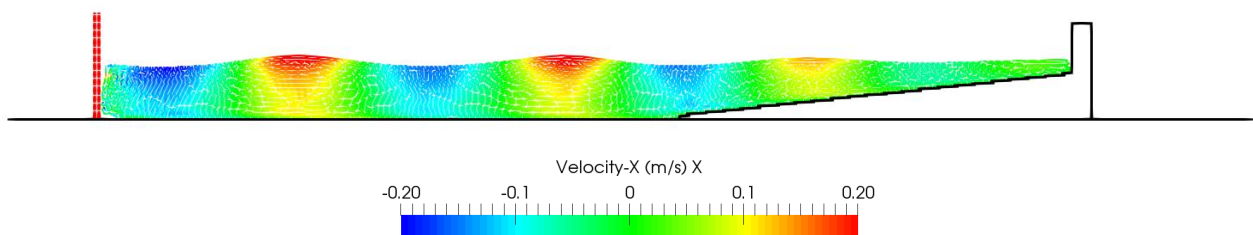
Open Paraview (Start → All Programs → Paraview)

File → Open “CaseWaveTank_Bound.vtk”

File → Open “PartFluid_..vtk” & File → Open “PartPiston_..vtk”

Play ► to visualise the simulation

Time: 2.80 s



8.22. Right panel: Post-processing section: **MeasureTool**

Output format: CSV

Variables to export: Mass

Check “Calculate water elevation”

Grid of points:

BeginX	BeginY	BeginZ	StepX	StepY	StepZ	CountX	CountY	CountZ	FinalX	FinalY	FinalZ
0.36	0.2	0.2	1	1	0.001	1.0	1.0	401	0.36	0.2	0.6

File name: SPH_wg1

Export

- 8.23. You can also try to **Export** the same results as in previous point but in VTK output format and plot those VTK files in Paraview:

Open Paraview (Start → All Programs → Paraview)

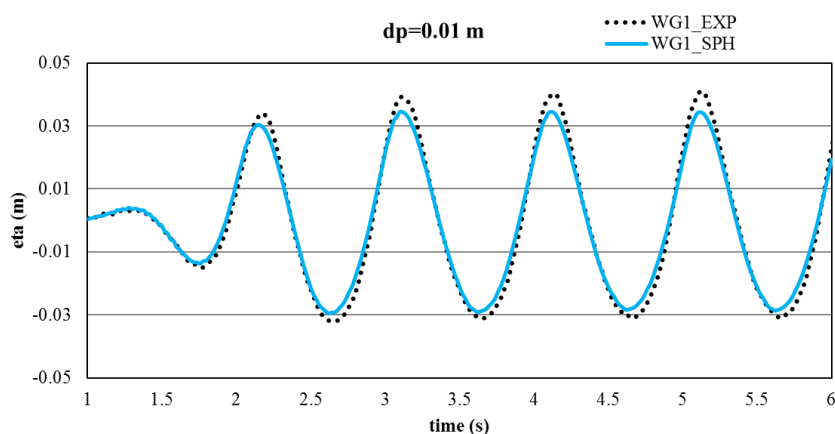
File → Open “SPH_WG_Height_..vtk” & File → Open “SPH_WG_Mass_..vtk”

Play ► to visualise the simulation

- 8.24. Open Material/CaseWaveTankFile_Validation_EXP.xls with EXCEL

Paste SPH results from SPH_wg1_Height.csv

Plot *Time* versus *Height_0*: time series of the water elevation at wg1 (x=0.36 m)



- 8.25. Right panel: Post-processing section: **ComputeForces**

This tool computes the force exerted by the fluid onto a boundary object

Output format: CSV

ID to process: **248-274** (you can find these values in Paraview)

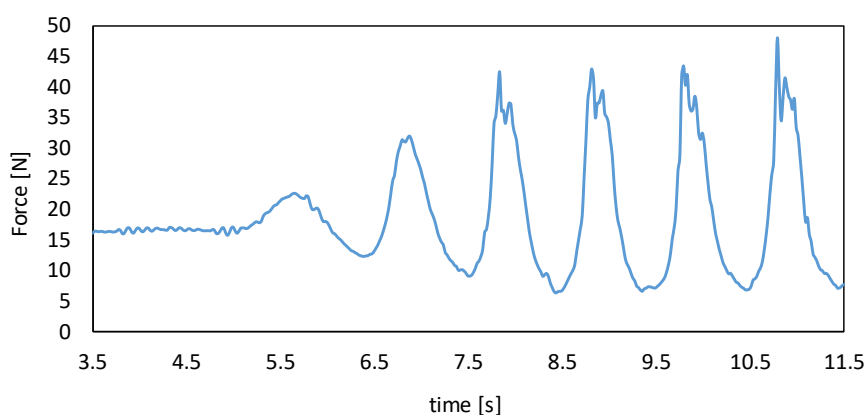
File Name: StructureForce

Export

- 8.26. Open “StructureForce.csv” with EXCEL and

Plot *Time* versus *Forces [N]*: time series of the force exerted onto the structure

NOTE that force includes the initial hydrostatic one (16 N)



CASESLOSHINGTANK (slow in low range CPU)

9.1. Open FreeCad and load macro


9.2. Right panel: Pre-processing section: **New Case**

9.3. Right panel: **Change 3D/2D** → New Y position (mm) = 0 mm

Left panel: Application/DSPH_Case: Case Limits (**2D**)


Base/Placement/Position: x=-500 mm, y=**0 mm**, z=-100 mm

Box: Length=1000 mm, **Width=1 μm**, Height=700 mm

9.4. “Create a cube solid” : Rename to **Water**

Base/Placement/Position: x=-450 mm, y=-50 mm, z=0 mm

Box: Length=900 mm, Width=100 mm, Height=100 mm

9.5. “Create a cube solid” : Rename to **Tank**

Base/Placement/Position: x=-450 mm, y=-50 mm, z=0 mm

Box: Length=900 mm, Width=100 mm, Height=500 mm

9.6. Left panel: Select object + **Add to DSPH Simulation**

Water: Type of object=Fluid, MKFluid=0, Fill mode=Full

Tank: Type of object=Bound, MKBound=0, Fill mode=Face

Motion: Configure. Set motion=True

Left: Global Movements: Create New → **Movement**

Select “New Movement”. Rename to **Sloshing Tank**

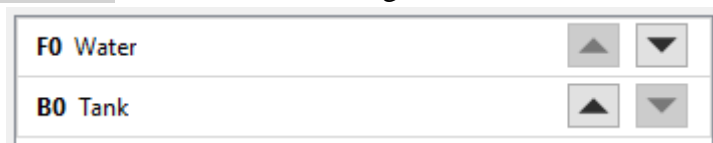
“Add a sinusoidal rotational motion” at right panel

Axis of rotation: (0, -1, 0.25) to (0, 1, 0.25).

Frequency=0.5; Amplitude=0.14 rad; Phase=0.0 rad

Duration=5.0 s

9.7. Right panel: **Object order**: Define the following order:



9.8. Right panel: Configuration: **Define Constants**: as it is (all by default)

9.9. Right panel: Configuration: **Execution Parameters**:

Step Algorithm: Symplectic

Viscosity value (alpha): 0.05

Time of simulation: 2.5 s; Time out data: 0.01 s

Fixed Domain: X Min:-0.55 X Max: 0.55

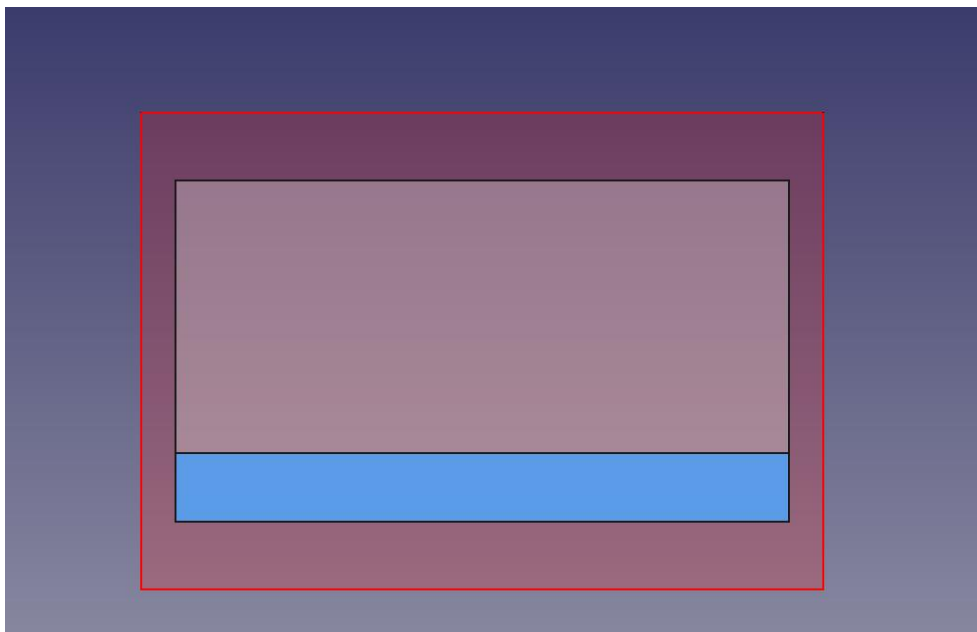
Y Min:-0.1 Y Max: 0.1

Z Min: -0.1 Z Max: 0.6

Domain limits need to be changed to allow boundary particles to move

9.10. Right panel: Configuration: **Inter-particle distance: 0.002 meters**

The case is ready now to be **generated**



9.11. Right panel: Pre-processing section: Save Case: **Save and run GenCase**
Name of the case: **CaseSloshingTank**

9.12. In the window “**Save & GenCase**”

You can read: *GenCase exported 23,850 particles.*

Show **Details** gives information about the execution of **GenCase**

Open CaseSloshingTank_All.vtk with **Paraview**

Click **Apply** on the Properties Tab under Object Inspector field (left-hand side)

Click on the **+Y** button on the toolbar

9.13. Open the folder **CaseSloshingTank_out** and check the content:

SloshingTank.bi4	SloshingTank.xml
SloshingTank_All.vtk	SloshingTank__Dp.vtk
SloshingTank_Bound.vtk	SloshingTank_Fluid.vtk

The case is ready now to be **simulated**

9.14. Right panel: Simulation control:

Case will be executed using **CPU**

▶Run to start execution of the SPH solver (indicates estimated time to finish)

Show **Details** gives information and shows the log file of the execution (with runtimes!)

9.15. When **simulation is complete**:

Open again the folder **CaseSloshingTank_out** and check the content:

PartInfo.ibi4, Part_Head.ibi4, PartOut_000.obt4

Part_XXXX.bi4 (100 output files)

Run.out (log file)

9.16. Right panel: Post-processing section: **PartVTK**

Output format: VTK

Types to export: All

File name: PartAll

Export

9.17. Visualise the simulation

Open Paraview (Start → All Programs → Paraview)

File → Open “PartAll_..vtk”

Play ► to visualise the simulation

