



# SPH Training Day

## 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](http://www.dual.sphysics.org)) to study fluid flows following various simulations, including *dam break*, *wave generation*, *wave absorption*, *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](http://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.

### FIRST:

1. Install Paraview <http://www.paraview.org/>
2. Install Notepad++ <http://notepad-plus-plus.org/>
3. Install FreeCad <https://www.freecadweb.org/>
4. Download the course from [www.design.sphysics.org/training/CPU\\_Beijing\\_2017](http://www.design.sphysics.org/training/CPU_Beijing_2017)

User: training

Password: DualSPHysics\_CoursE\_2017

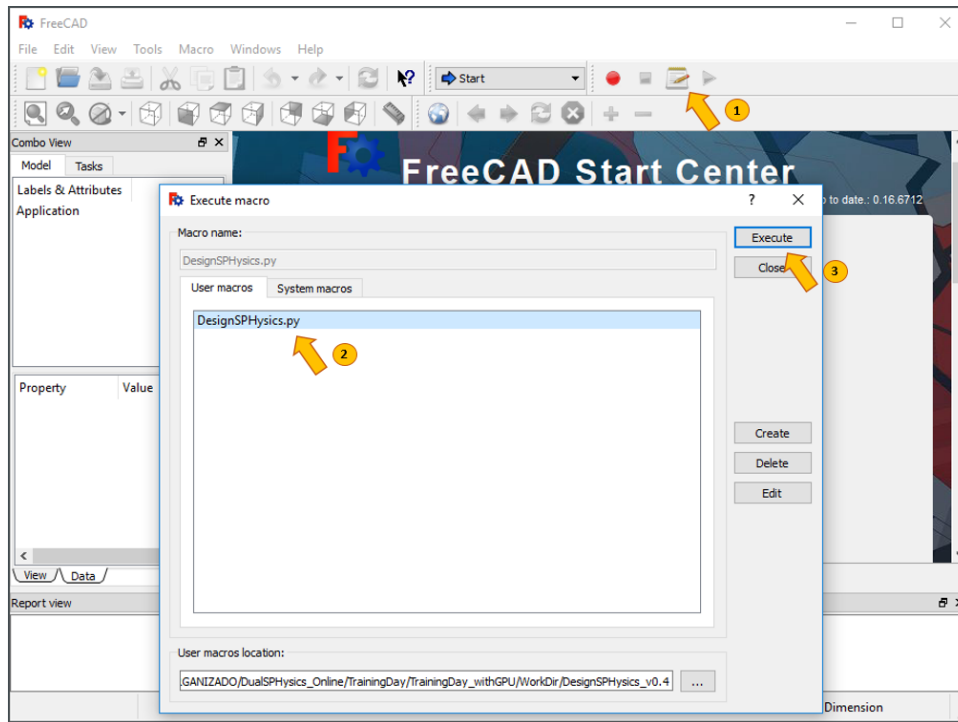
- **DesignSPHysics\_v0.4.zip:**  
package of the GUI application and executables of DualSPHysics package v4.2.019
- **Material.zip:**  
material needed for the course

### CASES OF EXAMPLE:

Name	dp (m)	Number of particles	Physical time (s)	Output files	Runtime Intel I5-3470 3.2GHz (seconds)	Runtime GeForce GTX 1080 (seconds)
CaseDambreak3D	0.0125	54,801	0.5	100	338	8
CaseDambreak2D	0.005	5,281	2.0	200	98	44
CaseFloatingSphere	0.025	9,801	6.0	300	689	174
CaseWaves	0.015	11,138	8.0	320	531	119
CaseWavesFloating	0.015	11,252	8.0	320	705	158
CaseWaveTank	0.01	13,649	6.0	300	693	97
CaseSloshingTank	0.002	23,850	2.5	250	1347	178

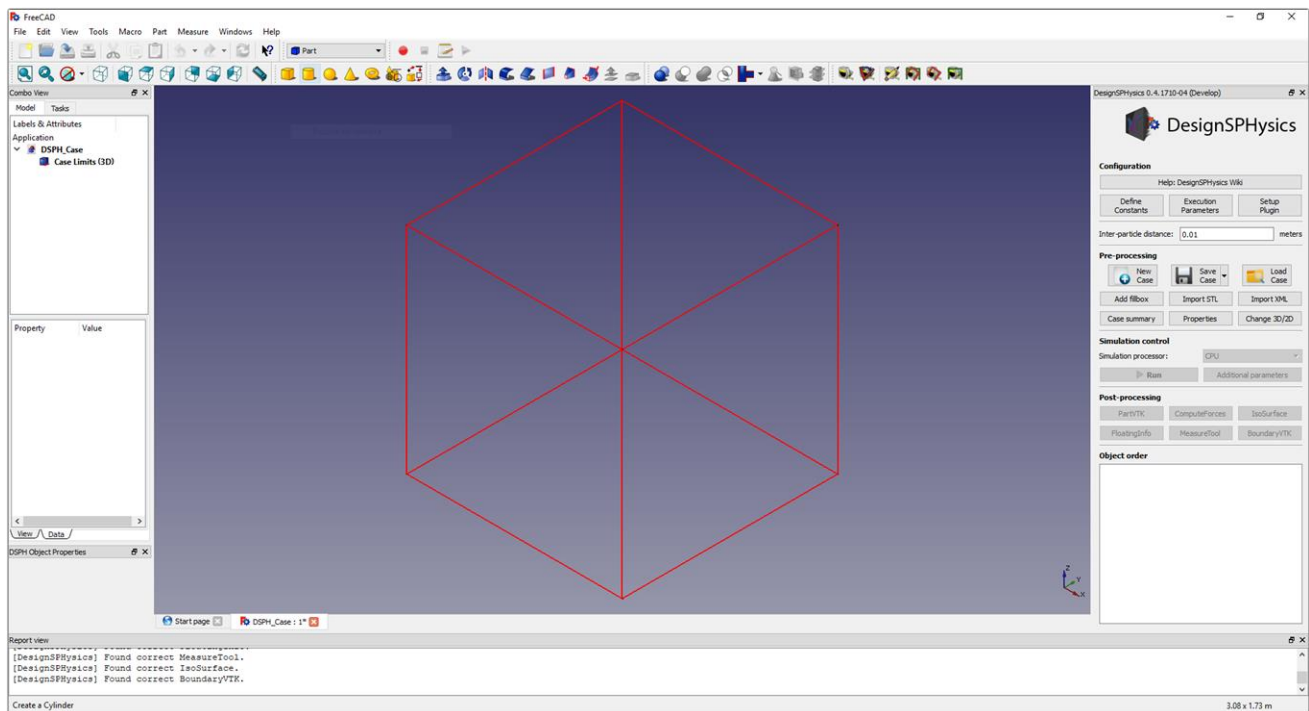
## TO START:

- Unzip **DesignSPHysics\_v0.4.zip** and **Material.zip**
- Open FreeCad
- Navigate to User Macros location and select folder: **DesignSPHysics\_v0.4**  
“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\_v0.4/dualsphysics/EXECS/**GenCase4\_win64.exe**
  - o DualSPHysics Path: DesignSPHysics\_v0.4/dualsphysics/EXECS/**DualSPHysics4\_win64.exe**
  - o PartVTK Path: DesignSPHysics\_v0.4/dualsphysics/EXECS/**PartVTK4\_win64.exe**
  - o ComputeForces Path: DesignSPHysics\_v0.4/dualsphysics/EXECS/**ComputeForces4\_win64.exe**
  - o FloatingInfo Path: DesignSPHysics\_v0.4/dualsphysics/EXECS/**FloatingInfo4\_win64.exe**
  - o MeasureTool Path: DesignSPHysics\_v0.4/dualsphysics/EXECS/**MeasureTool4\_win64.exe**
  - o IsoSurface Path: DesignSPHysics\_v0.4/dualsphysics/EXECS/**IsoSurface4\_win64.exe**
  - o BoundaryVTK Path: DesignSPHysics\_v0.4/dualsphysics/EXECS/**BoundaryVTK4\_win64.exe**
  - o ParaView Path: **C:/Program Files (x86)/ParaView 3.10.1/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=500 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

Import object name: **Tank**

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:



In this case, fluid SPH particles are first created and then the boundaries of the tank and the building.

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

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

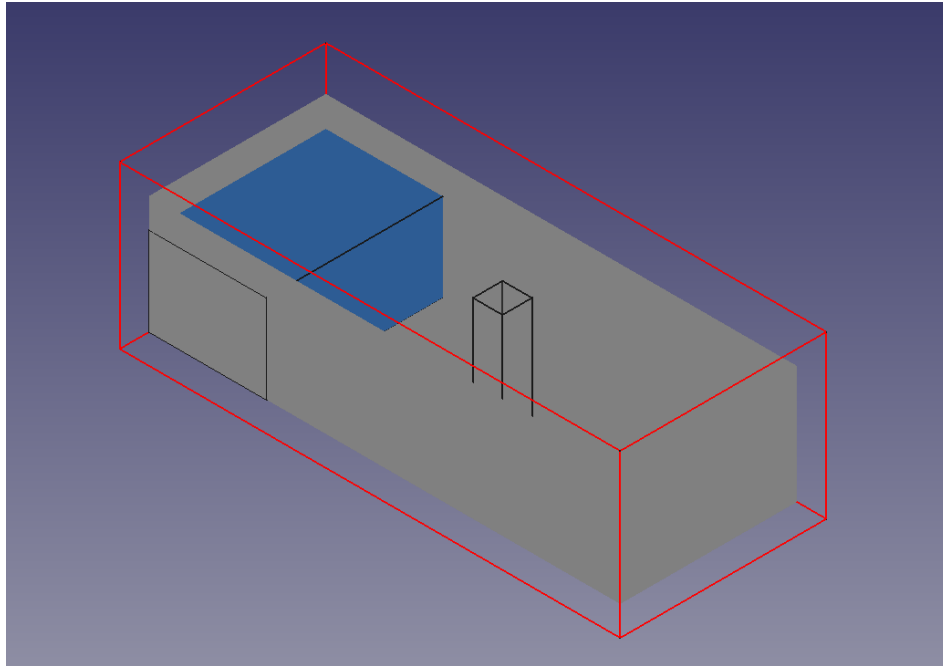
Viscosity value (alpha): 0.05

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: 0.0125 meters**

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 the same name **CaseDambreak3D**
- Creates geometries and files for FreeCad (DSPH\_Case.FCStd & casedata.dsphdata)
- Creates initial XML input file
- Exports STL if needed
- Generate particles (Executes **GenCase** code)

*“GenCase exported 99,981 particles. Press View Details to check the output information”*

1.15. Check the content of **Case summary**

1.16. 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

1.17. Check the particles initially created


**Open Paraview (Start → All Programs → Paraview)**

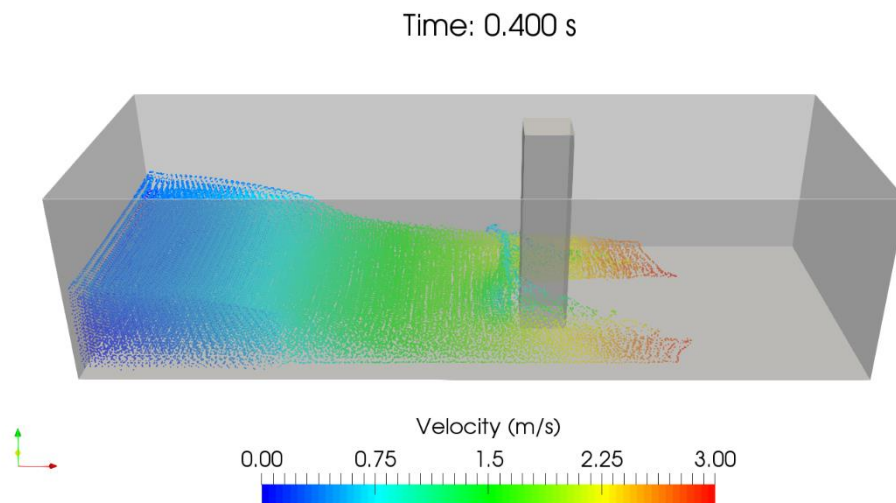
**Use: File → Open “CaseDambreak3D\_All.vtk”.**

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

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

The case is 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 (tells you when it finishes)  
Show **Details** gives information and shows the log file of the execution
- 1.19. When **simulation is complete**:  
Open again the folder **CaseDambreak3D\_Out** and check the content:  
Part\_XXXX.bi4  
PartOut\_000.obt4  
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”. Define Opacity:0.5  
File → Open “PartFluid\_..vtk”  
**Play**  to visualise the simulation



## CASEDAMBREAK2D

2.1. Do not close FreeCad... starting from CaseDambrek3D

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  $\mu$ 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**

*"GenCase exported 5,281 particles. Press View Details to check the output information"*

2.6. Right panel: Simulation control:

Case will be executed using **CPU**

**Run** to start execution of the SPH solver (tells you when it finishes)

Show **Details** gives information and shows the log file of the execution

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**

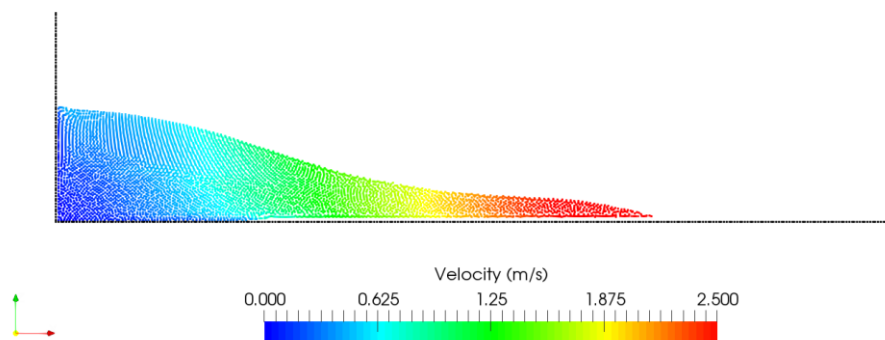
2.8. Visualise the simulation

Open Paraview (Start → All Programs → Paraview)

File → Open "PartAll\_..vtk"

**Play**  to visualise the simulation

Time: 0.350 s



## CASEFLOATINGSPHERE


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:  $x=-2000$  mm,  $y=-2000$  mm,  $z=-50$  mm

Box: Length=4000 mm, Width=4000 mm, Height=5000 mm

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


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

Box: Length=3000 mm, Width=3000 mm, Height=1 mm

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

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

Sphere: Radius=500 mm

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

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

Box: Length=3000 mm, Width=3000 mm, Height=2000 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/m<sup>3</sup>

**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:  $x=-2000$  mm,  $y=0$  mm,  $z=-50$  mm

Box: Length=4000 mm, **Width=1  $\mu$ m**, Height=5000 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

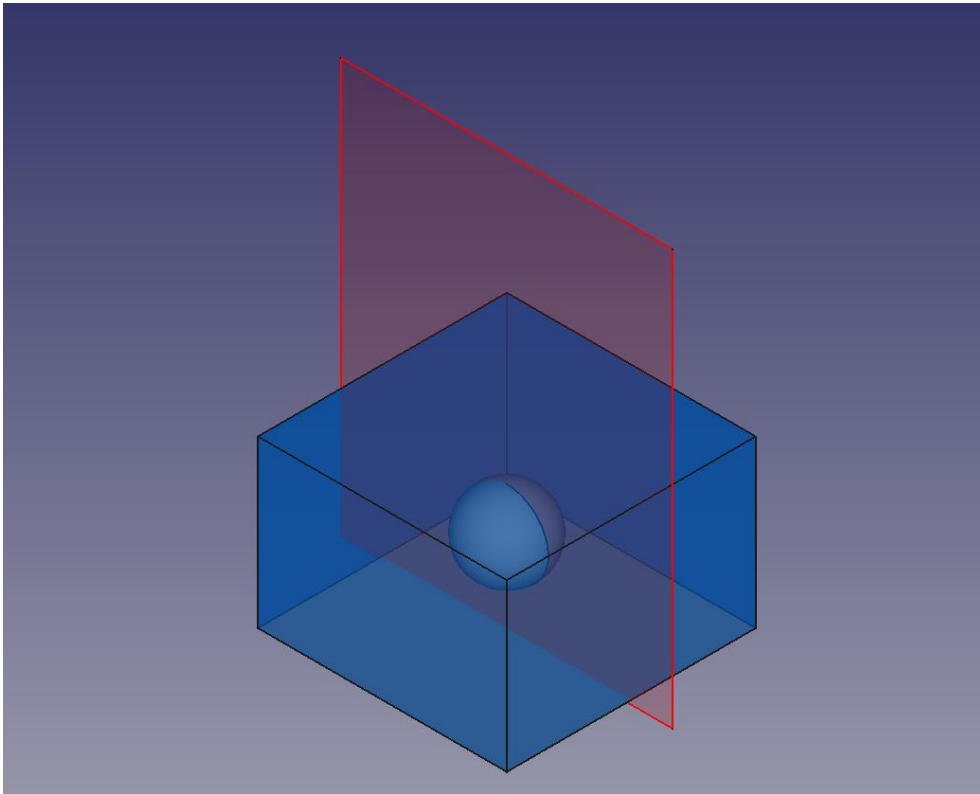
Time of simulation: 6 s; Time out data: 0.02 s

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**

*“GenCase exported 9,801 particles. Press View Details to check the output information”*

3.14. 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:

$$\text{mass}(\text{theory}) = \text{density} * \text{area} = 500 * \pi * \text{radius}^2 = 392.7 \text{ kg}$$

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

$$\text{mass}(\text{SPH}) = 415.313 \text{ kg (in CaseFloatingSphere.xml)}$$

3.15. Check the particles initially created


Open Paraview (Start → All Programs → Paraview)

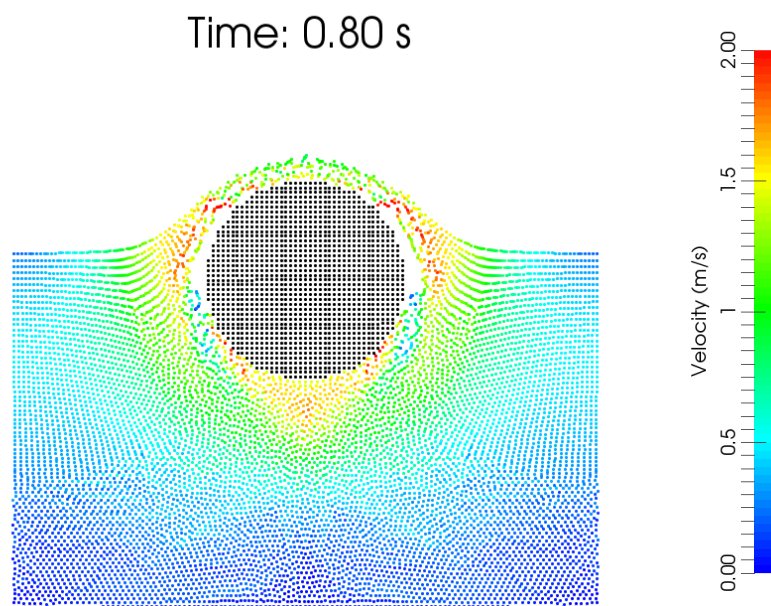
Use: File → Open “CaseFloatingSphere\_All.vtk”.

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

Click on the +Y button on the toolbar

The case 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 (tells you when it finishes)  
 Show **Details** gives information and shows the log file of the execution
- 3.17. When **simulation is complete**:  
 Open again the folder **CaseFloatingSphere\_Out** and check the content:  
 Part\_XXXX.bi4  
 PartOut\_000.obt4  
 Run.out (log file)
- 3.18. Right panel: Post-processing section: **PartVTK**  
 This tool generates files for visualisation  
 Output format: VTK  
 Types to export: Fluid  
 File name: PartFluid  
**Export**
- 3.19. Right panel: 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**)

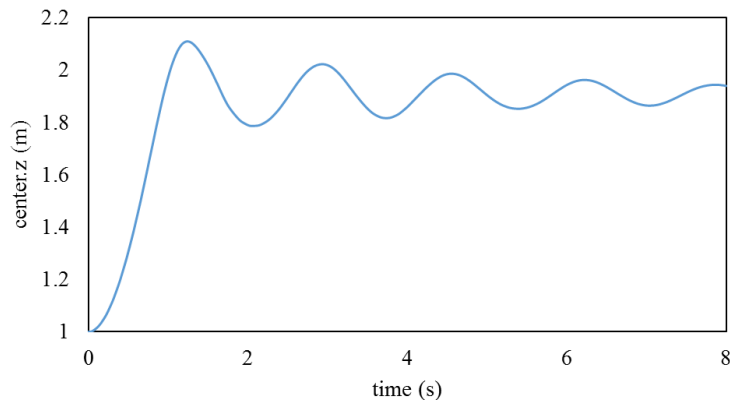
File Name: FloatingMotion

**Export**

3.22. Open “FloatingMotion\_mk12.csv” (from **CaseFloatingSphere\_Out**) with EXCEL

Plot *time* versus *center.z*: time series of the z-position of the center of gravity of the sphere

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



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**)

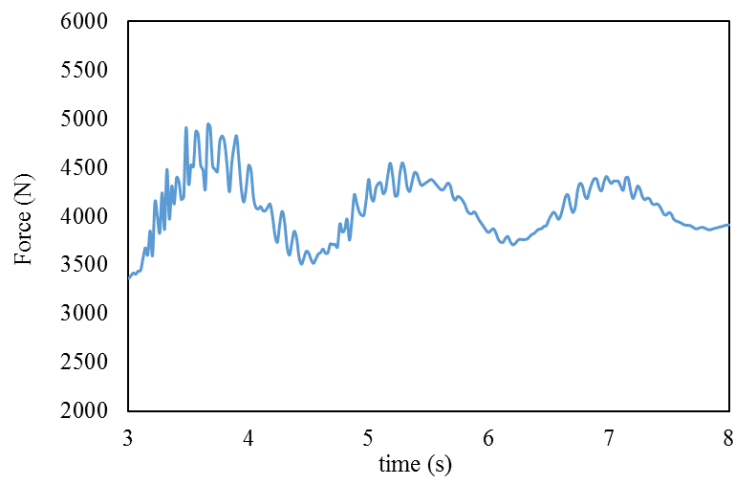
File Name: FloatingForce

**Export**

3.24. Open “FloatingForce.csv” with EXCEL

Plot *time* versus *Forces*: 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 4000 N



## 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=4100 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=4000 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=10 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

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**

Select “Regular wave generator”

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

Wave Order: 2<sup>nd</sup> Order      Depth: 0.4 m

Piston direction: (1,0,0)

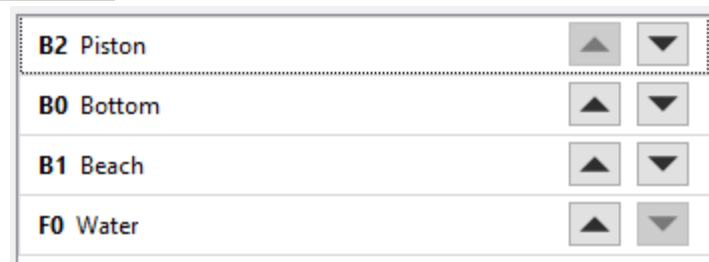
Wave height: 0.1 m, Wave period: 1.2 s

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

Save theoretical values: 10 periods, 20 period steps, xpos:2.5, 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

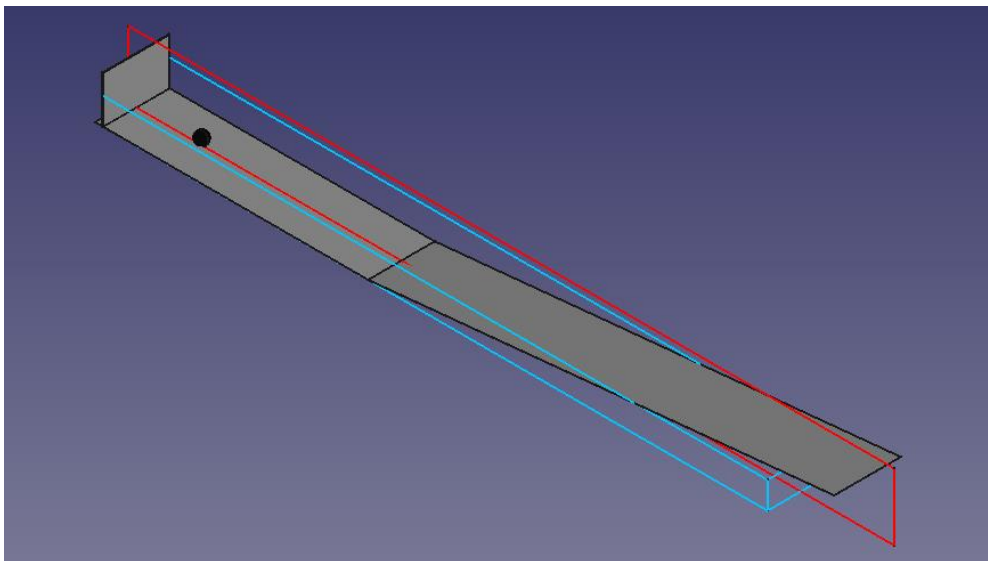
Viscosity factor with boundary: 0

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**

*“GenCase exported 11,138 particles. Press View Details to check the output information”*

4.14. Open the folder **CaseWaves\_Out** and check the content

4.15. Check the particles initially created

Open Paraview (Start → All Programs → Paraview)

Use: File → Open “CaseWaves\_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**

4.16. Right panel: Simulation control:

Case will be executed using **CPU**

**Run** to start execution of the SPH solver (tells you when it finishes)

Show **Details** gives information and shows the log file of the execution

4.17. When **simulation is complete**:

Open again the folder **CaseWaves\_Out** and check the content

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

This tool generates files for visualisation

Output format: VTK

Types to export: Fluid

File name: PartFluid

**Export**

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

This tool generates files for visualisation

Output format: VTK

Types to export: Moving

File name: PartPiston


**Export**

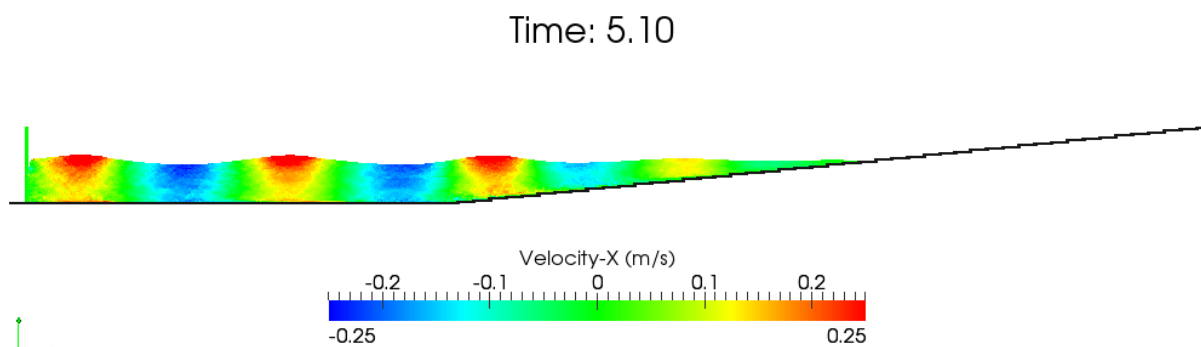
4.20. Visualise the simulation

Open Paraview (Start → All Programs → Paraview)

File → Open “CaseWaves\_Bound.vtk”

File → Open “PartFluid\_..vtk” & File → Open “PartPiston\_..vtk”

**Play**  to visualise the simulation



4.21. 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
3.0	0.0	0.0	1	1	0.001	1.0	1.0	4000	3.0	0.0	3.0

File name: SPH\_WG

**Export**

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

Output format: CSV

Variables to export: Velocity

**List of points:**

X	Y	Z
3.0	0.0	0.25

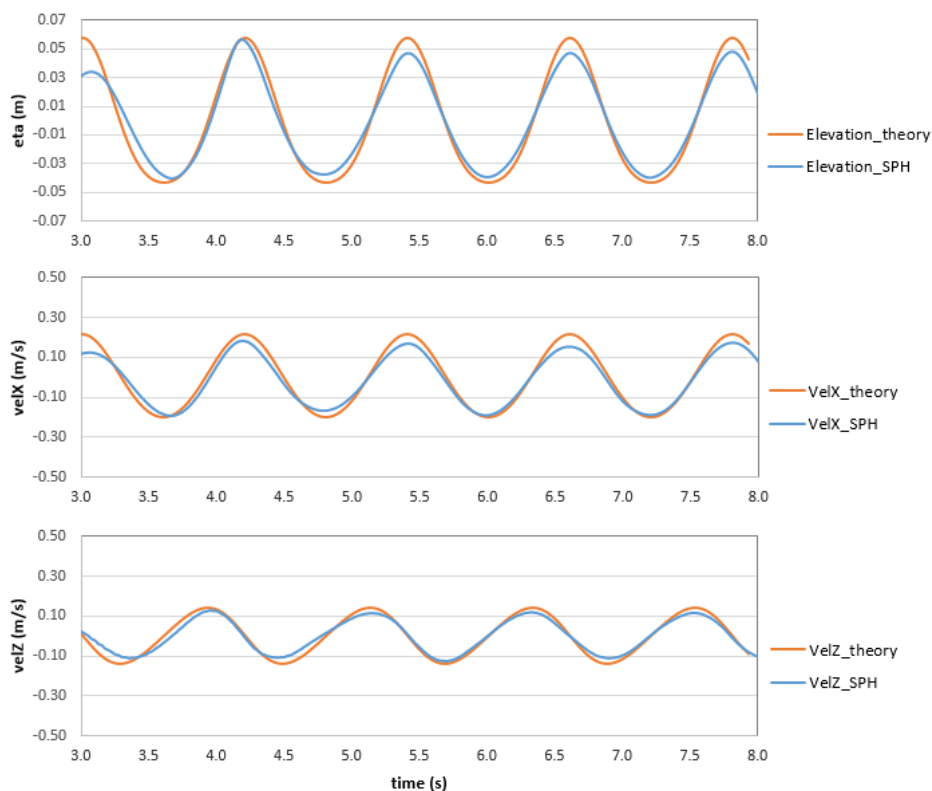
File name: SPH\_VG

**Export**

4.23. Open “CaseWaves\_Validation\_Theory.xls”


and paste SPH results (SPH\_WG\_Height.csv and SPH\_VG\_Vel.csv) in the shadow area.

(Theoretical results were taken from: WavePaddle\_mkb0002.csv)



## CASEWAVESFLOATING

5.1. Do not close FreeCad... starting from CaseWaves

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

5.2. 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

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

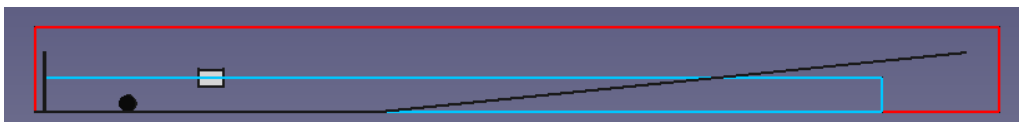
<b>B2</b> Piston	▲	▼
<b>B0</b> Bottom	▲	▼
<b>B1</b> Beach	▲	▼
<b>B3</b> Floater	▲	▼
<b>F0</b> Water	▲	▼

5.4. Right panel: Configuration: **Define Constants**: same as before

Right panel: Configuration: **Execution Parameters**: same as before

Right panel: Configuration: **Inter-particle distance: 0.015 meters** same as before

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



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

Name of the case: **CaseWavesFloating**

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

*“GenCase exported 11,252 particles. Press View Details to check the output information”*

5.7. Check the particles initially created

Open Paraview (Start → All Programs → Paraview)

Use: File → Open “CaseWavesFloating\_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**



5.8. Right panel: Simulation control:

Case will be executed using **CPU**

**Run** to start execution of the SPH solver (tells you when it finishes)

Show **Details** gives information and shows the log file of the execution

5.9. When **simulation is complete**:

Open again the folder **CaseWavesFloating\_Out** and check the content

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

This tool generates files for visualisation

Output format: VTK

Types to export: Fluid

File name: PartFluid

**Export**

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

This tool generates files for visualisation

Output format: VTK

Types to export: Moving

File name: PartPiston

**Export**

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

This tool generates files for visualisation

Output format: VTK

Types to export: Floating

File name: PartFloating

**Export**

5.13. Visualise the simulation

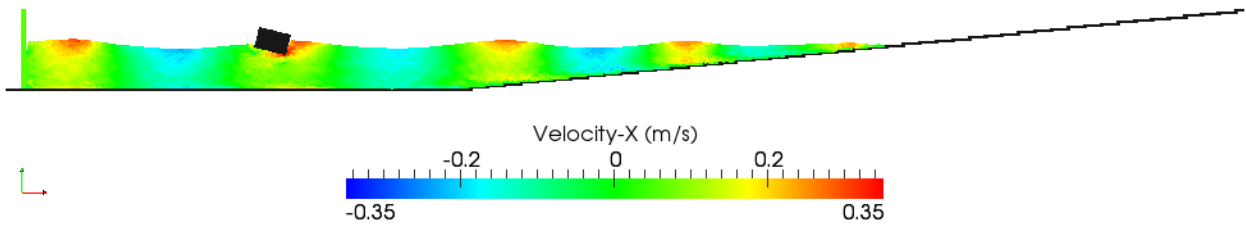
Open Paraview (Start → All Programs → Paraview)

File → Open “CaseWavesFloating\_Bound.vtk”

File → Open “PartFluid\_..vtk” & “PartPiston\_..vtk” & “PartFloating\_..vtk”

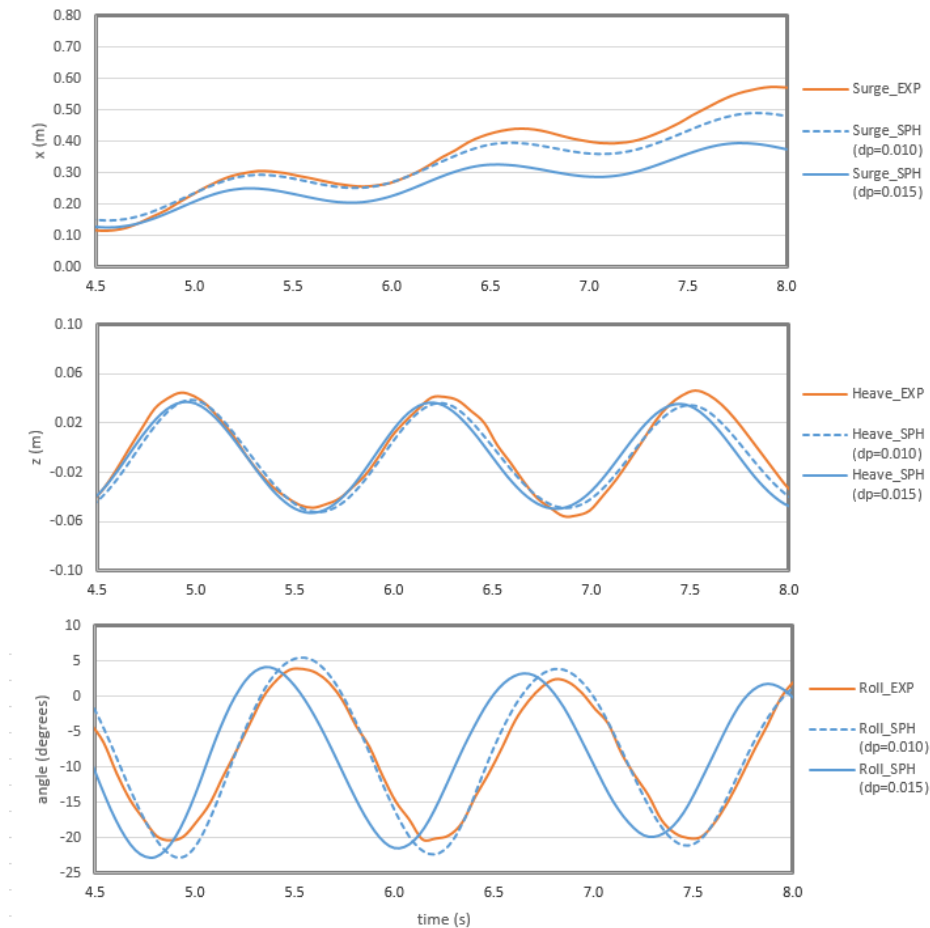
**Play**  to visualise the simulation

Time: 6.25



- 5.14. 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**

- 5.15. Open “CaseWavesFloating\_Validation\_EXP.xls” and paste SPH results (surge, heave and roll) from “FloatingMotion\_mk14.csv” in the shadow area.



## CASEWAVETANK

6.1. Open FreeCad and load macro

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

6.3. 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

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


Import STL options:

STL File: material/CaseWaveTank\_structure.stl

Scaling factor X:1000; Y:1000; Z:1000


Import object name: **Structure**

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

6.5. 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

6.6. “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

6.7. 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

6.8. 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**

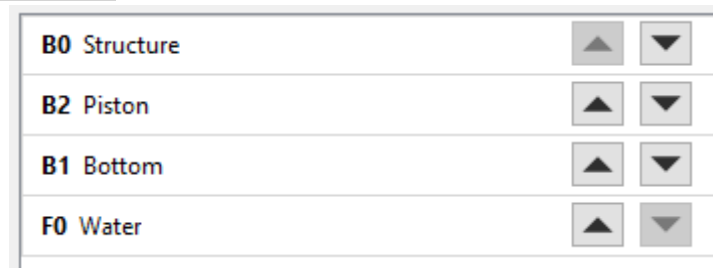
Select “Movement from a file”

File name: material/CaseWaveTank\_movement.dat

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

Duration: 25 s

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



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

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

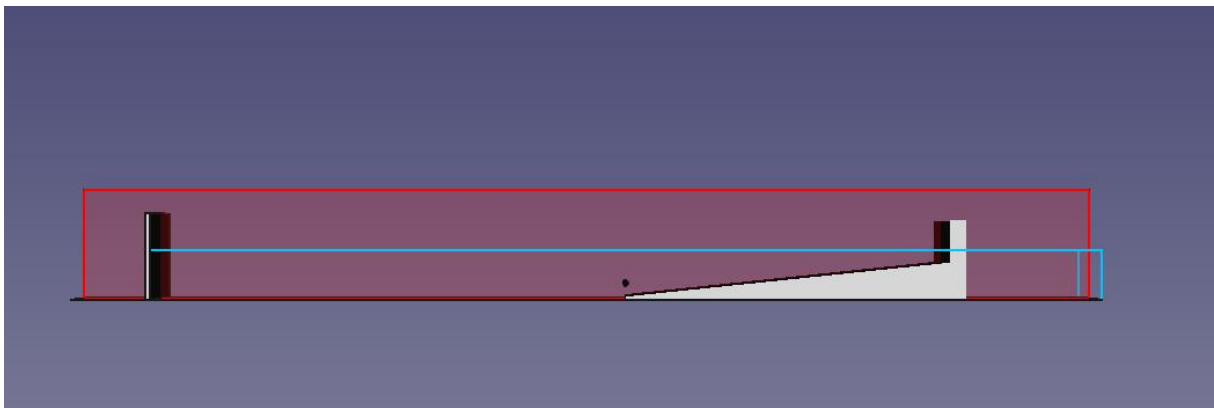
Step Algorithm: Symplectic

Viscosity value (alpha): 0.02

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

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

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



6.13. 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  $\mu$ m**, Height=700 mm

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

Name of the case: **CaseWaveTank**

*“GenCase exported 13,649 particles. Press View Details to check the output information”*

6.15. Open the folder **CaseWaveTank\_Out** and check the content

6.16. Check the particles initially created

Open Paraview (Start → All Programs → Paraview)

Use: File → Open “CaseWaveTank\_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**

6.17. Right panel: Simulation control:

Case will be executed using **CPU**

**Run** to start execution of the SPH solver (tells you when it finishes)

Show **Details** gives information and shows the log file of the execution

6.18. When **simulation is complete**:

Open again the folder **CaseWaveTank\_Out** and check the content

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

This tool generates files for visualisation

Output format: VTK

Types to export: Fluid

File name: PartFluid

**Export**

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

This tool generates files for visualisation

Output format: VTK

Types to export: Moving

File name: PartPiston


**Export**

6.21. Visualise the simulation

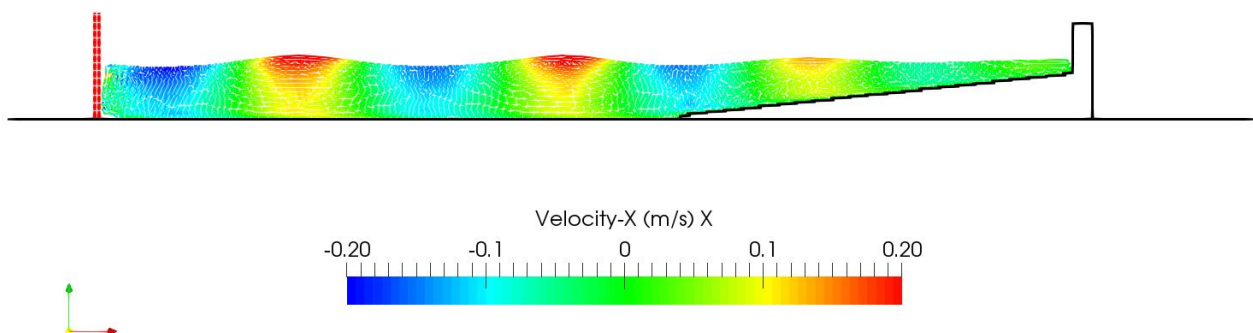
Open Paraview (Start → All Programs → Paraview)

File → Open “CaseWaveTank\_Bound.vtk”

File → Open “PartFluid\_..vtk” & File → Open “PartPiston\_..vtk”

**Play**  to visualise the simulation

Time: 2.80 s



6.22. 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

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	4000	<b>0.36</b>	<b>0.2</b>	<b>3.2</b>

File name: SPH\_wg1

**Export**

6.23. 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)

Use: File → Open “SPH\_WG\_Height\_..vtk” & File → Open “SPH\_WG\_Mass\_..vtk”

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

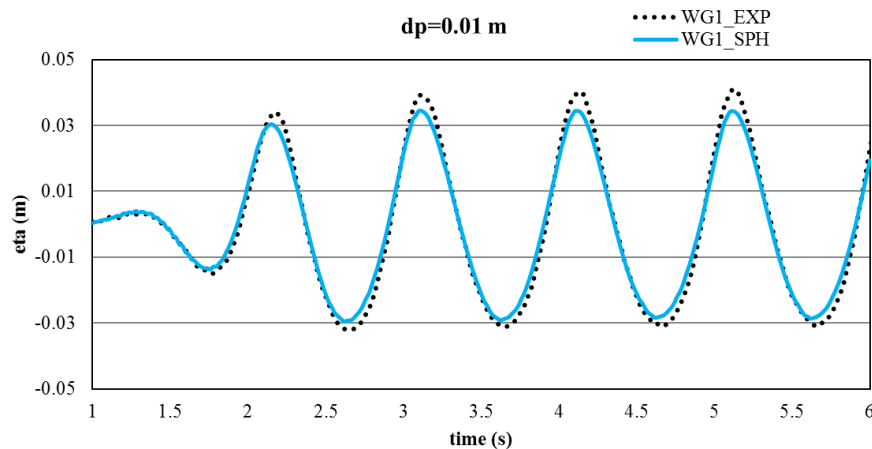
Click on the **+Y** button on the toolbar

6.24. Open “SPH\_wg1\_Height.csv” with EXCEL

Plot *Time* versus *Height\_0*: time series of the water elevation at wg1 (x=0.36 m)

Open “material/CaseWaveTank\_Validation\_EXP.xls”

and paste SPH results in the shadow area



6.25. 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: **11** (check the value of the structure in **Case summary**)

File Name: StructureForce

**Export**

6.26. Open “StructureForce.csv” with EXCEL and

Plot *time* versus *Forces*: time series of the force exerted onto the structure (includes initial hydrostatic force).

## CASESLOSHINGTANK

7.1. Open FreeCad and load macro


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

7.3. 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

7.4. “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

7.5. “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

7.6. 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**

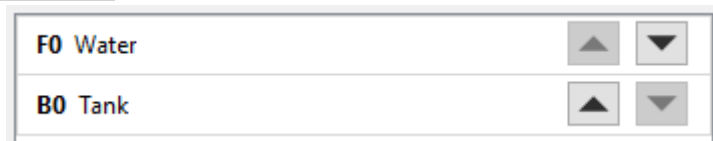
“Add a sinusoidal rotational motion”

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

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



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

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

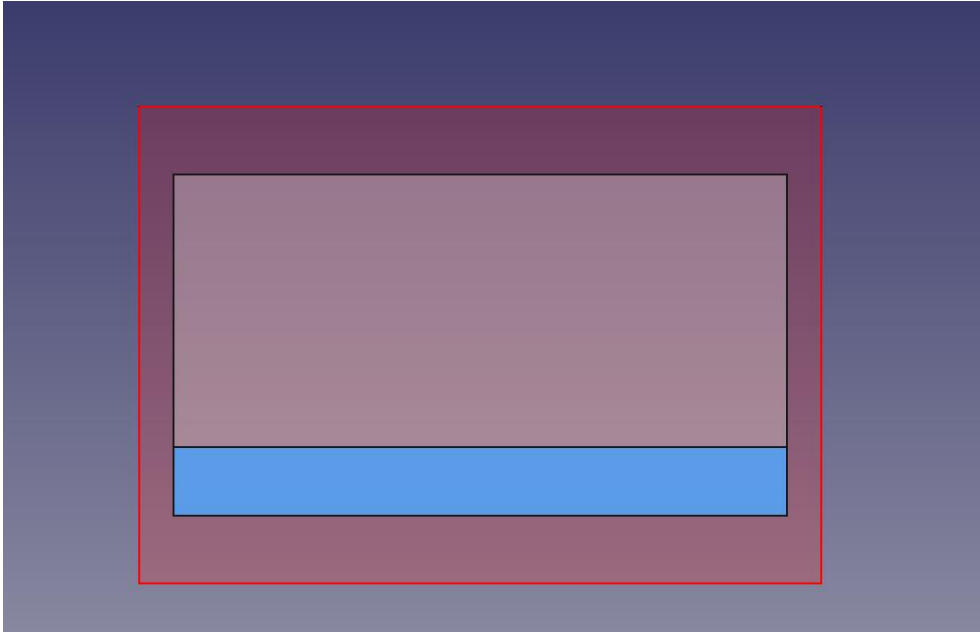
Step Algorithm: Symplectic

Viscosity value (alpha): 0.05

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

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

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



- 7.11. Right panel: Pre-processing section: Save Case: **Save and run GenCase**  
Name of the case: **CaseSloshingTank**  
*“GenCase exported 23,850 particles. Press View Details to check the output information”*
- 7.12. 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 |
- 7.13. Check the particles initially created  
**Open Paraview (Start → All Programs → Paraview)**  
**Use: File → Open “CaseSloshingTank\_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 almost ready to be **simulated**

- 7.14. Right panel: Simulation control:  
Case will be executed using **CPU**  
**Additional parameters: -domain\_fixed:-0.55:-0.1:-0.1:0.55:0.1:0.6 (Save Case)**  
**Domain limits need to be changed to allow boundary particles to move**  
**Run** to start execution of the SPH solver (tells you when it finishes)  
Show **Details** gives information and shows the log file of the execution



7.15. When **simulation is complete**:

Open again the folder **CaseSloshingTank\_Out** and check the content:

Part\_XXXX.bi4

PartOut\_000.obt4

Run.out (log file)

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

This tool generates files for visualisation

Output format: VTK

Types to export: All


File name: PartAll

**Export**

7.17. Visualise the simulation

Open Paraview (Start → All Programs → Paraview)

File → Open “PartAll .vtk”

**Play**  to visualise the simulation

