XML GUIDE FOR DUALSPHYSICS

Create your own case using the XML file

July 2020

DualSPHysics team

http://dual.sphysics.org

INDEX

1. Input & output file
1.1. Work flow
1.2. Format files

2. XML for GenCase
2.1. Main structure
2.2. SPH constants
2.3. MK configuration
2.4. Geometry
2.4.1. Domain limits
2.4.2. Drawing commands
- Command lists
- Drawing settings
- VTK output
- MK selection
- Transformations
- Draw lines
- Draw triangles
- Draw boxes
- Draw polyhedrons
- Draw rounded shapes
- External geometries
- Filling domains
- Redraw commands
- Freedraw mode
- Layers
- Variables

2.5. Initial transformations
2.5.1. Velocity
2.5.2. Position
2.6. Floating objects
2.7. Properties for MK
2.8. Motion
2.8.1. Simple rectilinear
2.8.2. Combined motions
2.8.3. Hierarchical motions
2.8.4. Accelerated rectilinear
2.8.5. Rotational
2.8.6. Circular
2.8.7. Sinusoidal
2.8.8. External motion

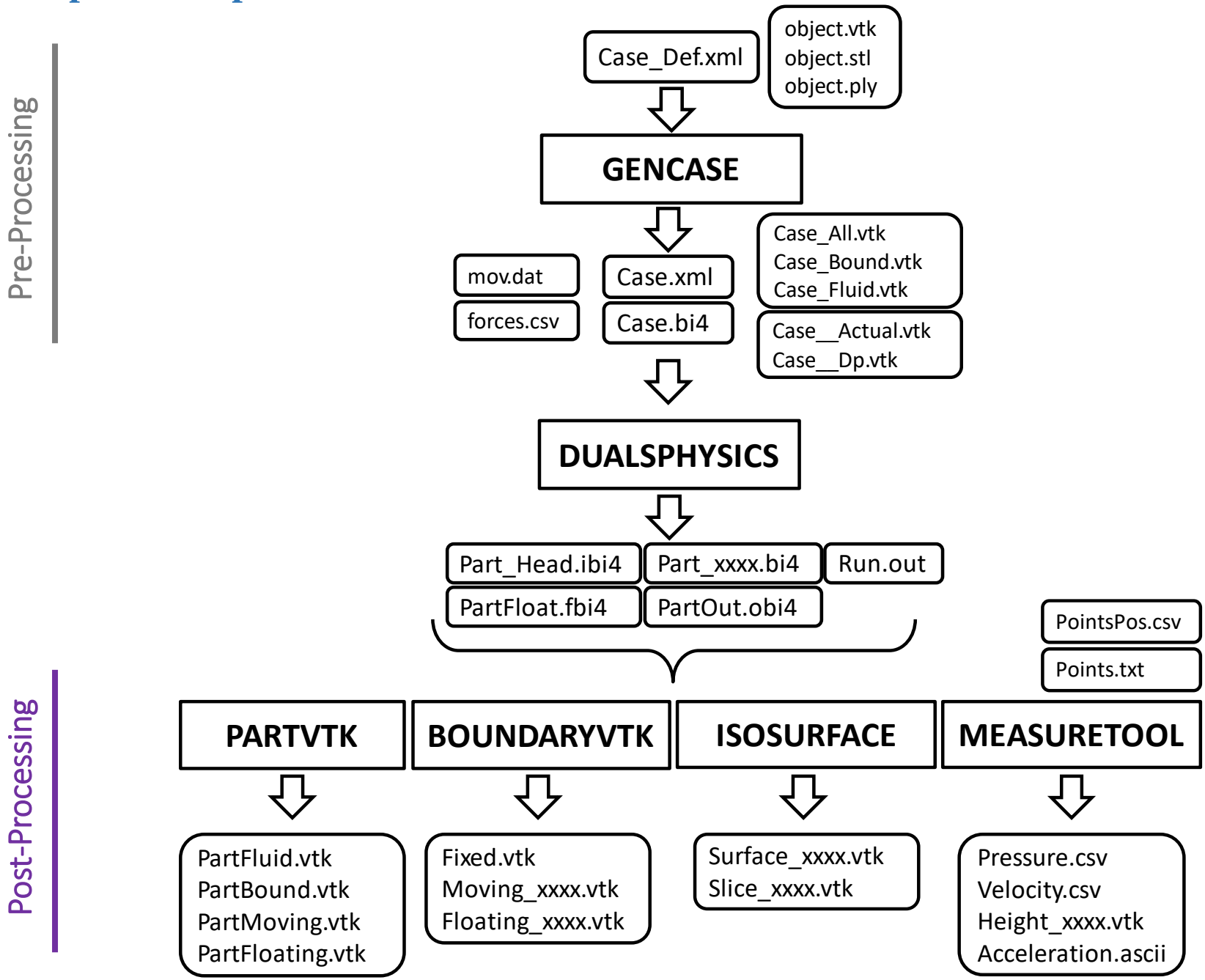
3. XML for DualSPHysics
3.1. Special features
3.1.1. Paddle wave generation
- Regular waves using piston
- Irregular waves using piston
- Regular waves using flap
- Irregular waves using flap
- Solitary waves using piston

3.1.2. Active absorption
3.1.3. Passive absorption
3.1.4. Multi-Layer pistons
3.1.5. Relaxation zone
- Regular & irregular waves
- External velocity (coupling)
- Uniform velocity profile

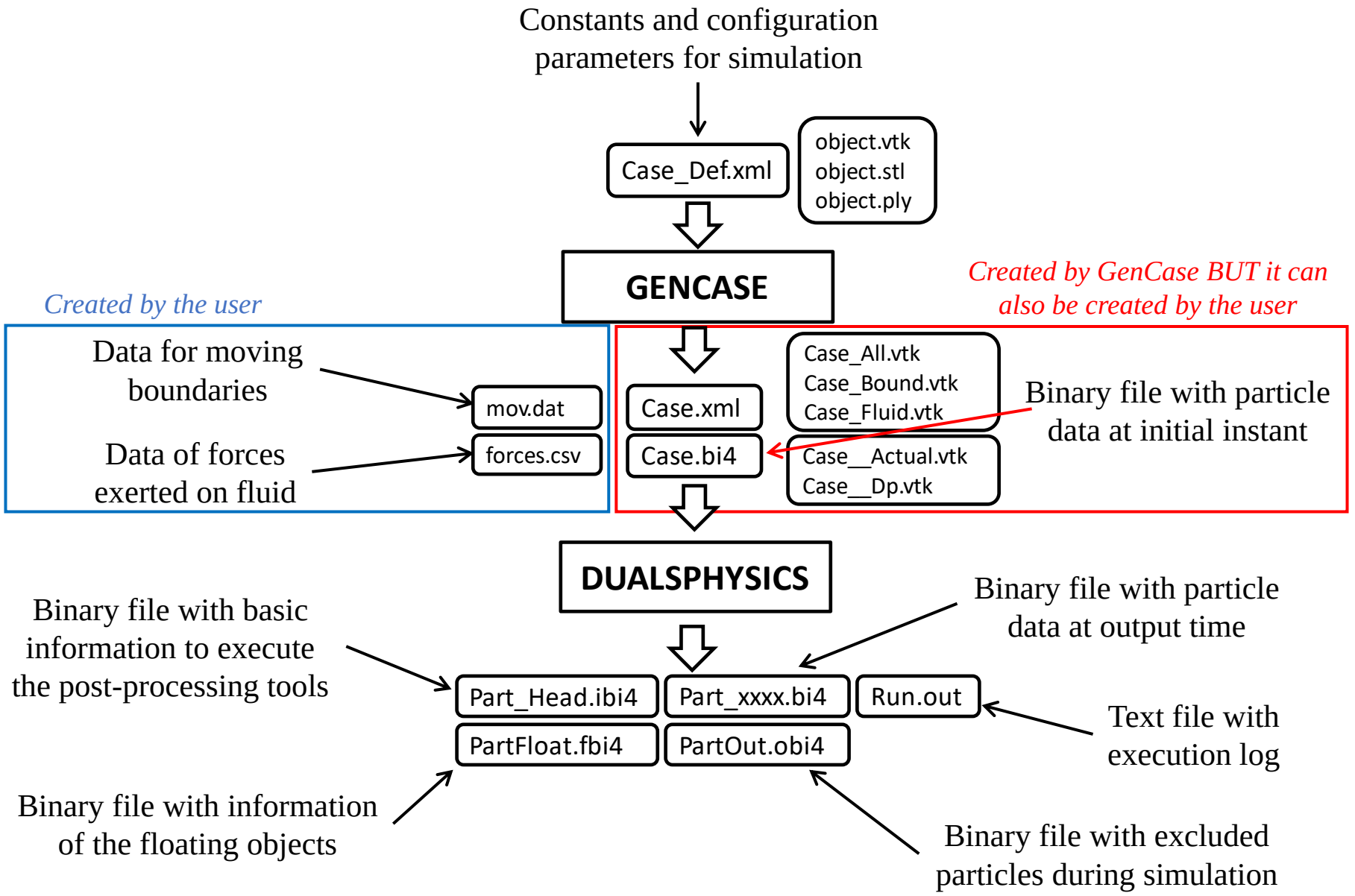
3.1.6. Gauge system
3.1.7. Imposed acceleration
3.1.8. Saves simulation Dt
3.1.9. Variable output data
3.2. Execution parameters
3.2.1. Double precision
3.2.2. Time integration scheme
3.2.3. Kernel function
3.2.4. Viscosity treatment
3.2.5. Density Diffusion Term
3.2.6. Shifting algorithm
3.2.7. Interaction between solids
3.2.8. Time step computation
3.2.9. Simulation time
3.2.10. Excluding particles
3.2.11. Simulation domain
3.2.12. Periodic conditions
3.3. Particle information
3.4. Constants for simulation

4. Simulation output
4.1. Log file
4.2. Other files

1.1. Input & output files: Work flow



1.1. Input & output files: Work flow



1.2. Input & output files: Format files

Case_Def.xml

Case.xml

XML File

- The eXtensible Markup Language is textual data format compatible with any hardware and software.
- Information is structured and organised by using labels.
- They can be easily edited using any text editor.

Case.bi4

Part_Head.ibi4
Part_xxxx.bi4
PartOut.obi4
PartFloat.fbi4

BINARY File

- Binary format consumes six times less memory than text format.
- Reading or writing is several times faster using a binary format.
- A special code is required to read the data (JPartDataBi4.cpp/.h).
- ".bi4" is the new binary format that also includes double precision.
- The user can also define new arrays that post-processing tools can automatic manage.

Particles_xxxx.vtk

Velocity_xxxx.vtk
Height_xxxx.vtk

Surface_xxxx.vtk
Slices_xxxx.vtk

VTK File

- VTK (Visualization ToolKit) files are used for final visualization
- These files are generated as a pre-processing step by GenCase or as output by DualSPHysics and post-processing tools.
- VTK files of POLYDATA type with legacy-binary format is used.
- VTK format is supported for many visualization applications, such as ParaView that is open-source and multi-platform .

2. XML file

Case_Def.xml → **GENCASE** → Case.xml

```
<case>
    <casedef>
        <constantsdef>
            <lattice bound="1" fluid="1" />
            <gravity x="0" y="0" z="-9.81" comment="Gravitational acceleration" units_comment="m/s^2" />
            <cflnumber value="0.2" comment="Coefficient to multiply Dt" />
            <hswl value="0" auto="true" comment="Maximum still water level to calculate speedofsound using coefsound" units_comment="metres (m)" />
            <speedsystem value="0" auto="true" comment="Maximum system speed (by default the dam-break propagation is used)" />
            <coefsound value="20" comment="Coefficient to multiply speedsystem" />
            <speedsound value="0" auto="true" comment="Speed of sound to use in the simulation (by default speedofsound=coefsound*speedsystem)" />
            <coefh value="1.0" comment="Coefficient to calculate the smoothing length (H=coefficient*sqrt(3*dp^2) in 3D)" />
            <gamma value="7" comment="Politropic constant for water used in the state equation" />
            <rhop0 value="1000" comment="Reference density of the fluid" units_comment="kg/m3" />
        </constantsdef>
        <mkconfig boundcount="240" fluidcount="10" />
        <geometry>
            <definition dp="0.01" units_comment="metres (m)">
                <pointmin x="-1" y="0" z="-1" />
                <pointmax x="4.5" y="0" z="3.5" />
            </definition>
            <commands>
                <mainlist>
                    <setdrawmode mode="full" />
                    <setmkfluid mk="0" />
                    <drawbox>
                        <boxfill>solid</boxfill>
                        <point x="0" y="-1" z="0" />
                        <size x="1" y="2" z="2" />
                    </drawbox>
                    <setmkbound mk="0" />
                    <drawbox>
                        <boxfill>bottom | left | right | front | back</boxfill>
                        <point x="0" y="-1" z="0" />
                        <size x="4" y="2" z="3" />
                    </drawbox>
                </mainlist>
            </commands>
        </geometry>
    </casedef>
    <execution>
        <parameters>
            <parameter key="StepAlgorithm" value="1" comment="Step Algorithm 1:Verlet, 2:Symplectic (default=1)" />
            <parameter key="VerletSteps" value="40" comment="Verlet only: Number of steps to apply Euler timestepping (default=40)" />
            <parameter key="Kernel" value="2" comment="Interaction Kernel 1:Cubic Spline, 2:Wendland (default=2)" />
            <parameter key="ViscoTreatment" value="1" comment="Viscosity formulation 1:Artificial, 2:Laminar+SPS (default=1)" />
            <parameter key="Visco" value="0.02" comment="Viscosity value" />
            <parameter key="ViscoBoundFactor" value="1" comment="Multiply viscosity value with boundary (default=1)" />
            <parameter key="DeltaSPH" value="0" comment="DeltaSPH value, 0.1 is the typical value, with 0 disabled (default=0)" />
            <parameter key="#Shifting" value="0" comment="Shifting mode 0:None, 1:Ignore bound, 2:Ignore fixed, 3:Full (default=0)" />
            <parameter key="#ShiftCoef" value="-2" comment="Coefficient for shifting computation (default=-2)" />
            <parameter key="#ShiftTFS" value="1.5" comment="Threshold to detect free surface. Typically 1.5 for 2D and 2.75 for 3D (default=0)" />
            <parameter key="RigidAlgorithm" value="1" comment="Rigid Algorithm 1:SPH, 2:DEM (default=1)" />
            <parameter key="FtPause" value="0.0" comment="Time to freeze the floatings at simulation start (warmup) (default=0)" units_comment="seconds" />
            <parameter key="CoefDtMin" value="0.05" comment="Coefficient to calculate minimum time step dtmin=coefdtemin*h/speedsound (default=0.05)" />
            <parameter key="#DtIni" value="0.0001" comment="Initial time step (default=h/speedsound" units_comment="seconds" />
            <parameter key="#DtMin" value="0.00001" comment="Minimum time step (default=coefdtmin*h/speedsound)" units_comment="seconds" />
            <parameter key="#DtFixed" value="DtFixed.dat" comment="Dt values are loaded from file (default=disabled)" />
            <parameter key="DtAllParticles" value="0" comment="Velocity of particles used to calculate DT. 1:All, 0:Only fluid/floating (default=0)" />
            <parameter key="TimeMax" value="0.72" comment="Time of simulation" units_comment="seconds" />
            <parameter key="TimeOut" value="0.01" comment="Time out data" units_comment="seconds" />
            <parameter key="IncZ" value="1" comment="Increase of Z+" units_comment="decimal" />
            <parameter key="PartsOutMax" value="1" comment="Allowed %/100 of fluid particles out the domain (default=1)" units_comment="decimal" />
            <parameter key="RhopOutMin" value="700" comment="Minimum rhop valid (default=700)" units_comment="kg/m3" />
            <parameter key="RhopOutMax" value="1300" comment="Maximum rhop valid (default=1300)" units_comment="kg/m3" />
        </parameters>
    </execution>
</case>
```

```
<case>
    <casedef>
        <constantsdef>
            <lattice bound="1" fluid="1" />
            <gravity x="0" y="0" z="-9.81" comment="Gravitational acceleration" units_comment="m/s^2" />
            <cflnumber value="0.2" comment="Coefficient to multiply Dt" />
            <hswl value="0" auto="true" comment="Maximum still water level to calculate speedofsound using coefsound" units_comment="metres (m)" />
            <speedsystem value="0" auto="true" comment="Maximum system speed (by default the dam-break propagation is used)" />
            <coefsound value="20" comment="Coefficient to multiply speedsystem" />
            <speedsound value="0" auto="true" comment="Speed of sound to use in the simulation (by default speedofsound=coefsound*speedsystem)" />
            <coefh value="1.0" comment="Coefficient to calculate the smoothing length (H=coefficient*sqrt(3*dp^2) in 3D)" />
            <gamma value="7" comment="Politropic constant for water used in the state equation" />
            <rhop0 value="1000" comment="Reference density of the fluid" units_comment="kg/m3" />
        </constantsdef>
        <mkconfig boundcount="240" fluidcount="10" />
        <geometry>
            <definition dp="0.01" units_comment="metres (m)">
                <pointmin x="-1" y="0" z="-1" />
                <pointmax x="4.5" y="0" z="3.5" />
            </definition>
            <commands>
                <mainlist>
                    <setdrawmode mode="full" />
                    <setmkfluid mk="0" />
                    <drawbox>
                        <boxfill>solid</boxfill>
                        <point x="0" y="-1" z="0" />
                        <size x="1" y="2" z="2" />
                    </drawbox>
                    <setmkbound mk="0" />
                    <drawbox>
                        <boxfill>bottom | left | right | front | back</boxfill>
                        <point x="0" y="-1" z="0" />
                        <size x="4" y="2" z="3" />
                    </drawbox>
                </mainlist>
            </commands>
        </geometry>
    </casedef>
    <execution>
        <parameters>
            <parameter key="StepAlgorithm" value="1" comment="Step Algorithm 1:Verlet, 2:Symplectic (default=1)" />
            <parameter key="VerletSteps" value="40" comment="Verlet only: Number of steps to apply Euler timestepping (default=40)" />
            <parameter key="Kernel" value="2" comment="Interaction Kernel 1:Cubic Spline, 2:Wendland (default=2)" />
            <parameter key="ViscoTreatment" value="1" comment="Viscosity formulation 1:Artificial, 2:Laminar+SPS (default=1)" />
            <parameter key="Visco" value="0.02" comment="Viscosity value" />
            <parameter key="ViscoBoundFactor" value="1" comment="Multiply viscosity value with boundary (default=1)" />
            <parameter key="DeltaSPH" value="0" comment="DeltaSPH value, 0.1 is the typical value, with 0 disabled (default=0)" />
            <parameter key="#Shifting" value="0" comment="Shifting mode 0:None, 1:Ignore bound, 2:Ignore fixed, 3:Full (default=0)" />
            <parameter key="#ShiftCoef" value="-2" comment="Coefficient for shifting computation (default=-2)" />
            <parameter key="#ShiftTFS" value="1.5" comment="Threshold to detect free surface. Typically 1.5 for 2D and 2.75 for 3D (default=0)" />
            <parameter key="RigidAlgorithm" value="1" comment="Rigid Algorithm 1:SPH, 2:DEM (default=1)" />
            <parameter key="FtPause" value="0.0" comment="Time to freeze the floatings at simulation start (warmup) (default=0)" units_comment="seconds" />
            <parameter key="CoefDtMin" value="0.05" comment="Coefficient to calculate minimum time step dtmin=coefdtemin*h/speedsound (default=0.05)" />
            <parameter key="#DtIni" value="0.0001" comment="Initial time step (default=h/speedsound" units_comment="seconds" />
            <parameter key="#DtMin" value="0.00001" comment="Minimum time step (default=coefdtmin*h/speedsound)" units_comment="seconds" />
            <parameter key="#DtFixed" value="DtFixed.dat" comment="Dt values are loaded from file (default=disabled)" />
            <parameter key="DtAllParticles" value="0" comment="Velocity of particles used to calculate DT. 1:All, 0:Only fluid/floating (default=0)" />
            <parameter key="TimeMax" value="0.72" comment="Time of simulation" units_comment="seconds" />
            <parameter key="TimeOut" value="0.01" comment="Time out data" units_comment="seconds" />
            <parameter key="IncZ" value="1" comment="Increase of Z+" units_comment="decimal" />
            <parameter key="PartsOutMax" value="1" comment="Allowed %/100 of fluid particles out the domain (default=1)" units_comment="decimal" />
            <parameter key="RhopOutMin" value="700" comment="Minimum rhop valid (default=700)" units_comment="kg/m3" />
            <parameter key="RhopOutMax" value="1300" comment="Maximum rhop valid (default=1300)" units_comment="kg/m3" />
        </parameters>
        <particles np="21001" nb="1001" nbf="1001" mkboundfirst="11" mkfluidfirst="1">
            <fixed mkbound="0" mk="11" begin="0" count="1001" />
            <fluid mkfluid="0" mk="1" begin="1001" count="20000" />
        </particles>
        <constants>
            <gravity x="0" y="0" z="-9.81" units_comment="m/s^2" />
            <cflnumber value="0.2" />
            <gamma value="7" />
            <rhop0 value="1000" units_comment="kg/m3" />
            <dp value="0.01" units_comment="metres (m)" />
            <h value="1.4142135624E-002" units_comment="metres (m)" />
            <b value="1.1155371429E+006" units_comment="metres (m)" />
            <massbound value="1.0000000000E-001" units_comment="kg" />
            <massfluid value="1.0000000000E-001" units_comment="kg" />
        </constants>
        <motion />
    </execution>
</case>
```

2.1. XML file: Main structure

Divided in two sections:

“casedef”

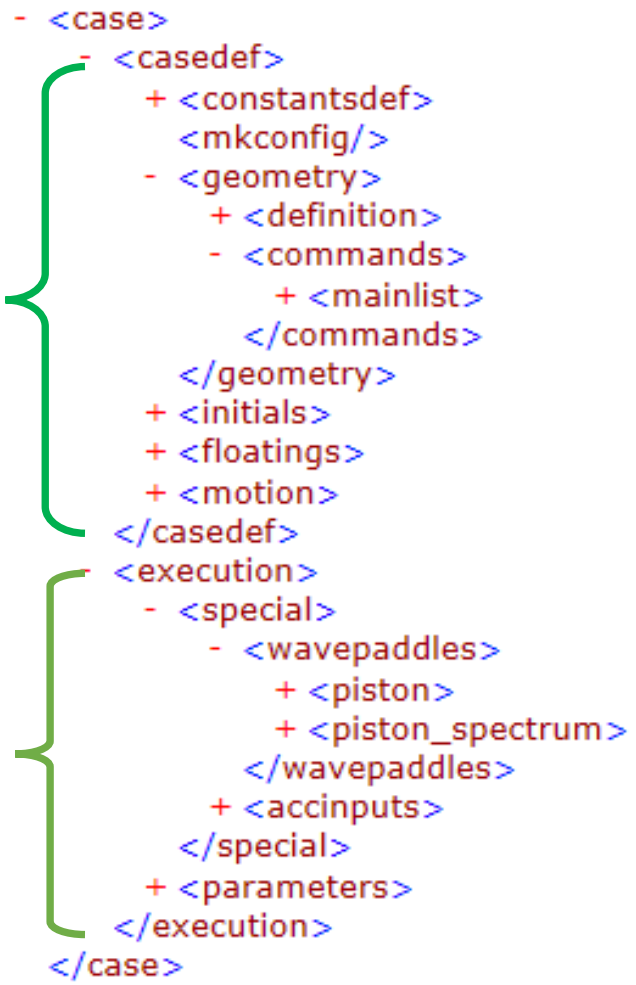
Definition of the case with initial geometry and configuration.

Created by the user and used by GenCase

“execution”

Information required to execute the case.

Created by the user, modified by GenCase and only used by DualSPHysics



2.1. XML file: Main structure

- **"casedef" :**

  - constantsdef constants needed in SPH

  - mkconfig label configuration

  - geometry system geometry (boundaries and fluid)
    - **definition**
    - **commands (list & mainlist)**

  - initials special features for boundary and fluid particles

  - floatings description of floating objects

  - motion description of boundary movement

- **"execution"**

  - special automatic wave generation and external forces
    - **wavepaddles (piston & piston_spectrum)**
    - **accinputs**

  - parameters execution parameters in DualSPHysics

2.2. XML file: Casedef-ConstantsDef


```
<constantsdef>
    <lattice bound="2" fluid="1" />
    <gravity x="0" y="0" z="-9.81" comment="Gravitational acceleration" units_comment="m/s^2" />
    <rhop0 value="1000" comment="Reference density of the fluid" units_comment="kg/m^3" />
    <hswl value="0" auto="true" comment="Maximum still water level to calculate speedofsound using coefsound"
                                units_comment="metres (m)" />
    <gamma value="7" comment="Polytropic constant for water used in the state equation" />
    <speedsystem value="0" auto="true" comment="Maximum system speed (by default the dam-break
                                                propagation is used)" />
    <coefsound value="10" comment="Coefficient to multiply speedsystem" />
    <speedsound value="0" auto="true" comment="Speed of sound to use in the simulation
                                               (by default speedofsound=coefsound*speedsystem)" />
    <coefh value="0.86" comment="Coefficient to calculate the smoothing length (h=coefh*sqrt(3*dp^2) in 3D)" />
    <_hdp value="2" comment="Coefficient to calculate the smoothing length (hdp=h/dp)" />
    <cflnumber value="0.2" comment="Coefficient to multiply dt" />
    <h value="0" auto="true" units_comment="metres (m)" />
    <b value="0" auto="true" units_comment="Pascal (Pa)" />
    <massbound value="0" auto="true" units_comment="kg" />
    <massfluid value="0" auto="true" units_comment="kg" />
</constantsdef>
```

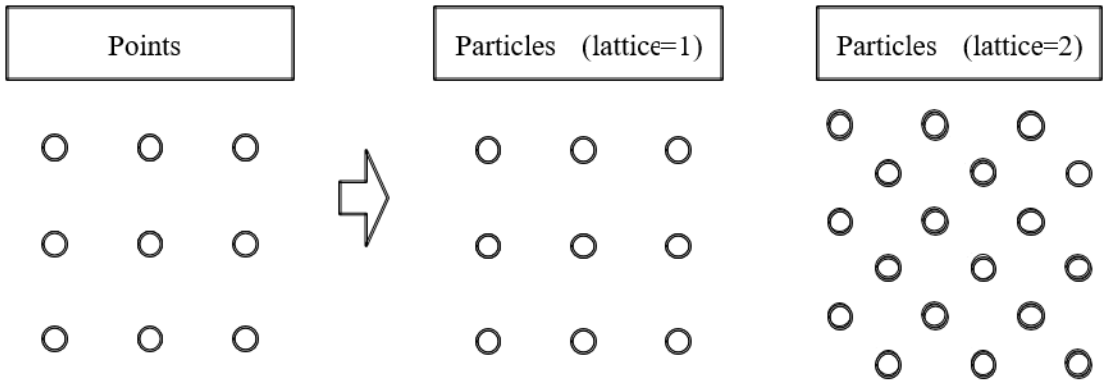
2.2. XML file: Casedef-ConstantsDef

Lattice

```
<constantsdef>
    <lattice bound="1" fluid="1" />
    <gravity x="0" y="0" z="-9.81" comment="Gravitational acceleration"
                                   units_comment="m/s^2" />
    <rhop0 value="1000" comment="Reference density of the fluid"
                                   units_comment="kg/m^3" />
    <hswl value="0" auto="true" comment="Maximum still water level to calculate speedofsound"
                                   units_comment="metres (m)" />
    <gamma value="7" comment="Polytropic constant for water used in the state equation" />
    <speedsystem value="0" auto="true" comment="Maximum system speed
                                   (by default the dam-break propagation is used)" />
    <coefsound value="20" comment="Coefficient to multiply speedsystem" />
    <speedsound value="0" auto="true" comment="Speed of sound to use in the simulation
                                   (by default speedofsound=coefsound*speedsystem)" />
    <coefh value="0.866025" comment="Coefficient to calculate the smoothing length
                                   (h=coefh*sqrt(3*dp^2) in 3D)" />
    <cflnumber value="0.2" comment="Coefficient to multiply dt" />
</constantsdef>
```

lattice: indicates the type of mesh to create particles:

- 1: one particle per point
- 2: two particles per point



2.2. XML file: Casedef-ConstantsDef

Gravity

```
<constantsdef>
    <lattice bound="1" fluid="1" />
    <gravity x="0" y="0" z="-9.81" comment="Gravitational acceleration"
                                   units_comment="m/s^2" />
    <rhop0 value="1000" comment="Reference density of the fluid"
                                   units_comment="kg/m^3" />
    <hswl value="0" auto="true" comment="Maximum still water level to calculate speedofsound"
                                   units_comment="metres (m)" />
    <gamma value="7" comment="Polytropic constant for water used in the state equation" />
    <speedsystem value="0" auto="true" comment="Maximum system speed
                                   (by default the dam-break propagation is used)" />
    <coefsound value="20" comment="Coefficient to multiply speedsystem" />
    <speedsound value="0" auto="true" comment="Speed of sound to use in the simulation
                                   (by default speedofsound=coefsound*speedsystem)" />
    <coefh value="0.866025" comment="Coefficient to calculate the smoothing length
                                   (h=coefh*sqrt(3*dp^2) in 3D)" />
    <cflnumber value="0.2" comment="Coefficient to multiply dt" />
</constantsdef>
```

$$\frac{d\boldsymbol{v}_a}{dt} = -\sum_b m_b \left(\frac{P_b + P_a}{\rho_b \cdot \rho_a} + \Pi_{ab} \right) \nabla_a W_{ab} + \boldsymbol{g}$$

2.2. XML file: Casedef-ConstantsDef

Speed of sound

```
<constantsdef>
    <lattice bound="1" fluid="1" />
    <gravity x="0" y="0" z="-9.81" comment="Gravitational acceleration"
                                   units_comment="m/s^2" />
    <rhop0 value="1000" comment="Reference density of the fluid"
                                   units_comment="kg/m^3" />
    <hswl value="0" auto="true" comment="Maximum still water level to calculate speedofsound"
                                   units_comment="metres (m)" />
    <gamma value="7" comment="Polytropic constant for water used in the state equation" />
    <speedsystem value="0" auto="true" comment="Maximum system speed
                                   (by default the dam-break propagation is used)" />
    <coefsound value="20" comment="Coefficient to multiply speedsystem" />
    <speedsound value="0" auto="true" comment="Speed of sound to use in the simulation
                                   (by default speedofsound=coefsound*speedsystem)" />
    <coefh value="0.866025" comment="Coefficient to calculate the smoothing length
                                   (h=coefh*sqrt(3*dp^2) in 3D)" />
    <cflnumber value="0.2" comment="Coefficient to multiply dt" />
</constantsdef>
```

$$speedsystem = \sqrt{g \cdot h_{swl}}$$

$$P = \frac{c_s^2 \rho_0}{\gamma}\left(\left(\frac{\rho}{\rho_0}\right)^{\gamma} - 1\right)$$

$$c_s = coef_{sound} \cdot \sqrt{g \cdot h_{swl}}$$

$$B = \frac{c_s^2 \cdot \rho_0}{\gamma} = \frac{coef_{sound}^{\,2} \cdot g \cdot h_{swl} \cdot \rho_0}{\gamma}$$

2.2. XML file: Casedef-ConstantsDef

Speed of sound

```
<constantsdef>
    <lattice bound="1" fluid="1" />
    <gravity x="0" y="0" z="-9.81" comment="Gravitational acceleration"
                                   units_comment="m/s^2" />
    <rhop0 value="1000" comment="Reference density of the fluid"
                                   units_comment="kg/m^3" />
    <hswl value="0" auto="true" comment="Maximum still water level to calculate speedofsound"
                                   units_comment="metres (m)" />
    <gamma value="7" comment="Polytropic constant for water used in the state equation" />
    <speedsystem value="0" auto="true" comment="Maximum system speed
                                   (by default the dam-break propagation is used)" />
    <coefsound value="20" comment="Coefficient to multiply speedsystem" />
    <speedsound value="0" auto="true" comment="Speed of sound to use in the simulation
                                   (by default speedofsound=coefsound*speedsystem)" />
    <coefh value="0.866025" comment="Coefficient to calculate the smoothing length
                                   (h=coefh*sqrt(3*dp^2) in 3D)" />
    <cflnumber value="0.2" comment="Coefficient to multiply dt" />
</constantsdef>
```

$$speedsystem = \sqrt{g \cdot h_{swl}}$$

$$P = \frac{c_s^2 \rho_0}{\gamma}\left(\left(\frac{\rho}{\rho_0}\right)^{\gamma} - 1\right)$$

$$c_s = coef_{sound} \cdot \sqrt{g \cdot h_{swl}}$$

$$B = \frac{c_s^2 \cdot \rho_0}{\gamma} = \frac{coef_{sound}{}^2 \cdot g \cdot h_{swl} \cdot \rho_0}{\gamma}$$

2.2. XML file: Casedef-ConstantsDef

Speed of sound

```
<constantsdef>
    <lattice bound="1" fluid="1" />
    <gravity x="0" y="0" z="-9.81" comment="Gravitational acceleration"
                                   units_comment="m/s^2" />
    <rhop0 value="1000" comment="Reference density of the fluid"
                                   units_comment="kg/m^3" />
    <hswl value="0" auto="true" comment="Maximum still water level to calculate speedofsound"
                                   units_comment="metres (m)" />
    <gamma value="7" comment="Polytropic constant for water used in the state equation" />
    <speedsystem value="0" auto="true" comment="Maximum system speed
                                   (by default the dam-break propagation is used)" />
    <coefsound value="20" comment="Coefficient to multiply speedsystem" />
    <speedsound value="0" auto="true" comment="Speed of sound to use in the simulation
                                   (by default speedofsound=coefsound*speedsystem)" />
    <coefh value="0.866025" comment="Coefficient to calculate the smoothing length
                                   (h=coefh*sqrt(3*dp^2) in 3D)" />
    <cflnumber value="0.2" comment="Coefficient to multiply dt" />
</constantsdef>
```

$$speedsystem = \sqrt{g \cdot h_{swl}}$$

$$P = \frac{c_s^2 \rho_0}{\gamma}\left(\left(\frac{\rho}{\rho_0}\right)^{\gamma} - 1\right)$$

$$c_s = coef_{sound} \cdot \sqrt{g \cdot h_{swl}}$$

$$B = \frac{c_s^2 \cdot \rho_0}{\gamma} = \frac{coef_{sound}^{\,2} \cdot g \cdot h_{swl} \cdot \rho_0}{\gamma}$$

2.2. XML file: Casedef-ConstantsDef

Kernel size

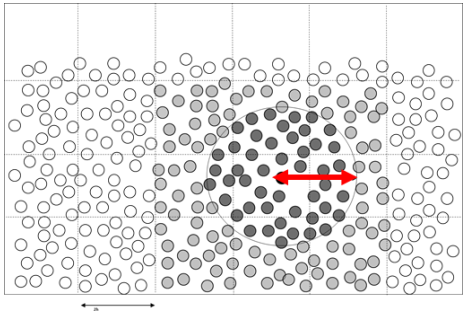
```
<constantsdef>
    <lattice bound="2" fluid="1" />
    <gravity x="0" y="0" z="-9.81" comment="Gravitational acceleration" units_comment="m/s^2" />
    <rhop0 value="1000" comment="Reference density of the fluid" units_comment="kg/m^3" />
    <hswl value="0" auto="true" comment="Maximum still water level to calculate speedofsound using coefsound"
                                units_comment="metres (m)" />
    <gamma value="7" comment="Polytropic constant for water used in the state equation" />
    <speedsystem value="0" auto="true" comment="Maximum system speed (by default the dam-break
                                                propagation is used)" />
    <coefsound value="10" comment="Coefficient to multiply speedsystem" />
    <speedsound value="0" auto="true" comment="Speed of sound to use in the simulation
                                               (by default speedofsound=coefsound*speedsystem)" />
    <coefh value="0.86" comment="Coefficient to calculate the smoothing length (h=coefh*sqrt(3*dp^2) in 3D)" />
    <_hdp value="2" comment="Coefficient to calculate the smoothing length (hdp=h/dp)" />
    <cflnumber value="0.2" comment="Coefficient to multiply dt" />
    <h value="0" auto="true" units_comment="metres (m)" />
    <b value="0" auto="true" units_comment="Pascal (Pa)" />
    <massbound value="0" auto="true" units_comment="kg" />
    <massfluid value="0" auto="true" units_comment="kg" />
</constantsdef>
```

coefh=1	typical value
coefh=1.2, 1.5	better for wave propagation

$$h = coefh \cdot \sqrt{dx^2 + dy^2 + dz^2}$$

$$h = coefh \cdot \sqrt{3 \cdot dp^2}$$

$$h = coefh \cdot \sqrt{3} \cdot dp$$



Other option is to define:

```
<hdp value="1.5" comment="Coefficient to calculate the smoothing length (hdp=h/dp)" />
```

$$hdp = h/dp$$

2.2. XML file: Casedef-ConstantsDef

Time-step

```
<constantsdef>
    <lattice bound="1" fluid="1" />
    <gravity x="0" y="0" z="-9.81" comment="Gravitational acceleration"
                                   units_comment="m/s^2" />
    <rhop0 value="1000" comment="Reference density of the fluid"
                                   units_comment="kg/m^3" />
    <hswl value="0" auto="true" comment="Maximum still water level to calculate speedofsound"
                                   units_comment="metres (m)" />
    <gamma value="7" comment="Polytropic constant for water used in the state equation" />
    <speedsystem value="0" auto="true" comment="Maximum system speed
                                   (by default the dam-break propagation is used)" />
    <coefsound value="20" comment="Coefficient to multiply speedsystem" />
    <speedsound value="0" auto="true" comment="Speed of sound to use in the simulation
                                   (by default speedofsound=coefsound*speedsystem)" />
    <coefh value="0.866025" comment="Coefficient to calculate the smoothing length
                                   (h=coefh*sqrt(3*dp^2) in 3D)" />
    <cflnumber value="0.2" comment="Coefficient to multiply dt" />
</constantsdef>
```

$$\Delta t = \mathrm{CFL} \cdot \min\left(\Delta t_f, \Delta t_{cv}\right)$$

$$\Delta t_f = \min_a \left(\sqrt{h / |f_a|} \right)$$

$$\Delta t_{cv} = \min_a \frac{h}{c_s + \max_b \left| \frac{h \mathbf{v}_{ab} \cdot \mathbf{r}_{ab}}{\mathbf{r}_{ab}^2} \right|}$$

2.2. XML file: Casedef-ConstantsDef

Other values

```
<constantsdef>
    <lattice bound="2" fluid="1" />
    <gravity x="0" y="0" z="-9.81" comment="Gravitational acceleration" units_comment="m/s^2" />
    <rhop0 value="1000" comment="Reference density of the fluid" units_comment="kg/m^3" />
    <hswl value="0" auto="true" comment="Maximum still water level to calculate speedofsound using coefsound"
                                units_comment="metres (m)" />
    <gamma value="7" comment="Polytropic constant for water used in the state equation" />
    <speedsystem value="0" auto="true" comment="Maximum system speed (by default the dam-break
                                                propagation is used)" />
    <coefsound value="10" comment="Coefficient to multiply speedsystem" />
    <speedsound value="0" auto="true" comment="Speed of sound to use in the simulation
                                               (by default speedofsound=coefsound*speedsystem)" />
    <coefh value="0.86" comment="Coefficient to calculate the smoothing length (h=coefh*sqrt(3*dp^2) in 3D)" />
    <_hdp value="2" comment="Coefficient to calculate the smoothing length (hdp=h/dp)" />
    <cflnumber value="0.2" comment="Coefficient to multiply dt" />
    <h value="0" auto="true" units_comment="metres (m)" />
    <b value="0" auto="true" units_comment="Pascal (Pa)" />
    <massbound value="0" auto="true" units_comment="kg" />
    <massfluid value="0" auto="true" units_comment="kg" />
</constantsdef>
```

Other option to define:

```
<h value="0" auto="true" units_comment="metres (m)" />
<b value="0" auto="true" units_comment="Pascal (Pa)" />
<massbound value="0" auto="true" units_comment="kg" />
<massfluid value="0" auto="true" units_comment="kg" />
```

2.3. XML file: Casedef-MkConfig


```
<mkconfig boundcount="240" fluidcount="10">
    <mkorientbound mk="0" orient="YxZ" />
    <mkorientfluid mk="1" orient="Xyz" />
    <mkorientfluid mk="2" orient="zyx" />
</mkconfig>
```

MK CONFIGURATION

mk: label used to

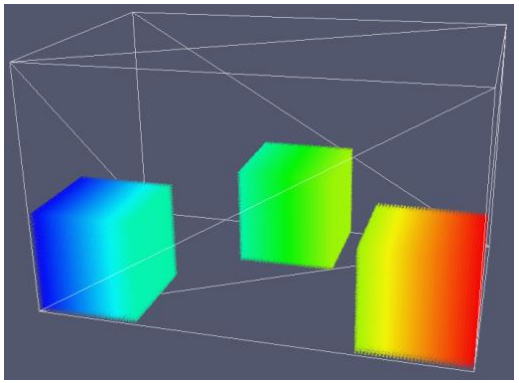
- defines the order objects are created
- applies specific features to the different set of points such as movement, rigid motion…

240 labels for boundary particles and

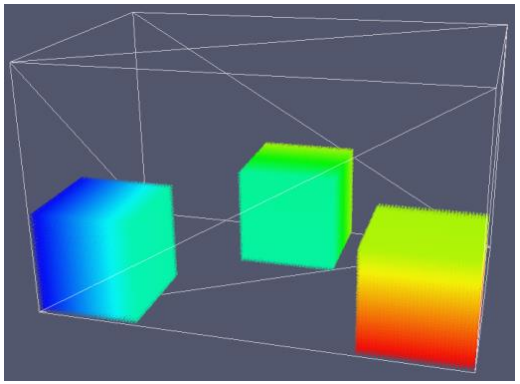
10 labels for fluid particles

The **maximum number of labels** is 250 using standard GenCase version and 65,530 using GenCase_MkWord. However, special compilation of DualSPHysics is necessary to support more than 2,048 different mk values.

mkorientfluid = “xyz”



mkorientfluid = “xyz”
mkorientfluid = “yzX”
mkorientfluid = “ZYx”



mkorientation: determines the order of particles when creating one object (useful for visualization with the variable *idp*)

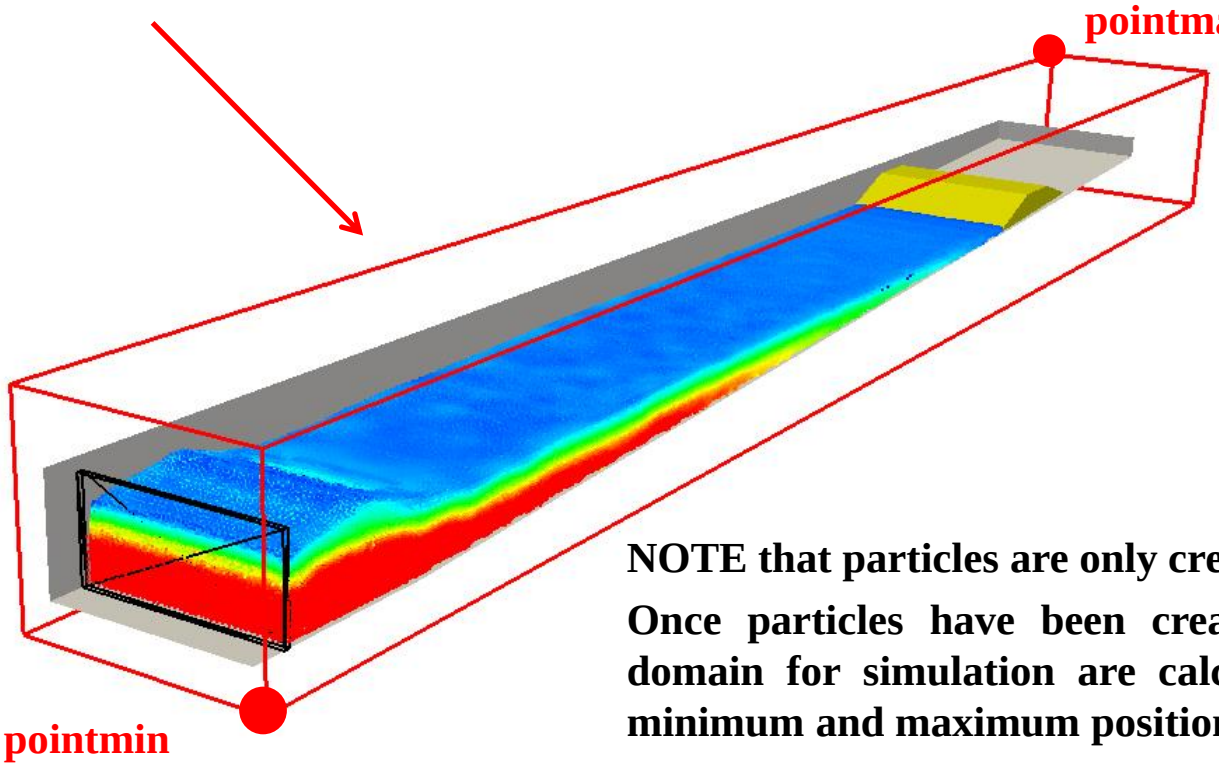
2.4.1. XML file: Casedef-Geometry-Definition


```
<!--DEFINITION OF DOMAIN WHERE PARTICLES WILL BE CREATED -->
<definition dp="0.005">
    <pointmin x="-0.05" y="0.1" z="-0.05" />
    <pointmax x=" 2.00" y="0.1" z=" 1.00" />
</definition>
```

dp defines the distance between particles

WHEN CHANGING THIS PARAMETER, THE TOTAL NUMBER OF PARTICLES IS MODIFIED

pointmin & pointmax defines the dimensions of the domain where particles can be created



NOTE that particles are only created within this domain.

Once particles have been created the dimensions of the domain for simulation are calculated again starting from minimum and maximum positions of the created particles.

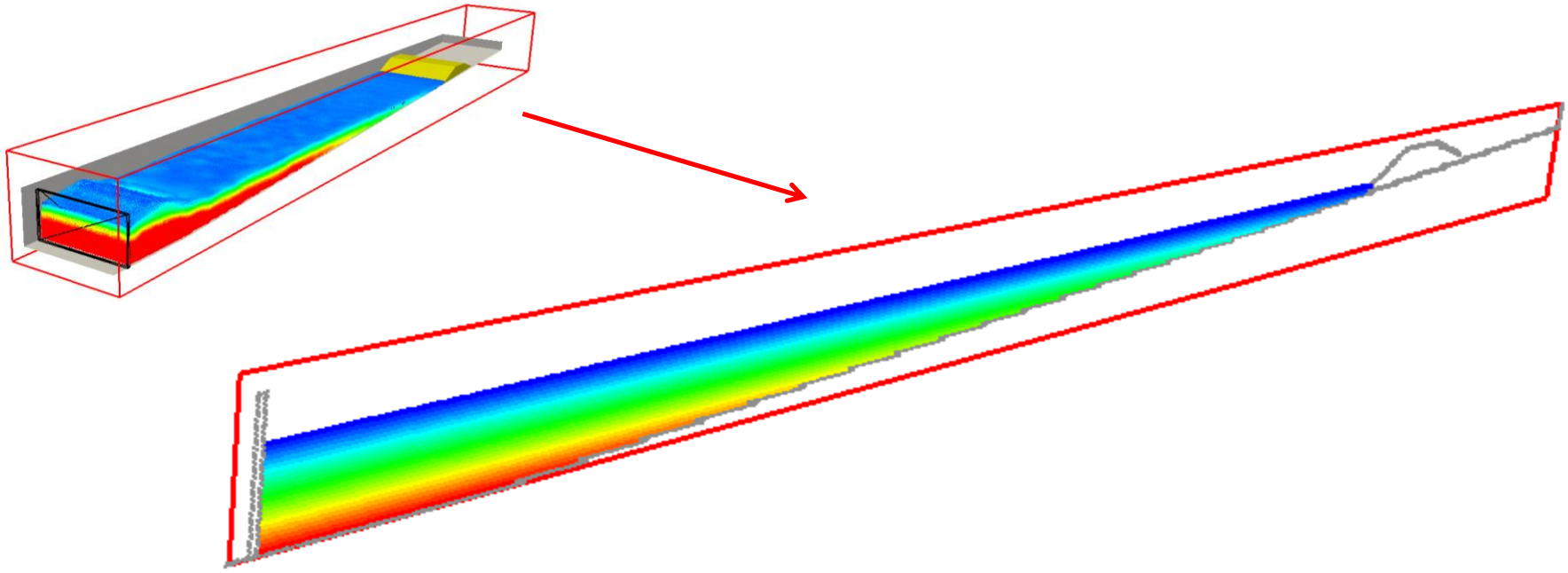
Simulation domain can be also configured in <parameters>

2.4.1. XML file: Casedef-Geometry-Definition


```
<!--DEFINITION OF DOMAIN WHERE PARTICLES WILL BE CREATED -->
<definition dp="0.005">
    <pointmin x="-0.05" y="0.1" z="-0.05" />
    <pointmax x=" 2.00" y="0.1" z=" 1.00" />
</definition>
```

A 2-D configuration can be generated by imposing the same values along Y-direction

<pointmin> = <pointmax>



2.4.2. XML file: Casedef-Geometry-Commands

Several commands in <mainlist> are executed by GenCase program to create the case geometry in SPH particles.

```
<commands>
    <mainlist>
        <setshapemode>dp | bound</setshapemode>
        <setdrawmode mode="full" />
        <!--CREATION OF FLUID PARTICLES (BOX OF WATER)-->
        <setmkfluid mk="0" />
        <drawbox>
            <boxfill>solid</boxfill>
            <point x="0" y="0" z="0" />
            <size x="0.4" y="0.67" z="0.3" />
        </drawbox>
        <!--CREATION OF BOUNDARY PARTICLES (WALLS OF TANK) -->
        <setmkbound mk="0" />
        <drawbox>
            <boxfill>bottom | left | right | front | back</boxfill>
            <point x="0" y="0" z="0" />
            <size x="1.6" y="0.67" z="0.4" />
        </drawbox>
        <shapeout file="" />
    </mainlist>
</commands>
```

Volume of fluid: *setmkfluid* mk=0,
solid to create particles within the specified volume
drawbox to plot a rectangular box defining a corner and its size in the 3 directions

Boundary Tank: *setmkbound* mk=0,
specify box faces on which particles are created (top is not used in this example)

2.4.2. XML file: Casedef-Geometry-Commands

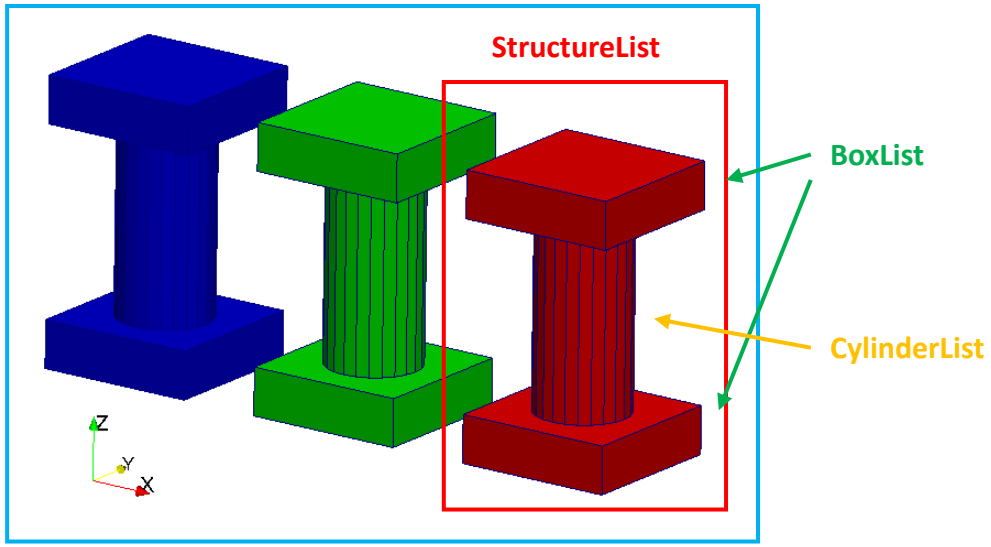

```
<geometry>
    <commands>
        <list name="BoxList">
            <drawbox>
                <boxfill>all</boxfill>
                <point x="0" y="0" z="0" />
                <size x="1" y="1" z="0.3" />
            </drawbox>
        </list>
        <list name="CylinderList">
            <drawcylinder radius="0.3">
                <point x="0" y="0" z="0" />
                <point x="0" y="0" z="1.2" />
            </drawcylinder>
        </list>
        <list name="StructureList">
            <matrixsave />
            <runlist name="BoxList" />
            <move x="0.5" y="0.5" z="0.3" />
            <runlist name="CylinderList" />
            <move x="-0.5" y="-0.5" z="1.2" />
            <runlist name="BoxList" />
            <matrixload />
        </list>
        <mainlist>
            <setshapemode>actual | bound</setshapemode>
            <setdrawmode mode="full" />
            <setmkbound mk="0" />
            <runlist name="StructureList" />
            <move x="1.5" y="0" z="0" />
            <setmkbound mk="1" />
            <runlist name="StructureList" />
            <move x="1.5" y="0" z="0" />
            <setmkbound mk="2" />
            <runlist name="StructureList" />
            <shapeout file="" />
        </mainlist>
    </commands>
</geometry>
```

COMMAND LISTS

Mainlist and auxiliary lists of commands.

- Several commands are executed by GenCase to create the geometry in SPH particles.
- Drawing commands are located in **mainlist** section, but other command lists can be created and these list can be called by mainlist or other lists using <*runlist*>.
- Multiple consecutive calls to <runlist> can be done using *times="number"*. For example: <*runlist name="StructureList" times="5" />*.

<mainlist>

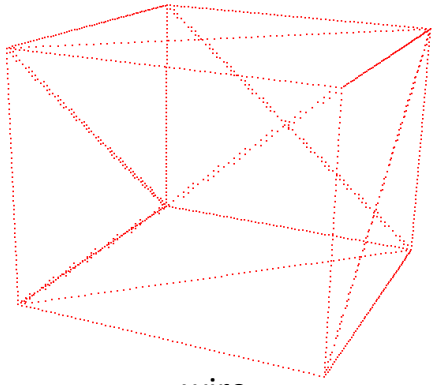


2.4.2. XML file: Casedef-Geometry-Commands

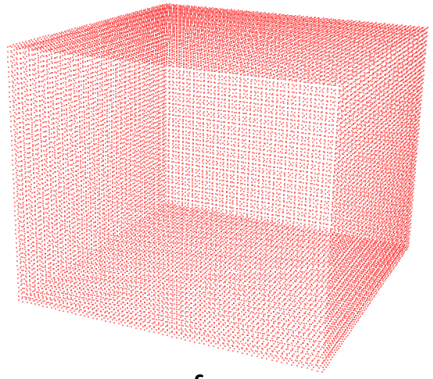
DRAWING SETTINGS

<setdrawmode>: This command indicates the mode to create points where particles will be generated

```
<mainlist>
    <setdrawmode mode="wire" />
    <setdrawmode mode="face" />
    <setdrawmode mode="solid" />
    <setdrawmode mode="full" />
</mainlist>
```



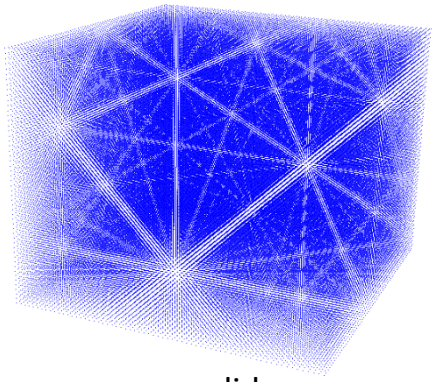
wire



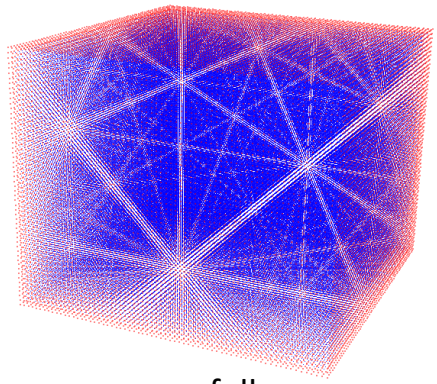
face

<setdrawmode>:

- "**wire**": wire mode
- "**face**": draw faces
- "**solid**": draw inside
- "**full**": combines *face* and *solid*



solid



full

2.4.2. XML file: Casedef-Geometry-Commands

VTK OUTPUT

```
<mainlist>
    <setshapemode>actual | bound</setshapemode>
    <setshapemode>actual | bound | fluid </setshapemode>
    <setshapemode>dp | bound</setshapemode>
    <setshapemode>dp | void</setshapemode>
    <setshapemode>actual | dp | void</setshapemode>
    <setshapemode>null</setshapemode>
</mainlist>
```

<setshapemode>: defines the draw operations to create VTK files (polygons)

- "**actual**": using the real coordinates
- "**dp**": adjusting coordinates to *dp*

- "**fluid**": operations with *mk-fluid*
- "**bound**": operations with *mk-bound*
- "**void**": operations with *mk-void*

- **"null"**: disable the generation of shapes.

  It is equivalent to: `<setactive drawshapes="false" />`

2.4.2. XML file: Casedef-Geometry-Commands

VTK OUTPUT

<setshapemode>: defines the draw operations to create a VTK files (polygons)

```
<mainlist>
    <setshapemode>dp | bound</setshapemode>
     <setdrawmode mode="full" />
     <setmkfluid mk="0" />
    +<drawbox></drawbox>
     <setmkbound mk="0" />
    +<drawbox></drawbox>
     <shapeout file="Box" />
     <setmkvoid />
    +<drawbox></drawbox>
     <setmkbound mk="1" />
    +<drawbox></drawbox>
     <shapeout file="Building" />
</mainlist>
```

```
<mainlist>
    <setshapemode>actual | dp | bound</setshapemode>
     <setdrawmode mode="full" />
     <setmkfluid mk="0" />
    +<drawprism mask="0"></drawprism>
     <setmkvoid />
    +<drawbox></drawbox>
     <setdrawmode mode="face" />
     <setmkbound mk="10" />
    +<drawbox></drawbox>
     <setmkbound mk="0" />
    +<drawprism mask="96"></drawprism>
     <shapeout file="" reset="true" />
</mainlist>
```

shapeout: creates VTK files (polygons) of only some *bound* objects
Case_Box_Dp.vtk
Case_Building_Dp.vtk

shapeout: creates VTK files (polygons) of all the *bound* objects
Case__Actual.vtk
Case__Dp.vtk

reset="true" objects created after this command will be saved on a different VTK file

2.4.2. XML file: Casedef-Geometry-Commands

DRAWING SETTINGS

<resetdraw>: Resets all options to their default value and deletes all points drawn.

```
<resetdraw />
```

<setactive>: Activates o deactivates the drawing of points (***drawpoints***), the generation of shapes for output VTK using <shapeout> (***drawshapes***) and the registration of output VTK files in <vtkout> section (***vtkout***). All values are activated by default.

```
<setactive drawpoints="false" drawshapes="true" vtkout="true" />
```

The drawing commands generate shapes for VTK files created by <shapeout>, but no points (particles) are drawn.

```
<setactive drawshapes="false" />
```

No shapes are generated. It is equivalent to: `<setshapemode>null</setshapemode>`

```
<setactive vtkout="false" />
```

The output VTK files created by <shapeout> are not added to the list of output files in the XML section <vtkout>.

2.4.2. XML file: Casedef-Geometry-Commands

MK SELECTION

These commands indicate the type of particles to be generated

```
<commands>
    <mainlist>
        <setmkvoid />
        <setmkfluid mk="0" />
        <setmkbound mk="0" />
        <setmknextfluid next="true" />
        <setmknextbound next="false" />
        <setmknextauto auto="true" />
    </mainlist>
</commands>
```

<setmkvoid>, **<setmkfluid>**, **<setmkbound>**: defines the label *mk* to draw points of type:

void (empty), fluid, bound

<setmknextfluid>, **<setmknextbound>**: increases (decreases) the value of *mk* with next=*true* (=*false*)

<setmknextauto>: after each draw command *mk* is increased automatically

2.4.2. XML file: Casedef-Geometry-Commands

TRANSFORMATIONS

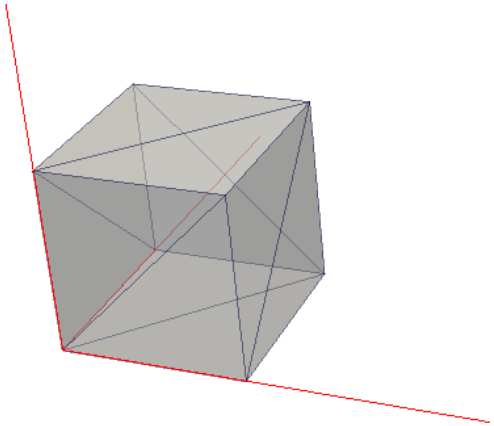
Transformation utilities

```
<mainlist>
    <setshapemode>dp | bound</setshapemode>
    <setmkbound mk="0" />
    <move x="0.5" y="0" z="0" />
   +<drawbox></drawbox>
    <shapeout file="BoxMove" reset="true" />
    <matrixreset />
    <matrixsave />
    <scale x="2" y="1.5" z="0.5" />
   +<drawbox></drawbox>
    <shapeout file="BoxScale" reset="true" />
    <stateload />
    <rotate ang="45" x="0" y="0" z="1" />
    <rotateline ang="45">
        <point x="0" y="0" z="0" />
        <point x="1" y="1" z="-1" />
    </rotateline>
   +<drawbox></drawbox>
    <shapeout file="BoxRotate" reset="true" />
</mainlist>
```

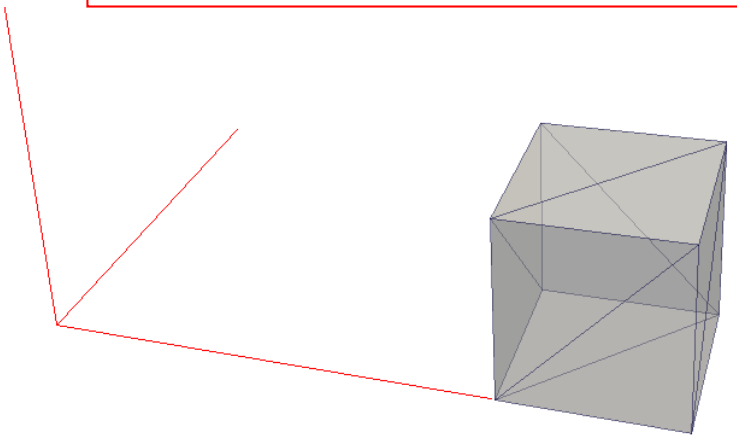
<move>: a displacement is applied to the transformation matrix
<scale>: scaling is applied to matrix
<rotate>:a starting vector and angle are given for object rotation
<rotateline>:a starting vector (defined by two points) and angle are given for object rotation
<matrixreset>: the modified matrix is replaced by the original one (identity matrix)
<matrixsave>: saves current transformation matrix in a stack
<matrixload>: recovers the last transformation matrix stored in the stack

2.4.2. XML file: Casedef-Geometry-Commands

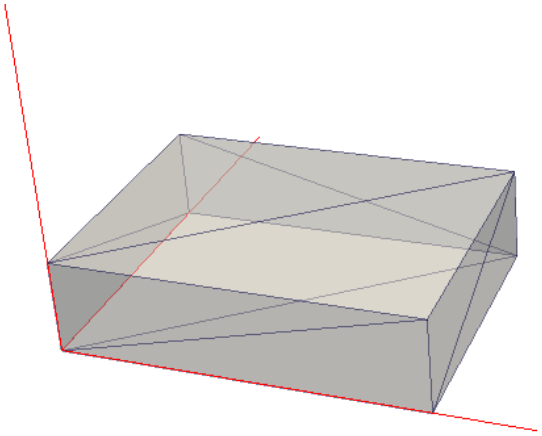
TRANSFORMATIONS



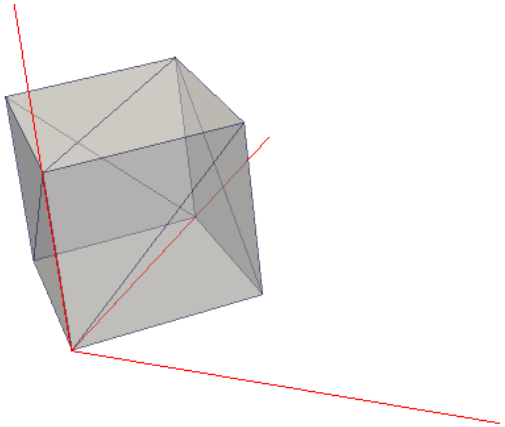
<drawbox …>



<move x="0.5" y="0" z="0"/>
<drawbox …>



<scale x="2" y="1.5" z="0.5"/>
<drawbox …>



<rotate x="0" y="0" z="1" ang="45"/>
<drawbox …>

2.4.2. XML file: Casedef-Geometry-Commands


```
- <mainlist>
    <setshapemode>dp | bound</setshapemode>
    <setmkbound mk="0"/>
  - <setlinebegin>
      <point x="0" y="0" z="0"/>
    </setlinebegin>
  - <drawlineto>
      <point x="0" y="1" z="0"/>
    </drawlineto>
    <setmknextbound next="true"/>
  - <drawline>
      <point x="0" y="1" z="0"/>
      <point x="1" y="1" z="0"/>
    </drawline>
    <setmknextbound next="true"/>
  - <drawline>
      <point x="1" y="1" z="0"/>
      <point x="1" y="0" z="0"/>
    </drawline>
    <setmknextbound next="true"/>
  - <drawlines>
      <point x="1" y="0" z="0"/>
      <point x="0" y="0" z="0.5"/>
      <point x="0" y="1" z="0.5"/>
      <point x="1" y="1" z="0.5"/>
      <point x="1" y="0" z="0.5"/>
    </drawlines>
    <shapeout file="Lines" reset="true"/>
  </mainlist>
```

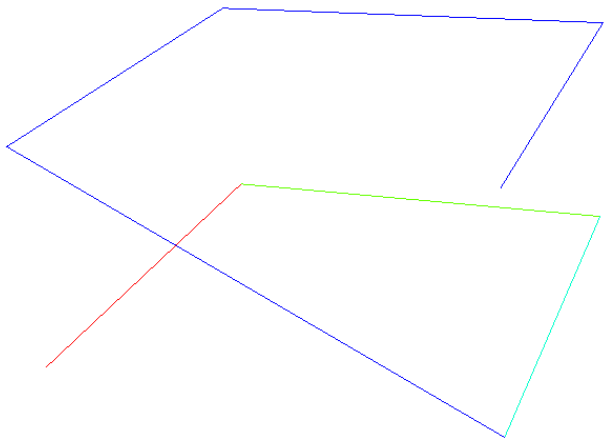
LINES

<setlinebegin>: sets the begining of the line with <drawlineto>

<drawlineto>: draws a line to a given point

<drawline>: draws a line between two points

<drawlines>: draws lines between several points



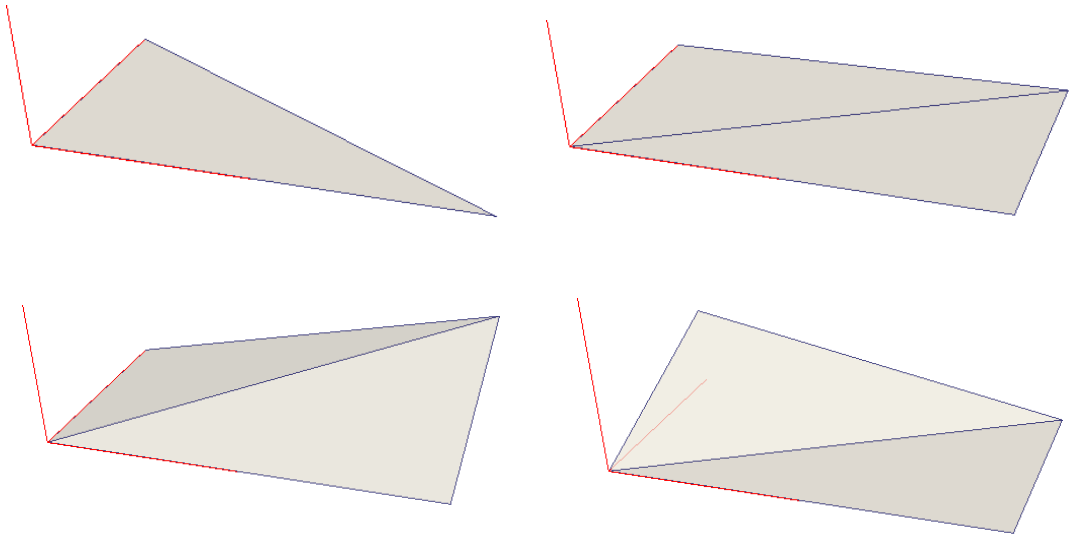
2.4.2. XML file: Casedef-Geometry-Commands


```
- <mainlist>
    <setshapemode>dp | bound</setshapemode>
    <setmkbound mk="0"/>
  - <drawtriangle>
      <point x="0" y="0" z="0"/>
      <point x="1" y="0" z="0"/>
      <point x="0" y="0.5" z="0"/>
    </drawtriangle>
    <shapeout file="Triangle" reset="true"/>
  - <drawquadri>
      <point x="0" y="0" z="0"/>
      <point x="1" y="0" z="0"/>
      <point x="1" y="0.5" z="0"/>
      <point x="0" y="0.5" z="0"/>
    </drawquadri>
    <shapeout file="Quadri" reset="true"/>
  - <drawquadri>
      <point x="0" y="0" z="0"/>
      <point x="1" y="0" z="0"/>
      <point x="1" y="0.5" z="0.2"/>
      <point x="0" y="0.5" z="0"/>
    </drawquadri>
    <shapeout file="Quadri2" reset="true"/>
  - <drawquadri>
      <point x="0" y="0" z="0"/>
      <point x="1" y="0" z="0"/>
      <point x="1" y="0.5" z="0"/>
      <point x="0" y="0.5" z="0.2"/>
    </drawquadri>
    <shapeout file="Quadri3" reset="true"/>
  </mainlist>
```

TRIANGLES

<drawtriangle>: draws a triangle with tree points (points must always go counterclockwise)

<drawquadri>: draws the quadrilateral described by four points (points may not be in the same plane)

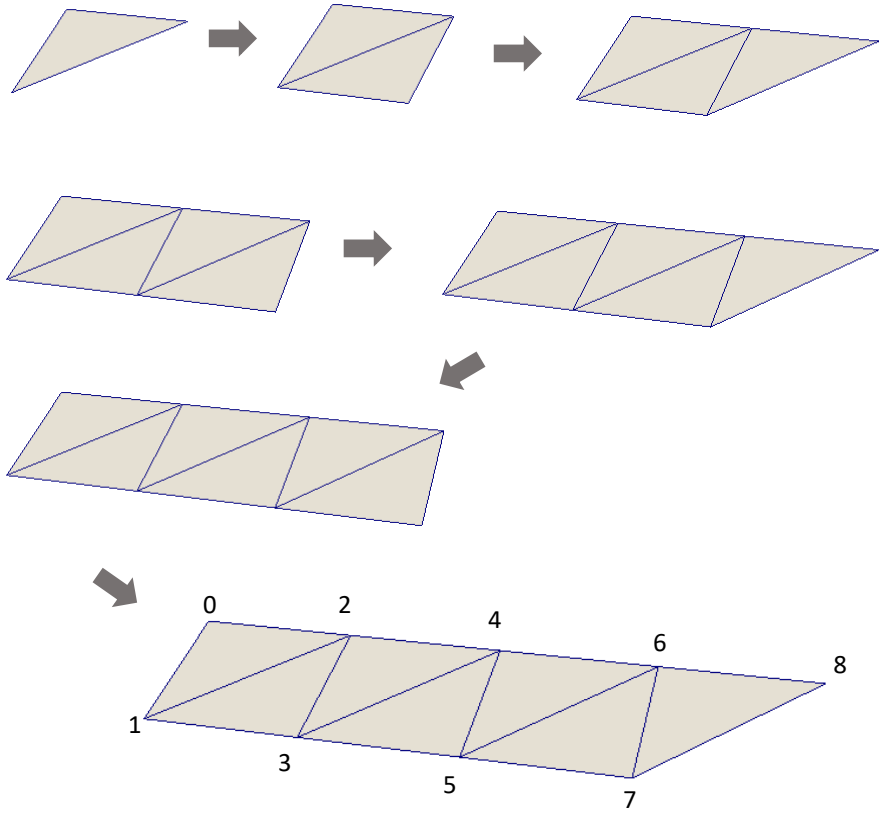


2.4.2. XML file: Casedef-Geometry-Commands


```
- <mainlist>
    <setshapemode>dp | bound</setshapemode>
    <setmkbound mk="0"/>
  - <drawtrianglesstrip>
      <point x="0" y="1" z="0"/>
      <point x="0" y="0" z="0"/>
      <point x="1" y="1" z="0"/>
      <point x="1" y="0" z="0"/>
      <point x="2" y="1" z="0"/>
      <point x="2" y="0" z="0"/>
      <point x="3" y="1" z="0"/>
      <point x="3" y="0" z="0"/>
      <point x="4" y="1" z="0"/>
    </drawtrianglesstrip>
    <shapeout file="TrianglesStrip9" reset="true"/>
  </mainlist>
```

TRIANGLES

<drawtrianglesstrip>: draws a series of chained triangles

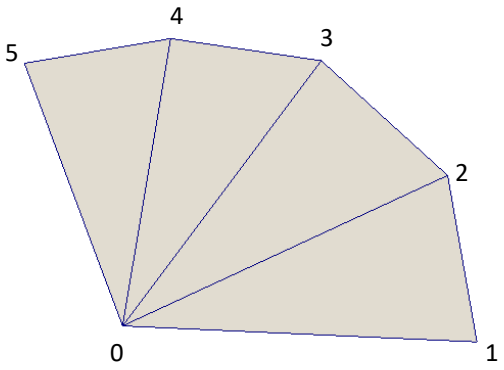


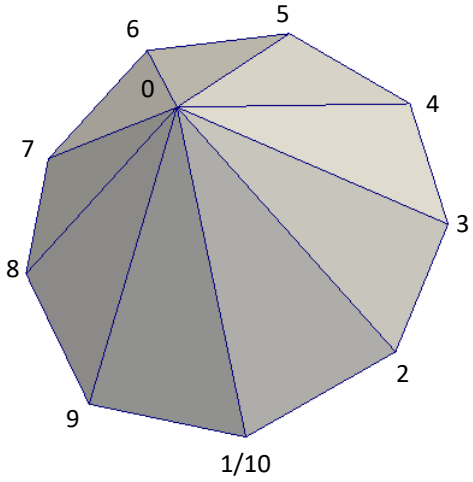
2.4.2. XML file: Casedef-Geometry-Commands


```
– <mainlist>
    <setshapemode>dp | bound</setshapemode>
    <setmkbound mk="0"/>
  – <drawtrianglesfan>
      <point x="0" y="0" z="0"/>
      <point x="1" y="0" z="0"/>
      <point x="0.9" y="0.5" z="0"/>
      <point x="0.5" y="0.9" z="0"/>
      <point x="0" y="1" z="0"/>
      <point x="-0.5" y="0.9" z="0"/>
    </drawtrianglesfan>
    <shapeout file="TrianglesFan" reset="true"/>
    <setmkbound mk="0"/>
  – <drawtrianglesfan>
      <point x="0" y="0" z="1"/>
      <point x="1" y="0" z="0"/>
      <point x="0.8" y="0.6" z="0"/>
      <point x="0.2" y="1" z="0"/>
      <point x="-0.5" y="0.9" z="0"/>
      <point x="-0.9" y="0.3" z="0"/>
      <point x="-0.9" y="-0.3" z="0"/>
      <point x="-0.5" y="-0.9" z="0"/>
      <point x="0.2" y="-1" z="0"/>
      <point x="0.8" y="-0.6" z="0"/>
      <point x="1" y="0" z="0"/>
    </drawtrianglesfan>
    <shapeout file="TrianglesFan2" reset="true"/>
  </mainlist>
```

TRIANGLES

<drawtrianglesfan>: draws a range of triangles



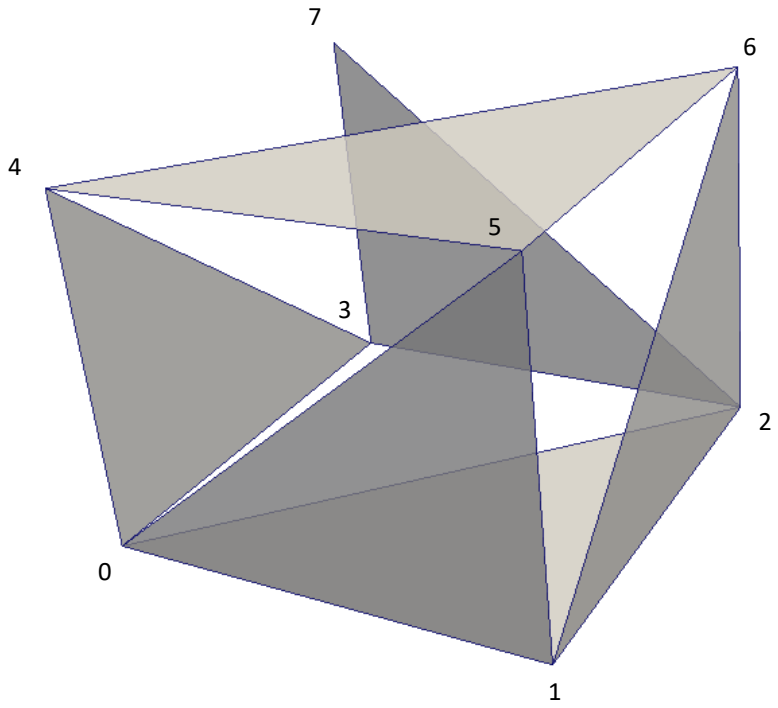


2.4.2. XML file: Casedef-Geometry-Commands


```
- <mainlist>
    <setshapemode>dp | bound</setshapemode>
    <setmkbound mk="0"/>
  - <drawtriangles>
    - <points>
        <point x="0" y="0" z="0"/>
        <point x="1" y="0" z="0"/>
        <point x="1" y="1" z="0"/>
        <point x="0" y="1" z="0"/>
        <point x="0" y="0" z="0.8"/>
        <point x="1" y="0" z="0.8"/>
        <point x="1" y="1" z="0.8"/>
        <point x="0" y="1" z="0.8"/>
      </points>
    - <triangles>
        <triangle x="0" y="1" z="5"/>
        <triangle x="1" y="2" z="6"/>
        <triangle x="2" y="3" z="7"/>
        <triangle x="3" y="0" z="4"/>
        <triangle x="0" y="2" z="1"/>
        <triangle x="4" y="5" z="6"/>
      </triangles>
    </drawtriangles>
    <shapeout file="Triangles" reset="true"/>
  </mainlist>
```

TRIANGLES

<drawtriangles>: draws a series of triangles defined by a set of points or a set of triangles



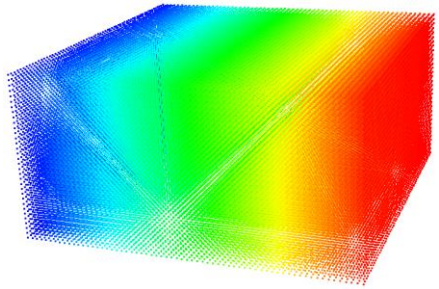
2.4.2. XML file: Casedef-Geometry-Commands


```
- <mainlist>
    <setshapemode>dp | bound</setshapemode>
    <setmkbound mk="0"/>
  - <drawbox>
      <boxfill>solid</boxfill>
      <point x="0" y="0" z="0"/>
      <size x="1" y="1" z="0.5"/>
    </drawbox>
    <shapeout file="BoxSolid" reset="true"/>
  - <drawbox>
      <boxfill>all</boxfill>
      <point x="0" y="0" z="0"/>
      <size x="1" y="1" z="0.5"/>
    </drawbox>
    <shapeout file="BoxA" reset="true"/>
  - <drawbox>
      <boxfill>all ^ top</boxfill>
      <point x="0" y="0" z="0"/>
      <size x="1" y="1" z="0.5"/>
    </drawbox>
    <shapeout file="BoxB" reset="true"/>
  - <drawbox>
      <boxfill>bottom | left | right</boxfill>
      <point x="0" y="0" z="0"/>
      <size x="1" y="1" z="0.5"/>
    </drawbox>
    <shapeout file="BoxC" reset="true"/>
</mainlist>
```

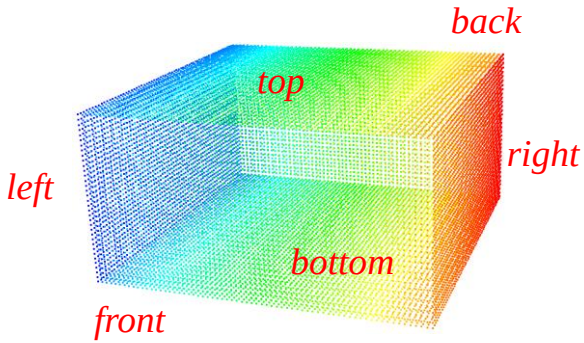
BOXES

<drawbox>: draws a box with an initial point and the size

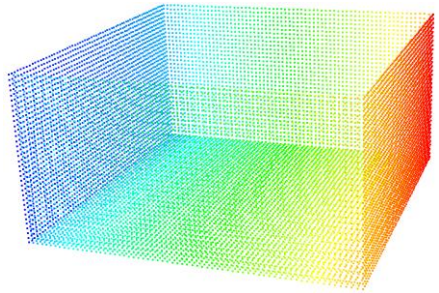
<boxfill> indicates if *solid* or *face* and the faces to be hidden



BoxSolid (solid)

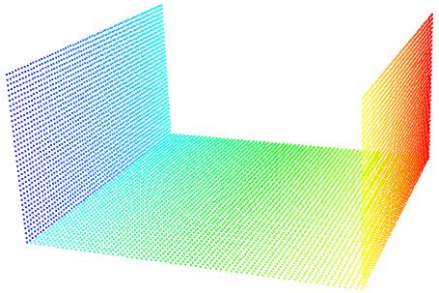


BoxA (all)



BoxB (all^top)

means
all faces excluding top



BoxC (bottom|left|right)

means only
bottom+left+right

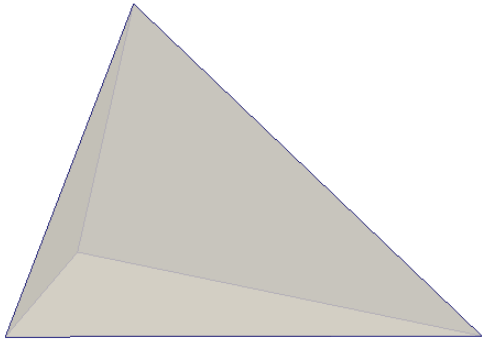
2.4.2. XML file: Casedef-Geometry-Commands

POLYHEDRONS: PYRAMID

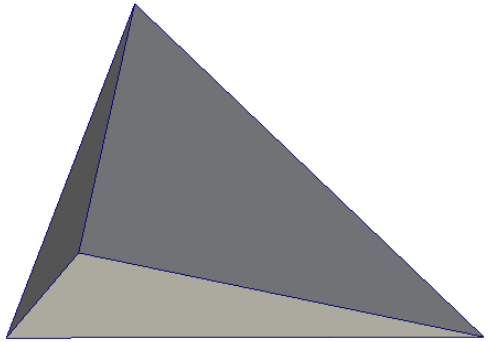
```
- <mainlist>
    <setshapemode>dp | bound</setshapemode>
    <setdrawmode mode="full"/>
    <setmkbound mk="0"/>
  - <drawpyramid mask="0">
      <point x="0.25" y="0.25" z="0.7"/>
      <point x="0" y="0" z="0"/>
      <point x="1" y="0" z="0"/>
      <point x="0" y="1" z="0"/>
    </drawpyramid>
    <shapeout file="Pyramid1" reset="true"/>
  - <drawpyramid mask="2">
      <point x="0.25" y="0.25" z="0.7"/>
      <point x="0" y="0" z="0"/>
      <point x="1" y="0" z="0"/>
      <point x="0" y="1" z="0"/>
    </drawpyramid>
    <shapeout file="Pyramid2" reset="true"/>
  </mainlist>
```

<drawpyramid>: draws a pyramid with the top point and other points of the base (minimum 3)

mask indicates the faces to be hidden with bits
the first bit always corresponds to the base and the rest to the faces following the order



Pyramid1
(*mask=0*)



Pyramid2
(*mask=2=0010*)

2.4.2. XML file: Casedef-Geometry-Commands

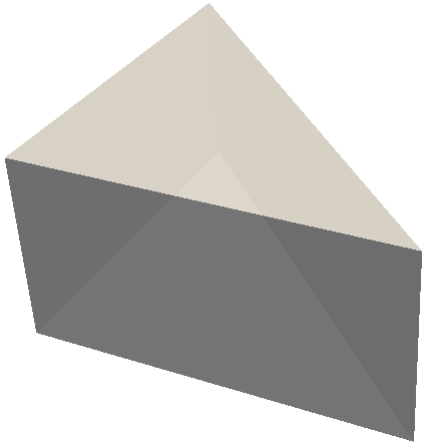
POLYHEDRONS: PRISM

```
- <mainlist>
    <setshapemode>dp | bound</setshapemode>
    <setdrawmode mode="full"/>
    <setmkbound mk="0"/>
  - <drawprism mask="0">
      <point x="0" y="0" z="0"/>
      <point x="1" y="0" z="0"/>
      <point x="0" y="1" z="0"/>
      <point x="0" y="0" z="0.5"/>
      <point x="1" y="0" z="0.5"/>
      <point x="0" y="1" z="0.5"/>
    </drawprism>
    <shapeout file="Prism1" reset="true"/>
  - <drawprism mask="2">
      <point x="0" y="0" z="0"/>
      <point x="1" y="0" z="0"/>
      <point x="0" y="1" z="0"/>
      <point x="0" y="0" z="0.5"/>
      <point x="1" y="0" z="0.5"/>
      <point x="0" y="1" z="0.5"/>
    </drawprism>
    <shapeout file="Prism2" reset="true"/>
</mainlist>
```

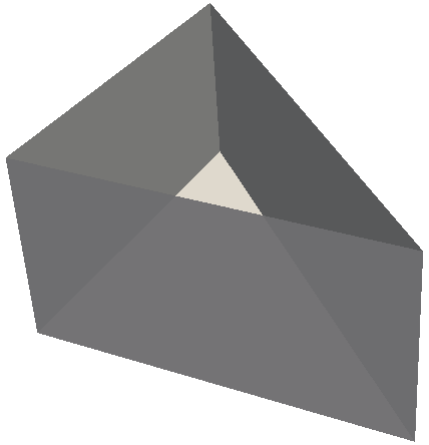
<drawprism>: draws a prism with a minimum of 6 points

The first half of points are the base and the second half the top (the number of points must be even)

mask indicates the faces to be hidden with bits
The first bit corresponds to the base, the second to the top and the rest to the faces following the order



Prism1 (*mask=0*)



Prism2 (*mask=2=00010*)

2.4.2. XML file: Casedef-Geometry-Commands

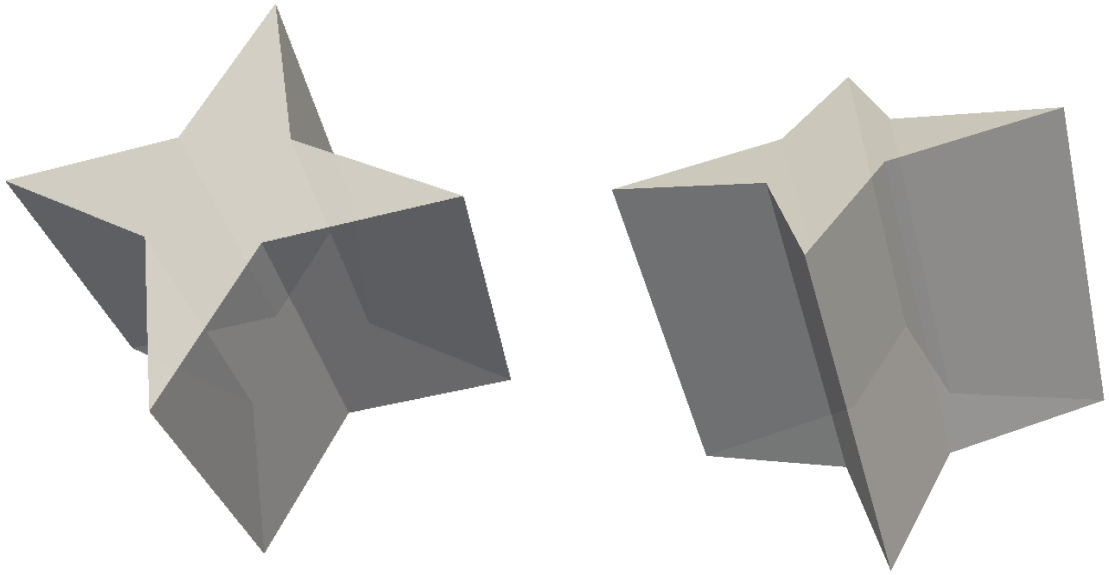
POLYHEDRONS: PRISM

```
- <mainlist>
    <setshapemode>dp | bound</setshapemode>
    <setdrawmode mode="full"/>
    <setmkbound mk="0"/>
  - <drawprism mask="0">
      <point x="0" y="0" z="0"/>
      <point x="1" y="-3" z="0"/>
      <point x="2" y="0" z="0"/>
      <point x="5" y="1" z="0"/>
      <point x="2" y="2" z="0"/>
      <point x="1" y="5" z="0"/>
      <point x="0" y="2" z="0"/>
      <point x="-3" y="1" z="0"/>
      <point x="0" y="0" z="6"/>
      <point x="1" y="-3" z="6"/>
      <point x="2" y="0" z="6"/>
      <point x="5" y="1" z="6"/>
      <point x="2" y="2" z="6"/>
      <point x="1" y="5" z="6"/>
      <point x="0" y="2" z="6"/>
      <point x="-3" y="1" z="6"/>
    </drawprism>
    <shapeout file="Prism3" reset="true"/>
  </mainlist>
```

<drawprism>: draws a prism with a minimum of 6 points

The first half of points are the base and the second half the top (the number of points must be even)

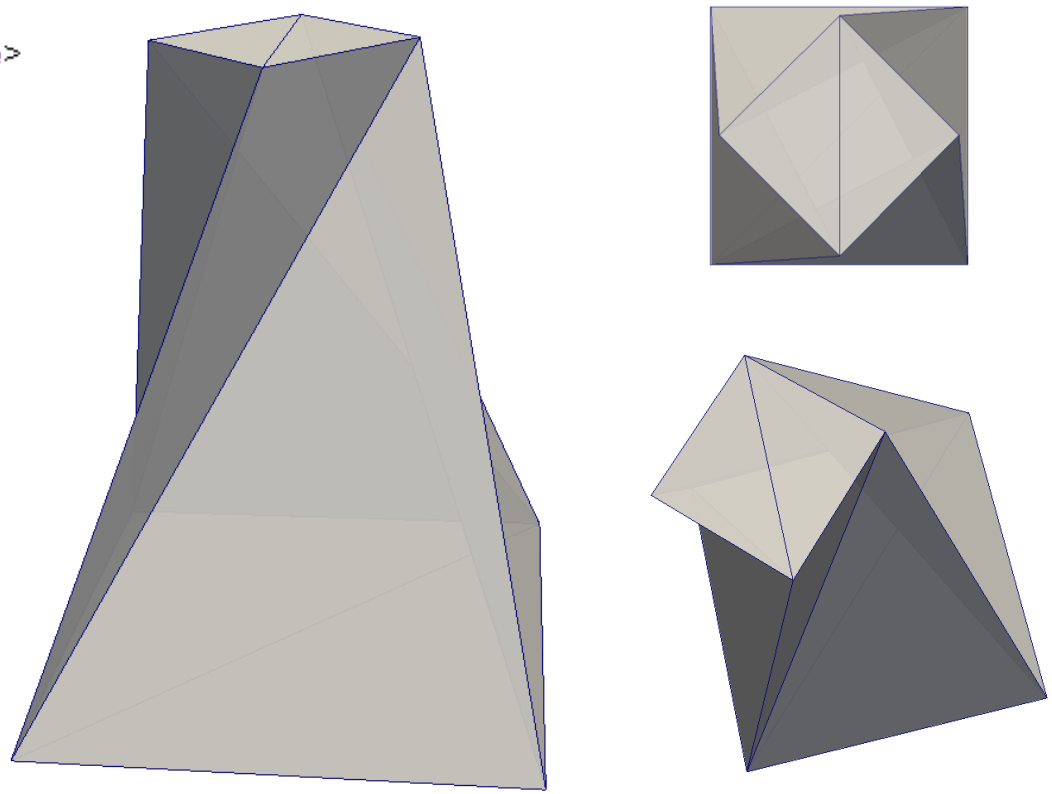
mask indicates the faces to be hidden with bits
The first bit corresponds to the base, the second to the top and the rest to the faces following the order



2.4.2. XML file: Casedef-Geometry-Commands

POLYHEDRONS: PRISM

```
- <mainlist>
    <setshapemode>dp | bound</setshapemode>
    <setdrawmode mode="full"/>
    <setmkbound mk="0"/>
  - <drawprism mask="0">
      <point x="0" y="0" z="0"/>
      <point x="4" y="0" z="0"/>
      <point x="4" y="4" z="0"/>
      <point x="0" y="4" z="0"/>
      <point x="2" y="1" z="5"/>
      <point x="3" y="2" z="5"/>
      <point x="2" y="3" z="5"/>
      <point x="1" y="2" z="5"/>
    </drawprism>
    <shapeout file="Prism4" reset="true"/>
  </mainlist>
```



2.4.2. XML file: Casedef-Geometry-Commands

POLYHEDRONS: MASK

mask indicates the faces to be hidden

Initially this is defined using BITS!!!

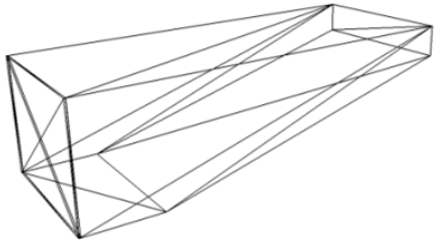
FOR EXAMPLE: OBJECT WITH 4 FACES:

mask="0"	decimal **0** is **0000** in binary	no faces are hidden
mask="1"	decimal **1** is **0001** in binary	first face is hidden
mask="2"	decimal **2** is **0010** in binary	second face is hidden
mask="4"	decimal **4** is **0100** in binary	third face is hidden
mask="8"	decimal **8** is **1000** in binary	fourth face is hidden
mask="12"	decimal **12** is **1100** in binary	third and fourth face are hidden

POLYHEDRONS: MASK

There is a second and easiest system to use mask

mask can be also defined using the index of the faces instead of bits



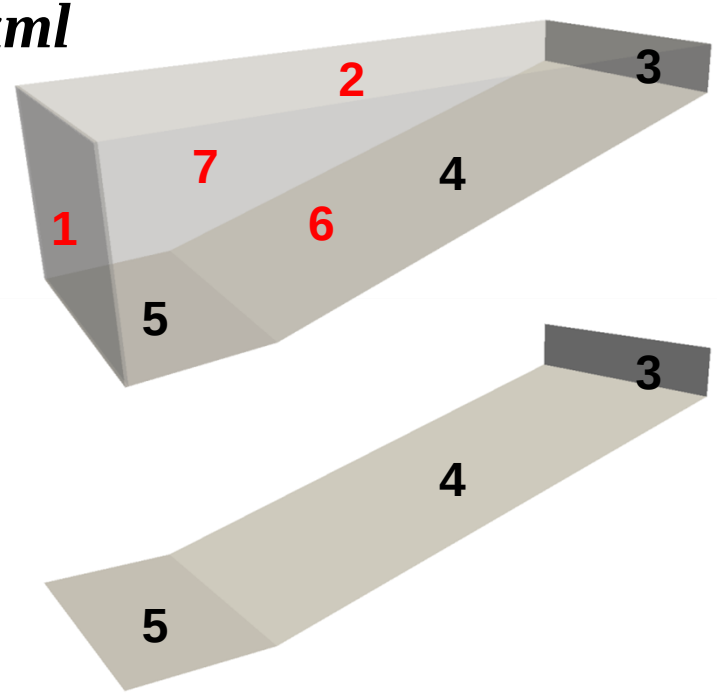
In this example, faces 1, 2, 6 and 7 are not created, only 3,4 and 5

It is important to use symbol "|" to detect this system!!!

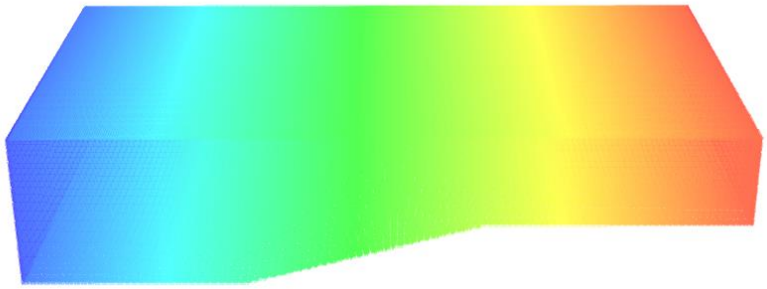
Use "X|X" if you want to specify only face X (otherwise X will be interpreted as bit)

EXAMPLE: *CaseWavemaker_Def.xml*

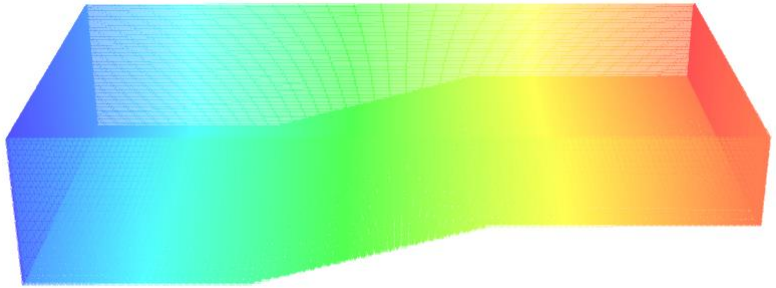
```
<setmkbound mk="0" />
<drawprism mask="1 | 2 | 6 | 7">
    <point x="5" y="0" z="1.5" />
    <point x="5" y="0" z="1.1" />
    <point x="1" y="0" z="0" />
    <point x="0" y="0" z="0" />
    <point x="0" y="0" z="1.5" />
    <point x="5" y="2" z="1.5" />
    <point x="5" y="2" z="1.1" />
    <point x="1" y="2" z="0" />
    <point x="0" y="2" z="0" />
    <point x="0" y="2" z="1.5" />
</drawprism>
```



2.4.2. XML file: Casedef-Geometry-Commands



BeachFace



BeachFace
(mask="128")

POLYHEDRONS: BEACH

```
<mainlist>
    <setshapemode>dp | bound</setshapemode>
    <setmkbound mk="10" />
    <setdrawmode mode="face" />
    <drawbeach mask="128">
        <point x="0" y="3" z="1.2" />
        <point x="0" y="3" z="0" />
        <point x="1.7" y="3" z="0" />
        <point x="3.5" y="3" z="0.5" />
        <point x="5.5" y="3" z="0.5" />
        <point x="5.5" y="3" z="1.2" />
    </drawbeach>
</mainlist>
```

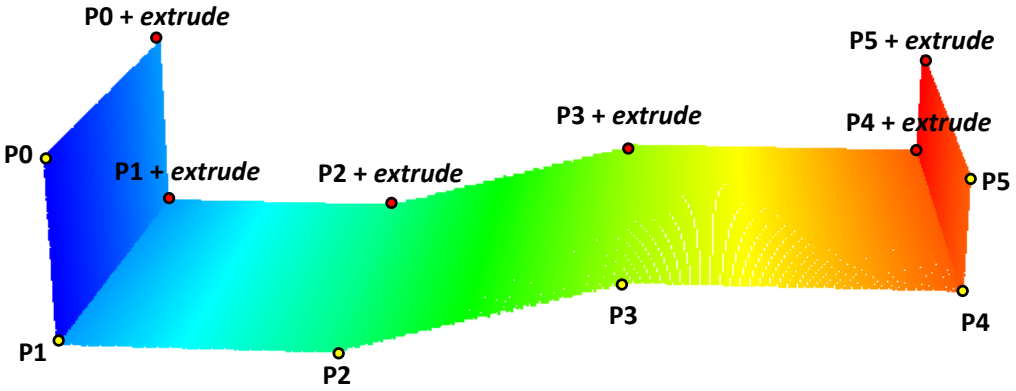
<drawbeach>: draws a beach with the lateral points that formed the profile of the beach

mask indicates the faces to be hidden .

mask="128" decimal **128** is **10000000** in binary eighth face is hidden

2.4.2. XML file: Casedef-Geometry-Commands

POLYHEDRONS: EXTRUDE




```
<mainlist>
    <setmkbound mk="0" />
    <drawextrude closed="false">
P0      <point x="0" y="0" z="1.2" />
P1      <point x="0" y="0" z="0" />
P2      <point x="1.7" y="0" z="0" />
P3      <point x="3.5" y="0" z="0.5" />
P4      <point x="5.5" y="0" z="0.5" />
P5      <point x="5.5" y="0" z="1.2" />
extrude <extrude x="0" y="3" z="0" />
    </drawextrude>
</mainlist>
```

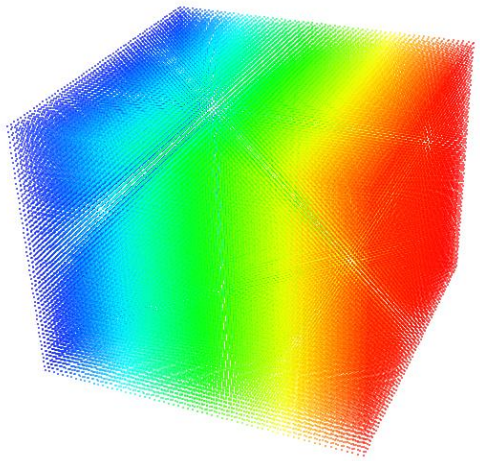
<drawextrude>: draws an extruded line of points to create a 3-D shape. It allows you to create complex shapes easily.

2.4.2. XML file: Casedef-Geometry-Commands

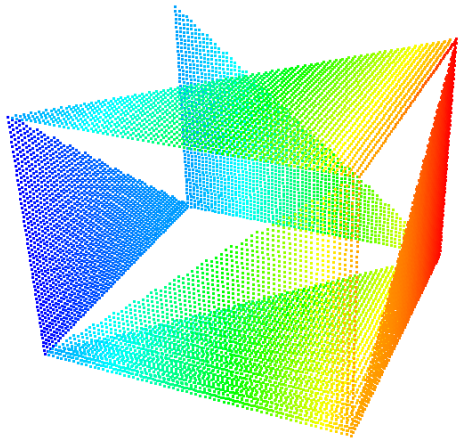
POLYHEDRONS: FIGURE

```
- <mainlist>
    <setshapemode>dp | bound</setshapemode>
    <setmkbound mk="0"/>
  - <drawfigure>
    - <points>
        <point x="0" y="0" z="0"/>
        <point x="1" y="0" z="0"/>
        <point x="1" y="1" z="0"/>
        <point x="0" y="1" z="0"/>
        <point x="0" y="0" z="0.8"/>
        <point x="1" y="0" z="0.8"/>
        <point x="1" y="1" z="0.8"/>
        <point x="0" y="1" z="0.8"/>
      </points>
    - <triangles>
        <triangle x="0" y="1" z="5"/>
        <triangle x="1" y="2" z="6"/>
        <triangle x="2" y="3" z="7"/>
        <triangle x="3" y="0" z="4"/>
        <triangle x="0" y="2" z="1"/>
        <triangle x="4" y="5" z="6"/>
      </triangles>
    </drawfigure>
  </mainlist>
```

<drawfigure>: draws a solid figure consisting of all the interior points to the planes formed by the given triangles



DrawFigure (*drawmode=solid*)



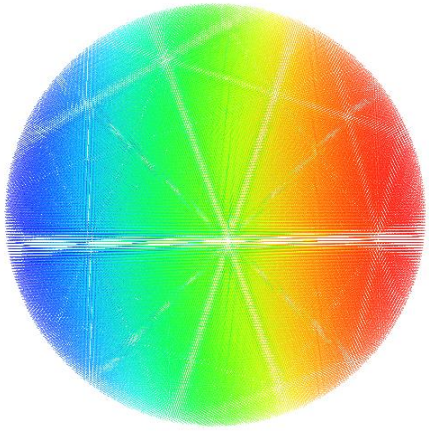
DrawTriangles or DrawFigure (*drawmode=face*)

2.4.2. XML file: Casedef-Geometry-Commands

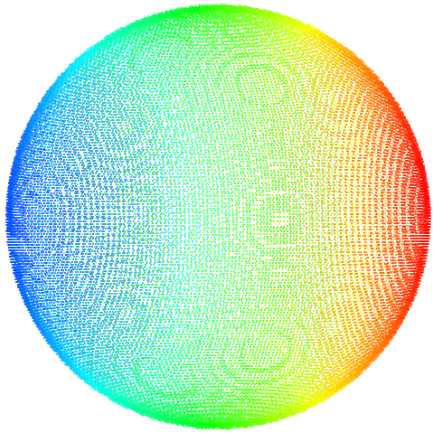
ROUNDED SHAPES: SPHERE

```
– <mainlist>
    <setmkbound mk="0"/>
    <setdrawmode mode="solid"/>
  – <drawsphere radius="0.8">
      <point x="1" y="1" z="1"/>
    </drawsphere>
    <setdrawmode mode="face"/>
  – <drawsphere radius="0.8">
      <point x="1" y="1" z="1"/>
    </drawsphere>
  </mainlist>
```

<drawsphere>: draws a sphere with the center point and the radius



Sphere
(*drawmode=solid*)



Sphere
(*drawmode=face*)

```
<setdpctes ctesphere="0.4"/>
<setdpctes ctespherenumsides="40"/>
```

when *face:*

ctesphere indicates the width of the sphere

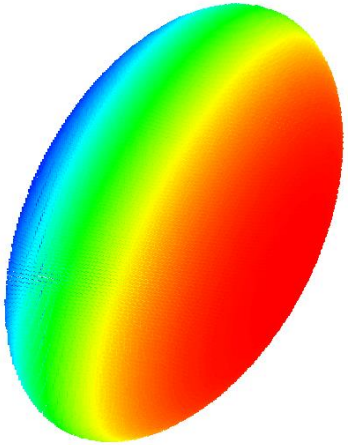
ctespherenumsides indicates the number of triangles used to create the VTK of polygons

2.4.2. XML file: Casedef-Geometry-Commands

ROUNDED SHAPES: ELLIPSOID

<drawellipsoid>: draws an ellipsoid with two center points and the radius

```
<mainlist>
    <setmkbound mk="0" />
    <setdrawmode mode="solid" />
    <drawellipsoid radius="1.8">
        <point x="1" y="0" z="0" />
        <point x="1" y="1" z="1" />
    </drawellipsoid>
    <setdrawmode mode="face" />
    <drawellipsoid radius="1.8">
        <point x="1" y="0" z="0" />
        <point x="1" y="1" z="1" />
    </drawellipsoid>
</mainlist>
```



Ellipsoid
(*drawmode=solid*)

Ellipsoid
(*drawmode=face*)

when *face:*

```
<setdpctes cteellipsoid="0.5" />
```

cteellipsoid indicates the width of the ellipsoid

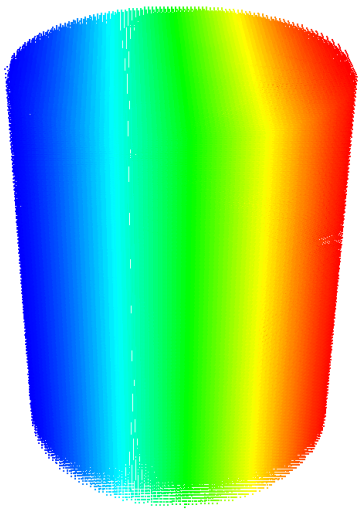
2.4.2. XML file: Casedef-Geometry-Commands

ROUNDED SHAPES: CYLINDER

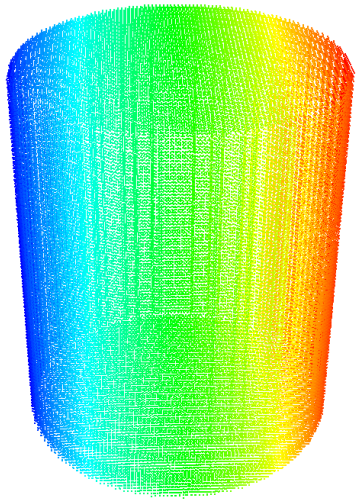
```
- <mainlist>
    <setmkbound mk="0"/>
    <setdrawmode mode="solid"/>
  - <drawcylinder radius="1" mask="0">
      <point  x="1.5" y="1.5" z="0.5"/>
      <point  x="1.5" y="1.5" z="3"/>
    </drawcylinder>
    <setdrawmode mode="face"/>
  - <drawcylinder radius="1" mask="0">
      <point  x="3.5" y="2.5" z="0.5"/>
      <point  x="3.5" y="2.5" z="3"/>
    </drawcylinder>
  </mainlist>
```

<drawcylinder>: draws a cylinder with two points and radius

ask indicates the faces to be hide



Cylinder
(*drawmode=solid*)



Cylinder
(*drawmode=face*)

```
<setdpctes ctecylindertube="0.6"/>
<setdpctes ctecylindercover="0.7"/>
<setdpctes ctecylindernumsides="40"/>
```

when *face:*

ctecylindertube indicates the width of the tube

ctecylindercover indicates the width of the covers

ctespherenumsides indicates the number of triangles used to create the VTK of polygons

2.4.2. XML file: Casedef-Geometry-Commands


```
- <mainlist>
    <setmkbound mk="0"/>
    <drawfilestl file="File.stl"/>
    <drawfileply file="File.ply"/>
    <drawfileply file="File.vtk"/>
  - <drawfilestl file="File.stl">
      <drawmove x="0.5" y="0" z="0"/>
      <drawrotate angx="10" angy="15" angz="30"/>
      <drawscale x="1" y="1" z="0.8"/>
    </drawfilestl>
  - <drawfileply file="File.ply">
      <drawmove x="0.5" y="0" z="0"/>
    </drawfileply>
  - <drawfileply file="File.ply">
      <drawmove x="0.5" y="0" z="0"/>
      <drawrotate angx="10" angy="15" angz="30"/>
    </drawfileply>
  - <drawfileply file="File.ply">
      <drawrotate angx="10" angy="15" angz="30"/>
    </drawfileply>
  - <drawfilevtk file="File.vtk">
      <polyselec>points</polyselec>
    </drawfilevtk>
  - <drawfilevtk file="File.vtk">
      <polyselec>points | lines</polyselec>
    </drawfilevtk>
  - <drawfilevtk file="File.vtk">
      <polyselec>triangles</polyselec>
    </drawfilevtk>
  - <drawfilevtk file="File.vtk">
      <polyselec>polygons</polyselec>
    </drawfilevtk>
</mainlist>
```

EXTERNAL GEOMETRIES

<drawfilevtk>: load a VTK file to be converted into points

<drawfileply>: load a PLY file to be converted into points

<drawfilestl>: load a STL file to be converted into points

Some modifications can be applied to the VTK, PLY or STL

drawmove a displacement is applied to the external object

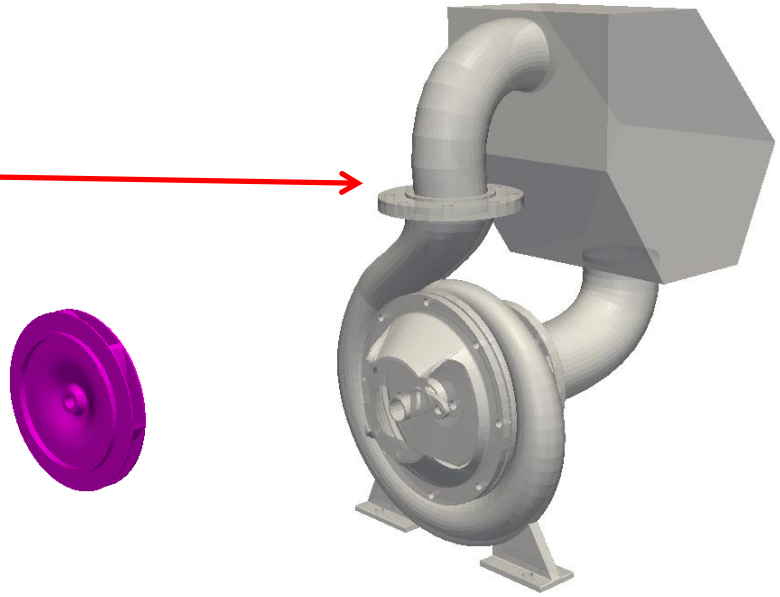
drawrotate a rotation is applied to the external object

drawscale scaling is applied to the external object

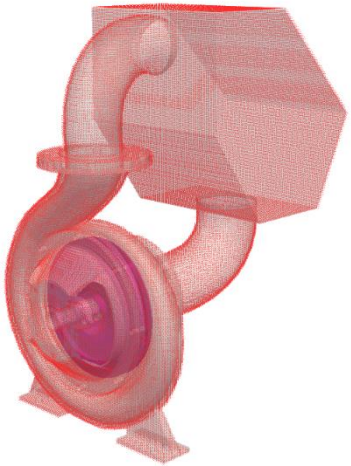
2.4.2. XML file: Casedef-Geometry-Commands

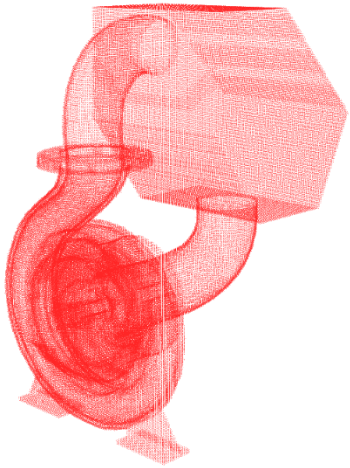
EXTERNAL GEOMETRIES

```
- <mainlist>
    <setshapemode>real | bound | dp</setshapemode>
    <setmkbound mk="0"/>
    <drawfilevtk file="pump_fixed.vtk"/>
    <setmkbound mk="1"/>
    <drawfilevtk file="pump_moving.vtk"/>
    <setmkfluid mk="0"/>
  - <fillbox x="0.14" y="-0.1" z="-0.39">
      <modefill>void</modefill>
      <point x="-0.6" y="-0.39" z="-0.8"/>
      <size x="0.9" y="0.68" z="0.52"/>
    </fillbox>
  </mainlist>
```



from VTK to points





2.4.2. XML file: Casedef-Geometry-Commands


```
- <mainlist>
    <setmkfluid mk="0"/>
    <fillvoidpoint x="3" y="2" z="1"/>
  - <fillpoint x="3" y="2" z="1">
      <modefill>void</modefill>
    </fillpoint>
  - <fillpoint x="1" y="1" z="1" mkfluid="0">
      <modefill>fluid</modefill>
    </fillpoint>
  - <fillpoint x="1" y="1" z="1" mkbound="0">
      <modefill>bound</modefill>
    </fillpoint>
  - <fillpoint x="2" y="2" z="2" mkfluid="2" mkbound="8">
      <modefill>border | void | fluid | bound</modefill>
    </fillpoint>
  - <fillbox x="0" y="1" z="0">
      <modefill>border</modefill>
      <point x="0.1" y="1" z="1.1"/>
      <size x="3" y="4" z="2"/>
    </fillbox>
  - <fillprism x="2" y="3" z="5">
      <point x="0" y="0" z="0"/>
      <point x="1" y="0" z="0"/>
      <point x="0" y="1" z="0"/>
      <point x="0" y="0" z="0.5"/>
      <point x="1" y="0" z="0.5"/>
      <point x="0" y="1" z="0.5"/>
      <modefill>void</modefill>
    </fillprism>
    <debugout/>
  </mainlist>
```

FILLING DOMAINS

<fillpoint>: fills with points starting from the seed

<fillbox>: fills with points starting from the seed within the limits defined by a box

<fillfigure>: fills with points starting from the seed within the limits defined by a figure

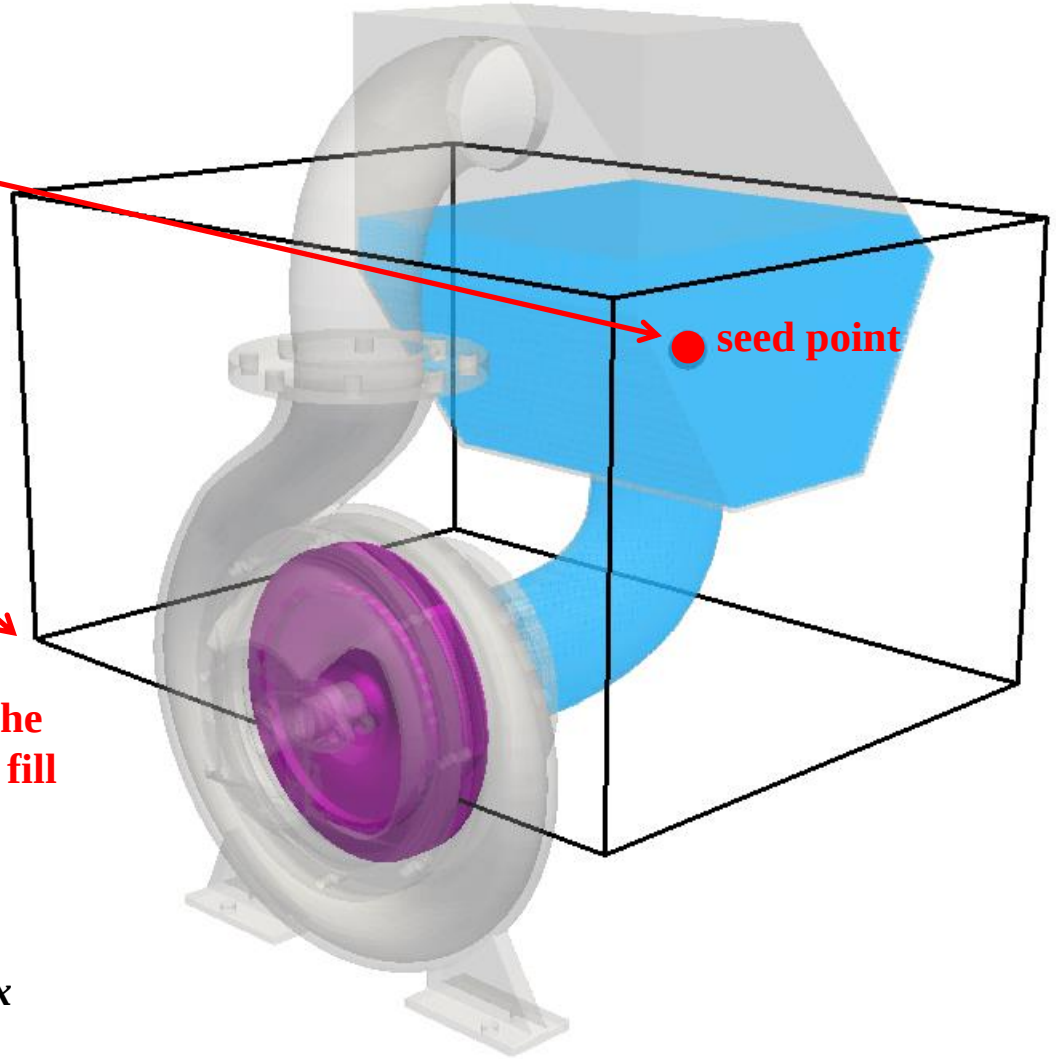
<fillprism>: fills with points starting from the seed within the limits defined by a prism

<modefill> indicates what type of points can be filled with *void*, *fluid*, *bound*, it fills with that type of points inside the specified limits or the presence of a given type of point using *border*

2.4.2. XML file: Casedef-Geometry-Commands


```
– <mainlist>
    <setshapemode>real | bound | dp</setshapemode>
    <setmkbound mk="0"/>
    <drawfilevtk file="pump_fixed.vtk"/>
    <setmkbound mk="1"/>
    <drawfilevtk file="pump_moving.vtk"/>
    <setmkfluid mk="0"/>
  – <fillbox x="0.14" y="-0.1" z="-0.39">
      <modefill>void</modefill>
      <point x="-0.6" y="-0.39" z="-0.8"/>
      <size x="0.9" y="0.68" z="0.52"/>
    </fillbox>
  </mainlist>
```

FILLING DOMAINS



filling with *fluid* while *void* and before the limits defined by a *box*

2.4.2. XML file: Casedef-Geometry-Commands

REDRAW

GenCase employs a 3-D Cartesian lattice to locate particles. These particles are created at the nodes of that lattice. A value of "mk" is assigned to each node according to the type of particle (boundary, fluid or void) and to define a special behaviour during the simulation.

```
<mainlist>
    <redraw />
    <redraw mkfluid="0" />
    <redraw mkbound="5" />
</mainlist>
```

<redraw> assigns the "mk" defined by the last <setmkvoid>, <setmkbound> or <setmkfluid> to all nodes that follow a given condition.

<redrawnear> allows to indicate the nodes that will be modified if there is a neighbouring node that follows some given condition.

```
<mainlist>
    <redrawnear targettp="fluid" bordertp="bound" />
    <redrawnear times="2" targettp="fluid" bordertp="bound" />
    <redrawnear targettp="void" bordertp="bound" />
    <redrawnear targettp="fluid" targetmk="0" bordertp="bound" />
    <redrawnear targettp="fluid" targetmk="0" bordertp="bound" bordermk="2" />
</mainlist>
```

```
<mainlist>
    <redrawbox mkfluid="0">
        <point x="0.1" y="1" z="1.1" />
        <size x="3" y="4" z="2" />
    </redrawbox>
    <redrawnearbox times="3" targettp="void">
        <point x="0.1" y="1" z="1.1" />
        <size x="3" y="4" z="2" />
    </redrawnearbox>
</mainlist>
```

<redrawbox> and **<redrawnearbox>** perform the same operation but limits the domain affected.

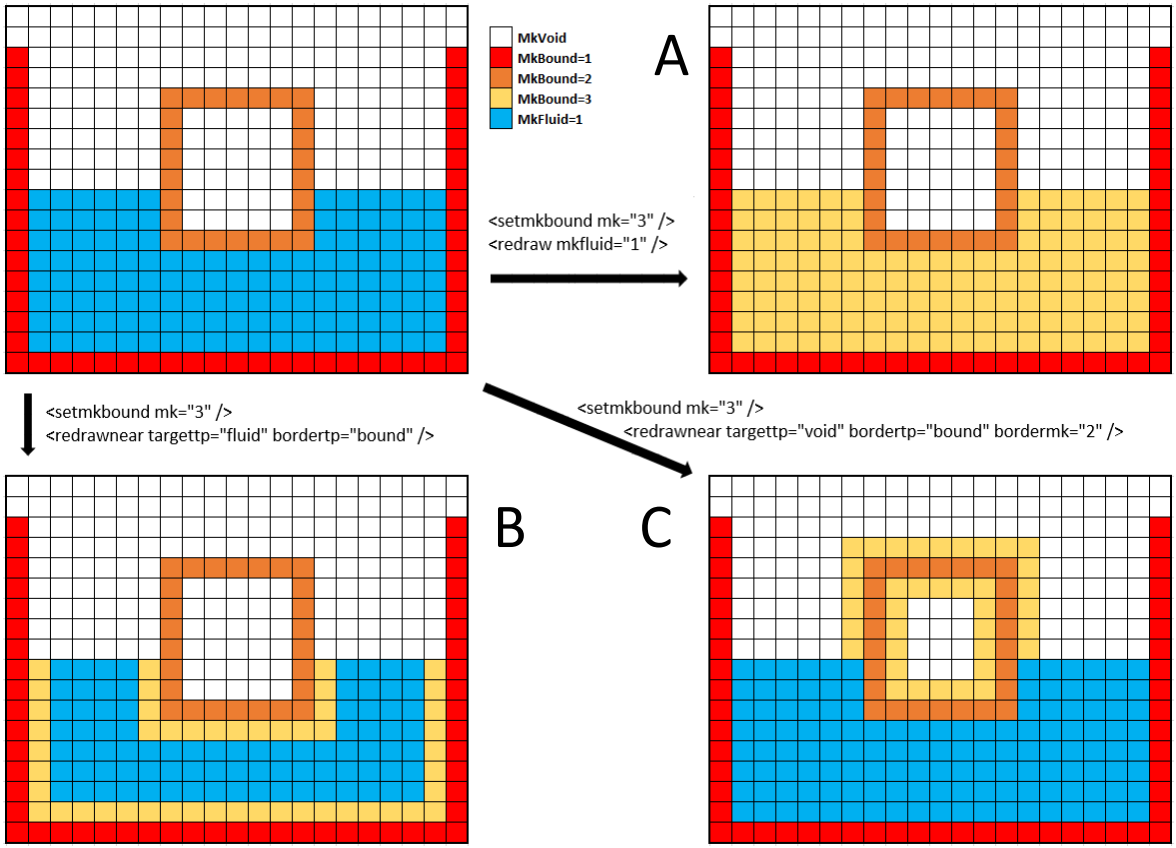
2.4.2. XML file: Casedef-Geometry-Commands

REDRAW

Example A:

```
<setmkbound mk="3" />
<redraw mkfluid="1" />
```

defines the "mk" for the nodes starting from here
changes all previous nodes with mkfluid=1



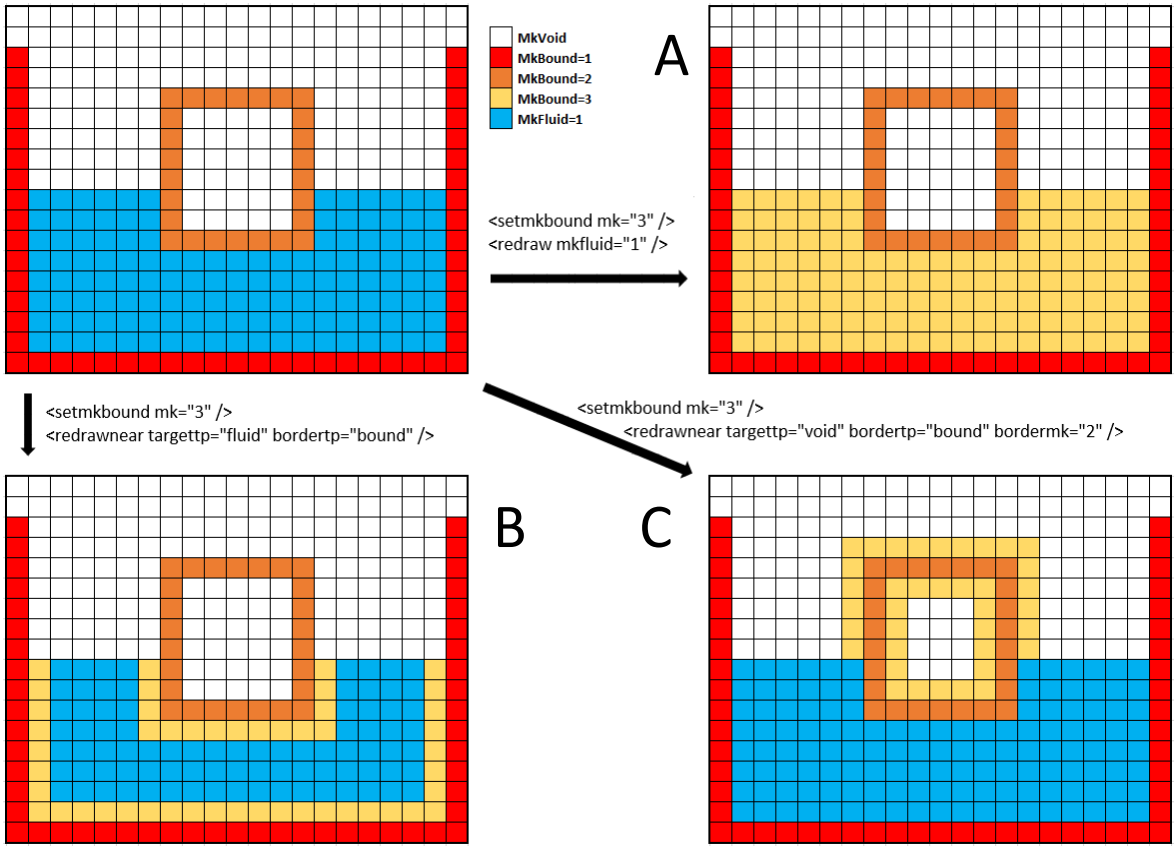
Note that squares are used here to represent the individual nodes (particles will be created in those nodes) for clarity.

2.4.2. XML file: Casedef-Geometry-Commands

REDRAW

Example B: modifies all nodes of fluid (targettp="fluid") that have a neighbouring node of type boundary (bordertp="bound")

```
<redrawnear targettp="fluid" bordertp="bound" />
```



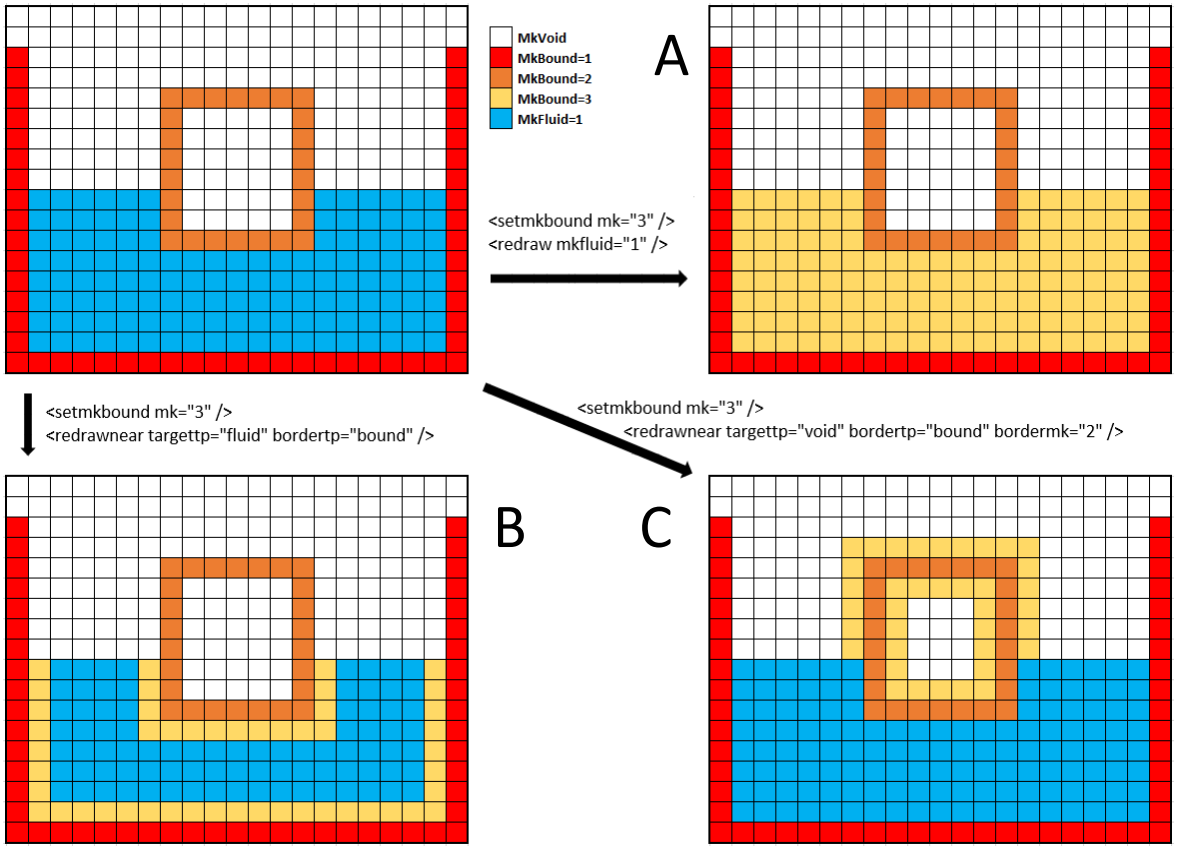
Note that squares are used here to represent the individual nodes (particles will be created in those nodes) for clarity.

2.4.2. XML file: Casedef-Geometry-Commands

REDRAW

Example C: modifies all nodes of type void (targettp="void") that have a neighbouring node of type boundary (bordertp="bound") and with mk=2 (bordermk="2")

```
<redrawnear targettp="void" bordertp="bound" bordermk="2" />
```

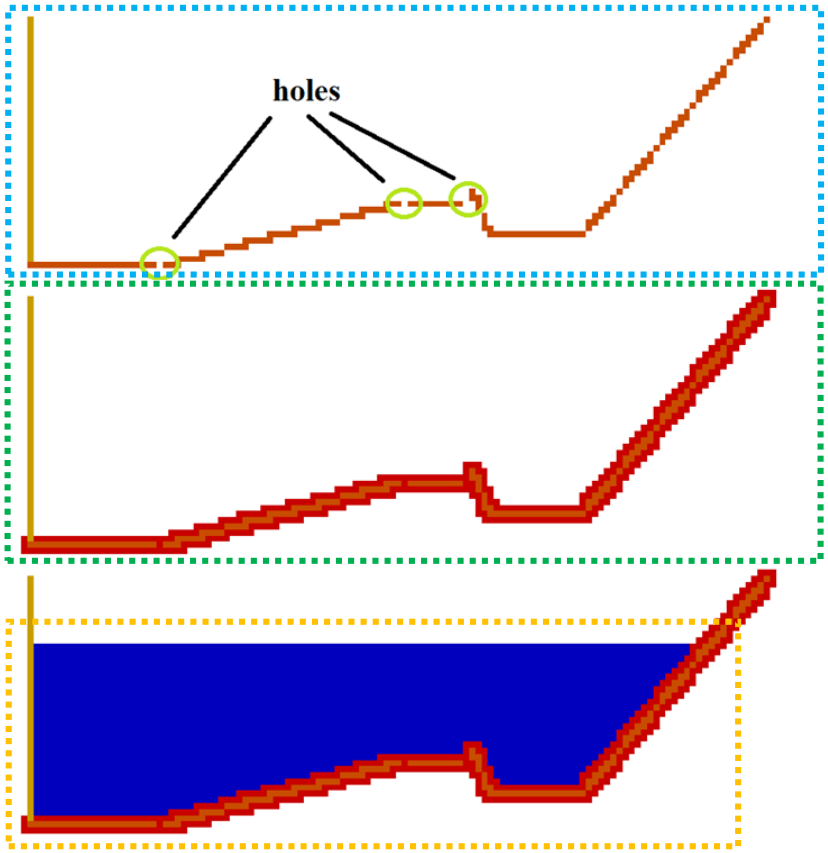


Note that squares are used here to represent the individual nodes (particles will be created in those nodes) for clarity.

2.4.2. XML file: Casedef-Geometry-Commands

REDRAW

Redraw commands can be used to close the holes in complex boundary geometries before filling areas with fluid particles.




```
<mainlist>
    <setshapemode>real | bound</setshapemode>
    <setdrawmode mode="full" />
    <!-- Creates vertical piston. -->
    <setmkbound mk="0" />
    <drawline>
        <point x="0" y="0" z="2" />
        <point x="0" y="0" z="0" />
    </drawline>
    <!-- Creates bottom from external geometry. -->
    <setmkbound mk="1" />
    <drawfilevtk file="Bottom.vtk" />
    <!-- Fills the holes. -->
    <setmkbound mk="2" />
    <redrawnear times="1" targettp="void"
        bordertp="bound" bordermk="1" />
    <!-- Fills fluid area. -->
    <setmkfluid mk="0" />
    <fillbox x="0.5" y="0" z="0.5">
        <modefill>void</modefill>
        <point x="0" y="-1" z="0" />
        <size x="6" y="2" z="1.5" />
    </fillbox>
</mainlist>
```

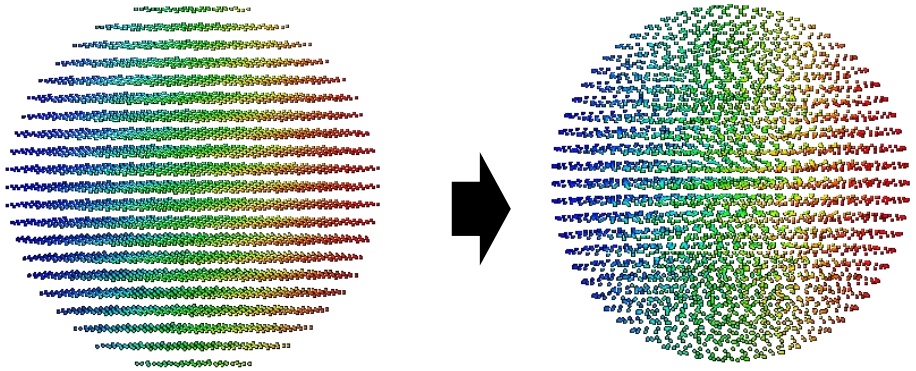
2.4.2. XML file: Casedef-Geometry-Commands

FREEDRAW

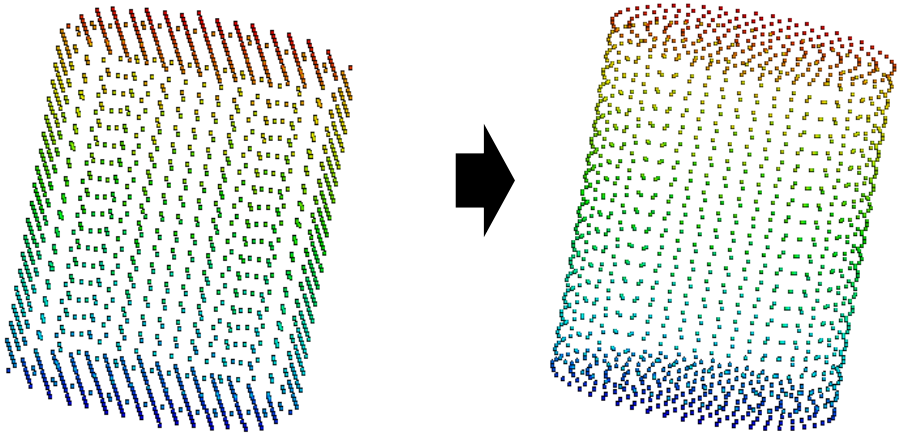
setfrdrawmode command is used to create particles at free nodes (not in the nodes of the cubic lattice) but maintaining the same initial particle distance “dp”

```
<setdrawmode mode="full" />
<setmkbound mk="0" name="Sphere" />
<setfrdrawmode auto="true" />
<drawsphere radius="5.0">
    <point x="0" y="0" z="5.0" />
</drawsphere>
<setfrdrawmode auto="false" />
```

Normal mode **vs.** **FrDraw mode**



```
<setdrawmode mode="face" />
<setmkbound mk="0" name="Cylinder" />
<setfrdrawmode auto="true" />
<drawcylinder radius="4.0" mask="0">
    <point x="0" y="0" z="0" />
    <point x="0" y="0" z="10" />
</drawcylinder>
<setfrdrawmode auto="false" />
```



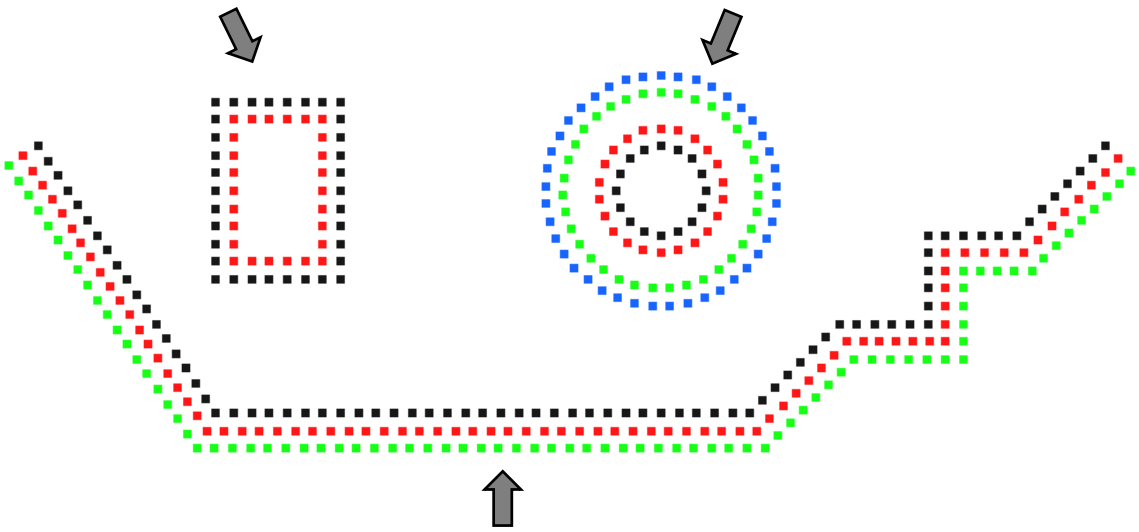
2.4.2. XML file: Casedef-Geometry-Commands

LAYERS

The following commands allow us to create **several layers automatically according to dp**: <drawbox>, <drawextrude>, <drawsphere> and <drawcylinder>.
The asterisk defines the shapes saved by <shapeout>.

```
<drawbox>
   <layers vdp="0,-1"/>
</drawbox>
```

```
<drawcylinder>
   <layers vdp="0,1,3,4"/>
</drawcylinder>
```



```
<drawextrude>
   <layers vdp="0,-1,-2"/>
</drawextrude>
```

```
<mainlist>
    <setshapemode>actual | bound</setshapemode>
    <setfrdrawmode auto="true" />
    <setmkbound mk="0" />
    <drawbox>
        <boxfill>all</boxfill>
        <point x="2" y="-1" z="1.5" />
        <size x="1.4" y="2" z="2" />
        <layers vdp="0*,-1" />
    </drawbox>
    <drawcylinder radius="0.5" mask="2">
        <point x="7" y="-1" z="2.5" />
        <point x="7" y="1" z="2.5" />
        <layers vdp="0,1,3,4*" />
    </drawcylinder>
    <drawextrude closed="false">
        <point x="0" y="0" z="3" />
        <point x="2" y="0" z="0" />
        <point x="8" y="0" z="0" />
        <point x="9" y="0" z="1" />
        <point x="10" y="0" z="1" />
        <point x="10" y="0" z="2" />
        <point x="11" y="0" z="2" />
        <point x="12" y="0" z="3" />
        <extrude x="0" y="1" z="0" />
        <layers vdp="0*,-1,-2" />
    </drawextrude>
    <shapeout file="" reset="true" />
</mainlist>
```

2.4.2. XML file: Casedef-Geometry-Commands

VARIABLES

New version of GenCase allows to define variables in the XML and numerical expressions with these variables are solved by GenCase and DualSPHysics programs.

The commands to define user variables are:

```
<newvar size="0.4" _rem="Defines size" _print="1" />
<newvar var1="10" var2="25.5" var3="var1+var2" var4="true" />
<newvarcte cte1="1.0" cte2="var1+var2/size" cte3="(var1>=cte2)!=var3" />
<newvarstr file1="cube.stl" file2="sphere.stl" />
<newvarstrcte text="The file is [file1]." />
```

<newvar>: creates one or several numerical variables (using double precision).

Values ***true*** and ***false*** are stored as 1 and 0.

The variables are created in order so previous variables can be used in the following ones.

E.g.: `var1=10, var2=25.5, var3=var1+var2, var4=true`

<newvarstr>: creates one or several text variables.

<newvarcte> and **<newvarstrcte>:** creates variables (numerical and text) as constants, so these variables cannot be changed.

_rem attribute is used for comments and **_print** equal to true (or not zero) shows the values stored in the variables on the screen .

2.4.2. XML file: Casedef-Geometry-Commands

VARIABLES

The user-defined variables can be defined in section **<predefinition>** and in the drawing **command lists**. The variables defined in **<predefinition>** can be used to define *Dp* and the limits of the domain. The user-defined variables are global, so they can be only created once and can be used anywhere.

```
<geometry>
    <predefinition>
        <newvarcte sizex="1.2" _rem="size domain"/>
        <newvarcte sizez="sizex*0.8" _rem="size domain"/>
        <newvarcte Dp="sizex/100" _rem="particle size"/>
        <newvarstr filestl="cube.stl"/>
    </predefinition>
    <definition dp="#Dp">
        <pointmin x="#-sizex/2" y="0" z="0"/>
        <pointmax x="#-sizex" y="0" z="#sizez"/>
    </definition>
    <commands>
        <list name="DrawShape">
            <newvar posx2="posx1+0.2"/>
            <drawfilestl file="$[filestl]"/>
            <move x="#sizex*posx2" y="0" z="0"/>
        </list>
        <mainlist>
            <newvarcte posx1="-sizex*0.5"/>
            <runlist name="DrawShape"/>
        </mainlist>
    </commands>
</geometry>
```

<predefinition> section

The variables created in <predefinition> are used to define the domain.

User-defined list of commands.

Main drawing list of commands.

2.4.2. XML file: Casedef-Geometry-Commands

VARIABLES

GenCase creates some variables automatically (as constants) according to values in the XML file, so the user-defined variables can be created from the previous ones. The available constants are:

- **Gravity_x, Gravity_y, Gravity_z, Rhop0**
  Gravity values and reference density of the fluid are loaded at the beginning, before evaluating <predefinition> section.

- **Dp**
  Initial inter-particle distance loaded from XML and after evaluating <predefinition> section. Dp value can be modified using variables in <predefinition>.

- **PosMin_x, PosMin_y, PosMin_z**
  PosMax_x, PosMax_y, PosMax_z
  Domain limits loaded or calculated from XML file.

- **CaseName**
  Data2D, Data2DPosy
  H, MassBound, MassFluid
  Constant variables calculated automatically starting from previous constants and configuration in XML file. These constants are created before evaluating the drawing command lists.

2.4.2. XML file: Casedef-Geometry-Commands

VARIABLES

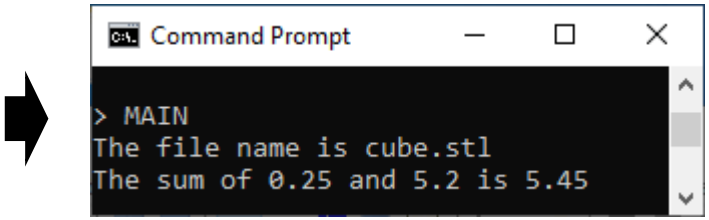
The commands **<setvar>** and **<setvarstr>** allow us to modify the user-defined variables (non-constant). **<setvar>** changes numerical variables and **<setvarstr>** text variables. The reference to text or numerical variables must use **[variable]** in text variable definitions.

```
<newvarcte Cte1="0.1" Cte2="0.2" Cte3="0.3" _rem="Constant values" _print="1" />
<newvar SizeX="0.25" SizeZ="(Dp+2)*2.8" dpd2="Dp/2" _rem="Some variables" />
<setvar SizeZ="SizeX+Cte1*Cte2" _rem="Changes value of SizeZ" _print="true" />
<setvar dpd2="dpd2+Dp" SizeX="SizeX*dpd2" />

<newvarstrcte Text1="Hello" Text2="bye" _rem="Text values" _print="true" />
<newvarstr Text3="AAAA" Text4="[Text1]_[Text2]. SizeX is [SizeX]" _rem="More text values" />
<setvarstr Text3="[Text1]" Text4="BBBB" _rem="Modify values" _print="1" />
```

The command **<printf>** shows the values of text variables, numerical variables and numerical expressions.

```
<mainlist>
    <newvarstr name="cube" ext=".stl" file="[name][ext]"/>
    <printf text="The file name is #file#"/>
    <newvar sx="0.25" sz="5.2"/>
    <printf text="Sum of #sx# and #sz# is #sx+sz#"/>
</mainlist>
```



2.4.2. XML file: Casedef-Geometry-Commands

VARIABLES

The user-defined variables can be used in most of the drawing commands and the different sections of the XML for GenCase and DualSPHysics programs.

```
<mainlist>
    <newvarstr filestl="cube.stl"/>
    <newvar inix="0.2" iniz="0.1"/>
    <move x="#inix" y="0" z="#iniz" />
    <drawbox>
        <boxfill>bottom</boxfill>
        <point x="#PosMin_x" y="0" z="#PosMin_z" />
        <size x="#Dp*20" y="1" z="#Dp*8" />
    </drawbox>
    <drawfilestl file="$[filestl]" if="Dp<=0.1" />
    <runlist name="DrawShape" times="#10/inix" />
</mainlist>
...
<initials>
    <velocity mkfluid="1" x="0" y="0" z="#-Gravity_z*2"/>
</initials>
...
<floatings>
    <floating mkbound="2">
        <center x="#inix" y="0" z="#iniz" />
    </floating>
</floatings>
```

The use of **numerical variables or expressions** must start with **#** in order to be recognized and evaluated.
Numerical or logical expressions can be used in the attribute **`if`** recognized by all drawing commands.

The use of **text variables** or expressions must start with **$** and the name of the variable **in square brackets**.

User-defined variables can be used in the following sections of XML: **<geometry>**, **<initials>**, **<floatings>**, **<properties>**, **<motion>**, **<normals>**, **<special>** and **<parameters>**

2.4.2. XML file: Casedef-Geometry-Commands

VARIABLES

The numerical or logical expressions can use: values, variables, mathematical operators, comparison operators, logical operators, parenthesis and functions.

- Mathematical operators: **+**, **-**, *****, **/**
- Comparison operators: **==** (equal), **!=** (not equal), **<**, **>**, **<=**, **>=**
- Logical operators: **!** (not), **||** (or), **@@** (and)
- Functions:
  pi(), **e**(),
  int(a), **float**(a), **abs**(a), **floor**(a), **ceil**(a), **round**(a), **fmod**(a,b), **fmodr**(a,b)=round(fmod(a,b)),
  min(a,b), **min**(a,b,c) , **min**(a,b,c,d), **max**(a,b), **max**(a,b,c), **max**(a,b,c,d),
  sqrt(a), **exp**(a), **log**(a), **log10**(a), **pow**(a,b),
  sin(radians), **cos**(radians), **tan**(radians), **sindg**(degrees), **cosdg**(degrees), **tandg**(degrees),
  sinh(a), **cosh**(a), **tanh**(a), **asin**(a), **acos**(a), **atan**(a),
  randinit(seed), **random**(), //to initialise the random generator and to obtain random numbers.
  eval(condition, v1, v2), //returns v1 when condition is true or v2 in the other case.
  wavelength(gravity, depth, waveperiod) //returns wave length.

2.4.2. XML file: Casedef-Geometry-Commands

VARIABLES

Use of variables in the XML for DualSPHysics program:
DualSPHysics defines a set of constant variables when execution starts. These variables and the user-defined variables in the <geometry> section can be used in the XML sections loaded by DualSPHysics (<special> and <parameters>).

List of variables automatically created by DualSPHysics: CaseName, Data2D, Data2DPosy, H, B, Gamma, Rhop0, Dp, Gravity_x, Gravity_y, Gravity_z, MassFluid, MassBound, TimeMax, TimeOut.

All user-defined variables in <geometry> section are included in <execution><uservars> by default. However, the list of variables to use in DualSPHysics can be defined using the command <exportvar>.

```
<casedef>
    <geometry>
        <mainlist>
            <newvarstr file="box.stl"/>
            <newvar numbers="12"/>
            <newvar sizex="0.25" sizez="5.2"/>
            <newvar sizetot="sizex+sizez"/>
            <newvar num1="0.1" num2="0.2"/>
            <exportvar vars="file,num*,sizez"/>
        </mainlist>
    </geometry>
</casedef>
```

```
<execution>
    <special/>
    <parameters/>
    <particles/>
    <constants/>
    <uservars>
        <varstr name="file" value="box.stl"/>
        <varnum name="num1" value="0.1"/>
        <varnum name="num2" value="0.2"/>
        <varnum name="numbers" value="12"/>
        <varnum name="sizez" value="5.2"/>
    </uservars>
    <motion/>
    <vtkout/>
</execution>
```

2.5. XML file: Casedef-Initials


```
- <geometry>
  - <definition dp="0.01">
      <pointmin x="-1" y="-0.05" z="-0.05"/>
      <pointmax x="2" y="1.1" z="2"/>
    </definition>
  - <commands>
    - <mainlist>
        <setshapemode>real | dp | bound</setshapemode>
        <setdrawmode mode="full"/>
        <setmkfluid mk="1"/>
      - <drawsphere radius="0.15">
          <point x="-0.55" y="0.5" z="0.18"/>
        </drawsphere>
        <setmkfluid mk="2"/>
      - <drawbox>
          <boxfill>solid</boxfill>
          <point x="1.4" y="0.35" z="0.01"/>
          <size x="0.3" y="0.3" z="0.3"/>
        </drawbox>
      </mainlist>
    </commands>
  </geometry>
- <initials>
    <velocity mkfluid="1" x="1.05" y="0" z="4.905"/>
    <velocity mkfluid="2" x="-0.875" y="0" z="5.886"/>
  </initials>
```

INITIAL VELOCITIES

<initials>: special behaviours can be imposed to a set of fluid particles labeled with a *mk*, such as:

<velocity> initial velocity defined by a vector

<velwave> a solitary wave defined by *depth* and *amplitude*

2.5. XML file: Casedef-Initials

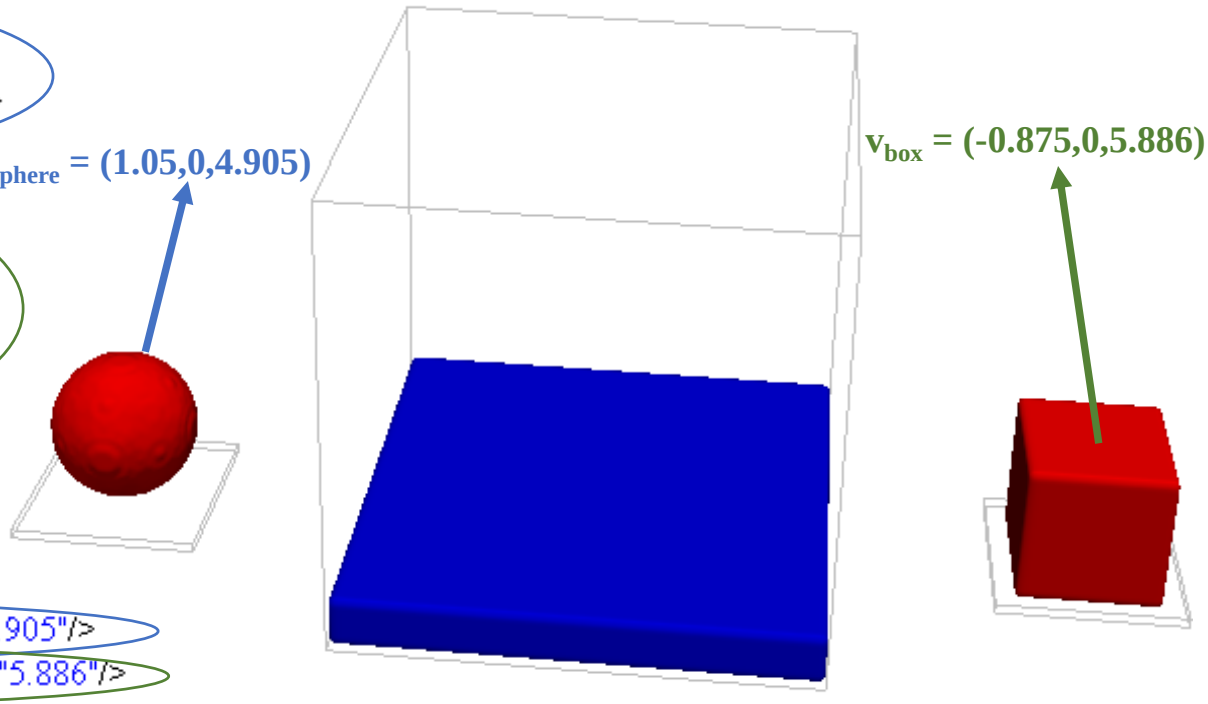

```
- <geometry>
  - <definition dp="0.01">
      <pointmin x="-1" y="-0.05" z="-0.05"/>
      <pointmax x="2" y="1.1" z="2"/>
    </definition>
  - <commands>
    - <mainlist>
        <setshapemode>real | dp | bound</setshapemode>
        <setdrawmode mode="full"/>
        <setmkfluid mk="1"/>
      - <drawsphere radius="0.15">
          <point x="-0.55" y="0.5" z="0.18"/>
        </drawsphere>
        <setmkfluid mk="2"/>
      - <drawbox>
          <boxfill>solid</boxfill>
          <point x="1.4" y="0.35" z="0.01"/>
          <size x="0.3" y="0.3" z="0.3"/>
        </drawbox>
      </mainlist>
    </commands>
  </geometry>
- <initials>
    <velocity mkfluid="1" x="1.05" y="0" z="4.905"/>
    <velocity mkfluid="2" x="-0.875" y="0" z="5.886"/>
  </initials>
```

INITIAL VELOCITIES

different initial velocities are imposed to two volumes of fluid *mk=1* (sphere) and *mk=2* (box)

v_{sphere} = (1.05,0,4.905)

v_{box} = (-0.875,0,5.886)

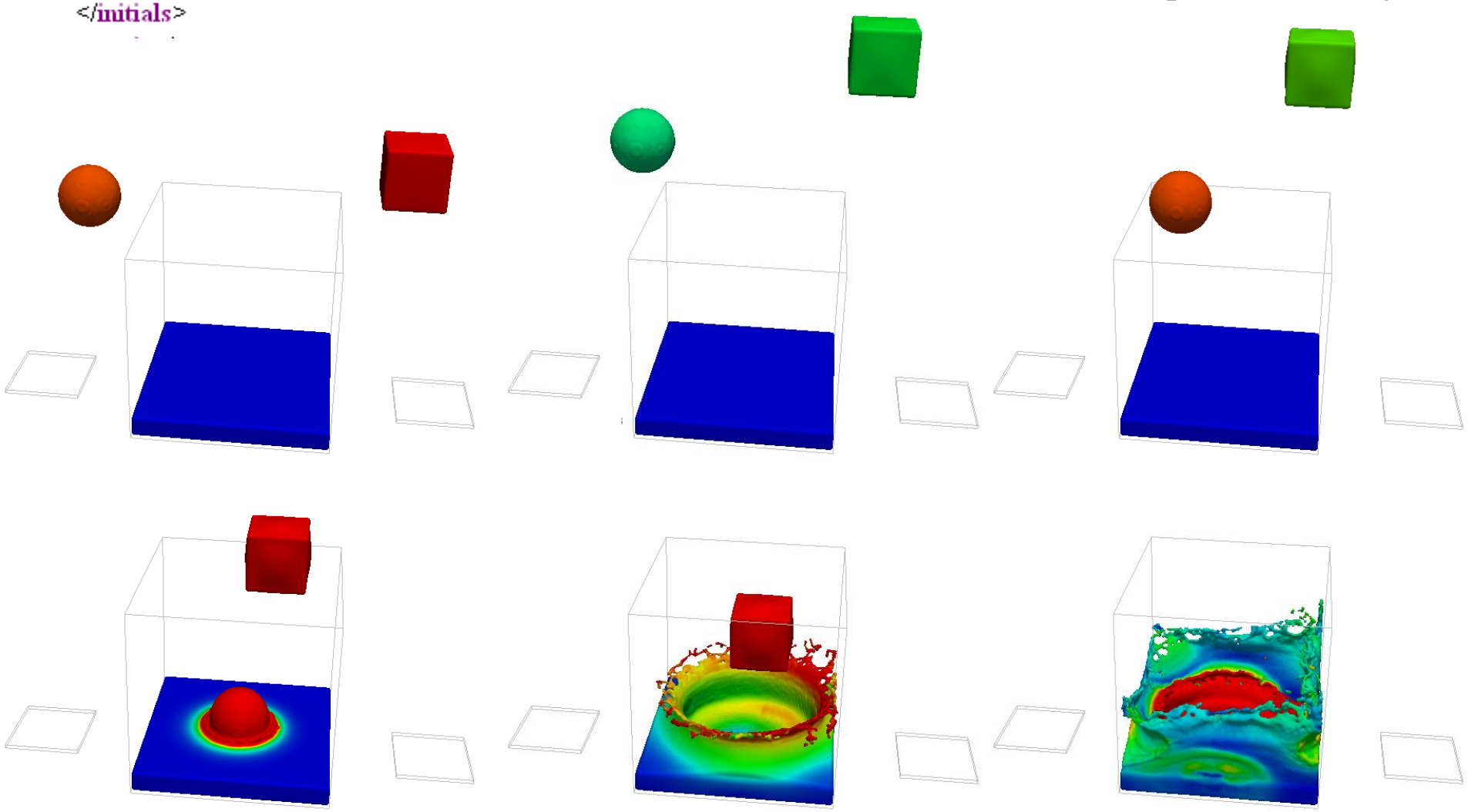


2.5. XML file: Casedef-Initials


```
– <initials>
    <velocity mkfluid="1" x="1.05" y="0" z="4.905"/>
    <velocity mkfluid="2" x="-0.875" y="0" z="5.886"/>
  </initials>
```

INITIAL VELOCITIES

colour represents velocity



2.5. XML file: Casedef-Initials

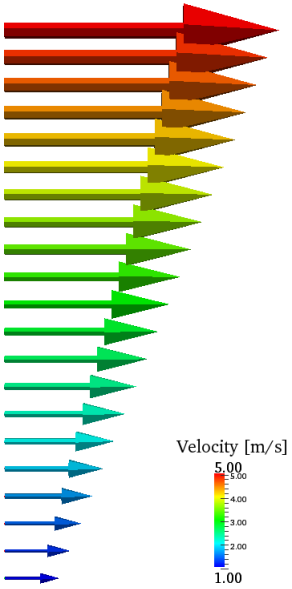
INITIAL VELOCITIES: PROFILES

Linear velocity profile from 1 to 5 m/s

```
<initials>
    <velocity mkfluid="7">
        <direction x="1" y="0" z="0" />
        <velocity2 v="1" z="0" v2="5" z2="1" units_comment="m/s" />
    </velocity>
</initials>
```

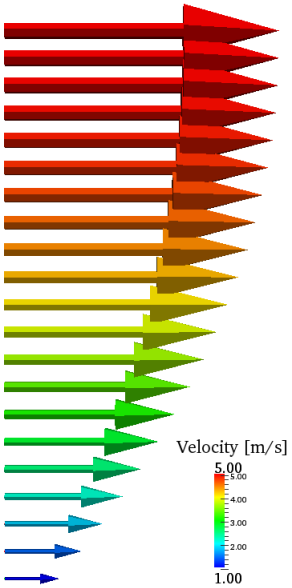
Parabolic velocity profile from 1 to 5 m/s

```
<initials>
    <velocity mkfluid="7">
        <direction x="1" y="0" z="0" />
        <velocity3 v="1" z="0" v2="3.1" z2="0.3" v3="5" z3="1" units_comment="m/s" />
    </velocity>
</initials>
```



Linear velocity:

z=1 m, v=5 m/s
z=0 m, v=1 m/s



Parabolic velocity:

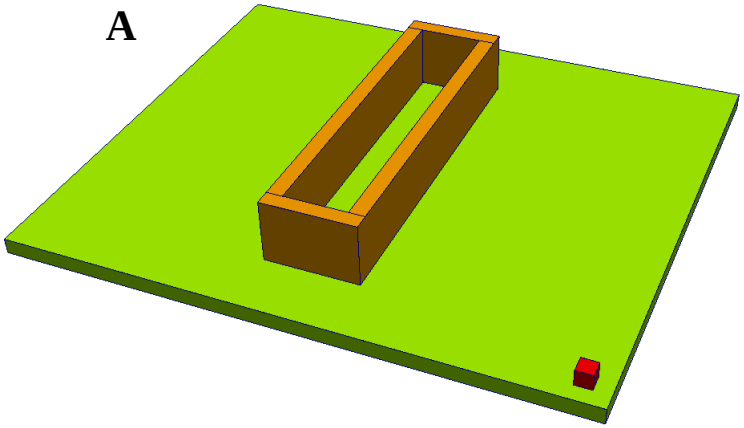
z=1.0 m, v=5.0 m/s
z=0.3 m, v=3.1 m/s
z=0.0 m, v=1.0 m/s

2.5. XML file: Casedef-Initials

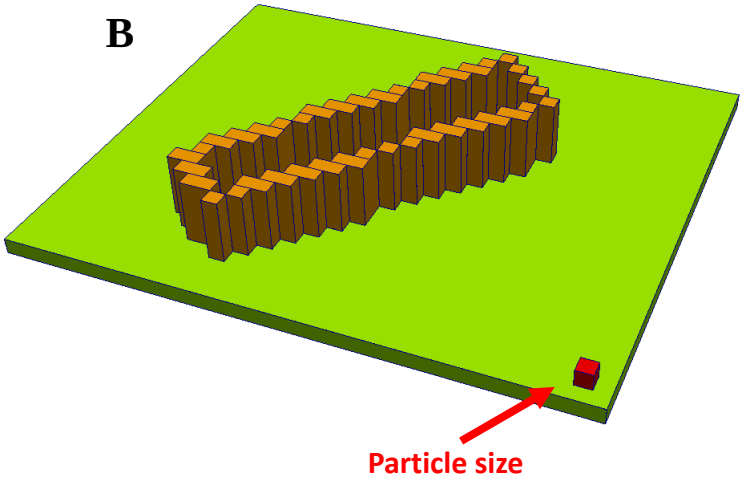
INITIAL DISPLACEMENT

Displacements (<move>) and **rotations** (<rotateaxis>) applied to final position of particles can be defined in <initials> section. It allows to change the position of particles without the limitations of the lattice.

A



B



A. *Creates a simple box.*

```
<mainlist>
    <!-- Box -->
    <setmkbound mk="2" />
    <drawbox>
        <boxfill>all ^ top ^ bottom</boxfill>
        <point x="0.15" y="0.04" z="0" />
        <size x="0.05" y="0.21" z="0.04" />
    </drawbox>
</mainlist>
```

B. *Draws a rotated box (using a lattice with very low resolution).*

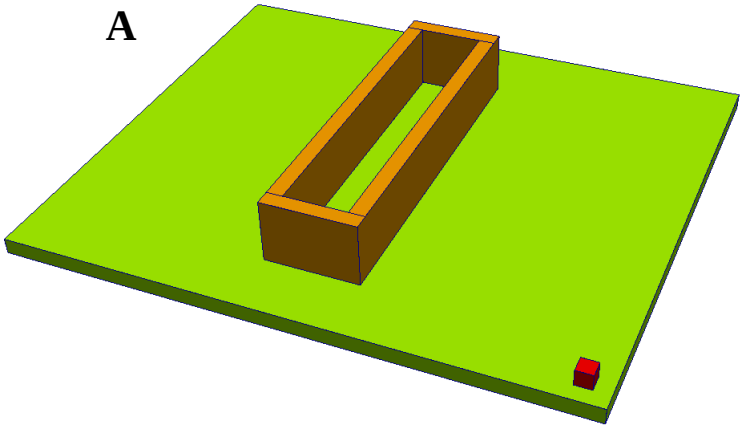
```
<mainlist>
    <!-- Rotation -->
    <rotateline ang="40">
        <point x="0.15" y="0.145" z="0" />
        <point x="0.15" y="0.145" z="1" />
    </rotateline>
    <!-- Box -->
    <setmkbound mk="2" />
    <drawbox>
        <boxfill>all ^ top ^ bottom</boxfill>
        <point x="0.15" y="0.04" z="0" />
        <size x="0.05" y="0.21" z="0.04" />
    </drawbox>
    <matrixreset />
</mainlist>
```

2.5. XML file: Casedef-Initials

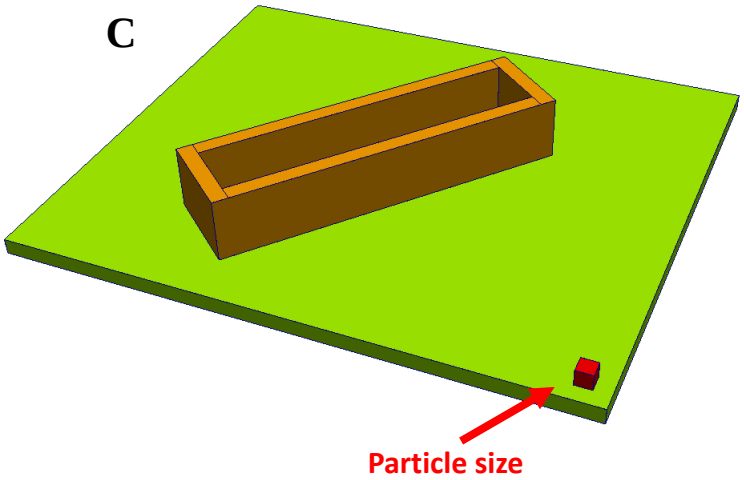
INITIAL DISPLACEMENT

Displacements (<move>) and **rotations** (<rotateaxis>) applied to final position of particles can be defined in <initials> section. It allows to change the position of particles without the limitations of the lattice.

A



C



C. *Applies rotation after creating the particles.*
It is useful to avoid the effects of the lattice when the resolution is low.

```
<geometry>
    <commands>
        <mainlist>
            <!-- Box -->
            <setmkbound mk="2" />
            <drawbox>
                <boxfill>all ^ top ^ bottom</boxfill>
                <point x="0.15" y="0.04" z="0" />
                <size x="0.05" y="0.21" z="0.04" />
            </drawbox>
        </mainlist>
    </commands>
</geometry>
...
<initials>
    <rotateaxis mkbound="2" angle="40" anglesunits="degrees">
        <axisp1 x="0.15" y="0.145" z="0" />
        <axisp2 x="0.15" y="0.145" z="1" />
    </rotateaxis>
</initials>
```

2.6. XML file: Casedef-Floatings


```
<floatings>
    <floating mkbound="0" rhopbody="1300" />
    <floating mkbound="1" relativeweight="1300">
        <linearvelini x="6" y="0" z="6" />
        <angularvelini x="0" y="8" z="0" />
    </floating>
    <floating mkbound="2">
        <massbody value="1300" />
        <center x="11" y="12" z="13" />
        <inertia x="20" y="22" z="24" />
    </floating>
    <floating mkbound="3">
        <massbody value="1300" />
        <center x="11" y="12" z="13" />
        <inertia x="20" y="22" z="24" />
        <linearvelini x="6" y="0" z="6" />
        <angularvelini x="0" y="8" z="0" />
    </floating>
    <floating mkbound="4">
        <massbody value="1300" />
        <inertia x="11" y="22" z="22" />
    </floating>
    <floating mkbound="5">
        <massbody value="1300" />
        <inertiafull>
            <values v11="11" v12="0" v13="0" />
            <values v21="0" v22="22" v23="0" />
            <values v31="0" v32="0" v33="33" />
        </inertiafull>
    </floating>
</floatings>
```

DEFINING FLOATINGS

<floatings>: indicates that a set of particles labelled with the same *mk* constitutes a floating object

Only one of these values can be defined:
rhopbody density of the object
relativeweight in relation to the reference density
massbody total mass of the object

The following variables are computed by GenCase or can be also specified by the user in advance:

center gravity center of the rigid object

inertia moment of inertia of the rigid object (only diagonal values) OR **inertiafull** defines all values *inertia with respect to axes parallel to the reference system and passing through the centre of the object*

linearvelini initial linear velocity of the object

angularvelini initial angular velocity of the object

2.6. XML file: Casedef-Floatings

DEFINING FLOATINGS

<floatings>: indicates that a set of particles labelled with the same *mk* constitutes a floating object

```
<floatings>
    <floating mkbound="1" relativeweight="0.5">
        <translationDOF x="0" y="1" z="1"  />
        <rotationDOF x="1" y="0" z="1"  />
    </floating>
</floatings>
```

<translationDOF> and **<rotationDOF>** can be used to restrain some of the six degrees of freedom (DOF)

<translationDOF> in x-axis (**surge**), in y-axis (**sway**) or in z-axis (**heave**)

x="0" to restrict surge motion OR x="1" to allow surge motion

y="0" to restrict sway motion OR y="1" to allow sway motion

z="0" to restrict heave motion OR z="1" to allow heave motion

<rotationDOF> along x-axis (**roll**), along y-axis (**pitch**) or along z-axis (**yaw**)

x="0" to restrict roll rotation OR x="1" to allow roll rotation

y="0" to restrict pitch rotation OR y="1" to allow pitch rotation

z="0" to restrict yaw rotation OR z="1" to allow yaw rotation

2.6. XML file: Casedef-Floatings

DEFINING FLOATINGS

```
<floatings>
    <floating mkbound="9" relativeweight="2" property="steel + userdef01" />
    <floating mkbound="10-79" relativeweight="0.50" property="pvc" />
</floatings>
<properties>
    <propertyfile file="Floating_Materials.xml" path="materials" />
    <property name="userdef01" Restitution_Coefficient_User="0.70"
              comment="User redefinition for Restitution Coefficient (-)" />
    <links>
        <link mkbound="0" property="steel + userdef01" comment="Property for the tank"/>
    </links>
</properties>
```

When the interaction of solids (boundaries or floatings) is computed using **Discrete Element Method (DEM)** some extra properties with parameters used in DEM are loaded from **"Floating_Materials.xml":**

```
<materials>
    <property name="steel">
        <Young_Modulus value="210000000000.0" comment="Young Modulus (N/m2)" />
        <PoissonRatio value="0.30" comment="Poisson Ratio (-)" />
        <Restitution_Coefficient value="0.80" comment="Restitution Coefficient (-)" />
        <Kfric value="0.45" comment="Kinetic friction coefficient" />
    </property>
</materials>
```

2.7. XML file: Casedef-Properties

DEFINING OTHER VARIABLES

Using section ***properties***, users can define variables to be assigned to one or more ***mk***.

```
<properties>
    <links>
        <link mkfluid="0" property="material_1" />
        <link mkbound="3-6,1" property="material_2+data_x" />
    </links>
    <propertyfile file="run/ftdata_ext.xml" path="case.materials" />
    <property name="material_1" weight="1.35" other="pepe"/>
    <property name="material_2" begin="168" count="973">
        <massbody value="4728.78" />
        <center x="4.99" y="5" z="7.03" />
    </property>
    <property name="data_x" weight="1.35" />
</properties>
```

Each label of ***property*** has a name and can group several values
that can be text (***other***) or a number (***weight***)

```
<property name="material_1" weight="1.35" other="pepe"/>
```

or other subvalues (***massbody*** and ***center***)

```
<property name="material_2" begin="168" count="973">
    <massbody value="4728.78" />
    <center x="4.99" y="5" z="7.03" />
</property>
```

2.7. XML file: Casedef-Properties

These properties can be loaded from an external file using ***propertyfile.***
In this case, users have to indicate file name and path to access section with properties.

```
<propertyfile file="run/ftdata_ext.xml" path="case.materials" />
```

Example of file “ftdata_ext.xml”:

```
<case>
    <materials>
        <property name="uno" value="1.35"/>
        <property name="dos" value="168">
            <massbody value="4728.78" />
        </property>
    </materials>
</case>
```

Section ***links*** assigns one or more ***property*** to one or several values of ***mk***:

- Values of ***material_1*** are assigned to fluid particles with ***mk***=0

```
<link mkfluid="0" property="material_1"/>
```

- Values of ***material_2*** and ***data_x*** are assigned to boundary particles with ***mk***=1,3,4,5,6

```
<link mkbound="3-6,1" property="material_2+data_x"/>
```

It is also possible to indicate one ***property*** directly in the definition of the ***floatings***:

```
<floatings>
    ...
    <floating mkbound="4" property="Material_2">
        <massbody value="1300" />
        <inertia x="20" y="22" z="24" />
    </floating>
</floatings>
```

2.7. XML file: Casedef-Properties

GenCase reads the information from ***case.casedef.properties*** and writes in ***case.execution.particles.properties***.

```
<particles np="1494" nb="313" nbf="313" mkboundfirst="11" mkfluidfirst="1">
    <fixed mkbound="0" mk="11" begin="0" count="229" />
    <fixed mkbound="1" mk="12" begin="229" count="28" property="data_x+material_2" />
    <fixed mkbound="2" mk="13" begin="257" count="28" />
    <fixed mkbound="4" mk="15" begin="285" count="28" property="data_x+material_2" />
    <fluid mkfluid="0" mk="1" begin="313" count="1146" property="material_1" />
    <fluid mkfluid="1" mk="2" begin="1459" count="35" />
    <properties>
        <links>
            <link mk="1" property="material_1" />
            <link mk="12,15" property="data_x+material_2" />
        </links>
        <property name="material_1" weight="1.35" other="pepe"/>
        <property name="material_2" begin="168" count="973">
            <massbody value="4728.78" />
            <center x="4.99" y="5" z="7.03" />
        </property>
        <property name="data_x" weight="1.35" />
    </properties>
</particles>
```

Thus, ***DualSPHysics*** can access to assigned values to each ***mk***.
The object of type JSpaceParts is used to obtain the assigned properties to each block of particles

2.8. XML file: Casedef-Motion

SIMPLE RECTILINEAR

•*Motion01*: uniform rectilinear motion (<**mvrect** />) that also includes pauses (<**wait** />)

```
- <motion>
  - <objreal ref="1">
      <begin mov="1" start="0" finish="5.4"/>
    - <mvrect id="1" duration="0.6" next="2">
        <vel x="1" y="0" z="0"/>
      </mvrect>
      <wait id="2" duration="0.3" next="3"/>
    - <mvrect id="3" duration="0.6" next="4">
        <vel x="1" y="0" z="0"/>
      </mvrect>
      <wait id="4" duration="0.3" next="5"/>
    - <mvrect id="5" duration="0.6" next="6">
        <vel x="1" y="0" z="0"/>
      </mvrect>
      <wait id="6" duration="0.3" next="7"/>
    - <mvrect id="7" duration="-1" next="1">
        <vel x="-1.8" y="0" z="0"/>
      </mvrect>
    </objreal>
  </motion>
```

movement defined for the set of particles with *mk=1*

**first mov=1 during 0.6s,
then wait=2 for 0.3s,
then mov=3 during 0.6s,
then wait=4 for 0.3s,
then mov=5 during 0.6s…**

<mvrect>: uniform rectilinear movement

vel indicates the constant velocity vector

2.8. XML file: Casedef-Motion

SIMPLE RECTILINEAR

•*Motion01*: uniform rectilinear motion (<**mvrect** />) that also includes pauses (<**wait** />)

Time: 0.00 s

Time: 0.30 s

Time: 0.60 s

Time: 0.90 s

Time: 1.20 s

Time: 1.50 s

Time: 1.80 s

Time: 2.10 s

Time: 2.40 s

Time: 2.70 s

2.8. XML file: Casedef-Motion

COMBINED MOTIONS

•*Motion02*: combination of two uniform rectilinear motions (<**mvrect** />)

```
- <motion>
  - <objreal ref="1">
      <begin mov="1" start="0"/>
    - <mvrect id="1" duration="2" next="2">
        <vel x="1" y="0" z="0"/>
      </mvrect>
    - <mvrect id="2" duration="1" next="1">
        <vel x="-2" y="0" z="0"/>
      </mvrect>
      <begin mov="3" start="0.5"/>
    - <mvrect id="3" duration="1.3" next="4">
        <vel x="0" y="1" z="0"/>
      </mvrect>
    - <mvrect id="4" duration="1.3" next="3">
        <vel x="0" y="-1" z="0"/>
      </mvrect>
    </objreal>
  </motion>
```

<mvrect>: uniform rectilinear movement

vel indicates the constant velocity vector

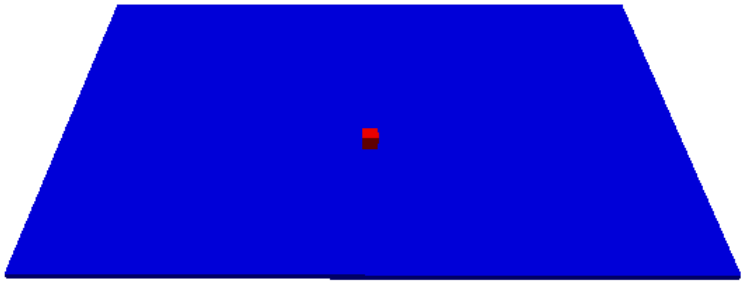
2.8. XML file: Casedef-Motion

COMBINED MOTIONS

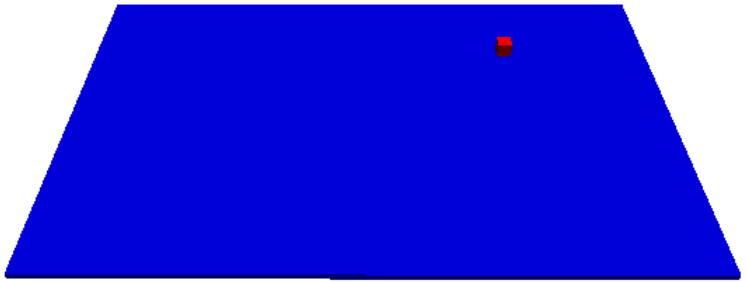
•*Motion02*: combination of two uniform rectilinear motions (<**mvrect** />)

Time: 0.00 s

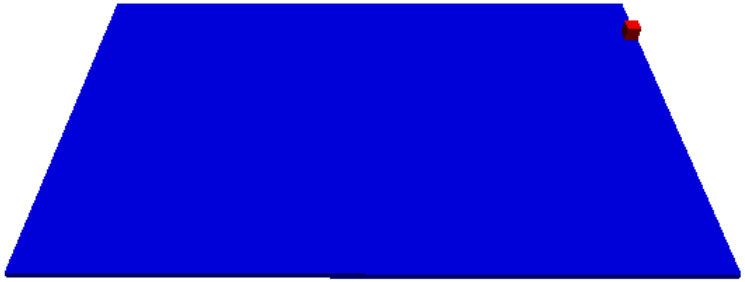
Time: 0.50 s



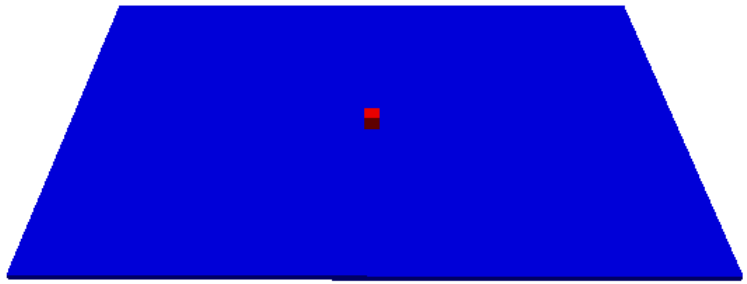
Time: 1.00 s



Time: 1.50 s



Time: 2.00 s



Time: 2.50 s

2.8. XML file: Casedef-Motion

HIERARCHICAL MOTIONS

•*Motion03*: movement of an object depending on the movement of another (hierarchy of objects)

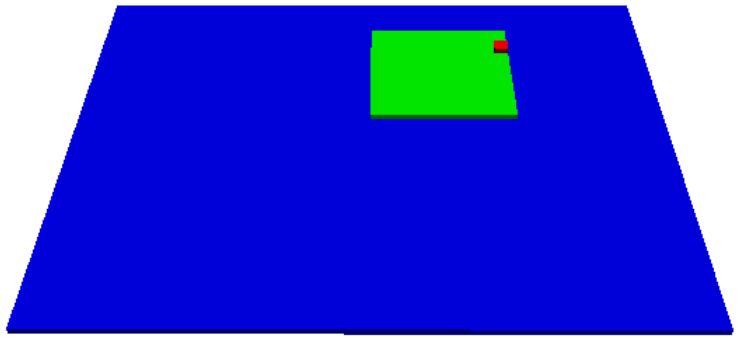
```
- <motion>
  - <objreal ref="1">
      <begin mov="1" start="0"/>
    - <mvrect id="1" duration="1.5" next="2">
        <vel x="1" y="0" z="0"/>
      </mvrect>
    - <mvrect id="2" duration="1.5" next="1">
        <vel x="-1" y="0" z="0"/>
      </mvrect>
      <begin mov="3" start="0.1"/>
    - <mvrect id="3" duration="1.1" next="4">
        <vel x="0" y="1" z="0"/>
      </mvrect>
    - <mvrect id="4" duration="1.1" next="3">
        <vel x="0" y="-1" z="0"/>
      </mvrect>
    - <objreal ref="2">
        <begin mov="1" start="0.2"/>
      - <mvrect id="1" duration="0.45" next="2">
          <vel x="1" y="0" z="0"/>
        </mvrect>
      - <mvrect id="2" duration="0.45" next="3">
          <vel x="0" y="1" z="0"/>
        </mvrect>
      - <mvrect id="3" duration="0.45" next="4">
          <vel x="-1" y="0" z="0"/>
        </mvrect>
      - <mvrect id="4" duration="0.45" next="1">
          <vel x="0" y="-1" z="0"/>
        </mvrect>
      </objreal>
    </objreal>
  </motion>
```

movement defined for the set of particles with *mk=2* that also moves according to the movement defined for *mk=1*

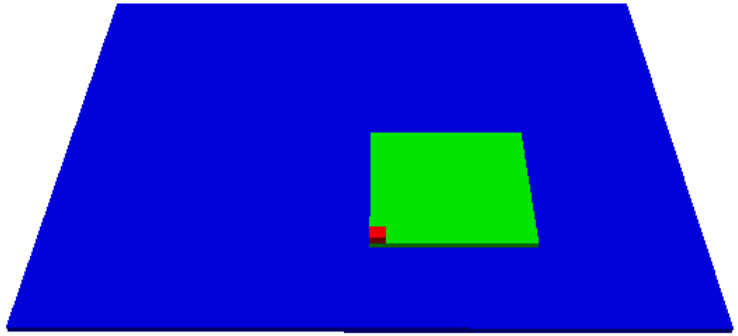
2.8. XML file: Casedef-Motion

•*Motion03*: movement of an object depending on the movement of another (hierarchy of objects)

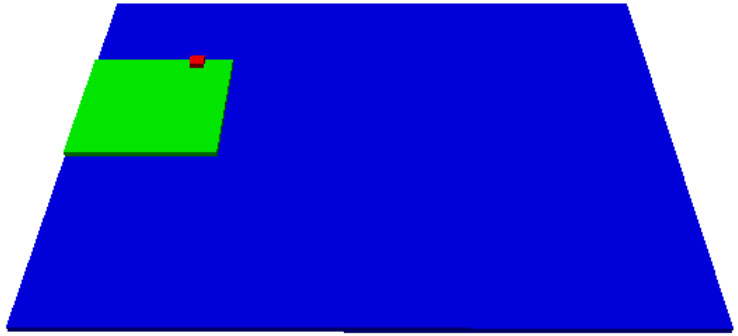
Time: 0.00 s



Time: 1.00 s



Time: 2.00 s



Time: 3.00 s

2.8. XML file: Casedef-Motion

ACELERATED RECTILINEAR

•*Motion04*: accelerated rectilinear motion (<**mvrectace** />)

```
- <motion>
  - <objreal ref="1">
      <begin mov="1" start="0"/>
    - <mvrectace id="1" duration="1.411" next="2">
        <velini x="0" y="0" z="0"/>
        <ace x="2" y="0" z="0"/>
      </mvrectace>
    - <mvrectace id="2" duration="1">
        <velini x="-2" y="5" z="0"/>
        <ace x="0" y="-10" z="0"/>
      </mvrectace>
    </objreal>
  </motion>
```

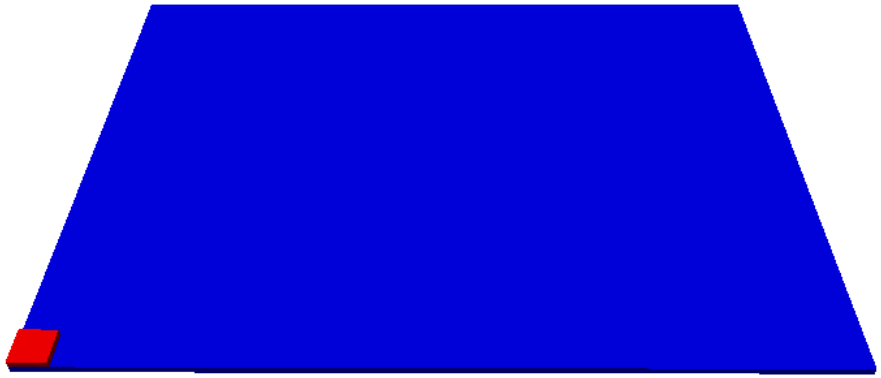
<mvrectace>: accelerated rectilinear movement

velini indicates the initial velocity vector
ace indicates the acceleration vector

2.8. XML file: Casedef-Motion

ACELERATED RECTILINEAR

•*Motion04*: accelerated rectilinear motion (<**mvrectace** />)



Time: 0.00 s

Time: 0.75 s

Time: 1.50 s

Time: 2.25 s

2.8. XML file: Casedef-Motion

ROTATIONAL

•*Motion05*: rotational motion (<**mvrot** />)

```
- <motion>
  - <objreal ref="3">
      <begin mov="1" start="0"/>
    - <mvrot id="1" duration="1000">
        <vel ang="20"/>
        <axisp1 x="0.5" y="0.5" z="0"/>
        <axisp2 x="0.5" y="0.5" z="1"/>
      </mvrot>
    </objreal>
  - <objreal ref="4">
      <begin mov="1" start="0"/>
    - <mvrot id="1" duration="1000">
        <vel ang="240"/>
        <axisp1 x="0.5" y="0.5" z="0"/>
        <axisp2 x="0.5" y="0.5" z="1"/>
      </mvrot>
    </objreal>
  </motion>
```

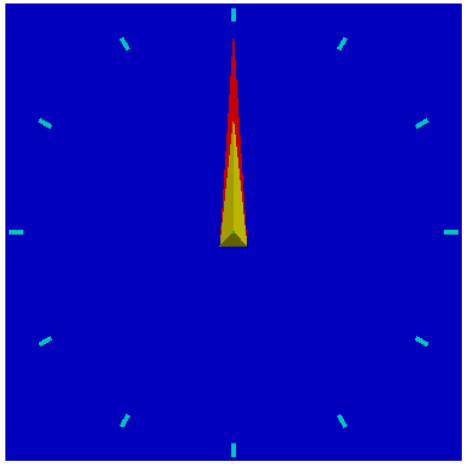
<mvrot>: rotational movement

vel indicates the angular velocity
axisp1 first point of the rotation axis
axisp2 second point of the rotation axis

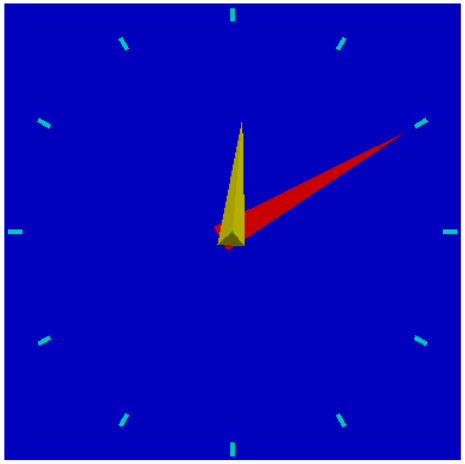
2.8. XML file: Casedef-Motion

ROTATIONAL

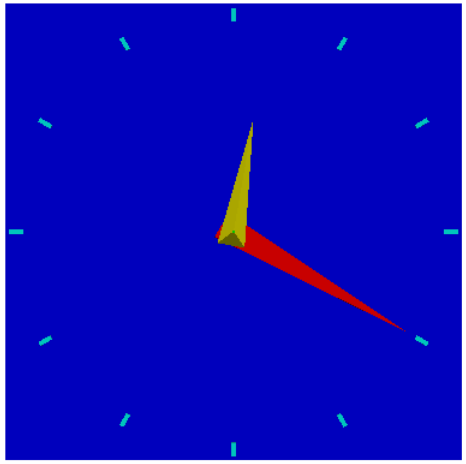
•*Motion05*: rotational motion (<**mvrot** />)



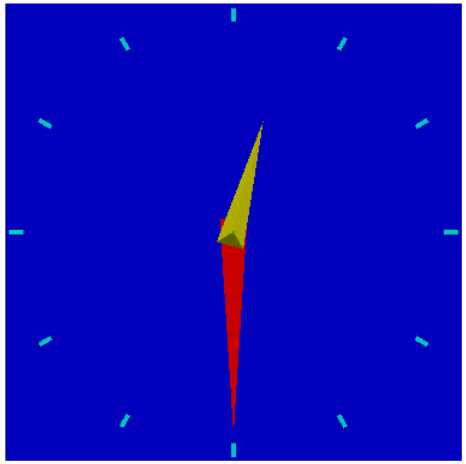
Time: 0.00 s



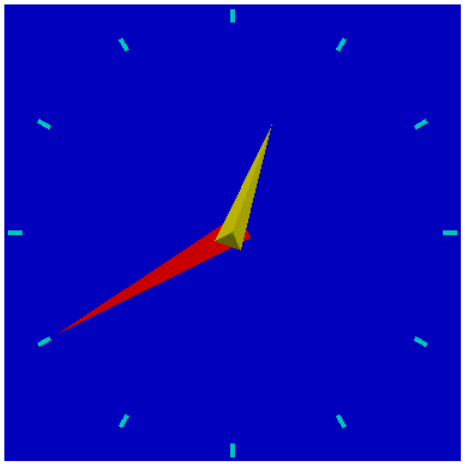
Time: 0.25 s



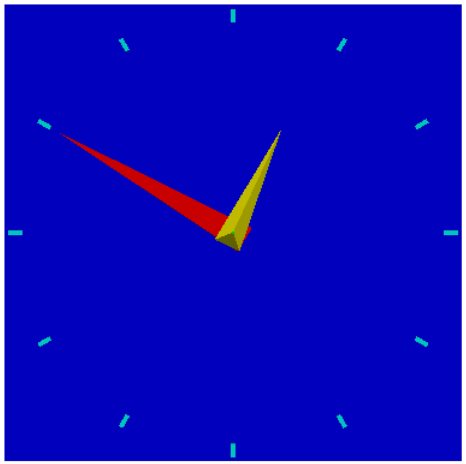
Time: 0.50 s



Time: 0.75 s



Time: 1.00 s



Time: 1.25 s

2.8. XML file: Casedef-Motion

ROTATIONAL & CIRCULAR

•*Motion06*: accelerated rotation motion (<**mvrotace** />) and accelerated circular motion (<**mvcirace** />).

```
- <motion>
  - <obj>
      <objreal ref="1"/>
      <objreal ref="3"/>
      <objreal ref="4"/>
      <begin mov="1" start="0"/>
    - <mvrotace id="1" duration="1000">
        <ace ang="9"/>
        <velini ang="-50"/>
        <axisp1 x="0" y="0" z="1.85"/>
        <axisp2 x="0" y="1" z="1.85"/>
      </mvrotace>
    </obj>
  - <objreal ref="5">
      <begin mov="1" start="0"/>
    - <mvcirace id="1" duration="1000">
        <ace ang="9"/>
        <velini ang="-50"/>
        <ref x="1.3" y="-0.7" z="1.85"/>
        <axisp1 x="0" y="0" z="1.85"/>
        <axisp2 x="0" y="1" z="1.85"/>
      </mvcirace>
    </objreal>
</motion>
```

<mvrotace>: accelerated rotational movement

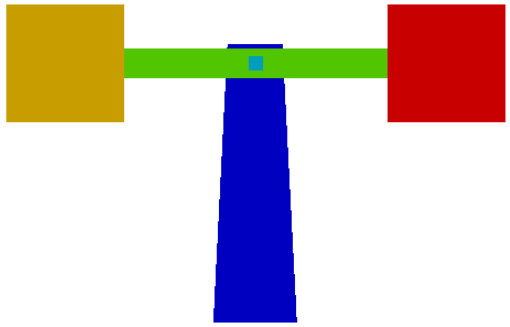
ace indicates the angular acceleration
velini indicates the initial angular velocity
axisp1 first point of the rotation axis
axisp2 second point of the rotation axis

<mvcirace>: accelerated circular movement

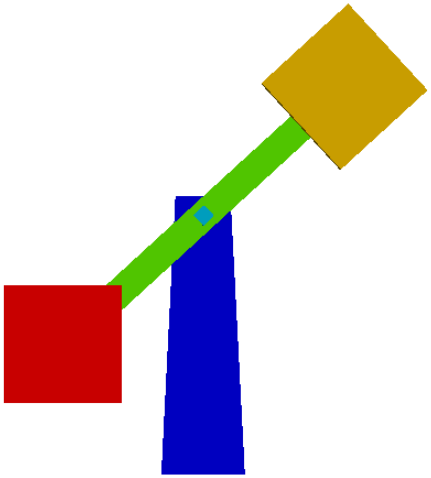
ace indicates the angular acceleration
ref indicates the point of the object that rotates with the axis
velini indicates the initial angular velocity
axisp1 first point of the rotation axis
axisp2 second point of the rotation axis

2.8. XML file: Casedef-Motion

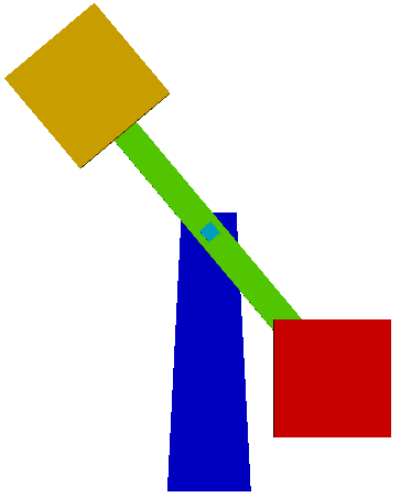
•*Motion06*: accelerated rotation motion (<**mvrotace** />) and accelerated circular motion (<**mvcirace** />).



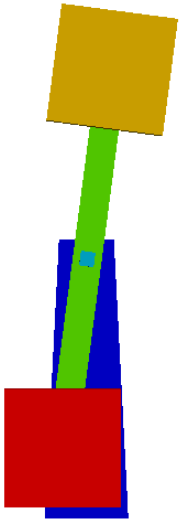
Time: 0.00 s



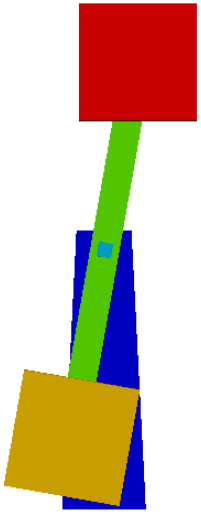
Time: 5.00 s



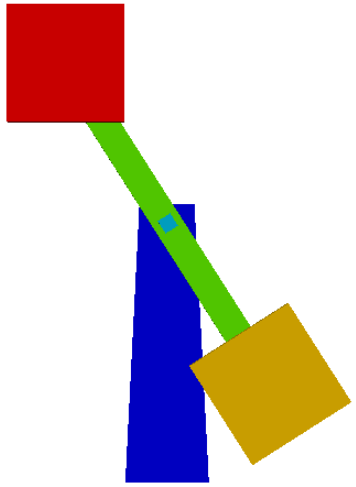
Time: 10.00 s



Time: 15.00 s



Time: 20.00 s



Time: 25.00 s

2.8. XML file: Casedef-Motion

SINUSOIDAL

•*Motion07*: sinusoidal movement (**<mvrectsinu />, <mvrotsinu />, <mvcirsinu />**)

```
- <motion>
  - <objreal ref="4">
      <begin mov="1" start="0"/>
    - <mvrotsinu id="1" duration="5" next="2">
        <axisp1 x="0" y="0" z="2.85"/>
        <axisp2 x="0" y="1" z="2.85"/>
        <freq v="0.2"/>
        <ampl v="60"/>
        <_phase v="0"/>
      </mvrotsinu>
    - <mvrotsinu id="2" duration="5" next="1">
        <axisp1 x="0" y="0" z="2.85"/>
        <axisp2 x="0" y="1" z="2.85"/>
        <freq v="0.4"/>
        <ampl v="75"/>
      </mvrotsinu>
    </objreal>
  - <objreal ref="5">
      <begin mov="1" start="0"/>
    - <mvcirsinu id="1" duration="5" next="2"
        <ref x="0" y="-0.7" z="0.2"/>
        <axisp1 x="0" y="0" z="2.85"/>
        <axisp2 x="0" y="1" z="2.85"/>
        <freq v="0.2"/>
        <ampl v="60"/>
        <phase v="0"/>
      </mvcirsinu>
    - <mvcirsinu id="2" duration="5" next="1">
        <ref x="0" y="-0.7" z="0.2"/>
        <axisp1 x="0" y="0" z="2.85"/>
        <axisp2 x="0" y="1" z="2.85"/>
        <freq v="0.4"/>
        <ampl v="75"/>
        <phase v="0"/>
      </mvcirsinu>
    </objreal>
  - <objreal ref="6">
      <begin mov="1" start="0"/>
    - <mvrectsinu id="1" duration="5" next="2">
        <freq x="0.2" y="0" z="0"/>
        <ampl x="2.30" y="0" z="0"/>
        <phase x="0" y="0" z="0"/>
      </mvrectsinu>
    - <mvrectsinu id="2" duration="5" next="1">
        <freq x="0.4" y="0" z="0"/>
        <ampl x="2.55" y="0" z="0"/>
        <phase x="0" y="0" z="0"/>
      </mvrectsinu>
    </objreal>
</motion>
```

<mvrectsinu>: sinusoidal rectilinear movement

<mvrotsinu>: sinusoidal rotational movement

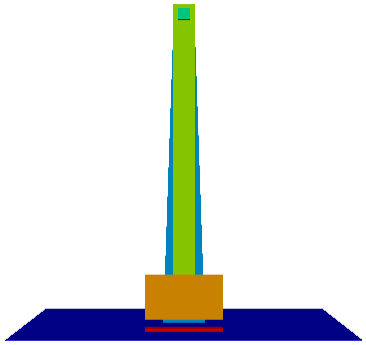
<mvcirsinu>: sinusoidal circular movement

axisp1 first point of the rotation axis
axisp2 second point of the axis
freq frequency
ampl amplitude
phase phase

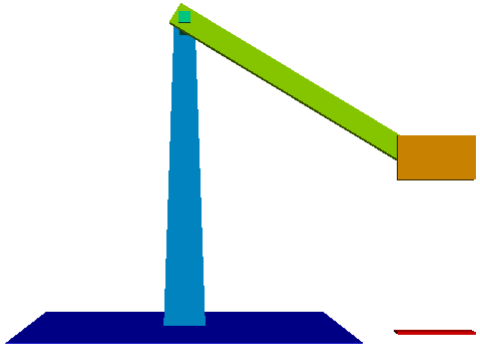
2.8. XML file: Casedef-Motion

SINUSOIDAL

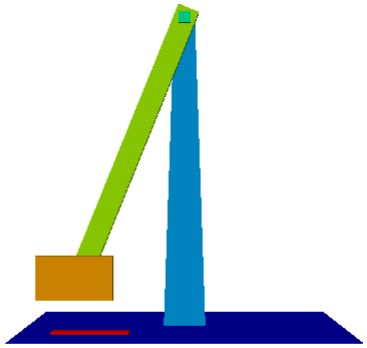
•*Motion07*: sinusoidal movement (**<mvrectsinu />, <mvrotsinu />, <mvcirsinu />**)



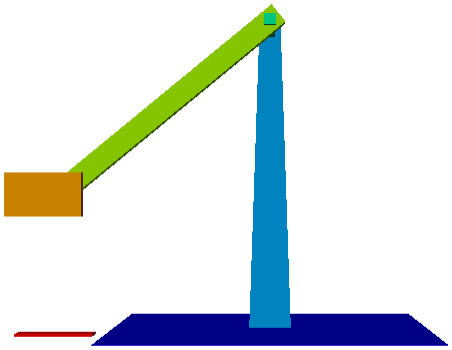
Time: 0.00 s



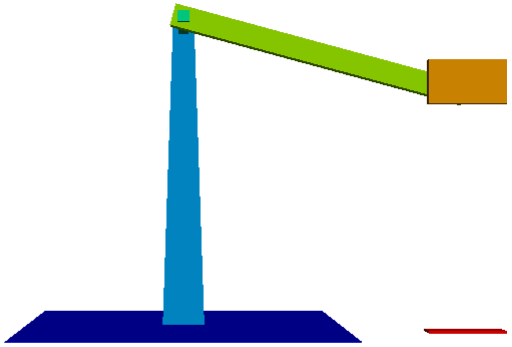
Time: 1.40 s



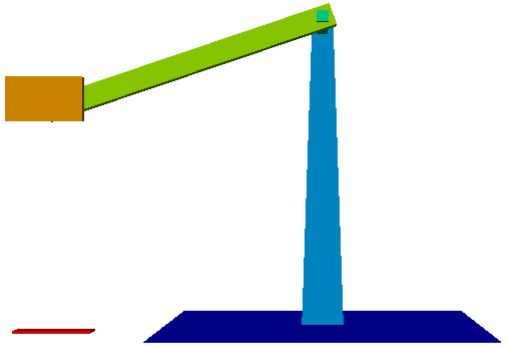
Time: 2.80 s



Time: 4.20 s



Time: 5.60 s



Time: 7.00 s

2.8. XML file: Casedef-Motion

EXTERNAL MOTION: RECTILINEAR

•*Motion08*: predefined movement with data from an external file (**<mvpredef /> or <mvfile />**)

```
- <motion>
  - <objreal ref="200">
      <begin mov="1" start="0"/>
    - <mvpredef id="1" duration="10">
        <file name="motion08mov_f3.out" fields="4" fieldtime="0" fieldx="1" fieldy="2" fieldz="3"/>
      </mvpredef>
    </objreal>
  - <objreal ref="150">
      <begin mov="1" start="0"/>
    - <mvpredef id="1" duration="8" next="2">
        <file name="motion08mov_f3.out" fields="4" fieldtime="0" fieldx="1" fieldy="2"/>
      </mvpredef>
    - <mvrect id="2" duration="-1">
        <vel x="0" y="0" z="-0.02"/>
      </mvrect>
    </objreal>
  - <objreal ref="151">
      <begin mov="1" start="0"/>
    - <mvpredef id="1" duration="10">
        <file name="motion08mov_f3.out" fields="4" fieldtime="0" fieldx="1" fieldz="3"/>
      </mvpredef>
    </objreal>
  - <objreal ref="152">
      <begin mov="1" start="0"/>
    - <mvpredef id="1" duration="10">
        <file name="motion08mov_f3.out" fields="4" fieldtime="0" fieldy="2" fieldz="3"/>
      </mvpredef>
    </objreal>
</motion>
```

<mvpredef /> or <mvfile />:
prescribed motion loaded from a file

name name of the file
fields number of columns of the file
fieldtime column with time
fieldx column with X-position
fieldy column with Y-position
filedz column with Z-position

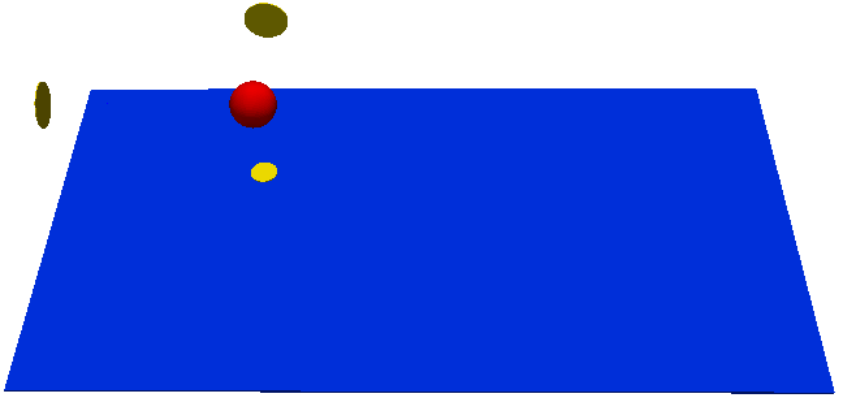
first field (or column) has reference "0"
second field (or column) has reference "1"

2.8. XML file: Casedef-Motion

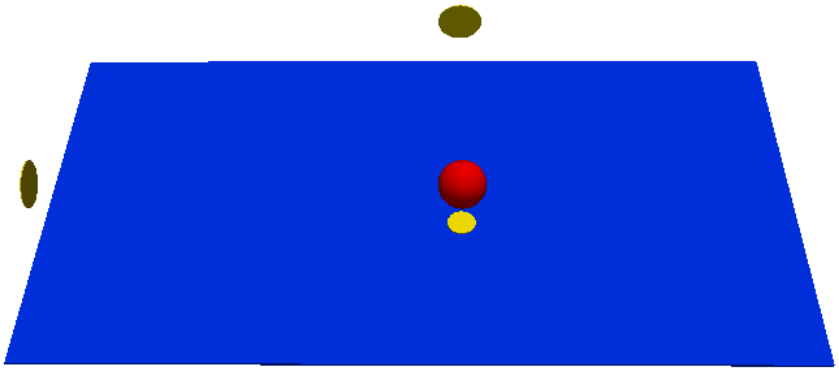
EXTERNAL MOTION: RECTILINEAR

•*Motion08*: predefined movement with data from an external file (**<mvpredef /> or <mvfile />**)

Time: 0.00 s



Time: 3.00 s



Time: 6.00 s

Time: 9.00 s

2.8. XML file: Casedef-Motion

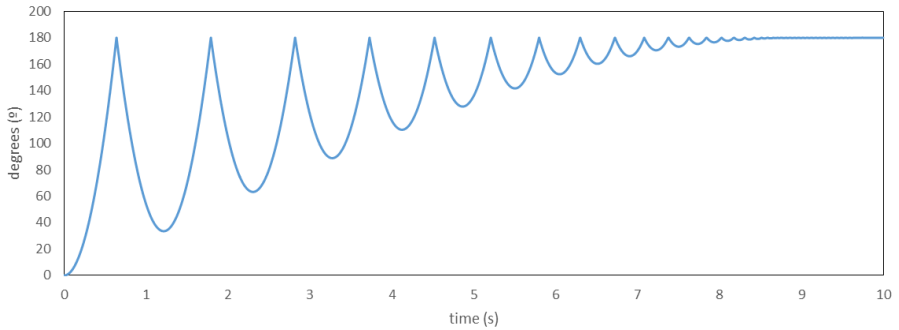
EXTERNAL MOTION: ROTATIONAL

•*Motion09*: predefined movement with data from an external file (<**mvrotfile** />)

<**mvrotfile** />**:** prescribed motion loaded from a file *with degrees*

name name of the file
axisp1 & axisp2 two points to define the axis of rotation

```
<motion>
    <objreal ref="1">
        <begin mov="1" start="0" finish="100" />
        <mvrotfile id="1" duration="9" next="2" anglesunits="degrees">
            <file name="Motion09mov_deg.csv" />
            <axisp1 x="1" y="1" z="0.03" />
            <axisp2 x="1" y="-1" z="0.03" />
        </mvrotfile>
        <mvrotfile id="2" duration="9" anglesunits="radians">
            <file name="Motion09mov_rad.csv" />
            <axisp1 x="1" y="-1" z="0.03" />
            <axisp2 x="1" y="1" z="0.03" />
        </mvrotfile>
    </objreal>
</motion>
```

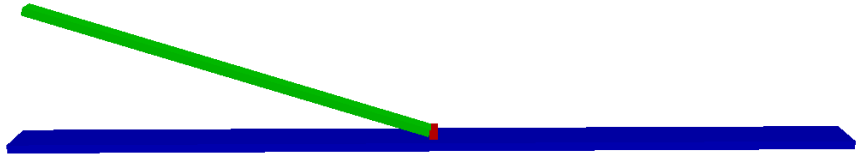


2.8. XML file: Casedef-Motion

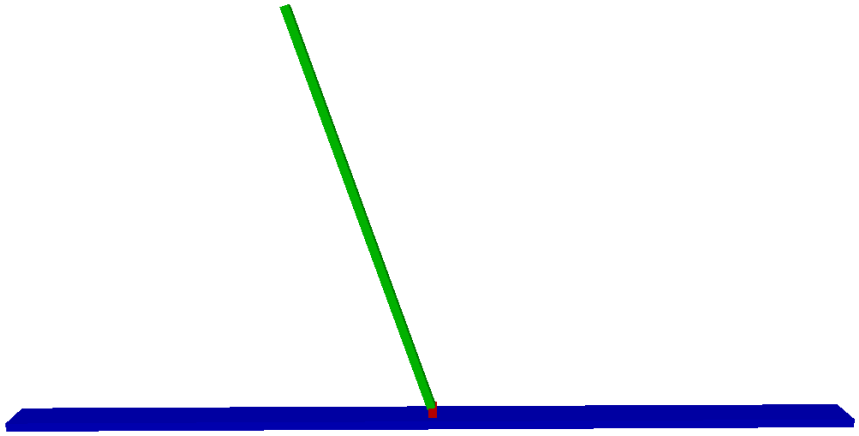
EXTERNAL MOTION: ROTATIONAL

•*Motion09*: predefined movement with data from an external file (<**mvrotfile** />)

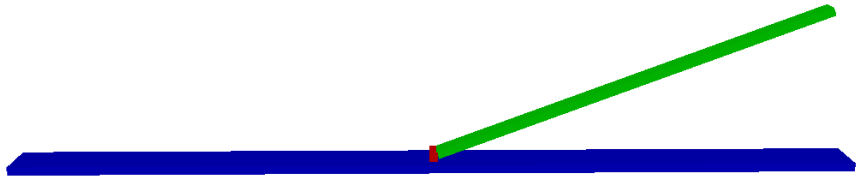
Time: 0 s



Time: 0.2 s



Time: 0.4 s



Time: 0.6 s

3.1. XML file: Execution-Special

The section <special> defines the configuration of different features used in DualSPHysics simulation.

```
<special>
    <accinputs>       <!-- *** Configures force to be applied to different blocks of particles. *** -->
    <boundcorr>       <!-- *** Configures boundary extrapolated correction configuration (used in combination with InOut). *** -->
    <chrono>          <!-- *** Defines geometry of bodies and links between bodies using Chrono (used in combination with
    <parameter key="RigidAlgorithm"/>). *** -->
    <damping>         <!-- *** Configures diferent damping zones. *** -->
    <gauges>          <!-- *** Configures automatic measurements during the simulation for use in the simulation itself. *** -->
        <velocity>              <!-- Calculates velocity in one point and changes default configuration parameters. -->
        <swl>                   <!-- Calculates Surface Water Limit in a line of points. -->
        <maxz>                  <!-- Calculates maximum z of fluid at distance of a vertical line. -->
        <force>                 <!-- Calculates force sumation on selected fixed or moving particles (using only fluid particles). -->
    </gauges>
    <initialize>      <!-- *** Configures initialization modes for particle data before starting the simulation (created  framwork
    to be modified by the user). *** -->
    <inout>           <!-- *** Configures inlet and outlet boundary conditions. *** -->
    <mlayerpistons>   <!-- *** Configures Multi-Layer Pistons using external velocity data (e.g. from SWASH) to create waves 1D or
    2D *** -->
        <piston1d>
        <piston2d>
    </mlayerpistons>
    <relaxationzones><!-- *** Configures Relaxation Zones (RZ) to manage the behaviour of the fluid. *** -->
        <rzwaves_regular>     <!-- REGULAR wave generation and absorption using Relaxation Zone (RZ). -->
        <rzwaves_spectrum>    <!-- IRREGULAR wave generation and absorption using Relaxation Zone (RZ). -->
        <rzwaves_external_1d> <!-- EXTERNAL wave generation and absorption using Relaxation Zone (RZ). -->
        <rzwaves_uniform>     <!-- Uniform velocity using Relaxation Zone (RZ). -->
    </relaxationzones>
    <savedt>          <!-- *** Special configuration to save dt values used in the simulation. *** -->
    <timeout>         <!-- *** Configures variable timing to save simulation results (particle data) . *** -->
    <wavepaddles>     <!-- *** Configures automatic wave generation using PISTON or FLAP. *** -->
        <piston>              <!-- PISTON to create REGULAR waves *** -->
            <awas_zsurf>      <!-- AWAS (only for Piston) with regular and irregular waves. -->
        </piston>
        <flap>                <!-- FLAP to create REGULAR waves. -->
        <piston_spectrum>     <!-- PISTON to create IRREGULAR waves. -->
        <flap_spectrum>       <!-- FLAP to create IRREGULAR waves. -->
        <piston_solitary>     <!-- PISTON to create SOLITARY waves. -->
    </wavepaddles>
</special>
```

3.1.1. XML file: Special-WavePaddles

Automatic wave generation for Regular & Irregular waves using a Piston or Flap and Solitary waves using a Piston.

Piston or Flap is made of moving boundary particles with a MkBound value. These particles are moved according wave parameters.

```
<casedef>
    ...
    <motion>
        <objreal ref="27">     <!-- Defines MkBound=26 as moving particles (piston or flap). -->
            <begin mov="1" start="0" />
            <mvnull id="1" />
        </objreal>
    </motion>
</casedef>
<execution>
    <special>
        <!-- *** Configures automatic wave generation using PISTON or FLAP. *** -->
        <wavepaddles>
            <piston>           <!-- PISTON to create REGULAR waves *** -->
                <mkbound value="27" comment="Mk-Bound of selected particles to simulate the piston" />
                <awas_zsurf>  <!-- AWAS (only for Piston) with regular and irregular waves. -->
            </piston>
            <piston_spectrum> <!-- PISTON to create IRREGULAR waves. -->
                <awas_zsurf>
            </piston_spectrum>
            <flap>             <!-- FLAP to create REGULAR waves. -->
            <flap_spectrum>    <!-- FLAP to create IRREGULAR waves. -->
            <piston_solitary> <!-- PISTON to create SOLITARY waves. -->
        </wavepaddles>
</special>
```

3.1.1. XML file: Special-WavePaddles

GENERATION OF **REGULAR** WAVES: **PISTON**


```
<wavepaddles>
    <piston>
        <mkbound value="10" comment="Mk-Bound of selected particles" />
        <start value="0" comment="Start time (default=0)" />
        <duration value="0" comment="Movement duration, Zero is the end of simulation (default=0)" />
        <depth value="30" comment="Water depth (default=0)" />
        <pistondir x="1" y="0" z="0" comment="Movement direction (default=(1,0,0))" />
        <waveorder value="1" comment="Order wave generation 1:1st order, 2:2nd order (default=1)" />
        <waveheight value="1.5" comment="Wave height" />
        <waveperiod value="9.5" comment="Wave period" />
        <gainstroke value="1" comment="Gain factor to amplify/reduce the paddle stroke (default=1)" />
        <phase value="0" comment="Initial wave phase in function of PI (default=0)" />
        <ramp value="0" comment="Periods of ramp (default=0)" />
        <_usefiledata file="PistonFile_WallRegLong.csv" swlmod="-0.266916" comment="Uses motion and
        elevation from file" />
        <savemotion periods="24" periodsteps="20" xpos="2" zpos="-0.5" comment="Saves motion data. xpos and
        zpos are optional. zpos=-depth of the measuring point" />
    </piston>
</wavepaddles>
```

***waveorder*:** order of wave generation,1st order or 2nd order (i.e. superharmonics)

***depth*:** depth at the piston location

***waveheight*:** wave height H

***waveperiod*:** wave period T

***gainstroke*:** factor to scale the amplitude of the piston displacement time series, that has been previously calculated based on H,T and depth.

***ramp*:** number of periods to smooth the movement of the piston

***savemotion*:** saves theoretical results of elevation and orbital velocities at *xpos* and *zpos*

(being *zpos=-depth* of the measuring point)

3.1.1. XML file: Special-WavePaddles

***waveorder*:** order of wave generation,1st order or 2nd order (i.e. superharmonics)

***depth*:** depth at the piston location

***waveheight*:** wave height H

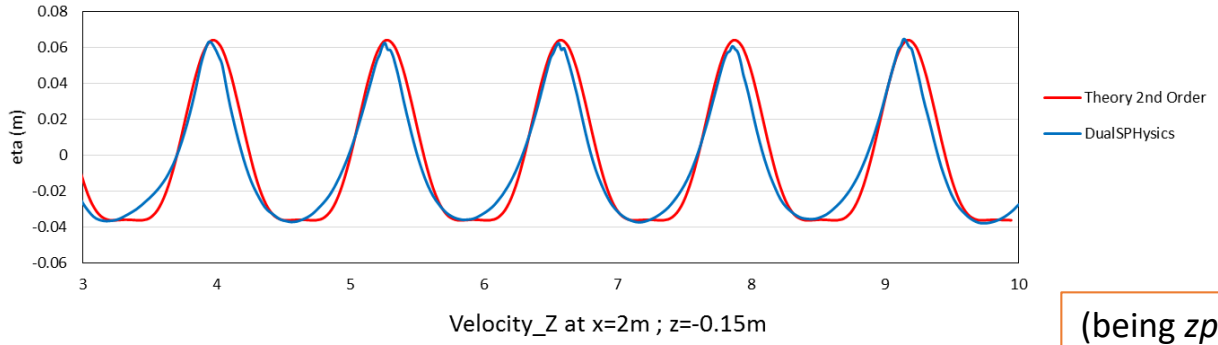
***waveperiod*:** wave period T

***gainstroke*:** factor to scale the amplitude of the piston displacement time series, that has been previously calculated based on H,T and depth.

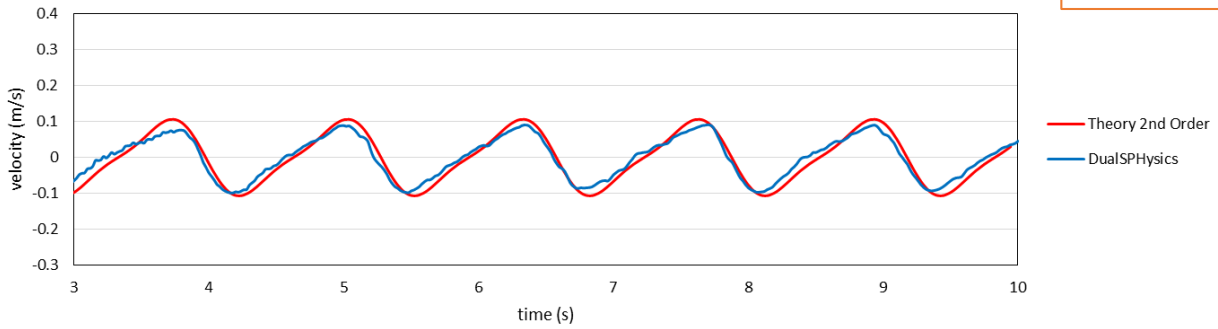
***ramp*:** number of periods to smooth the movement of the piston

***savemotion*:** saves theoretical results of elevation and orbital velocities at *xpos* and *zpos*

```
<savemotion periods="24" periodsteps="20" xpos="2" zpos="-0.15" />
```



Velocity_Z at x=2m ; z=-0.15m



(being *zpos=-depth* of the measuring point)

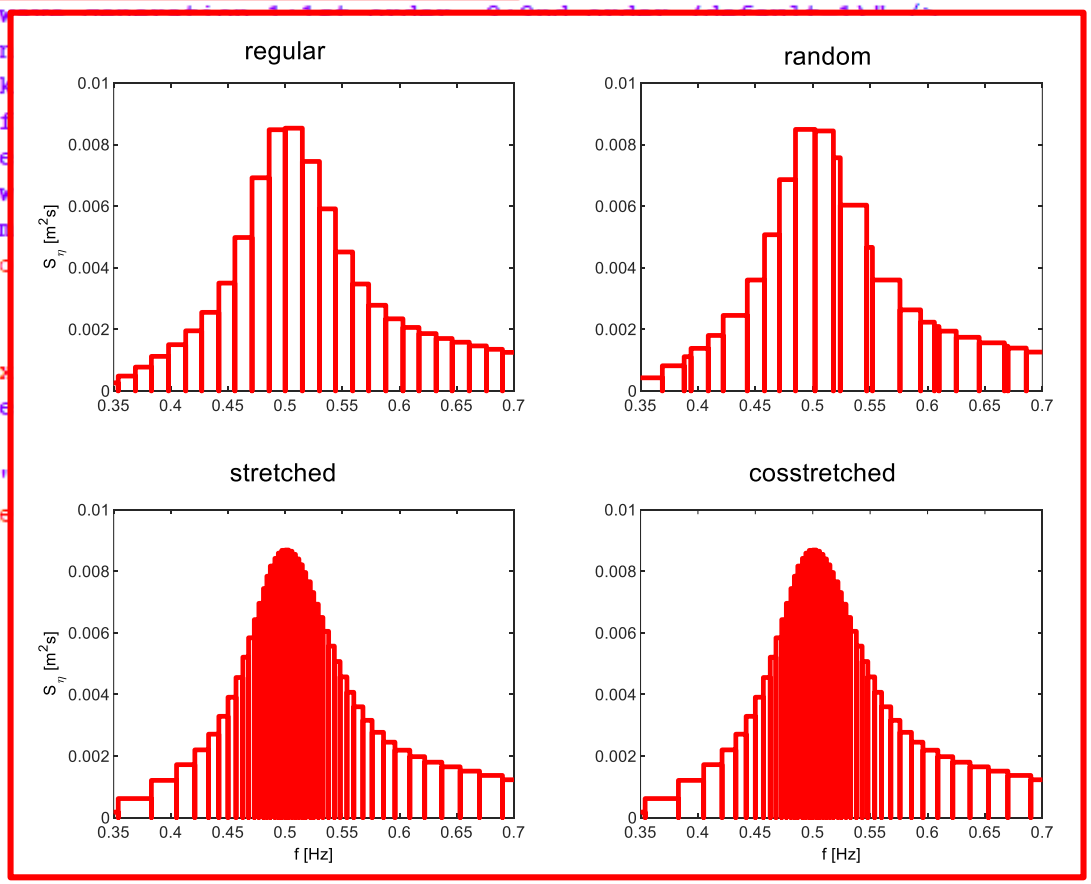
3.1.1. XML file: Special-WavePaddles


```
<wavepaddles>
    <piston_spectrum>
        <mkbound value="12" comment="Mk-Bound of selected particles" />
        <depth value="50" comment="Water depth (default=0)" />
        <start value="0" comment="Start time (default=0)" />
        <duration value="0" comment="Movement duration, Zero is the end of simulation (default=0)" />
        <pistondir x="1" y="0" z="0" comment="Movement direction (default=(1,0,0))" />
        <spectrum value="jonswap" comment="Spectrum type: jonswap,pierson-moskowitz" />
        <discretization value="stretched" comment="Spectrum discretization:
        regular,random,stretched,cosstretched (default=stretched)" />
        <waveorder value="1" comment="Order wave generation 1:1st order, 2:2nd order (default=1)" />
        <waveheight value="1.5" comment="Significant Wave Height (Hs)" />
        <waveperiod value="9.5" comment="Peak Period" />
        <gainstroke value="1" comment="Gain factor to amplify/reduce the paddle stroke (default=1)" />
        <peakcoef value="3.3" comment="Peak enhancement coefficient (default=3.3)" />
        <waves value="3" comment="Number of waves to create irregular waves (default=50)" />
        <randomseed value="2" comment="Random seed to initialize a pseudorandom number generator" />
        <serieini value="0" autofit="true" comment="Initial time in irregular wave serie (default=0 and
        autofit=false)" />
        <ramptime value="0" comment="Time of ramp (default=0)" />
        <savemotion time="10" timedt="0.02" xpos="0.1" zpos="-0.5" comment="Saves motion data. xpos and
        zpos are optional. zpos=-depth of the measuring point" />
        <saveserie timemin="0" timemax="300" timedt="0.05" xpos="0" comment="Saves serie data (optional)" />
        <saveseriewaves timemin="0" timemax="1000" xpos="0" comment="Saves serie heights" />
        <calcserielength timemax="1000" comment="Calculates serie length (optional)" />
    </piston_spectrum>
</wavepaddles>
```

3.1.1. XML file: Special-WavePaddles

GENERATION OF **IRREGULAR** WAVES: **PISTON**


```
<wavepaddles>
    <piston_spectrum>
        <mkbound value="12" comment="Mk-Bound of selected particles" />
        <depth value="50" comment="Water depth (default=0)" />
        <start value="0" comment="Start time (default=0)" />
        <duration value="0" comment="Movement duration, Zero is the end of simulation (default=0)" />
        <pistondir x="1" y="0" z="0" comment="Movement direction (default=(1,0,0))" />
        <spectrum value="jonswap" comment="Spectrum type: jonswap,pierson-moskowitz" />
        <discretization value="stretched" comment="Spectrum discretization:
        regular,random,stretched,cosstretched (default=stretched)" />
        <waveorder value="1" comment="Order w
        <waveheight value="1.5" comment="Sign
        <waveperiod value="9.5" comment="Peak
        <gainstroke value="1" comment="Gain f
        <peakcoef value="3.3" comment="Peak e
        <waves value="3" comment="Number of w
        <randomseed value="2" comment="Random
        <serieini value="0" autofit="true" co
        autofit=false)" />
        <ramptime value="0" comment="Time of
        <savemotion time="10" timedt="0.02" x
        zpos are optional. zpos=-depth of the
        <saveserie timemin="0" timemax="300"
        <saveseriewaves timemin="0" timemax="
        <calcserielength timemax="1000" comme
    </piston_spectrum>
</wavepaddles>
```



3.1.1. XML file: Special-WavePaddles

GENERATION OF **IRREGULAR** WAVES: **PISTON**

***waveorder*:** order of wave generation,1st order or 2nd order (i.e. subharmonics or bound long waves)

***spectrum*:** type of spectrum (Jonswap or Pierson-Moskowitz)

***waveheight*:** significant wave height H_s

***waveperiod*:** peak wave period T_p

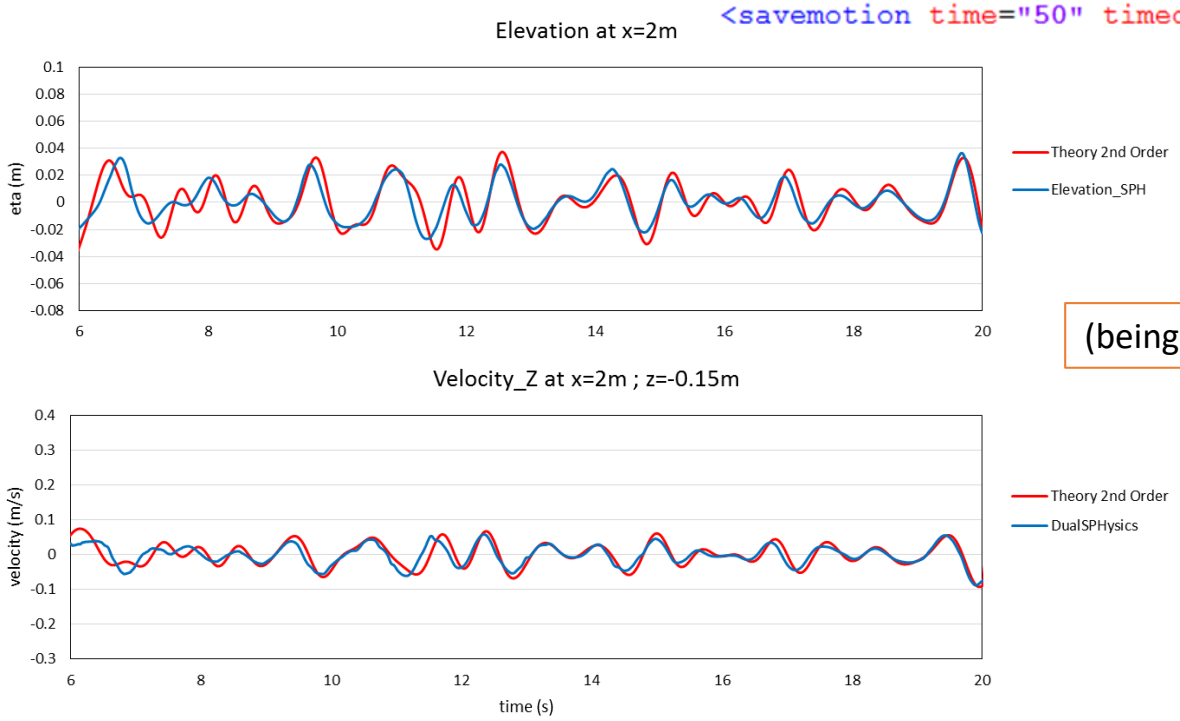
***waves*:** number of wave components to discretize the wave spectrum

***serieini*:** initial series of the irregular train is chosen from "WavePaddle_mkb0010_Serie.csv"

***ramptime*:** time to slowly start a smoothed movement of the piston

***savemotion*:** saves theoretical results of elevation and orbital velocities at *xpos* and *zpos*

```
<savemotion time="50" timedt="0.05" xpos="2/> pos="-0.15"
```



(being *zpos=-depth* of the measuring point)

3.1.1. XML file: Special-WavePaddles

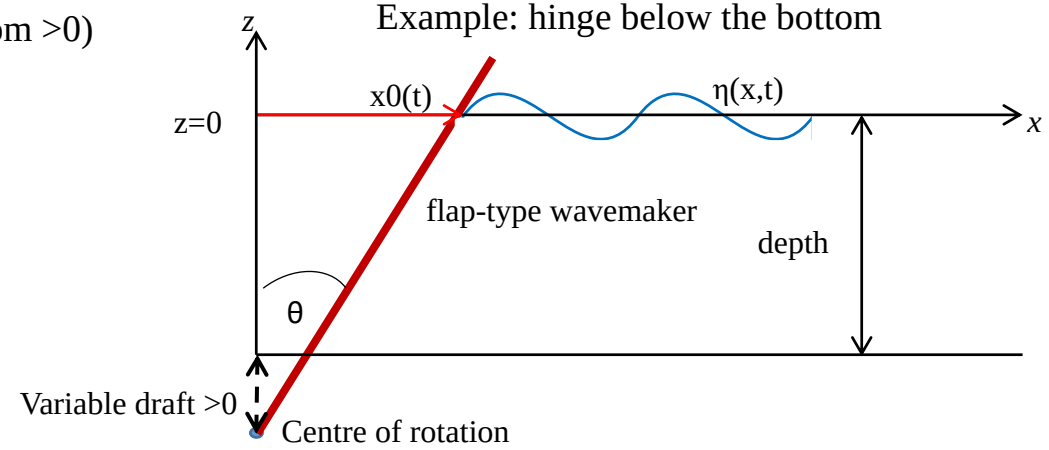

```
<wavepaddles>
    <flap>
        <mkbound value="11" comment="Mk-Bound of selected particles" />
        <start value="2" comment="Start time (default=0)" />
        <duration value="0" comment="Movement duration, Zero is the end of simulation (default=0)" />
        <depth value="1.4" comment="Water depth (default=0)" />
        <variabledraft value="0" comment="Position of the wavemaker hinge (above the bottom <0; below the
        bottom >0) (default=0)" />
        <flapaxis0 x="3" y="0" z="0" comment="Point 0 of axis rotation" />
        <flapaxis1 x="3" y="1" z="0" comment="Point 1 of axis rotation" />
        <waveheight value="0.1" comment="Wave height" />
        <waveperiod value="5" comment="Wave period" />
        <gainstroke value="1" comment="Gain factor to amplify/reduce the paddle stroke (default=1)" />
        <phase value="1" comment="Initial wave phase in function of PI (default=0)" />
        <ramp value="3" comment="Periods of ramp (default=0)" />
        <savemotion periods="24" periodsteps="20" xpos="2" zpos="-0.5" comment="Saves motion data. xpos
        and zpos are optional. zpos=-depth of the measuring point" />
    </flap>
</wavepaddles>
```

***variabledraft*:** position of the flap hinge with respect to the bottom (above the bottom <0; below the bottom >0) (default=0)

***flapaxis0*:** initial point defining the rotation axis

***flapaxis1*:** final point defining the rotation axis

Example: hinge below the bottom



3.1.1. XML file: Special-WavePaddles

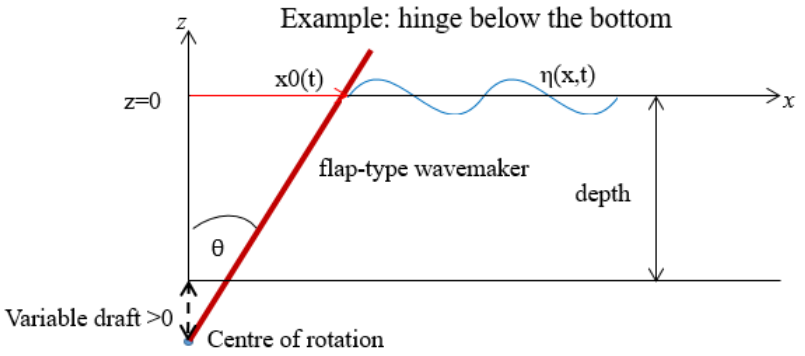
GENERATION OF **IRREGULAR** WAVES: **FLAP**


```
<wavepaddles>
    <piston_spectrum>
        <mkbound value="12" comment="Mk-Bound of selected particles" />
        <depth value="50" comment="Water depth (default=0)" />
        <start value="0" comment="Start time (default=0)" />
        <duration value="0" comment="Movement duration, Zero is the end of simulation (default=0)" />
        <pistondir x="1" y="0" z="0" comment="Movement direction (default=(1,0,0))" />
        <spectrum value="jonswap" comment="Spectrum type: jonswap,pierson-moskowitz" />
        <discretization value="stretched" comment="Spectrum discretization:
        regular,random,stretched,cosstretched (default=stretched)" />
        <waveorder value="1" comment="Order wave generation 1:1st order, 2:2nd order (default=1)" />
        <waveheight value="1.5" comment="Significant Wave Height (Hs)" />
        <waveperiod value="9.5" comment="Peak Period" />
        <gainstroke value="1" comment="Gain factor to amplify/reduce the paddle stroke (default=1)" />
        <peakcoef value="3.3" comment="Peak enhancement coefficient (default=3.3)" />
        <waves value="3" comment="Number of waves to create irregular waves (default=50)" />
        <randomseed value="2" comment="Random seed to initialize a pseudorandom number generator" />
        <serieini value="0" autofit="true" comment="Initial time in irregular wave serie (default=0 and
        autofit=false)" />
        <ramptime value="0" comment="Time of ramp (default=0)" />
        <savemotion time="10" timedt="0.02" xpos="0.1" zpos="-0.5" comment="Saves motion data. xpos and
        zpos are optional. zpos=-depth of the measuring point" />
        <saveserie timemin="0" timemax="300" timedt="0.05" xpos="0" comment="Saves serie data (optional)" />
        <saveseriewaves timemin="0" timemax="1000" xpos="0" comment="Saves serie heights" />
        <calcserielength timemax="1000" comment="Calculates serie length (optional)" />
    </piston_spectrum>
</wavepaddles>
```

***variabledraft*:** position of the flap hinge with respect to the bottom (above the bottom <0; below the bottom >0) (default=0)

***flapaxis0*:** initial point defining the rotation axis

***flapaxis1*:** final point defining the rotation axis



3.1.1. XML file: Special-WavePaddles


```
<wavepaddles>
    <piston_solitary>
        <mkbound value="10" comment="Mk-Bound of selected particles" />
        <theory value="1" comment="Theory of generation 1:Rayleigh (Serre 1953), 2: Boussinesq
        (Goring 1978) 3: KdV (Clamond and Germain 1999) (def=2)" />
        <start value="0" comment="Start time (def=0)" />
        <depth value="0.4" comment="Water depth" />
        <waveheight value="0.12" comment="Wave height" />
        <durationcoef value="1" comment="Coefficient of movement duration (def=1)" />
        <pistondir x="1" y="0" z="0" comment="Movement direction (def=(1,0,0))" />
        <savemotion time="6" timedt="0.02" xpos="8" comment="Saves motion data. xpos is
        optional for elevation calculation" />
    </piston_solitary>
</wavepaddles>
```

***theory*:** theory of wave generation to move the piston

***depth*:** depth at the piston location

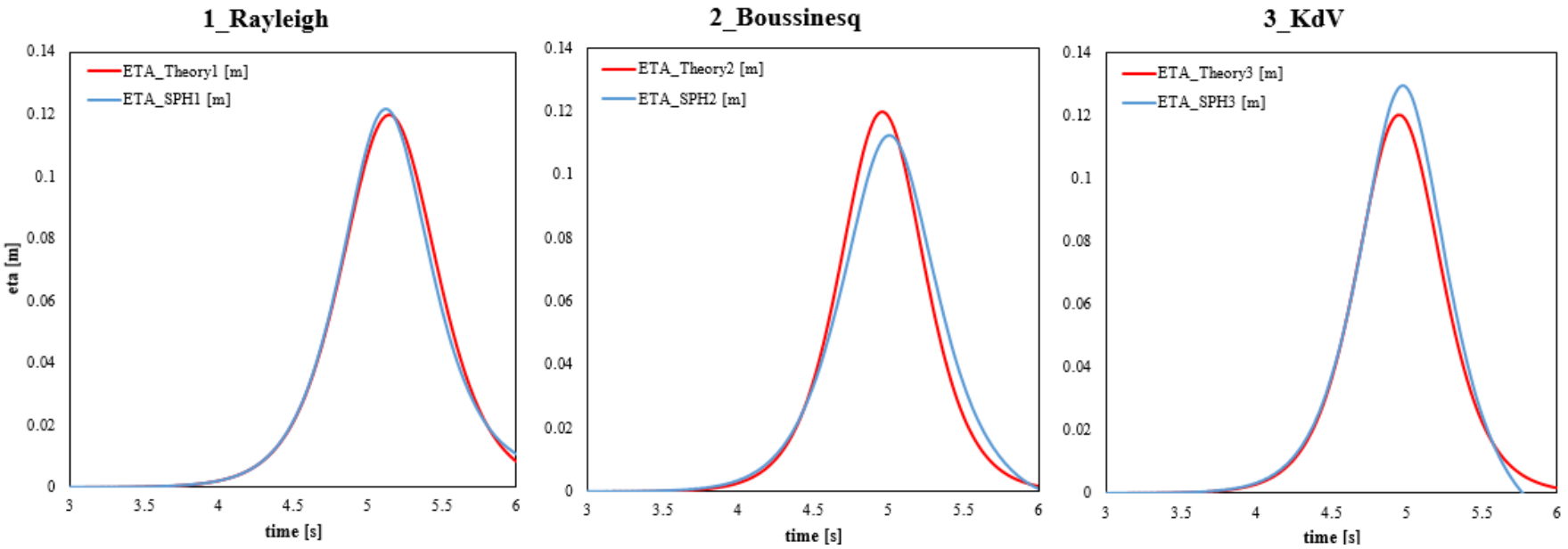
***durationcoef*:** coefficient to increase or reduce the movement duration

***waveheight*:** wave height H

***savemotion*:** saves piston movement and theoretical results of elevation at *xpos*

3.1.1. XML file: Special-WavePaddles


```
<wavepaddles>
    <piston_solitary>
        <mkbound value="10" comment="Mk-Bound of selected particles" />
        <theory value="1" comment="Theory of generation 1:Rayleigh (Serre 1953), 2: Boussinesq
        (Goring 1978) 3: KdV (Clamond and Germain 1999) (def=2)" />
        <start value="0" comment="Start time (def=0)" />
        <depth value="0.4" comment="Water depth" />
        <waveheight value="0.12" comment="Wave height" />
        <durationcoef value="1" comment="Coefficient of movement duration (def=1)" />
        <pistondir x="1" y="0" z="0" comment="Movement direction (def=(1,0,0))" />
        <savemotion time="6" timedt="0.02" xpos="8" comment="Saves motion data. xpos is
        optional for elevation calculation" />
    </piston_solitary>
</wavepaddles>
```



3.1.1. XML file: Special-WavePaddles

GENERATION OF **MULTIPLE SOLITARY** WAVES


```
<wavepaddles>
    <piston_solitary>
        <mkbound value="10" comment="Mk-Bound of selected particles" />
        <theory value="1" comment="Theory of generation 1:Rayleigh (Serre 1953), 2: Boussinesq (Goring 1978) 3: KdV
        (Clamond and Germain 1999) (def=2)" />
        <start value="0" comment="Start time (def=0)" />
        <pistondir x="1" y="0" z="0" comment="Movement direction (def=(1,0,0))" />
        <savemotion time="8" timedt="0.02" xpos="8" comment="Saves motion data. xpos is optional for elevation
        calculation" />
        <depth value="0.4" comment="Water depth" />
        <durationcoef value="1" comment="Coefficient of movement duration (def=1)" />
        <waveheight value="0.12" comment="Wave height" />
        <waves value="3" comment="Number of solitary waves (def=1)" />
        <startcoef2 value="0.6" comment="Defines start of next wave according to period of previous wave. (def=1)" />
        <waveheight2 value="0.18" comment="Wave height for 2nd wave (def=value of previous wave" />
        <startcoef3 value="0.6" comment="Defines start of next wave according to period of previous wave. (def=1)" />
        <waveheight3 value="0.24" comment="Wave height for 3rd wave (def=value of previous wave" />
    </piston_solitary>
</wavepaddles>
```

***waves*:** number of solitary waves

***startcoef2, startcoef3…*:** start of wave 2, 3… according to period of previous wave

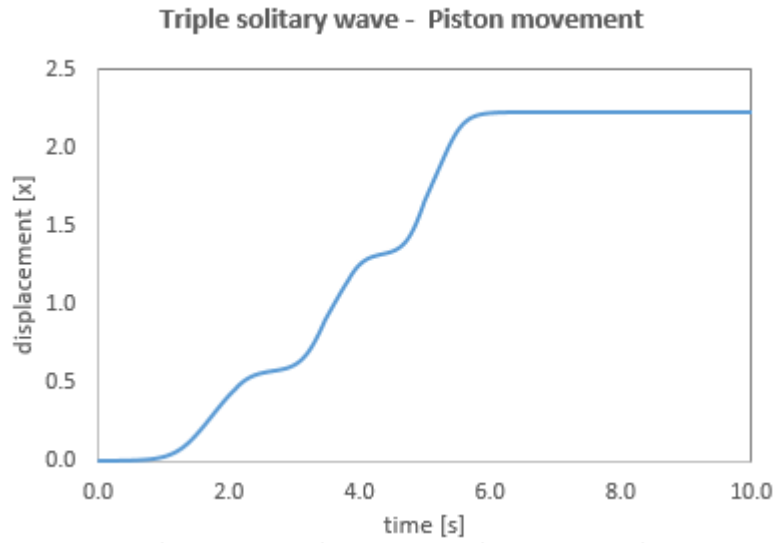
***durationcoef2, durationcoef3…*:** coefficient to increase or reduce the movement duration for each wave

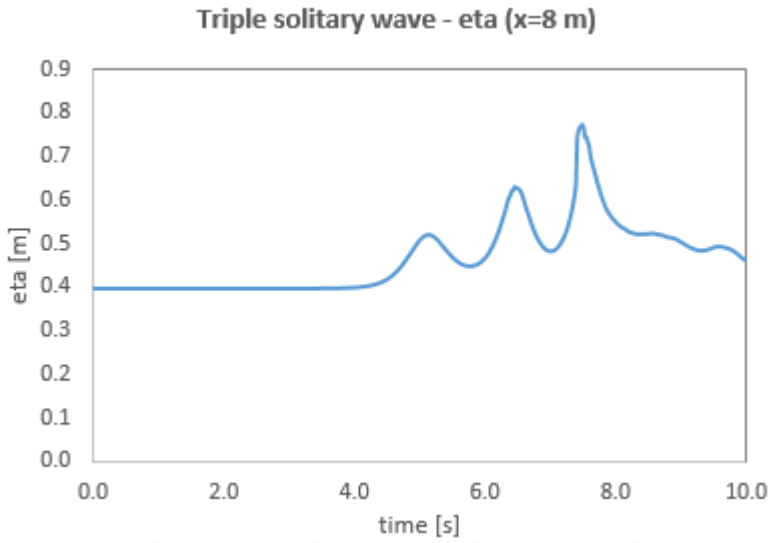
***waveheight2, waveheight3…*:** height H for each wave

3.1.1. XML file: Special-WavePaddles

GENERATION OF **MULTIPLE SOLITARY** WAVES


```
<wavepaddles>
    <piston_solitary>
        <mkbound value="10" comment="Mk-Bound of selected particles" />
        <theory value="1" comment="Theory of generation 1:Rayleigh (Serre 1953), 2: Boussinesq (Goring 1978) 3: KdV
        (Clamond and Germain 1999) (def=2)" />
        <start value="0" comment="Start time (def=0)" />
        <pistondir x="1" y="0" z="0" comment="Movement direction (def=(1,0,0))" />
        <savemotion time="8" timedt="0.02" xpos="8" comment="Saves motion data. xpos is optional for elevation
        calculation" />
        <depth value="0.4" comment="Water depth" />
        <durationcoef value="1" comment="Coefficient of movement duration (def=1)" />
        <waveheight value="0.12" comment="Wave height" />
        <waves value="3" comment="Number of solitary waves (def=1)" />
        <startcoef2 value="0.6" comment="Defines start of next wave according to period of previous wave. (def=1)" />
        <waveheight2 value="0.18" comment="Wave height for 2nd wave (def=value of previous wave" />
        <startcoef3 value="0.6" comment="Defines start of next wave according to period of previous wave. (def=1)" />
        <waveheight3 value="0.24" comment="Wave height for 3rd wave (def=value of previous wave" />
    </piston_solitary>
</wavepaddles>
```





3.1.2. XML file: Special-WavePaddles-AWAS

ACTIVE WAVE ABSORPTION


```
<wavepaddles>
    <piston>
        <awas_zsurf>
            <startawas value="0" comment="Time to start AWAS correction (default=start+ramp*waveperiod)" />
            <swl value="0.266" comment="Still water level (free-surface water)" />
            <elevation value="2" comment="Order wave to calculate elevation 1:1st order, 2:2nd order (default=2)" />
            <gaugex value="0.05" comment="Position in X from piston to measure free-surface water (default=5*Dp)" />
            <_gaugex valueh="3" comment="Position in X from piston to measure free-surface water (according H value)" />
            <_gaugex valuedp="5" comment="Position in X from piston to measure free-surface water (according Dp value)" />
            <gaugey value="1" comment="Position in Y to measure free-surface water" />
            <gaugezmin value="0.05" comment="Minimum position in Z to measure free-surface water, it must be in water
            (default=domain limits)" />
            <gaugezmax value="0.5" comment="Maximum position in Z to measure free-surface water (default=domain limits)" />
            <gaugedp value="0.25" comment="Resolution to measure free-surface water, it uses Dp*gaugedp (default=0.1)" />
            <_coefmasslimit value="0.4" comment="Coefficient to calculate mass of free-surface (default=0.5 on 3D and 0.4
            on 2D)" />
            <savedata value="1" comment="Saves CSV with information 1:by part, 2:more information, 3:by step (default=0)" />
            <limitace value="2" comment="Factor to limit maximum value of acceleration, with 0 disabled (default=2)" />
            <correction coefstroke="1.8" coefperiod="1" powerfunc="3" comment="Drift correction configuration (default=no
            applied)" />
        </awas_zsurf>
    </piston>
</wavepaddles>
```

***swl*:** z-coordinate of still water level with respect to the reference system

***gaugex*:** position from piston to measure free-surface elevation

- *value*: expressed in [m]
- *valueh*: expressed as multiple of the smoothing length, *h*.
- *valuedp*: expressed as multiple of the *dp*.

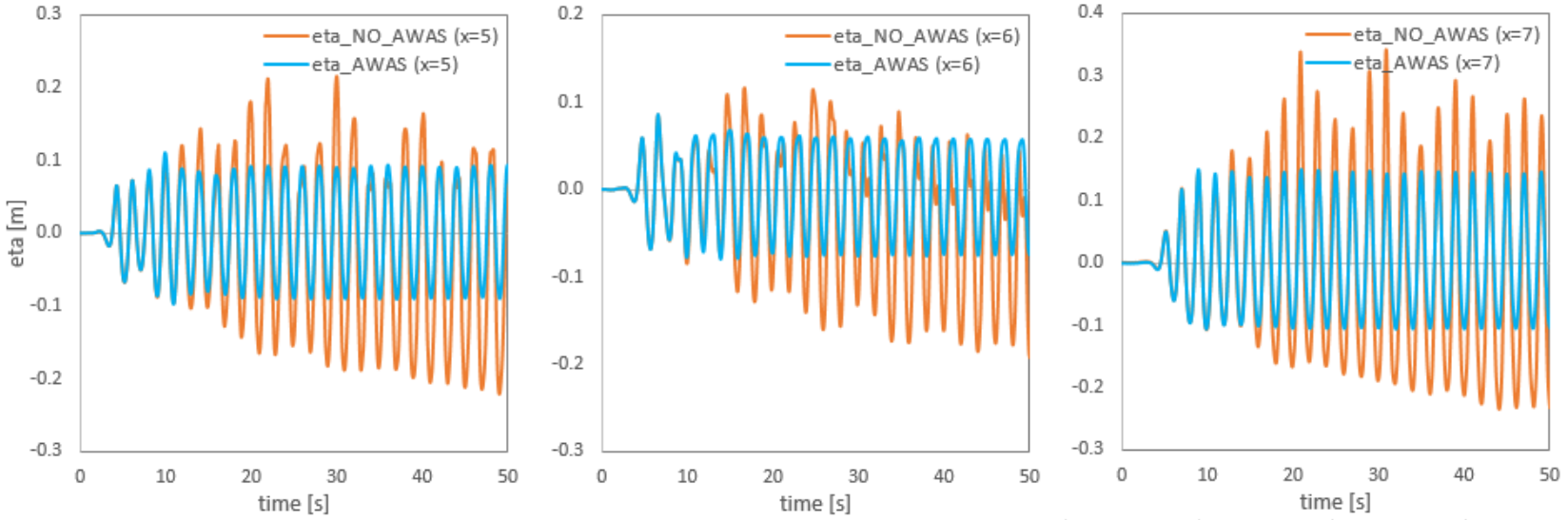
***limitace*:** factor to limit maximum value of acceleration of the avoid modes of vibration of very high frequencies

***correction*:** compensation of piston drift if the displacement is bigger than *coefstroke* times the piston stroke applied during a time equal to *coefperiod**T

3.1.2. XML file: Special-WavePaddles-AWAS

ACTIVE WAVE ABSORPTION


```
<wavepaddles>
    <piston>
        <awas_zsurf>
            <startawas value="0" comment="Time to start AWAS correction (default=start+ramp*waveperiod)" />
            <swl value="0.266" comment="Still water level (free-surface water)" />
            <elevation value="2" comment="Order wave to calculate elevation 1:1st order, 2:2nd order (default=2)" />
            <gaugex value="0.05" comment="Position in X from piston to measure free-surface water (default=5*Dp)" />
            <_gaugex valueh="3" comment="Position in X from piston to measure free-surface water (according H value)" />
            <_gaugex valuedp="5" comment="Position in X from piston to measure free-surface water (according Dp value)" />
            <gaugey value="1" comment="Position in Y to measure free-surface water" />
            <gaugezmin value="0.05" comment="Minimum position in Z to measure free-surface water, it must be in water
            (default=domain limits)" />
            <gaugezmax value="0.5" comment="Maximum position in Z to measure free-surface water (default=domain limits)" />
            <gaugedp value="0.25" comment="Resolution to measure free-surface water, it uses Dp*gaugedp (default=0.1)" />
            <_coefmasslimit value="0.4" comment="Coefficient to calculate mass of free-surface (default=0.5 on 3D and 0.4
            on 2D)" />
            <savedata value="1" comment="Saves CSV with information 1:by part, 2:more information, 3:by step (default=0)" />
            <limitace value="2" comment="Factor to limit maximum value of acceleration, with 0 disabled (default=2)" />
            <correction coefstroke="1.8" coefperiod="1" powerfunc="3" comment="Drift correction configuration (default=no
            applied)" />
        </awas_zsurf>
    </piston>
</wavepaddles>
```



3.1.3. XML file: Special-Damping


```
<damping>
    <dampingzone>
        <limitmin x="4" y="0" z="0" comment="Location where minimum reduction is applied" />
        <limitmax x="4.5" y="0" z="0" comment="Location where maximum reduction is applied" />
        <overlimit value="1" comment="The scope of maximum reduction over limitmax (default=0)" />
        <redumax value="10" comment="Maximum reduction in location limitmax (default=10)" />
        <factorxyz x="1" y="1" z="1" comment="Application factor in components (default x=1,y=1,z=1)" />
        <domain zmin="-1.2" zmax="0" comment="Box domain to apply damping (default=all domain)">
            <point x="5" y="2" />
            <point x="10" y="2" />
            <point x="10" y="8" />
            <point x="5" y="8" />
        </domain>
    </dampingzone>
</damping>
```

***limitmin*:** location where minimum reduction to velocity is applied ($\boldsymbol{x}_0$)

***limitmax*:** location where maximum reduction to velocity is applied ($\boldsymbol{x}_1$)

***overlimit*:** extends the area of maximum velocity reduction

***redumax*:** coefficient to increase or reduce damping ($\boldsymbol{\beta}$)

***factorxyz*:** allows to apply damping in some axis only

***domain*:** defines the area where damping is applied

$$\mathbf{v} = \mathbf{v}_0 \cdot f(x, dt)$$

$\mathbf{v}_0$ initial velocity of particle

$\mathbf{v}$ final velocity

$f(x, dt)$ reduction function

$$f(x, dt) = 1 - dt \cdot \beta \cdot \left(\frac{x - x_0}{x_1 - x_0} \right)^2$$

dt duration of time step

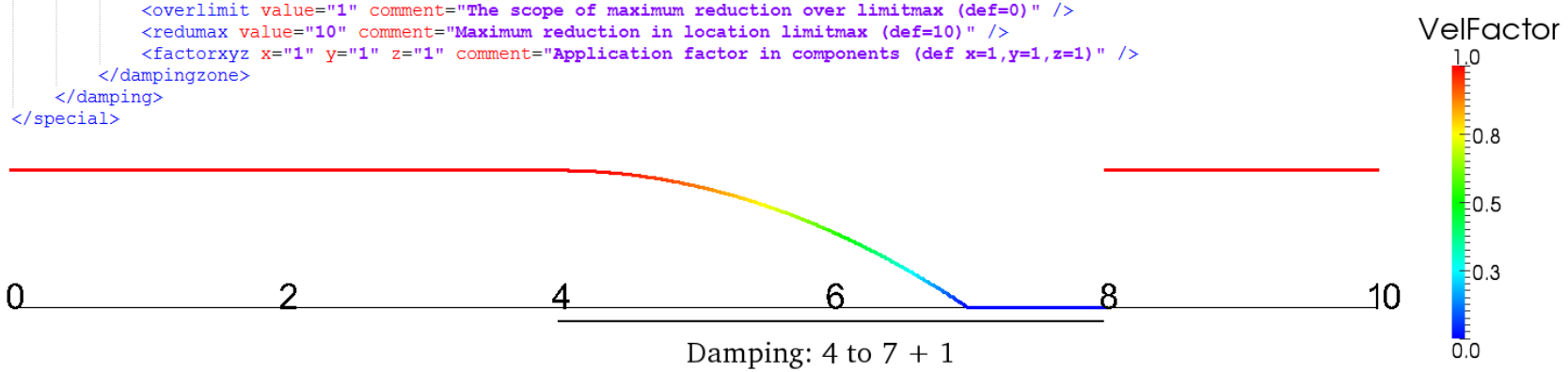
$\boldsymbol{x}$ particle position

$\boldsymbol{x}_0$ and $\boldsymbol{x}_1$ intial and final position of damping zone

3.1.3. XML file: Special-Damping

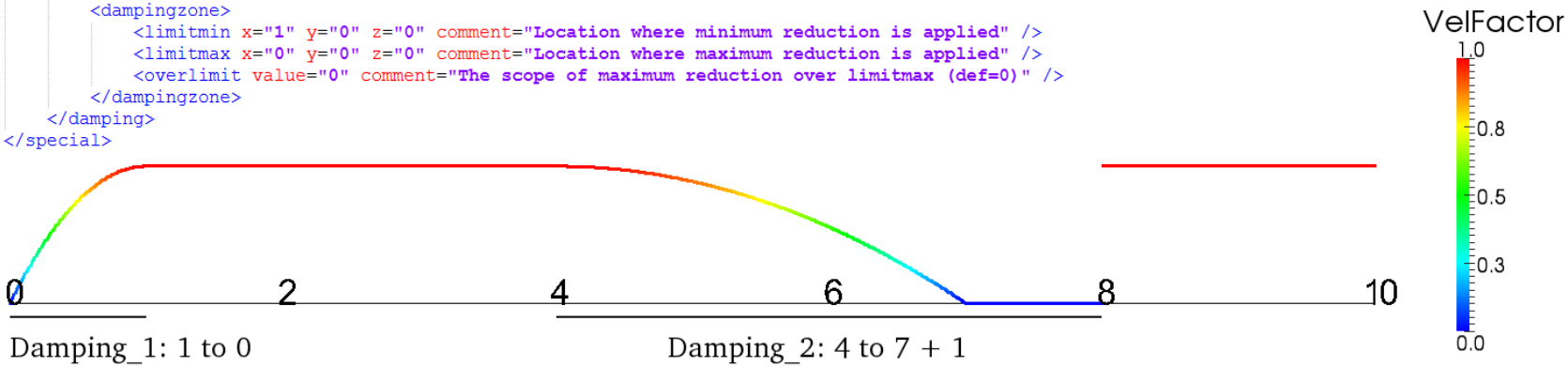
Example 1: standard damping on X direction

```
<special>
    <damping>
        <dampingzone>
            <limitmin x="4" y="0" z="0" comment="Location where minimum reduction is applied" />
            <limitmax x="7" y="0" z="0" comment="Location where maximum reduction is applied" />
            <overlimit value="1" comment="The scope of maximum reduction over limitmax (def=0)" />
            <redumax value="10" comment="Maximum reduction in location limitmax (def=10)" />
            <factorxyz x="1" y="1" z="1" comment="Application factor in components (def x=1,y=1,z=1)" />
        </dampingzone>
    </damping>
</special>
```



Example 2: two damping definitions

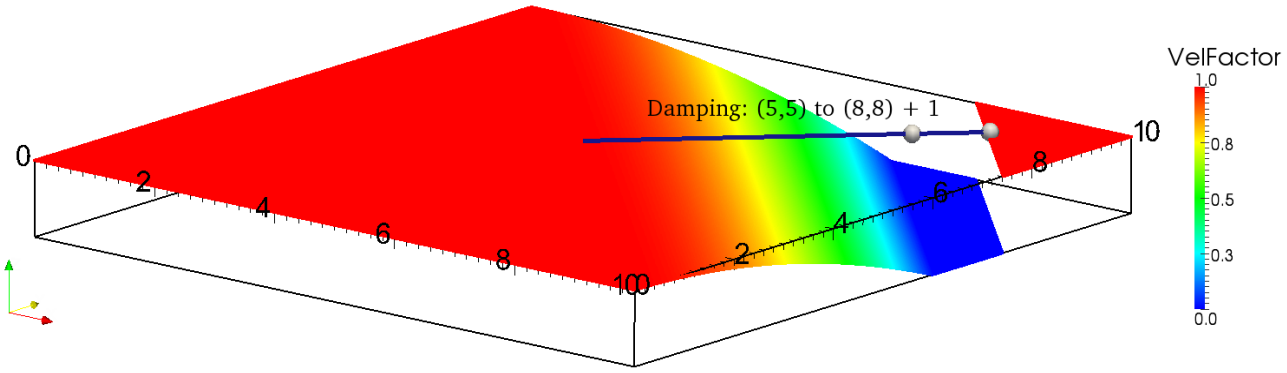
```
<special>
    <damping>
        <dampingzone>
            <limitmin x="4" y="0" z="0" comment="Location where minimum reduction is applied" />
            <limitmax x="7" y="0" z="0" comment="Location where maximum reduction is applied" />
            <overlimit value="1" comment="The scope of maximum reduction over limitmax (def=0)" />
        </dampingzone>
        <dampingzone>
            <limitmin x="1" y="0" z="0" comment="Location where minimum reduction is applied" />
            <limitmax x="0" y="0" z="0" comment="Location where maximum reduction is applied" />
            <overlimit value="0" comment="The scope of maximum reduction over limitmax (def=0)" />
        </dampingzone>
    </damping>
</special>
```



3.1.3. XML file: Special-Damping

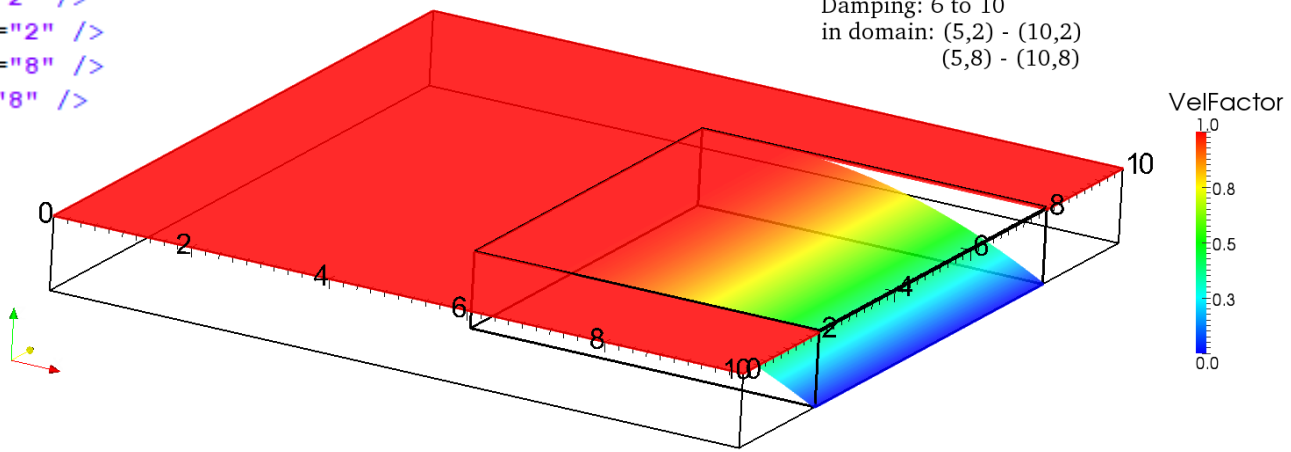
Example 3: damping on arbitrary direction

```
<damping>
    <dampingzone>
        <limitmin x="5" y="5" z="0" comment="Location where minimum reduction is applied" />
        <limitmax x="8" y="8" z="0" comment="Location where maximum reduction is applied" />
        <overlimit value="1" comment="The scope of maximum reduction over limitmax (default=0)" />
    </dampingzone>
</damping>
```



Example 4: using damping in a defined domain

```
<damping>
    <dampingzone>
        <limitmin x="6" y="0" z="0" comment="Location where minimum reduction is applied" />
        <limitmax x="10" y="0" z="0" comment="Location where maximum reduction is applied" />
        <overlimit value="0" comment="The scope of maximum reduction over limitmax (default=0)" />
        <domain zmin="0" zmax="10" comment="Box domain to apply damping (default=all domain)">
            <point x="5" y="2" />
            <point x="10" y="2" />
            <point x="10" y="8" />
            <point x="5" y="8" />
        </domain>
    </dampingzone>
</damping>
```

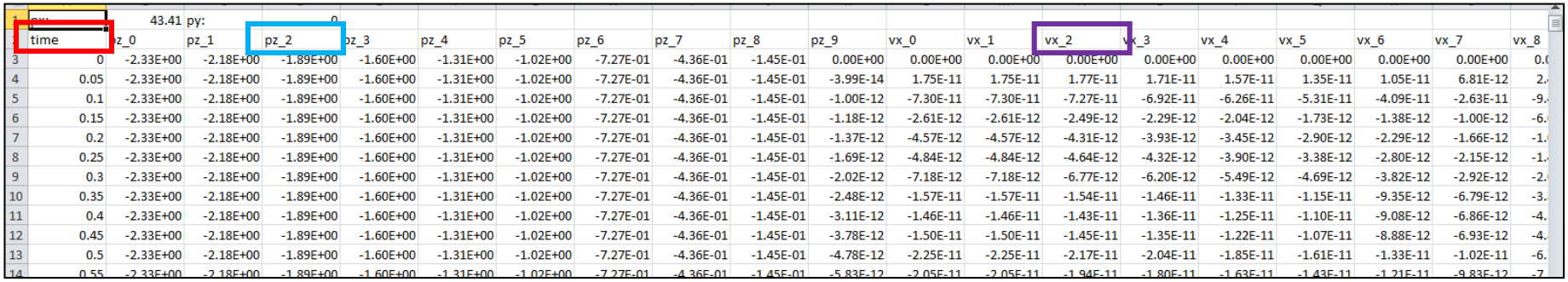


3.1.4. XML file: Special-MultiLayerPiston

Multi-Layer Pistons using external velocity data (for one-way offline coupling)

```
<mlayerpistons>
    <pistonId>
        <mkbound value="10" comment="Mk-Bound of selected particles" />
        <filevelx value="AwaReg1L_velx_x10_y00.csv" comment="File name with X velocity" />
        <incz value="0" comment="Z offset (def=0)" />
        <timedataini value="0" comment="Time offset (def=0)" />
        <smooth value="5" comment="Smooth motion level (def=0)" />
    </pistonId>
</mlayerpistons>
```

***filevelx*:** .csv file containing the information of horizontal velocity to be applied to the boundary particles forming the multi-layered piston. It contains time, z-position of each layer and velocity of each layer



px:		43.41	py:		0																	
time	z_0	pz_1	pz_2	pz_3	pz_4	pz_5	pz_6	pz_7	pz_8	pz_9	vx_0	vx_1	vx_2	vx_3	vx_4	vx_5	vx_6	vx_7	vx_8			
0	-2.33E+00	-2.18E+00	-1.89E+00	-1.60E+00	-1.31E+00	-1.02E+00	-7.27E-01	-4.36E-01	-1.45E-01	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.0
0.05	-2.33E+00	-2.18E+00	-1.89E+00	-1.60E+00	-1.31E+00	-1.02E+00	-7.27E-01	-4.36E-01	-1.45E-01	-3.99E-14	1.75E-11	1.75E-11	1.77E-11	1.71E-11	1.57E-11	1.35E-11	1.05E-11	6.81E-12	2.			
0.1	-2.33E+00	-2.18E+00	-1.89E+00	-1.60E+00	-1.31E+00	-1.02E+00	-7.27E-01	-4.36E-01	-1.45E-01	-1.00E-12	-7.30E-11	-7.30E-11	-7.27E-11	-6.92E-11	-6.26E-11	-5.31E-11	-4.09E-11	-2.63E-11	-9.			
0.15	-2.33E+00	-2.18E+00	-1.89E+00	-1.60E+00	-1.31E+00	-1.02E+00	-7.27E-01	-4.36E-01	-1.45E-01	-1.18E-12	-2.61E-12	-2.61E-12	-2.49E-12	-2.29E-12	-2.04E-12	-1.73E-12	-1.38E-12	-1.00E-12	-6.			
0.2	-2.33E+00	-2.18E+00	-1.89E+00	-1.60E+00	-1.31E+00	-1.02E+00	-7.27E-01	-4.36E-01	-1.45E-01	-1.37E-12	-4.57E-12	-4.57E-12	-4.31E-12	-3.93E-12	-3.45E-12	-2.90E-12	-2.29E-12	-1.66E-12	-1.			
0.25	-2.33E+00	-2.18E+00	-1.89E+00	-1.60E+00	-1.31E+00	-1.02E+00	-7.27E-01	-4.36E-01	-1.45E-01	-1.69E-12	-4.84E-12	-4.84E-12	-4.64E-12	-4.32E-12	-3.90E-12	-3.38E-12	-2.80E-12	-2.15E-12	-1.			
0.3	-2.33E+00	-2.18E+00	-1.89E+00	-1.60E+00	-1.31E+00	-1.02E+00	-7.27E-01	-4.36E-01	-1.45E-01	-2.02E-12	-7.18E-12	-7.18E-12	-6.77E-12	-6.20E-12	-5.49E-12	-4.69E-12	-3.82E-12	-2.92E-12	-2.			
0.35	-2.33E+00	-2.18E+00	-1.89E+00	-1.60E+00	-1.31E+00	-1.02E+00	-7.27E-01	-4.36E-01	-1.45E-01	-2.48E-12	-1.57E-11	-1.57E-11	-1.54E-11	-1.46E-11	-1.33E-11	-1.15E-11	-9.35E-12	-6.79E-12	-3.			
0.4	-2.33E+00	-2.18E+00	-1.89E+00	-1.60E+00	-1.31E+00	-1.02E+00	-7.27E-01	-4.36E-01	-1.45E-01	-3.11E-12	-1.46E-11	-1.46E-11	-1.43E-11	-1.36E-11	-1.25E-11	-1.10E-11	-9.08E-12	-6.86E-12	-4.			
0.45	-2.33E+00	-2.18E+00	-1.89E+00	-1.60E+00	-1.31E+00	-1.02E+00	-7.27E-01	-4.36E-01	-1.45E-01	-3.78E-12	-1.50E-11	-1.50E-11	-1.45E-11	-1.35E-11	-1.22E-11	-1.07E-11	-8.88E-12	-6.93E-12	-4.			
0.5	-2.33E+00	-2.18E+00	-1.89E+00	-1.60E+00	-1.31E+00	-1.02E+00	-7.27E-01	-4.36E-01	-1.45E-01	-4.78E-12	-2.25E-11	-2.25E-11	-2.17E-11	-2.04E-11	-1.85E-11	-1.61E-11	-1.33E-11	-1.02E-11	-6.			
0.55	-2.33E+00	-2.18E+00	-1.89E+00	-1.60E+00	-1.31E+00	-1.02E+00	-7.27E-01	-4.36E-01	-1.45E-01	-5.83E-12	-2.05E-11	-2.05E-11	-1.94E-11	-1.80E-11	-1.63E-11	-1.43E-11	-1.21E-11	-9.83E-12	-7			

***incz*:** offset to apply to the z-coordinate contained in the .csv file. Usually it is the difference in z between the coordinate of the still water level from the wave propagation model and the one in DualSPHysics.

***smooth*:** number of neighbor particles considered to apply a smoothing average on the velocity along the z-axis and avoid solution of continuity in the multi-layered piston.

3.1.5. XML file: Special-RelaxationZones

Relaxation zone to generate **regular waves** (DualSPHysics stand alone)

```
<relaxationzones>
    <rzwaves_regular>
        <start value="0" comment="Start time (default=0)" />
        <duration value="0" comment="Movement duration, Zero is the end of simulation (default=0)" />
        <waveorder value="2" comment="Order wave generation 1:1st order, 2:2nd order (default=1)" />
        <waveheight value="0.16" comment="Wave height" />
        <waveperiod value="2" comment="Wave period" />
        <depth value="0.7" comment="Water depth for wave generation" />
        <swl value="0.7" comment="Still water level (free-surface)" />
        <center x="0" y="0" z="0" comment="Central point of application" />
        <width value="0.2" comment="Width for generation" />
        <phase value="0" comment="Initial wave phase in function of PI (default=0)" />
        <ramp value="1" comment="Periods of ramp (default=0)" />
        <savemotion periods="24" periodsteps="20" xpos="13" zpos="-0.5" comment="Saves motion data. xpos and zpos
        are optional. zpos=-depth of the measuring point" />
        <coefdir x="1" y="0" z="0" comment="Coefficients for each direction (default=(1,0,0))" />
        <coefdt value="1000" comment="Multiplies by dt value in the calculation (using 0 is not applied)
        (default=1000)" />
        <function psi="0.9" beta="1" comment="Coefficients in funtion for velocity (def. psi=0.9, beta=1)" />
        <driftcorrection value="1.0" comment="Coefficient of drift correction applied in velocity X. 0:Disabled,
        1:Full correction (def=0)" />
    </rzwaves_regular>
</relaxationzones>
```

***center*:** coordinates defining the center of the relaxation zone

***width*:** relaxation zone width, defined as the total width with center specified above

***function*:** values of the parameters ψ and β used to calculate the shape of the weighting function applied within the relaxation zone

***driftcorrection*:** weight to apply to the drift correction, calculated starting from Stokes' second-order wave theory

3.1.5. XML file: Special-RelaxationZones

Relaxation zone to generate **irregular waves** (DualSPHysics stand alone)

```
<relaxationzones>
    <rzwaves_spectrum>
        <start value="0" comment="Start time (def=0)" />
        <duration value="0" comment="Movement duration, Zero is the end of simulation (def=0)" />
        <peakcoef value="3.3" comment="Peak enhancement coefficient (default=3.3)" />
        <spectrum value="jonswap" comment="Spectrum type: jonswap,pierson-moskowitz" />
        <discretization value="stretched" comment="Spectrum discretization: regular,random,stretched,cosstretched
        (default=stretched)" />
        <waveorder value="1" comment="Order wave generation 1:1st order, 2:2nd order (default=1)" />
        <waveheight value="0.1" comment="Significant Wave Height" />
        <waveperiod value="3" comment="Peak Period" />
        <waves value="1024" comment="Number of waves to create irregular waves (default=50)" />
        <randomseed value="2" comment="Random seed to initialize a pseudorandom number generator" />
        <depth value="0.7" comment="Water depth for wave generation" />
        <swl value="0.7" comment="Still water level (free-surface)" />
        <center x="0" y="0" z="0" comment="Central point of application" />
        <width value="0.2" comment="Width for generation" />
        <ramptime value="0" comment="Time of initial ramp (default=0)" />
        <serieini value="0" comment="Initial time in irregular wave serie (default=0)" />
        <savemotion time="50" timedt="0.1" xpos="14" zpos="-0.5" comment="Saves motion data. xpos and zpos are
        optional. zpos=-depth of the measuring point" />
        <saveserie timemin="0" timemax="1000" timedt="0.1" xpos="0" comment="Saves serie data (optional)" />
        <saveseriewaves timemin="0" timemax="1000" xpos="0" comment="Saves serie heights" />
        <_calcserielength timemax="1000" comment="Calculates serie length (optional)" />
        <coefdir x="1" y="0" z="0" comment="Coefficients for each direction (default=(1,0,0))" />
        <coefdt value="1000" comment="Multiplies by dt value in the calculation (using 0 is not applied)
        (default=1000)" />
        <function psi="0.9" beta="1" comment="Coefficients in funtion for velocity (def. psi=0.9, beta=1)" />
        <driftcorrection value="1.0" comment="Coefficient of drift correction applied in velocity X. 0:Disabled,
        1:Full correction (def=0)" />
    </rzwaves_spectrum>
</relaxationzones>
```

Parameters coincide with the generation of irregular waves using paddles

3.1.5. XML file: Special-RelaxationZones

Relaxation zone to generate waves using **external time series** (offline coupling with wave propagation model)

```
<relaxationzones>
    <rzwaves_external_ld>
        <start value="0" comment="Start time (def=0)" />
        <duration value="0" comment="Movement duration, Zero is the end of simulation (def=0)" />
        <depth value="0.7" comment="Water depth. It is necessary for drift correction (def=0)" />
        <swl value="0.7" comment="Still water level (free-surface). It is necessary for drift correction (def=0)" />
        <filesvel value="SgWave4i3L_corr" comment="Main name of files with velocity to use" />
        <filesvelx initial="2" count="5" comment="First file and count to use" />
        <usevelz value="false" comment="Use velocity in Z or not (def=false)" />
        <movedata x="1" y="0" z="0" comment="Movement of data in CSV files" />
        <dpz valuedp="1" comment="Distance between key points in Z (def=2)" />
        <smooth value="5" comment="Smooth motion level (def=0)" />
        <center x="0" y="0" z="0" comment="Central point of application" />
        <width value="0.2" comment="Width for generation" />
        <coefdir x="1" y="0" z="0" comment="Coefficients for each direction (default=(1,0,0))" />
        <coefdt value="1000" comment="Multiplies by dt value in the calculation (using 0 is not applied)
        (default=1000)" />
        <function psi="0.9" beta="1" comment="Coefficients in funtion for velocity (def. psi=0.9, beta=1)" />
        <driftcorrection value="1.0" comment="Coefficient of drift correction applied in velocity X. 0:Disabled,
        1:Full correction (def=0)" />
        <driftinitialramp value="10" comment="Ignore waves from external data in initial seconds (def=0)" />
    </rzwaves_external_ld>
</relaxationzones>
```

***center*:** coordinates defining the center of the relaxation zone

***width*:** relaxation zone width, defined as the total width with center specified above

***function*:** values of the parameters ψ and β used to calculate the shape of the weighting function applied within the relaxation zone

***driftcorrection*:** weight to apply to the drift correction, calculated starting from Stokes' second-order wave theory

***driftinitialramp*:** delay to apply drift correction

3.1.5. XML file: Special-RelaxationZones

OFFLINE COUPLING

Relaxation zone to generate waves using **external time series** (offline coupling with wave propagation model)

```
<relaxationzones>
    <rzwaves_external_ld>
        <start value="0" comment="Start time (def=0)" />
        <duration value="0" comment="Movement duration, Zero is the end of simulation (def=0)" />
        <depth value="0.7" comment="Water depth. It is necessary for drift correction (def=0)" />
        <swl value="0.7" comment="Still water level (free-surface). It is necessary for drift correction (def=0)" />
        <filesvel value="SgWave4i3L_corr" comment="Main name of files with velocity to use" />
        <filesvelx initial="2" count="5" comment="First file and count to use" />
        <usevelz value="false" comment="Use velocity in Z or not (def=false)" />
        <movedata x="1" y="0" z="0" comment="Movement of data in CSV files" />
        <dpz valuedp="1" comment="Distance between key points in Z (def=2)" />
        <smooth value="5" comment="Smooth motion level (def=0)" />
        <center x="0" y="0" z="0" comment="Central point of application" />
        <width value="0.2" comment="Width for generation" />
        <coefdir x="1" y="0" z="0" comment="Coefficients for each direction (default=(1,0,0))" />
        <coefdt value="1000" comment="Multiplies by dt value in the calculation (using 0 is not applied)
        (default=1000)" />
        <function psi="0.9" beta="1" comment="Coefficients in funtion for velocity (def. psi=0.9, beta=1)" />
        <driftcorrection value="1.0" comment="Coefficient of drift correction applied in velocity X. 0:Disabled,
        1:Full correction (def=0)" />
        <driftinitialramp value="10" comment="Ignore waves from external data in initial seconds (def=0)" />
    </rzwaves_external_ld>
</relaxationzones>
```

***depth*:** water depth at wave generation location (i.e. relaxation zone location)

***swl*:** z-coordinate of the still water level

filesvel*:** name of the .csv files (SgWave4i3L_corr_velz_xNN_yMM***) generated by the wave propagation model and containing the horizontal velocity information. If SWASH model (http://swash.sourceforge.net/) is used as wave propagation model, the .csv files can be generated using *ReadSwash2_win64.exe*

***filesvelX*:** index of the first velocity file to use and total number of files to be considered

3.1.5. XML file: Special-RelaxationZones

UNIFORM VELOCITY

Relaxation zone to impose uniform velocity profiles

```
<relaxationzones>
    <rzwaves_uniform>
        <start value="0" comment="Start time (default=0)" />
        <duration value="0" comment="Duration, Zero is the end of simulation (default=0)" />
        <domainbox>
            <point x="0.3" y="-0.05" z="0" />
            <size x="0.4" y="0.1" z="0.2" />
            <direction x="-1" y="0" z="0" />
            <rotateaxis angle="-45" anglesunits="degrees">
                <point1 x="0.3" y="0" z="0" />
                <point2 x="0.3" y="1" z="0" />
            </rotateaxis>
        </domainbox>
        <velocity value="0.3" comment="Velocity to impose (it is ignored when velocitytimes is defined)" />
        <velocitytimes comment="Uniform velocity in time">
            <timevalue time="0.0" v="0" />
            <timevalue time="2.0" v="0.5" />
            <timevalue time="4.0" v="0" />
            <timevalue time="5.0" v="0" />
            <timevalue time="7.0" v="0.4" />
        </velocitytimes>
        <coefdt value="1000" comment="Multiplies by dt value in the calculation (using 0 is not applied)
        (default=1000)" />
        <function psi="0.9" beta="1" comment="Coefficients in funtion for velocity (def. psi=0.9, beta=1)" />
    </rzwaves_uniform>
</relaxationzones>
```

***velocity*:** constant velocity value applied within the relaxation zone

***velocitytimes*:** it is used if a variation in time of the velocity is required: "time" defines the initial time when the new value of velocity "v" is applied.

***domainbox*:** domain within the velocity profile is applied (=relaxation zone)

3.1.6. XML file: Special-Gauges

Gauges section configures multiple measurements (fluid velocity, free surface limit, maximum elevation, force exerted by the fluid) that are performed during the simulation and their results can be used in the simulation itself.

```
<gauges>
    <!-- Defines the default configuration for all gauges. -->
    <default>
        <savevtkpart value="false" comment="Creates VTK files for each PART (default=false)" />
        <computedt value="0.001" comment="Time between measurements. 0:all steps (default=TimeOut)" units_comment="s" />
        <computetime start="0.1" end="0.2" comment="Start and end of measures. (default=simulation time)" units_comment="s" />
        <output value="true" comment="Creates CSV files of measurements (default=false)" />
        <outputdt value="0" comment="Time between output measurements. 0:all steps (default=TimeOut)" units_comment="s" />
        <outputtime start="0" end="10" comment="Start and end of output measures. (default=simulation time)" units_comment="s" />
    </default>
    <!-- Calculates velocity in one point and changes default configuration parameters. -->
    <velocity name="Vel-0">
        <point x="1.0" y="0" z="0.1" comment="Measurement position" units_comment="m" />
    </velocity>
    <!-- Only "start" is defined in <compute> and <output>. -->
    <velocity name="Vel_start">
        <point x="1.0" y="0" z="0.1" comment="Measurement position" units_comment="m" />
    </velocity>
    <!-- Calculates Surface Water Level in a line of points. -->
    <swl name="Swl_x09">
        <masslimit value="0.1" comment="Mass value to calculate free-surface (default=uses coefmasslimit)" units_comment="kg" />
        <pointdp value="0.2" comment="Distance between check points" units_comment="m" />
        <point0 x="0.9" y="0" z="0" comment="Initial point" units_comment="m" />
        <point2 x="0.9" y="0" z="2.1" comment="Final point" units_comment="m" />
    </swl>
    <!-- Calculates maximum z of fluid at distance of a vertical line. -->
    <maxz name="MaxX_x09a">
        <point0 x="0.9" y="0" z="0" comment="Bottom point" units_comment="m" />
        <height value="2.1" comment="Maximum height to check" units_comment="m" />
        <distlimit value="0.04" comment="Radio to check particles" units_comment="m" />
        <_distlimit coefdp="2" comment="Radio to check particles (coefdp*Dp)" />
        <_distlimit coefh="0.1" comment="Radio to check particles (coefh*H)" />
    </maxz>
    <!-- Calculates force sumation on selected fixed or moving particles (using only fluid particles). -->
    <force name="Force1">
        <target mkbound="1" comment="Indicates the mkbound of selected particles to compute forces" />
    </force>
</gauges>
```

3.1.6. XML file: Special-Gauges


```
<gauges>
    <!-- Defines the default configuration for all gauges. -->
    <default>
        <savevtkpart value="false" comment="Creates VTK files for each PART (default=false)" />
        <computedt value="0.001" comment="Time between measurements. 0:all steps (default=TimeOut)"
        units_comment="s" />
        <computetime start="0.1" end="0.2" comment="Start and end of measures. (default=simulation time)"
        units_comment="s" />
        <output value="true" comment="Creates CSV files of measurements (default=false)" />
        <outputdt value="0" comment="Time between output measurements. 0:all steps (default=TimeOut)"
        units_comment="s" />
        <outputtime start="0" end="10" comment="Start and end of output measures. (default=simulation time)"
        units_comment="s" />
    </default>
    <!-- Calculates velocity in one point and changes default configuration parameters. -->
```

Configuration applied **by default** to all gauges. Each gauge can modify these parameters.

```
    <!-- Calculates Surface Water Level in a line of points. -->
    <swl name="Swl_x09">
        <masslimit value="0.1" comment="Mass value to calculate free-surface (default=uses coefmasslimit)"
        units_comment="kg" />
        <pointdp value="0.2" comment="Distance between check points" units_comment="m" />
        <point0 x="0.9" y="0" z="0" comment="Initial point" units_comment="m" />
        <point2 x="0.9" y="0" z="2.1" comment="Final point" units_comment="m" />
    </swl>
    <!-- Calculates maximum z of fluid at distance of a vertical line. -->
    <maxz name="MaxZ_x09a">
        <point0 x="0.9" y="0" z="0" comment="Bottom point" units_comment="m" />
        <height value="2.1" comment="Maximum height to check" units_comment="m" />
        <distlimit value="0.04" comment="Radio to check particles" units_comment="m" />
        <_distlimit coefdp="2" comment="Radio to check particles (coefdp*Dp)" />
        <_distlimit coefh="0.1" comment="Radio to check particles (coefh*H)" />
    </maxz>
    <!-- Calculates force sumation on selected fixed or moving particles (using only fluid particles). -->
    <force name="Force1">
        <target mkbound="1" comment="Indicates the mkbound of selected particles to compute forces" />
    </force>
</gauges>
```

3.1.6. XML file: Special-Gauges

VELOCITY


```
<gauges>
    <!-- Defines the default configuration for all gauges. -->
    <default>
        <savevtkpart value="false" comment="Creates VTK files for each PART (default=false)" />
        <computedt value="0.001" comment="Time between measurements. 0:all steps (default=TimeOut)"
        units_comment="s" />
        <computetime start="0.1" end="0.2" comment="Start and end of measures. (default=simulation time)"
        units_comment="s" />
        <output value="true" comment="Creates CSV files of measurements (default=false)" />
```

Velocity: Measures the fluid velocity in a position (x, y, z). Each gauge is identified by a unique name.

```
    <!-- Calculates velocity in one point and changes default configuration parameters. -->
    <velocity name="Vel-0">
        <point x="1.0" y="0" z="0.1" comment="Measurement position" units_comment="m" />
    </velocity>
    <swl name="Swl_x09">
        <masslimit value="0.1" comment="Mass value to calculate free-surface (default=uses coefmasslimit)"
        units_comment="kg" />
        <pointdp value="0.2" comment="Distance between check points" units_comment="m" />
        <point0 x="0.9" y="0" z="0" comment="Initial point" units_comment="m" />
        <point2 x="0.9" y="0" z="2.1" comment="Final point" units_comment="m" />
    </swl>
    <!-- Calculates maximum z of fluid at distance of a vertical line. -->
    <maxz name="MaxZ_x09a">
        <point0 x="0.9" y="0" z="0" comment="Bottom point" units_comment="m" />
        <height value="2.1" comment="Maximum height to check" units_comment="m" />
        <distlimit value="0.04" comment="Radio to check particles" units_comment="m" />
        <_distlimit coefdp="2" comment="Radio to check particles (coefdp*Dp)" />
        <_distlimit coefh="0.1" comment="Radio to check particles (coefh*H)" />
    </maxz>
    <!-- Calculates force sumation on selected fixed or moving particles (using only fluid particles). -->
    <force name="Force1">
        <target mkbound="1" comment="Indicates the mkbound of selected particles to compute forces" />
    </force>
</gauges>
```

3.1.6. XML file: Special-Gauges

SWL


```
<gauges>
    <!-- Defines the default configuration for all gauges. -->
    <default>
        <savevtkpart value="false" comment="Creates VTK files for each PART (default=false)" />
        <computedt value="0.001" comment="Time between measurements. 0:all steps (default=TimeOut)"
        units_comment="s" />
        <computetime start="0.1" end="0.2" comment="Start and end of measures. (default=simulation time)"
        units_comment="s" />
        <output value="true" comment="Creates CSV files of measurements (default=false)" />
        <outputdt value="0" comment="Time between output measurements. 0:all steps (default=TimeOut)"
        units_comment="s" />
        <outputtime start="0" end="10" comment="Start and end of output measures. (default=simulation time)"
        units_comment="s" />
    </default>
```

Swl: Calculates the Free Surface Limit along a line (between *point0* and *point1*).

```
    <!-- Calculates Surface Water Level in a line of points. -->
    <swl name="Swl_x09">
        <masslimit value="0.1" comment="Mass value to calculate free-surface (default=uses coefmasslimit)"
        units_comment="kg" />
        <pointdp value="0.2" comment="Distance between check points" units_comment="m" />
        <point0 x="0.9" y="0" z="0" comment="Initial point" units_comment="m" />
        <point2 x="0.9" y="0" z="2.1" comment="Final point" units_comment="m" />
    </swl>
    <maxz name="MaxZ_x09a">
        <point0 x="0.9" y="0" z="0" comment="Bottom point" units_comment="m" />
        <height value="2.1" comment="Maximum height to check" units_comment="m" />
        <distlimit value="0.04" comment="Radio to check particles" units_comment="m" />
        <_distlimit coefdp="2" comment="Radio to check particles (coefdp*Dp)" />
        <_distlimit coefh="0.1" comment="Radio to check particles (coefh*H)" />
    </maxz>
    <!-- Calculates force sumation on selected fixed or moving particles (using only fluid particles). -->
    <force name="Force1">
        <target mkbound="1" comment="Indicates the mkbound of selected particles to compute forces" />
    </force>
</gauges>
```

3.1.6. XML file: Special-Gauges

MAXZ


```
<gauges>
    <!-- Defines the default configuration for all gauges. -->
    <default>
        <savevtkpart value="false" comment="Creates VTK files for each PART (default=false)" />
        <computedt value="0.001" comment="Time between measurements. 0:all steps (default=TimeOut)"
        units_comment="s" />
        <computetime start="0.1" end="0.2" comment="Start and end of measures. (default=simulation time)"
        units_comment="s" />
        <output value="true" comment="Creates CSV files of measurements (default=false)" />
        <outputdt value="0" comment="Time between output measurements. 0:all steps (default=TimeOut)"
        units_comment="s" />
        <outputtime start="0" end="10" comment="Start and end of output measures. (default=simulation time)"
        units_comment="s" />
    </default>
    <!-- Calculates velocity in one point and changes default configuration parameters. -->
    <velocity name="Vel-0">
        <point x="1.0" y="0" z="0.1" comment="Measurement position" units_comment="m" />
    </velocity>
    <!-- Calculates Surface Water Level in a line of points. -->
    <swl name="Swl_x09">
        <masslimit value="0.1" comment="Mass value to calculate free-surface (default=uses coefmasslimit)"
```

Maxz: Obtains maximum z of fluid particles in a zone defined by the position *point0* and a search distance according Dp, Coefh or guiven value.

```
    </swl>
    <!-- Calculates maximum z of fluid at distance of a vertical line. -->
    <maxz name="MaxZ_x09a">
        <point0 x="0.9" y="0" z="0" comment="Bottom point" units_comment="m" />
        <height value="2.1" comment="Maximum height to check" units_comment="m" />
        <distlimit value="0.04" comment="Radio to check particles" units_comment="m" />
        <_distlimit coefdp="2" comment="Radio to check particles (coefdp*Dp)" />
        <_distlimit coefh="0.1" comment="Radio to check particles (coefh*H)" />
    </maxz>
    <!-- Calculates force sumation on selected fixed or moving particles (using only fluid particles). -->
    <force name="Force1">
        <target mkbound="1" comment="Indicates the mkbound of selected particles to compute forces" />
    </force>
</gauges>
```

3.1.6. XML file: Special-Gauges

FORCE


```
<gauges>
    <!-- Defines the default configuration for all gauges. -->
    <default>
        <savevtkpart value="false" comment="Creates VTK files for each PART (default=false)" />
        <computedt value="0.001" comment="Time between measurements. 0:all steps (default=TimeOut)"
        units_comment="s" />
        <computetime start="0.1" end="0.2" comment="Start and end of measures. (default=simulation time)"
        units_comment="s" />
        <output value="true" comment="Creates CSV files of measurements (default=false)" />
        <outputdt value="0" comment="Time between output measurements. 0:all steps (default=TimeOut)"
        units_comment="s" />
        <outputtime start="0" end="10" comment="Start and end of output measures. (default=simulation time)"
        units_comment="s" />
    </default>
    <!-- Calculates velocity in one point and changes default configuration parameters. -->
    <velocity name="Vel-0">
        <point x="1.0" y="0" z="0.1" comment="Measurement position" units_comment="m" />
    </velocity>
    <!-- Calculates Surface Water Level in a line of points. -->
    <swl name="Swl_x09">
        <masslimit value="0.1" comment="Mass value to calculate free-surface (default=uses coefmasslimit)"
        units_comment="kg" />
        <pointdp value="0.2" comment="Distance between check points" units_comment="m" />
        <point0 x="0.9" y="0" z="0" comment="Initial point" units_comment="m" />
        <point2 x="0.9" y="0" z="2.1" comment="Final point" units_comment="m" />
    </swl>
    <!-- Calculates maximum z of fluid at distance of a vertical line. -->
    <maxz name="MaxZ_x09a">
        <point0 x="0.9" y="0" z="0" comment="Bottom point" units_comment="m" />
```

Force: Obtains the force summation exerted by fluid on fixed or moving particles according the MkBound value.

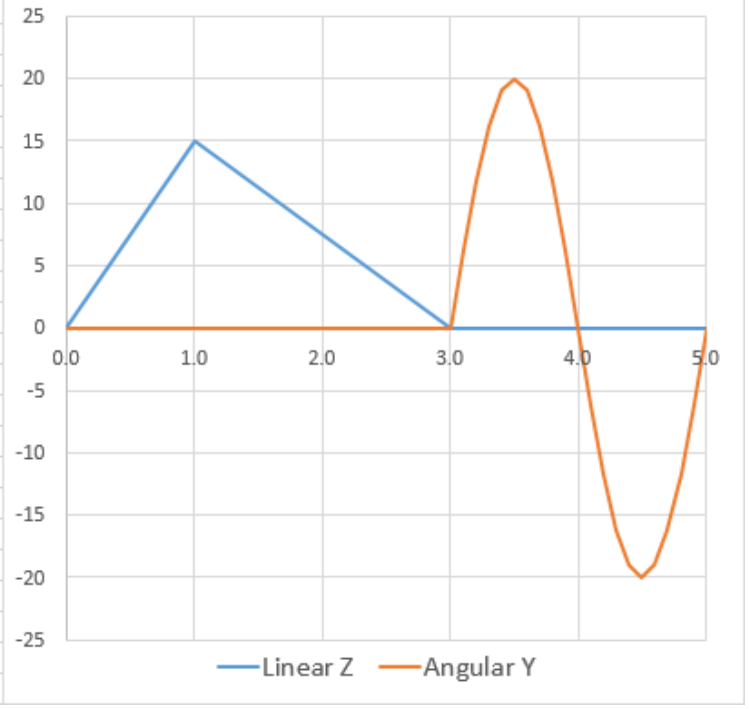
```
    <!-- Calculates force sumation on selected fixed or moving particles (using only fluid particles). -->
    <force name="Force1">
        <target mkbound="1" comment="Indicates the mkbound of selected particles to compute forces" />
    </force>
</gauges>
```

3.1.7. XML file: Special-Accinputs

Imposed acceleration to fluid particles. Linear and angular acceleration according time is obtained from an external CSV file. Imposed acceleration is applied according to MkFluid.

```
<accinputs>
    <accinput>
        <mkfluid value="14" comment="Mk-Fluid of selected particles" />
        <acccentre x="0" y="0" z="0" comment="Center of acceleration" />
        <globalgravity value="0" comment="Global gravity enabled (1) or disabled (0)" />
        <datafile value="CaseForcesData_0.csv" comment="File with acceleration data" />
    </accinput>
</accinputs>
```

	#Time	LinearAccX	LinearAccY	LinearAccZ	AngularAccX	AngularAccY	AngularAccZ
2	0.0	0	0	0	0	0	0
3	1.0	0	0	15	0	0	0
4	3.0	0	0	0	0	0	0
5	3.1	0	0	0	0	6.18033989	0
6	3.2	0	0	0	0	11.755705	0
7	3.3	0	0	0	0	16.1803399	0
8	3.4	0	0	0	0	19.0211303	0
9	3.5	0	0	0	0	20	0
10	3.6	0	0	0	0	19.0211303	0
11	3.7	0	0	0	0	16.1803399	0
12	3.8	0	0	0	0	11.755705	0
13	3.9	0	0	0	0	6.18033989	0
14	4.0	0	0	0	0	0	0
15	4.1	0	0	0	0	-6.1803399	0
16	4.2	0	0	0	0	-11.755705	0
17	4.3	0	0	0	0	-16.18034	0
18	4.4	0	0	0	0	-19.02113	0
19	4.5	0	0	0	0	-20	0
20	4.6	0	0	0	0	-19.02113	0
21	4.7	0	0	0	0	-16.18034	0
22	4.8	0	0	0	0	-11.755705	0
23	4.9	0	0	0	0	-6.1803399	0
24	5.0	0	0	0	0	0	0



3.1.8. XML file: Special-SaveDt

Saves Dt values used in the DualSPHysics simulation using different configuration parameters

```
<savedt>
    <start value="0" comment="Initial time (def=0)" />
    <finish value="1000" comment="End time (def=0, no limit)" />
    <interval value="0.01" comment="Time between output data (def=TimeOut)" />
    <fullinfo value="1" comment="Saves AceMax, ViscDtMax and VelMax (def=0)" />
    <alldt value="1" comment="Saves file with all dt values (def=0)" />
</savedt>
```

3.1.9. XML file: Special-TimeOut

Defines a **variable frequency to save results** (binary files with particle data) of the simulation

```
<timeout>
    <tout time="0" timeout="0.01" />
    <tout time="0.1" timeout="0.5" />
    <tout time="0.2" timeout="0.001" />
    <tout time="0.3" timeout="0.01" />
</timeout>
```

EXECUTION-PARAMETERS

Parameters for execution in DualSPHysics

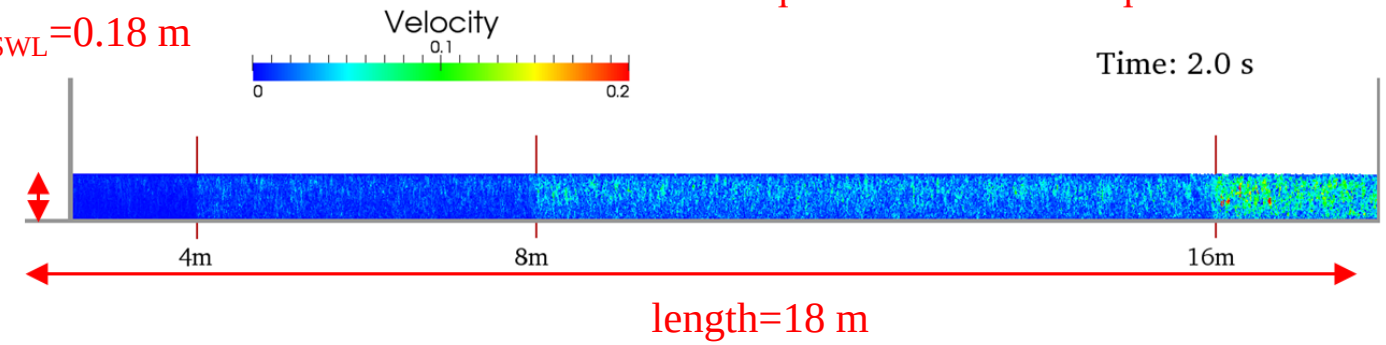
```
<parameters>
    <parameter key="SavePosDouble" value="0" comment="Saves particle position using double precision (default=0)" />
    <parameter key="StepAlgorithm" value="1" comment="Step Algorithm 1:Verlet, 2:Symplectic (default=1)" />
    <parameter key="VerletSteps" value="40" comment="Verlet only: Number of steps to apply Euler timestepping (default=40)" />
    <parameter key="Kernel" value="1" comment="Interaction Kernel 1:Cubic Spline, 2:Wendland (default=2)" />
    <parameter key="ViscoTreatment" value="1" comment="Viscosity formulation 1:Artificial, 2:Laminar+SPS (default=1)" />
    <parameter key="Visco" value="0.1" comment="Viscosity value" />
    <parameter key="ViscoBoundFactor" value="1" comment="Multiply viscosity value with boundary (default=1)" />
    <parameter key="DensityDT" value="2" comment="Density Diffusion Term 0:None, 1:Molteni, 2:Fourtakas, 3:Fourtakas(full) (default=0)" />
    <parameter key="DensityDTvalue" value="0.1" comment="DDT value (default=0.1)" />
    <parameter key="#Shifting" value="0" comment="Shifting mode 0:None, 1:Ignore bound, 2:Ignore fixed, 3:Full (default=0)" />
    <parameter key="#ShiftCoef" value="-2" comment="Coefficient for shifting computation (default=-2)" />
    <parameter key="#ShiftTFS" value="1.5" comment="Threshold to detect free surface. Typically 1.5 for 2D and 2.75 for 3D (default=0)" />
    <parameter key="RigidAlgorithm" value="1" comment="Rigid Algorithm 1:SPH, 2:DEM , 3:CHRONO (default=1)" />
    <parameter key="FtPause" value="0.0" comment="Time to freeze the floatings at simulation start (warmup) (default=0)" units_comment="seconds" />
    <parameter key="CoefDtMin" value="0.05" comment="Coefficient to calculate minimum time step dtmin=coefdtmin*h/speedsound (default=0.05)" />
    <parameter key="#DtIni" value="0.0001" comment="Initial time step (default=h/speedsound)" units_comment="seconds" />
    <parameter key="#DtMin" value="0.00001" comment="Minimum time step (default=coefdtmin*h/speedsound)" units_comment="seconds" />
    <parameter key="#DtFixed" value="DtFixed.dat" comment="Dt values are loaded from file (default=disabled)" />
    <parameter key="DtAllParticles" value="0" comment="Velocity of particles used to calculate DT. 1:All, 0:Only fluid/floating (default=0)" />
    <parameter key="TimeMax" value="1.6" comment="Time of simulation" units_comment="seconds" />
    <parameter key="TimeOut" value="0.01" comment="Time out data" units_comment="seconds" />
    <parameter key="PartsOutMax" value="1" comment="%/100 of fluid particles allowed to be excluded from domain (default=1)" units_comment="decimal" />
    <parameter key="RhopOutMin" value="700" comment="Minimum rhop valid (default=700)" units_comment="kg/m^3" />
    <parameter key="RhopOutMax" value="1300" comment="Maximum rhop valid (default=1300)" units_comment="kg/m^3" />
    <simulationdomain comment="Defines domain of simulation (default=Uses minimun and maximum position of the generated particles)">
        <posmin x="default" y="default" z="default" comment="e.g.: x=0.5, y=default-1, z=default-10%" />
        <posmax x="default" y="default" z="default + 50%" />
    </simulationdomain>
</parameters>
```

EXECUTION-PARAMETERS

Double precision

distance of particle interaction: dp= 0.012374m

depth: h_{SWL}=0.18 m



length=18 m

The problems of precision mainly appear when the domain is huge in comparison to the distance of interaction between particles

length>depth>>dp

```
<parameter key="SavePosDouble" value="0"
comment="Saves particle position using double precision (default=0)" />
```

The code uses double precision for **position** and to update magnitudes.
- GPU implementation includes an auxiliary array with position as function of the cell
- CPU implementation solves position in double precision

The user can choose whether or not to save position in double precision in PART_XXX.bi4 files

Time integrator scheme

```
<parameter   key="StepAlgorithm" value="1"
             comment="Step Algorithm 1:Verlet, 2:Symplectic (default=1)" />
<parameter   key="VerletSteps" value="40"
             comment="Verlet only: Number of steps to apply Euler timestepping (default=40)" />
```

Verlet algorithm

$$\boldsymbol{v}_a^{n+1} = \boldsymbol{v}_a^{n-1} + 2\Delta t \boldsymbol{F}_a^n$$
$$\boldsymbol{r}_a^{n+1} = \boldsymbol{r}_a^n + \Delta t \boldsymbol{V}_a^n + 0.5\Delta t^2 \boldsymbol{F}_a^n$$
$$\rho_a^{n+1} = \rho_a^{n-1} + 2\Delta t D_a^n$$

once every **M** time steps

$$\boldsymbol{v}_a^{n+1} = \boldsymbol{v}_a^n + \Delta t \boldsymbol{F}_a^n$$
$$\boldsymbol{r}_a^{n+1} = \boldsymbol{r}_a^n + \Delta t \boldsymbol{V}_a^n + 0.5\Delta t^2 \boldsymbol{F}_a^n$$
$$\rho_a^{n+1} = \rho_a^n + \Delta t D_a^n$$

```
<parameter   key="StepAlgorithm" value="2"
             comment="Step Algorithm 1:Verlet, 2:Symplectic (default=1)" />
```

Symplectic algorithm

$$\boldsymbol{r}_a^{n+\frac{1}{2}} = \boldsymbol{r}_a^n + \frac{\Delta t}{2}\boldsymbol{v}_a^n$$
$$\rho_a^{n+\frac{1}{2}} = \rho_a^n + \frac{\Delta t}{2} D_a^n$$

Predictor

$$\boldsymbol{v}_a^{n+1} = \boldsymbol{v}_a^{n+\frac{1}{2}} + \frac{\Delta t}{2}\boldsymbol{F}_a^{n+\frac{1}{2}}$$
$$\boldsymbol{r}_a^{n+1} = \boldsymbol{r}_a^{n+\frac{1}{2}} + \frac{\Delta t}{2}\boldsymbol{v}_a^{n+1}$$

Corrector

EXECUTION-PARAMETERS

Kernel function

$$\frac{d\boldsymbol{v}_a}{dt} = -\sum_b m_b \left(\frac{P_b + P_a}{\rho_b \cdot \rho_a} + \Pi_{ab} \right) \nabla_a W_{ab} + \boldsymbol{g}$$

```
<parameter key="Kernel" value="2"
           comment="Interaction Kernel 1:Cubic Spline, 2:Wendland (default=2)" />
```

Cubic Spline

$$W(r,h) = \alpha_D \begin{cases} 1 - \frac{3}{2}q^2 + \frac{3}{4}q^3 & 0 \le q \le 1 \\ \frac{1}{4}(2-q)^3 & 1 \le q \le 2 \\ 0 & q \ge 2 \end{cases}$$

where α_D is equal to $10/7\pi h^2$ in 2-D and $1/\pi h^3$ in 3-D

Wendland

$$W(r,h) = \alpha_D \left(1 - \frac{q}{2}\right)^4 (2q+1) \qquad 0 \le q \le 2$$

where α_D is equal to $7/4\pi h^2$ in 2-D and $21/16\pi h^3$ in 3-D

EXECUTION-PARAMETERS

Viscosity treatment

```
<parameter   key="ViscoTreatment" value="1"
             comment="Viscosity formulation 1:Artificial, 2:Laminar+SPS (default=1)" />
<parameter   key="Visco" value="0.02" comment="Viscosity value" />
<parameter   key="ViscoBoundFactor" value="1"
             comment="Multiply viscosity value with boundary (default=1)" />
```

$$\frac{d\boldsymbol{v}_a}{dt} = -\sum_b m_b \left(\frac{P_b + P_a}{\rho_b \cdot \rho_a} + \Pi_{ab} \right) \nabla_a W_{ab} + \boldsymbol{g}$$

$$\Pi_{ab} = \begin{cases} \dfrac{-\alpha \overline{c_{ab}} \mu_{ab}}{\overline{\rho_{ab}}} & \boldsymbol{v}_{ab} \cdot \boldsymbol{r}_{ab} < 0 \\ 0 & \boldsymbol{v}_{ab} \cdot \boldsymbol{r}_{ab} > 0 \end{cases}$$

α=0.01 for wave tanks

higher values of α for dam-break (depends on *dp*)

EXECUTION-PARAMETERS

Viscosity treatment

```
<parameter   key="ViscoTreatment" value="1"
             comment="Viscosity formulation 1:Artificial, 2:Laminar+SPS (default=1)" />
<parameter   key="Visco" value="0.02" comment="Viscosity value" />
<parameter   key="ViscoBoundFactor" value="1"
             comment="Multiply viscosity value with boundary (default=1)" />
```

$$\frac{d\boldsymbol{v}_a}{dt} = -\sum_b m_b \left(\frac{P_b + P_a}{\rho_b \cdot \rho_a} + \Pi_{ab} \right) \nabla_a W_{ab} + \boldsymbol{g}$$

$$\Pi_{ab} = \begin{cases} \dfrac{-\alpha \overline{c_{ab}} \mu_{ab}}{\overline{\rho_{ab}}} & \boldsymbol{v}_{ab} \cdot \boldsymbol{r}_{ab} < 0 \\ 0 & \boldsymbol{v}_{ab} \cdot \boldsymbol{r}_{ab} > 0 \end{cases}$$

$\boldsymbol{\alpha_{FF}}$ *for interaction fluid-fluid*

$\boldsymbol{\alpha_{FB}}$ *for interaction fluid-boundary*

$\boldsymbol{\alpha_{FB}}$= ViscoBoundFactor · $\boldsymbol{\alpha_{FF}}$

Viscosity treatment

```
<parameter   key="ViscoTreatment" value="2"
             comment="Viscosity formulation 1:Artificial, 2:Laminar+SPS (default=1)" />
<parameter   key="Visco" value="0.000001" comment="Viscosity value" units_comment="m^2/s" />
```

$$\frac{d\boldsymbol{v}_a}{dt} = -\sum_b m_b\left(\frac{P_b}{\rho_b^2}+\frac{P_a}{\rho_a^2}\right)\nabla_a W_{ab} + \boldsymbol{g} + \sum_b m_b\left(\frac{4\upsilon_0 r_{ab}\cdot\nabla_a W_{ab}}{(\rho_a+\rho_b)(r_{ab}^2+\eta^2)}\right)\boldsymbol{v}_{ab} + \sum_b m_b\left(\frac{\vec{\tau}_{ij}^{\,b}}{\rho_b^2}+\frac{\vec{\tau}_{ij}^{\,a}}{\rho_a^2}\right)\nabla_a W_{ab}$$

$\boldsymbol{\upsilon_o}$ is kinematic viscosity (typically 10^{-6} m^2s for water

Density diffusion formulation

```
<parameter key="DensityDT" value="2"
comment="Density Diffusion Term 0:None, 1:Molteni, 2:Fourtakas, 3:Fourtakas(full) (default=0)" />

<parameter key="DensityDTvalue" value="0.1"
comment="DDT value (default=0.1)" />
```

1:Molteni

Molteni and Colagrossi, 2009:

$$\frac{d\rho_i}{dt} = \sum_j^N m_j \mathbf{u}_{ij} \cdot \nabla W_{ij} + \delta h c_i \sum_j^N \psi_{ij} \cdot \nabla W_{ij} V_j \qquad \psi_{ij} = 2\left(\rho_j - \rho_i\right) \frac{\mathbf{x}_{ij}}{\left\|\mathbf{x}_{ij}\right\|^2}$$

2:Fourtakas

3:Fourtakas(full)

2: Fourtakas is applied to fluid particles that do not interact with boundaries
3: Fourtakas (full) is applied to all the fluid particles

Fourtakas et al, 2019:

$$\frac{d\rho_i}{dt} = \sum_j^N m_j \mathbf{u}_{ij} \cdot \nabla W_{ij} + \delta h c_i \sum_j^N \psi_{ij} \cdot \nabla W_{ij} V_j \qquad \psi_{ij} = 2\left(\rho_i^D - \rho_j^D\right) \frac{\mathbf{x}_{ij}}{\left\|\mathbf{x}_{ij}\right\|^2}$$

with, $y_{ij} = 2\left(r_{ji}^{\mathsf{T}} - r_{ij}^{\mathsf{H}}\right) \frac{\mathbf{x}_{ij}}{\left\|\mathbf{x}_{ij}\right\|^2}$ $\qquad \rho_{ij}^{\mathrm{H}} = \rho_0 \left(\sqrt[\gamma]{\frac{P_{ij}^{\mathrm{H}} + 1}{C_B}} - 1 \right) \rightarrow \quad P_{ij}^{\mathrm{H}} = \rho_0 g z_{ij}$

- Hydrostatic pressure is calculated locally
- Dependence on gravity (applicable to gravity driven flows)
- Applicable to BCs with severe kernel truncation

XML file

EXECUTION-PARAMETERS

Shifting algorithm

```
<parameter   key="Shifting" value="2"
             comment="Shifting mode 0:None, 1:Ignore bound, 2:Ignore fixed, 3:Full (default=0)" />
<parameter   key="ShiftCoef" value="-2"
             comment="Coefficient for shifting computation (default=-2)" />
<parameter   key="ShiftTFS" value="1.5"
             comment="Threshold to detect free surface. Typically 1.5 for 2D and 2.75 for 3D" />
```

$$\delta\mathbf{r} = D \cdot \nabla C_i$$

$$D = A\, h \left\|\mathbf{u}\right\|_i \Delta t$$

Shifting update

$$\nabla C_i = \sum_j \frac{m_j}{\rho_j} \nabla W_{ij}$$

Concentration gradient

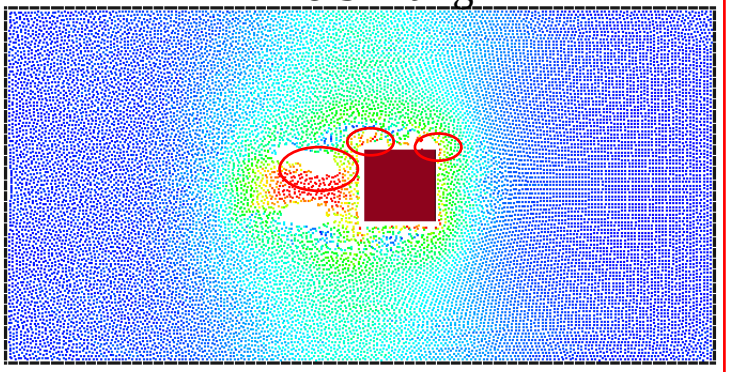
SHIFTING
SHIFTING IN THE NORMAL DIRECTION
IS NOT APPLIED FOR PARTICLES
AT THE FREE SURFACE

$$\nabla \cdot \mathbf{r} > 1.5$$

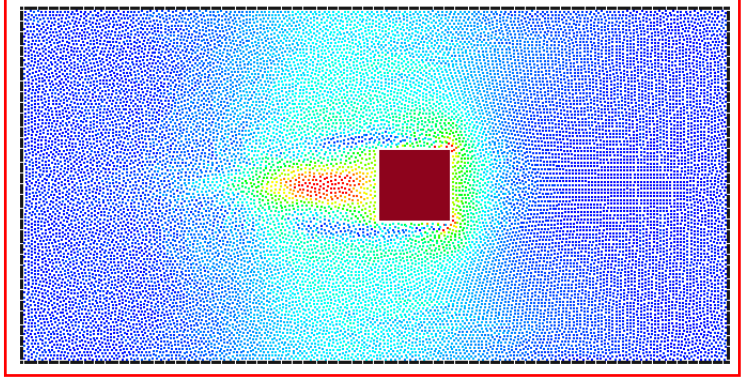
Particle divergence

$$\nabla \cdot \mathbf{r} = \sum_j \frac{m_j}{\rho_j} \mathbf{r}_{ij} \cdot \nabla_i W_{ij}$$

No Shifting



Shifting that ignores fixed boundaries



EXECUTION-PARAMETERS

Interaction between solids

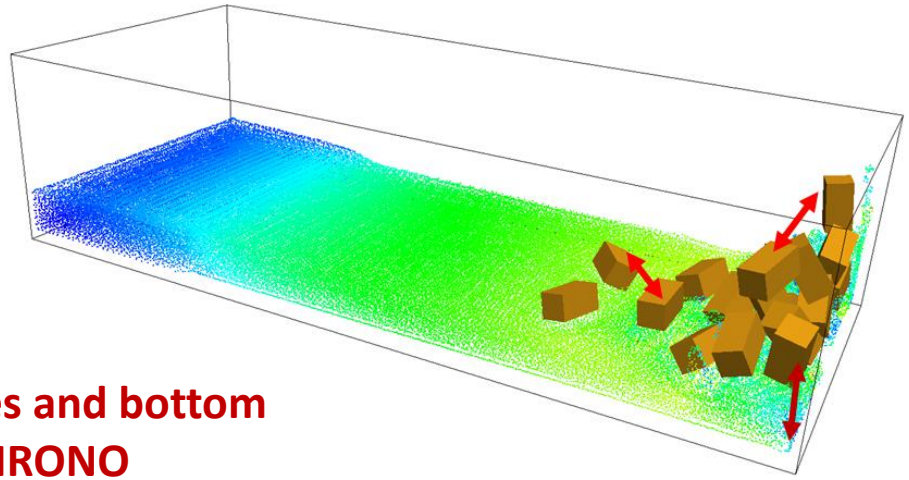
```
<parameter key="RigidAlgorithm" value="1"
           comment="Rigid Algorithm 0:collision-free, 1:SPH, 2:DEM, 3:Chrono (default=1)" />
```

0: boundary elements will not interact at all

1: SPH interactions are allowed between boundary objects

2: solid interactions are solved using DEM (Discrete Element Method)

3: solid interaction are solved using the contact methods of Project Chrono



Interaction between floating boxes and bottom
NONE or SPH or DEM or CHRONO

EXECUTION-PARAMETERS

Time step computation

```
<parameter key="CoefDtMin" value="0.05"
  comment="Coefficient to calculate minimum time step dtmin=coefdtmin*h/speedsound (default=0.05)" />
<parameter key="DtIni" value="0"
  comment="Initial time step. Use 0 to defult use (default=h/speedsound)" units_comment="seconds" />
<parameter key="DtMin" value="0"
  comment="Minimum time step. Use 0 to defult use (default=coefdtmin*h/speedsound)" units_comment="seconds" />
<parameter key="DtFixed" value="0"
  comment="Fixed Dt value. Use 0 to disable (default=disabled)" units_comment="seconds" />
<parameter key="DtFixedFile" value="NONE"
  comment="Dt values are loaded from file. Use NONE to disable (default=disabled)" units_comment="milliseconds"/>
```

$$\Delta t_{minimum} = 0.05 \cdot \frac{h}{c_s}$$

$$\Delta t_{init} = \frac{h}{c_s}$$

```
<parameter key="DtAllParticles"
          comment="Velocity of particles used to calculate DT.
                   1:All, 0:Only fluid/floating (default=0)" />
```

$$\Delta t = 0.3 \cdot \min\left(\Delta t_f, \Delta t_{cv}\right)$$

$$\Delta t_f = \min_a \left(\sqrt{h/|f_a|}\right)$$

$$\Delta t_{cv} = \min_a \frac{h}{c_s + \max_b \left| \frac{h \mathbf{v}_{ab} \cdot \mathbf{r}_{ab}}{\mathbf{r}_{ab}^2} \right|}$$

0: $b \in$ fluid/floating
OR
1: $b \in$ fluid/floating + boundaries

EXECUTION-PARAMETERS

Physical time and frequency to store data

```
<parameter   key="TimeMax" value="10"
             comment="Time of simulation" units_comment="seconds" />
<parameter   key="TimeOut" value="0.1"
             comment="Time out data" units_comment="seconds" />
```

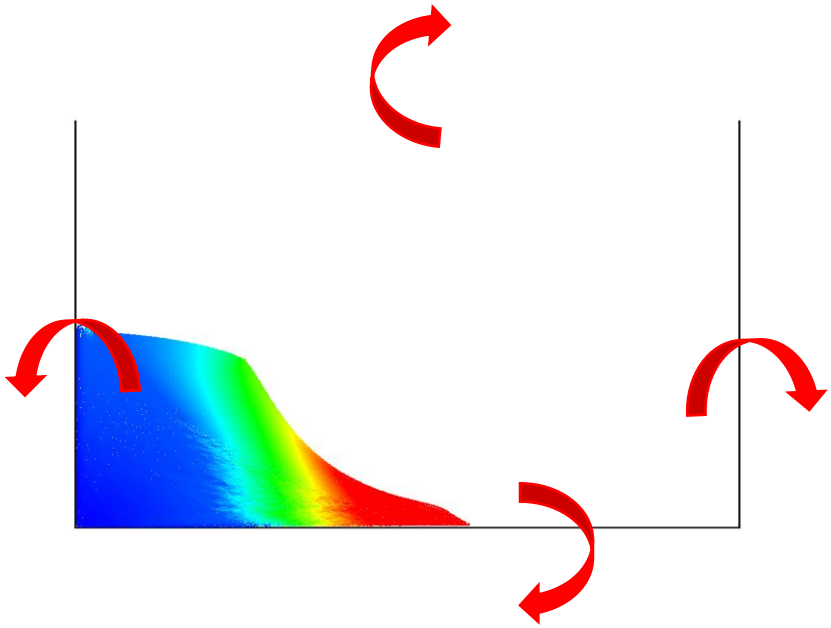
Number of output files = TimeMax / TimeOut
= 10 / 0.1 = 100 files

TimeMax can be changed during the simulation execution by creating a file named TERMINATE in the output directory. The value in **TERMINATE file** replaces the current TimeMax. If TERMINATE is empty or its value is lower than current time of the simulation then TimeMax changes to current simulation time.
TERMINATE file is checked at the end of each PART.

EXECUTION-PARAMETERS


```
<parameter   key="PartsOutMax" value="1"
             comment="%/100 of fluid particles allowed to be excluded
             from domain before termination of simulation (default=1)"
             units_comment="decimal" />
```

Excluding particles by position



```
<parameter   key="RhopOutMin" value="700"
             comment="Minimum rhop valid (default=700)" units_comment="kg/m^3" />
<parameter   key="RhopOutMax" value="1300"
             comment="Maximum rhop valid (default=1300)" units_comment="kg/m^3" />
```

Excluding particles by density

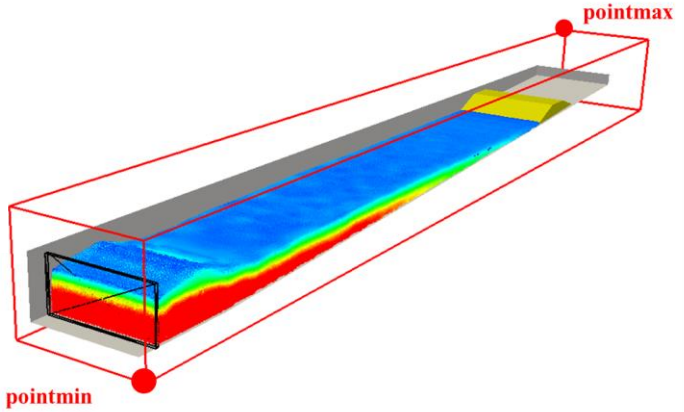
$$700 < \rho_0 < 1300$$

EXECUTION-PARAMETERS

Simulationdomain

```
<!--DEFINITION OF DOMAIN WHERE PARTICLES WILL BE CREATED -->
<definition dp="0.005">
    <pointmin x="-0.05" y="0.1" z="-0.05" />
    <pointmax x=" 2.00" y="0.1" z=" 1.00" />
</definition>
```

pointmin & pointmax defines the dimensions of the domain where particles can be created



NOTE that once particles have been created the dimensions of the domain for simulation are calculated again starting from minimum and maximum positions of the created particles.

BUT this simulation domain can be also configured in <parameters>

```
<simulationdomain comment="Defines domain of simulation ">
    <posmin x="0.5" y="default - 2" z="default - 20%" />
    <posmax x="default" y="default + 1" z="default + 10%" />
</simulationdomain>
```

default: value calculated starting from positions of the created particles

default - 2: default value MINUS 2 meters

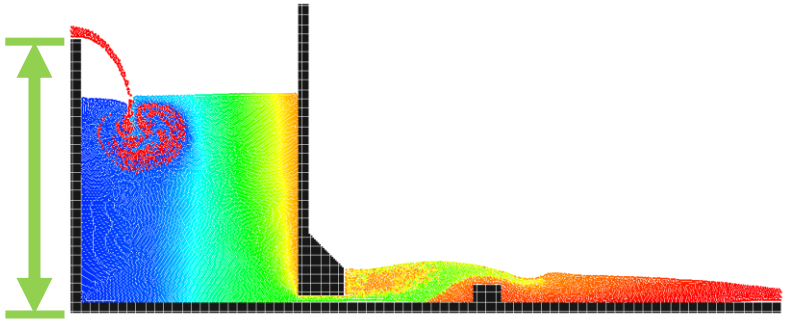
default + 10%: default value PLUS 10% of the size in that direction

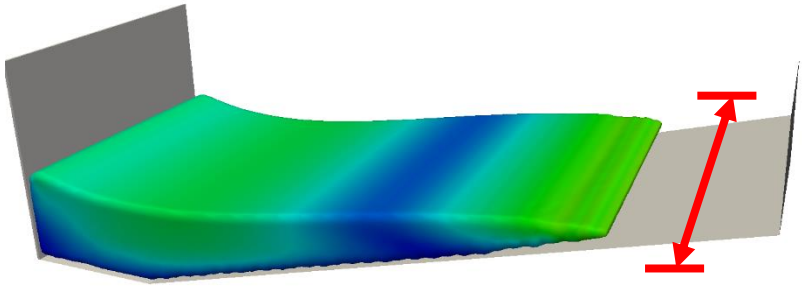
EXECUTION-PARAMETERS

Periodicity

```
<parameter   key="XPeriodicIncZ" value="0.3"
             comment="Increase of Z with periodic BC" units_comment="metres (m)" />
<parameter   key="YPeriodicIncZ" value="0.0"
             comment="Increase of Z with periodic BC" units_comment="metres (m)" />
```

Δz=0.3 m





Δz=0 m

XML file


```xml
<case>
    <casedef>
        <constantsdef>
            <lattice bound="1" fluid="1" />
            <gravity x="0" y="0" z="-9.81" comment="Gravitational acceleration" units_comment="m/s^2" />
            <cflnumber value="0.2" comment="Coefficient to multiply Dt" />
            <hswl value="0" auto="true" comment="Maximum still water level to calculate speedofsound using coefsound" units_comment="metres (m)" />
            <speedsystem value="0" auto="true" comment="Maximum system speed (by default the dam-break propagation is used)" />
            <coefsound value="20" comment="Coefficient to multiply speedsystem" />
            <speedsound value="0" auto="true" comment="Speed of sound to use in the simulation (by default speedofsound=coefsound*speedsystem)" />
            <coefh value="1.0" comment="Coefficient to calculate the smoothing length (H=coefficient*sqrt(3*dp^2) in 3D)" />
            <gamma value="7" comment="Politropic constant for water used in the state equation" />
            <rhop0 value="1000" comment="Reference density of the fluid" units_comment="kg/m3" />
        </constantsdef>
        <mkconfig boundcount="240" fluidcount="10" />
        <geometry>
            <definition dp="0.01" units_comment="metres (m)">
                <pointmin x="-1" y="0" z="-1" />
                <pointmax x="4.5" y="0" z="3.5" />
            </definition>
            <commands>
                <mainlist>
                    <setdrawmode mode="full" />
                    <setmkfluid mk="0" />
                    <drawbox>
                        <boxfill>solid</boxfill>
                        <point x="0" y="-1" z="0" />
                        <size x="1" y="2" z="2" />
                    </drawbox>
                    <setmkbound mk="0" />
                    <drawbox>
                        <boxfill>bottom | left | right | front | back</boxfill>
                        <point x="0" y="-1" z="0" />
                        <size x="4" y="2" z="3" />
                    </drawbox>
                </mainlist>
            </commands>
        </geometry>
    </casedef>
    <execution>
        <parameters>
            <parameter key="StepAlgorithm" value="1" comment="Step Algorithm 1:Verlet, 2:Symplectic (default=1)" />
            <parameter key="VerletSteps" value="40" comment="Verlet only: Number of steps to apply Euler timestepping (default=40)" />
            <parameter key="Kernel" value="2" comment="Interaction Kernel 1:Cubic Spline, 2:Wendland (default=2)" />
            <parameter key="ViscoTreatment" value="1" comment="Viscosity formulation 1:Artificial, 2:Laminar+SPS (default=1)" />
            <parameter key="Visco" value="0.02" comment="Viscosity value" />
            <parameter key="ViscoBoundFactor" value="1" comment="Multiply viscosity value with boundary (default=1)" />
            <parameter key="DeltaSPH" value="0" comment="DeltaSPH value, 0.1 is the typical value, with 0 disabled (default=0)" />
            <parameter key="#Shifting" value="0" comment="Shifting mode 0:None, 1:Ignore bound, 2:Ignore fixed, 3:Full (default=0)" />
            <parameter key="#ShiftCoef" value="-2" comment="Coefficient for shifting computation (default=-2)" />
            <parameter key="#ShiftTFS" value="1.5" comment="Threshold to detect free surface. Typically 1.5 for 2D and 2.75 for 3D (default=0)" />
            <parameter key="RigidAlgorithm" value="1" comment="Rigid Algorithm 1:SPH, 2:DEM (default=1)" />
            <parameter key="FtPause" value="0.0" comment="Time to freeze the floatings at simulation start (warmup) (default=0)" units_comment="seconds" />
            <parameter key="CoefDtMin" value="0.05" comment="Coefficient to calculate minimum time step dtmin=coefdtmin*h/speedsound (default=0.05)" />
            <parameter key="#DtIni" value="0.0001" comment="Initial time step (default=h/speedsound" units_comment="seconds" />
            <parameter key="#DtMin" value="0.00001" comment="Minimum time step (default=coefdtmin*h/speedsound)" units_comment="seconds" />
            <parameter key="#DtFixed" value="DtFixed.dat" comment="Dt values are loaded from file (default=disabled)" />
            <parameter key="DtAllParticles" value="0" comment="Velocity of particles used to calculate DT. 1:All, 0:Only fluid/floating (default=0)" />
            <parameter key="TimeMax" value="0.72" comment="Time of simulation" units_comment="seconds" />
            <parameter key="TimeOut" value="0.01" comment="Time out data" units_comment="seconds" />
            <parameter key="IncZ" value="1" comment="Increase of Z+" units_comment="decimal" />
            <parameter key="PartsOutMax" value="1" comment="Allowed %/100 of fluid particles out the domain (default=1)" units_comment="decimal" />
            <parameter key="RhopOutMin" value="700" comment="Minimum rhop valid (default=700)" units_comment="kg/m3" />
            <parameter key="RhopOutMax" value="1300" comment="Maximum rhop valid (default=1300)" units_comment="kg/m3" />
        </parameters>
    </execution>
</case>
```

```xml
<case>
    <casedef>
        <constantsdef>
            <lattice bound="1" fluid="1" />
            <gravity x="0" y="0" z="-9.81" comment="Gravitational acceleration" units_comment="m/s^2" />
            <cflnumber value="0.2" comment="Coefficient to multiply Dt" />
            <hswl value="0" auto="true" comment="Maximum still water level to calculate speedofsound using coefsound" units_comment="metres (m)" />
            <speedsystem value="0" auto="true" comment="Maximum system speed (by default the dam-break propagation is used)" />
            <coefsound value="20" comment="Coefficient to multiply speedsystem" />
            <speedsound value="0" auto="true" comment="Speed of sound to use in the simulation (by default speedofsound=coefsound*speedsystem)" />
            <coefh value="1.0" comment="Coefficient to calculate the smoothing length (H=coefficient*sqrt(3*dp^2) in 3D)" />
            <gamma value="7" comment="Politropic constant for water used in the state equation" />
            <rhop0 value="1000" comment="Reference density of the fluid" units_comment="kg/m3" />
        </constantsdef>
        <mkconfig boundcount="240" fluidcount="10" />
        <geometry>
            <definition dp="0.01" units_comment="metres (m)">
                <pointmin x="-1" y="0" z="-1" />
                <pointmax x="4.5" y="0" z="3.5" />
            </definition>
            <commands>
                <mainlist>
                    <setdrawmode mode="full" />
                    <setmkfluid mk="0" />
                    <drawbox>
                        <boxfill>solid</boxfill>
                        <point x="0" y="-1" z="0" />
                        <size x="1" y="2" z="2" />
                    </drawbox>
                    <setmkbound mk="0" />
                    <drawbox>
                        <boxfill>bottom | left | right | front | back</boxfill>
                        <point x="0" y="-1" z="0" />
                        <size x="4" y="2" z="3" />
                    </drawbox>
                </mainlist>
            </commands>
        </geometry>
    </casedef>
    <execution>
        <parameters>
            <parameter key="StepAlgorithm" value="1" comment="Step Algorithm 1:Verlet, 2:Symplectic (default=1)" />
            <parameter key="VerletSteps" value="40" comment="Verlet only: Number of steps to apply Euler timestepping (default=40)" />
            <parameter key="Kernel" value="2" comment="Interaction Kernel 1:Cubic Spline, 2:Wendland (default=2)" />
            <parameter key="ViscoTreatment" value="1" comment="Viscosity formulation 1:Artificial, 2:Laminar+SPS (default=1)" />
            <parameter key="Visco" value="0.02" comment="Viscosity value" />
            <parameter key="ViscoBoundFactor" value="1" comment="Multiply viscosity value with boundary (default=1)" />
            <parameter key="DeltaSPH" value="0" comment="DeltaSPH value, 0.1 is the typical value, with 0 disabled (default=0)" />
            <parameter key="#Shifting" value="0" comment="Shifting mode 0:None, 1:Ignore bound, 2:Ignore fixed, 3:Full (default=0)" />
            <parameter key="#ShiftCoef" value="-2" comment="Coefficient for shifting computation (default=-2)" />
            <parameter key="#ShiftTFS" value="1.5" comment="Threshold to detect free surface. Typically 1.5 for 2D and 2.75 for 3D (default=0)" />
            <parameter key="RigidAlgorithm" value="1" comment="Rigid Algorithm 1:SPH, 2:DEM (default=1)" />
            <parameter key="FtPause" value="0.0" comment="Time to freeze the floatings at simulation start (warmup) (default=0)" units_comment="seconds" />
            <parameter key="CoefDtMin" value="0.05" comment="Coefficient to calculate minimum time step dtmin=coefdtmin*h/speedsound (default=0.05)" />
            <parameter key="#DtIni" value="0.0001" comment="Initial time step (default=h/speedsound" units_comment="seconds" />
            <parameter key="#DtMin" value="0.00001" comment="Minimum time step (default=coefdtmin*h/speedsound)" units_comment="seconds" />
            <parameter key="#DtFixed" value="DtFixed.dat" comment="Dt values are loaded from file (default=disabled)" />
            <parameter key="DtAllParticles" value="0" comment="Velocity of particles used to calculate DT. 1:All, 0:Only fluid/floating (default=0)" />
            <parameter key="TimeMax" value="0.72" comment="Time of simulation" units_comment="seconds" />
            <parameter key="TimeOut" value="0.01" comment="Time out data" units_comment="seconds" />
            <parameter key="IncZ" value="1" comment="Increase of Z+" units_comment="decimal" />
            <parameter key="PartsOutMax" value="1" comment="Allowed %/100 of fluid particles out the domain (default=1)" units_comment="decimal" />
            <parameter key="RhopOutMin" value="700" comment="Minimum rhop valid (default=700)" units_comment="kg/m3" />
            <parameter key="RhopOutMax" value="1300" comment="Maximum rhop valid (default=1300)" units_comment="kg/m3" />
        </parameters>
    </execution>
    <particles np="21001" nb="1001" nbf="1001" mkboundfirst="11" mkfluidfirst="1">
        <fixed mkbound="0" mk="11" begin="0" count="1001" />
        <fluid mkfluid="0" mk="1" begin="1001" count="20000" />
    </particles>
    <constants>
        <gravity x="0" y="0" z="-9.81" units_comment="m/s^2" />
        <cflnumber value="0.2" />
        <gamma value="7" />
        <rhop0 value="1000" units_comment="kg/m3" />
        <dp value="0.01" units_comment="metres (m)" />
        <h value="1.4142135624E-002" units_comment="metres (m)" />
        <b value="1.1155371429E+006" units_comment="metres (m)" />
        <massbound value="1.0000000000E-001" units_comment="kg" />
        <massfluid value="1.0000000000E-001" units_comment="kg" />
    </constants>
    <motion />
</case>
```

XML file

Info of the created particles

np=total number of particles
nb=boundary particles
nbf=fixed boundary particles

```
<particles np="8079" nb="1795" nbf="1659" mkboundfirst="10" mkfluidfirst="1">
    <_summary>
        <positions units_comment="metres (m)">
            <posmin x="-0.5" y="0" z="-0.06" />
            <posmax x="16.006" y="0" z="1.01" />
        </positions>
        <fixed count="1659" id="0-1658" mkcount="1" mkvalues="10" />
        <moving count="136" id="1659-1794" mkcount="1" mkvalues="20" />
        <fluid count="6284" id="1795-8078" mkcount="1" mkvalues="1" />
    </_summary>
    <fixed mkbound="0" mk="10" begin="0" count="1659" />
    <moving mkbound="10" mk="20" begin="1659" count="136" refmotion="0" />
    <fluid mkfluid="0" mk="1" begin="1795" count="6284" />
</particles>
```

NOTE value of final "mk"
mk=mkbound+1+fluidcount(9)
mk=mkfluid+1

YOU SHOULD ALWAYS CHECK
Case_All.vtk, Case_Bound.vtk, Case_Fluid.vtk

XML file

Summary of the computed constants

$$h = coefh \cdot \sqrt{3} \cdot dp$$

$$B = \frac{c_s^2 \cdot \rho_0}{\gamma} = \frac{coef_{sound}{}^2 \cdot g \cdot h_{swl} \cdot \rho_0}{\gamma}$$

```
<constants>
    <data2d value="true" />
    <data2dposy value="0" units_comment="metres (m)" />
    <gravity x="0" y="0" z="-9.81" units_comment="m/s^2" />
    <cflnumber value="0.2" />
    <gamma value="7" />
    <rhop0 value="1000" units_comment="kg/m^3" />
    <dp value="0.03" units_comment="metres (m)" />
    <h value="5.0911688245E-02" units_comment="metres (m)" />
    <b value="3.3634285714E+05" units_comment="Pascal (Pa)" />
    <massbound value="9.0000000000E-01" units_comment="kg" />
    <massfluid value="9.0000000000E-01" units_comment="kg" />
</constants>
```

mass=*rhop0*dp*dp*dp* in 3D
mass=*rhop0*dp*dp* in 2D

Input & output files

Run.out

Text file with execution log

```
DualSPHysics v4 (10-11-2015)
================================
[Select CUDA Device]
Device 0: "GeForce GTX 590"
  Compute capability:        2.0
  Multiprocessors:           16 (512 cores)
  Memory global:             1536 MB
  Clock rate:                1.23 GHz
  Run time limit on kernels: Yes
  ECC support enabled:       No
Device 1: "GeForce 8400 GS"
  Compute capability:        1.1
  Multiprocessors:           1 (8 cores)
  Memory global:             512 MB
  Clock rate:                1.63 GHz
  Run time limit on kernels: Yes
  ECC support enabled:       No

[GPU Hardware]
Device default: 0  "GeForce GTX 590"
Compute capability: 2.0
Memory global: 1536 MB
Memory shared: 49152 Bytes
[Initialising JSphGpuSingle v0.70  24-11-2015 15:40:36]
**Basic case configuration is loaded
**Special case configuration is loaded
Loading initial state of particles...
Loaded particles: 5281
MapRealPos(border)=(-7.07107e-006,0.0999929,-7.07107e-006)-(1.60001,0.100007,0.400007)
MapRealPos(final)=(-7.07107e-006,0.0999929,-7.07107e-006)-(1.60001,0.100007,0.600014)
**Initial state of particles is loaded
**2D-Simulation parameters:
CaseName="CaseDambreak2D"
RunName="CaseDambreak2D"
SvDouble=False
SvTimers=True
SvTimersStep=0.000000
StepAlgorithm="Verlet"
VerletSteps=40
Kernel="Wendland"
Viscosity="Artificial"
Visco=0.300000
ViscoBoundFactor=0.000000
DeltaSph="None"
Shifting="None"
RenCorrection=0.000000
Splitting=False
FloatingFormulation="None"
FloatingCount=0
```

Domain dimensions computed starting from minimum and maximum positions of the particles created initially

Input & output files

Run.out

Text file with execution log

```
CaseNp=5281
CaseNbound=481
CaseNfixed=481
CaseNmoving=0
CaseNfloat=0
CaseNfluid=4800
PeriodicActive=0
Dx=0.005
H=0.007071
CoefficientH=1
CteB=165368.578125
Gamma=7.000000
RhopZero=1000.000000
Eps=0
Cs0=34.0232
CFLnumber=0.200000
DtIni=0.000207831
DtMin=1.03915e-005
DtAllParticles=False
MassFluid=0.025000
MassBound=0.025000
Bwen (wendland)=-7877736.000000
TimeMax=2
TimePart=0.02
Gravity=(0.000000,0.000000,-9.810000)
NpMinimum=481
RhopOut=True
RhopOutMin=700.000000
RhopOutMax=1300.000000
**Requested gpu memory for 5281 particles: 0.6 MB.
CellOrder="XYZ"
CellMode="2H"
Hdiv=1
MapCells=(114,1,43)
DomCells=(114,1,43)
DomCellCode="13_8_11"

PtxasFile="../../EXECS/DualSPHysics_win64_ptxasinfo"
Use code for compute capability 2.0 on hardware 2.0
BsForcesBound=128 (36 regs)
BsForcesFluid=128 (50 regs)

**CellDiv: Requested gpu memory for 5545 particles: 0.0 MB.
**CellDiv: Requested gpu memory for 1488 cells (CellMode=2H): 0.0 MB.
RunMode="Pos-Simple, Single-Gpu"
Allocated memory in CPU: 475290 (0.45 MB)
Allocated memory in GPU: 745752 (0.71 MB)
Part_0000        5281 particles successfully stored
```

Input & output files

Run.out

Text file with execution log

```
**CellDiv: Requested gpu memory for 5545 particles: 0.0 MB.
**CellDiv: Requested gpu memory for 1488 cells (CellMode=2H): 0.0 MB.
RunMode="Pos-Simple, Single-Gpu"
Allocated memory in CPU: 475290 (0.45 MB)
Allocated memory in GPU: 745752 (0.71 MB)
Part_0000        5281 particles successfully stored

[Initialising simulation (m15001k0)  24-11-2015 15:40:36]
PART       PartTime      TotalSteps    Steps    Time/Seg   Finish time
=========  ============  ============  =======  =========  ===================
Part_0001      0.020027           484      484      47.06  24-11-2015 15:42:10
Part_0002      0.040010           970      486      47.60  24-11-2015 15:42:10
Part_0003      0.060019          1460      490      48.01  24-11-2015 15:42:11
Part_0004      0.080034          1953      493      47.73  24-11-2015 15:42:11
Part_0005      0.100001          2447      494      48.86  24-11-2015 15:42:11
...
...
...
Part_0100      2.000038         49060      484      67.06  24-11-2015 15:42:14

[Simulation finished  24-11-2015 15:42:14]
Particles of simulation (initial): 5281
DTs adjusted to DtMin............: 0
Excluded particles...............: 0
Total Runtime....................: 98.489380 sec.
Simulation Runtime...............: 98.427155 sec.
Time per second of simulation....: 49.212650 sec.
Steps per second.................: 498.439697
Steps of simulation..............: 49060
PART files.......................: 101
Maximum number of particles......: 5281
Maximum number of cells..........: 2850
CPU Memory.......................: 475290 (0.45 MB)
GPU Memory.......................: 777144 (0.74 MB)

[GPU Timers]
VA-Init..........................: 0.036532 sec.
NL-Limits........................: 2.969780 sec.
NL-PreSort.......................: 0.258268 sec.
NL-RadixSort.....................: 33.559845 sec.
NL-CellBegin.....................: 4.946213 sec.
NL-SortData......................: 0.639390 sec.
NL-OutCheck......................: 0.114177 sec.
CF-PreForces.....................: 3.701347 sec.
CF-Forces........................: 11.354674 sec.
SU-ComputeStep...................: 0.961830 sec.
SU-Floating......................: 0.000000 sec.
SU-Motion........................: 0.000000 sec.
```

Input & output files