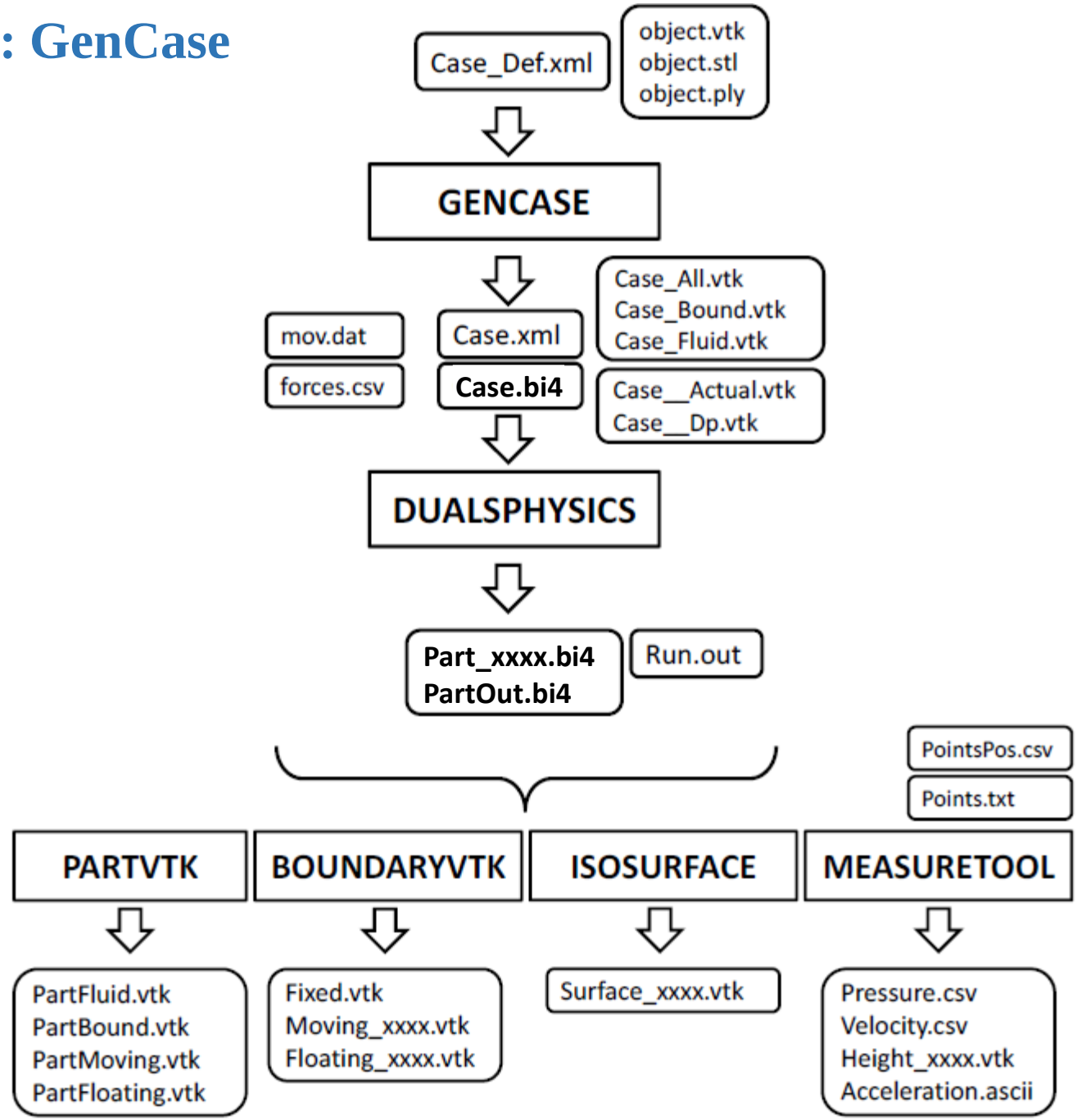
Pre-processing and User Friendly Interface

Dr Alejandro CRESPO
Universidade de Vigo
SPAIN

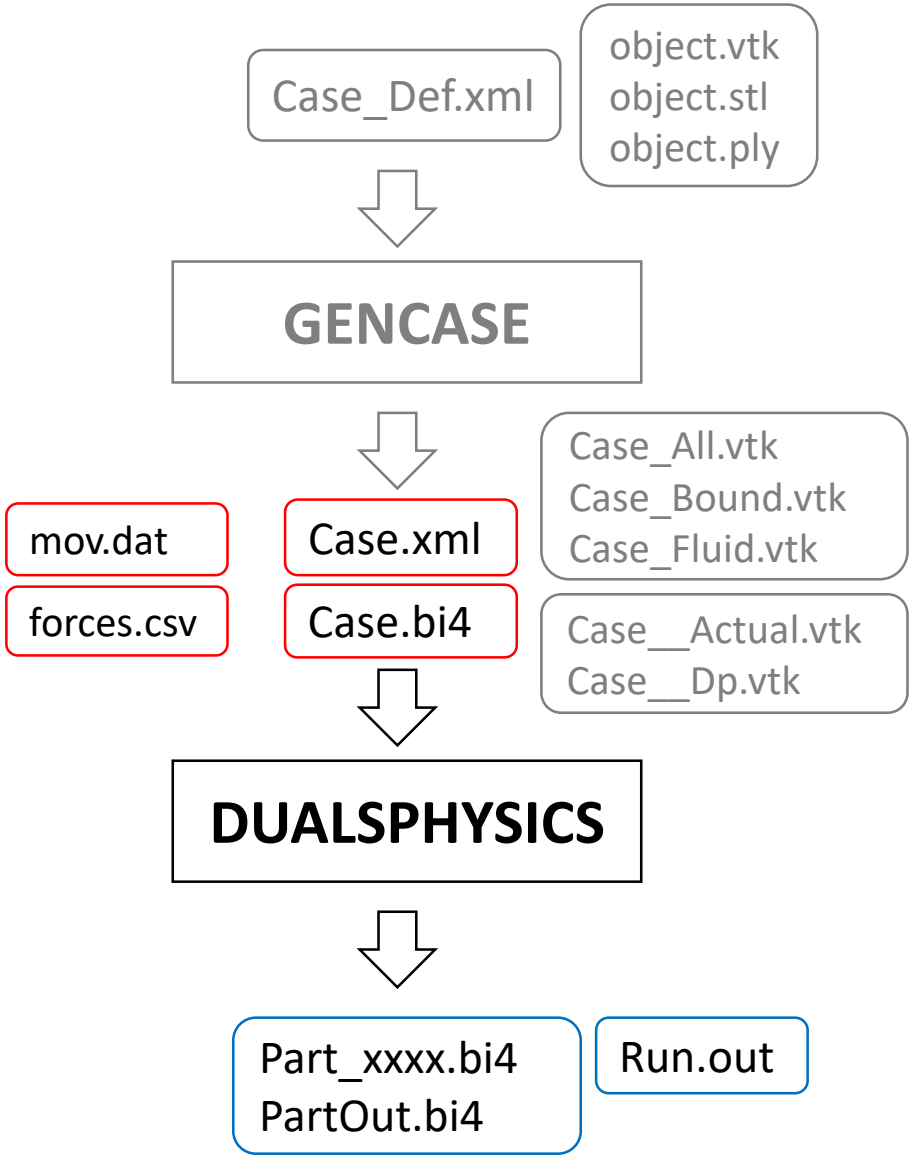
Outline of Presentation

- Pre-processing tool: GenCase
- XML input file
- DesignSPHysics
- Future developments

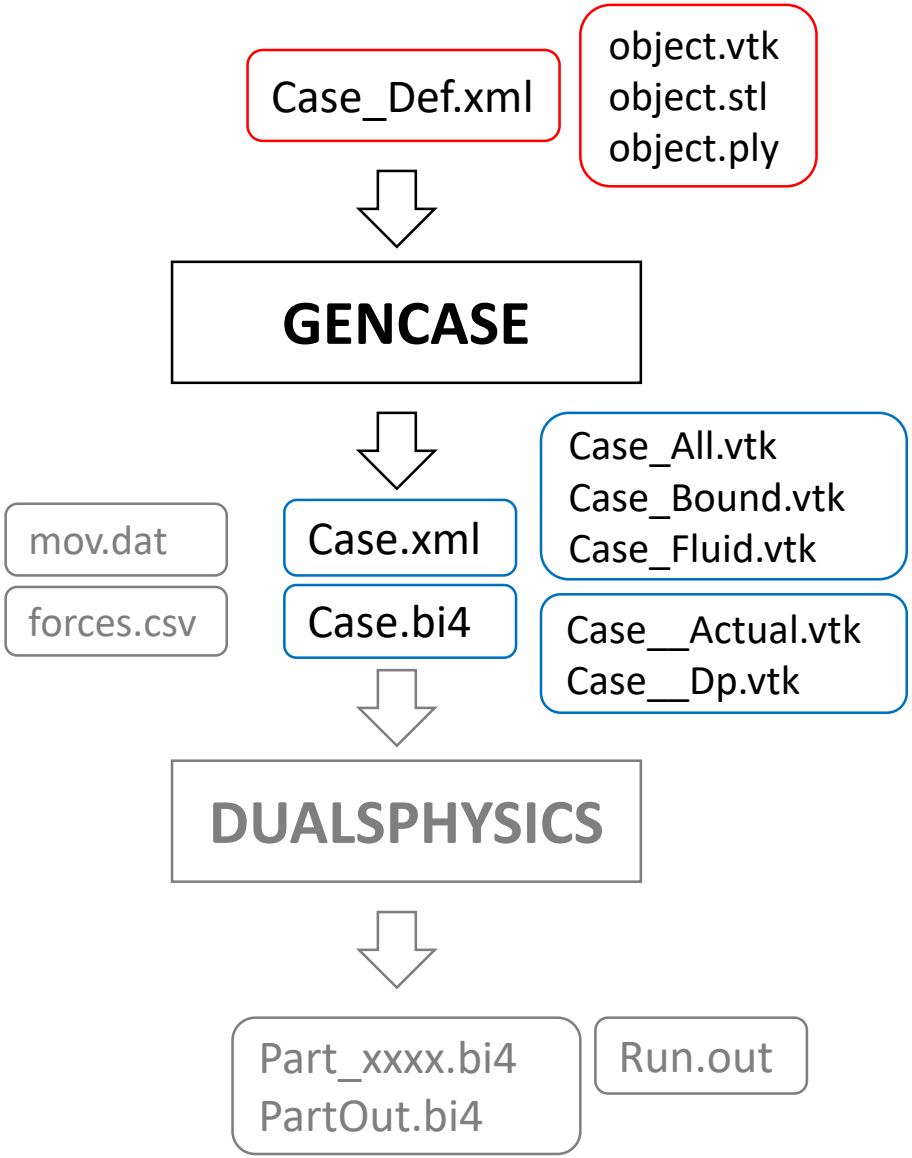
Pre-processing: GenCase



Pre-processing: GenCase



Pre-processing: GenCase

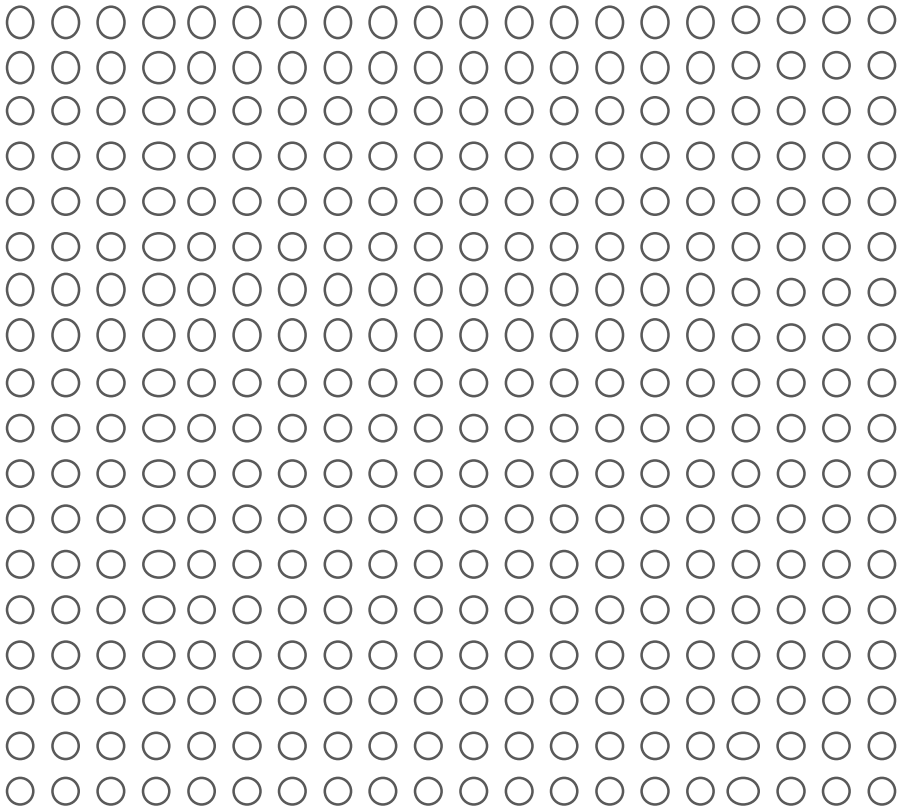


Draw points

- **GenCase** is a drawing application that creates **points** that will be converted **into particles**.
- It employs a **3D lattice to locate points** which represents possible positions of the particles.

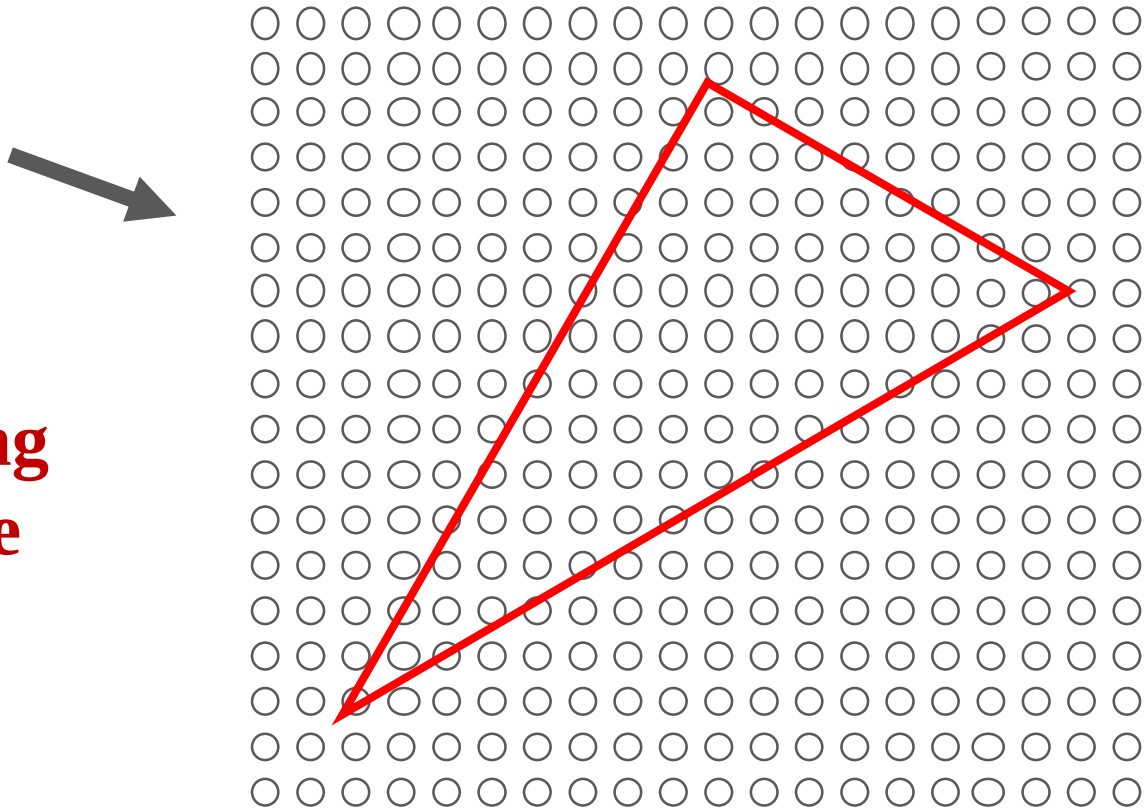
Draw points

Lattice



Draw points

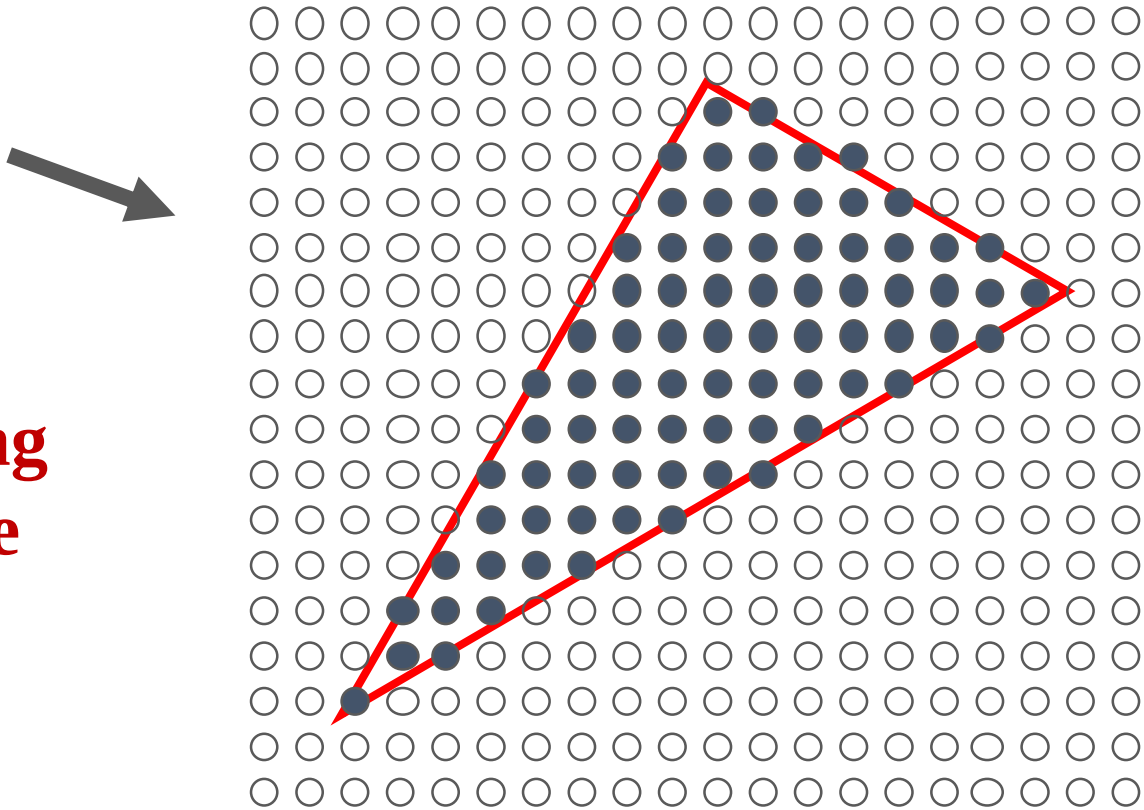
Lattice



Drawing triangle

Draw points

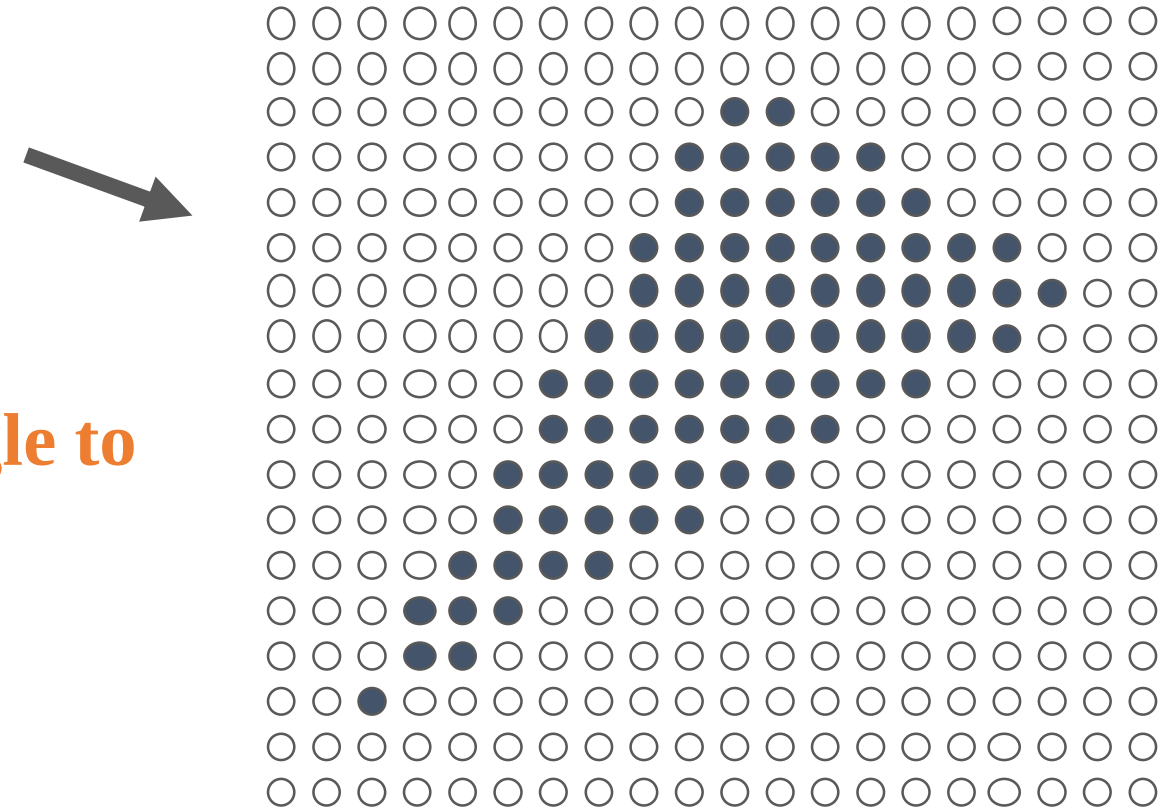
Lattice



Drawing triangle

Draw points

Lattice



Triangle to points

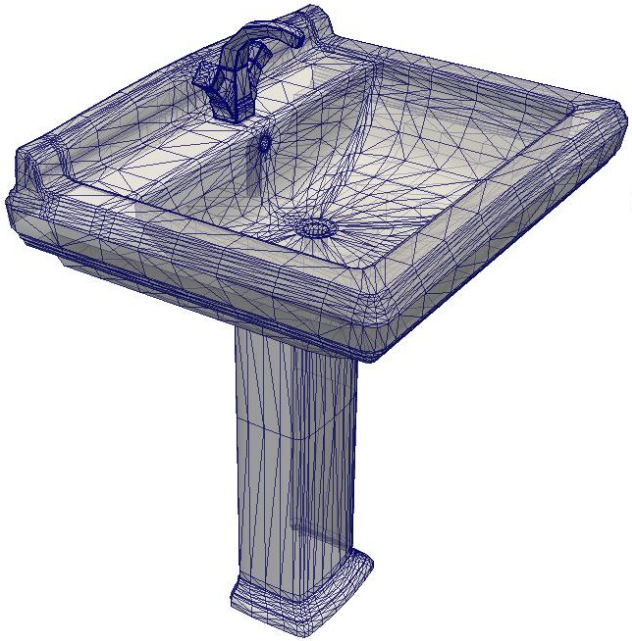
Draw points

Lattice

Triangle to points

Draw points

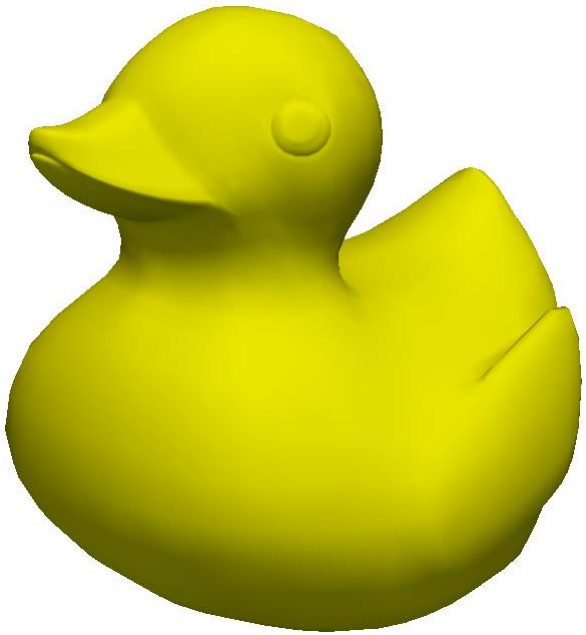
3D model



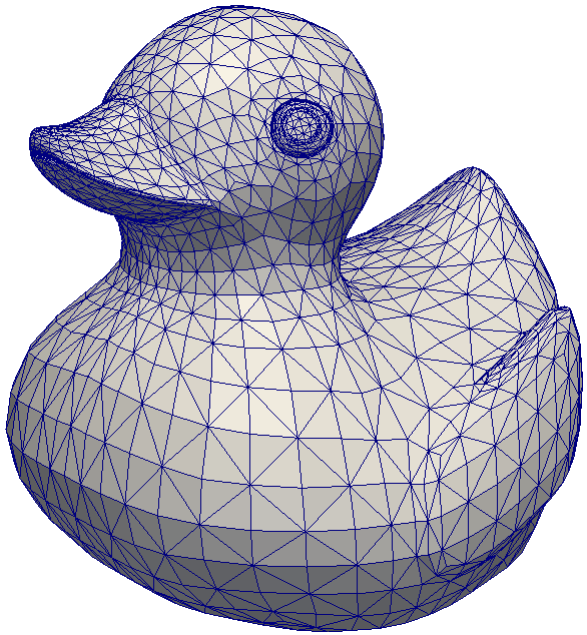
Triangles

Points

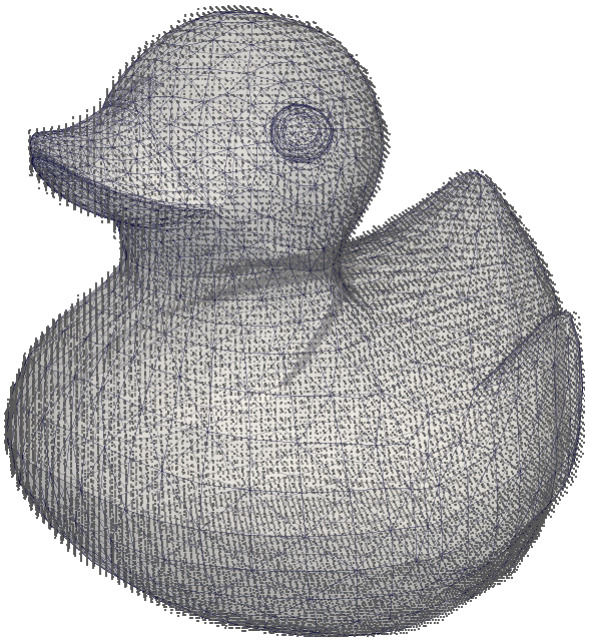
Draw points



3D model



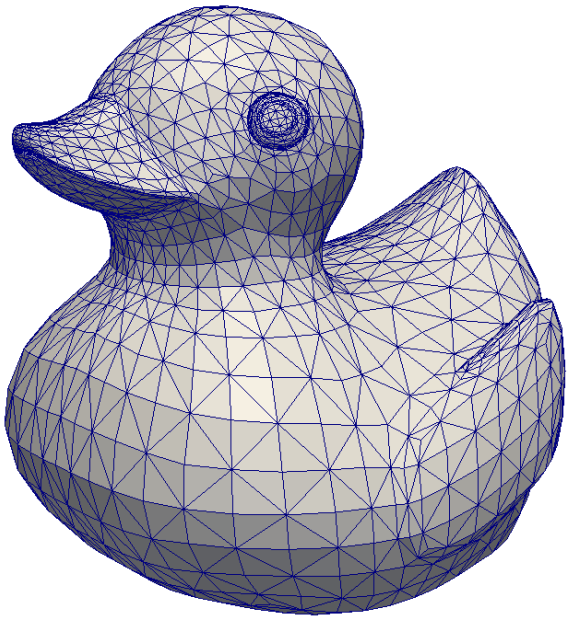
Triangles



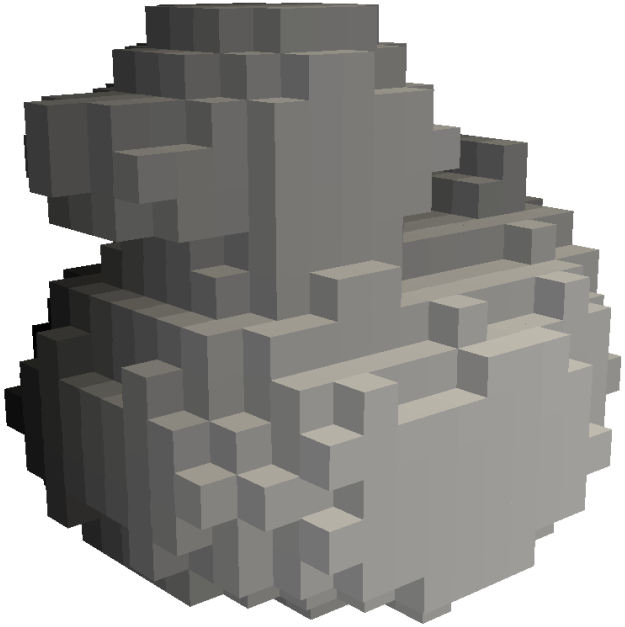
Points

Draw points

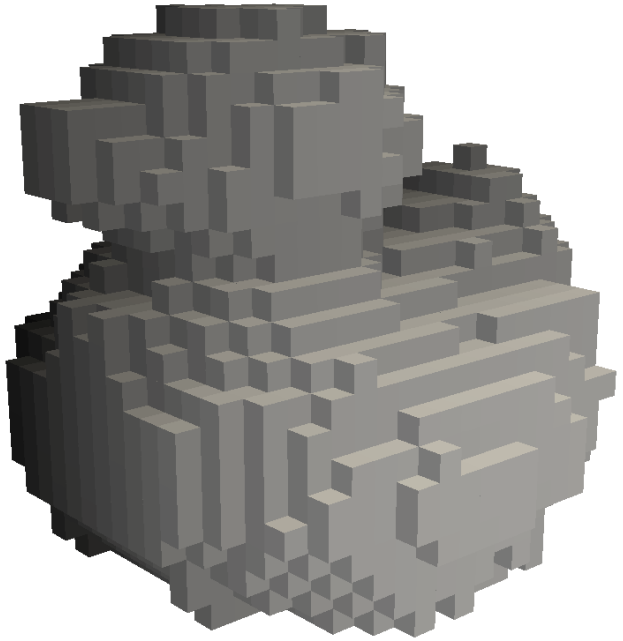
Particles are depicted as cubes



Triangles

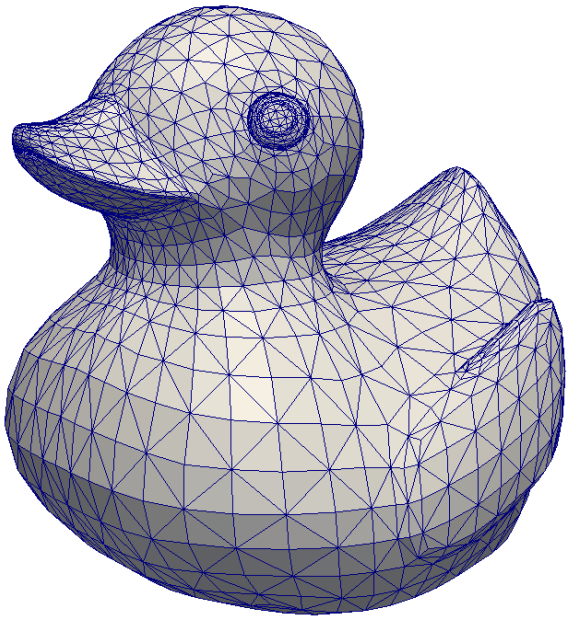


Low resolution
836 particles

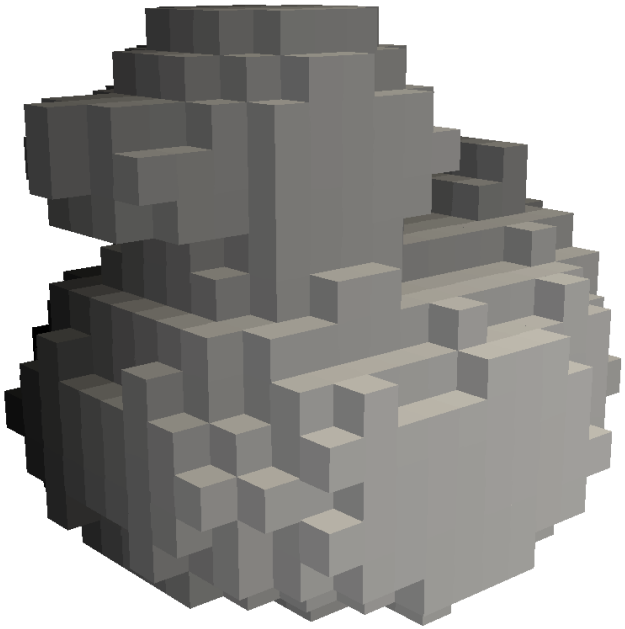


Higher resolution
1,836 particles

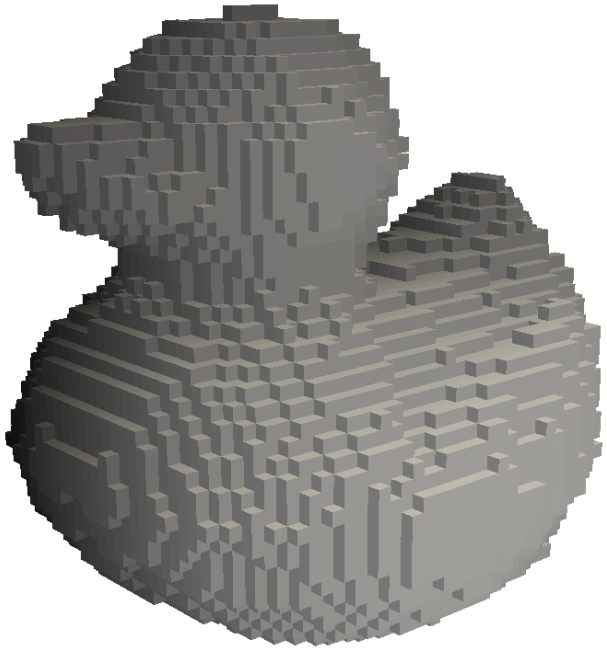
Draw points



Triangles

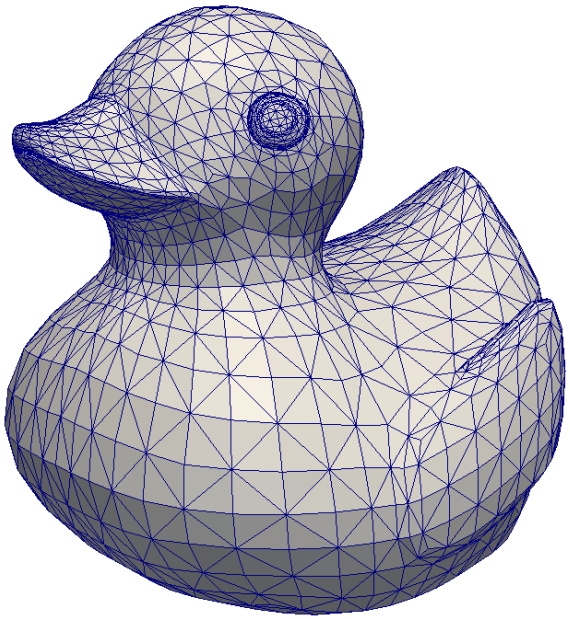


Low resolution
836 particles

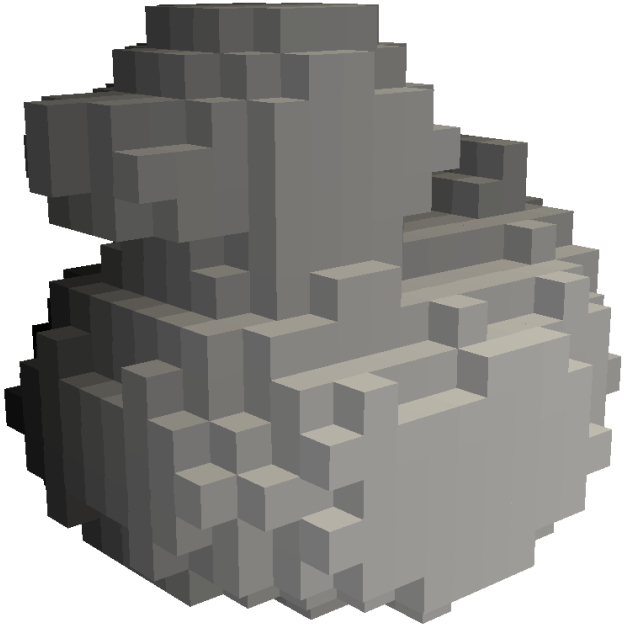


Higher resolution
6,934 particles

Draw points



Triangles

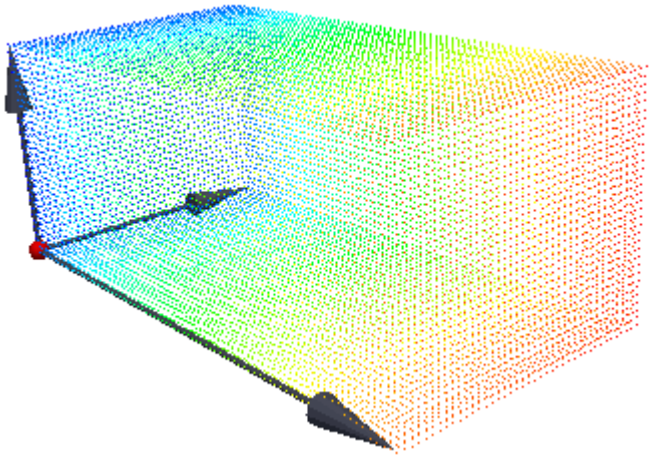


Low resolution
836 particles

Higher resolution
27,474 particles

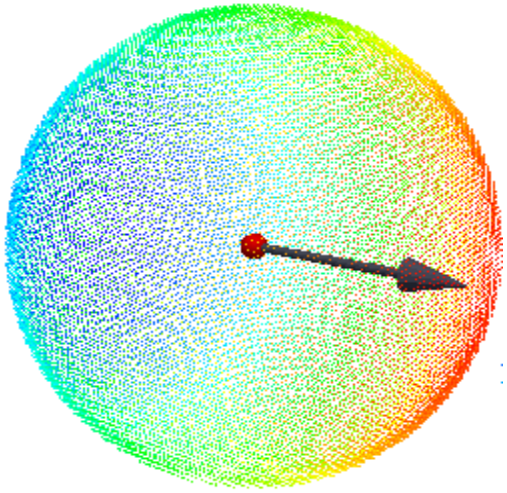
Predefined objects

Predefined shapes can be added to the simulation just by setting up some configuration parameters:



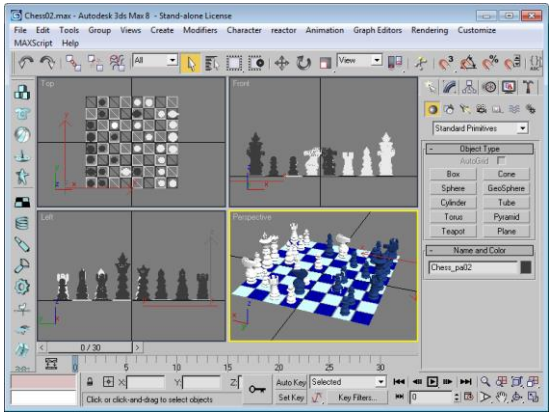
BOX: a **corner** and the **size** are required

SPHERE: the **centre** and **radius** are needed

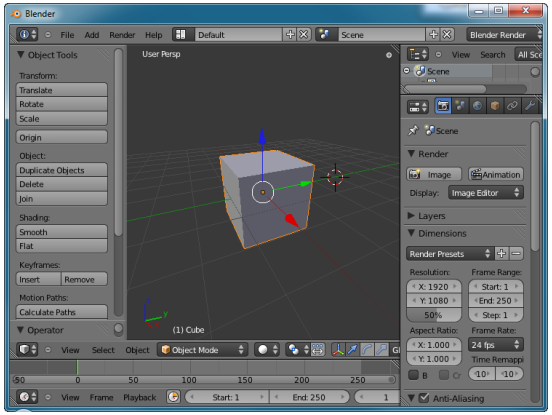


External objects

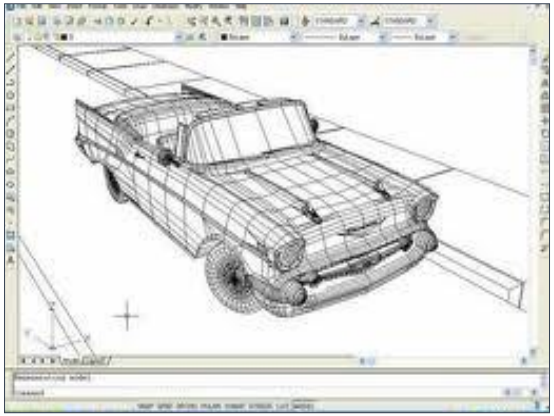
GenCase is able to load external object designed with different softwares.



3D Studio Max



Blender



AutoCAD

3DS	DXF	DWG	GIS	H5PART	CSV	MAX	
SHP	CAD	PLY	STL	VTK			

External objects

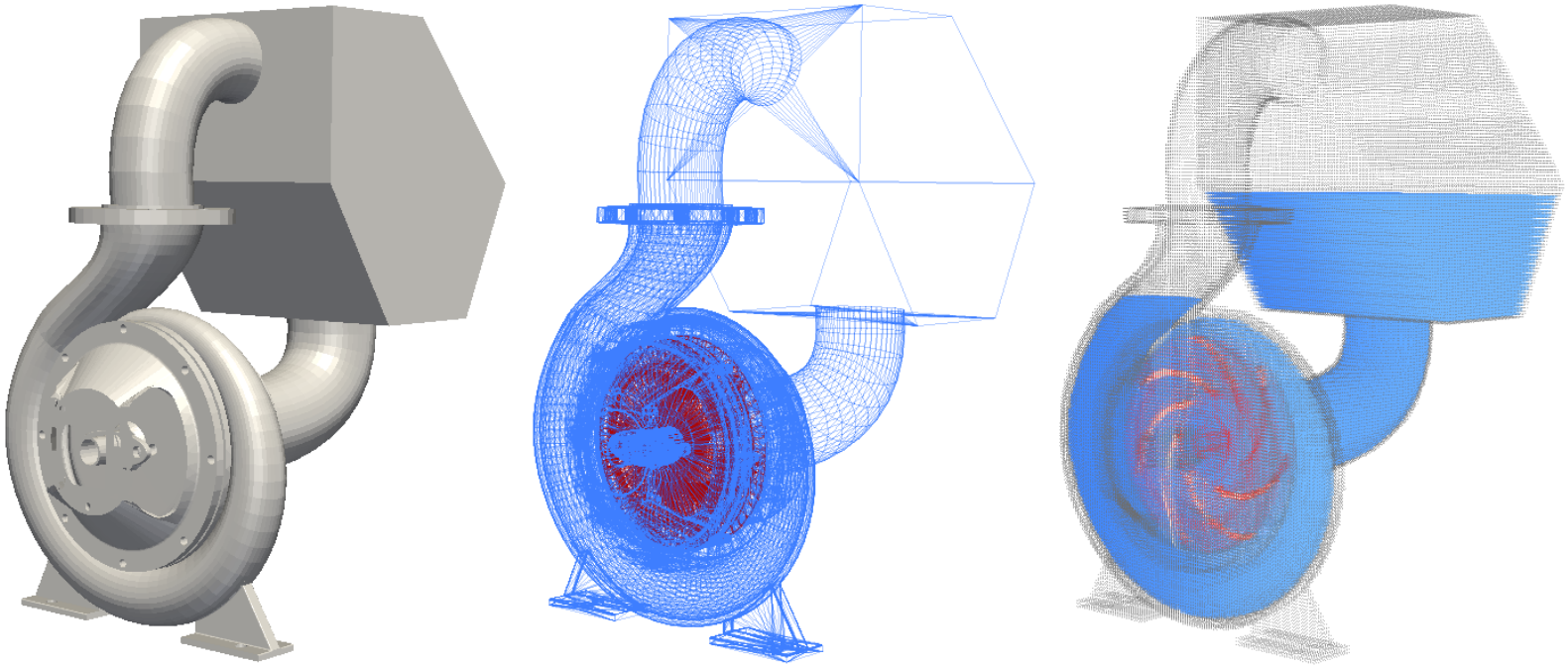
3DS DXF DWG GIS H5PART CSV MAX
SHP CAD PLY STL VTK

PLY -> exportable using BLENDER
STL -> exportable using 3DSTUDIO
VTK -> PARAVIEW

PLY, STL, VTK can be loaded by **GenCase**

External objects

CAD files

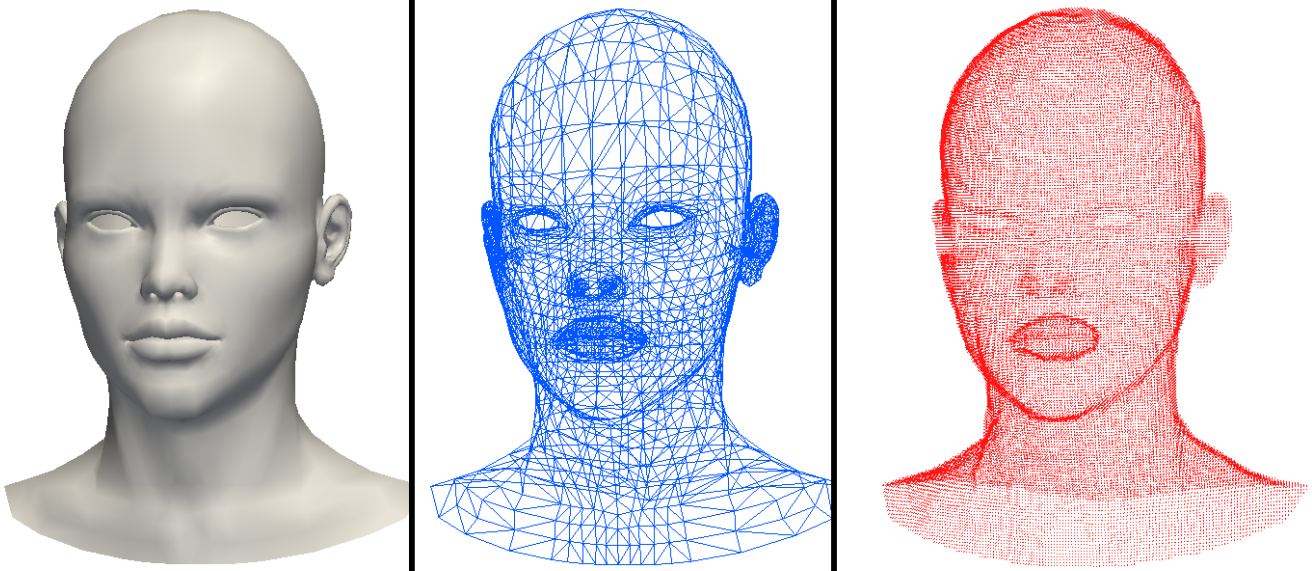


PLY, STL, VTK can be loaded by **GenCase**

Pre-processing: GenCase

External objects

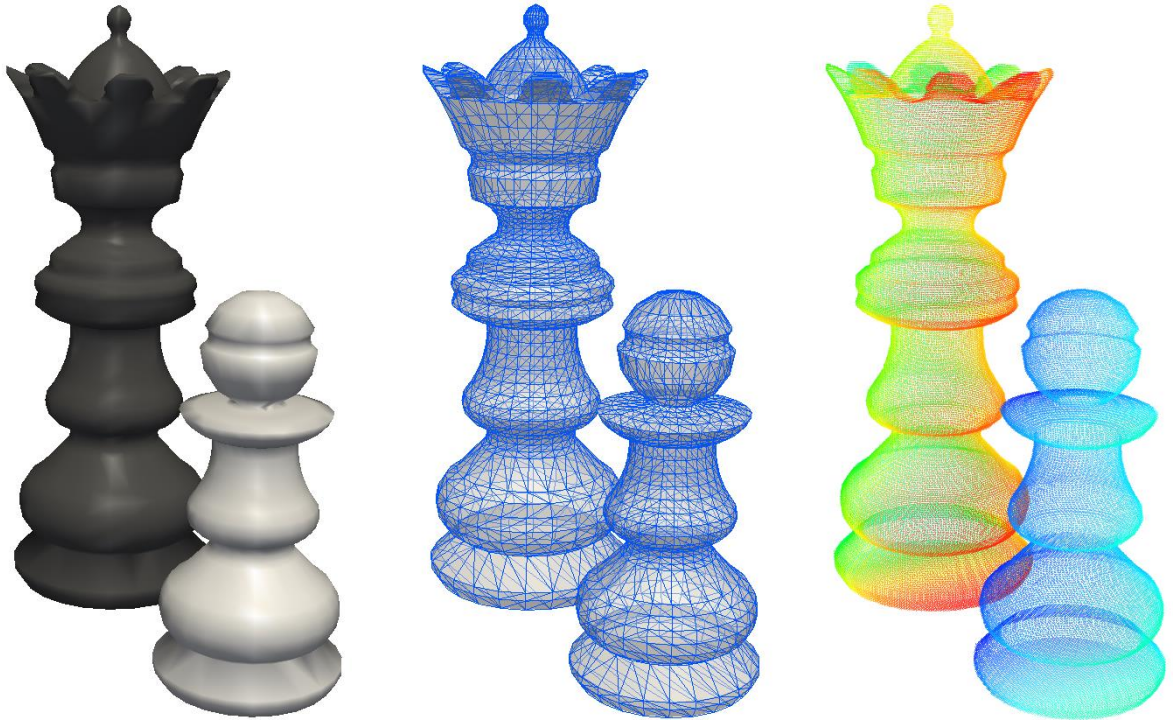
3D Studio objects



PLY, STL, VTK can be loaded by **GenCase**

External objects

3D Studio objects



PLY, STL, VTK can be loaded by **GenCase**

Outline of Presentation

- Pre-processing tool: GenCase
- XML input file
- DesignSPHysics
- Future developments

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=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>
```

```
<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>
        <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>
```

STRUCTURE OF THE XML FILE

http://dual.sphysics.org/index.php/downloads/

- **DUALSPHYSICS DOCUMENTATION:**
  - **DualSPHysics_v4.0_GUIDE.pdf**
  - **XML_GUIDE_v4.0.pdf**
  - **ExternalModelsConversion_GUIDE.pdf**
  - **PostprocessingCalculations.pdf**

- **DUALSPHYSICS PACKAGE:**
  - **DualSPHysics_v4.0_Linux_x64.zip**
  - **DualSPHysics_v4.0_Windows_x64.zip**

STRUCTURE OF THE XML FILE

Divided in two sections:

“casedef”

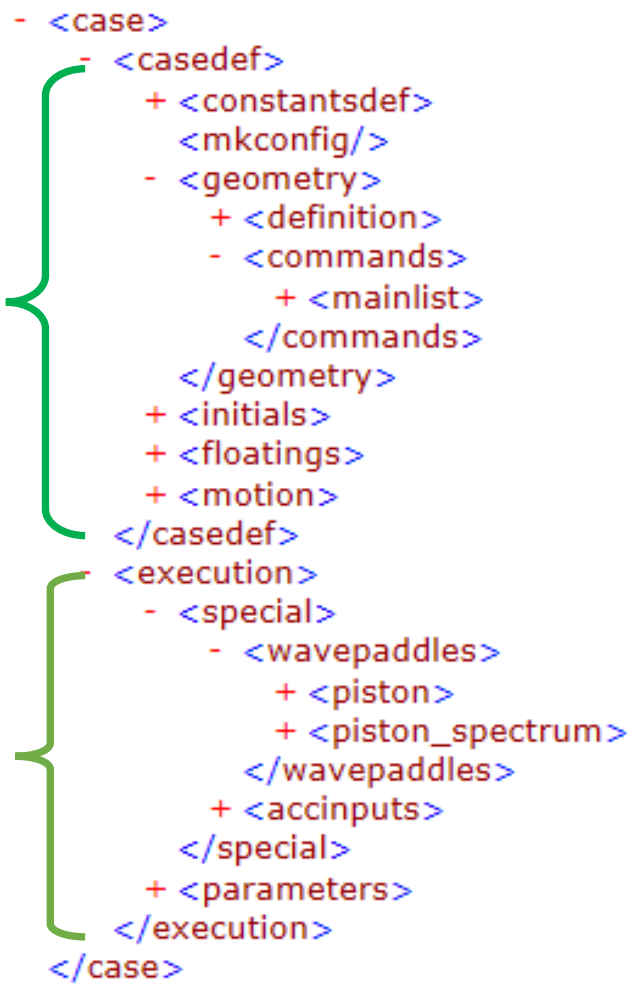
Definition of the case with initial geometry and configuration.

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



STRUCTURE OF THE XML FILE

- **"casedef" :**

  - constantsdef constants needed in SPH

  - mkconfig label configuration

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

  - initials special features for 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

XML file

CASEDEF-CONSTANTSDEF

```
<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>
```

XML file

CASEDEF-MKCONFIG

```
– <mkconfig boundcount="240" fluidcount="10">
    <mkorientbound mk="0" orient="YxZ"/>
    <mkorientfluid mk="1" orient="yzX"/>
    <mkorientfluid mk="2" orient="ZYx"/>
  </mkconfig>
```

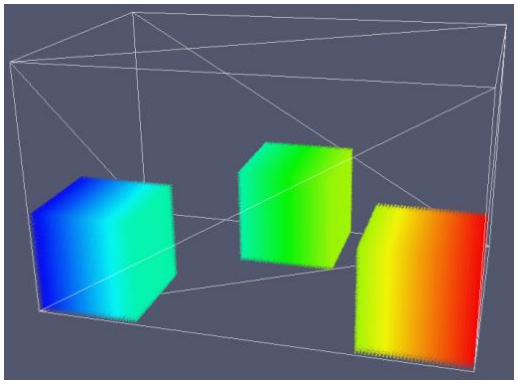
mk: label used to

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

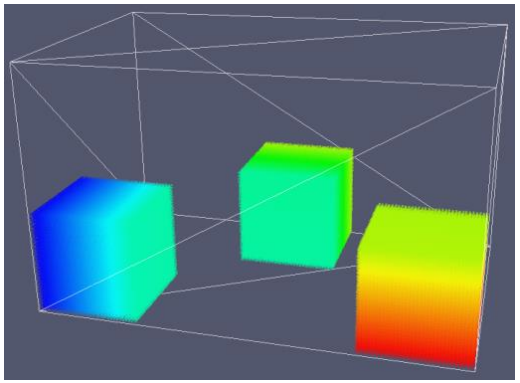
240 labels for boundary particles and

10 labels for fluid particles

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*)

XML file

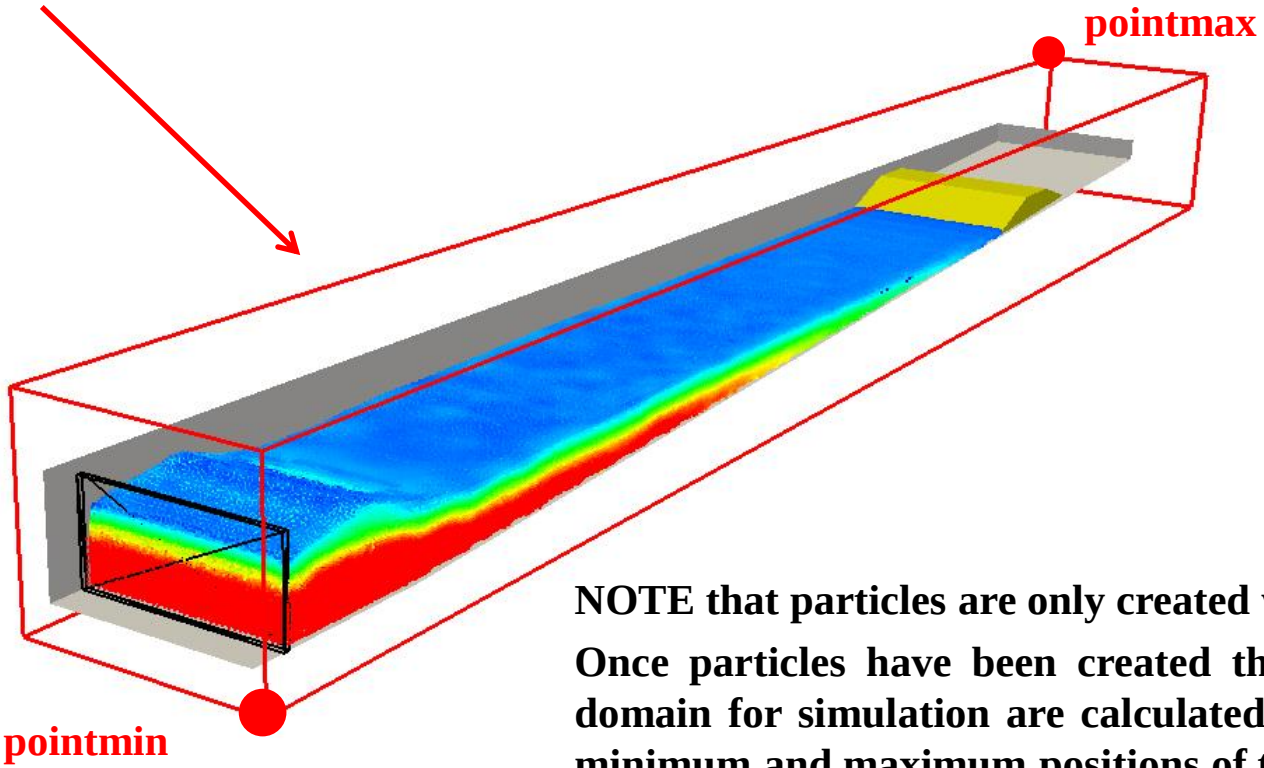
CASEDEF-GEOMETRY-DEFINITION

```
<!--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.

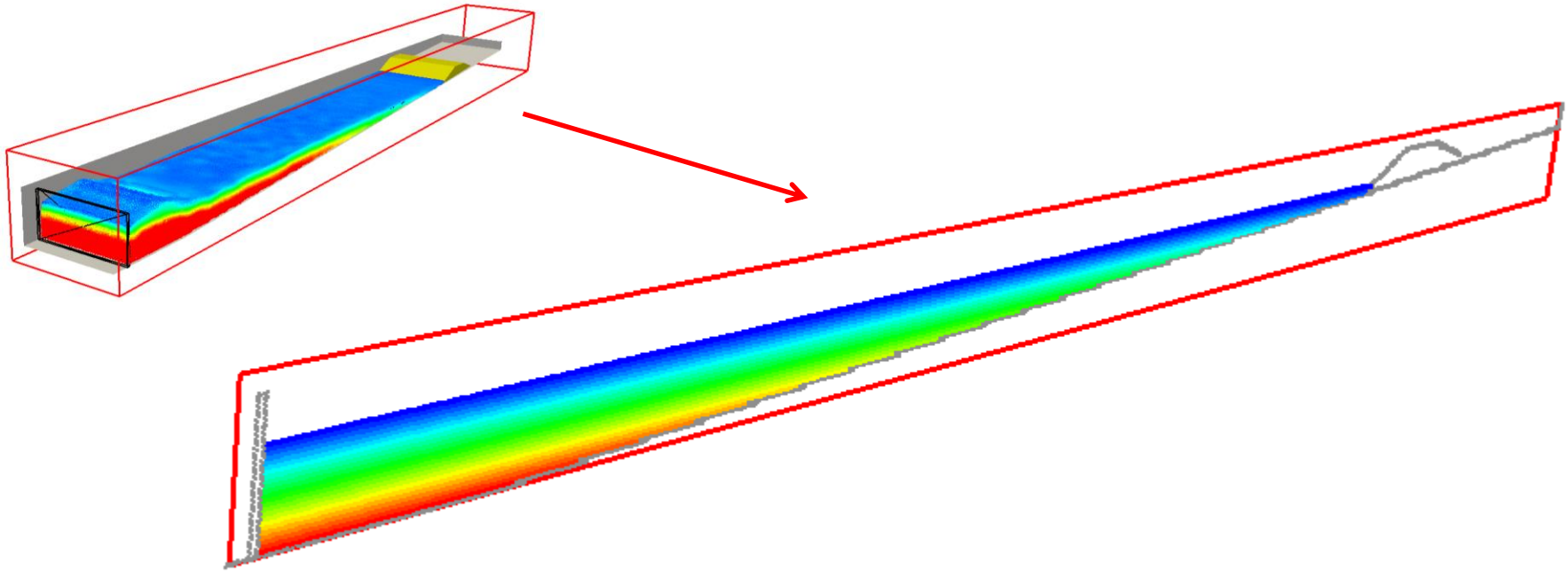
XML file

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

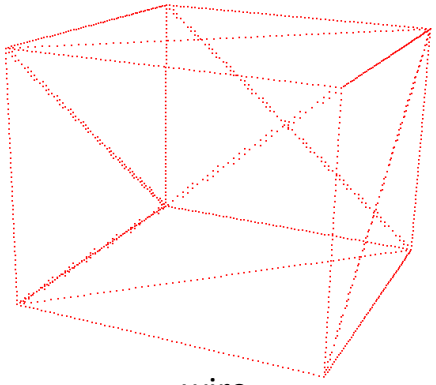
<pointmin> = <pointmax>



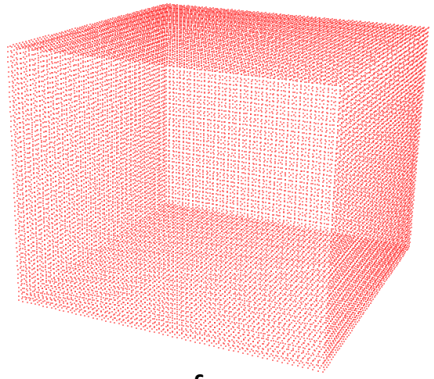
CASEDEF-GEOMETRY-COMMANDS-MAINLIST

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>
```



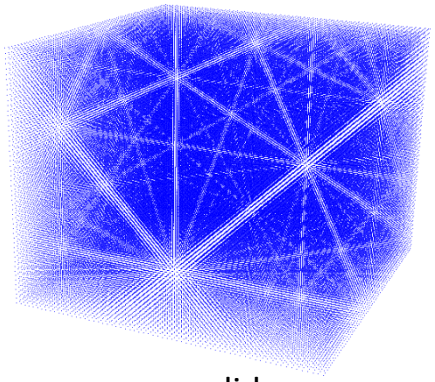
wire



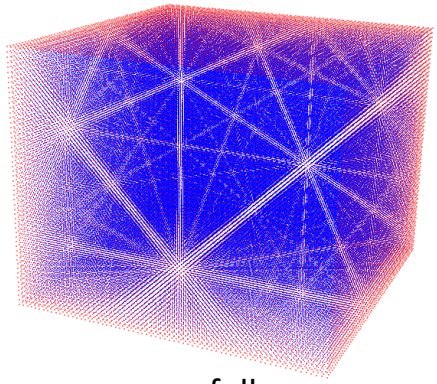
face

<setdrawmode>:

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



solid



full

CASEDEF-GEOMETRY-COMMANDS-MAINLIST

```
- <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>
```

IMPORTING 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

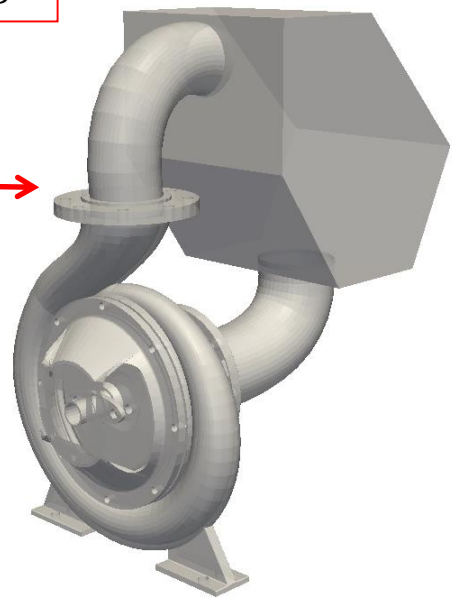
drawrotate a rotation is applied to the external object

drawscale scaling is applied to the external object

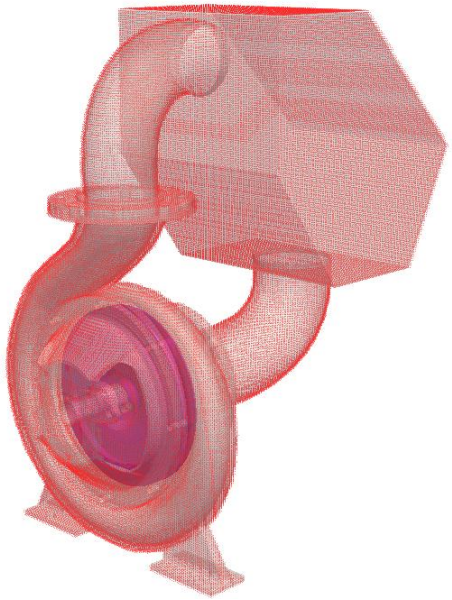
CASEDEF-GEOMETRY-COMMANDS-MAINLIST

IMPORTING 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



CASEDEF-GEOMETRY-COMMANDS-MAINLIST

```
- <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

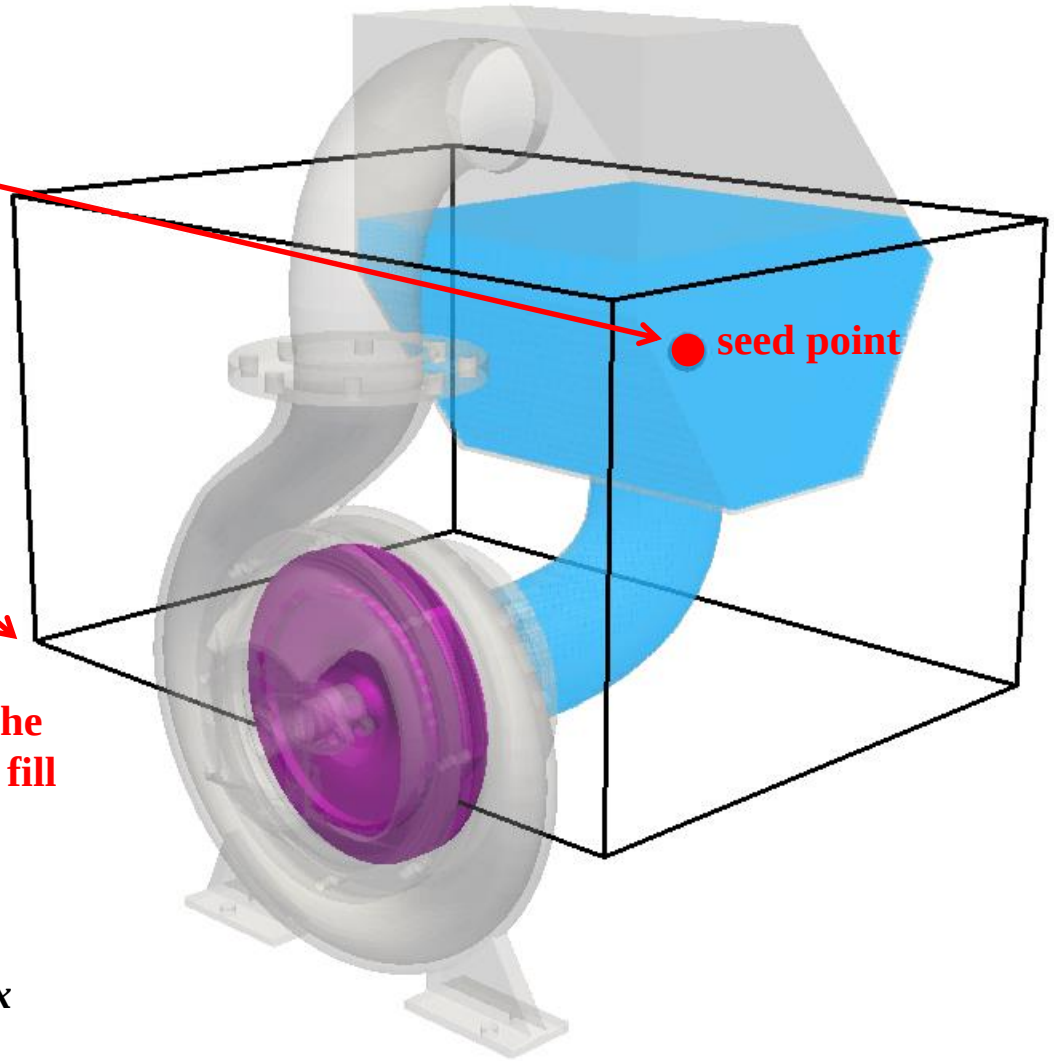
<fillprism>: fills with points starting from the seed within the limits defined by a prism

<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*

CASEDEF-GEOMETRY-COMMANDS-MAINLIST

```
- <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*

XML file

CASEDEF-FLOATINGS

```
- <floatings>
    <floating mkbound="0" relativeweight="1.3"/>
  - <floating mkbound="1" relativeweight="1.3">
      <velini x="1" y="3" z="2"/>
      <omegaini x="0.2" y="0.4" z="0.6"/>
    </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"/>
      <velini x="1" y="3" z="2"/>
      <omegaini x="0.2" y="0.4" z="0.6"/>
    </floating>
  - <floating mkbound="4">
      <massbody value="1300"/>
      <inertia x="20" y="22" z="24"/>
    </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

So that, the mass of a floating particles is:
masspart = massbody / nfloat **or**
masspart = relativeweight * rhop0 * dp^3 **or**
masspart = rhopbody * dp^3

These variables are computed by GenCase or can be also specified in advance:
center gravity center of the rigid object
inertia momentum of inertia of the rigid object
velini initial linear velocity of the object
omegaini initial angular velocity of the object

XML file

CASEDEF-MOTION

•*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

XML file

CASEDEF-MOTION

•*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

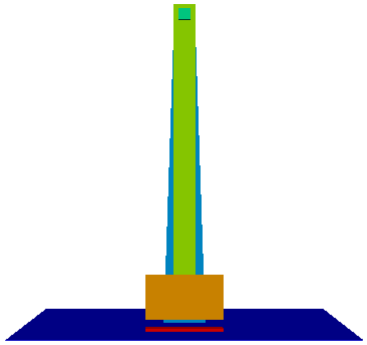
<mvcirsinu>: sinusoidal circular movement

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

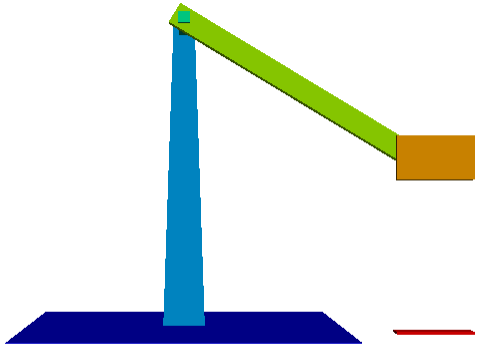
XML file

CASEDEF-MOTION

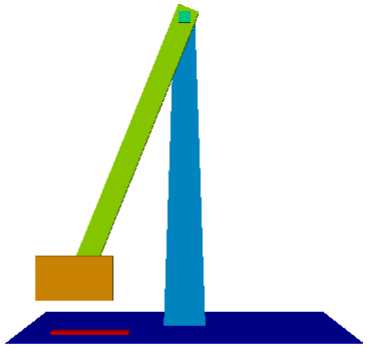
•*Motion07*: sinusoidal movement (<**mvrectsinu** />, <**mvrotsinu** />, <**mvcirsinu** />)



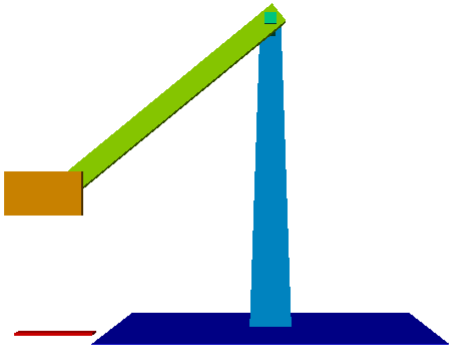
Time: 0.00 s



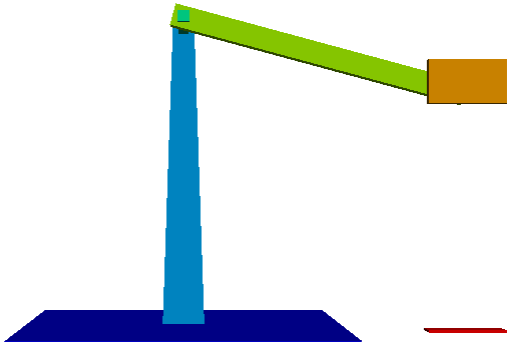
Time: 1.40 s



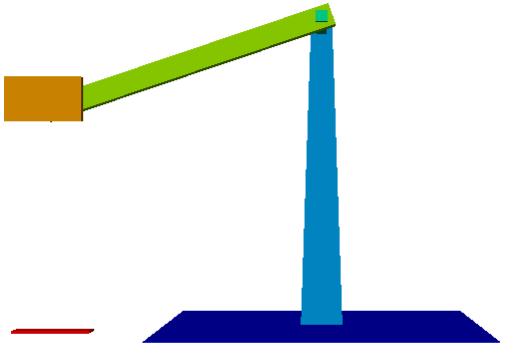
Time: 2.80 s



Time: 4.20 s



Time: 5.60 s



Time: 7.00 s

XML file

CASEDEF-MOTION

•*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"

XML file

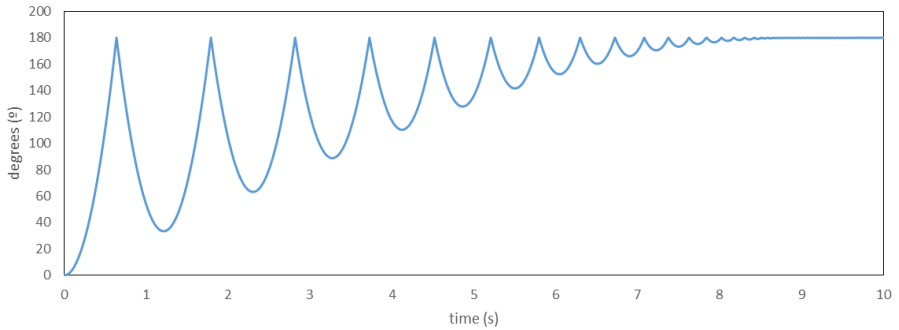
CASEDEF-MOTION

•*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>
```



XML file EXECUTION-SPECIAL-WAVEPADDLES-PISTON

Generation of regular waves

```
<piston>
    <mkbound value="10" comment="Mk-Bound of selected particles" />
    <waveorder value="2" comment="Order wave generation 1:1st order, 2:2nd order (def=1)" />
    <start value="0" comment="Start time (def=0)" />
    <duration value="0" comment="Movement duration, Zero is the end of simulation (def=0)" />
    <depth value="0.27" comment="Fluid depth (def=0)" />
    <fixeddepth value="0" comment="Fluid depth without paddle (def=0)" />
    <pistondir x="1" y="0" z="0" comment="Movement direction (def=(1,0,0))" />
    <waveheight value="0.1" comment="Wave height" />
    <waveperiod value="1.3" comment="Wave period" />
    <phase value="0" comment="Initial wave phase in function of PI (def=0)" />
    <ramp value="0" comment="Periods of ramp (def=0)" />
    <savemotion periods="24" periodsteps="20" xpos="2" zpos="-0.15"
                comment="Saves motion data. xpos and zpos are optional. zpos=-depth" />
</piston>
```

- *waveorder*: order of wave generation (1st order or 2nd order)
- *depth*: depth at front of the piston
- *waveheight*: wave height H
- *waveperiod*: wave period T
- *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)

EXECUTION-SPECIAL-WAVEPADDLES-PISTON_SPECTRUM

Generation of irregular waves

```
<piston_spectrum>
    <mkbound value="10" comment="Mk-Bound of selected particles" />
    <waveorder value="2" comment="Order wave generation 1:1st order, 2:2nd order (def=1)" />
    <start value="0" comment="Start time (def=0)" />
    <duration value="0" comment="Movement duration, Zero is the end of simulation (def=0)" />
    <depth value="0.27" comment="Fluid depth (def=0)" />
    <fixeddepth value="0" comment="Fluid depth without paddle (def=0)" />
    <pistondir x="1" y="0" z="0" comment="Movement direction (def=(1,0,0))" />
    <spectrum value="jonswap" comment="Spectrum type: jonswap,pierson-moskowitz" />
    <discretization value="stretched"
                comment="Spectrum discretization: regular,random,stretched,cosstretched (def=stretched)" />
    <waveheight value="0.1" comment="Wave height" />
    <waveperiod value="1.3" comment="Wave period" />
    <peakcoef value="3.3" comment="Peak enhancement coefficient (def=3.3)" />
    <waves value="128" comment="Number of waves to create irregular waves (def=50)" />
    <randomseed value="2" comment="Random seed to initialize a pseudorandom number generator" />
    <serieini value="2.8" comment="Initial time in irregular wave serie (def=0)" />
    <ramptime value="1" comment="Time of ramp (def=0)" />
    <savemotion time="50" timedt="0.05" xpos="2" zpos="-0.15"
                comment="Saves motion data. xpos and zpos are optional. zpos=-depth" />
    <saveserie timemin="0" timemax="1300" timedt="0.05" xpos="0" comment="Saves serie data (optional)" />
    <saveseriewaves timemin="0" timemax="1000" xpos="2" comment="Saves serie heights" />
</piston_spectrum>
```

XML file

EXECUTION-PARAMETERS

Parameters for execution in DualSPHysics

```
<parameters>
    <parameter key="PosDouble" value="1" comment="Precision in particle interaction 0:Simple, 1:Double, 2:Uses and saves double (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="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="1.5" 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/m^3" />
    <parameter key="RhopOutMax" value="1300" comment="Maximum rhop valid (default=1300)" units_comment="kg/m^3" />
</parameters>
```

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=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>
```

```
<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

Summary of the number of created particles and computed constants

```
<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/m^3" />
    <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 />
```

np=total number of particles
nb=boundary particles
nbf=fixed boundary particles and final ***mk*** of the objects

NOTE value of final "mk"
mk=mkbound+11
mk=mkfluid+1

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

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

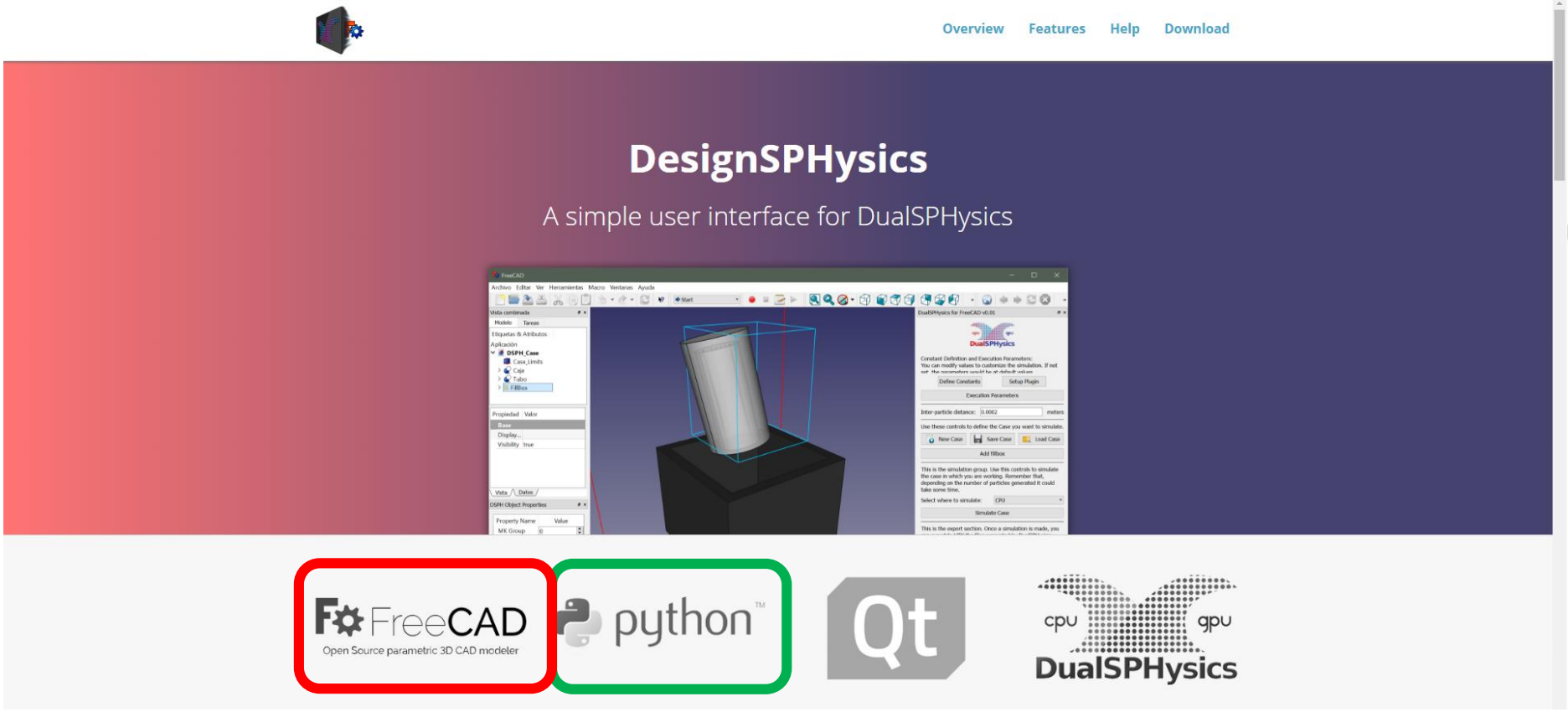
Outline of Presentation

- Pre-processing tool: GenCase
- XML input file
- DesignSPHysics: **new GUI !!!**
- Future developments

DesignSPHysics

A simple user interface for DualSPHysics

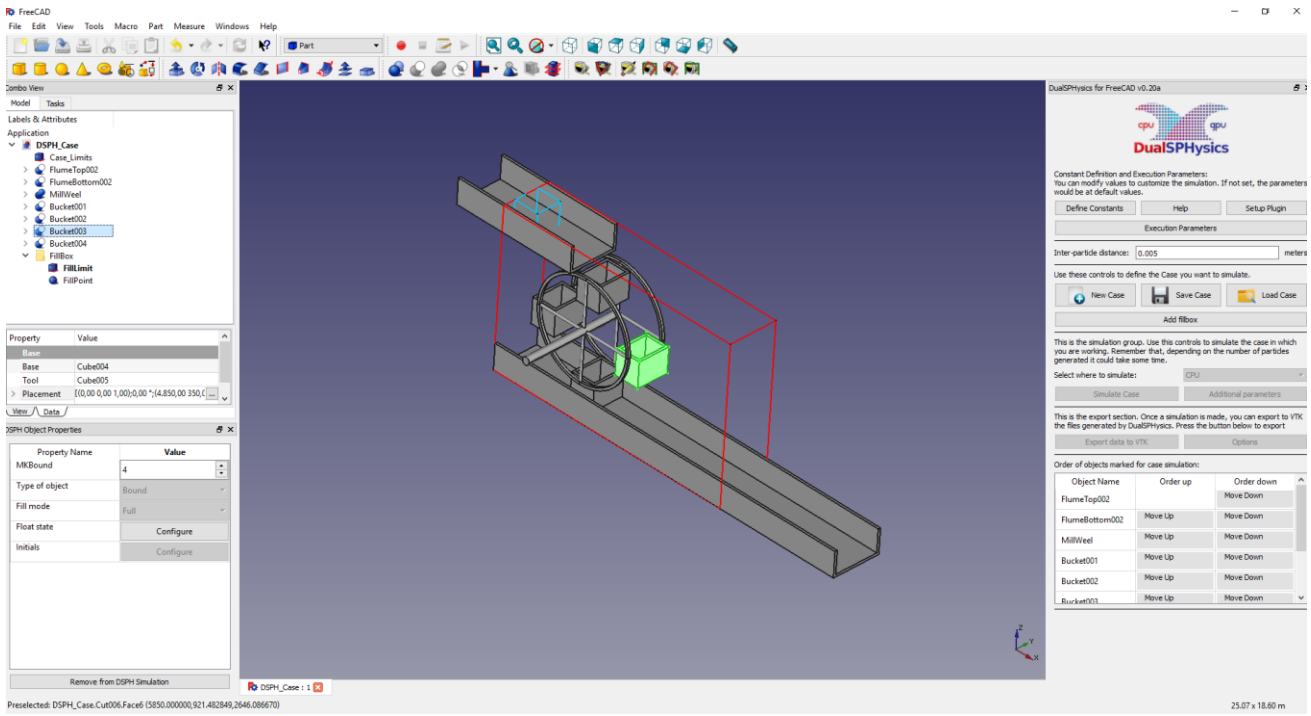
Andrés Vieira



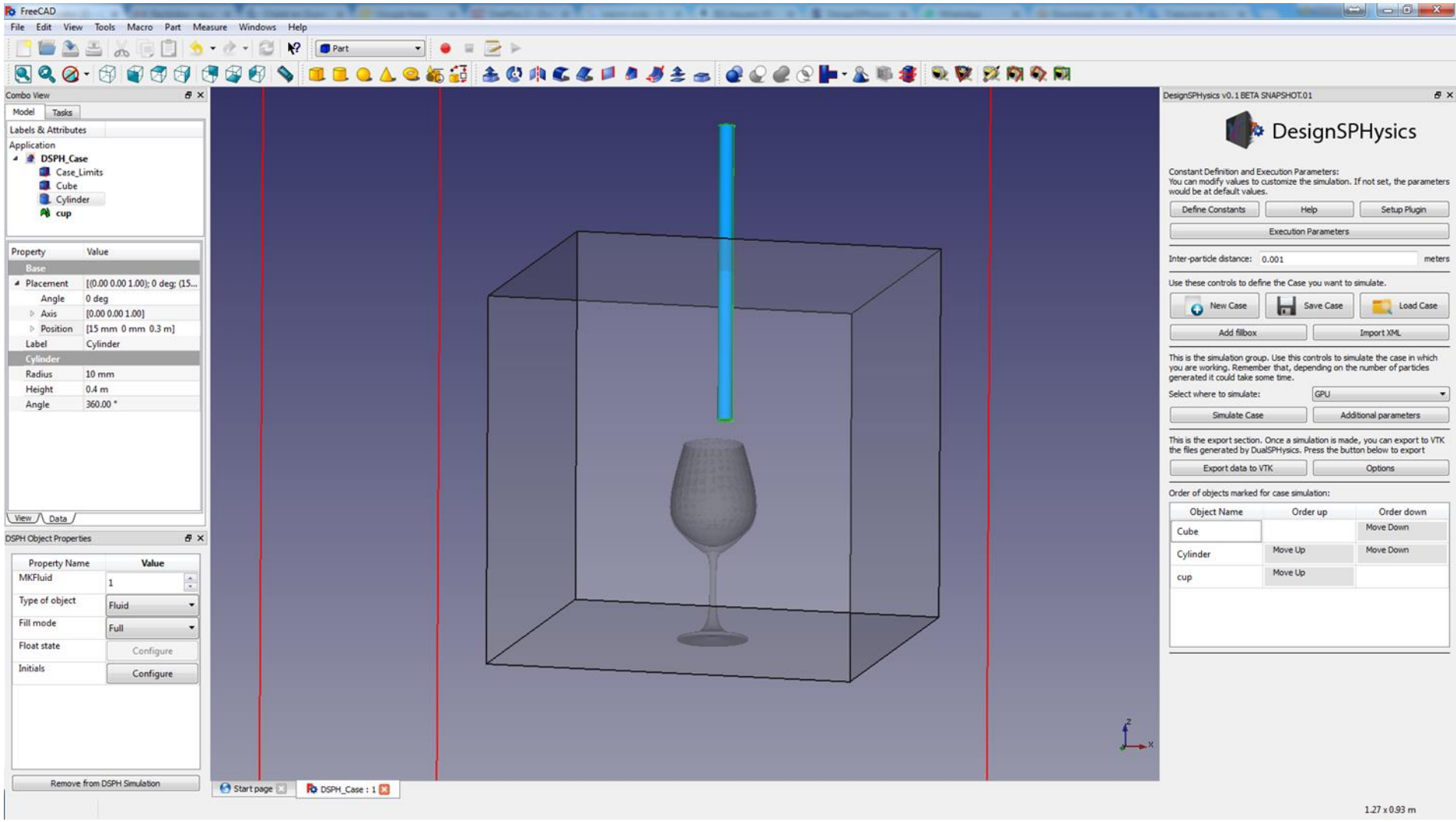
DesignSPHysics

A simple user interface for DualSPHysics

Modular: Integrates with FreeCAD, it is simple and easy to use.
Configurable: No need to edit external files or work with messy code.
Not only case design: It can run a simulation clicking on a button.
Coherent GUI: Easy to use, easy to learn.

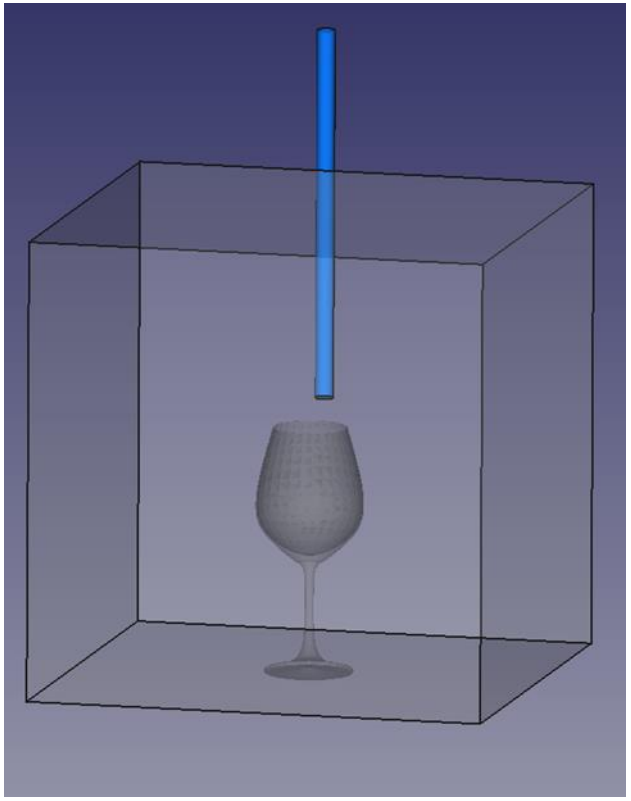


Let us try DesignSPHysics



STEPS in DesignSPHysics

- Install FreeCAD version 0.16 or higher (https://sourceforge.net/projects/free-cad/)
- Download DesignSPHysics beta-version-> Installer.exe
- Open FreeCAD
- *Opens a dialog to let you execute a recorded macro*: Use DSPH.py at %appdata%/FreeCAD/Macro
- New Case: DSPH_Case (**CaseWine**)
  - Case_Limits
    - Base/Placement/Position: x=-0.5m, y=-0.5m, z=-0.5m
    - Box: Length=1m, Width=1m, Height=1.5m
  - Cube
    - Base/Placement/Position: x=-0.25m, y=-0.25m, z=0m
    - Box: Length=0.5m, Width=0.5m, Height=0.5m
  - ADD TO DSPH SIMULATION
    - MKBound=0, Type of object=Bound, Fill mode=Wire
  - Cylinder
    - Base/Placement/Position: x=0.015m, y=0m, z=0.3m
    - Cylinder: Radius=0.01m, Height=0.4m, Angle=360º
  - ADD TO DSPH SIMULATION
    - MKFluid=1, Type of object=Fluid, Fill mode=Full
  - Import: cup.stl (reads assuming units in mm)
  - ADD TO DSPH SIMULATION
    - MKBound=1, Type of object=Bound, Fill mode=Face
  - Inter-particle distance=0.001 meters
  - Define Constants
  - Execution Parameters
    - Time of simulation=1 seconds
    - Time out data=0.005 seconds
  - Setup Plugin: path of GenCase4, DualSPHyics4, PartVTK4
  - Save Case: Executes GenCase4 (creates .XML and initial .BI4)
  - Simulate Case: Executes DualSPHysics4 (CPU or GPU)
  - Export data to VTK: Executes PartVTK4 (VTK of all particles)



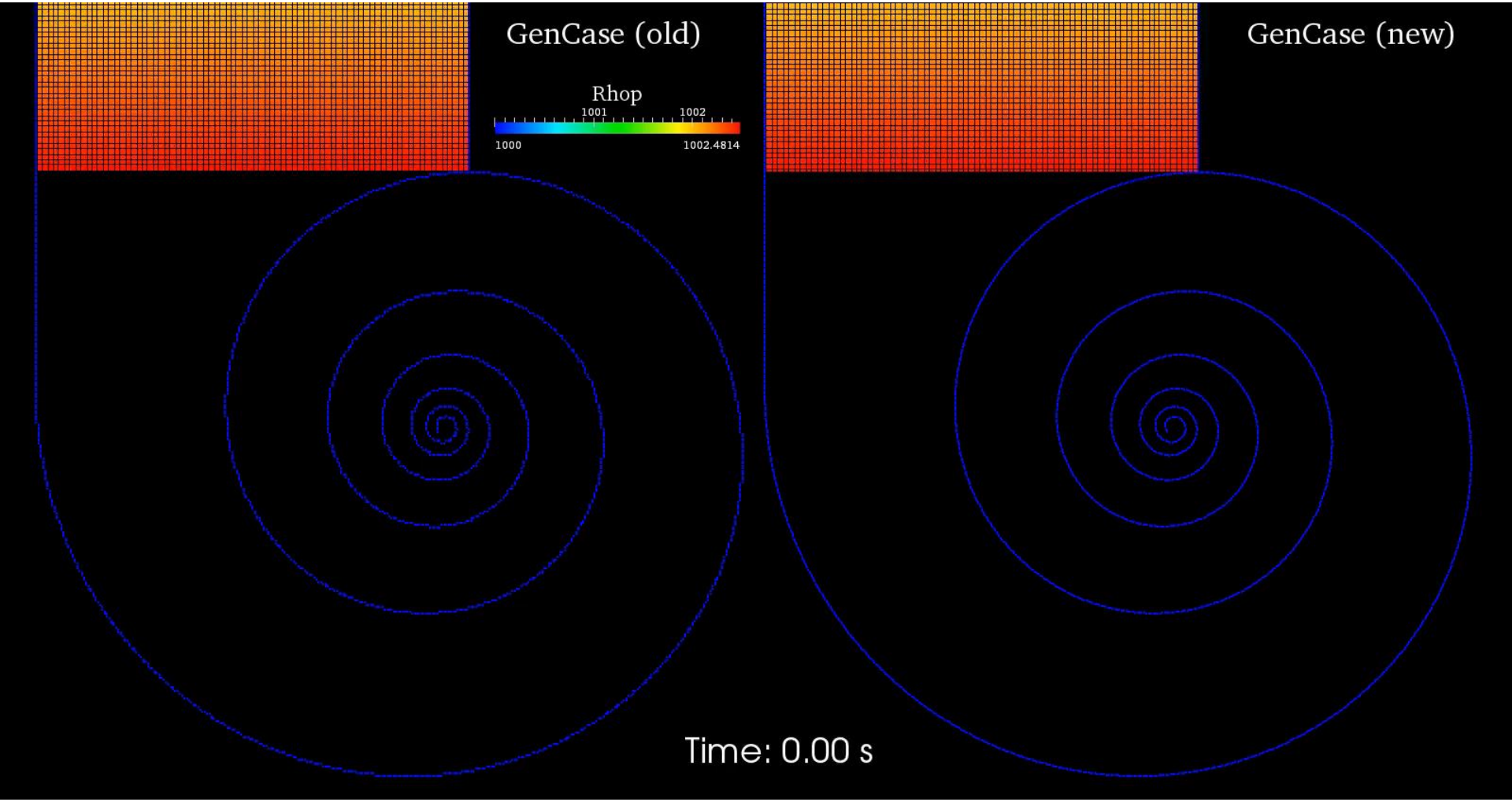
Visualisation in Paraview loading VTK files

Outline of Presentation

- Pre-processing tool: GenCase
- XML input file
- DesignSPHysics: **new GUI !!!**
- Future developments

Future developments

- FreePoints



Future developments

- FreePoints
- Computation of normals (new BCs)
- Release of DesignSPHysics

MANCHESTER 1824

The University of Manchester

Thank you

Acknowledgements

- DualSPHysics team: all developers and contributors
- Andrés Vieira

Website

Free open-source **DualSPHysics** code:

http://www.dual.sphysics.org