Two-phase liquid-sediment implementation in DualSPHysics

1. Introduction

This guide provides a depiction of the case set up for the multi-phase liquid-sediment model implemented in DualSPHysics solver [1]. The model is capable of simulating problems involving liquid-sediment phases with the addition of highly non-linear deformations and free-surface flows which are frequently encountered in applied hydrodynamics. More specifically, the two-phase liquid-solid model is aimed at flow-induced erosion of fully saturated sediment. These multi-phase sediment scouring phenomena are induced by rapid flows creating shear forces at the surface of the sediment which causes the surface to yield and produce a shear layer of sediment suspended particles at the interface and finally sediment suspension in the fluid. Applications include scouring in industrial tanks, port hydrodynamics, wave breaking in coastal applications and scour around structures in civil and environmental engineering flows among others.

The reader is referred to the work of Fourtakas *et al*. [2-3] for more details regarding the liquid-sediment two phase liquid, where a detailed description of the model and validation can be found.

2. Case setup

Although the two-phase implementation enhances the capabilities of DualSPHysics some functionality has been disabled in this BETA version to ensure the stability of the multi-phase code and use options that are compatible with the two-phase implementation such as the viscous formulation. Due to the large computational cost the current version runs only on the GPU. These options are described bellow in the *CaseTwoPhases* test case included in the package.

A 2-D erodible dam break case is included along this guide as demonstration test case to assist with the creation of new cases. The cases consist of a dam breaking over an erodible bed as depicted in Figure 1.

The case setup is described in the remaining of this guide however, the readers should be familiar with the DualSPHysics package documentation and *XML* guide of the pre-processing tool *GenCase*.

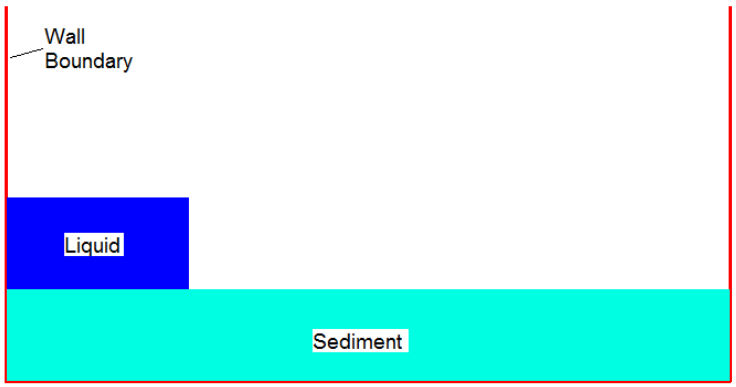

Figure 1. Domain setup for the erodible dam break.

The input *XML* file has been modified to include some additional *parameters* as an input to the Drucker-Prager yield criterion, the Herschel-Bulkley-Papanastasiou model and the setup of the different phases in the DualSPHysics solver, a detailed description follows.

The `<constantsdef> ... </constantsdef>` section of the *XML* has no changes and the user should follow the *XML GenCase* guide for the setup of this section. An example follows,

```
<constantsdef>
      <lattice bound="2" fluid="1" />
      <gravity x="0" y="0" z="-9.81" />
      <cflnumber value="0.1" />
      <hswl value="0" auto="true" />
      <coefsound value="20" />
      <coefficient value="0.92" />
      <gamma value="7" />
      <rhop0 value="1000" />
      <eps value="0.0" />
</constantsdef>
```

(Note that the `coefsound` is depreciated since the used can input the C_{s0} manually for each phase as will be explained shortly but it is required by DualSPHysics when reading the *XML* file and therefore it acts as a dummy input.)

The generation of the geometry of the domain under `<geometry> ... </geometry>` is used to generate the different phases using the `setmkfluid mk` flag. In the example *XML* file the sediment domain is marked by `mk=1` and the fluid domain as `mk=0`. Note, that both phases must be identified by the `setmkfluid` flag. The configuration setup uses the `mk=0` as a fluid and `mk=1` as a sediment phase in conjunction with the parameters section `<parameters> ... </parameters>`. Boundary particles with flag `setmkbound` are not affected. An example follows.

Creating the sediment phase with `setmkfluid mk="1"`:

```
<setmkfluid mk="1" />
      <drawbox>
      <boxfill>solid</boxfill>
      <point x="0" y="0" z="0" />
      <size x="4" y="2" z="0.5" />
</drawbox>
```

Creating the liquid phase with `setmkfluid mk="0"`:

```
<setmkfluid mk="0" />
      <drawbox>
      <boxfill>solid</boxfill>
      <point x="0" y="0" z="0.51" />
      <size x="1" y="2" z="0.5" />
</drawbox>
```

After the creation of the geometry the `<parameters> ... </parameters>` section is modified to load the variables and parameters of each different phase. Although the structure of the parameters section remains mostly unchanged, in order to ensure stability in this BETA version, some functionalities of the DualSPHysics solver have been disabled. Table 1 summarises the changes and restrictions.

Parameter key	Comment/Restriction
`StepAlgorithm`	Only the use of Symplectic is allowed
`Kernel`	Only the use of Wendland kernel is allowed
`ViscoTreatment`	The implementation uses the new viscous formulation
`ShepardSteps`	The Shepard filer has been disabled
`DeltaSPH`	Recommended value of 0.1
`Floating bodies`	Disabled in this BETA version
`Periodic boundaries`	Disabled in this BETA version
`CPU implementation`	Disabled in this BETA version

Table 1. The `parameter` section of the xml file.

In addition to these input parameters for the single-phase DualSPHysics solver, a number of constants and variables are required for the definition of the two phases and the rheological characteristics of the phases. Firstly the number of phases is defined as

```
<parameter key="PhaseCount" value="2"
```

Secondly, the constants and parameters of each phase are defined in the XML file. The user is required to setup the constants and parameters for each phase differently by identifying the phase by the `mk` value of the geometry section, i.e. in the attached xml file the liquid phase has an `mk=0` thus

```
<parameter key="PhaseMkFluid_0" value="0" comment="MkFluid of fluid" />
```

denotes the liquid phase.

The remaining parameters are summarised in the following table.

Parameter key	Value	Comment
`PhaseRhop0_0`	1000	Density of the phase
`PhaseCsound_0`	80	The numerical speed of sound for the phase
`PhaseGamma_0`	7	Polytropic index of the phase
`PhaseVisco_0`	0.001	Viscosity or consistency index (liquid and sediment respectively)
`HB_parameter_0`	1	Parameter *n* of the Herschel-Bulkley-Papanastasiou model. Use 1 to reduce to Bingham model
`HB_Papanastasiou_0`	0	Parameter *m* of the Herschel-Bulkley-Papanastasiou model. Use 0 to reduce to Herschel-Buckley. Note that the combination of n=1 and m=0 reduces the model to a Newtonian constitutive equation
`Coh_0`	0	Cohesion coefficient of the phase, use 0 for non cohesive material
`PhiF_0`	0	Internal friction angle of the phase, use 0 for liquids

YieldStrength_0	0	Yield strength of sediment, set to 0 to use the Drucker-Prager formulation or input the yield strength of the material in *Pa*
Phase_0	0	Use a Newtonian or Non-Newtonian formulation without the particle resuspension model respectively

Table 2. XML parameters defining the properties and constants of the liquid phase.

Please note the "_0" of all the parameters in Table 2. The "0" ending denotes the phase of mk=0. In the example *XML* the sediment phase parameters are ending in "_1" since the mk of the sediment phase is mk=1.

An example is given bellow in Figure 2.

```
<parameters>
<parameter key="DeltaSPH" value="0.1" comment="DeltaSPH parameter, 0 not applied (def=0.1)" />
    <parameter key="DtIni" value="0.0000000001" comment="Initial time step" />
    <parameter key="DtMin" value="0.0000000001" comment="Minimum time step (def=0.00001)" />
    <parameter key="TimeMax" value="2.0" comment="Time of simulation" />
    <parameter key="TimeOut" value="0.01" comment="Time out data" />
    <parameter key="IncZ" value="5" comment="Increase of Z+" />
    <parameter key="PartsOutMax" value="1" comment="Proportion of fluid particles out allowed (def=1)" />

    <parameter key="PhaseCount" value="2" comment="Number of different fluids" />

    <parameter key="PhaseMkFluid_0" value="0" comment="MkFluid of phase 1" />
    <parameter key="PhaseRhop0_0" value="1000" comment="Rhop value for phase 1" />
    <parameter key="PhaseCsound_0" value="80" comment="Csound value for phase 1" />
    <parameter key="PhaseGamma_0" value="7" comment="Gamma value for phase 1" />
    <parameter key="PhaseVisco_0" value="0.001" comment="Viscosity value for phase" />
    <parameter key="HB_parameter_0" value="1" comment="Use 1 to reduce to Bingham for Newtonian liquid" />
    <parameter key="HB_Papanastasiou_0" value="0" comment="Use 0 to reduce to HB for Newtonian liquid" />
    <parameter key="Coh_0" value="0" comment="Cohesion coefficient of the phase" />
    <parameter key="PhiF_0" value="0" comment="Internal friction angle of the phase" />
    <parameter key="YieldStrength_0" value="0" comment="Yield strength of phase if constant Yield strength is being used (def=0)" />
    <parameter key="Phase_0" value="0" comment="Newtonian=0, Non-Newtonian" />

    <parameter key="PhaseMkFluid_1" value="1" comment="MkFluid of phase 1" />
    <parameter key="PhaseRhop0_1" value="1500" comment="Rhop value for phase 1" />
    <parameter key="PhaseCsound_1" value="81" comment="Csound value for phase 1" />
    <parameter key="PhaseGamma_1" value="7" comment="Gamma value for phase 1" />
    <parameter key="PhaseVisco_1" value="0.002" comment="Viscosity value for phase" />
    <parameter key="HB_parameter_1" value="1" comment="Use 1 to reduce to Bingham for Newtonian liquid" />
    <parameter key="HB_Papanastasiou_1" value="100" comment="Use 0 to reduce to HB for Newtonian liquid" />
    <parameter key="Coh_1" value="1" comment="Cohesion coefficient of the phase" />
    <parameter key="PhiF_1" value="35" comment="Internal friction angle of the phase" />
    <parameter key="YieldStrength_1" value="0" comment="Yield strength of phase if constant Yield strength is being used (def=0)" />
   <parameter key="Phase_1" value="1" comment="Newtonian=0, Non-Newtonian" />
</parameters>
```

Figure 2. Parameters setup in the example *XML*.

Summarising the parameters section of Figure 2, the first line refers to the number of phases (restricted to 2 in this BETA version). Next, the constants and properties of each phase are set equal to their corresponding value.

Note that this is a BETA version and although we have taken every possible precaution, the two-phase model may underperform in certain citations that it was not designed to model.

References:

1. Crespo, A.J.C., *et al.*, *DualSPHysics: Open-source parallel CFD solver based on Smoothed Particle Hydrodynamics (SPH).* Computer Physics Communications, 2015. **187**: p. 204-216.
2. Fourtakas, G., B.D. Rogers, and D.R. Laurence. *3-D SPH Modelling of Sediment Scouring Induced by Rapid Flows*. in *9th International SPHERIC SPH Workshop*. 2014. Paris, France.
3. Fourtakas, G., B.D. Rogers, and D.R. Laurence, *Modelling sediment resuspension in industrial tanks using SPH.* Houille Blanche-Revue Internationale De L Eau, 2013(2): p. 39-45.