interDebrisFoam

Contents

1 Introduction

The *interDebrisFoam* is a solver for three incompressible fluids, two of which are miscible, that uses the Volume of Fluid (VOF) method to capture the interface. For example, slurry and gravels are mixed in the debris flow, and there is an interface between the mixture and the air. The interaction between the slurry and gravels is modeled based on velocity slip. Two rheology models named Herschel-Bulkley-Papanastasiou (H-B-P) model for the slurry phase and the Coulomb-viscoplastic model von Boetticher et al. (2016) for the gravel phase are given in this solver.

2 How to install

The *interDebrisFoam* is based on OpenFOAM, a free, open-source, parallel processing software backed by a large user-driven support community, vertion 6.0. It is necessary to install OpenFOAM-6.0 to install *interDebrisFoam*.

2.1 Installing OpenFOAM-6.0

The OpenFOAM foundation has given an installation tutorial for OpenFOAM-6.0 at Installation of Open-FOAM.

2.2 Installing *interDebrisFoam*

After a successful OpenFOAM-6.0 installation, *interDebrisFoam* can be downloaded and installed.

2.2.1 Downloading the code

The code of *interDebrisFoam* can be downloaded from the supplement files. Extracting the folder "interDebrisFoam" into the $HOME directory.

2.2.2 Compiling the rheology models

1 Change to the folder where the rheology models is

```
cd $HOME/interDebrisFOAM/solver/viscosityModels/
```

2 Type

```
wmake
```

2.2.3 Compiling the relativevelocity model

1 Change to the folder where the relativevelocity models is

```
cd $HOME/interDebrisFOAM/solver/relativeVelocityModels/
```

2 Type

```
wmake
```

2.2.4 Compiling the solver

1 Change to the solver folder

```
cd $HOME/interDebrisFOAM/solver/
```

2 Type

```
wmake
```

Then, the solver *interDebrisFoam* is installed.

3 Getting started

The debris flow case in the Aiwa Watershed is included as an example case in the $HOME/interDebrisFoam/run/ folder. In order for users to better test the case, the number of cells in this case has been reduced.

   1 Change to the run folder

```
cd \$HOME/interDebrisFoam/run/
```

   2 Type

```
interDebrisFoam
```

For parallel computation

   1 Change to the run folder

```
cd \$HOME/interDebrisFoam/run/
```

   2 Type

```
decomposePar
```

   3 Using the parallel commands to start.

The material properties such as the density, the viscosity, the relative velocity, and so, are set in "constant/transportProperties". The time step size and writting interval are set in "system/controlDict".

References

von Boetticher, A., Turowski, J. M., McArdell, B. W., Rickenmann, D., and Kirchner, J. W. (2016). Debrisintermixing-2.3: a finite volume solver for three-dimensional debris-flow simulations with two calibration parameters – part 1: Model description. *Geoscientific Model Development*, 9(9):2909–2923.