

## ReadMe: installation instructions for debrisInterMixing-6

This is a short introduction to compile the Open-Source code debrisInterMixing developed at ETH Zürich for the simulation of viscous debris flows. For a description of the method and implementation, please see *von Boetticher, A., Turowski, J. M., McArdell, B. W., Rickenmann, D., and Kirchner, J. W.: DebrisInterMixing-2.3: a finite volume solver for three-dimensional debris-flow simulations with two calibration parameters – Part 1: Model description, Geosci. Model Dev., 9, 2909-2923, <https://doi.org/10.5194/gmd-9-2909-2016>, 2016.*

### 1 OpenFOAM-6

The debris flow solver debrisInterMixing-6 is based on the open-source CFD code OpenFOAM<sup>®</sup>, version 6.0. To install debrisInterMixing-6, it is necessary to compile OpenFOAM-6.

The following text is a selection of copied installation instructions for OpenFOAM-6 from the OpenFOAM Foundation, <http://www.openfoam.org> and the OpenFOAMWiki, <http://openfoamwiki.net>.

Downloading the source code for OpenFOAM-6 is straight forward. However, compiling OpenFOAM-6 needs some system-specific steps as described at <https://openfoamwiki.net/index.php/Installation/Linux/OpenFOAM-6/Ubuntu> or, for other Linux distributions:

<https://openfoamwiki.net/index.php/Installation/Linux/OpenFOAM-6>

#### 1.1 Git Software

The sources can be obtained using the Git open source version control system that is commonly available on Linux distributions. Those with administrator permission can install the git software, e.g. by first switching to root user with sudo by typing in a terminal

```
sudo su –
```

On Ubuntu, git can be installed by executing the following command in a terminal prompt

```
apt-get install git-core
```

Similarly it can be installed on OpenSuSE by the following command:

```
zypper install git-core
```

Also it can be installed on RHEL by the following command.

```
yum install git
```

## 1.2 Downloading the Sources

The user should choose a directory location to download and unpack the files, which will become the installation directory of OpenFOAM. If the installation is for a single user only, or if the user does not have root access to the machine, we would recommend the installation directory is  $\$HOME/OpenFOAM$  (i.e. a directory OpenFOAM in the user's home directory). If the installer has root permissions and the installation is for more than one user, one of the 'standard' locations can be used, e.g.  $/usr/local/OpenFOAM$ ,  $/opt/OpenFOAM$ , or just  $/opt$ . After the installation directory is chosen (and, if necessary, created) download and unpack OpenFOAM-6 and ThirdParty-6 as follows.

### 1.2.1 OpenFOAM-6

The OpenFOAM-6 repository is available at <https://github.com/OpenFOAM/OpenFOAM-6>. For downloading the sources follow the steps in section 1.3.

### 1.2.2 ThirdParty

ThirdParty includes necessary software for pre- and post-processing. For downloading the sources follow the steps in section 1.3.

## 1.3 Compiling OpenFOAM

We recommend Ubuntu and to follow the steps at

<https://openfoamwiki.net/index.php/Installation/Linux/OpenFOAM-6/Ubuntu>

for Ubuntu or

[https://openfoamwiki.net/index.php/Installation/Linux/OpenFOAM-6/CentOS\\_SL\\_RHEL](https://openfoamwiki.net/index.php/Installation/Linux/OpenFOAM-6/CentOS_SL_RHEL)

or

<https://openfoamwiki.net/index.php/Installation/Linux/OpenFOAM-6/openSUSE>

## 2 debrisInterMixing-6

After a successful OpenFOAM-6 installation in  $\$HOME/OpenFOAM/$ , debrisInterMixing-6 can be downloaded and installed.

The debrisInterMixing solver is based on an adaptation of the interMixingFoam solver and is used in combination with two rheology models, named HerschelBulkleyDebrisFlow for the slurry and PudasainiCoulombViscoPlastic for the gravel phase. In order to make the solver and the viscosity models available within the OpenFOAM installation, one needs to download and compile the corresponding code in a folder 'userName-6' at  $\$HOME/OpenFOAM/$ , where 'userName' should be the user's name. A user named henry would need to download & compile the code in a folder  $\$HOME/OpenFOAM/henry-6$ .

### 2.1 Downloading the Sources

Download and extract the supplement and move the folder "applications" into the  $\$HOME/OpenFOAM/'userName-6'$  directory.

### 2.2 Compiling the solver

In a terminal window, change to the folder (cd command)  $\$HOME/OpenFOAM/'userName-6'/applications/solvers/multiphase/interFoam$

It might be necessary to type

of6

in the terminal window to make OpenFOAM-6 available.  
Then type:

```
wmake debrisInterMixingFoam
```

to compile the solver. Typing debrisInterMixingFoam after this should give an output starting with the OpenFOAM header and ending with FOAM exiting following the IO ERROR in case the installation was successful.

### 2.3 Compiling the rheology models

Now you have a solver that can handle three phases, the first phase stays separated and the latter two can be mixed. The phases are solved by a single volume-averaged Navier-Stokes equation, and besides the forces at the surface between the mixed and non-mixed phase there is no momentum exchange between phases due to an assumed high drag between the mixing phases. The debris flow physics are introduced by two rheology models, one that accounts of the shear-thinning behavior of fine sediment suspensions and one that accounts for the pressure-dependent coulomb friction between shearing gravel grains.

To install these, change to the folder `$HOME/OpenFOAM/'userName-6'/applications/transportModels/` and type:

```
wmake incompressible
```

or

```
wmake libso incompressible
```

In case of difficulties it may help to replace the folder `$HOME/OpenFOAM/'userName-6'/applications/transportModels` with the folder `$HOME/OpenFOAM/OpenFOAM-6/src/transportModels`. This will overwrite the general rheology source code with the one from your actual OpenFOAM installation and thus account for local settings. However, you now need to insert the debris flow rheology models of debrisInterMixing into the source code once again, which means some handwork: Manually copy the folders HerschelBulkleyDebrisFlow and PudasainiCoulombViscoPlastic from the applications/transportModels/incompressible/viscosityModels directory from the debrisInterMixing download folder to the `$HOME/OpenFOAM/'userName-6'/applications/transportModels/incompressible/viscosityModels` folder (userName stands for your user name). Now you need to add the lines

```
viscosityModels/HerschelBulkleyDebrisFlow/HerschelBulkleyDebrisFlow.C  
viscosityModels/PudasainiCoulombViscoPlastic/PudasainiCoulombViscoPlastic.C
```

before `transportModel/transportModel.C` in the 'files' file in the directory `HOME/OpenFOAM/'userName-6'/applications/transportModels/incompressible/Make`.

Now try again and Change to the folder `$HOME/OpenFOAM/'userName-6'/applications/transportModels/` and type

```
wmake incompressible
```

In case compiling the viscosity models works without errors, but the solver can't find the new viscosity models when getting started as in the following section, one can copy the folders `viscosityModels/HerschelBulklesDebrisFlow` and `viscosityModels/PudasainiCoulombViscoPlastic` to

```
$HOME/OpenFOAM-6/src/transportModels/incompressible/viscosityModels/
```

and add the lines

```
viscosityModels/HerschelBulkleyDebrisFlow/HerschelBulkleyDebrisFlow.C  
viscosityModels/PudasainiCoulombViscoPlastic/PudasainiCoulombViscoPlastic.C
```

before `transportModel/transportModel.C` in the 'files' file in the directory `HOME/OpenFOAM/OpenFOAM-6/transportModels/incompressible/Make`.

Then go to `HOME/OpenFOAM/OpenFOAM-6/transportModels` in a terminal and type

```
wmake incompressible
```

(after sourcing OpenFoam by typing of6)

### 3 Getting started

The calibration experiment of Hürlimann et al. (2015) used in von Boetticher et al. (2015b) is included as an example case in the `applications/run` folder.

In a terminal, change the directory to

```
$HOME/OpenFOAM/'userName-6'/applications/run/10_4060_0,285
```

and type the three commands:

```
blockMesh  
setFields  
debrisInterMixingFoam
```

to run the case in serial. For parallel computation please refer to the OpenFOAM tutorials at [www.openfoam.org](http://www.openfoam.org). Time step and writing interval settings are defined in `system/controlDict`, material properties are set in `constant/transportProperties`.

## 4 References

- [1] Hürlimann, M., McArdell, W., and Rickli, C.: Field and laboratory analysis of the runout characteristics of hillslope debris flows in Switzerland, *Geomorphology*, 232, 20–32, doi:10.1016/j.geomorph.2014.11.030,2015.
  
- [2] v. Boetticher, A., Turowski, J. M., McArdell, W. B., Rickenmann, D., Hürlimann, M., Scheidl, C., and Kirchner, J. W.: DebrisInterMixing-2.3: A Finite Volume solver for three dimensional debris flow simulations based on two calibration parameters. Part two: model validation with experiments. *Geoscientific Model Development*, 10, 11: 3963-3978. doi: 10.5194/gmd-10-3963-2017

## 5 Support

For more information and assistance, please contact

Dr. Albrecht v. Boetticher  
[albrecht.vonboetticher@wasserbau.ch](mailto:albrecht.vonboetticher@wasserbau.ch)  
**Staubli, Kurath & Partner AG**  
Ingenieurbüro SIA USIC  
8048 Zürich · Bachmattstrasse 53 · Tel. 043 336 40 50  
[www.wasserbau.ch](http://www.wasserbau.ch)