OpenFOAM-2.3.x

The debris flow solver debrisInterMixing-2.3 is based on the open-source CFD code OpenFOAM, version 2.3.x. To install debrisInterMixing-2.3, it is necessary to compile OpenFOAM-2.3.x.

The following text is a selection of copied installation instructions for OpenFOAM-2.3.x from the OpenFOAM Foundation, http://www.openfoam.org/

Downloading the source code for OpenFOAM-2.3.x should be possible following the instructions. However, compiling OpenFOAM-2.3.x needs some system-specific steps as described at http://www.openfoam.org/download/git.php

Version 2.3.x: This page describes how to download the source code of OpenFOAM-2.3.x

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

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-2.3.x and ThirdParty-2.3.0 as follows.

OpenFOAM-2.3.x

The OpenFOAM-2.3.x repository is available at https://github.com/OpenFOAM/OpenFOAM-2.3.x. From the OpenFOAM installation root directory the repository can be obtained using the https protocol (https://):

- git clone https://github.com/OpenFOAM/OpenFOAM-2.3.x

Note: Copy-pasting the link in Ubuntu may cause a difference in the '-' character. One may need to search under https://github.com/OpenFOAM for the correct link. Furthermore, while the TCP port for https is rarely blocked by a firewall, download hangs have been experienced; upgrading to git version 1.7 seems to overcome this problem.

Either of the commands above will create an OpenFOAM-2.3.x directory that the user can subsequently be updated to the latest published copy using

- cd OpenFOAM-2.3.x
- git pull

Please compile OpenFOAM-2.3.x in analogy to the instructions given in http://www.openfoam.org/download/git.php but please rename the ThirdParty folder to ThirdParty-2.3.x and palce it in the OpenFOAM directory and add

source $HOME/OpenFOAM/OpenFOAM-2.3.x/etc/bashrc

OR

source $HOME/OpenFOAM/OpenFOAM-2.3.x/etc/cshrc

to your $HOME/.bashrc or $HOME/.cshrc file

After a successful OpenFOAM-2.3.x installation in $HOME/OpenFOAM/, debrisInterMixing-2.3 can be downloaded and installed:

debrisInterMixing-2.3

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-2.3.x' 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-2.3.x.

Downloading the Sources

Download and extract the supplement and move the folder "applications" into the $HOME/OpenFOAM/'userName-2.3.x' directory. Updates and news can be found on http://www.wsl.ch/fe/gebirgshydrologie/downloads/index_DE

Compiling the solver

In a terminal window, change to the folder (cd command) $HOME/OpenFOAM/'userName-2.3.x'/applications/solvers/multiphase/interFoam

Then type

wmake debrisInterMixingFoam

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

Compiling the rheology models

Change to the folder $HOME/OpenFOAM/'userName-2.3.x'/applications/transportModels/ and type

wmake incompressible

to make HerschelBulkleyDebrisFlow and PudasainiCoulombViscoPlastic available.

In case of difficulties it may help to replace the folder

$HOME/OpenFOAM/'userName-2.3.x'/applications/transportModels

with the folder

$HOME/OpenFOAM/OpenFOAM-2.3.x/src/transportModels

and manually copy the folders HerschelBulkleyDebrisFlow and PudasainiCoulombViscoPlastic from the applications/transportModels/incompressible/viscosityModels

directory of the supplement to the

$HOME/OpenFOAM/'userName-2.3.x'/applications/transportModels/incompressible/viscosityModels

folder. But in this case one needs 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-2.3.x'/applications/transportModels/incompressible/Make

Getting started

The calibration experiment of Hürlimann et al. (2015) used in von Boetticher et al. (2017) is included as an example case in the applications/run folder.

In a terminal, change the directory to

$HOME/OpenFOAM/'userName-2.3.x'/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. Time step and writing interval settings are defined in system/controlDict, material properties are set in constant/transportProperties. A coarser mesh with doubled cell sizes in all directions can be created by deleting the constant/polyMesh/blockMeshDict, renaming blockMeshDict_coarse to blockMeshDict. If the dense mesh case was created before, the 0 directory needs to be replaced with the initial 0 directory as downloaded.

References:

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.

v. Boetticher, A., Turowski, J. M., McArdell,W. B., Rickenmann, D., Hürlimann, M., Scheidl, C., and Kirchner, J. W.: (submitted) DebrisInterMixing-2.3: a finite volume solver for three-dimensional debris-flow simulations with two calibration parameters - Part 2: Model validation, Geoscientific Model Development, 2017.