A multiphaseInterFoam Tutorial

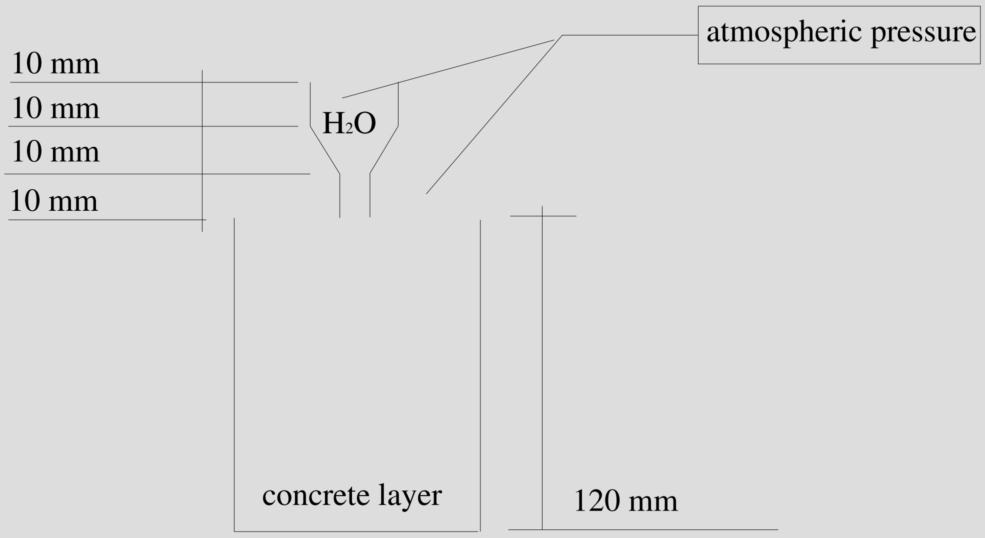
Annika Gram

- multiphaseInterFoam
- the Case
- Code
- Results

multiphaseInterFoam

- based on the VOF method
- Gamma
- incompressible
- multi phase
- transient

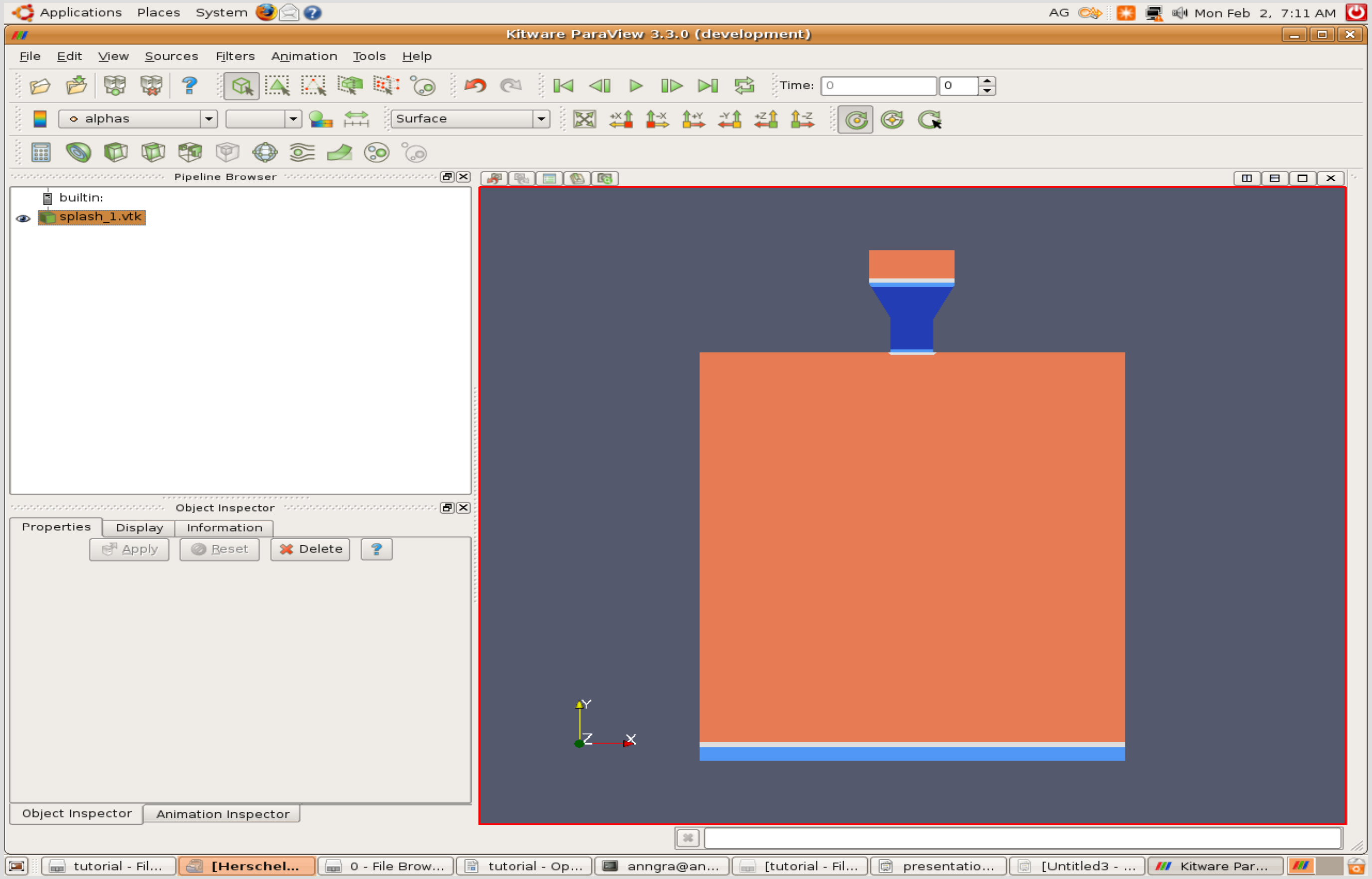
the Case


atmospheric pressure
10 mm
10 mm
10 mm
10 mm
H_2O
concrete layer
120 mm

the Case

$$\tau = \tau_0 + \mu_{pl}\, \gamma^n$$

the Case


Applications Places System
Mon Feb 2, 7:11 AM
Kitware ParaView 3.3.0 (development)
File Edit View Sources Filters Animation Tools Help
Time: 0
alphas
Surface
Pipeline Browser
builtin:
splash_1.vtk
Object Inspector
Properties Display Information
Apply Reset Delete
Object Inspector Animation Inspector

Code

`$FOAM_RUN/splash`

In this directory the following folders are to be seen:

`0` holding files alphas, alphaair, -crete, -newcrete, -water, pd and U as well as subdirectory hide for the original alpha files

`constant` holding files dynamicMeshDict, environmentalProperties and transportProperties
as well as subdirectory polyMesh for the files blockMeshDict, boundary, faces, neighbour, owner and points

`system` holding files controlDict, decomposeParDict, fvSchemes, fvSolution, setFieldsDict

Code

`$FOAM_RUN/splash/0`

Intitial and boundary conditions are specified in the `0` directory. The following holds for this case:

Variable	Initial Conditions
pd	zeroGradient, fixedValue and uniform 0 for atmosphere
U	fixedValue uniform (0 0 0), inletOutlet for atmosphere
	alphas
	zeroGradient
aphaair	alphaContactProperties
	thetaProperties 90 0 0 0 inletOutlet for atmosphere
	uniform 0

Code

`$FOAM_RUN/splash/constant`

Material properties can be found in the `constant` directory.

File	Initial Conditions
environmentalProperties	gravity
transportProperties	material constants

Code

`$FOAM_RUN/splash/system`

Material properties can be found in the `constant` directory.

<u>File</u>

`setFields`

`controlDict` computation time 0.2 s

Code

myHerschelBulkley.C

```
return (min(nu0_,(tau0_ + k_*(pow(sr(), n_)

        - pow(tau0_/nu0_,n_))) / (max(sr(),
dimensionedScalar

        ("VSMALL", dimless/dimTime, VSMALL)))));
```

Code

```
cd
$WM_PROJECT_USER_DIR/src/transportModels/incompr
essible/Make

wmake libsowmake libso
```

Code

```
cd $FOAM_RUN/splash



blockMesh
checkMesh
setFields
multiphaseInterFoam
```

Results

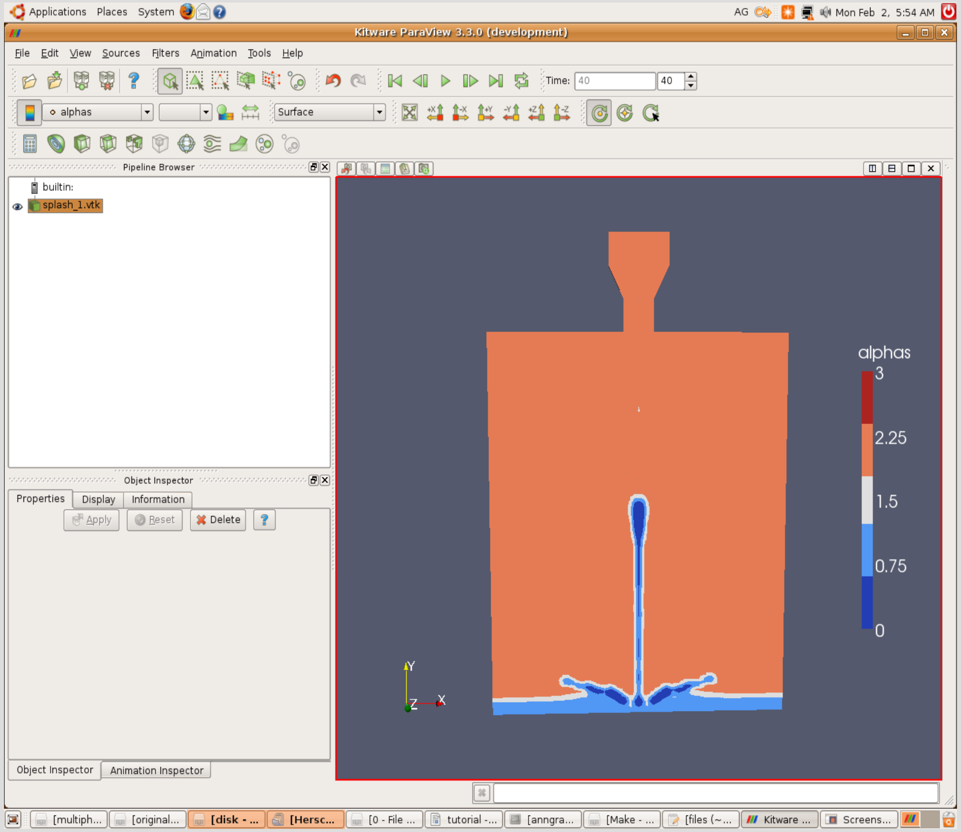
```
cd $FOAM_RUN/splash

paraFoam
```

Results

```
cd $FOAM_RUN/splash
```

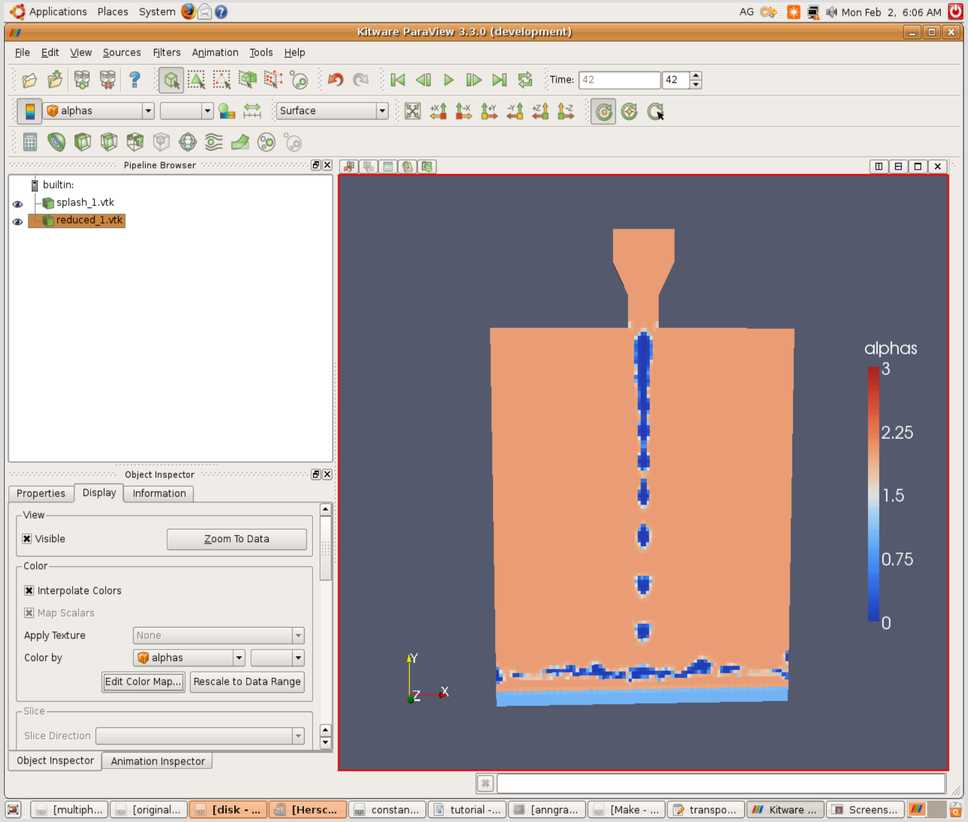
```
paraFoam
```



Results

```
cd $FOAM_RUN/splash
```

```
paraFoam
```



Results

- changing transportProperties
- adding phases

Questions