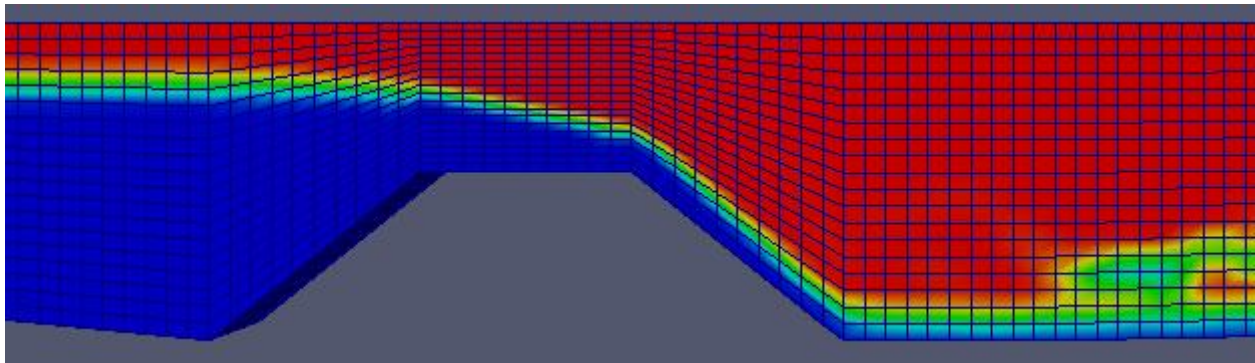


## ***OpenFOAM* homework**

*OpenFOAM* is the most popular open-source CFD program in the world today. In this homework we will use the program to determine the coefficient of discharge for a broad-crested spillway.



*OpenFOAM* is considered to be a difficult program to learn. This is partially due to that it is designed to run on *Linux*, and partly that the grid generation tools are complicated. In the current homework, we will use a Windows version of *OpenFOAM*. Instead of using the *OpenFOAM* grid editor, *snappyHexMesh*, we will use *SSIM 2*. This will make the learning curve less steep, and hopefully make it possible to finish the homework in a reasonable time (around 3-10 hours).

The homework is divided into different parts, which should be carried out sequentially.

### **1. Installation of blueCFD**

*BlueCFD* is an *OpenFOAM* version for *Windows*. It can be downloaded from <http://code.google.com/p/bluecfd-singlecore/>. The download can be done by clicking on the left side of the page: *blueCFD-SingleCore-2.0-1-setup.exe*. Download the program and install it. It is possible to install newer versions, but there have been some problems with these versions, so it is recommended to use version 2.0-1.

The post-processor in *OpenFOAM* is called *ParaView*. This can be downloaded from <http://www.paraview.org>. The *OpenFOAM* window calls *ParaView* through a file called *ParaFoam.bat*. To make *ParaView* accessible when the file is called, you can add the following line at the top of the *ParaFoam.bat* file: `set PATH=%PATH%; c:\Program Files\ParaView 3.4.12\bin`

This will only work if you have installed *ParaView* version 3.4.12 in the directory *Program Files*. If you have installed another version in another directory, you have to modify the line accordingly.

The *ParaFoam.bat* file is found in the directory

```
c:\Program Files\blueCFD-SingleCore-2.0\OpenFOAM-2.0\bin.
```

## 2. Running a tutorial

After installation, you have a program *blueCFD-SingleCore-2.0* installed on your PC. Click on the program, and on the directory *OpenFOAM*, and choose the option *OpenFOAM Users Guide*. This is a PDF file. You can start with running the *Dam Break* tutorial, described in Chapter 2.3 in the *Users Guide*. The tutorial is carried out by opening the *Command Line with ming-w32 DP*, which is in the *blueCFD-SingleCore-2.0* program. This command prompt is used to start the *OpenFOAM* solvers, grid generators and post-processing programs.

In the command prompt, use the command *cd* to go to the directory: *OpenFOAM-2.0 -> tutorials -> multiphase -> interFoam -> ras*. Copy the input files using the command

```
xcopy /S damBreak c:\damBreak\.
```

Answer *D* on the question about the directory. After the files are copied, go to the root directory with the command *cd \*. Then start the grid generator by the command

```
blockMesh -case damBreak
```

After the grid is made, initiate the variables with the command

```
setFields -case damBreak
```

Then do the computation by writing the command

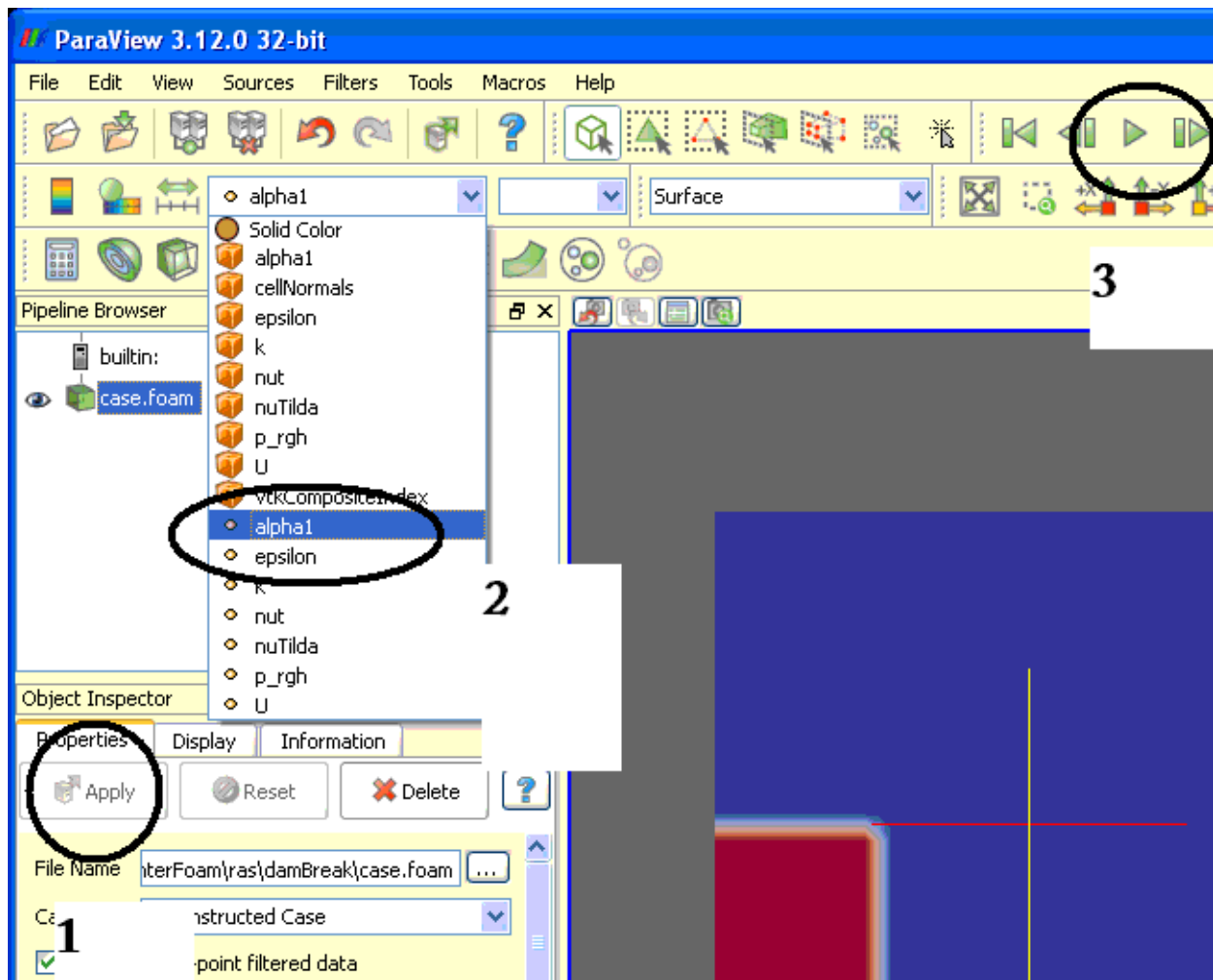
```
interFoam -case damBreak
```

The program then runs for a while. After it has stopped, view the result by the command

```
paraFoam -case damBreak
```

After the *ParaFoam* window opens, click on the green button *Apply*. Then click on the box where it says *p*, and choose *alpha1*. Then click on the triangular green “play” button at the top of the menu.

The buttons are circled in the figure on the next page, where also the order they should be pushed is given with numbers.

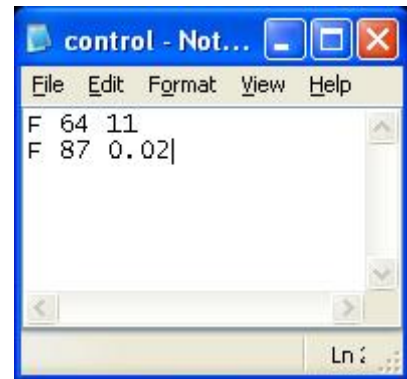


### 3. Making the grid for the spillway

One of the most effective ways of learning new computer programs is to run a tutorial and modify it so that it fits the problem we want to solve. We will modify the *damBreak* tutorial so that it contains the grid of the spillway. The spillway will be made with *SSIM 2*.

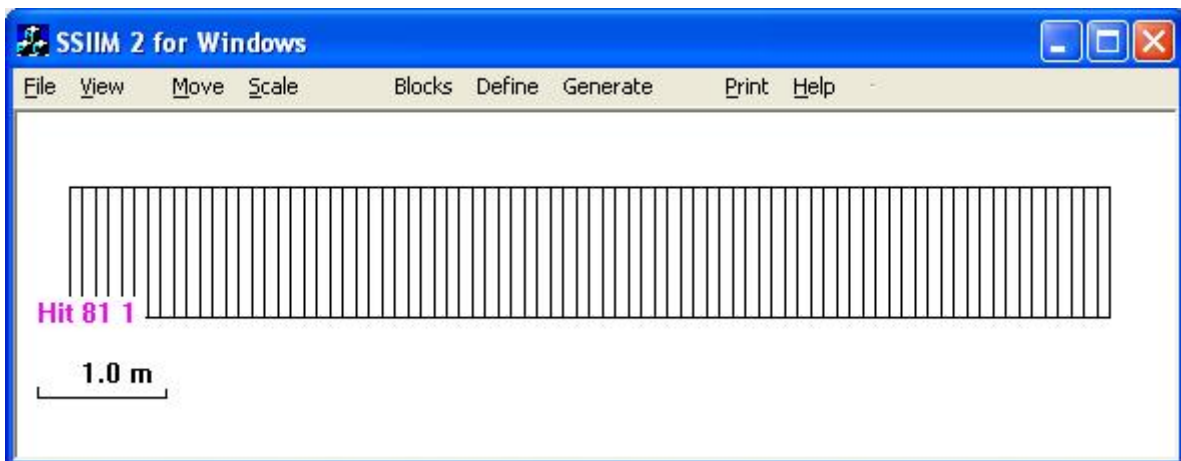
First, install *SSIIM 2* on your computer. It can be downloaded from the web page <http://folk.ntnu.no/nilsol/ssiim>. Choose the latest 32 bits version. Remember to download the DLL files and unzip these in the same directory as you used for the executable. You may for example use the directory *c:\ssiim\*. Make a control file, and add the following options:

```
F 87 0.02      uniform distribution of vertical cells
F 159 0 0 0 1 0 0  2d grid allowed
G 1 100 11 21 1   allocation of arrays
```



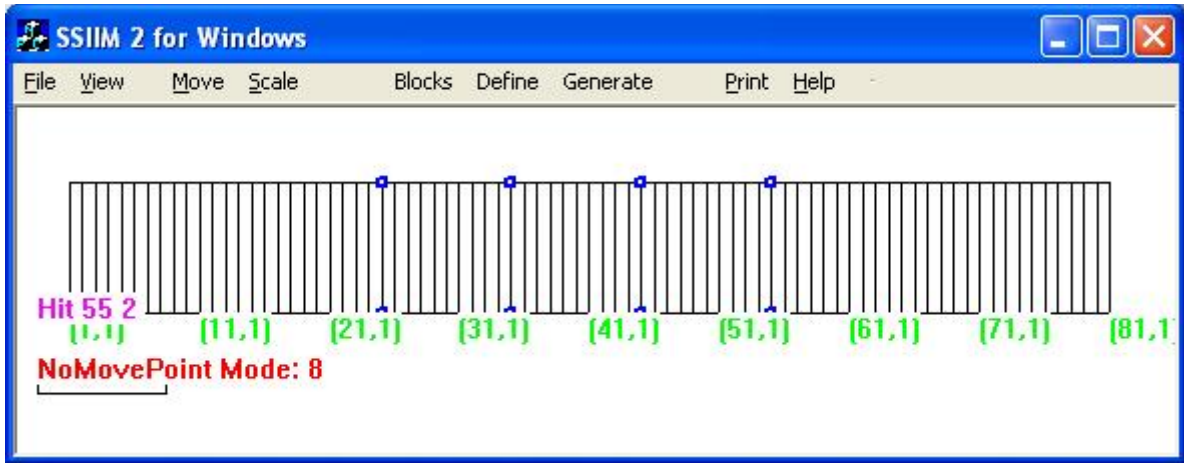
Note that the control file must not have any *.txt* or other extension. If the file has a *.txt* extension, it has to be renamed.

Then start the program. A window comes up. Go to the menu option *View -> GridEditor*. We now make the plan view of the grid, which will be a channel that is 8 meters long and 1 meter wide. Click on *Blocks -> Add block*, and then click on four points in the white area, roughly where the four corners of the channel is. In plan view. No accuracy needed here. The points have to be given clockwise, otherwise an error message is given. After you see the green outline of the channel, choose the menu option *Blocks -> Size block*. Choose 81 lines in *i*-direction and 2 in *j*-direction. Then move the four corners to their exact points. This is done by first clicking on one point and then choosing the menu option *Define -> Give coordinates*. Give the coordinates (0,0), (0,1), (8,1) and (8,0). Then choose the menu option *Generate -> boundary*. The outline of the channel is then given.

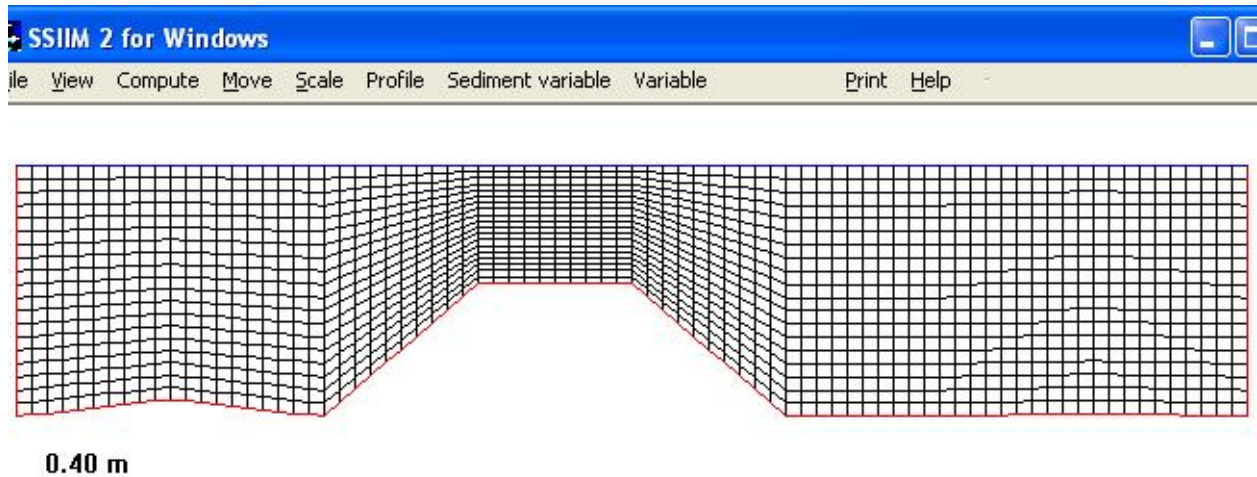


Next, we want to make the geometry of a broad-crested weir (<http://folk.ntnu.no/nilsol/cases/langkronet>). First, choose in the menu *View -> Legend*. This shows the grid line numbering. Then choose in the menu *Define -> NoMovePoints mode*. Click on the grid line intersections for lines *i*=25, *i*=35, *i*=45 and *i*=55. Note that there are two intersection for each *i*. So you will get 8 blue markers on the grid. If you make a mistake, choose the menu option *Define -> Delete NoMove-*

Point. After this is ok, choose the menu option *Define -> SetNoPointMode*. This will allow you to click on the grid without adding new *NoMovePoints*.



Then we will give an elevated level for the crest of the weir. This is done by clicking on the four central blue *NoMovePoints*, and giving a level by the menu option *Define -> Give coordinates*. Choose the *z* level 0.6 meters. Afterwards, choose the menu option *Generate -> Boundary*. This creates straight lines between the *NoMovePoints*. Then choose *Generate -> 3D grid*. Save the grid by choosing *File -> Write unstruc file*. End the program and restart it. Choose the menu option *File -> Read unstruc file*. You can see the grid by choosing *View -> Profile*.



Before making the OpenFOAM files, the discharge in and out have to be defined. This is done by *View-> DischargeEditor*. The grid seen from above emerges. Choose *Side Discharge -> Choose Group -> 1*. Then *Side Discharge -> Give values*. A dialog box emerges, where the water discharge is specified in  $\text{m}^3/\text{s}$ . Give a value of 1.0 for the inflow discharge. Then choose *OK*. Then click with the mouse on the left sides of the channel, at section 1. Once clicked, the lines change

colour. Then choose *Side Discharge* -> *Choose Group* -> 2. Then *Side Discharge* -> *Give values*. In the dialog box, click on the *Inflow* button. This specifies an outflow of the geometry. Give a value of 1.0 for the discharge. Then choose *OK*. Then click on the right side of the grid and see this section change colour. Then save the *unstruc* file again and end the program.

#### 4. Making the input files for *OpenFOAM*.

The general idea now is to make the grid and boundary files for the SSIIM 2 geometry and copy these into the directory of the *damBreak* tutorial. First, edit SSIIM 2's control file and add the data set *F 298 0 1.0e20 0.24*. Restart SSIIM 2 and read the *unstruc* file. Note that version 142 or later must be used. Then choose on the menu *File* -> *Write OpenFOAM files*. This will make the OpenFOAM grid files and boundary files. Copy the *blockMeshDict* file from the *ssiim* directory to the directory *damBreak/constant/polymesh*. Copy the files *U*, *alpha1*, *k*, *epsilon*, *p\_rgh* and *nut* to the directory *damBreak/0*. You have to specify the inflow and outflow water velocity in the *U* file. Edit the *U* file and give an inlet velocity of 0.4 m/s, instead of the default 0.3. Also, you need to specify the outlet velocity. This is done by using the continuity equation and the inflow and outflow areas given at the top of the file. Divide the *inlet* area with the *outlet1* area. Multiply this with 0.4 and give this number as the velocity in the *outlet1* boundary field. Use many decimals, otherwise the program will give a continuity error. Set the *outlet2* velocity to zero. The *outlet1* area is lower than *outlet2*, so this will prevent high downstream water levels. The last value on the *F 298* data set in the SSIIM 2 *control* file is the level between the two outlet areas.

#### 5. Running OpenFOAM

Open the command prompt from *blueCFD-SingleCore-2.0* as for the tutorial, and do the same commands for grid generation, running the program and post-processing. Before you do this, you need to edit the gravity file called *g*, to get the gravity to be in the *z* direction instead of the *y* direction. Use the *Wordpad* for doing this. The *g* file is located in the *damBreak/constant* directory. Change the vector  $(0 -9.81 0)$  to  $(0 0 -9.81)$ .

If you want to make the program run for a longer time, you have to increase the running time by editing the *controlDict* file in the *damBreak/system* directory.

If you encounter the error "*pRefCell or pRefValue not found*" when running *interFoam*, then you need to add some data to an input file. In the directory *damBreak/system*, there is a file called *fvSolution*. Open this file in an editor, for example *WordPad*. In the file, there is a section called *PIMPLE*. In the brackets enclosing this section, add the following two lines:

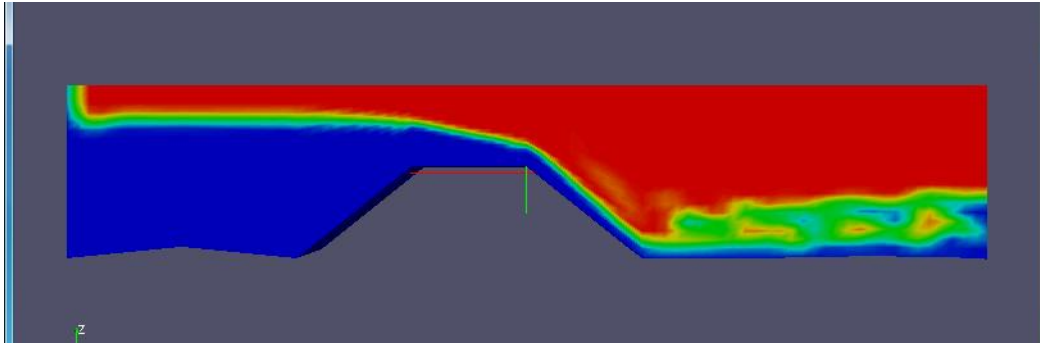
```
pRefCell 0
pRefValue 0
```

Save the file and try running the *interFoam* again.

If you get error messages for files that we are not using, for example *alpha1.org* and *nuTilda*, then delete these files.

When looking at the results with *paraFoam*, you may have to rotate the figure with the mouse, or use one of the buttons on the menu to see the spillway from the side. Also, you may need to rescale the colours. Choose the variable  $U$  first and then *alpha1*.

## 6. Determining the coefficient of discharge



*OpenFOAM* computes the water fraction in each cell, which is given by the parameter *alpha1*. The coefficient of discharge can be computed from the water discharge and the water level behind the weir. The water discharge is given as the inflow velocity times the inflow area. The water level behind the weir is more problematic to find. Approximate values can be taken from the graphics. For more accurate values, it is necessary to find where the line of  $\alpha = 0.5$  is located. This must be interpolated from values in the grid. To facilitate this interpolation, SSIIM writes a file called *levels*, which gives the cell number and its level in all the grid cells. This file can be taken into a spreadsheet, together with the *alpha1* values at the last time step from *OpenFOAM*. The values from the *alpha1* file and *levels* file will correspond. A figure showing the *alpha1* file in *WordPad* is given at the end of this document. Viewing the graphics of cell numbers in SSIIM simultaneously as the *paraFoam* graphics, enables a rough choice of cells where  $\alpha = 0.5$  is located. The more accurate value can then be interpolated using the spreadsheet with the *levels/alpha1* values. ‘

The figure on the next page shows the cell numbers for the grid as given by *SSIIM 2*.



	19	38	57	76	95	114	133	152	171	190	209	228	247	266
	18	37	56	75	94	113	132	151	170	189	208	227	246	265
	17	36	55	74	93	112	131	150	169	188	207	226	245	264
	16	35	54	73	92	111	130	149	168	187	206	225	244	263
	15	34	53	72	91	110	129	148	167	186	205	224	243	262
	14	33	52	71	90	109	128	147	166	185	204	223	242	261
	13	32	51	70	89	108	127	146	165	184	203	222	241	260
	12	31	50	69	88	107	126	145	164	183	202	221	240	259
	11	30	49	68	87	106	125	144	163	182	201	220	239	258
	10	29	48	67	86	105	124	143	162	181	200	219	238	257
	9	28	47	66	85	104	123	142	161	180	199	218	237	256
	8	27	46	65	84	103	122	141	160	179	198	217	236	255
	7	26	45	64	83	102	121	140	159	178	197	216	235	254
	6	25	44	63	82	101	120	139	158	177	196	215	234	253
	5	24	43	62	81	100	119	138	157	176	195	214	233	252
	4	23	42	61	80	99	118	137	156	175	194	213	232	251
0.20	3	22	41	60	79	98	117	136	155	174	193	212	231	250
0.4	2	21	40	59	78	97	116	135	154	173	192	211	230	249
	1	20	39	58	77	96	115	134	153	172	191	210	229	248

The coefficient of discharge,  $C$ , is computed by using the following formula:

$$Q = CBH^{1.5}$$

where  $Q$  is the discharge,  $B$  is the width of the spillway and  $H$  is the vertical distance from the top of the spillway to the water level behind the spillway.



```
format      ascii;
class      volScalarField;
location   "20";
object     alpha1;
}
// * * * * *
// * * * * *

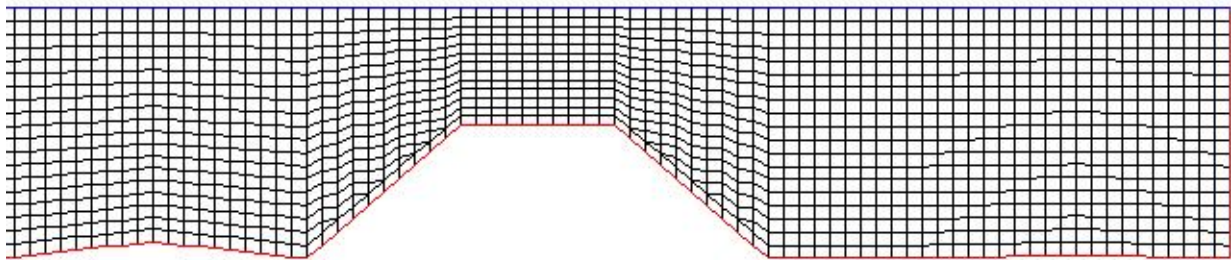
dimensions      [0 0 0 0 0 0];

internalField   nonuniform List<scalar>
1520
(
1 ← water fraction in cell 1
1
1
1
1
1
1
1
1
1
1
1
1
1
1
1
0.999995
0.999599
0.99151
0.926032
0.736555 ← water fraction in cell 17
0.653514
```

## 7. Further information

The accuracy of the results may be affected by the number of cells in the grid. It is possible to increase this. The maximum cells in the vertical direction is given on the *G 1* data set in the control file of *SSIIM 2*. It is also possible to investigate other numerical parameters, such as the time step, by modifying the *OpenFOAM* input files. For example the *controlDict* file.

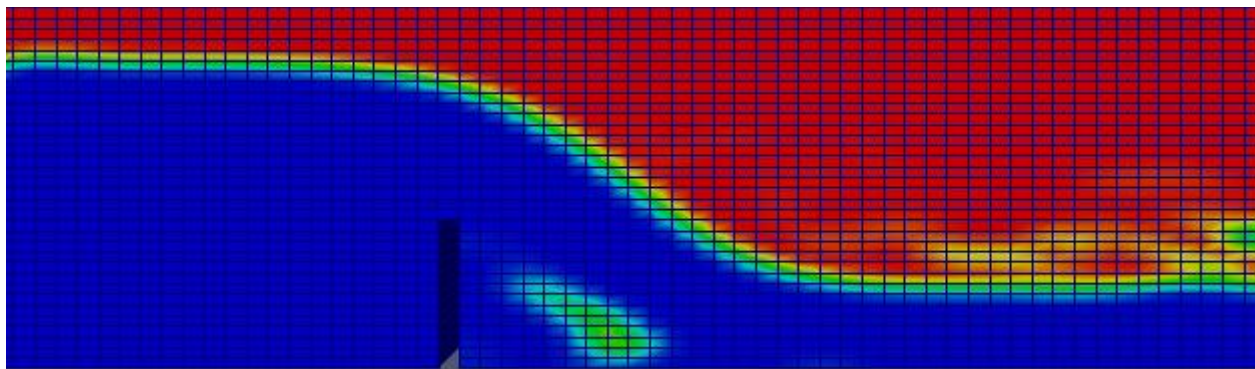
The *F 87* data set in the control file of *SSIIM 2* decides the distribution of the number of vertical cells as the depth varies. The low value given makes *SSIIM 2* generate a grid with equal amount of cells in the vertical direction over the whole channel. The default value of the *F 87* data set is 0.7, which gives some wedge cells in the grid. To avoid getting problems with the wedge cells in the *OpenFOAM* grid, the *F 299 1* data set has to be given in the *control* file of *SSIIM 2*.



0.50 m

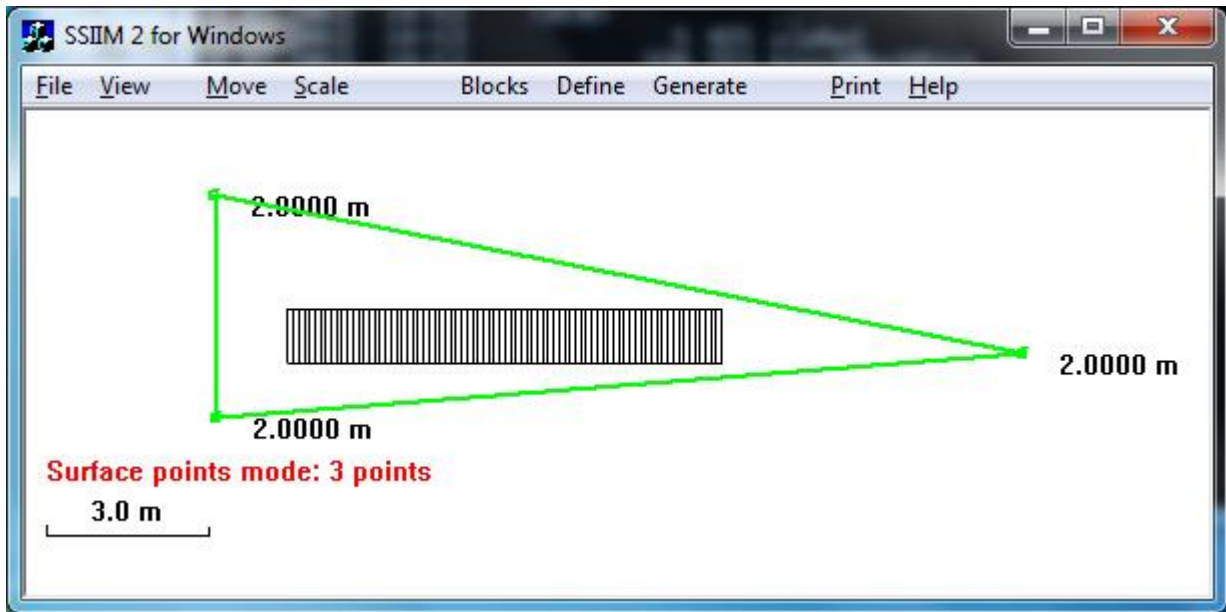
## 8. Vertical spillway walls

The *F 64 11* option makes the grid follow the ground without any vertical walls. To model flow over a vertical plate, the *F 64 0* option can be used. The grid will then be orthogonal, seen in a longitudinal profile. By giving higher bed levels at two neighbouring cross-sections, a wall will be made.



## 9. Level of top of the grid

The default level of the top of the grid in SSIIM 2 is 1 meter. This can be changed by giving in at least three surface points in the Grid Editor. These points are given by the menu option *Define -> Set surfacepoint*. Clicking with the mouse in the window will define a point. The three first points will form a triangle. Make sure the grid is inside the triangle. Then choose *Generate -> Surface* and *Generate -> 3D Grid*. A new grid is then made with the new top level of the grid.



## 10. Limitations

SSIIM 2 uses an unstructured grid, where it is possible to have varying number of grid cells in the vertical direction, depending on the water depth. This is done by including tetrahedral cells in the grid. Unfortunately, the tetrahedral grid cells seem to be problematic when transferring the grid from SSIIM 2 to OpenFOAM. At least if there is a 3D geometry, instead of a 2D like in the current example.