Homework: Sediment deposition in a sand trap

The purpose of this homework is to show the students how to compute the trap efficiency of a sand trap/desilting basin using a 3D CFD model. The geometry used in the homework is identical to a laboratory model study done by Olsen and Skoglund in 1994. In this study, the water velocities and sediment concentrations were measured in the flume. The velocities and concentrations that are computed can thereby be compared with the measurements.

Step 1: Getting the data and information

The homework should start by reading the article from the flume study, and looking at the photos from the web page: *http://folk.ntnu.no/nilsol/cases/doris*. (Doris is the name of the laboratory flume). The article reference is:

Olsen, N. R. B. and Skoglund, M. (1994) "Three-dimensional numerical modeling of water and sediment flow in a sand trap", IAHR Journal of Hydraulic Research, Vol. 32, No. 6. The article can be found on the web page:

http://www.tandfonline.com/doi/abs/10.1080/00221689409498693.

The SSIIM 2 program will be used in this homework. Download the 64 bits Windows version to your PC, from the web page: *http://folk.ntnu.no/nilsol/ssiim*. If you have a PC with 32 bits Windows, you have to use the 32 bit version of SSIIM. Then you also have to download the DLL's from the same web page and copy these to the same directory as SSIIM 2.

Step 2: Making the grid

The flume is a square box, 1 m wide, 1.54 m tall and 23.5 meters long. The entrance region is given on the picture below. The water flow direction is from left to right.

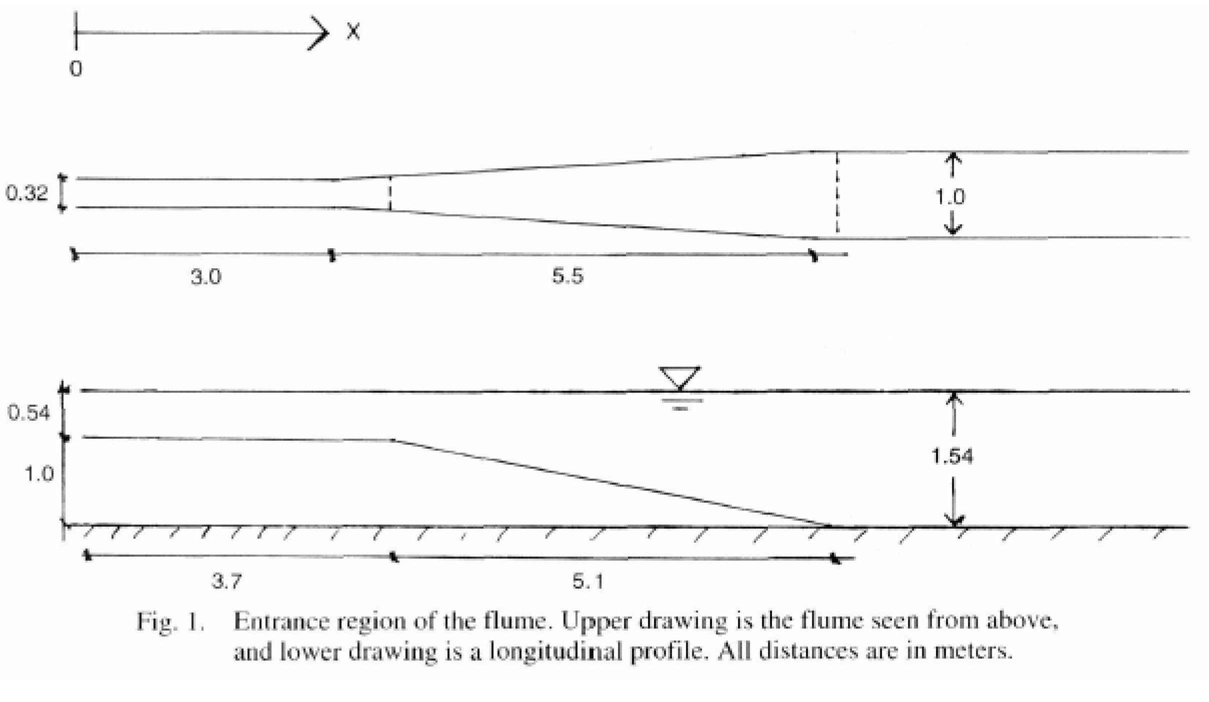

Fig. 1. Entrance region of the flume. Upper drawing is the flume seen from above, and lower drawing is a longitudinal profile. All distances are in meters.

Before you start, copy the *control.g2* file from the *http://folk.ntnu.no/nilsol/cases/doris* web page to your computer. To the same location as you run SSIIM 2 from. Rename the file to *control* without extension. Edit the file and remove the *F 2 YH* data set from the file before saving it.

Start SSIIM 2 and go to the *GridEditor.* This is done from the menu: *View->GridEditor.* Add a block from the menu, by choosing *Blocks->Add a block.* Then click on four points in the white area, in clockwise direction. The four points give the a box where the water is flowing into one side and out of the opposite side. The two other sides are walls. The first point should be the upstream right bank corner. The second point is the upstream left bank corner. The third point is the left bank downstream corner, and the fourth point is the right bank downstream corner. The exact location of the points is not important at this stage. Clicking the four points should give a green rectangle. If you choose the points in counter clockwise direction, you get an error message. Then choose the number of grid cells. This is done from the menu: *Blocks->Size block*. A dialog box emerges. Choose 236 lines in the i-direction and 21 lines in the j-direction. This means the cell sizes will be 23.5 m/(236-1) = 10 cm long. And 5 cm wide. Click *OK* and you see the grid is made.

The shape of the grid is now probably not what we want. When you made the grid by clicking on four points earlier, you made the four corners of the domain, seen from above. The first point you clicked on is the upstream right corner of the flume. This should have coordinates (0,0.34,1.0), according to Fig. 1. Click on this point in the figure, and then from the menu choose *Define -> Give coordinates*. A dialog box emerges, and you can give the coordinates x=0, y=0.34, z=1.0. Click *OK*, and you see the corner point has moved. Do the same with the other three corner points. Note that the outflow boundary has an x coordinate of 23.5 m. After all the points are given, then from the menu choose *Generate -> Boundary* and *Generate -> TransfiniteI* and *Generate -> 3D Grid.* Then save the grid: *File -> Write unstruc file*.

What you see now is that you don't have the correct geometry at the inflow region. For example, the first 3.0 meters is a straight flume with constant width and depth. To change the grid accordingly, you need to give two *NoMovePoints* at grid lines i=31,j=1 and i=31,j=21, and specify the coordinates at these two points. From the menu choose *Define->Set NoMovePoints mode*. Then from the menu choose *View->Legend*. Then enlarge the grid with the *PageUp* button. You can also use the arrow keys to move the grid. Make the grid large around the points (31,1) and (31,21). Click on these two grid intersections. You will see a blue square emerge. If you click on the wrong point, go to the menu and choose *Define-> Delete NoMovePoint*. After the two points are defined, from the menu choose *Define -> Set NoMovePoints* mode again. Then click on the point (31,1), and from the menu choose Define -> Give coordinates. Then give the coordinates x=3, y=0.34, z=1.0. Click *OK* and see the point move. Then do the same with point (31,21), which should get coordinates: x=3,y=0.66, z=1.0. Afterwards, from the menu choose *Generate->Boundary* and *Generate-> TransfiniteI* and *Generate->3D grid*. Now you have made the first section of the flume.

Then continue with the next section, by adding more *NoMovePoints*. The next section will be at grid line 38, the last section which has vertical level 1.0 m. Then there will be a section at grid line 86 and 89. At the end, save the *unstruc* file.

Step 3: Water velocities

The next step is to compute the water velocities. Before this is done, the discharges need to be specified. This can be done in the *DischargeEditor*, or alternatively, by using the *F 314* and *F 237* data sets, as given in the control file from the web site *http://folk.ntnu.no/nilsol/cases/doris*. Edit the *control* file and replace the *YH* on the *F 2* data set with *UW*. Or use the *control.u2* file from the web site, renamed without extension. Then restart the program and wait until it converges. The measured water velocities are given in the *verify.u* file. This file was used in *SSIIM 1* to compare measured and computed values directly in the *SSIIM 1* graphics. It is actually better to use a spreadsheet for this comparison - the graphics will be better, with scales, legends etc. The *verify* file gives the velocities at profiles specified with (x,y) coordinates. This is the same coordinate system as used for making the grid. Comparing the computed and measured results, it is necessary to produce profiles of computed velocities at the same location as the measurements. This is done with the *interpol* file. A file called *interpol.u* is given on the web page. This can be downloaded and put in the working directory. It also has to be renamed *interpol*, without an extension. Then, the *control* file has to contain the data set *F 48 2*. When this data set and the *interpol* file is used and *SSIIM 2* writes the results, *SSIIM 2* will also produce a file called *interres*. This contains the computed velocity profiles at the locations from the *interpol* file. Which is the same as the measured profiles. The data from the *interres* file and the *verify.u* file can then be taken into a spreadsheet and compared.

Step 4: Sediment concentrations

The last step is to compute the sediment concentrations. To do this, the *control* file has to be modified, or the *control.s2* file used. Add *F 37 2*, *F 33 10.0 10* and *F 68 2*. The *F 68 2* data set will invoke a computation where the water velocities are not recomputed for each time step, only the sediment concentrations. This will save computational time and not deteriorate the results as the bed elevation changes were very small. Also, replace *UW* on the *F 2* data set with *F 2 URSM*. The *M* invokes a function writing a new *interres* file. Also, change the *F 48* data set to *F 48 6* instead of *2*, which will produce concentrations instead of velocities. This time, we will use the *intepol.c* file, which can be downloaded and renamed *interpol*. Also, add an *F 1 D* data set in the *control* file. This will write the sediment fluxes to the *boogie* file, enabling computation of the trap efficiency for the sand trap. The first parameter on the *K 1* data set can be reduced to 100, to save computational time. This means only 100 time steps are used. A transient computation of the sediment concentration is done here, but over a long time so that a steady state is produced at the end of the computation. Also, the *timei* file from the web site has to be used.

Common mistakes:

If the sediments drops to the bed of the flume at the entrance, and do not continue into the sand trap, the reason is most probably that an error has occurred reading the *result* file. You can check this by looking at the velocity vectors in the SSIIM graphics. The velocity field is stored in the *result* file, so make sure that this file exist in your working directory and that it contains the correct velocity field.

If the student attempts to compute the water flow and the residuals are not changing and the program seems not to solve the Navier-Stokes equations, this is because the *F 68 2* option is given in the *control* file.

To be handed in:

A text of what you have done and how the computed water velocities and sediment concentrations compare. Figures that shows the computed and measured water velocities and concentrations. Figure of the grid, seen in plan view and in a longitudinal profile. Velocity vector figures.