From the menu, choose *View* and *Text*. Then choose *Compute* and *Waterflow*. Then push the *F 10* button on the keyboard repeatedly. The window shows how the residuals decrease. When they are below 0.001, the program will converge. The water velocities have then been computed.

Step 6. Looking at the results

From the main menu, choose View and Map. Then choose *Variable* and *Surface+bed vectors*. Scale the vectors with the *F7* and *F8* keys on the keyboard until they are of appropriate length. Also, you can scale or move the view with the arrow keys and the keyboard keys *PgUp* and *PgDn*. See how the secondary currents cause a deviation between the bed and surface velocity vectors at the end of the canal.

Then from the main menu choose *Sediment variable* and *Secondary current angle*. This shows the angle in degrees between the surface and bed vectors.

2.9 Tutorial 7. Natural river using SSIIM 2

We will compute the flow in a natural river using SSIIM 2. SSIIM 2 includes wetting and drying, and it is convenient to use this algorithm to get a grid that follows the bank of the river as the water level rises and sinks. This tutorial will only work directly with SSIIM 2 versions made after 12. December 2013. Earlier versions need to start with a *control* file that has the *F 64 11* data set.

Step 1. Making the grid

The geometry of a natural river is most often fairly complex. The most used way to describe the geometry is by measuring a large number of coordinates (*x,y,z* values) of points at the river bed. SSIIM use these points in a file called *geodata*. Each point is given on one line in the file, with the three floating point numbers, *x,y* and *z*. There has to be a capital *E* at the start of each line, before the numbers, and the numbers must be separated by one or more spaces.

In the present tutorial, we will use a geodata file made up from a virtual case. The file can be downloaded from the web page: *http://folk.ntnu.no/nilsol/cases/tutorial_river/geodata*.

Make a new directory on your PC, download this geodata file to the directory and also download SSIIM 2 with the DLL's to the same directory. Then start the program. In the main menu, go to *View* and *Grid Editor*. Then go to *View* and *geodata points*. The measured points now emerge in the window as seen in Fig. 2.9.1.

The colours of the geodata points shows the water depth. Red is high water depths and blue is low water depths. Green is in between blue and red. Fig. 2.9.1 shows that the river is flowing in a bend. We now assume the flow direction is from the top of the figure to the right in the figure.

The first thing we need to do is to make a grid block covering the river. This is done from the menu, by choosing *Blocks* and *Add block*. Then you click on the four corners of a structured grid.

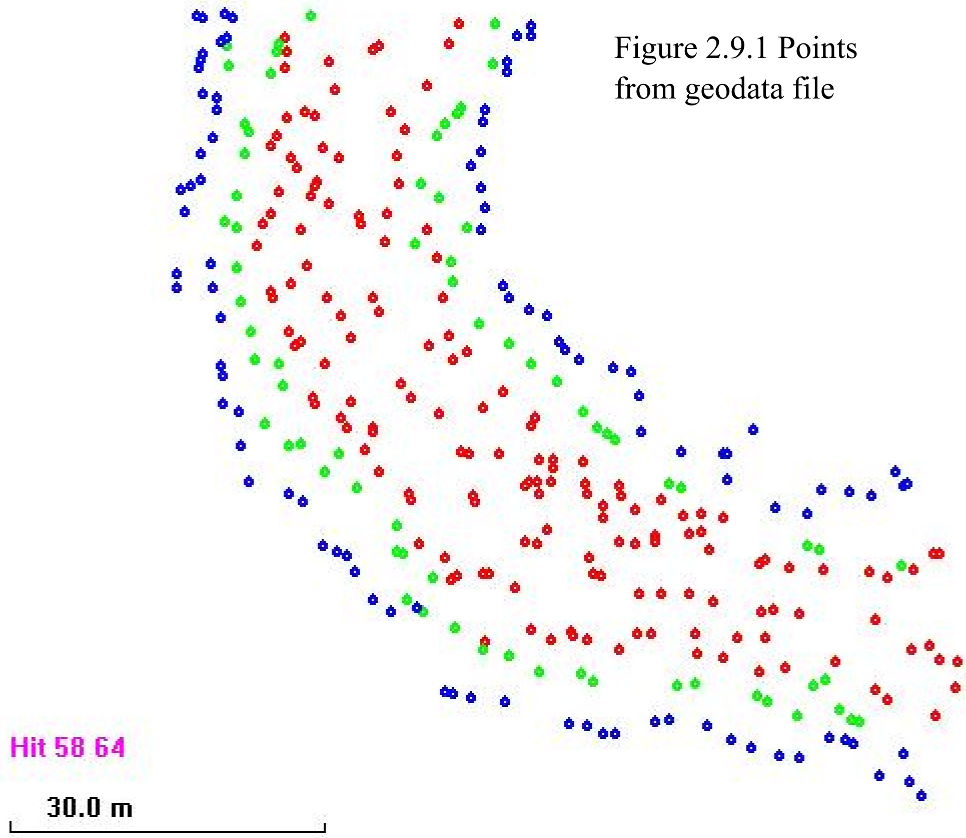

Figure 2.9.1 Points from geodata file

The first point should be on the right bank of the upstream cross-section. The second point on the left bank of the upstream cross-section. The third point on the left bank of the downstream cross-section and the fourth point on the right bank of the downstream cross-section. After clicking the second, third and fourth point, green lines emerge, which shows the border of the structured grid. This is shown in Fig. 2.9.2.

The next step is to fill this area with a structured grid. On the menu, choose *Blocks* and *Size block*. A dialog box emerges, with question about the grid size. Choose 71 lines in the *i*-direction (streamwise) and 21 lines in the *j*-direction (cross-section). Then click on *OK* and the grid is made as seen in the window.

This grid has straight sides and does not follow the river. To make the sides more curved, it is possible to click on a grid intersection at the sides of the grid and drag it towards the correct location. However, if the grid is large, this will be a time-consuming process. Instead, we select some points called *NoMovePoints*,

The maximum number of grid lines in *i* and *j* direction is 300 x 300 by default. If you need a larger grid, add a *G 1* data set to the control file, specifying the size. There is no limit on how large you can make the grid in SSIIM, only a practical limit on how long time you want to wait for the computations to finish.

which we will drag to the sides. Then straight lines will be made between these points.

From the menu, choose *Define* and *Set NoMovePoints mode*. Then click with the mouse on some of the grid line intersections on the left side of the grid (right bank). Each time a click is made, a *NoMovePoint* is made. These are indicated with blue squares. Make 6-7 *NoMove-Points* on the left side of the grid. Then choose on the menu Define and Set *NoMovePoints* mode again. This allows you to click on the grid and move grid intersections without creating more *NoMovePoints*.

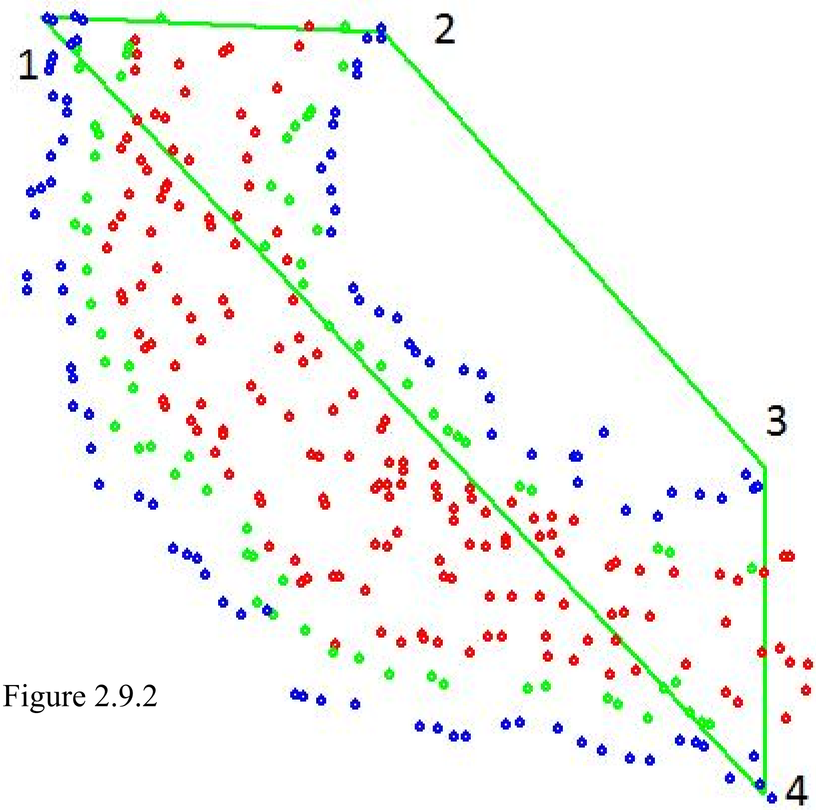

Figure 2.9.2

The next step is to move the *NoMovePoints* to the boundary of the river, given by the *geodata* points. This is done by clicking on a *NoMovePoint* and dragging it outwards to the blue geo-data points. Figure 2.9.3 shows this, where one *NoMovePoint* has been dragged.

Next, repeat the dragging of the other *NoMovePoints* to the bank of the river. Then from the menu, choose Generate and Boundary. And then choose Generate and TransfiniteI. The grid is then moves along the outer bank of the river. Repeat the same procedure with the *NoMove-Points* on the inner side of the curve. Note that it may be an advantage to scale the figure in the window with the *PgUp* / *PgDn* buttons on the keyboard, and also use the arrow keys. If you make a mistake, and for example put a *NoMovePoint* in the interior of the grid, this can be removed with the menu option *Define* and *Delete NoMovePoint*. Then the last *NoMovePoint* is deleted.

From the menu, choose *Generate* and *Boundary*, and then *Generate* and *TransfiniteI*. You can also move individual *NoMovePoints* and repeat the boundary and transfinite grid generation, until you are happy with the grid. At the end, you can choose *Generate* and Elliptic. This usually gives the best grid. Figure 2.9.4 shows an example of how the grid can look like.

The grid can then be saved to the unstruc file. This is done by the menu options *Generate* and *3D grid*, and then *File* and *Write unstruc file*.

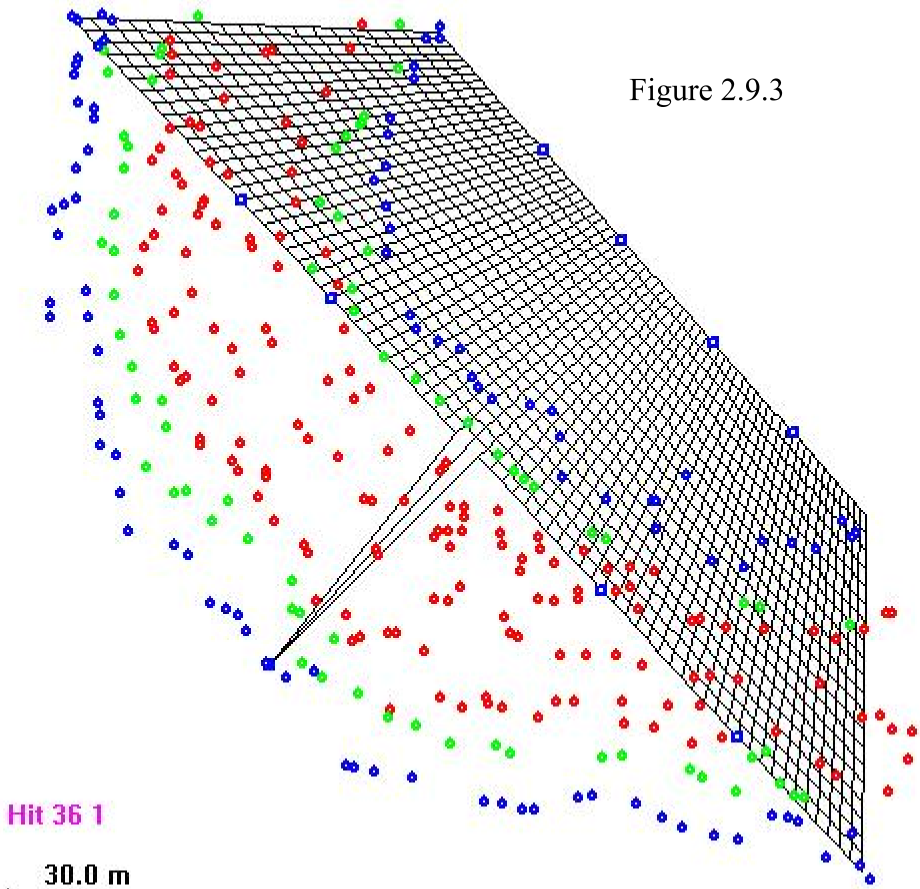

Figure 2.9.3

The procedure so far has given the horizontal layout of the grid. The vertical extension of the grid also need to be decided. This is done by using the geodata points to determine the bed level of the grid by interpolation. The menu option is *Generate* and *bed levels*. After this, use the menu again with *Generate* and *3D Grid*. The grid should then be saved to the *unstruc* file again.

A common mistake by SSIIM novices is to make the initial grid too fine. Then the computational time is long and it takes considerable time before the mistakes are seen and the program is learned. It is better to start with a coarse grid. And then make a new and finer grid once the program is learned and all other probles are sorted out. Usually, it does not take too long time to make a grid, and the second grid you make is usually better than the first.

The grid we have made may not be optimal. If we end the SSIIM 2 program and restart it, and then read the *unstruc* file, we can see how it looks by choosing on the menu: *View* and *Map*. Choose on the menu *Variable* and *bed level*. Then on the menu choose *View* and *Legend*. Then the geometry may look like what is shown in Fig. 2.9.5. There may be holes or hills where the bed interpolation algorithm has not done a good job. Then it is possible to go back to the *Grid Editor* and give in the vertical level on individual grid intersections. Click on the grid intersection and from the menu choose *Define* and *Give coordinates*. Then give a better *z* value in the dialog box. Repeat this for all problem areas and then from the menu choose *Generate* and *3D Grid*, and then save the *unstruc* file. Do not generate new bed levels, as the algorithm with overwrite any manual changes to the coordinates.

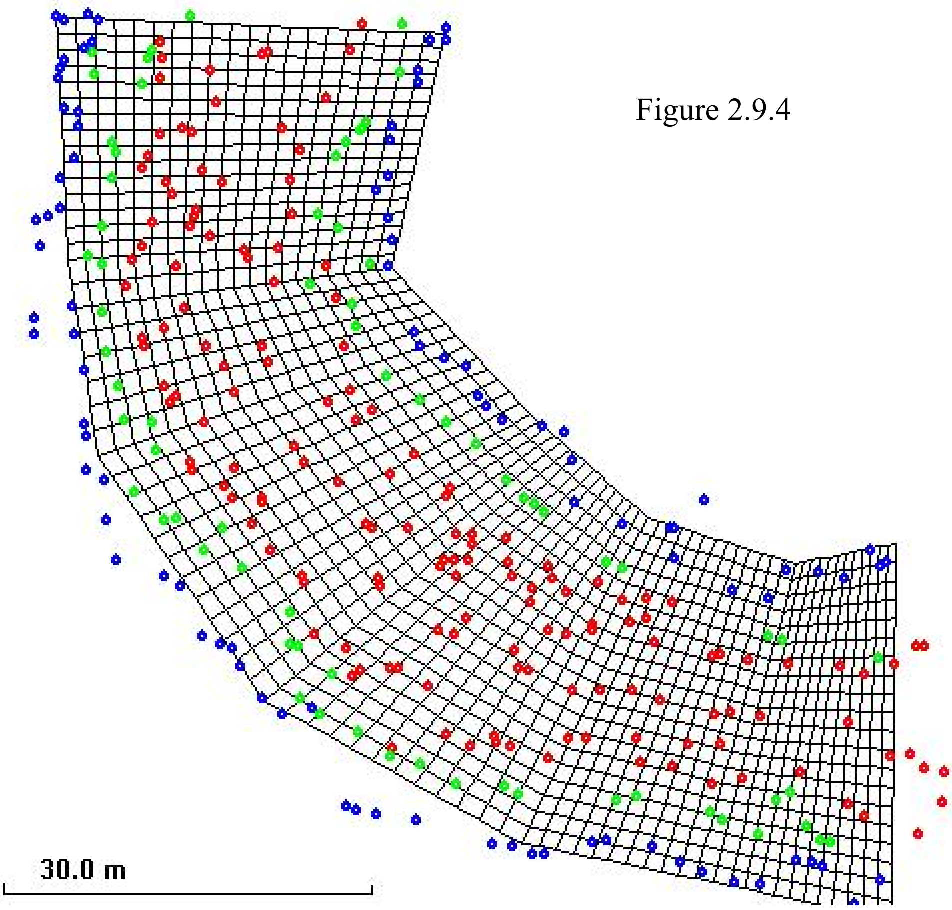

Figure 2.9.4

Note that it is also possible to improve the grid by adding or removing *geodata* points in the *Grid Editor*.

Step 2. Specifying a water discharge

The water discharge can be specified using the *Discharge Editor*, as described in the earlier tutorials. However, for a river where there is only one block and the water is flowing into one of the four sides and out of the opposite side, like we have in the current case, there is another option. This is to specify the following data sets in the *control* file:

F 314 1 1
F 237 1 1.5
F 237 2 1.5

The *F 337* data sets specify the water discharge in m^3/s, while the *F 314* data set sets the grid ends to inflow and outflow.

If we open SSIIM 2 again, read the *unstruc* file and look in the *Discharge Editor*, we can see the coloured lines from the inflow and outflow. If we look in the dialog box, we see that the water discharge is the same as specified in the *control* file.

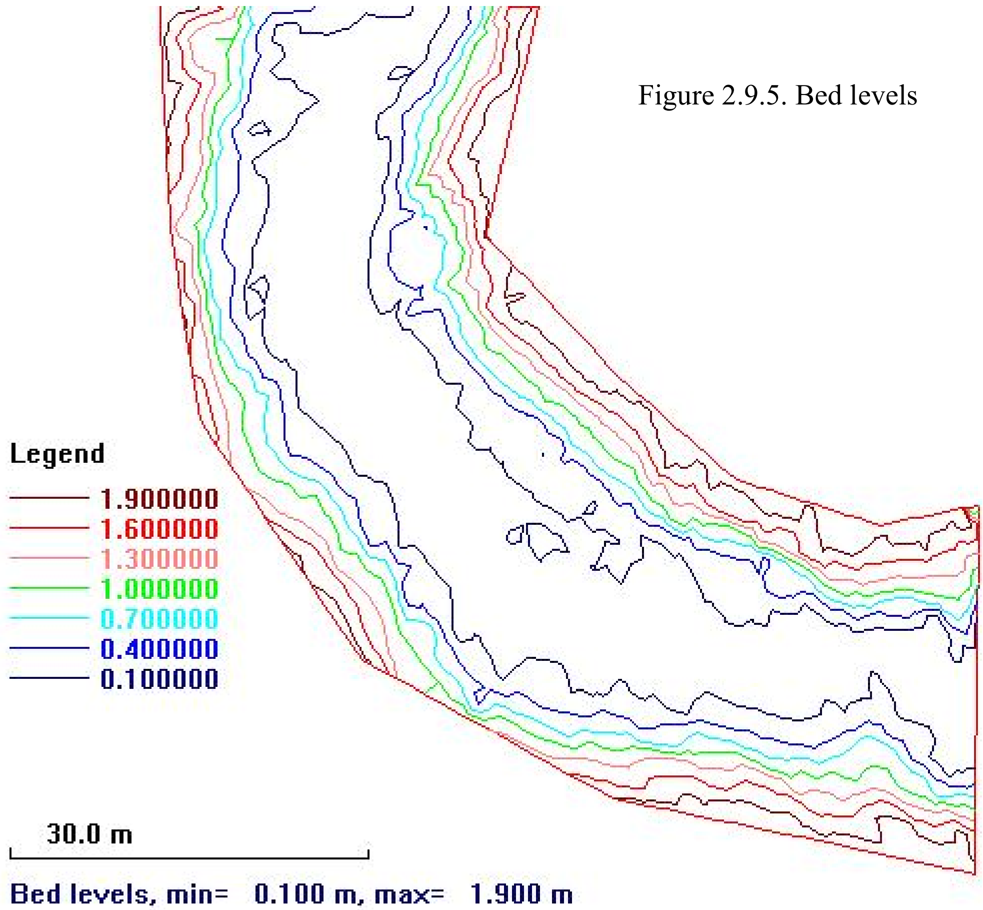

Figure 2.9.5. Bed levels

Step 3. Computing the water flow

Start up SSIIM 2, read the unstruc file and from the menu choose *Compute* and *Waterflow* Push the *F10* button on the keyboard and watch the residuals decrease. When they are under 0.001, the solution is converged.

On the menu, choose *View* and *Map,* and then *Variable* and *Velocity vectors*. Scale the vectors with the *F7* or *F8* button on the keyboard until they look clear.

Step 4. Making an initial water surface

The top of the grid made in Step 1 is horizontal and at the level of the highest *geodata* point. This is at 2 meters with the current data. Often the geometrical points measured in the river include the overbanks. This makes sense when modelling a flood. However, if we want to model a lower water discharge, we also want a lower water level. The easiest way to lower the water level is by specifying *surface points* in the geometry. This is done by first reading the *unstruc* file, and then go to the *Grid Editor*. Then from the menu choose *Define* and *Set surfacepoint*. A dialog box emerges, asking for the level of the surface point. Normally, we choose three surface points, all outside the geometry of the grid. The three points will form a triangle with a defined surface. This will be the new surface.

Scale the grid down with the *PgDn* button on the keyboard and move it to the center of the window. Then click with the mouse outside the grid. Go back to the menu and repeat the procedure two more times. The three emerged surface points will form a triangle. The grid should be inside the triangle, as given in Fig. 2.9.6.

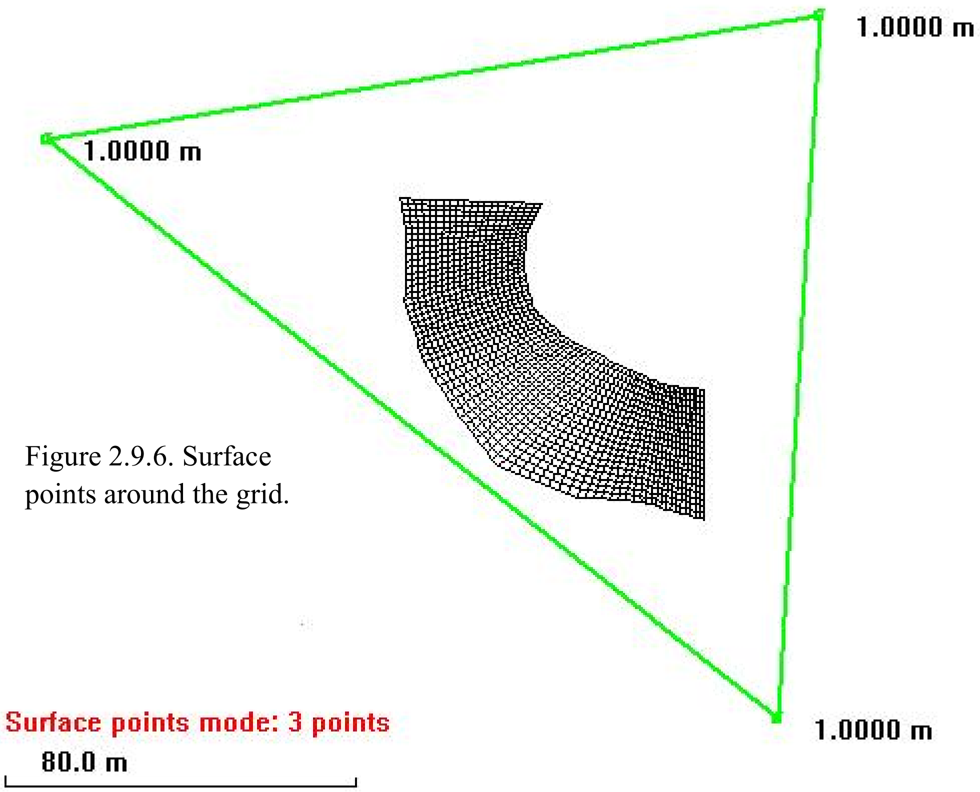

Figure 2.9.6. Surface points around the grid.

Then go to the menu option *Generate* and *Surface*, and then *Generate* and *3D Grid*. Then write the *unstruc* file. Also, you can look at the *Map* in the *View* menu, and choose *Water level* or *Depth* from the *Variable* menu option. You can see that the water surface elevation has been reduced to 1 meter. You can now recompute the water velocities with this water level.

Note that you may choose a different level on each of the three *surface points*. Then a sloping water level is given.

Looking at the grid, the edges following the banks may not look too smooth. To improve the river bank cells, the *control* file needs to be used. Open the file with an editor and add the data sets *F 102 1*. Save the file, restart SSIIM 2, go to the *Grid Editor* and choose from the menu *Generate* and *3D Grid*. Go back to the *Map* view and see how the edges are smoother, similar to Fig. 2.9.7. Then save the *unstruc* file.

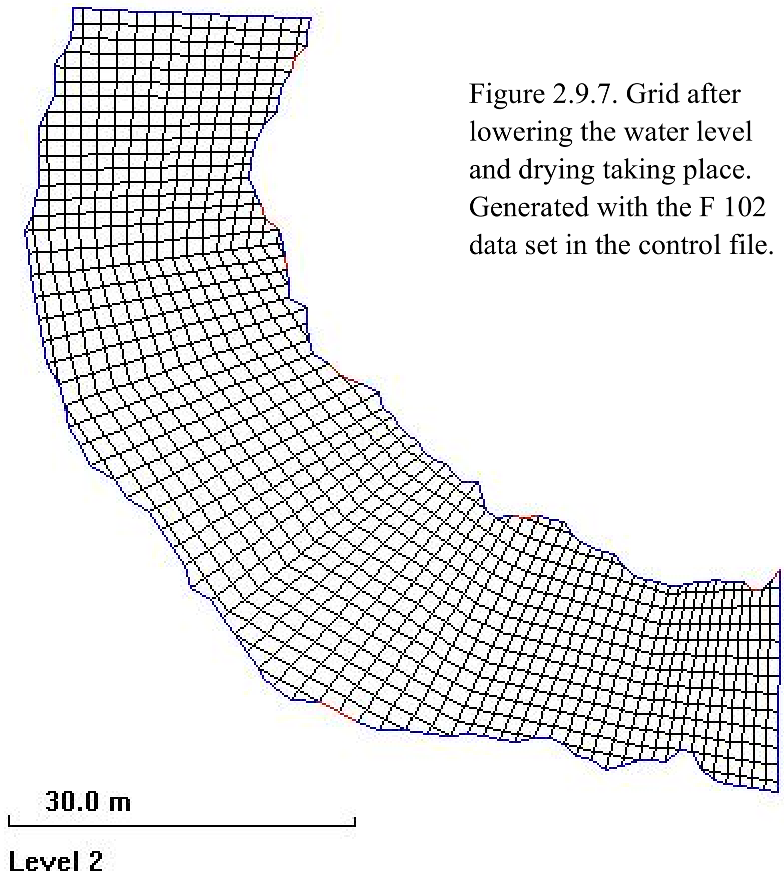

Figure 2.9.7. Grid after lowering the water level and drying taking place. Generated with the F 102 data set in the control file.

When you want to compute the water flow for this grid, the solution may not converge. This is because some of the grid cells will have a very low height to length ratio, or the area between the cells is very small. There are several methods to improve convergence. This is further described in Chapter 3.2. The methods involve using stabilizing procedures by adding data sets to the *control* file. The first thing to do is to lower the relaxation coefficients by adding a K 3 data set. If that doesn't work one or more of the following additional data sets may be used:

F 94 0.02 0.1 minimum cell corner heights
F 168 8 multi-block solver
F 159 1 9 0 1 0 disconnecting cells in shallow areas
F 235 10 triangular cell damping
F 292 0.1 inner time step
K 3 0.3 0.3 0.3 0.05 0.2 0.2 lowered relaxation coefficients
K 5 0 0 0 10 0 0 multi-block sover

The *F 94* and *F 159* data sets give parameters for generating the grid. This means the grid needs to be regenerated after this data set is given in the control file. The regeneration is done in the *Grid Editor*. Remember to save the *unstruc* file afterwards.

Step 5. Computing the water level more accurately

Although the chosen flat level of the water may correspond reasonably well to the real world, it may sometimes be necessary to make a more accurate model. There are several algorithms in SSIIM 2 for computing the water level. The most relevant ones for river modelling are based on approximately subcritical flow, where the downstream water level is specified. The user then has to specify a cell with a given water level, which is unaffected by the water surface computation. The reference cell must be specified on the *G 6* data set in the *control* file. The next step therefore involves which cell to choose and how to choose it.

On the menu, choose *View* and *Map*. Then choose *Variable* and *Cell numbers*. Enlarge the downstream part of the grid using the *PgUp* and the arrows buttons on the keyboard. You will see something like what is given in Fig. 2.9.8.

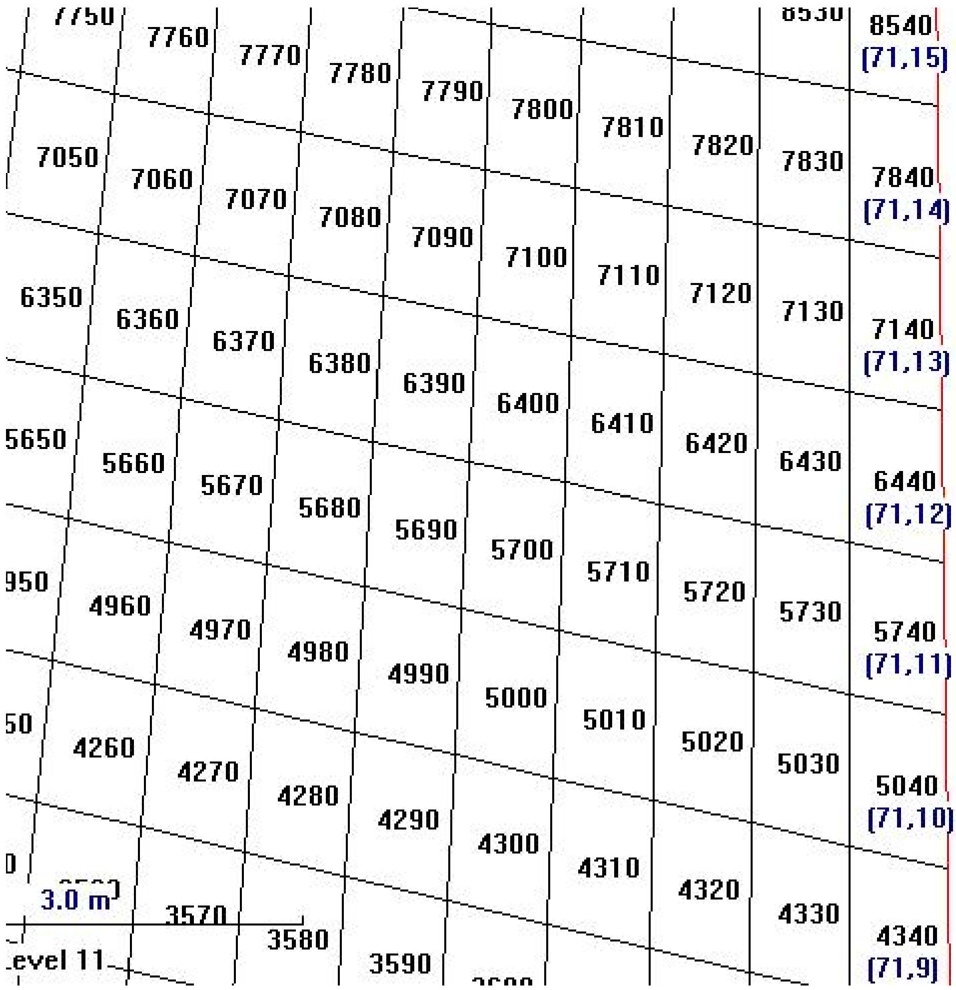

Figure 2.9.8

Cell numbers for the downstream part of the grid

Push the *F12* button on the keyboard repeatedly, until the numbers don't change any more. Then choose one of the number in the cell where the outflow is. Typically in the middle of the outflow region. For example 6440 in Fig. 2.9.8. Then give the following data set in the *control* file: *G 6 6440 0 0 0.01 0.1*

It is also necessary to specify a time step and which algorithm is to be used for the free water surface computation. To the *control* file, add

F 33 2.0 100
F 36 2

Then restart the program, read the *unstruc* file and start the water flow computations. In the *Map* graphics, look at how the water level changes. Note that the water level will only be updated if the residuals are fairly low. Lowering the time step or increasing the number of innier iteratios on the F 33 data set may be necessary to reduce the residuals.

2.10 Examples

Example cases with input files can be downloaded from the *WWW*, at one of the addresses:

http://folk.ntnu.no/nilsol/ssiim

Some of the files are located in the package containing the OS/2 version of the program. This can be downloaded at: *http://folk.ntnu.no/nilsol/ssiim/ssiimos2.zip*.

Some of the examples are explained in the following:

Water quality with Streeter-Phelps model (SSIIM 1)

The input files have extension *.qua*

The water quality in a river is modelled with the Streeter-Phelps model. This has two water quality constituents: Organic substance measured in Biological Oxygen Demand (BOD) and Oxygen Saturation Deficit (OSD). The convection-diffusion equation for each constituent is solved, including source terms for biochemical reactions.

Start by calculating the water flow field using *MB-Flow2D* or *MB-Flow3D*. Afterwards, start the water quality calculation. You may use the *calculation* menu.

The OpenGL 2D graphics is well suited to show the results.

This case is used for testing the numerical model against an analytical solution.

Curved channel (SSIIM 1)

The files have extensions *.svi*. The channel is used for testing the programs ability to calculate the secondary flow pattern in a curved channel. It is also used for testing the routine that recalculates the water surface location based on the 3D flow field. The cross-sectional slope corresponds very well to theoretical solutions.