Mohid Sample Applications

Overview

This chapter describes how to set up the sample applications to run the MOHID model. First you should download the ZIP files which contain the MOHID GUI and the MOHID sample applications from the member's area of the MOHID homepage (www.mohid.com). Then follow the steps below:

1. Extract the MOHID GUI to any folder you wish (for example c:\program files\MOHID)
2. Extract the sample application to any folder you wish, using full paths during the extraction.
3. Map the root folder (MohidProject) of the sample applications as drive M:\. This can be done choosing Tools->Map Network Drive… from the Windows Explorer Menu. A dialog box like shown in Figure 1 appears.

Figure 1: Map a Network drive

Once mapped drive M:\ you should start the MOHID GUI by double clicking on the icon Mohid_GUI.exe. This file is located in the folder where you placed the MOHID GUI (c:\program files\MOHID in the description above). The main window of the MOHID GUI, like shown in Figure 2, should appear.

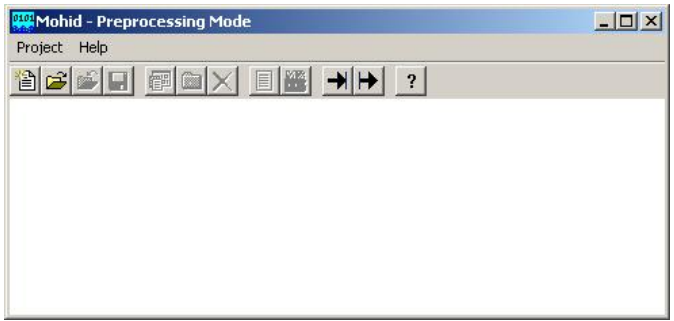

Figure 2: The MOHID GUI main window

First Sample Application - Simple Hydrodynamics

The first sample application shows the steps necessary to run an application which just simulates the hydrodynamics inside a schematic estuary.

Prepare simulation

Start the MOHID GUI and choose Project->New. A dialog box which asks for the project name and its directory appears. Give the project by any name you want and choose the directory where it should be stored (for example M:\).

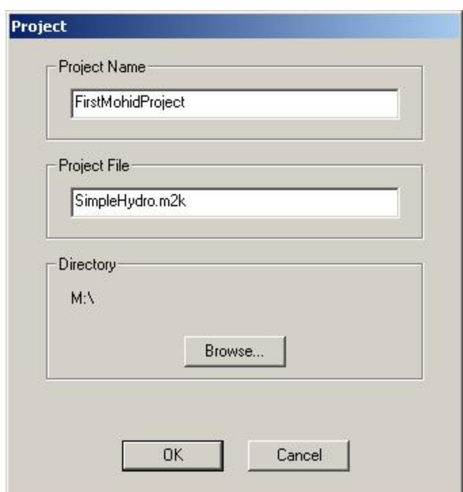

Figure 3: The project dialog box

On the left side of the main window, the project tree appears the icon of the project. Select this icon and choose Tree->Insert->Simulation. The Simulation dialog box appears (as shown in Figure 4). In this dialog box you must specify:

- the name of the simulation (can be any name related to the type of simulation you want to run, without white spaces)

- the file which contains the bathymetric data (for the present example choose the file SampleBatim.dat from the folder M:\GeneralData)
- the file which contains the tidal data (TideM2.dat from the folder M:\GeneralData)
- the MOHID executable (Mohid_v3.exe from the folder M:\GeneralData)

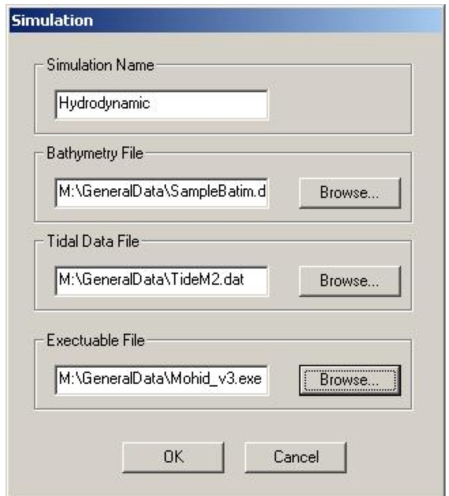

Figure 4: The simulation dialog box

The main window of the MOHID GUI should look like in Figure 5.

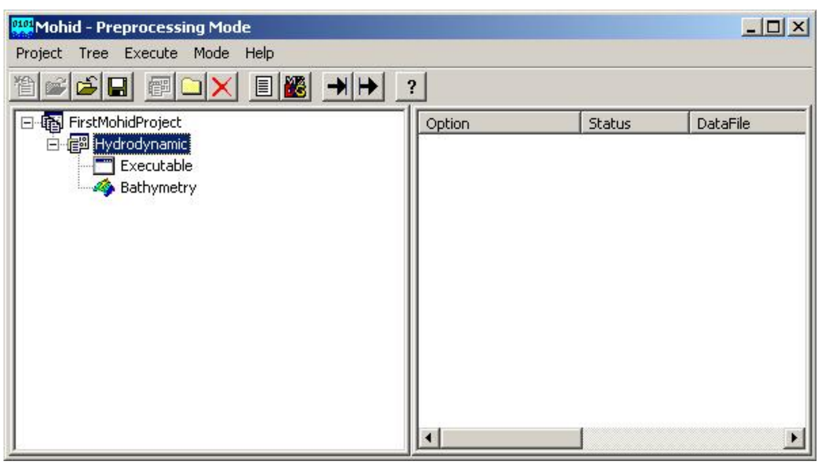

Figure 5: The MOHID GUI main window

By selecting the icon *Bathymetry* and double-clicking on the icon which appears in the right list view, one can open a window which displays the bathymetric data in a graphical way. The functionalities of this window are given below.

View Bathymetric Data

After double-clicking on the icon of the bathymetric data, a window which displays the bathymetry in a graphical way appears.

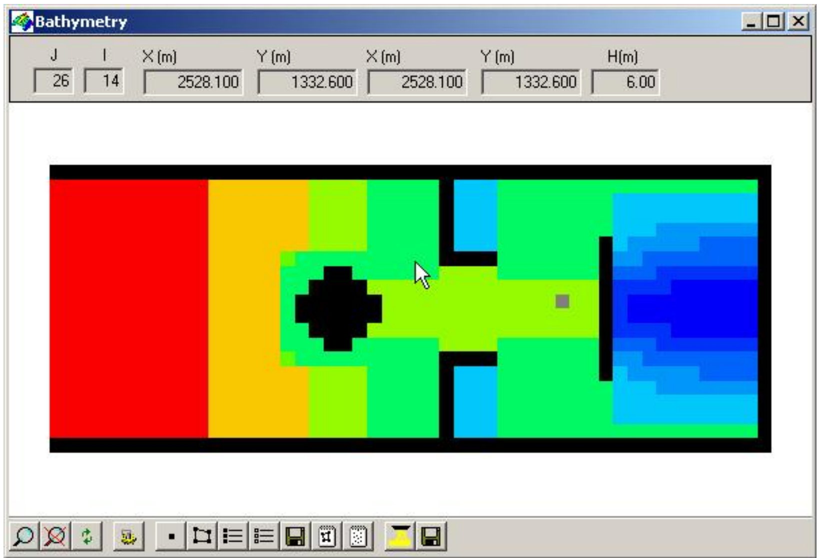

Figure 6: Display of the bathymetric data

In the upper part of the window some information about the bathymetry, for the present cursor position, is displayed. This information includes the grid cell, the distance to the lower left corner and the distance to the origin of the bathymetry. The depth is also displayed (note that a depth of -99.0m indicates land points).

In the lower part of the window one can find a menu bar, which lets the user zoom in/out, specify general settings, define points/box and change the depth of the bathymetry.

For the sample application, we will define a point at the grid location J=26 and I=11, which we will use later to write time series. Choose the symbol  from the toolbar and press the right mouse button at the location of J=26 and I=11 and then the left mouse button the finish the selection of points. When you close the window (by selecting ) the program ask you to save the selected points and boxes. Choose yes.

Prepare a run

A project consists of one or more simulations and a simulation of one or more runs. All runs inside a simulation use the same bathymetry and the same tidal components. To add a new run select the place where the run should be insert in the project tree view. For the present example, choose the simulation (since it doesn't have any runs yet) and choose Tree->Insert->Run. A dialog box which let you specify the option of the run appears (Figure 7). Here one can specify the start and the end of the run as the time step. Also it is possible to choose the modules which you want to use during the simulation. For now just change the time settings to the ones shown in Figure 7 and leave the rest.

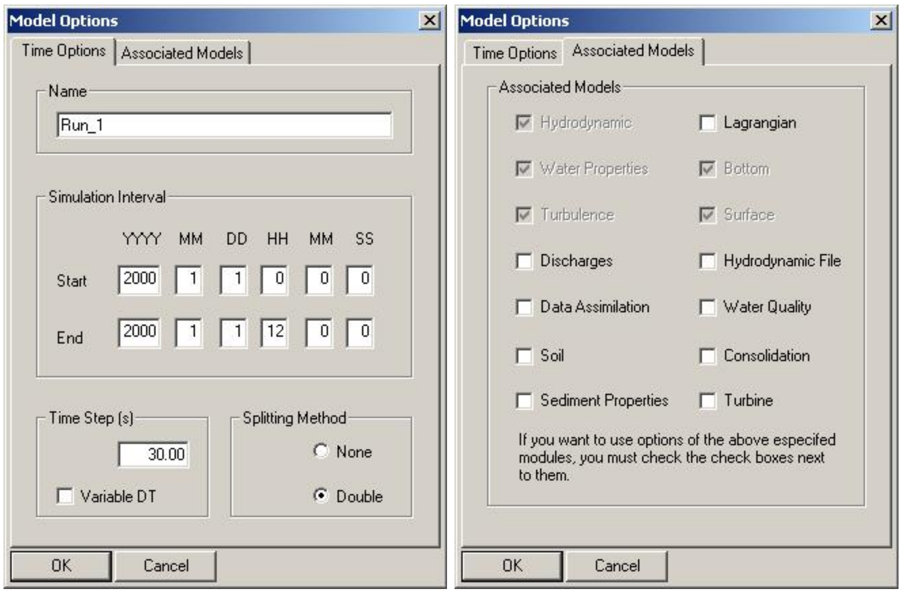

Figure 7: The Model option dialog box

When you are finished with all data choose OK. On the right side a list with all modules which are used appear. By double clicking any of them, the compute options for this module can be specified. This can normally be done in two ways. Or one uses the Notepad or the GUI. Using the GUI is more user friendly (but not yet finished for all modules), using the Notepad gives the user more liberty over the compute options. If the status displays *"?"* means that options for this module haven't be processed yet.

For the sample application choose double-click on the *Hydrodynamic* icon and choose GUI. A window like the one shown in Figure 8 will appear.

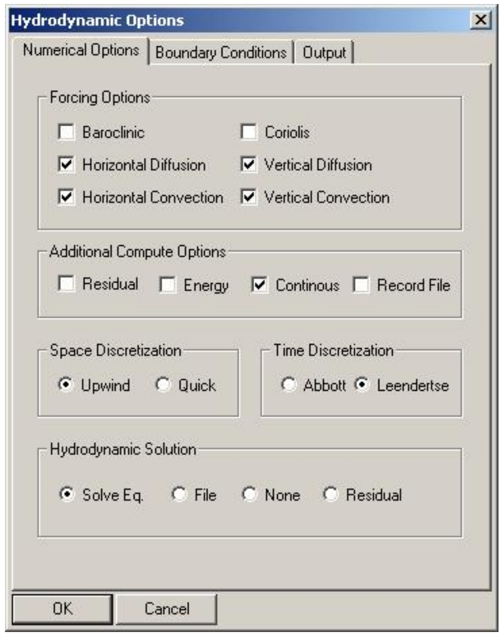

Figure 8: The Hydrodynamic options (1/3)

Just remove the option *Coriolis* under *Forcing Options*. This will allow us to obtain a symmetric flow, once the bathymetry is symmetric.

As next step choose the tab *Boundary Condition* and enable the option *Tide.* This will impose the tidal harmonics from the tidal data file (Figure 9).

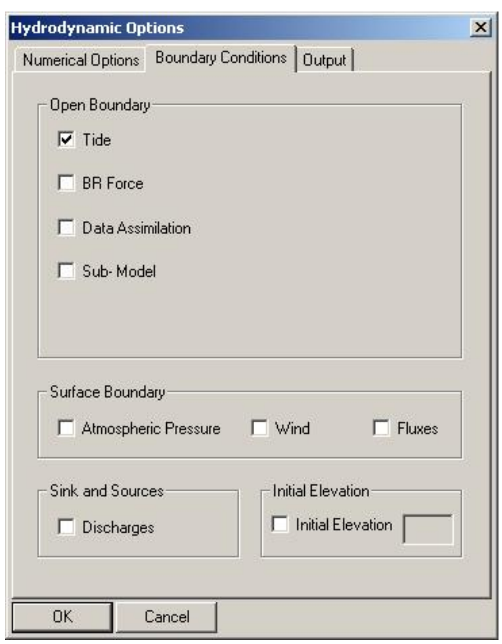

Figure 9: The hydrodynamic options (2/3)

As last step choose the tab *Output* and enable the HDF output and the time series output. Use the options like shown in Figure 10. If you press *Add*… to add time series locations, the list of points previously specified in the bathymetry window appears.

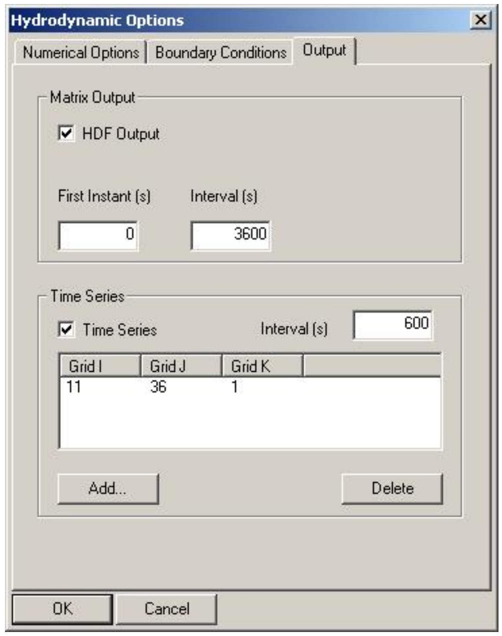

Figure 10: The hydrodynamic options (3/3)

Close the window by choosing OK at the bottom.

As next step double-click on the Geometry icon and choose GUI. A window like the one shown in will appear. Here it's necessary to specify the vertical domain composition. Choose *Add*… to add a new vertical domain.

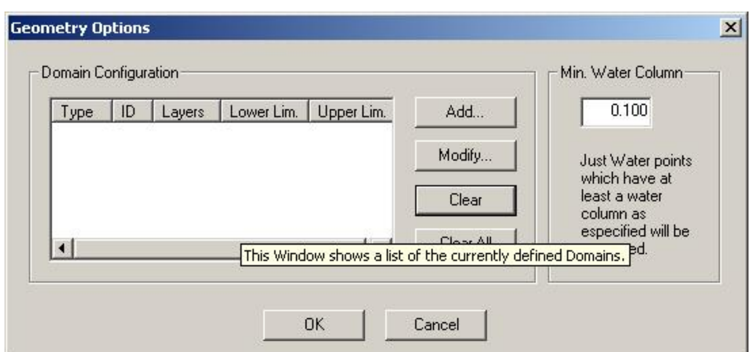

Figure 11: The geometry options

A new window, with all possible options for the vertical domain appears. Choose as *Domain Type* Sigma and choose 5 vertical layers. The window should look like the one shown in Figure 12.

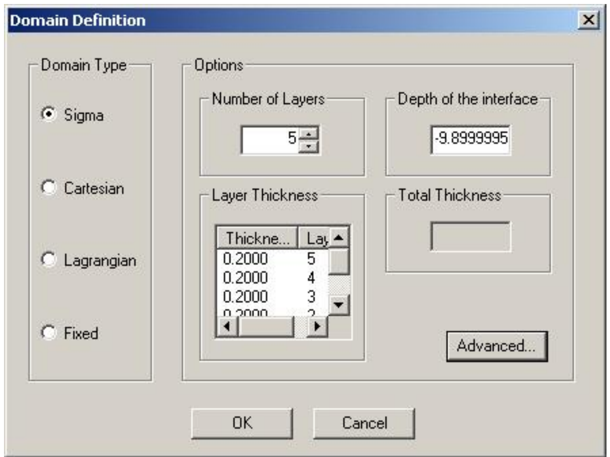

Figure 12: Sub-domain configuration

Choose OK to close the window and again OK to close the geometry options.

Once we are running a sample estuary with a grid resolution of 50m, open the *Turbulence* options, by double clicking on the respective icon. The Notepad (there is no GUI yet) with two keywords appears. Change the values to the ones indicated in Figure 13.

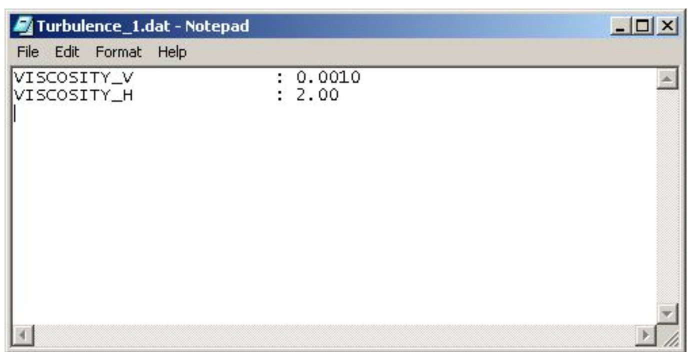

Figure 13: The turbulence options

Now all data files, except the one which says *Filenames* should have the status OK. To run the models choose Execute->Create Nomfichs from the main window menu. This procedure prepares all files for a posterior run (and the status of *Filenames* passes to OK). In the window which appears, select Run_1 and choose *Create*….

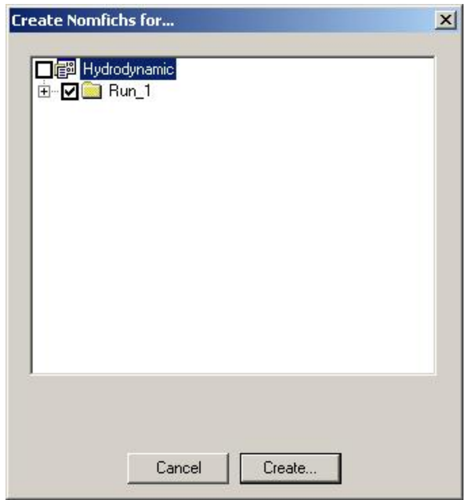

Figure 14: The create filename window

To run the Run_1, you must now choose Execute->Select Runs To Execute from the menu of the main window. A window similar to the one in Figure 14 appears. Select once again the Run_1 and choose *Run*…. A DOS window will open, with MOHID executing the Run_1 (Figure 15).

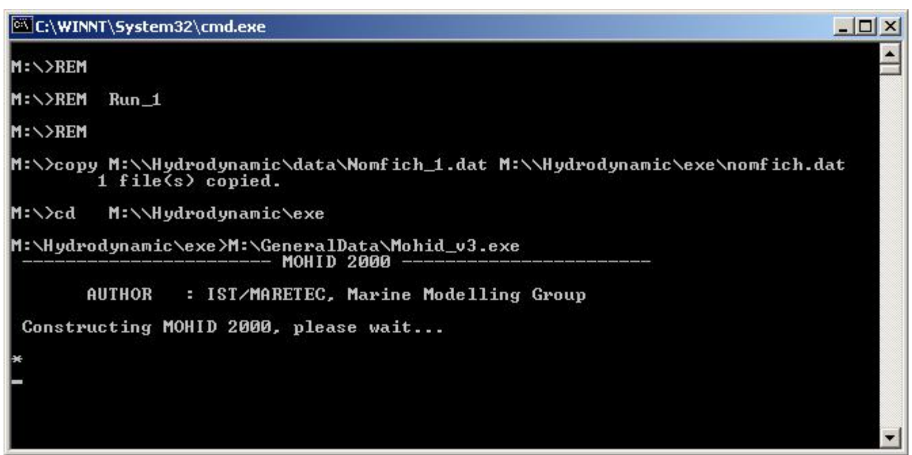

Figure 15: MOHID executing in a DOS window

Depending on your computer speed, the run just set up will take about 80 seconds to finish. At the end of the run, the DOS window should look like the one shown in Figure 16. Be aware that without the completed execution of the model, it is impossible to analyze the results.

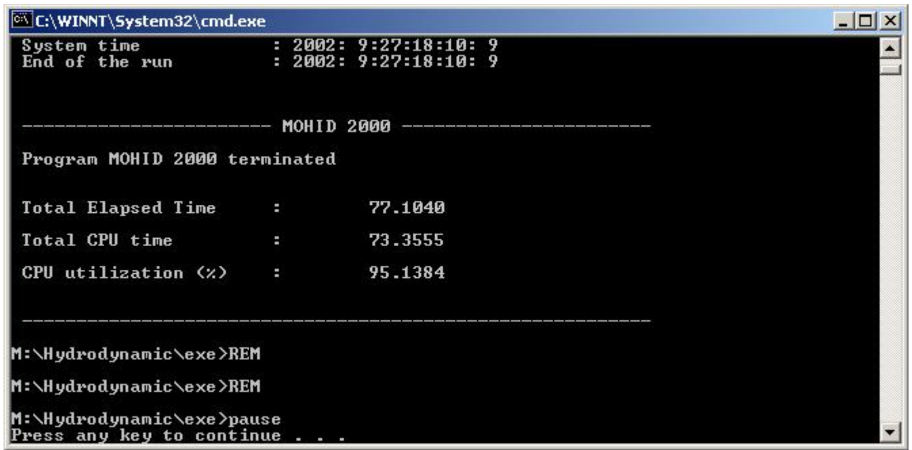

Figure 16: Completed execution of MOHID

Analyzing the results

The MOHID GUI main window has two “modes”. The pre-processing mode and the post-processing mode. Until now you were working in the pre-processing mode to prepare the data files. To analyze the results you must switch to the post-processing mode by choosing Mode-Post Processor from the main menu. A window like shown in Figure 17 appears and in the right list-view of the main window the result files of the modules are displayed.

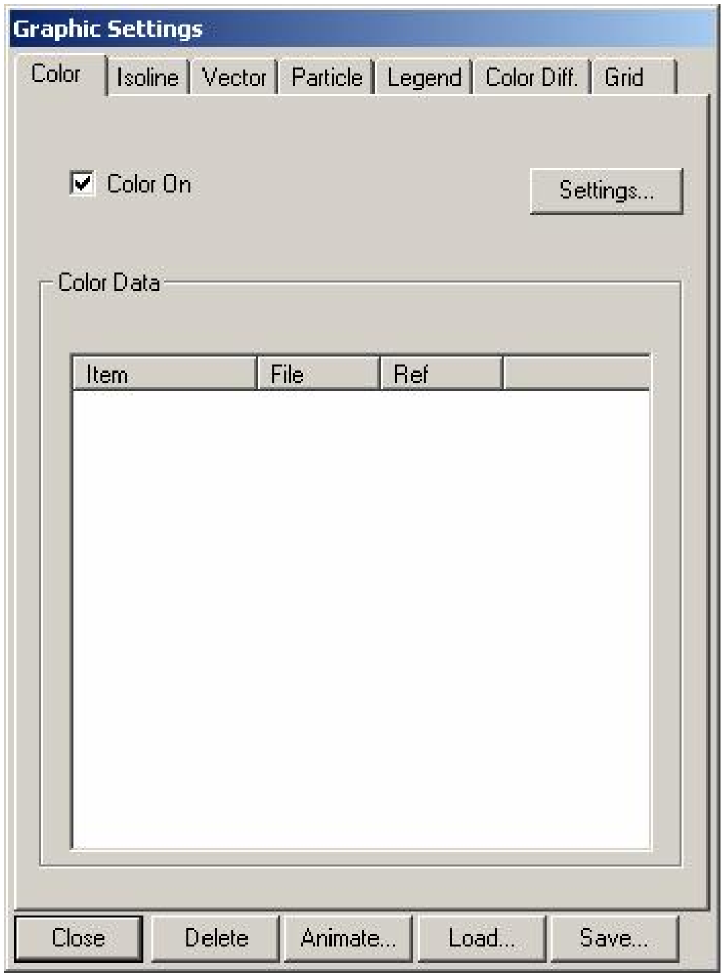

Figure 17: The Post Processor Data Selection Window

To visualize the results of the hydrodynamic simulation, double-click on the icon of the hydrodynamic module. A window with the contents of the file will appear (see Figure 18 – left side). To display any information as color map, select the information with the left mouse button and then click the right mouse button. A pop-up menu appears. Choose Add to Color (see Figure 18 – right side).

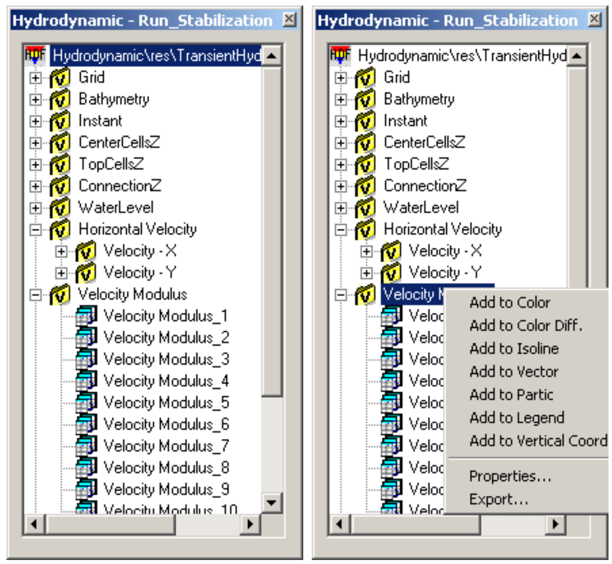

Figure 18: Results of the hydrodynamic module.

The select information will appear in the post processor data selection window and Color will be turned on (Figure 19).

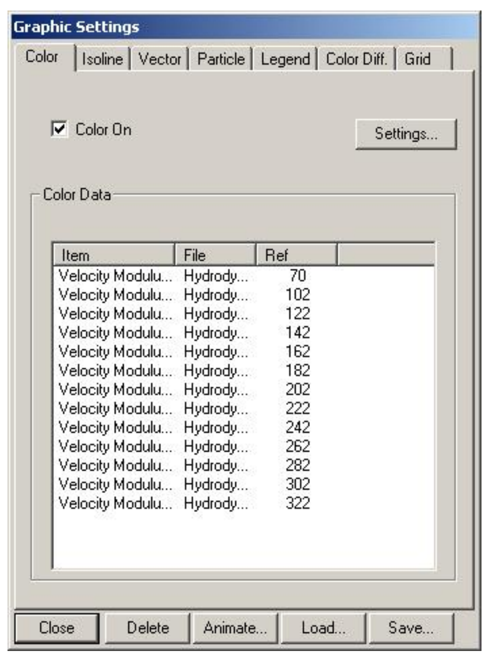

Figure 19: The Post Processor Data Selection Window with Color Data

Click on "Animate" to view the results. A window with the results will open (Figure 20). If the mouse is over the image (and the window active) with the following commands you can manipulate the image:

```
s, S    - Settings      brings up settings dialog box
w, W    - Write         brings up dialog box to write images to file
```

```
g, G     - Goto           lets user choose new instant to render
m        - minimize       resizes window to 600x400
M        - Maximize       displays image in FullScreen
r        - render         renders next instant
R        - Render All     render all instants, from current to last
x, X     - exit           closes OpenGL window
f, F     - Flight         start flight (defined by flight settings dialog
box)
0        - Light0         toggles on/off state off light 0
1        - Light1         toggles on/off state off light 1
2        - Light2         toggles on/off state off light 2
3        - Light3         toggles on/off state off light 3
4        - Light4         toggles on/off state off light 4
P        -                Moves maximum texture X coordinate right
p        -                Moves maximum texture X coordinate left
O        -                Moves minimum texture X coordinate right
o        -                Moves minimum texture X coordinate left
K        -                Moves maximum texture Y coordinate up
k        -                Moves maximum texture Y coordinate down
I        -                Moves minimum texture Y coordinate up
i        -                Moves minimum texture Y coordinate down
```

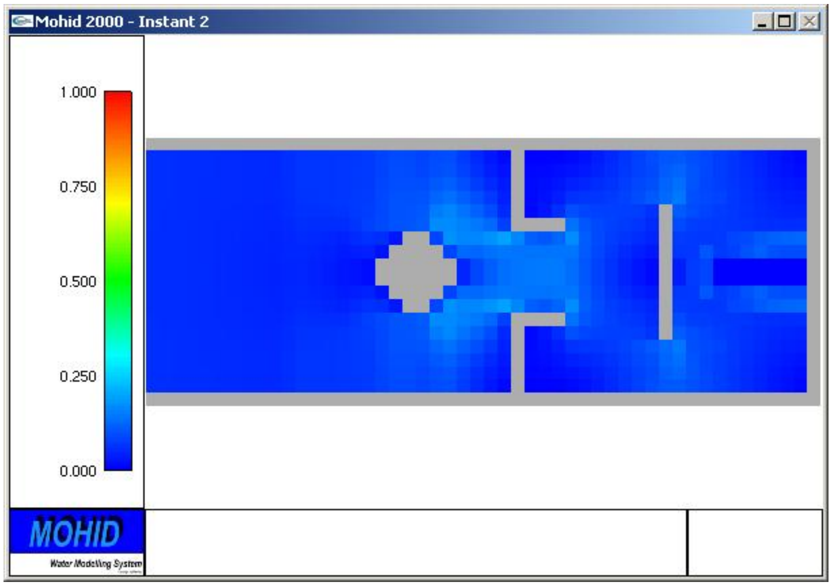

Figure 20: The window with the results

If you press the key "s" to access a dialog box which lets you adjust the settings of the image. Use the right mouse button to rotate, zoom and scale the image.

To be continued….