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 (<u>www.mohid.com</u>). 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.
- 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:\).

1		
Project File	ro.m2k	
Directory		
M:V		
	Browse	

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)

Hydr	odynamic	1	18	
Bathyr	netry File-			
M:\0	ieneralDa	ta\SampleE	atim. d	Browse
Fidal C	ata File -			
M:\0	ieneralDa	ta\TideM2.	dat	Browse
Exectu	iable File-			
M:\0	ieneralDa	ta\Mohid_v	/3.exe	Browse

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 $\square \square$ 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 $\square \square$) 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.

Model Options X	Model Options	×
Time Options Associated Models	Time Options Associated Models	
Name	Associated Models	
Run_1	🔽 Hydrodynamic 🔲 Lagrangian	
	Water Properties 🕅 Bottom	
- Simulation Interval	🔽 Turbulence 🔽 Surface	
YYYY MM DD HH MM SS Start 2000 1 1 0 0 0	🗖 Discharges 🗖 Hydrodynamic Fi	ïle
Fred 2000 1 1 1 12 0 0	🗖 Data Assimilation 📄 Water Quality	
	🗖 Soil 🗖 Consolidation	
Time Step (s) Splitting Method	🗖 Sediment Properties 🗖 Turbine	
30.00 O None	If you want to use options of the above especife modules, you must check the check boxes next to them.	d
Variable DT O Double		
OK Cancel	OK Cancel	

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.

e
3

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).

imerical Options Boundary Co	onditions Output
Open Poundani	
Open Boundary	
I ∽ Tide	
BR Force	
Data Assimilation	
Sub-Model	
- Surface Boundary	
- Surface Boundary T Atmospheric Pressure	□ Wind □ Fluxes
Surface Boundary Atmospheric Pressure Sink and Sources	Wind Fluxe: Initial Elevation
Surface Boundary Atmospheric Pressure Sink and Sources	Wind Fluxes
Surface Boundary C Atmospheric Pressure Sink and Sources Discharges	Wind Fluxes
Surface Boundary Atmospheric Pressure Sink and Sources Discharges	Wind Fluxe: Initial Elevation Initial Elevation

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.

umerical Optio	ns Boundai	ry Conditions Ou	itput
Matrix Outp	ut		
I▼ HDF 0	lutput		
First Instar	ıt(s) İnt	erval (s)	
	οΓ	3600	
- Time Series		later of t	. 600
J✓ Time S	eries	Interval (s	
0.11		O LIV	
Grid I 11	Grid J 36	Grid K 1	
Grid I 11	Grid J 36	Grid K	
Grid I 11 Add	Grid J 	Grid K	Delete
Grid I 11 Add		Grid K	Delete
Grid I 11 Add	Gind J36	Grid K	Delete
Grid I 11 Add		Grid K 1	Delete

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.

Туре	ID	Layers	Lower Lim.	Upper Lim.	Add	0.100
					Modify	Just Water points
					Clear	which have at least a water
					Close All	especified will be

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.

	Number of Layers	Depth of the interface
• Sigma	5.*	-9.8999995
C Cartesian	Layer Thickness	Total Thickness
C Lagrangian	Thickne Lay.▲ 0.2000 5	
	0.2000 4	. <u></u> ,
C Fixed		Advanced

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.

Turbulence_1.dat - Not	epad	
File Edit Format Help		
VISCOSITY_V VISCOSITY_H	: 0.0010 : 2.00	*
		v
<u></u>		▶ <i>I</i> ii

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*....

Create Nomfichs for	×
☐(ﷺ Hydrodynamic ⊕- ☑ Run_1	
Cancel 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.

C:\WINNT\System32\cmd.ex	æ		- O ×
System time End of the run	: 2002: : 2002:	9:27:18:10: 9 9:27:18:10: 9	
	MOHID :	2000	
Program MOHID 2000 te	rminated		
Total Elapsed Time		77.1040	
Total CPU time		73.3555	
CPU utilization (%)		95.1384	
M:\Hydrodynamic\exe>RE	м		
M:\Hydrodynamic\exe>RE	м		
M:\Hydrodynamic\exe>pa Press any key to conti	use nue		-

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.

Graphic Set	tings				
Color Iso	line Vector	Particle	Legend	Color Dif	f. Grid
Color Dat	r On			9	Settings
Item	(File	Ref	1	
	1	1	1		
Close	Delete	Animate	Loa	ad	Save

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).

🔽 Color On			Settings.
olor Data			
		D (7
Item Velecity Meduly	File	Het 70	
Velocity Modulu	Hydrody	102	
Velocity Modulu	Hudrody	122	
Velocity Modulu	Hudrody	142	
Velocity Modulu	Hudrody	162	
Velocity Modulu	Hydrody	182	
Velocity Modulu	Hydrody	202	
Velocity Modulu	Hydrody	222	
Velocity Modulu	Hydrody	242	
Velocity Modulu	Hydrody	262	
Velocity Modulu	Hydrody	282	
Velocity Modulu	Hydrody	302	
Velocity Modulu	Hydrody	322	

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	s di	alog	g box			
w,	W	-	Write	brings	up	dialog b	xoo	to v	write	images	to	file

g,	G	- Goto	lets user choose new instant to render
m		- minimize	resizes window to 600x400
М		- Maximize	displays image in FullScreen
r		- render	renders next instant
R		- Render All	render all instants, from current to last
x,	Х	- exit	closes OpenGL window
f,	F	- Flight	start flight (defined by flight settings dialog
boz	c)		
0		- Light0	toggles on/off state off light 0
1		- Lightl	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
Ρ		-	Moves maximum texture X coordinate right
р		-	Moves maximum texture X coordinate left
0		-	Moves minimum texture X coordinate right
0		-	Moves minimum texture X coordinate left
Κ		-	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





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....