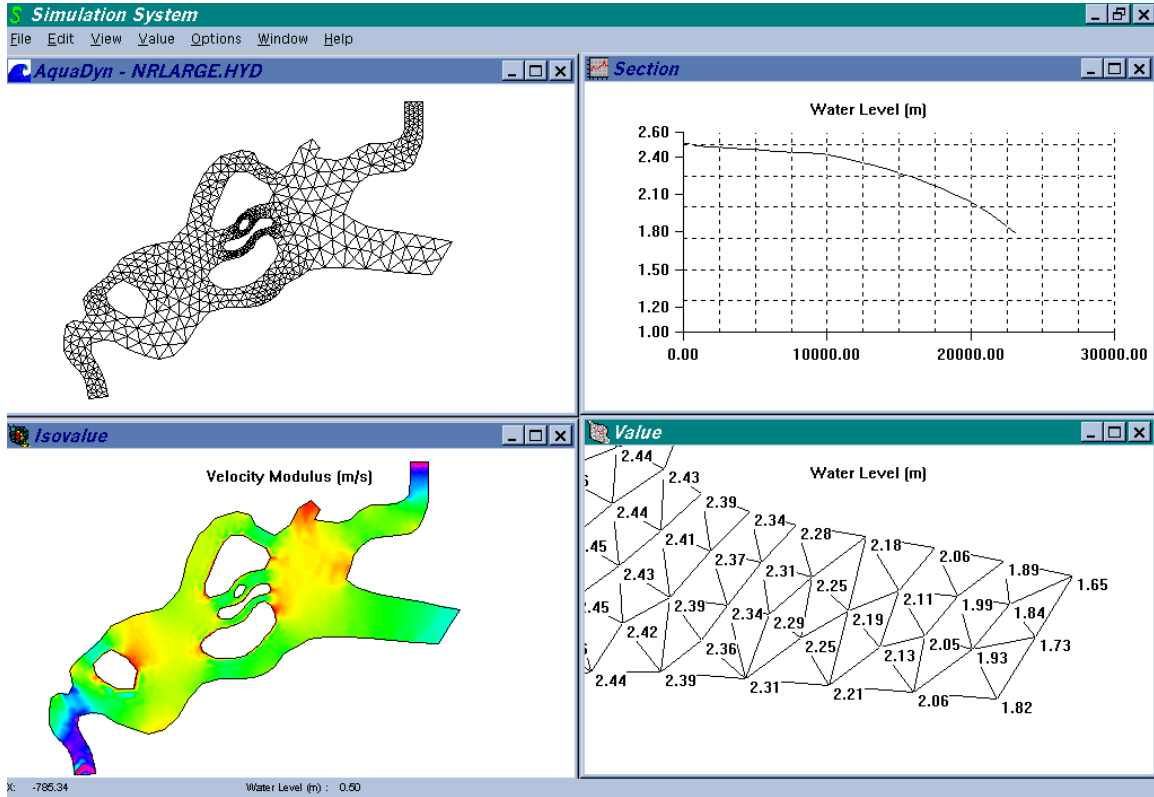


# Tutorial for AquaDyn 2.0



## Table of Content

Introduction .....	3
Getting Started .....	4
Exploring the examples .....	5
New Simulation .....	5
Visualization.....	11

## Introduction

AquaDyn is a powerful, an easy-to-use, and a completely integrated hydrodynamic simulation package. It includes a complete on line documentation, automatic mesher, input editors, the solver and intuitive visualization features. AquaDyn simulates the flow of rivers, lakes and estuaries by solving the two dimensional shallow water equations using the finite element method. AquaDyn provides a reliable way to forecast the consequences of different activities such as dredging, and building dikes, bridges, piers, and embankments.

The purpose of this document is give the new user a quick reference to learn AquaDyn.

This document describes the following topics:

- Getting Started
- Exploring Examples
- Starting a New Simulation

The Getting Started topic explains how to install the software and how to run it.

The Exploring Examples topic allows to quickly learn how to navigate to find and visualize the inputs and solution. It explains also how to perform a new simulation from that an existing simulation.

The Starting a New Simulation topic, leads the user step by step to perform a simulation from the beginning to the end.

## Getting Started

There are four easy steps to follow to get AquaDyn up running:

- Installing the simulation system
- Installing AquaDyn
- Running the Simulation System
- Load the AquaDyn model in the Simulation System

### Installing the Simulation System

Place the Simulation System diskette marked 1/3 in the Floppy Disk drive.  
From the Windows Explorer or File Manager, double click on the INSTALL.EXE file  
Follow the instructions given by the program to continue the installation process.

### Installing AquaDyn

Place the AquaDyn diskette marked 1/1 in the Floppy Disk drive.  
From the Windows Explorer or File Manager, double click on the INSTALL.EXE file  
Follow the instructions given by the program to continue the installation process.

### Running the Simulation System

Run the Simulation System using the following procedure:

For Windows 3.1x or Windows NT: Double-click the Simulation System icon in the Simulation System group of Windows Program Manager.

For Windows 95: Click Start, choose Programs and then choose Simulation System or run the command line: shellen.exe shellen.w3 with the appropriate path preceding shelle.exe)

### Load the AquaDyn model in the Simulation System

From the File Menu, choose Models. The Model Manager dialog box appears.  
Click Add. The Open dialog box appears.  
Using the drivers, directories and list boxes, locate and choose the file Aquaen.mod which is in the directory AquaDyn.

To start a new project, from the File menu, choose New. In the Available Models dialog box, select AquaDyn.

You are now ready to begin exploring some of the sample files we've included for you using the File/Open command.

## Exploring the examples

The directories where AquaDyn has been installed contains examples of simulation performed with AquaDyn (all the files with a .hyd extension)

Open an example: river.hyd, reserv.hyd or squah1hnp.hyd

Explore the following features:

- Practice Zooming, use the right mouse button or the **View/Zoom...** submenus
- Display: click the **View/Display** submenu, hold the shift key down to select several items
- Analyse the boundary conditions: click the **Preprocessing/Boundary Conditions** submenu
- Visualize the Bathymetry and Manning
- Visualize other parameters such the water level, the water velocities and the Froude number
- Modify some input parameters: change the viscosity for instance
- Re-simulate: click the **Solution/Calculate** submenu
- Visualize the results again

## New Simulation

To perform a new simulation from scratch, follow the following steps, step by steps in order.

- Start a new simulation
- Defining the Boundaries
- Defining the Internal Nodes
- Defining the Mesh
- Optimizing the Mesh Bandwidth
- Inputting the Bathymetry
- Inputting the Manning Coefficient
- Inputting the Wind Fields
- Adjusting Physical Parameters
- Defining the Boundary Conditions
- Defining the Initial States
- Visualizing Input Field
- Adjusting the Solver Control Parameters
- Check the memory requirements
- Calculating a solution
- Visualizing solution

### Start a new simulation: File/New submenu

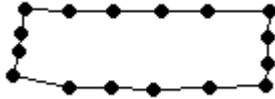
To start a new simulation project, first Run the Simulation System.  
From the **File** menu, choose **New**.  
In the Available Models dialog box, select AquaDyn and click OK.

### Defining the Boundaries: Preprocessing/Boundaries submenu

Enter the boundary of region used for the simulation.

First enter the external boundary (only one): click the **external boundary** command. Then click the left mouse button to insert first boundary node, drag the mouse and click the left mouse button to enter the second boundary node, and so forth. To exit the input mode, click the right mouse button and select the confirm command, the boundary will be closed automatically.

Here is an example of an external boundary:



Second enter the **internal boundaries** (islands) if desired: proceed as for the external boundary

```
*** put intbnd.gif
```

We can modify the boundary using the command **move, insert, delete node**. Remember to exit the move, insert or delete node command with the command **confirm** found by clicking the right mouse button.

The Boundaries can also be imported from an ASCII file using the **Read Boundary File** command.

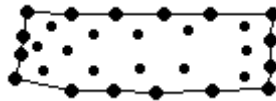
*Tip: keep the boundary nodes equally spaced, this will lead to a nicer mesh*

## Defining the Internal Nodes: Preprocessing/Mesh Nodes submenu

Enter the internal node which will be used to form the mesh.

For small mesh, it is best to enter the internal node manually with the command Create/Delete: click the **Create/Delete** Command. Then drag the mouse to the location you want a node and then click the left mouse button. To remove the node, click again over the node.

Here is an example of internal nodes:



*Tip: enter the internal nodes starting near the boundary nodes placing each internal nodes at equidistance from to boudnary node to form equilateral triangles.*

For large mesh, it is best to enter the internal automatically with the command Automatic: click the **Automatic** command. Then move, create or delete the nodes appropriately.

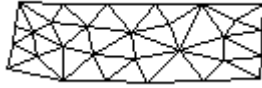
*Tip: adjust the position of the internal nodes near the boundary such they form equilateral triangles with the neighboring nodes.*

## Defining the Mesh: Preprocessing/Mesh submenu

To generate the mesh automatically click the **Auto Mesh** command.

To make the triangle equilateral click **Regulate (\*\*regulate the more than one\*\*)** .

Here is an example of the resulting mesh:



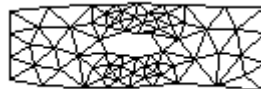
## Refining the Mesh: Preprocessing/Refine Mesh submenu

This submenu is useful for refining the mesh at some or all elements. Try the command **At Node**, **On Element**, **On Part of Mesh** or **Over all Mesh**. Remember, to exit the refining mode, click the right mouse button and select the confirm command.

\*\*\*Now to pursue the example above, it is important to refine the mesh near the island boundary because there are links which touch the internal and external boundary. Click the **On Element** command and select few elements as shown here,



than click the right mouse button and select the confirm command. Finally regulate the mesh few times. We get:



\*\*\*\*

Tip: if you do not like the change you have just made, use the **Edit/Undo** submenu.

## Optimizing the Mesh Bandwidth: Preprocessing/Bandwidth submenu

This step optimizes to the internal numbering of the mesh node in order to minimize the bandwidth of the resolution matrix. The purpose of this optimization is to reduce the amount of memory require by the solver.

Click the **Bandwidth** Command. A dialog Box appears.

Click **Optimize** button.

Within 1 to 30 seconds (depending on the mesh size) the **minimal width** will start decreasing. The optimization will stop automatically if the maximal number of iteration is reached (**Max**) or the number of iteration without a change in the minimal width (**Best**) is reached.

To stop the optimization manually, click **Stop**.

To close the dialog box, click **Done**.

Tip: For a large and complex mesh, set **Max** and the **Min** parameter to a very large number (say 10000) and let run for 5 to 10 minutes. Click **Stop** when the minimal width has not

*change for few minutes. Note that the minimal bandwidth expected is to be roughly equal to 15 times the number of element across the widest section of the mesh.*

### **Inputting the Bathymetry: Preprocessing/Fluids & Milieu submenu**

The Bathymetry defines the bed elevation of the water course you want to simulate. To define the Bathymetry, you need to enter the value of the Bathymetry at each mesh node.

Click the **Bathymetry** command.

Enter the node value of the bathymetry manually or import them from a file using the **File** Button. The node number appears on the mesh next to each node. The default value for the bathymetry is zero. You can keep the default values for now.

The bathymetry is the elevation measured from a given reference level. If the reference level is lower than the deepest point of the water course than the bathymetry will be positive for all nodes.

Tip: generally for real case studies, the bathymetry is imported from a file which contains a list of bathymetry measurement on the field of the form X, Y coordinates with the bed elevation. AquaDyn interpolates these values at the mesh nodes.

### **Inputting the Manning: Preprocessing/Fluids & Milieu submenu**

Follow the same step as for the Bathymetry.

Click the **Manning** command.

The Manning represent the water friction with the water course bed. A typical value for the Manning is 0.03. See the documentation for more details. You can keep the default values for now.

Tip: *the water course velocity varies roughly as 1 over the Manning. Therefore the manning is a powerful parameter for calibrating the simulation to match the measurements.*

### **Inputting the Wind Fields: Preprocessing/Fluids & Milieu submenu**

Follow similar step as the Bathymetry.

Click the **Wind** command.

You can keep the default values for now.

### **Adjusting Physical Parameters: Preprocessing/Fluids & Milieu submenu**

You can also modified the **gravity**, **coriolis**, **viscosity** and **turbulence** constants. You can keep the default values for now.

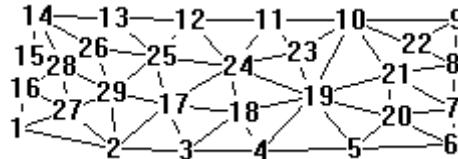
Tips: of these parameters, the viscosity is more important. The finite element method requires the viscosity to not be much smaller than 1, otherwise convergence to a solution may be difficult. Typical value of the viscosity range from 1 to 100. The viscosity is used to calibrate the solution. A large viscosity tends to smooth out the solution.

### **Defining the Boundary Conditions: Preproces./Bnd. Conditions Submenu**

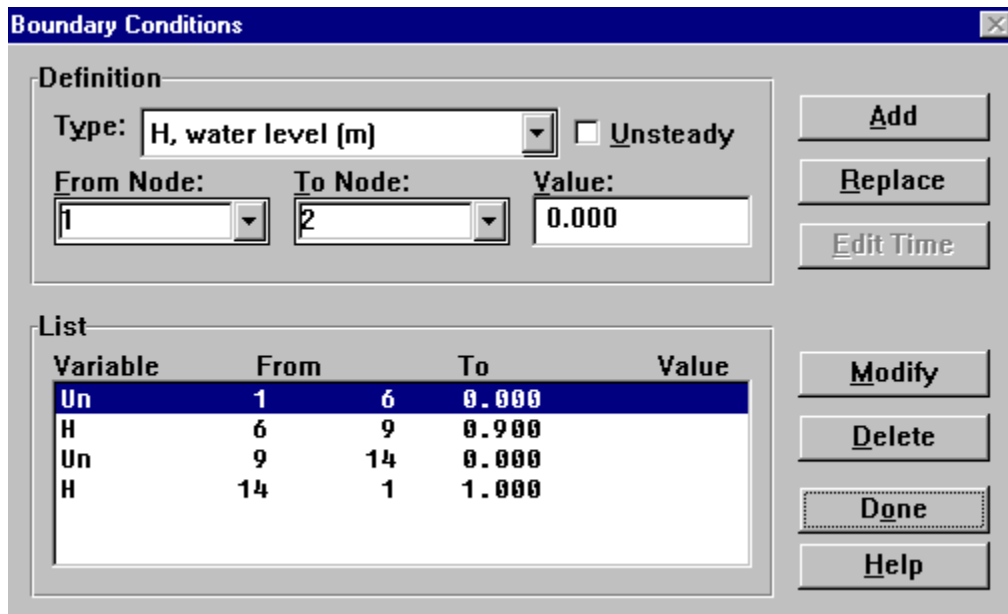
This step specifies the physical conditions to impose at the boundary of the mesh.

Click **Boundary Conditions** submenu. The dialog box which appears allows you to specify different types of boundary conditions along segments. The segments are defined by specifying the node number of the two end node always in a **counter clockwise fashion**.  
 \*\*\*For closed boundary the two end points happened to be the same node: fill the **From** and **To** control with the number node of any one node of the closed boundary.\*\*\*

For a simple river, you can specify the water level, H, along a segment upstream and a segment downstream\*\*\*, \*\*\*and\*\*\* require that the normal velocity be zero on the left and right coasts,\*\*\*and require the velocity be zero on the island\*\*\*. For example, given the node numbers:



you enter the following:



### Defining the Initial States: Preprocessing/Initial State submenu

Enter the initial configuration for the water level and water velocities. For unsteady simulation, the water elevation will evolve starting from this initial configuration. For steady simulation, the initial configuration is used as the first guess in the iterative process to find the exact solution.

Click the **Auto** button. This will set the water level automatically and reset the velocities to zero. The water depth being the difference between the water level and the bathymetry. So to prevent negative depth, we need the water level to be greater to the bathymetry.

Click the **All** button.

Set the initial velocities to your best guess. Level the water level unchanged.

Tip: *never leave the initial velocities to zero for steady state simulation: the solver will have difficulty to converge.*

## Visualizing Input Field

Once the initial state has been initialized, the Visualization menu becomes available. You can visualize. Use the visualization menu to visualize the water level, the water velocities, the Manning and the bathymetry entered. See Visualization section below for details on how to take advantage of this menu.

## Adjusting the Solver Control Parameters: Solution menu

For steady simulation, the solver starts from the initial state and through an iterative process try to obtain a converging solution.

For unsteady simulation, the solver uses an iterative process for each time step.

Click the **Precision** submenu and enter the precision desired for the water level and water velocities in the dialog box. You can leave the default values unchanged for now.

Click the **Convergence** submenu to enter the **Relaxation Parameter**, the **Minimum Water Depth** and the **Convection**. You can leave the default values unchanged for now.

Tip: *the relaxation parameter an important parameter to adjust to increase the stability of the convergence process. If the solution is not converging, decrease the relaxation parameter to 0.1 and try again. Note that to obtain convergence, you must first make sure have entered the best initial state you can come up with.*

## Check the memory requirements: Solution menu

Before launching the solver, verify the memory requirements.

Click the **Memory** command.

The memory required by the solver is indicated.

The available memory offered by the Simulation System is also indicated (it is not the memory available on your machine). If the available memory is not sufficient, click the **Option/Set Memory** submenu and enter the amount of memory you require in kilobyte. You must restart the Simulation System for this change to take effect. First save your project, exit the simulation system, restart the simulation system and load your project.

## Calculating a solution: Solution menu

To launch the calculation, click the **Calculate** submenu. A form, showing the convergence graph of the simulation appears.

To stop the simulation, click the **View/Stop Calculate** submenu. The computation will stop once the current iteration is completed (this may take few minutes for large simulation).

Note: if the principal window is on focus (the window with the mesh), the Stop submenu is under the Preprocessing menu.

Tip: *if the water level correction is higher than 10. Click the **View/Stop Calculate** submenu. And change your guess for the initial water velocities with the **Preprocessing/Initial State** submenu. Click **Calculate** again. Proceed iteratively till you find the initial water velocities which make the first water level correction minimal.*

## Visualizing solution

See Visualization section below for details on how to take advantage of this menu.

## Visualization

The visualization features are used to visualize the Input field data( Bathymetry, Manning and Wind components) , the computed solution (Water Level, X and Y water velocities), and several mathematical combinations of those fields (i.e.: Water Depth, Froude Number, etc).

When the Visualization menu is enabled a specific parameter (or field data) can be visualized at a specific time as (all submenus of the Visualization menu)

Section:	graphic representing the variation of the parameter between two spatial points
Values:	show the values of the parameter at all nodes or elements
Isovalue:	colored regions each representing a specific range of values
Contour Line:	lines along which the parameter as a specific value
Time Curve:	graphic representing the time variation of the parameter at a specific node

Moreover: the water velocity field can viewed as an **ARROW** field and the water **discharge** across a section can be computed

We explain here how to exploit visualization features.

You need first to select a parameter using the Select Parameter Submenu. If you don't select one, you will be prompt later to select it or the current selected parameter is used. The Select Parameter dialogue box offers you a choice of three classes (Hydraulic, Engineering and Numerical); choose one class and then select a parameter from that class and press OK.

If there is a current solution which is the result of an unsteady simulation, you can select the time at which you want to visualize the parameter. If you don't select one, the current time is taking by default. If the solution results from a steady simulation the select time will not be accessible.

Now you need to select one of the 5 visualization options: pick one of the 5 submenu which we describe briefly below:

### Section: Visualization/Section submenu

To define the section along which you want to see the spatial variation of the selected parameter select the command **Using Mouse** or **Using Coordinate**. Select the command **Using Mouse** to select the points by clicking with the left mouse button or select the cascade menu **Using Coordinate** to specify the coordinate of the two points manually.

### Value: Visualization/Value

click **Value** submenu.

### Isovalue: Visualization/Isovalue

click **Isovalue** submenu

### Contour Line: Visualization/Contour Line

click **Coutour Line** submenu

### **Time Curve: Visualization/Time Curve**

click **Time Curve** submenu

To visualize the water velocity as an arrow field, pick the **Speed** Submenu

To obtain the water discharge across a section which you define select the **Discharge** Submenu. The section can be defined by specifying to points. Select the command **Using Mouse** to select the points by clicking with the left mouse button or select the command **Using Coordinate** to specify the coordinate of the two points manually.