

EcoDynamo

Ecological Dynamics Model Application



António Pereira & Pedro Duarte

*University Fernando Pessoa
CEMAS - Centre for Modelling and Analysis of
Environmental Systems*

January 2005

Table of Contents

Introduction	3
Main Window	4
Model Menu	5
New Model	6
Open Model.....	7
Close Model	8
Remove Model	9
Defaults Model	10
Exit	11
Specs Menu	12
Morphology.....	13
Time specs	14
Classes	16
Variables.....	17
Sub Domain.....	18
Enable / Disable Communications Area	19
Output Menu	20
Output File.....	21
Output Chart.....	23
Output Table.....	24
Output Logs.....	25
Execute Menu	26
Help Menu	27
About.....	28
Simulation Panel	29
Output Panel.....	30
Communications Area.....	31
Configuration Files	32
Morphology File.....	33
Classes File.....	35
Variables File.....	36
Parameters File	37
Loads File.....	38
Points File.....	39
Output Files.....	40
General Output File	41
Mean Values Files.....	42
Log Files.....	43
Properties Files	45
Model Properties File	46
EcoDynamo Properties File.....	47

Introduction

EcoDynamo (Ecological Dynamics Model) is an application built to enable physical and biogeochemical simulation processes of aquatic ecosystems. It's an object oriented program application, built in C++ language, with a shell that manages the graphical user interface, the communications between classes and the output devices where the simulation results are saved.

The simulated processes include:

- hydrodynamics of aquatic systems: current speeds and directions;
- thermodynamics: energy balances between water and atmosphere and water temperature;
- biogeochemical: nutrient and biological species dynamics;
- anthropogenic pressures, such as biomass harvesting.

The ecosystem characteristic properties are described in a model database (Configuration Files): morphology, geometric representation of the model, dimensions, number of cells, classes, variables, parameter initial values and ranges.

The user can choose between file, chart or table to store the simulation results. These output formats are compatible with some commercial software (like MatLab[®]) products, enabling their posterior treatment.

Different classes simulate different variables and processes, with proper parameter and process equations. Classes can be selected or deselected from shell dialogs determining its inclusion or exclusion in each simulation run of the model.

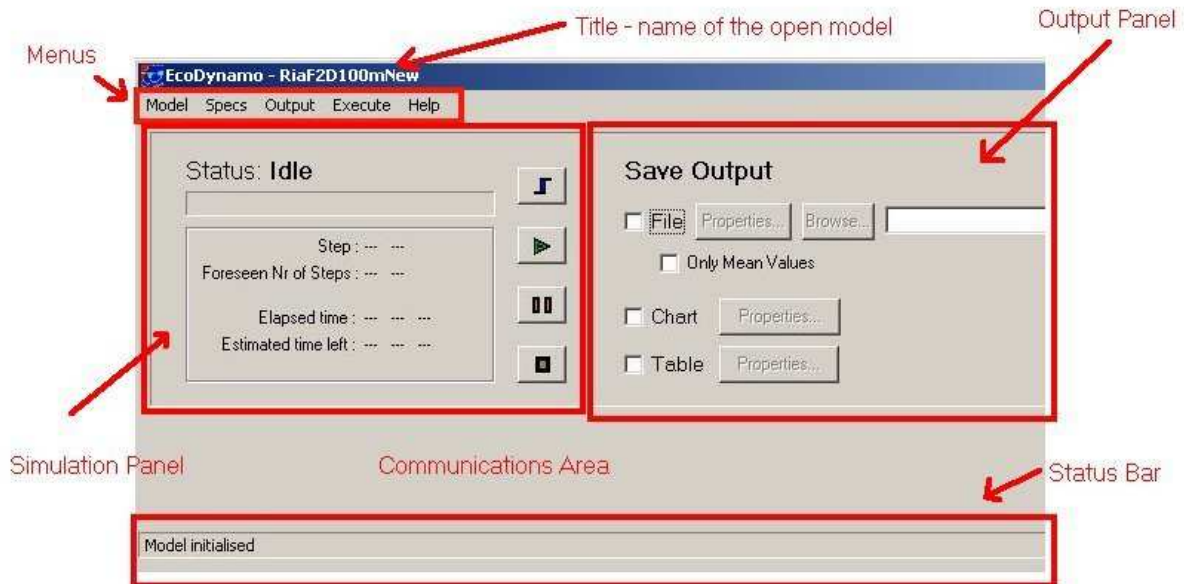
This application has an interface module (implementing the EcoDynamo Protocol) that enables communications with other programs for external control. For example, the simulation runs can be controlled by commands like start/stop/pause/restart/step simulation.

Simulation activity can be spied with the help of log files, activated previously before the simulation run.

This document describes the operation of the EcoDynamo application and the files used for configuration, properties and output results.

Main Window

The EcoDynamo's main window is the privileged interface for the user to manipulate the models simulation.



When a model is opened, its name is added to the name of the application in the title area.

The Main Window has five principal areas: Menu, Simulation Panel (also called Execute Panel), Output Panel, Communications Area and Status Bar.

The Status Bar area indicates the current task or the last action made by EcoDynamo. The other areas are described in detail in the following pages.

Model Menu

The Model Menu is used to define new models, open and remove existing models, close the working model and set the default model when the application starts.

The exit command is also present in this menu.



New Model... - define a new model with proper Configuration Files.

Open Model... - open a defined model.

Close Model - close the working model.

Remove Model... - remove an existing model.

Defaults Model... - set the default model for the application.

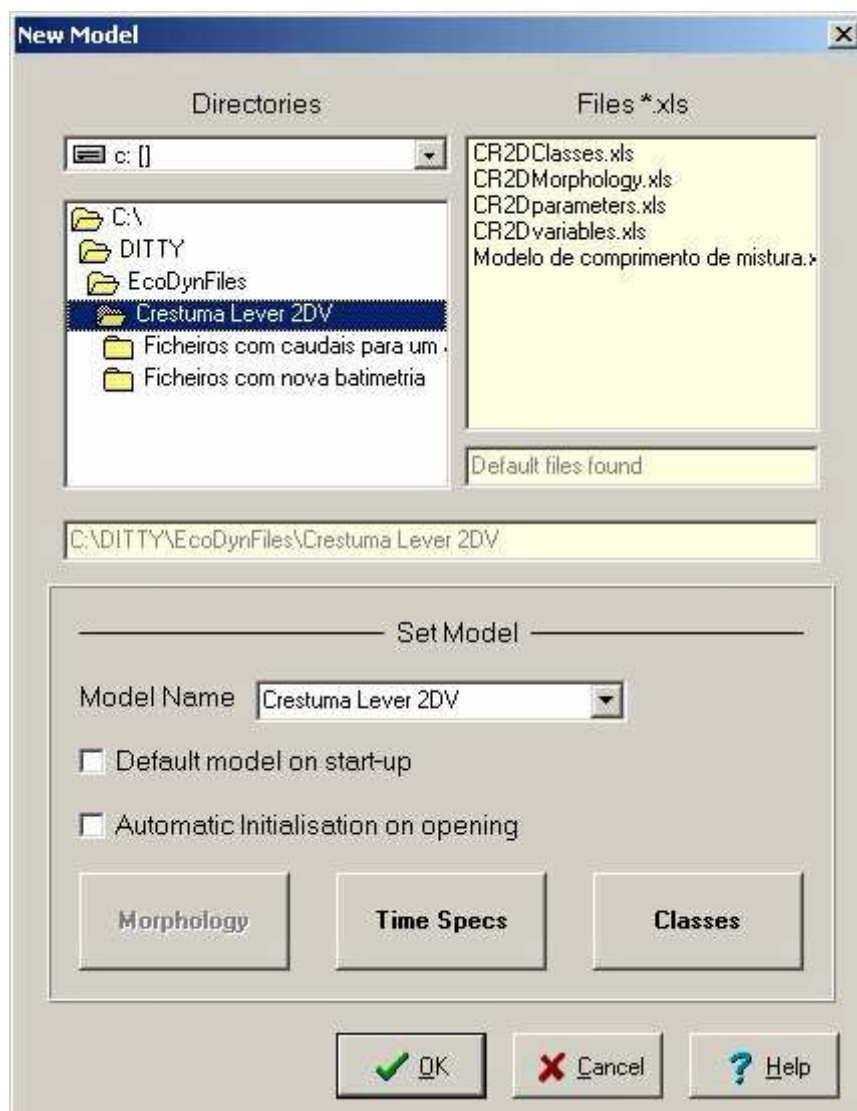
Exit - exits EcoDynamo application.

New Model

The New submenu opens a dialog where the user defines a new model for simulation.

In this dialog the user must browse for the directory where the model is defined (Configuration Files present), supply a name for the model, define if the model will be the default model on start-up of the EcoDynamo application and if the model is automatically initialised on opening.

When the entire mandatory Configuration Files are present in the directory browsed, the dialog looks like (if some configuration file misses background will be grey):



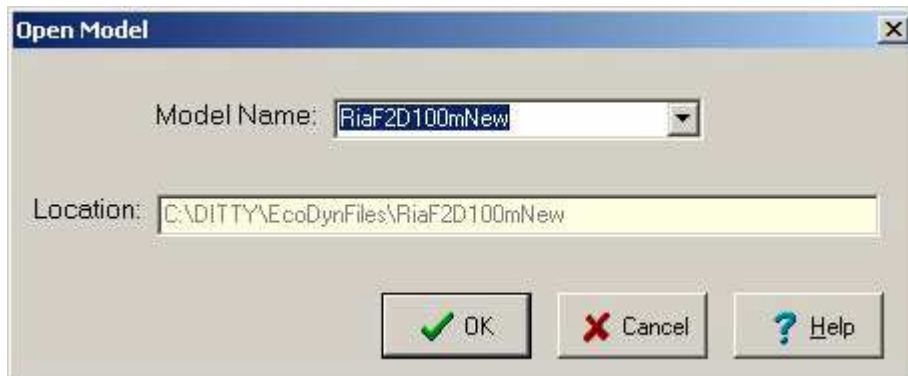
The user can select the Classes, Variables and Time specs for the simulation.

After click OK button, one entry is added to the Model Properties File.

Open Model

The Open submenu opens a dialog where the user selects one already created model for simulation (saved in Model Properties File).

If the model was defined as automatically initialised on opening, it is initialised after the click on the OK button.



If one model was in use, the EcoDynamo Properties File is saved in the model directory, with the simulation properties (Time specs, Classes and Variables).

Close Model

The Close menu closes the working model.

The EcoDynamo Properties File, with the simulation properties, is saved in the model directory.

Remove Model

The Remove submenu opens a dialog where the user selects one already created model for deletion.

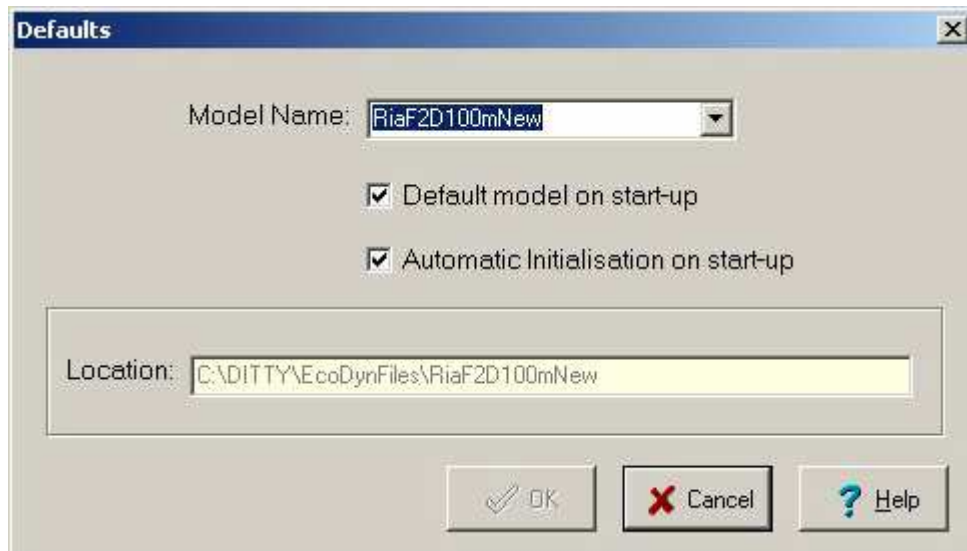
The corresponding entry in the Model Properties File is removed.



Defaults Model

The Defaults submenu opens a dialog where the user selects the default behaviour for each model and the default model on start-up.

The corresponding entries in the Model Properties File are changed.



For the default model on start-up the background colour of the Location textbox is changed to smooth yellow.

Exit

This option exits the EcoDynamo application.

If one model is in use, the corresponding EcoDynamo Properties File is saved.

Specs Menu

The Specs Menu is used to define the simulation options for the model run, like time specs, classes selected, variables to output and sub domain of the model.

The user can enable/disable the communications to trace the messages between EcoDynamo and the other applications (behind the EcoDynamo Protocol) on the communications area.



Morphology... - check morphology values

Time specs... - define time specs for simulation and output register.

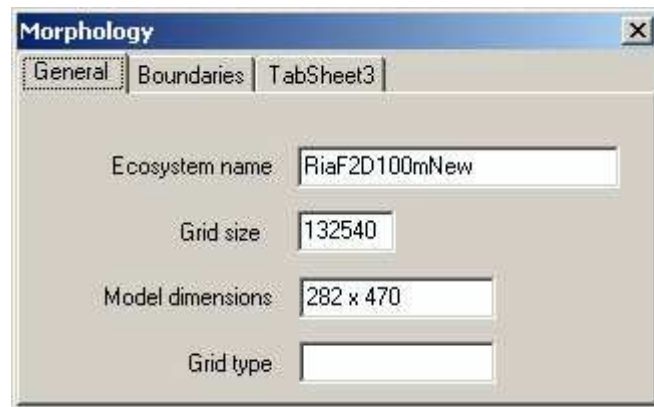
Classes... - select Classes for simulation and Variables for output.

Sub Domain... - select domain for simulation.

Enable / Disable Communications Area - enable / disable the communications area.

Morphology

This dialog shows the model's general morphology - name, grid size, number of lines vs. number of columns.



The information of this panel is filled only after the model initialisation.

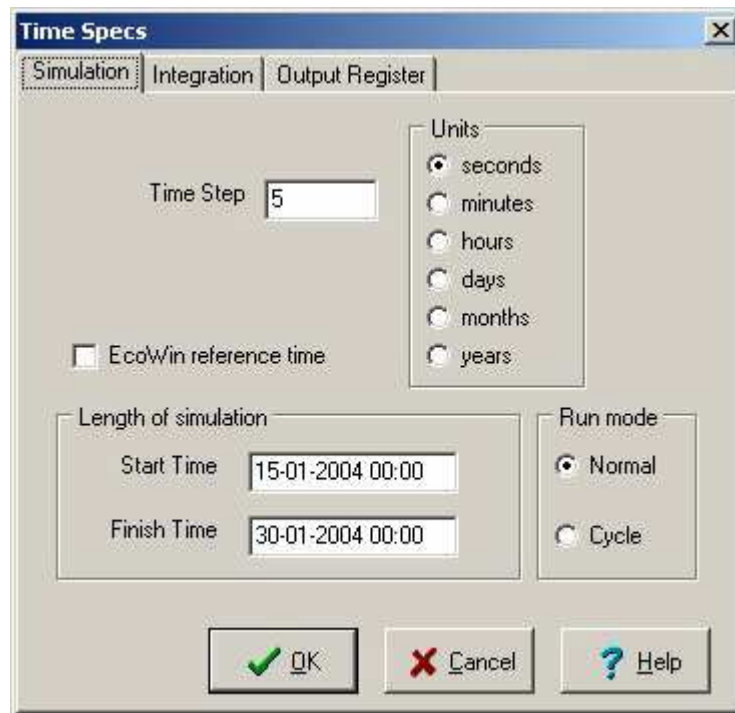
The panels "Boundaries" and "TabSheet3" are not used in this version of EcoDynamo.

The field "Grid type" is not used in this version of EcoDynamo.

Time specs

The Time Specs submenu opens a dialog where the user defines the time specifications for simulation and output in different tabs.

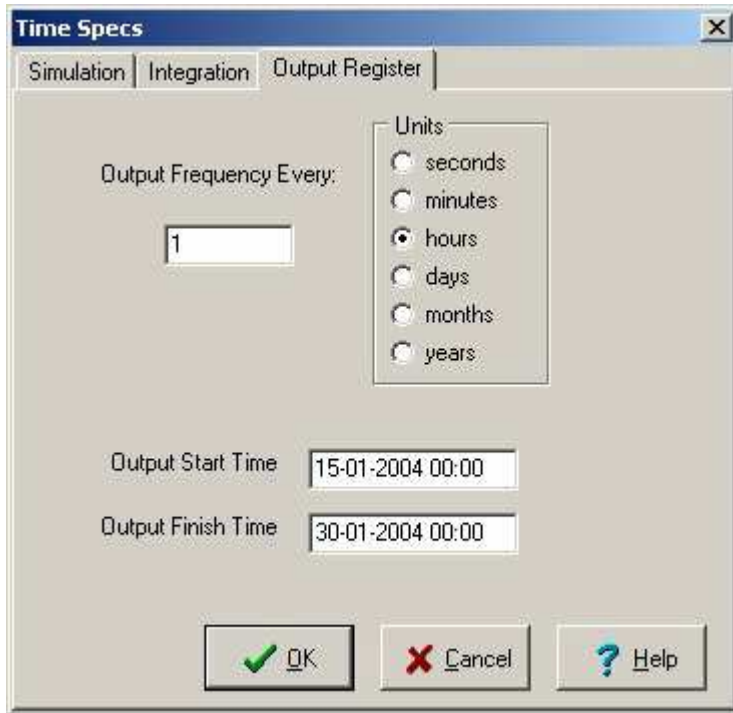
The Simulation tab selects the simulation time step and length:



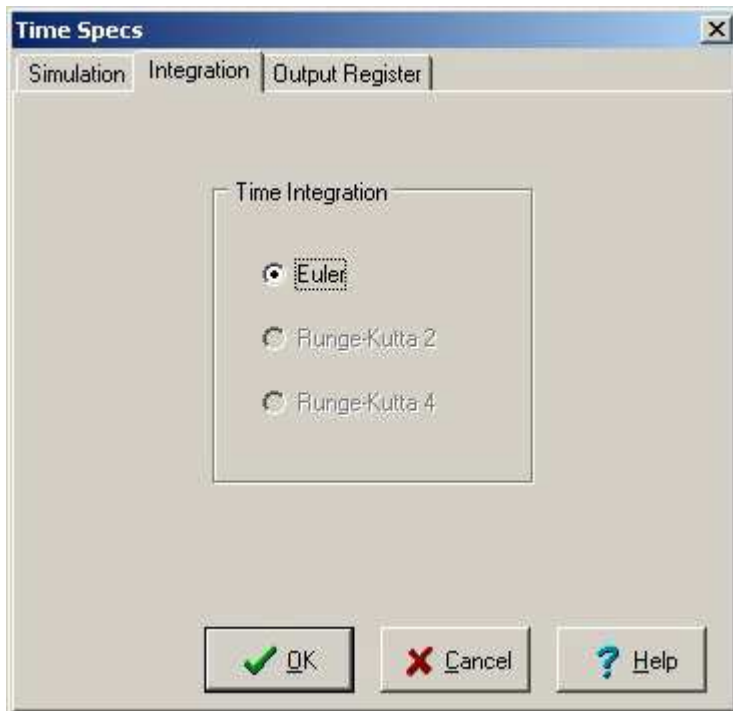
If the checkbox "EcoWin reference time" is selected the time for each time step is based on the EcoWin[®] reference time.

The "Run mode" option is not yet implemented.

The Output Register tab select the output register frequency and length:



The Integration tab is:

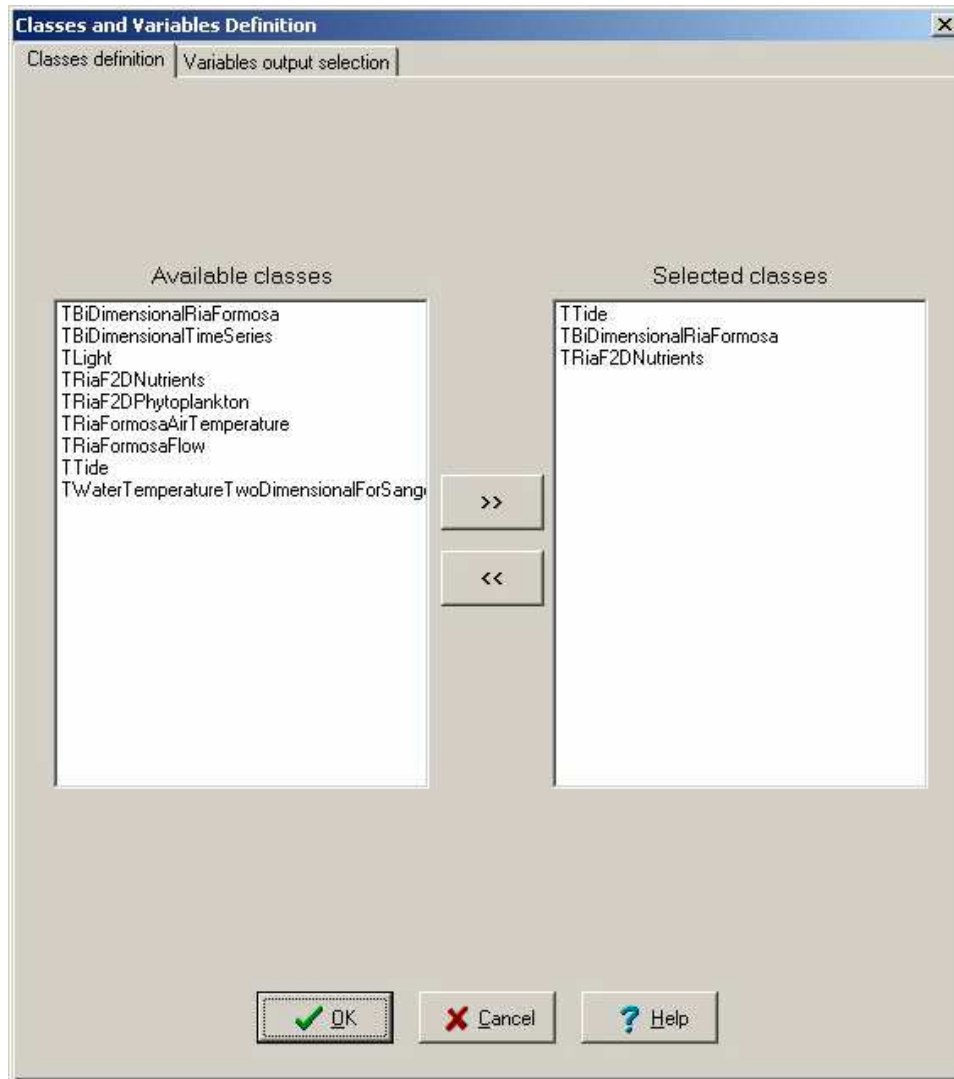


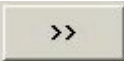
Only "Euler" integration mode is implemented in this version.

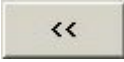
Classes

The Classes submenu opens a dialog where the user selects the classes for simulation and the variables to output results in different tabs.

The Classes definition tab selects the classes for simulation:



The  button selects classes for simulation (transfers the classes marked in the "Available classes" list to the "Selected classes" list).

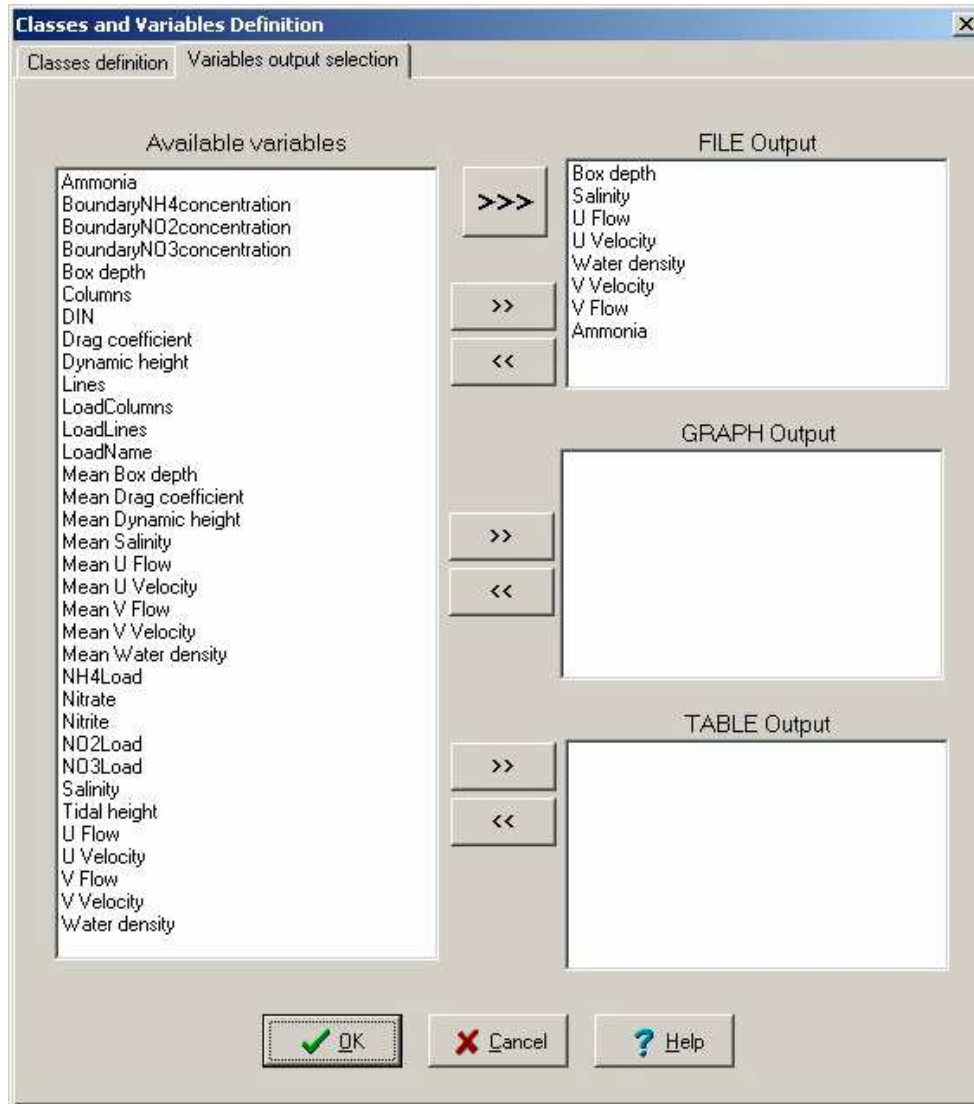
The  button deselects the classes marked in the "Selected classes" list.


In the "Variables output selection" tabbed panel the user selects the variables for output register.

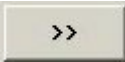
Variables


From the Classes submenu the user selects the variables for output results.

The tab for variables output selection is:



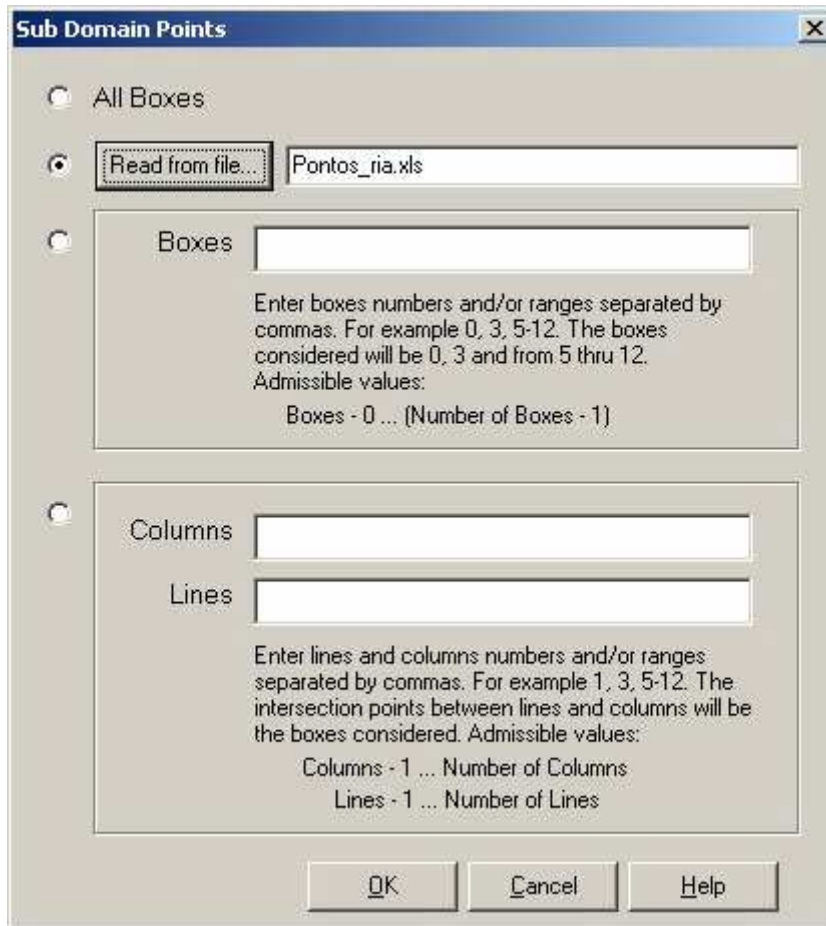
The  button transfers the selected variables in the "Available variables" area to all output options.

Each  button copies the variables selected in "Available variables" list to the related output area.

Each  button removes the variables selected in the corresponding output area.

Sub Domain

The Sub domain submenu opens a dialog where the user selects the points for simulation:



The different exclusive options for points selection are:

"All Boxes" - complete domain.

"Read from file..." - read sub domain points from a file. Button click opens a dialog enabling the user to browse the file's location. Files must have the format described in Points File to be considered.

"Boxes" - select only the boxes listed in the textbox.

"Columns" and "Lines" - the boxes selected are the intersection of the line and column numbers listed in the textboxes.

Enable / Disable Communications Area

This submenu shows / hides the Communications Area panel.

The selection of this menu when its text is "Enable Communications", displays the communications area panel and its text changes to "Disable Communications".

The selection of this menu when its text is "Disable Communications", hides the communications area panel and its text changes to "Enable Communications".

Output Menu

The Output Menu is used to select the options for register the model output results.

The user can choose different output devices like file, chart and table.

It is also possible to log the messages exchanged between the different classes and at different simulation steps.



Output File... - select output filename and register points.

Output Chart... - select output chart register points.

Output Table... - select output table register points.

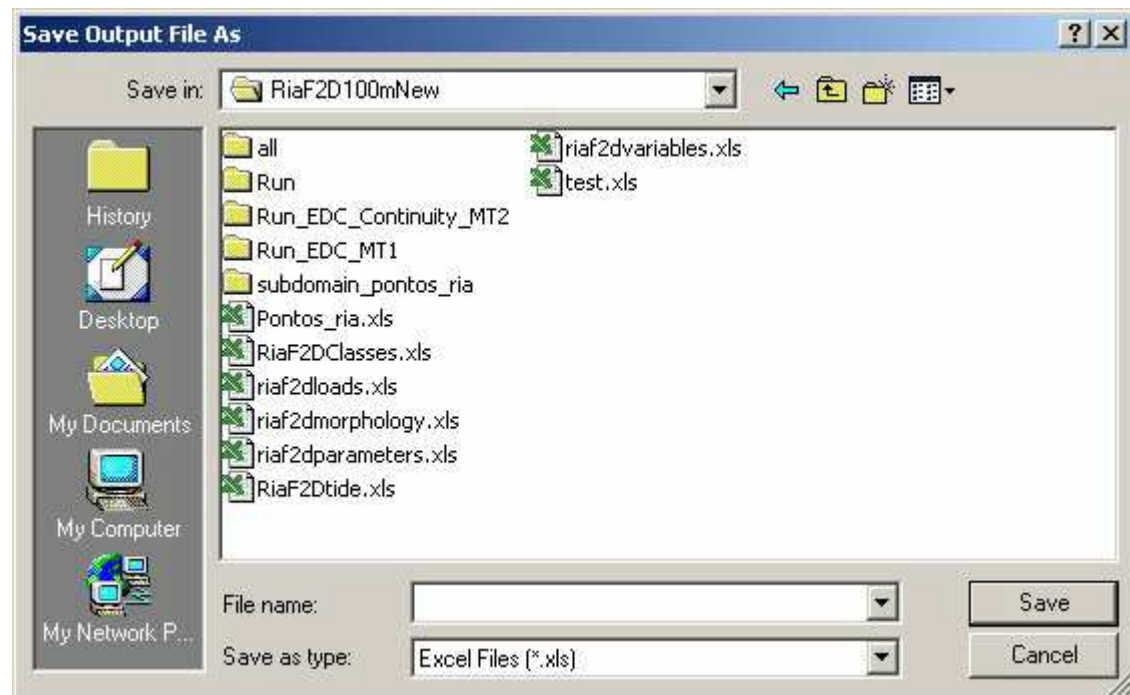
Output Logs... - select log simulation steps and file type

The selection of the output options can be done in the Output Panel

Output File

The File submenu selects the filename and the boxes for register the simulation variables output results in two steps:

Step 1 - Select file name



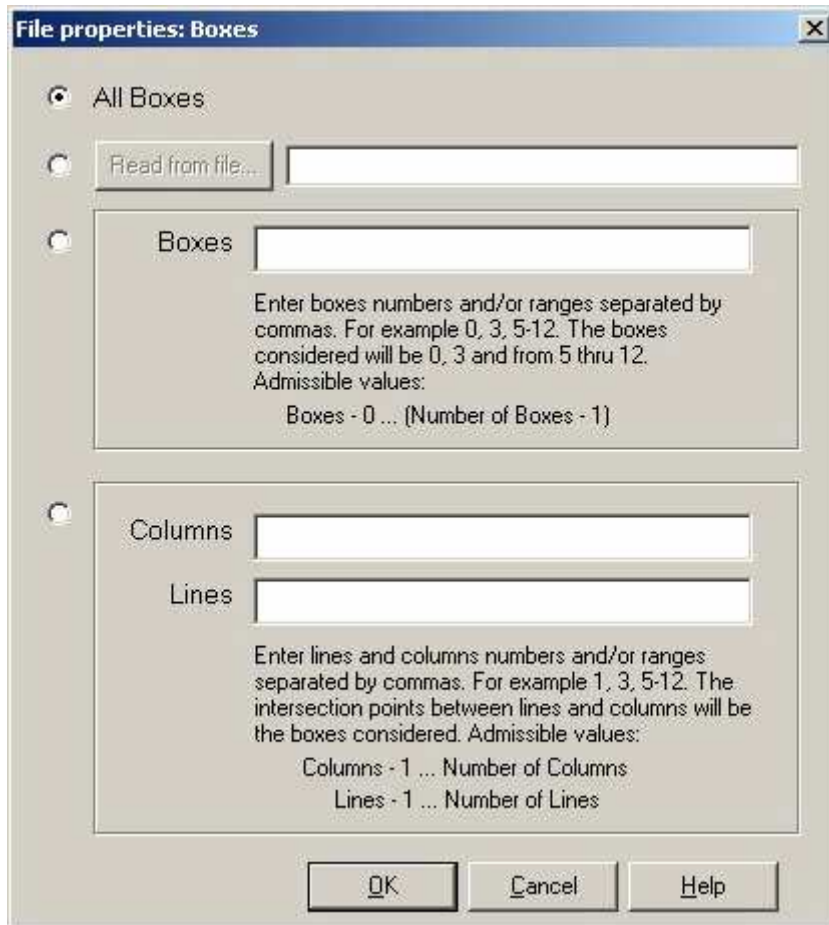
The file types can be "xls", "hdf" or "txt".

The "xls" format is obtained with Tab Separated Values and is similar to the "txt" format.

The "hdf" format follows the HDF specifications - see NCSA http server (<http://hdf.ncsa.uiuc.edu/>)

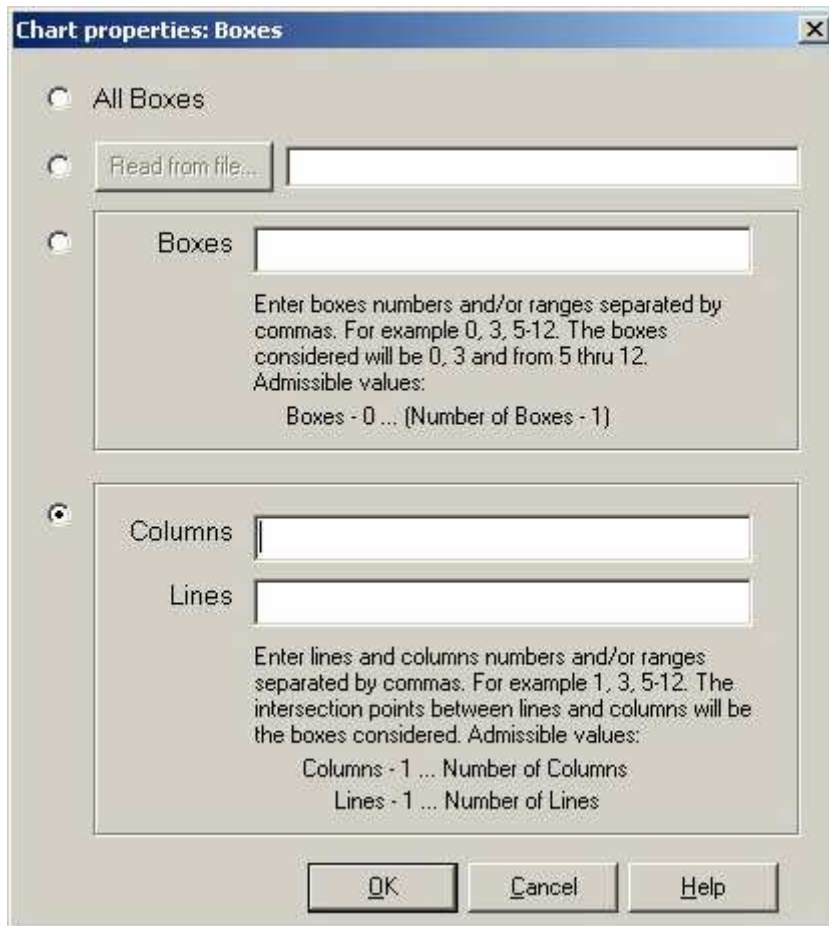
Step 2 - Select points to register

After entering the filename and click on the "Save" button, one dialog (similar to Sub Domain) appears for select the register points:



Output Chart

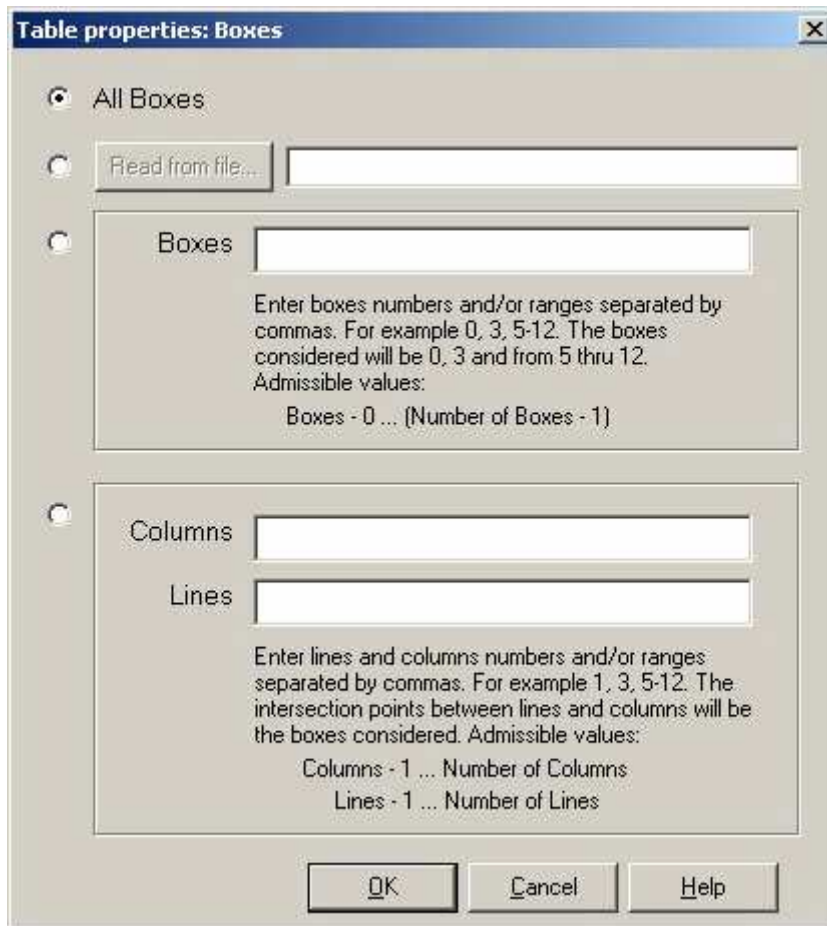
The Chart submenu selects the boxes for register the simulation variables output results in chart format. One dialog (similar to Sub Domain) appears for select the register points:



To create the output charts, one for each variable, the EcoDynamo launches the MatLab[®] application and benefits from all the features that MatLab[®] supplies to manipulate the chart images.

Output Table

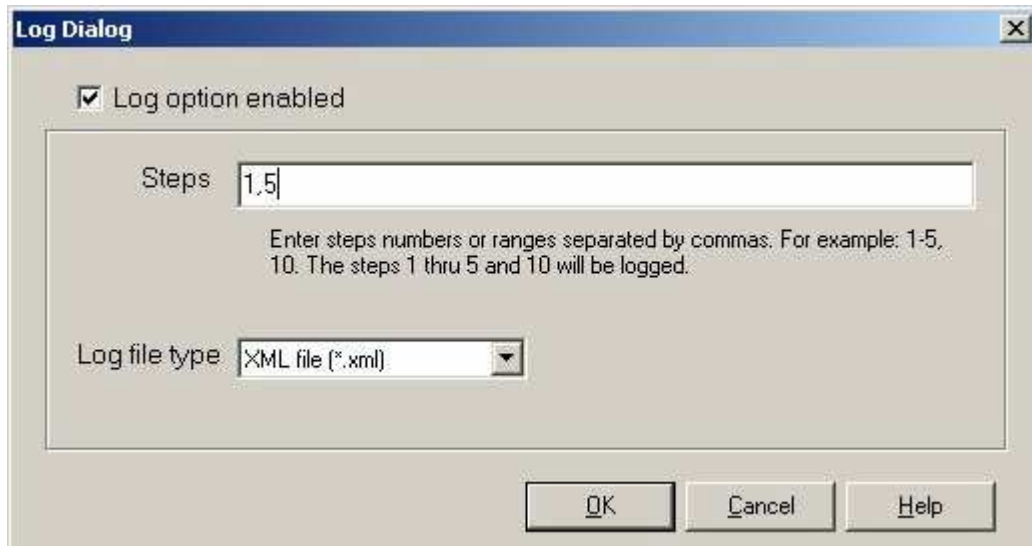
The Table submenu selects the boxes for register the simulation variables output results in table format. One dialog (similar to Sub Domain) appears for select the register points:



The table output format isn't implemented in this version.

Output Logs

The Logs submenu presents a dialog to specify the steps where the messages exchanged between the different classes are logged:



The log file format can be specified by the user within "xml", "xls" and "txt".

The "xls" format is obtained with Tab Separated Values and is similar to the "txt" format.

The "xml" format is more elaborated and is used for more sophisticated post-processing methods.

The format of the file behind each type is described in Log Files.

Execute Menu

The Execute Menu controls the simulation runs.



Initialise - initialise model for simulation.

Step - run the simulation step by step.

Run - run the simulation.

Pause - pause the simulation.

Stop - stop / end the simulation.

The control of the simulation runs can be done in the Simulation Panel

Help Menu

The Help Menu presents the contents of the manual and the about screen.



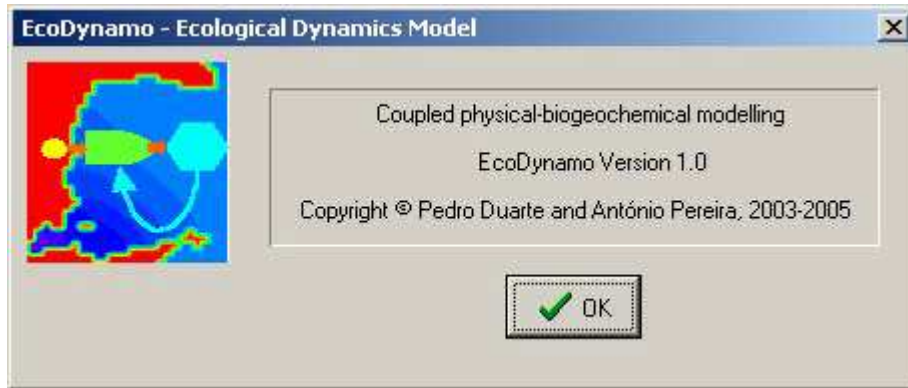
Contents - presents this help manual.

Commands - presents the main window commands

About... - presents the application's authors, version and rights information.

About

The About Menu presents the application's authors, version and rights:



Simulation Panel

The Simulation Panel presents the simulation state and enables the control of the simulation runs. This panel synthesizes the Execute Menu.



The buttons have the general symbols used in hi-fi control panels:



Step button - run one step of the simulation.



Run button - execute the simulation until stopped or paused.



Pause button - exchange between pause and run mode.



Stop button - stops the simulation run.

The labels has the following meaning:

"Status" - simulation status (Idle, Running, Paused, Stopped).

"Step" - simulation step number.

"Foreseen Nr of Steps" - expected number of steps for all simulation.

"Elapsed time" - simulation time from the beginning.

"Estimated time left" - presents the expected time for simulation end.

Output Panel

The Output Panel presents the outputs selected for the simulation run and enables the control of the output type.



The checkboxes enable / disable the corresponding output device (File, Chart and Table).

Each **Properties...** button opens the dialog to select the points for the corresponding output (see Output File, Output Chart and Output Table).

The **Browse...** button opens the dialog to select the filename of the output file (see Output File). The filename is printed in the combo box area.

The checkbox "Only Mean Values" enables a special kind of output register that saves only the mean values (see Mean Values Files). In this case nothing is done with the filename.

This panel is an extension of the Output Menu.

Communications Area

The Communications Area panel enables the visualisation of the messages exchanged between EcoDynamo and other applications.



In the area "MSGs:" the messages received and transmitted are displayed with the prefixes "R: " and "T: ", respectively, followed by the name of the host and the message.

The user can format messages and send them to other applications. The messages must follow the EcoDynamo Protocol format and are entered in the "Send MSG:" area; when the "Enter" key is pressed, the message is sent.

For information on messages format see the EcoDynamo Protocol documentation.

Configuration Files

The configuration files are the files used to initialise and run the model for simulation. Each file has its own significance and utility.

The mandatory files are:

1. Morphology File - describe the morphology of the ecosystem, including the number of cells, localisation, geometry and boundary types of each one.
2. Classes File - list all classes that can be used by the simulation.
3. Variables File - list the variables treated by each class and their initial values.
4. Parameters File - list the parameters used by each class and their initial, minimum, maximum and increment values.

The next files are not mandatory but can be useful in some ecosystems or simulations:

5. Loads File - requested when the loads process is active.
6. Points File - list sub domain points.

All these files are saved in text format, with Tab Separated Values, and can be accessed and modified by any text editor or commercial application (Excel, Word, Wordpad, Notepad,...).

The name of the first five files must have a common prefix; the suffix distinguishes the file type – see following pages.

Morphology File

The morphology filename must terminate with the suffix "Morphology.xls".

It must have the format showed in the next figure:

	A	B	C	D	E	F	G	H	I	J	K	L
1	NumberOfColumns	470										
2	NumberOfLines	282										
3	NumberOfBoxes	132540										
4	Columns	Lines	BoxDepth	BoxLength	BoxWidth	BoxElevation	BoxType	Nboundary	Eboundary	Sboundary	Wboundary	ModelType
5	1	282	57.62843	100	100	2.628824	1	0	0	2	2	
6	2	282	57.62843	100	100	2.628824	1	0	0	2	0	
7	3	282	57.62843	100	100	2.628824	1	0	0	2	0	
8	4	282	57.62843	100	100	2.628824	1	0	0	2	0	
9	5	282	57.62843	100	100	2.628824	1	0	0	2	0	
10	6	282	57.62843	100	100	2.628824	1	0	0	2	0	
11	7	282	57.62843	100	100	2.628824	1	0	0	2	0	
12	8	282	57.62843	100	100	2.628824	1	0	0	2	0	
13	9	282	57.62843	100	100	2.628824	1	0	0	2	0	
14	10	282	57.62843	100	100	2.628824	1	0	0	2	0	
15	11	282	57.62843	100	100	2.628824	1	0	0	2	0	
16	12	282	57.62843	100	100	2.628824	1	0	0	2	0	
17	13	282	57.62843	100	100	2.628824	1	0	0	2	0	
18	14	282	57.62843	100	100	2.628824	1	0	0	2	0	
19	15	282	57.62843	100	100	2.628824	1	0	0	2	0	
20	16	282	57.62843	100	100	2.628824	1	0	0	2	0	
21	17	282	57.62843	100	100	2.628824	1	0	0	2	0	
22	18	282	57.62843	100	100	2.628824	1	0	0	2	0	
23	19	282	57.62843	100	100	2.628824	1	0	0	2	0	
24	20	282	57.62843	100	100	2.628824	1	0	0	2	0	
25	21	282	57.62843	100	100	2.628824	1	0	0	2	0	

The field names **NumberOfColumns**, **NumberOfLines** and **NumberOfBoxes** are mandatories and the columns that follow each one of them define the dimensions of the model grid:

NumberOfColumns - number of cells in the longitude direction

NumberOfRows - number of cells in the latitude direction

NumberOfBoxes - product of the two previous values

After the model initialisation these values are displayed in the dialog of the Morphology menu.

There must be a headline with the following mandatory labels:

Columns - number of cell column

Lines - number of cell line

BoxDepth - cell depth (meters)

BoxLength - cell length (meters)

BoxWidth - cell width (meters)

BoxElevation - cell elevation of free surface (meters)

BoxType - cell type (1 - border cell, 0 - inner cell)

Nboundary - cell North boundary type

Eboundary - cell East boundary type

Sboundary - cell South boundary type

Wboundary - cell West boundary type

Boundary types are coded like:

0 - no frontier

- 1 - river boundary
- 2 - sea boundary
- 4 - solid boundary (earth)

The column with the label **ModelType** indicates the model type (p.e. 2DH means two-dimensional horizontal). In this version of EcoDynamo this field is not used.

The following lines are filled with the values for the corresponding fields.

NOTE: The grid is viewed as a rectangle:

line number increases from North to South (*top-down*)

column number increases from West to East (*left-right*).

Classes File

The classes filename must terminate with the suffix "Classes.xls".

It must have the format showed in the next figure:

	A	B
1	Number of Classes	9
2	Class name	Status
3	TTide	Active
4	TBiDimensionalRiaFormosa	Active
5	TRiaF2DNutrients	Active
6	TRiaF2DPhytoplankton	Active
7	TLight	Active
8	TWaterTemperatureTwoDimensionalForSango	Active
9	TRiaFormosaAirTemperature	Active
10	TBiDimensionalTimeSeries	Active
11	TRiaFormosaFlow	Active
12		
13		
14		

The labels **Number of Classes**, **Class name** and **Status** are mandatory.

The column that follows the label **Number of Classes** must define the number of classes that will be used by the model.

The line with labels **Class name** and **Status** head those with the classes names and their initial state of activation. There must be an agree between the number of classes defined and the number of lines with class names.

The names of the classes are the names of the class objects defined by each DLL, included with the EcoDynamo code, which implements the simulation of the different processes.

In this version of EcoDynamo the field **Status** is not taken into account because the class activation is done in the Classes menu and activation changes are saved when the model is closed in the EcoDynamo Properties File.

Variables File

The variables filename must terminate with the suffix "Variables.xls".

This file lists all the classes' variables, with their initial values, and each class imposes its own format for the variables filling.

For further information read the EcoDynamo manual.

Parameters File

The parameters filename must terminate with the suffix "Parameters.xls".

This file lists all the classes' parameters and their initial, minimum, maximum and increment values, and each class imposes its own format for the parameters filling.

For further information read the EcoDynamo manual.

Loads File

The loads filename must terminate with the suffix "Loads.xls".

It must have the format showed in the next figure:

	A	B	C	D	E	F	G	H
1	NumberOfLoads	NumberOfDaysForLoads	LoadLines	LoadColumns	LoadName	Flow1	Flow2	Flow3
2	3	365	72	94	ETAR1	0	0	0
3			73	94	ETAR2	0	0	0
4			49	147	ETAR3	0	0	0
5						0	0	0
6						0	0	0
7						0	0	0
8						n	n	n

The field names **NumberOfLoads**, **NumberOfDaysForLoads**, **LoadLines**, **LoadColumns** and **LoadName** are mandatories:

NumberOfLoads - The line behind contains the number of load discharges points.

NumberOfDaysForLoads - The line behind contains the number of days with load discharges.

LoadLines - The lines behind contain the line numbers of each load discharge point. The number of lines filled must agree with the number of loads defined.

LoadColumns - The lines behind contain the column numbers of each load discharge point. The pair Line x Column defines the cell point. The number of lines filled must agree with the number of loads defined.

LoadName - The lines behind contain the names of the load discharges entities. The number of lines filled must agree with the number of loads defined.

For each discharge point, one column labelled **FlowN** must be added to the file (**N** is the flow number) with **NumberOfDaysForLoads** lines, one load value for each day.

NOTE: The grid is viewed as a rectangle:

line number increases from North to South (*top-down*)

column number increases from West to East (*left-right*).

Points File

The points file should have a format like the one showed in the next figure:

	A	B	C	D
1	X	Y	COLUNA_X_	LINHA_Y_
2	222000	-400	131	251
3	222100	-400	132	251
4	222200	-400	133	251
5	221700	-300	128	250
6	221800	-300	129	250
7	221900	-300	130	250
8	222000	-300	131	250
9	222100	-300	132	250
10	222200	-300	133	250
11	222300	-300	134	250
12	222400	-300	135	250
13	222500	-300	136	250

The field names **COLUNA_X_** and **LINHA_Y_** are mandatory:

COLUNA_X_ - The lines behind contain the column numbers of each point. The pair Line x Column defines the cell point.

LINHA_Y_ - The lines behind contain the line numbers of each point. The pair Line x Column defines the cell point.

In the example shown above the values of columns **COLUNA_X_** and **LINHA_Y_** were built from X and Y (extracted with the help of one GIS application).

NOTE: The grid is viewed as a rectangle:

line number increases from North to South (*top-down*)

column number increases from West to East (*left-right*).

Output Files

The output files are the files generated by the EcoDynamo application during the simulation.

The results can be saved in one General Output File or several Mean Values Files.

The Log Files save the communications between classes during the simulation in some specified steps.

General Output File

The General Output File have, by default, 3 formats: "xls", "hdf" or "txt".

The "xls" and "txt" formats are text files saved with tabs separating values. The "xls" format is used for Excel application quick view with Tab Separated Values. The format is:

	A	B	C	D	E	F	
1	Time(hours)	TimeStep	GridColumn	GridLine	BoxNumber	DIN	Phyto
2	1	1	1	35	0	0	
3	1	1	2	35	1	0	
4	1	1	3	35	2	0	
5	1	1	4	35	3	0	
6	1	1	5	35	4	0	
7	1	1	6	35	5	0	
8	1	1	7	35	6	0	
9	1	1	8	35	7	0	
10	1	1	9	35	8	0	
11	1	1	10	35	9	0	
12	1	1	11	35	10	0	
13	1	1	12	35	11	0	
14	1	1	13	35	12	0	
15	1	1	14	35	13	0	
16	1	1	15	35	14	0	
17	1	1	16	35	15	0	
18	1	1	17	35	16	0	
19	1	1	18	35	17	0	
20	1	1	19	35	18	0	
21	1	1	20	35	19	0	
22	1	1	21	35	20	3.529125	
23	1	1	22	35	21	3.529125	

The header row contains the fields significance. The first five columns are fixed for all registers: time (in hours), register step, column, line and cell numbers.

After the fifth column appear the variables in the order selected in Variables Dialog to file output.

Only the cells selected in the Points File are registered.

The "hdf" format follows the HDF specifications - see NCSA http server (<http://hdf.ncsa.uiuc.edu/>). When this option is chosen all the points of the model domain are saved.

Mean Values Files

The option to generate only Mean Values Files enables the user to run the model with a minor time step (normally only the hydrodynamic part of the model) and save the mean values of flows and velocities in files. To do that the classes file must include classes with this feature implemented.

The user will use this files to run the model later with a time step greater than the used in the previous simulation.

The Mean Values Files are formatted as "xls" files and their names follow a special order. Each file saves 10 register steps.

The first file to be saved has the name "HydroTimeSeriesValues_0.xls", the second "HydroTimeSeriesValues_1.xls", and so on.

	A	B	C	D	E	F	G
1	Time(seconds)	TimeStep	BoxNumber	Mean U Flow	Mean U Velocity	Mean V Flow	Mean V Velocity
2	980877897	1	14700	0	0	0	0
3	980877897	1	14701	0	0	-0.813602	-0.0020099
4	980877897	1	14702	0.04518862	0.00012979	-0.6975267	-0.00182016
5	980877897	1	15167	0	0	0	0
6	980877897	1	15168	0	0	0	0
7	980877897	1	15169	0	0	0	0
8	980877897	1	15170	0	0	0	0
9	980877897	1	15171	0	0	0	0
10	980877897	1	15172	0	0	0	0
11	980877897	1	15173	0	0	0	0
12	980877897	1	15174	0	0	0	0
13	980877897	1	15175	0	0	0	0
14	980877897	1	15176	0	0	-0.4541567	-0.00122211
15	980877897	1	15629	0	0	0	0
16	980877897	1	15630	0	0	0	0

The header row contains the fields significance. The first three columns are fixed for all registers: time (in seconds from January 1, 1970 00:00), register step and cell number.

After the third column appear the mean variables in the order selected in Variables Dialog to file output.

Only the cells selected in the Points File are registered.

Log Files

The log files save the communications between classes during the simulation in some specified steps (specified in Output Logs). Each step is saved in one separated file named "LogfileN", where N is the step number. The extension name reflects the format used.

The "xls" format is obtained with Tab Separated Values and is similar to the "txt" format. The header row describes each column field:

STEP - time step

Class_Name - origin class of the communication

Type - method invoked: "Update" or "Inquiry"

Data_Class - destination class of the communication

Variable - variable of destination class

Value - value of the variable

Box_Number - number of cell

The **Type** "Update" means that class **Class_Name** will update the variable **Variable** in the class **Data_Class** with the value **Value** in the cell **Box_Number**.

The **Type** "Inquiry" means that class **Class_Name** asks for the value of the variable **Variable** in the class **Data_Class** in the cell **Box_Number**. The value is returned in **Value**.

	A	B	C	D	E	F	G
1	STEP	Class_Name	Type	Data_Class	Variable	Value	Box_Number
11123	89	TSangoPhytoplankton	Update	TSangoNutrients	Ammonia	0	108
11124	89	TSangoPhytoplankton	Update	TSangoResuspendDeposit	POM	5.90E-08	108
11125	89	TSangoPhytoplankton	Update	TSangoResuspendDeposit	TPM	5.90E-08	108
11126	89	TSangoPhytoplankton	Inquiry	TBiDimensionalSango3	Box depth	6.370691	109
11127	89	TSangoPhytoplankton	Inquiry	TBiDimensionalSango3	Box depth	6.370691	109
11128	89	TSangoPhytoplankton	Inquiry	TSangoResuspendDeposit	Extinction coefficient Ktot	1.36419	109
11129	89	TSangoPhytoplankton	Inquiry	TLight	PAR surface irradiance	0	109
11130	89	TLight	Inquiry	TSangoResuspendDeposit	Extinction coefficient Ktot	1.36419	109
11131	89	TSangoPhytoplankton	Inquiry	TLight	Sub-surface PAR irradiance	0	109
11132	89	TSangoPhytoplankton	Update	TSangoNutrients	Ammonia	0.000107	109
11133	89	TSangoPhytoplankton	Update	TSangoNutrients	Ammonia	0	109
11134	89	TSangoPhytoplankton	Update	TSangoResuspendDeposit	POM	5.90E-08	109
11135	89	TSangoPhytoplankton	Update	TSangoResuspendDeposit	TPM	5.90E-08	109
11136	89	TSangoPhytoplankton	Inquiry	TBiDimensionalSango3	Box depth	6.404196	110

The "xml" format is more elaborated and is used for more sophisticated post-processing methods:

```
<?xml version="1.0" encoding="UTF-8"?>|
<LOG>
<STEP Number="89">
<REGISTRY>
  <Class_Name>TWaterTemperatureTwoDimensionalForSango</Class_Name>
  <Type>Inquiry</Type>
  <Data_Class>TBiDimensionalSango3</Data_Class>
  <Variable>Salinity</Variable>
  <Value>-99</Value>
  <Box_Number>0</Box_Number>
</REGISTRY>
<REGISTRY>
  <Class_Name>TWaterTemperatureTwoDimensionalForSango</Class_Name>
  <Type>Inquiry</Type>
  <Data_Class>TBiDimensionalSango3</Data_Class>
  <Variable>Box depth</Variable>
  <Value>-10</Value>
  <Box_Number>0</Box_Number>
</REGISTRY>
<REGISTRY>
  <Class_Name>TWaterTemperatureTwoDimensionalForSango</Class_Name>
  <Type>Inquiry</Type>
  <Data_Class>TBiDimensionalSango3</Data_Class>
```

The root of the XML file is the **LOG** element.

This element is composed by one **STEP** element, with the time step number, and multiple **REGISTRY** elements, each one with 6 elements containing the information described previously (one registry for each line).

Properties Files

There are two kinds of properties files:

1. Model Properties File saves the models already created by the EcoDynamo application.
2. EcoDynamo Properties File saves, in the model directory, the simulation properties defined in the last usage by EcoDynamo application.

Model Properties File

The file "Models.properties" belongs to the EcoDynamo application's directory.

It saves the models used by EcoDynamo application and is created when the application ends.

Here is an example of model properties file:

```
DefinedModels=2
Model_0=RiaF2D100mNew
Path_0=C:\DITTY\EcoDynFiles\RiaF2D100mNew
Default_0=true
AutoInit_0=true
Model_1=Sango
Path_1=C:\DITTY\EcoDynFiles\Sango
Default_1=false
AutoInit_1=false
```

The first line contains the number of models already created and opened by EcoDynamo (property name is **DefinedModels**).

For each model created 4 properties are included:

Model - name of the model

Path - location of the model directory

Default – “true” if the model is the default model, “false” otherwise

AutoInit – “true” if the model is initialised when opened, “false” otherwise.

Each model property has the suffix **_N** where **N** is the order number of the model in the file.

EcoDynamo Properties File

Each model created by EcoDynamo application will have a file named "EcoDynamo.properties" in its directory.

The file is created when the model is closed, and saves the properties of the simulation.

Here is an example of one EcoDynamo properties file:

```
PrefixName=RIAF2D
Type=
PathName=C:\DITTY\EcoDynFiles\RiaF2D100mNew
NrClassesAvailable=8
Available_0=TTide
Available_1=TBiDimensionalRiaFormosa
Available_2=TRiaF2DNutrients
Available_3=TRiaF2DPhytoplankton
Available_4=TLight
Available_5=TWaterTemperatureTwoDimensionalForSango
Available_6=TRiaFormosaAirTemperature
Available_7=TBiDimensionalTimeSeries
NrClassesSelected=2
Selected_0=TTide
Selected_1=TBiDimensionalRiaFormosa
NrVariablesAvailable=16
AvailableVars_0=Tidal height
AvailableVars_1=Columns
AvailableVars_2=Lines
AvailableVars_3=U Flow
AvailableVars_4=V Flow
AvailableVars_5=Dynamic height
AvailableVars_6=Salinity
AvailableVars_7=U Velocity
AvailableVars_8=V Velocity
AvailableVars_9=Water density
AvailableVars_10=Drag coefficient
AvailableVars_11=Box depth
AvailableVars_12=Mean U Flow
AvailableVars_13=Mean V Flow
AvailableVars_14=Mean U Velocity
AvailableVars_15=Mean V Velocity
NrVariablesSelectedFileOutput=7
SelectedVarsFileOutput_0=Box depth
SelectedVarsFileOutput_1=Salinity
SelectedVarsFileOutput_2=U Flow
SelectedVarsFileOutput_3=U Velocity
SelectedVarsFileOutput_4=Water density
SelectedVarsFileOutput_5=V Velocity
SelectedVarsFileOutput_6=V Flow
NrVariablesSelectedGraphOutput=0
NrVariablesSelectedTableOutput=0
StartTime=980877600
FinishTime=983383200
TimeStep=3
TimeUnit=Seconds
```

```
RunMode=Normal
Integration=Euler
OutputStartTime=980877600
OutputFinishTime=983383200
OutputFrequency=300
OutputTimeUnit=Minutes
```

The properties saved in the file are:

PrefixName - prefix that heads each mandatory file
Type - not used in this version of EcoDynamo
PathName - location of the model directory
NrClassesAvailable - number of classes available in classes file
Available_N - list of available classes [**N** is the class order number]
NrClassesSelected - number of classes selected [Classes dialog]
Selected_N - list of selected classes [**N** is the class order number]
NrVariablesAvailable - number of variables available from classes selected
AvailableVars_N - list of available variables [**N** is the variable order number]
NrVariablesSelectedFileOutput - number of variables selected for file output [Variables dialog]
SelectedVarsFileOutput_N - list of selected variables to file output [**N** is the variable order number]
NrVariablesSelectedGraphOutput - number of variables selected for graph output [Variables dialog]
SelectedVarsGraphOutput_N - list of selected variables to graph output [**N** is the variable order number]
NrVariablesSelectedTableOutput - number of variables selected for table output [Variables dialog]
SelectedVarsTableOutput_N - list of selected variables to table output [**N** is the variable order number]
StartTime - simulation start time (seconds from January 1, 1970 00:00) [Time specs dialog (Simulation tab)]
FinishTime - simulation finish time (seconds from January 1, 1970 00:00) [Time specs dialog (Simulation tab)]
TimeStep - simulation time step (in seconds) [Time specs dialog (Simulation tab)]
TimeUnit - simulation time step unit [Time specs dialog (Simulation tab)]
RunMode - simulation run mode [Time specs dialog (Simulation tab)]
Integration - simulation time integration [Time specs dialog (Integration tab)]
OutputStartTime - output register start time (seconds from January 1, 1970 00:00) [Time specs dialog (Output Register tab)]
OutputFinishTime - output register finish time (seconds from January 1, 1970 00:00) [Time specs dialog (Output Register tab)]
OutputFrequency - output frequency time (in seconds) [Time specs dialog (Output Register tab)]
OutputTimeUnit - output frequency time unit [Time specs dialog (Output Register tab)]