Volume

1

INSTITUTO SUPERIOR TÉCNICO - MARETEC

MOHID a Water System model

Hydrodynamic Module (Mohid) User Guide

INSTITUTO SUPERIOR TÉCNICO - MARETEC

MOHID Hydrodynamic Module User Guide

INSTITUTO SUPERIOR TÉCNICO – MARETEC
Avenida Rovisco Pais, nº1 1096 Lisboa Codex
Phone 214239016 • Fax 214211272

Table of Contents

Table Index

Figures Index

General Overview

Introduction

The "hydrodynamic" module of the Mohid system is able to simulate the flow in water masses where the flow velocity is lower than the celerity of the pressure wave. This module has been used to simulate hydrodynamic processes in oceans (Atlantic NE and Atlantic SW), in coastal areas (several areas along the Portuguese and Brazil coasts), in more than 30 estuaries and lagoons (European, African and Brazilian estuaries and lagoons) and in water dams (south of Portugal). All these study areas have complex flows and intense ecological activity with a strong relation with hydrodynamic processes. The "hydrodynamic" module aims to be a numerical tool oriented to help understanding biogeochemical processes and resolve ecological problems associated with human activity.

Approximations

Hydrostatic flow (in the near future a non-hydrostatic version will be available)

Boussinesq approximation

Main numerical characteristics

Spatial discretization	Finite volumes
Horizontal Grid	Orthogonal
Vertical Grid	Generic coordinates
Computation points distribution	Arakawa C
Time discretization	ADI

Forces discretization

Forces computed explicitly	Coriolis, tide potential, baroclinic pressure gradient, atmosphere forcing (wind stress and pressure), horizontal advection and diffusion of momentum
Forces computed implicitly	Barotropic pressure gradient, bottom friction, vertical advection and diffusion of momentum

Baroclinic pressure spatial discretization	Cartesian referential (or z level referential)
Horizontal advection of momentum	Hybrid (upwind + central differences) 2º order upwind
Vertical advection of momentum	Hybrid (upwind + central differences)
Diffusion of momentum	Central differences

Boundary conditions

Barotropic pressure gradient: Water level Barotropic velocity	▪ Imposed ▪ Null gradient ▪ Cyclic ▪ Radiation - Flather, 1976 ▪ Radiation – Blumberg & Kantha, 1985 ▪ Flow relaxation The last three boundary conditions use a reference solution that can be imposed using two methodologies: ▪ Input data – the solution is imposed as model input data ▪ One way nesting - the solution is compute by a courser grid model
Baroclinic pressure gradient: Baroclinic velocity Temperature and salinity	▪ Imposed ▪ Null gradient ▪ Radiation – Marchesiello et al., 2001 1. Celerity constant 2. Celerity – Orlansky, 1976 ▪ Flow relaxation

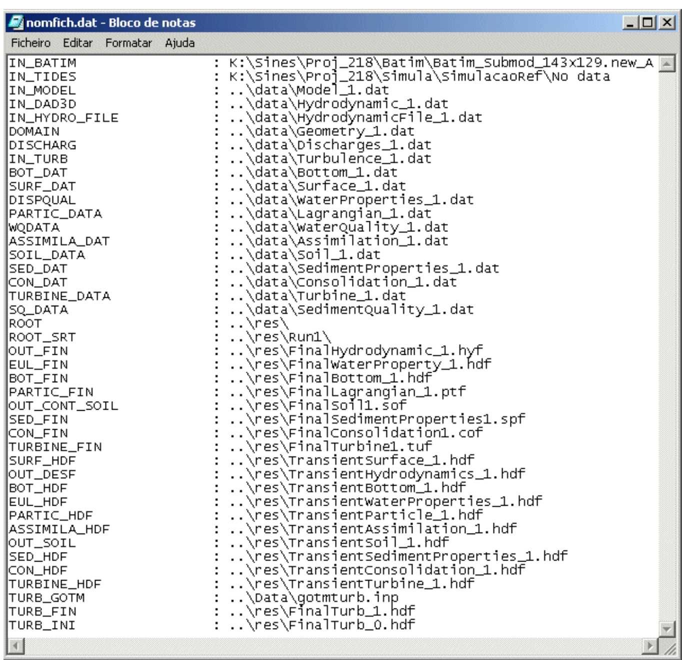
	The last two boundary conditions use a reference solution that can be imposed using two methodologies: ▪ Input data ▪ One way nesting

Manual organization

The manual is subdivided in 4 chapters: discretization, boundaries, turbulence parameterization and output control. A chapter is divided in several sub items where is presented a table with the keywords and the input data file where the user can control the options available. After the keywords table is presented an example of a data file.

Files organization

Along the entire manual is done several references to input and output files. The name and paths of these files are defined in a file named nomfich.dat (Figure 1). This file is placed always in the directory where the model is run. In this file is defined a list of KEYWORDS and each keyword correspond a input or output file. Along this manual these keywords are used to make reference to a specific input or output file. For example the input file where the vertical geometry is defined corresponds to the keyword DOMAIN.


nomfich.dat - Bloco de notas

Ficheiro Editar Formatar Ajuda

```
IN_BATIM                : K:\Sines\Proj_218\Batim\Batim_Submod_143x129.new_A
IN_TIDES                : K:\Sines\Proj_218\Simula\SimulacaoRef\No data
IN_MODEL                : ..\data\Model_1.dat
IN_DAD3D                : ..\data\Hydrodynamic_1.dat
IN_HYDRO_FILE           : ..\data\HydrodynamicFile_1.dat
DOMAIN                  : ..\data\Geometry_1.dat
DISCHARG                : ..\data\Discharges_1.dat
IN_TURB                 : ..\data\Turbulence_1.dat
BOT_DAT                 : ..\data\Bottom_1.dat
SURF_DAT                : ..\data\Surface_1.dat
DISPQUAL                : ..\data\WaterProperties_1.dat
PARTIC_DATA             : ..\data\Lagrangian_1.dat
WQDATA                  : ..\data\WaterQuality_1.dat
ASSIMILA_DAT            : ..\data\Assimilation_1.dat
SOIL_DATA               : ..\data\Soil_1.dat
SED_DAT                 : ..\data\SedimentProperties_1.dat
CON_DAT                 : ..\data\Consolidation_1.dat
TURBINE_DATA            : ..\data\Turbine_1.dat
SQ_DATA                 : ..\data\SedimentQuality_1.dat
ROOT                    : ..\res\
ROOT_SRT                : ..\res\Run1\
OUT_FIN                 : ..\res\FinalHydrodynamic_1.hyf
EUL_FIN                 : ..\res\FinalWaterProperty_1.hdf
BOT_FIN                 : ..\res\FinalBottom_1.hdf
PARTIC_FIN              : ..\res\FinalLagrangian_1.ptf
OUT_CONT_SOIL           : ..\res\FinalSoil1.sof
SED_FIN                 : ..\res\FinalSedimentProperties1.spf
CON_FIN                 : ..\res\FinalConsolidation1.cof
TURBINE_FIN             : ..\res\FinalTurbine1.tuf
SURF_HDF                : ..\res\TransientSurface_1.hdf
OUT_DESF                : ..\res\TransientHydrodynamics_1.hdf
BOT_HDF                 : ..\res\TransientBottom_1.hdf
EUL_HDF                 : ..\res\TransientWaterProperties_1.hdf
PARTIC_HDF              : ..\res\TransientParticle_1.hdf
ASSIMILA_HDF            : ..\res\TransientAssimilation_1.hdf
OUT_SOIL                : ..\res\TransientSoil_1.hdf
SED_HDF                 : ..\res\TransientSedimentProperties_1.hdf
CON_HDF                 : ..\res\TransientConsolidation_1.hdf
TURBINE_HDF             : ..\res\TransientTurbine_1.hdf
TURB_GOTM               : ..\Data\gotmturb.inp
TURB_FIN                : ..\res\FinalTurb_1.hdf
TURB_INI                : ..\res\FinalTurb_0.hdf
```

Figure 1 - File where the name and paths of the input and output files is defined. This file name is always nomfich.dat and is placed always in the directory where the model is run.

Table 1 – An example of the file where the name and paths of the input and output files is defined. This file name is always nomfich.dat and is placed always in the directory where the model is run.

```
IN_BATIM          : K:\Sines\Proj_218\Batim\Batim_Submod_143x129.new_A_B_C_D
IN_TIDES          : C:\Work\Aplica\Sines\Proj_218\No data
IN_MODEL          : ..\data\Model_5.dat
IN_DAD3D          : ..\data\Hydrodynamic_5.dat
IN_HYDRO_FILE     : ..\data\HydrodynamicFile_5.dat
DOMAIN            : ..\data\Geometry_5.dat
DISCHARG          : ..\data\Discharges_5.dat
IN_TURB           : ..\data\Turbulence_5.dat
BOT_DAT           : ..\data\Bottom_5.dat
SURF_DAT          : ..\data\Surface_5.dat
DISPQUAL          : ..\data\WaterProperties_5.dat
PARTIC_DATA       : ..\data\Lagrangian_5.dat
WQDATA            : ..\data\WaterQuality_5.dat
ASSIMILA_DAT      : ..\data\Assimilation_5.dat
SOIL_DATA         : ..\data\Soil_5.dat
SED_DAT           : ..\data\SedimentProperties_5.dat
CON_DAT           : ..\data\Consolidation_5.dat
TURBINE_DATA      : ..\data\Turbine_5.dat
SQ_DATA           : ..\data\SedimentQuality_5.dat
ROOT              : ..\res\
ROOT_SRT          : ..\res\Run5\
OUT_FIN           : ..\res\FinalHydrodynamic_5.hyf
EUL_FIN           : ..\res\FinalWaterProperty_5.hdf
BOT_FIN           : ..\res\FinalBottom_5.hdf
PARTIC_FIN        : ..\res\FinalLagrangian_5.ptf
OUT_CONT_SOIL     : ..\res\FinalSoil5.sof
SED_FIN           : ..\res\FinalSedimentProperties5.spf
CON_FIN           : ..\res\FinalConsolidation5.cof
TURBINE_FIN       : ..\res\FinalTurbine5.tuf
SURF_HDF          : ..\res\TransientSurface_5.hdf
OUT_DESF          : ..\res\TransientHydrodynamics_5.hdf
BOT_HDF           : ..\res\TransientBottom_5.hdf
EUL_HDF           : ..\res\TransientWaterProperties_5.hdf
PARTIC_HDF        : ..\res\TransientParticle_5.hdf
ASSIMILA_HDF      : ..\res\TransientAssimilation_5.hdf
OUT_SOIL          : ..\res\TransientSoil_5.hdf
SED_HDF           : ..\res\TransientSedimentProperties_5.hdf
CON_HDF           : ..\res\TransientConsolidation_5.hdf
TURBINE_HDF       : ..\res\TransientTurbine_5.hdf
TURB_GOTM         : ..\Data\gotmturb.inp
TURB_FIN          : ..\res\FinalTurb_5.hdf
TURB_INI          : ..\res\FinalTurb_0.hdf
```

Default data file

The Mohid graphical interface allows the definition of a hydrodynamic input data file (Table 2). However, the interface does not give access to all the modelo options for that is necessary to edit the file and put by hand the extra options. After doing this the user can not edit the hydrodynamic input data file with the graphical mode only using text editors like notepad. This manual describes all the model options that can be used to simulate the flow properties evolution.

Table 2 – An example a default hydrodynamic input data file generate bu the Mohid graphical interface.

```
Hydrodynamic data file. DO NOT EDIT
BAROCLINIC              : 1
HORIZONTALDIFFUSION     : 1
HORIZONTALADVECTION     : 1
VERTICALDIFFUSION       : 1
VERTICALADVECTION       : 1
CORIOLIS                : 1
RESIDUAL                : 0
ENERGY                  : 0
```

```
CONTINUOUS            : 0
UPSTREAM              : Upwind
DISCRETIZATION        : 2
EVOLUTION             : Solve_Equations
TIDE                  : 1
DATA_ASSIMILATION     : 0
BRFORCE               : 0
SUBMODEL              : 1
ATM_PRESSURE          : 0
WIND                  : -1
SURFACEWATERFLUX      : 0
WATER_DISCHARGES      : 1
RECORDING             : 0
INITIAL_ELEVATION     : 0
 OUTPUT_TIME          : 0 900
TIME_SERIE            : 1
<BeginTimeSerie>
LOCALIZATION_I        : 16
LOCALIZATION_J        : 43
LOCALIZATION_K        : 7
<EndTimeSerie>
<BeginTimeSerie>
LOCALIZATION_I        : 1
LOCALIZATION_J        : 39
LOCALIZATION_K        : 7
<EndTimeSerie>
DT_OUTPUT_TIME        : 300
```

Discretization

Discretization of the primitive equations in time and space

The model uses a finite volume approach for space discretization allowing a great flexibility in the grid definition ().

Horizontal Discretization

Is possible in the Mohid system define the bathymetry using several coordinate systems: geographic (Figure 2), circular (Figure 3), Portuguese military (Figure 4).

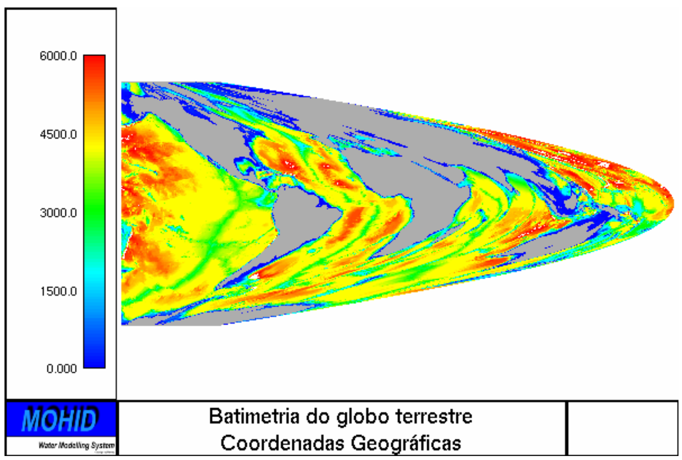


Figure 2 – Global bathymetry defined using geographic coordinates.

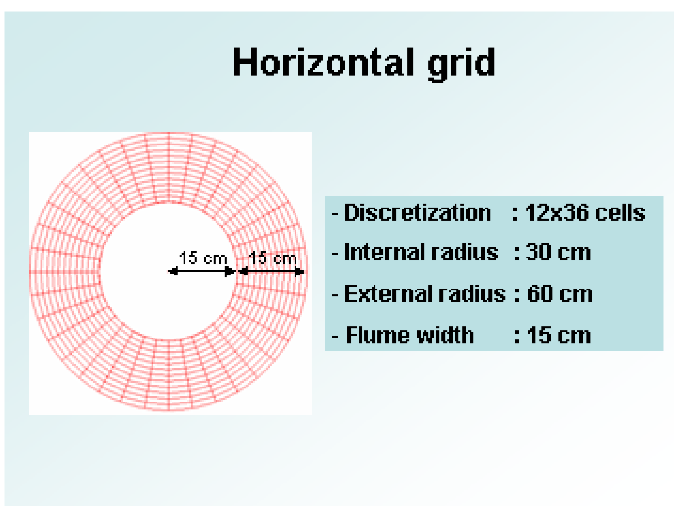


Figure 3 – Bathymetry describing a small annular flume. Cylindrical coordinates.

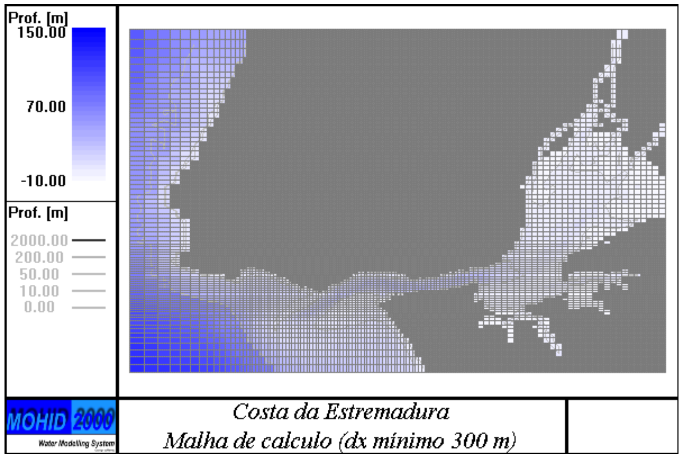


Figure 4 – Tagus estuary bathymetry defined using military coordinates.

Table 3 – Keywords used to define the model bathymetry.

Input data file		IN_BATIM (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
ILB_IUB	*	integer	ILB_IUB : 1 2	Lower line
JLB_JUB	*	Integer	JLB_JUB : 1 2	Upper line
COORD_TIP	3	integer	COORD_TIP : 3	1-geographic/ellipsoid 2-UTM 3-Portuguese military 4-geographic/spheroid 5-grid coordinate (this coordinate where the origin is the left corner grid and the x and y axes are along the lines and column direction) 6-circular
ZONE	29	integer	ZONE : 29	Definition of UTM zone for Portugal the zone is 29
LONGITUDE	*	real	LONGITUDE :-9	Average longitude in the domain. Redundant in the case of geographic coordinates
LATITUDE	*	real	LATITUDE : 38	Average latitude in the domain. Redundant in the case of geographic coordinates
ORIGIN	0.0 0.0	real,real	1200. 4200.	X and Y location of the left lower corner of the grid.
ROTATION	0	real	ROTATION : 0	Rotation of the grid in relation to the axes of the coordinate system adopted. Valid only for the case of metric coordinates (UTM, Portuguese military)
<BeginBathymetry> <EndBathymetry>		Block	<BeginXX> 5 5	Cell depths in meters. The reading sequence is the following:

			5 5 <EndBathymetry>	do i=ILB,IUB do j=JLB,JUB Bathymetry(i,j) enddo enddo
<BeginXX> <EndXX>	-	Block	<BeginXX> 0 10 20 <EndXX>	Distance along the X direction between the grid lower left corner and the cell faces aligned with the Y direction
<BeginYY> <EndYY>	-	Block	<BeginYY> 0 10 20 <EndYY>	Distance along the Y direction between the grid lower left corner and the cell faces aligned with the X direction

* - The user must give a value to this keyword or else the model do not run.

Table 4 – An example of a bathymetry file (IN_BATIM see Table 1).

```
ILB_IUB : 1 2
JLB_JUB : 1 2
COORD_TIP : 3
ZONE : 29
LONGITUDE :-9
LATITUDE : 38
1200. 4200.
ROTATION : 0
<BeginXX>
5
5
5
5
<EndBathymetry>
<BeginXX>
0
10
20
<EndXX>
<BeginYY>
0
10
20
<EndYY>
```

Vertical Discretization

Actually the module Geometry can divide the water column in different vertical coordinates: Sigma (Figure 5), Cartesian (Figure 6), Lagrangian (based on Sigma or based on Cartesian), "Fixed Spacing" and Harmonic. A subdivision of the water column into different domains is also possible (Figure 7). The Sigma and the Cartesian coordinates are the classical ones. The Cartesian coordinate can be used with or without "shaved cells". The Lagrangian coordinate moves the upper and lower faces with the vertical flow velocity. The "Fixed Spacing" coordinate allows the user to study flows close to the bottom and the Harmonic coordinate works like the Cartesian coordinate, just that the horizontal faces close to the surface expand and collapse depending on the variation of the surface elevation. This coordinate was implemented in the geometry module to simulate reservoirs.

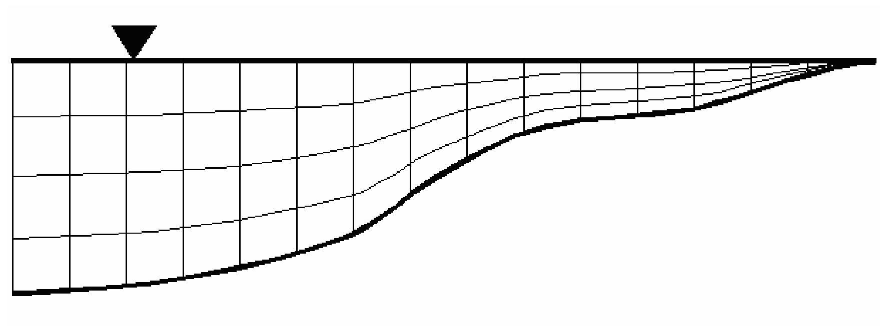


Figure 5 - Sigma domain with 4 Layers.

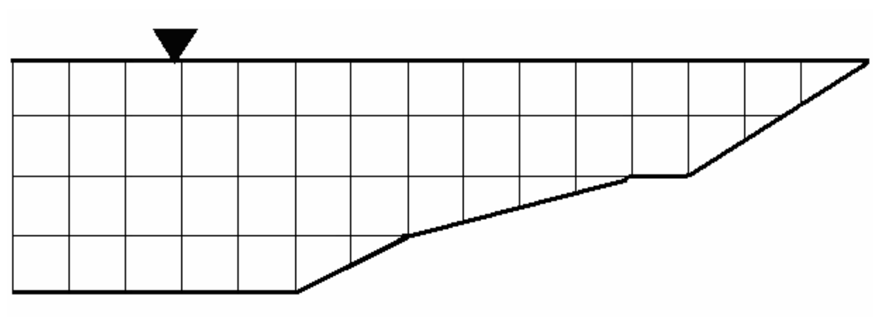


Figure 6 - Cartesian domain with 4 Layers (shaved cells).

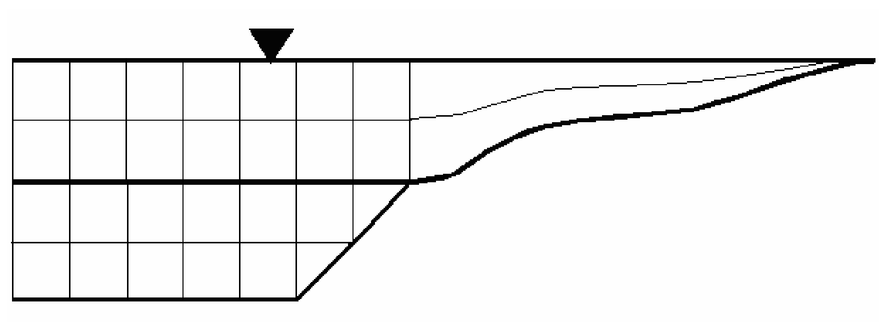


Figure 7 – Sub-division of the water column in a Cartesian domain (inferior) and a Sigma domain (superior).

Table 5 – Keywords used to define the vertical geometry.

Input data file		DOMAIN (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
MINIMUMDEPTH	0.1	real	MINIMUMDEPTH : 0.1	Water column thickness below which the cell is consider uncovered
FACESOPTION	1	integer	FACESOPTION : 1	Methodology to compute areas between cells: 1 – minimum thickness of the adjacent water columns; 2 -average thickness of the adjacent water columns;
<begindomain> <enddomain>	*	block	<begindomain> ID: 1 TYPE : SIGMA LAYERS : 5 LAYERTHICKNESS : .6 .2 .1 .05 .05 DOMAINDEPTH : 3 TOLERANCEDEPTH : 0.05 <enddomain>	In a block is defined a domain limited below by is DOMAINDEPTH and above by the DOMAINDEPTH of the domain locate above. If does not exist a domain above the upper limit is the surface level. If the DOMAINDEPTH is greater than bottom depth then bottom is the domain lower limit.
ID	*	integer	ID : 1	The ID of the domain. The ID=1 is the lowest domain.
TYPE	*	string	TYPE : CARTESIAN	The vertical coordinate of the domain: 1. CARTESIAN 2. SIGMA 3. LAGRANGIAN 4. HARMONIC 5. FIXSPACING 6. FIXSEDIMENT
LAYERS	*	integer	LAYERS : 4	Number of layers
EQUIDISTANT	0.0	real	EQUIDISTANT : 0.5	Thickness of layers admitting that all the layers have the same thickness
LAYERTHICKNESS	*	vector	LAYERTHICKNESS : 2 3 3 2	Thickness of layers. The number of values must be equal to the number of layers. The order is from bottom to surface
MININITIALLAYERTHICKNESS		real	MININITIALLAYERTHICKNESS : 5	
TOLERANCEDEPTH	0.05	real	TOLERANCEDEPTH : 0.05	Thickness of layer in meters below which the bathymetry is corrected. Valid only for the sigma and lagrangian (sigma initialization) coordinate.

TOTALTHICKNESS	**	real	TOTALTHICKNESS : 1	Total thickness in meters of the domain Valid only for the FixSpacing, FixSediment coordinates.
DOMAINDEPTH	*	real	DOMAINDEPTH : 100	The depth of the domain lower limit
MINEVOLVELAYERTHICKNESS	0	real [%]		Distortion in % of the initial thickness 0 – maximum distortion 0.5 – minimum distortion
GRIDMOVEMENTDUMP	0	real		
DISPLACEMENT_LIMIT	1000	real		DisplacementLimit is the maximum displacement that the model allow cell faces to move vertically in meters
INITIALIZATION_METHOD	SIGMA	string	INITIALIZATION_METHOD : SIGMA	Type of initialization used in the case of a lagrangian coordinate. This is also the reference coordinate in relation to which the lagrangian coordinate suffers distortion function of the vertical velocity

* - The user must give a value to this keyword or else the model do not run.

** - Valid only for the FixSpacing, FixSediment coordinates.

Table 6 – An example of a input data file used to define the vertical geometry of the model.

```
MINIMUMDEPTH           : 0.1
FACES_OPTION           : 2
<begindomain>
ID                   : 1
TYPE                 : SIGMA
LAYERS               : 1
LAYERTHICKNESS         : 1.
DOMAINDEPTH            : 4
TOLERANCEDEPTH         : 0.0500
<enddomain>

<begindomain>
ID                   : 2
TYPE                 : CARTESIAN
LAYERS               : 4
!LAYERTHICKNESS         : .6 .55. .5 .45 .4 .35 .3 .25
EQUIDISTANT           : 1
A Domain Depth of -99 equals the suface
DOMAINDEPTH            : 0
MININITIALLAYERTHICKNESS: 0.05

<enddomain>

<begindomain>
ID                   : 3
TYPE                 : SIGMA
LAYERS               : 5
EQUIDISTANT           : 0.2
!LAYERTHICKNESS         : -9e15
DOMAINDEPTH            : -99.00
TOLERANCEDEPTH         : 0.0500
<enddomain>
```

Time Discretization

In time uses a semi-implicit descritization to resolve the 2D mass conservation equation used to estimate the hydrostatic pressure. In the calculation of the horizontal velocity the bottom stress and the vertical transport of momentum are computed implicitly. The user in respect to the time discretization can control the run period and the time step. Related with the time evolution of the hydrodynamic properties the user can decide if he wants to solve the primitive equations or to read the hydrodynamic solution of a file or admitted the hydrodynamic properties stationary equal to the residual values of a past run or equal to the default initial conditions of the model (velocity null and no water level gradients).

Table 7 – Keyword in the hydrodynamic input data file that controls the time discretization.

Input data file		IN_MODEL (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
START	*	6*Integer	START : 2000 1 1 0 0 0	Start date of the run
BEGIN	*	6*Integer	END : 2000 1 1 0 0 0	End date of the run
DT	*	Real (seconds)	DT : 1.5	Time step
SPLITTING	Double_Splitting	String	SPLITTING : No_Splitting	The Double_Splitting means that the model will solve the primitive equations. If the users

				wants a stationary solution or read the hydrodynamic properties of a file then the option is No_Splitting
VARIABLEDT	0	Integer	VARIABLEDT : 0	Check if the user wants variable time step. The users of the hydrodynamic module should disregard this option.

Table 8 – An example on how the time discretization can be control in the IN_MODEL input data file (see Table 1).

```
START              : 1999. 12. 24.  6.  0.  0.
 END               : 1999. 12. 24. 15.  0.  0.
DT                 : 5.00
VARIABLEDT               : 0
SPLITTING            : Double_Splitting
```

Table 9 – Keywords used to control the time discretization in the hydrodynamic input data file (IN_DAD3D see Table 1).

Input data file		IN_DAD3D (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
CONTINUOUS	0	Integer	CONTINUOUS : 0	Check if the user wants to continuum a past run (1) or not (0)
DISCRETIZATION	2	Integer	DISCRETIZATION : 1	Checks if the user want to solve 3 equation for each half step (2) Leendertse method (Leendertse, 1967) or 2 eq. (1) Abbott method (Abbott et al., 1973).
EVOLUTION	Solve_Equations	String	EVOLUTION : Read_File	The user have 5 options for the hydrodynamic properties evolution: • Solve_Equations (solve the primitive equations) • Read_File (read the hydrodynamic properties from a file) • No_hydrodynamic (stationary solution with null velocity) • Residual_hydrodynamic (stationary solution equal to the residual field of a past run) • Run_Off (water level tend always to a null gradient and velocities are considered null)

Table 10 – An example of time discretization control in the hydrodynamic input data file (IN_DAD3D see Figure 1).

```
CONTINUOUS          : 0
DISCRETIZATION        : 2
```

EVOLUTION : Solve_Equations

Hydrodynamic Solution Input

Table 11 – Keywords use to control the hydrodynamic solution when is read from a file.

Input data file		IN_HYDRO_FILE (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
INPUT	0	Integer	INPUT : 1	Check if the user wants to read an hydrodynamic solution from a file
IN_FILE_VERSION	2	Integer	IN_FILE_VERSION : 1	The user can choose from two options. One more old (1) and another more recent and more optimized (2). This last option should be used always. The old option was maintained only to allow the used of files written by model old versions.
IN_FILE_TYPE	M2_Tide_type	String	IN_FILE_TYPE : BeginEnd_type	The user can choose from two options: • BeginEnd_type : the hydrodynamic properties evolution is along a specific period; • M2_Tide_type : the hydrodynamic evolution is cyclic repeats is self after a M2 (12.425 hours) period. Associated to this file exist a initial date and this allow to define a hydrodynamic evolution to any period posterior the initial date.

Table 12 – A pratical application of the keywords defined in Table 11.

INPUT : 1 IN_FIELD : InHydroFile.bin IN_FILE_VERSION : 2 IN_FILE_TYPE : M2_Tide_type

Forces Discretization

Basically this model aims to compute the velocities and cells volume evolution. The horizontal velocity results from computing the local acceleration that equal to a sum of the follow forces:

- Inertia (volume variation, advection, diffusion and coriolis);
- Pressure (atmospheric, barotropic and baroclinic);

- Astronomic forces (tide potential);
- Bottom and surface stress (vertical boundaries);
- Imposed sinks and sources of momentum (ex: river discharges).

The barotropic pressure, the bottom stress and the inertia forces associated with vertical transport have stability limits very restrictive so they are computed implicitly all the other forces are computed explicitly. The user can disconnect all forces except the local acceleration (the unknown variable), the barotropic pressure and the bottom stress. The last one can be disconnected but for that the user need to go to the input data file of the modulo bottom (BOT_DATA - see Figure 1) and consider a null rugosity coefficient (RUGOSITY : 0). For more details see the chapter about the bottom boundary.

Table 13 – Options available to control the forces discretization.

Input data file		IN_DAD3D (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
BAROCLINIC	0	integer	BAROCLINIC : 1	Check if the user wants to consider the baroclinic pressure (1) or not (0)
RAMP	0	integer	RAMP : 1	Check if the user wants to start with baroclinic force null and only after a specific period the total force is compute (1) or not (0)
INERTIAL_PERIODS	1	real	INERTIAL_PERIODS : 2.5	The period after which the total effect of the baroclinic force is compute. The time units are inertial periods 2*pi/ f
RAMP_START	*	6*real	RAMP_START : 2002 1 1 0 0 0	In the case of the run be continuation of a past run and the RAMP option is active then the user must define the initial date from which the RAMP option started.
HORIZONTALDIFFUSION	1	integer	HORIZONTALDIFFUSION : 1	Check if the user wants to consider the horizontal advection of momentum (1) or not (0)
HORIZONTALADVECTION	1	integer	HORIZONTALADVECTION : 1	Check if the user wants to consider the horizontal diffusion of momentum (1) or not (0)
VERTICALDIFFUSION	1	Integer	VERTICALDIFFUSION : 1	Check if the user wants to consider the vertical diffusion of momentum (1) or not (0)
VERTICALADVECTION	1	Integer	VERTICALADVECTION : 1	Check if the user wants to consider the vertical advection of momentum (1) or not (0)
CORIOLIS	1	Integer	CORIOLIS : 1	Check if the user wants to consider the coriolis force (1) or not (0)
VOLUMEVARIATION	1	Integer	VOLUMEVARIATION : 1	Check if the user wants to consider the volume

				variation (1) or not (0)
ATM_PRESSURE	0	Integer	ATM_PRESSURE : 1	Check if the user wants to consider the atmospheric pressure (1) or not (0)
TIDEPOTENTIAL	0	Integer	TIDEPOTENTIAL : 1	Check if the user wants to consider the astronomic forces (1) or not (0)
WIND	0	Integer	WIND : 2	Check if the user wants to consider the wind stress (1) or not (0) or wind stress with a smooth initial period (2)
WIND_SMOOTH_PERIOD	86400.	Real (seconds)	WIND_SMOOTH_PERIOD : 172800.	The user can impose a specific period in seconds after which the model considers the total effect of wind stress. Along this period the wind stress amplitude is multiplied by a coefficient that has a linear evolution between 0 and 1. By default this period is zero seconds
UPSTREAM	Upwind	String	UPSTREAM : Quick	Check if the user wants to consider a first order (Upwind) or a second order (Quick) upwind scheme to solve the horizontal advection of momentum
UP_CENTER	1	Real	UP_CENTER : 0.5	The advection algorithm is a hybrid one and can be total upwind (1) or total Center differences (0).
HMIN_ADVECTION	0.5	Real (m)	HMIN_ADVECTION : 1.1	The user can impose a specific water column height below which the horizontal advection is not compute. By default when the water column has less then 0.5 m the advection in not compute.
CONSERVATIVE_HOR_DIF	0	integer	CONSERVATIVE_HOR_DIF : 1	Check if the user wants to compute the horizontal diffusion in a conservative way (1) or not (0).
WATERCOLUMN2D	-9.9e15	real	WATERCOLUMN2D : 1.1	Water column thickness below which the vertical transport of momentum is disconnected
BOTTOMVISC_LIM	0	integer	BOTTOMVISC_LIM : 1	Check if the user wants to limit the viscosity at the bottom. This can be important due to the fact of being use explicit approach between the bottom layer and above layer for the vertical mixing when the water flux is compute implicit in conservation equation where an estimate of the barotropic pressure is made.
BOTTOMVISC_COEF	5.	real	BOTTOMVISC_COEF : 10.	Factor that multiplies diffusion number for imposing a maximum viscosity at bottom layer (coefficient of turbulence transport between layers kbottom and kbottom +1, i.e. viscosity(kbottom+1))

				Maximum viscosity = BottomVisc_MAX*dz*dz/2/dt/Viscosity(kbottom+1)
HMIN_CHEZY	0.1	Real (m)	HMIN_CHEZY : 0.2	Checks the minimum water column height below which the chezy coefficient is constant. By default Hmin_Chezy is equal to 10 cm
VMIN_CHEZY	0.1	Real (m/s)	VMIN_CHEZY : 0.3	Checks the minimum velocity (Vmin_Chezy) below which the chezy coefficient is constant if the water column is smaller than Hmin_Chezy. By default Vmin_Chezy is equal to 0.10 m/s.
MOMENTUM_DISCHARGE	0	integer	MOMENTUM_DISCHARGE : 1	Checks if the user wants to do a discharge of momentum. By default the model do not have momentum discharges
WATER_DISCHARGES	0	integer	WATER_DISCHARGES : 1	Check if the user wants to water discharges.
CORRECT_WATERLEVEL	0	integer	CORRECT_WATERLEVEL : 1	By default the model corrects the water level when the water column tend to be lower then zero but in this case the model can also corrected (1) or not (0) the water level when it is lower than a reference water level
MIN_WATERLEVEL	0	real	MIN_WATERLEVEL : 1.2	Reference level below which the water level is corrected.

Table 14 – Pratical application of the keywords used to control the forces discretization.

```
BAROCLINIC            : 1
HORIZONTALDIFFUSION   : 1
HORIZONTALADVECTION   : 1
VERTICALDIFFUSION     : 1
VERTICALADVECTION     : 1
CORIOLIS              : 1
UPSTREAM              : Upwind
ATM_PRESSURE          : 0
WIND                  : 1
```

Initial condition

Table 15 – Keywords used to define the initial hydrodynamic properties condition.

Input data file		IN_DAD3D (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
CONTINUOUS	0	integer	CONTINUOUS : 1	Check if the user wants to continuum a past run (1) or not (0)
INITIAL_ELEVATION	0	integer	INITIAL_ELEVA	Checks if the user wants to

			TION : 1	impose a initial elevation (1) or not (0)
INITIAL_ELEVATION_VALUE	0	real	INITIAL_ELEVATION_VALUE : 1	The user define with this keyword the initial elevation value
INITIAL_VEL_U	0.	Real	INITIAL_VEL_U : 0.2	Checks if the user pretends to impose an initial velocity U (X direction) different from zero.
INITIAL_VEL_V	0.	real	INITIAL_VEL_V : 0.3	Checks if the user pretends to impose an initial velocity V (Y direction) different from zero.

Table 16 – An example on how initial hydrodynamic properties condition can be defined.

```
CONTINUOUS              : 0
INITIAL_ELEVATION       : 1
INITIAL_ELEVATION_VALUE  : 2.08
INITIAL_VEL_U : 0.3
INITIAL_VEL_V : .2
```

Boundaries

Horizontal Boundaries

Open

Table 17 –Keywords that the user can use to define a open boundary condition in the horizontal direction in the input data file of the module hydrodynamic (IN_DAD3D - see Figure 1).

Input data file		IN_DAD3D (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
TIDE	0	integer	TIDE : 1	Checks if the user wants to impose in the boundary points the water level define in the input data file of the module gauge (IN_TIDES - see Figure 1)
SLOWSTART	0	Real (seconds)	SLOWSTART : 86400.	The user can impose a specific period in seconds after which the model consider the total imposed boundary wave. Along this period the wave amplitude is multiplied by A coefficient that has a linear evolution between 0 and 1. By default this period is zero seconds
DATA_ASSIMILATION	0	Integer	DATA_ASSIMILATION : 1	Check if the user wants to assimilate hydrodynamic properties define in the input data file of Data Assimilation module (1) or not (0)
BRFORCE	0	Integer	BRFORCE : 1	
SUBMODEL	0	Integer	SUBMODEL : 1	Check if the user wants to run this model as a submodel (1) or not (0).
MISSING_NULL	0	integer	MISSING_NULL : 1	When the option "SUBMODEL" is active check if the user wants to replace the missing values (where the coarser grid do not have information) by zero.
DEADZONE	0		DEADZONE	Check if the user wants to define a dead zone where the submodel do not look for information in the father (coarser) model
DEADZONE_FILE	******.***		DEADZONE_FILE : deadboxes.txt	file name where the dead zone is defined was polygon
RADIATION	0	Integer	RADIATION : 1	1 – Flather solution applied to case where the reference solution are waves with a specific direction (ex: wind waves) $c(\eta - \eta_{wave}\cos(\Theta)) = q$ Θ - angle between the wave direction and the

				normal boundary vector; 2 – Flather solution when a reference water level and a reference water flow are defined $\left(c\left(\eta-\eta_{ref.}\right)=q-q_{ref.}\right)$; 3 – Blumberg and Kantha (19855). Radiation of water level plus a decay term $\left(\frac{\partial\eta}{\partial t}+\sqrt{gh}\frac{\partial\eta}{\partial x}=\frac{\eta-\eta_{impose}}{T_{decay}}\right)$.
ENTERING_WAVE	0	Integer	ENTERING_WAVE : 1	Checks if the user wants to impose a wave with a specific direction in the boundary. Valid only if RADIATION : 1
WAVE_DIRECTION	0	Real (degrees)	WAVE_DIRECTION : 90.	Wave direction in degrees (0 – East; 90-North), imposed in the boundary . Valid only if RADIATION : 1
TLAG_FILE			TLAG_FILE : TdecayKantha.txt	The name file where are define the relaxation times for the Blumberg and Kantha (1985) radiation boundary condition
LOCAL_SOLUTION	0	integer	LOCAL_SOLUTION : 2	Check what type o local (or reference) solution the user wants to use as a reference for the radiative and relaxation boundary conditions 1 – No local solution; 2 – A coarser grid model is the local solution. In this case the SUBMODEL option must be active; 3 – A field define in the assimilation module is the local solution; 4 – The velocities and water levels defined in points in the tidal gauges module are the local solution. In this case the field is construct by triangulation; 5 – In his case the local solution results from the sum of the field define in the assimilation module and the solution of a coarser model. In this case the SUBMODEL option must be active;
VELTANGENTIALBOUNDARY	2	integer	VELTANGENTIALBOUNDARY : 1	Checks the velocities the user want to impose between two boundary points: 1 – null value 2 – null gradient
VELNORMALBOUNDARY	2	integer	VELNORMALBOUNDARY : 1	Checks the velocities the user want to impose in the exterior faces: 1 – null value 2 – null gradient
NULL_BOUND_HORADV	1	integer	NULL_BOUND_HORADV : 0	Check if the user wants to turn off the horizontal transport of momentum in the boundary (1) or not (0)
CYCLIC_BOUNDARY	0	integer	CYCLIC_BOUN	Check if the user wants to

			DARY : 1	impose a CYCLIC boundary condition (1) or not (0).
BAROCLINIC_RADIATION	0	integer	BAROCLINIC_RADIATION : 1	Check if the user wants to radiate internal waves. The options are: 0 – No radiation, 1 – The horizontal baroclinic velocities in the exterior faces are estimated with a radiation equation 2 – In this case the vertical in the boundary column are estimated with a radiation equation
CELERITY_TYPE	1	integer	CELERITY_TYPE :	The options to compute the internal waves celerity are: 0 – Based on the internal variability (Orlanski, 1976) 1 – Value defined by the user 2 – $c = \sqrt{10^{-3} gh}$ (Oey and Chen, 1992)
INTERNAL_CELERITY	2.	Real (m/s)	INTERNAL_CELERITY : 1.2	In case of option CELERITY_TYPE : 1 the user can define the internal waves celerity with this keyword
DECAY_IN	86400		DECAY_IN	When the CELERITY_TYPE : 2 is active a decay term is add to the radiation equation when the internal variability says that the waves are entering in the domain then the celerity is set to zero. The decay time inward is defined with this keyword (see Marchesiello et al., 2001)
DECAY_OUT	864000		DECAY_OUT	When the CELERITY_TYPE : 1 or 2 is active a decay term is add to the radiation equation. For option CELERITY_TYPE : 2 this decay time is only used when the waves are leaving the domain. The decay time outward is defined with this keyword (see Marchesiello et al., 2001)

Table 18 – An example of a possible open boundary condition. In this case the model radiates the difference between the computed barotropic flow and the bartropic flow of the coarser grid.

```
TIDE              : 0
DATA_ASSIMILATION : 0
BRFORCE           : 0
SUBMODEL          : 1
RADIATION         : 2
LOCAL_SOLUTION    : 2
```

Table 19 – Definition of decay times for the Blumberg and Kantha (1985) boundary condition

Input data file		TLAG_FILE (see Table 17)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
<TlagBegin> <TlagEnd>	*	block	<TlagBegin> 1000 10 1000 10 <TlagEnd>	Decay times per cell in seconds. The reading sequence is the following: do i=ILB,IUB do j=JLB,JUB Tdecay(i,j) enddo enddo $\left(\frac{\partial \eta}{\partial t}+\sqrt{gh}\frac{\partial \eta}{\partial x}=\frac{\eta-\eta_{impose}}{T_{decay}}\right)$

Tidal Gauges input

The user can specified for several points the evolution of water level and of velocity. The points not define are obtained by triangulation from the defined points. These points are defined in the input data file of the module gauge (IN_TIDES - see Figure 1). Initially these points were only used to define water level variability that why in the code they are call tidal gauges, but now the user can also define associate to this points velocities.

Table 20 – Definition of keywords in the input data file of the module gauges (IN_TIDES - see Figure 1). This module allow the user to define as an input data the evolution of water level and of velocity in specific points.

Input data file		IN_TIDES (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
<begingauge> <endgauge>	*	String (block)	<begingauge> <endgauge>	Keywords block where a tidal gauge characteristics are describe (ex: location, reference level)
NAME	*	String	NAME : Sines	The name the user wants to give to the tidal gauge
LONGITUDE	*	3*real	LONGITUDE : -9 1 45	Longitude in degrees minutes and seconds
LATITUDE	*	3*real	LATITUDE : 38 33 54	Latitude in degrees minutes and seconds
METRIC_X	*	Real	METRIC_X : -9.029	Location in the X direction in the coordinates use to define the bathymetry
METRIC_Y	*	Real	METRIC_Y : 38.565	Location in the Y direction in the coordinates use to define the bathymetry
REF_LEVEL	*	Real	REF_LEVEL : 2.08	Reference level of the tidal Gauge
TIME_REF	*	Real	TIME_REF : 1	Tidal gauge time reference (0 – GMT)
HARMONICS	**	string string+2*real	HARMONICS Q1 0.01 267 O1 0.06 314	This is alwys the last KEYWORD define in guage block and below it is

			K1 0.07 63. 2N2 0.03 42	define the amplitude (m) and the phase (degrees) of each tidal component (ex: M2).
EVOLUTION	Harmonics	String	EVOLUTION : Time Serie	The water level variability can be defined with tidal harmonics (Harmonics) or with a time serie (Time Serie) where to several specific dates are associated water levels.
EVOLUTION_VEL	Harmonics	String	EVOLUTION : Time Serie	The velocity variability in the tidal gauge location can be defined with tidal harmonics (Harmonics) or with a time serie (Time Serie) where to a several specific dates is associated velocities
EVOLUTION_REF	Constant	String	EVOLUTION : Time Serie	The reference variability in the tidal gauge location can be defined with a constant value (Constant) given in the keyword REF_LEVEL or with a time serie (Time Serie) where to a several specific dates are associated reference levels.
COVERED_COLUMN	**	Integer	COVERED_COLUMN : 14	In case of any of the tidal gauge properties variability is defined using a time serie then is necessary to define a column of the time serie file that indicates if the tidal gauge is under water or not along time. The first column is reserve to define time.
LEVEL_COLUMN	**	Integer	LEVEL_COLUMN : 12	In case of water level variability is defined using a time serie then is necessary to define a column of the time serie file where the water level is defined.
REFLEVEL_COLUMN	**		REFLEVEL_COLUMN : 12	In case of the reference level variability is defined using a time serie then is necessary to define a column of the time serie file where the reference level is defined
VELU_COLUMN	**		VELU_COLUMN : 10	In case of the velocity variability is defined using a time serie then is necessary to define a column of the time serie file where the velocity in the X direction is defined
VELV_COLUMN	**		VELV_COLUMN : 11	In case of the velocity variability is defined using a time serie then is necessary to define a column of the time serie file where the velocity in the Y direction is defined

*- necessary to define this keyword;

** - In case of using a time serie to define a specific property (water level, reference level, velocity).

Table 21 – An example of input data file of the module gauge (IN_TIDES - see Figure 1).

```
<begingauge>
NAME       : Cascais
LONGITUDE :     -9.0000     11.0000     59.9993
LATTITUDE  :     40.0000      5.0000     59.9945
METRIC_X   :     -9.2000
METRIC_Y   :     40.1000
REF_LEVEL  :      1.9600
TIME_REF   :      0.0000
HARMONICS
Q1      0.016959    267.9709
O1      0.062444    314.7122
K1      0.066952     63.3073
2N2     0.029998     42.3924
N2      0.216744     51.6643
M2      1.028724     70.8528
<endgauge>

<begingauge>
NAME       : Peniche
LONGITUDE :     -9.0000     36.0000      0.0014
LATITUDE   :     40.0000      5.0000     59.9945
METRIC_X   :     -9.6000
METRIC_Y   :     40.1000
REF_LEVEL  :      1.9600
TIME_REF   :      0.0000
HARMONICS
Q1      0.016840    268.4873
O1      0.061739    315.2913
K1      0.066442     63.8877
2N2     0.029742     42.5366
N2      0.214662     51.8412
M2      1.016305     70.9776
<endgauge>
```

Assimilation Data Input

Table 22 – Keyword that can be use to give data input to the assimilation module.

Input data file		ASSIMILA_DAT (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
<beginproperty> <endproperty>	*	Block	<beginproperty> NAME : water level UNITS : m DEFAULTVALUE : 0.0 DEFAULT_COEF : 1e32 DIMENSION : 2D IN_SPACE : FILE2D FILENAME_COEF: ..\..\boundaries\Vigo\DecayTimeZVigo.dat FILENAME_PROP: ..\..\boundaries\Vigo\DecayTimeZVigo.dat <endproperty>	This file function as a data base where several field properties are defined and can be use as reference solution to condition the evolution of the water and hydrodynamic properties.

NAME	**	string	NAME : water level	Property name and is units don't have a default value. The program stops when it is not specified the property name and units
UNITS	**	String	m	See cell above
DESCRIPTION	No description available.	String	DESCRIPTION : satellite altimetry	The property description is a character*132 where the user can store information about the property.
IN_TIME	CONSTANT	string	IN_TIME : VARIABLE	The water or hydrodynamic property can be CONSTANT in time or VARIABLE (not active yet)
IN_SPACE	CONSTANT	string	IN_SPACE : FILE2D	The user can choose 5 types of initialization of the property values: CONSTANT - a constant value is admitted in all the domain BOXES - a constant value for each box is admitted FILE1D - the property values are read from a ASCII file in 1D. This methodology is use to define a profile assumed equal in all domain. FILE2D - the property values are read from a ASCII file in 2D. If the property is 3D in this way is consider a homogenous profile. FILE3D - the property values are read from a ASCII file in 3D
IN_SPACE_COEF	CONSTANT	string	IN_SPACE_COEF : FILE3D	The user can choose 5 types of initialization of the decay times associated with a property (see above cell).
DIMENSION	3D	string	DIMENSION : 2D	The user can define properties 3D (ex: temperature) or 2D (ex: water level)
DEFAULTVALUE	-9.9e15	real	DEFAULTVALUE : 12	Property default value
DEFAULT_COEF	0	Real	DEFAULT_COEF : 3600.	Decay time default value
RANDOM_COMPONENT	0	Real	RANDOM_COMPONENT : 1	Interval of a random uniform distribution centered in the property value. For example a random component of 1 means that the property value (P) will oscillate randomly between P-0.5 and P+0.5.
COLD_RELAX_PERIOD	0.	Real (s)	COLD_RELAX_PERIOD : 3600.	The user specify the period along which wants the relaxation have a linear growth
FILENAME_PROP	***		FILENAME_PROP : d:\Temp.txt	This keyword is used to give the model the filename where the

				values are defined or where the boxes structure is defined.
FILENAME_COEF	***		FILENAME_COEF : d:\DecayTemp.txt	See above cell
BOXES_PROP	****	n*real	BOXES_PROP : 2 3.2 4	The n boxes property value is defined by this keyword. The first value corresponds to the first box and so one.
BOXES_COEF	****	n*real	BOXES_ COEF : 200 320 400	Definition of the decay times of specific property using boxes. The first value corresponds to the first box and so one.

*- The block type keyword do not have default values associated.
** - A valid name and units must be defined
*** - If an option different from constant in space is active then this keyword must be defined.
**** - If the property or the decay time is defined using boxes then this keyword must be defined.

Table 23 – An example of data assimilation property fields and correspondent decay times definition.

```
<beginproperty>
NAME         : water level
UNITS        : m
DEFAULTVALUE : 0.0
DEFAULT_COEF : 1e32
DIMENSION    : 2D
IN_SPACE     : FILE2D
FILENAME_COEF: ..\..\boundaries\Vigo\DecayTimeZVigo.dat
FILENAME_PROP: ..\..\boundaries\Vigo\DecayTimeZVigo.dat
<endproperty>


<beginproperty>
NAME         : velocity U
UNITS        : m/s
DEFAULTVALUE : 0.0
DEFAULT_COEF : 1e32
DIMENSION    : 3D
IN_SPACE     : FILE2D
FILENAME_COEF: ..\..\boundaries\Vigo\DecayTimeUVigo.dat
FILENAME_PROP: ..\..\boundaries\Vigo\DecayTimeUVigo.dat
<endproperty>

<beginproperty>
NAME         : velocity V
UNITS        : m/s
DEFAULTVALUE : 0.0
DEFAULT_COEF : 1e32
DIMENSION    : 3D
IN_SPACE     : FILE2D
FILENAME_COEF: ..\..\boundaries\Vigo\DecayTimeVVigo.dat
FILENAME_PROP: ..\..\boundaries\Vigo\DecayTimeVVigo.dat
<endproperty>

<beginproperty>
NAME         : temperature
UNITS        : ºC
DEFAULTVALUE : 0.0
DEFAULT_COEF : 1e32
DIMENSION    : 3D
IN_SPACE     : FILE2D
FILENAME_COEF: ..\..\boundaries\Vigo\DecayTimeZVigo.dat
FILENAME_PROP: ..\..\boundaries\Vigo\DecayTimeZVigo.dat
<endproperty>
```

Table 24 – File format to be used when the user wants to define the rugosity absolute coefficient or manning coefficient variable in space.

Input data file		FILENAME_PROP or FILENAME_COEF (see Table 22)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
<ValueBegin> <ValueEnd>	*	block	<ValueBegin> 1 1 0.002 1 2 0.003 2 1 0.0034 2 2 0.0035 <ValueEnd>	The reading sequence is the following for the assimilation field values or decay times if in each line is only one number: do i=ILB,IUB do j=JLB,JUB Value(i,j,k) enddo enddo In this case the number of values must be equal to the number of grid cells. However if for line exists 4 values then the models reads the follow values until the end of the block: i j k value(i,j,k) In this way the user can define only some cells and for the other the default values area assumed. If only exists three values per line then the model read: i j Value(i,j,k) and assumes a homogeneous profile. If exist only 2 values per line the model reads: k Value(i, j, k) In this case model assumes no horizontal gradients.

Land

The fluxes of mass along the land boundary are null and by default the momentum fluxes are also null. However the user can active a no slip boundary condition.

Table 25 – Keywords available to control the land boundary condition.

Input data file		IN_DAD3D (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
SLIPPING_CONDITION	1	integer	SLIPPING_CONDITION : 0	Checks if the user want to consider the slipping condition for horizontal diffusion (1) or not (0).

Vertical Boundaries

Surface

Table 26 – Keywords used in the hydrodynamic input data file (IN_DAD3D see Figure 1 and Table 1) to control the surface boundary condition.

Input data file		IN_DAD3D (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
SURFACEWATERFLUX	0	integer	SURFACEWATERFLUX : 1	Checks if the user want to consider the effect of precipitation and evaporation
ATM_PRESSURE	0	Integer	ATM_PRESSURE : 1	Check if the user wants to consider the atmospheric pressure (1) or not (0)
WIND	0	Integer	WIND : 2	Check if the user wants to consider the wind stress (1) or not (0) or wind stress with a smooth initial period (2)
WIND_SMOOTH_PERIOD	86400.	Real (seconds)	WIND_SMOOTH_PERIOD : 172800.	The user can impose a specific period in seconds after which the model considers the total effect of wind stress. Along this period the wind stress amplitude is multiplied by a coefficient that has a linear evolution between 0 and 1. By default this period is zero seconds

Surface properties input

Table 27 – Definition of the surface fluxes in the input data file of the module surface.

Input data file		SURF_DAT (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
RUGOSITY	0.0025	Real (m)	RUGOSITY : 0.003	Rugosity coefficient. This coefficient is used in vertical turbulence parameterization
<beginproperty> <endproperty>	*	Block	<beginproperty> NAME : atmospheric pressure UNITS : atm IN_TIME : CONSTANT IN_SPACE : CONSTANT DEFAULTVALUE 1e5 <endproperty>	This block is used to define the surface properties that are necessary to compute the fluxes of momentum and mass between the atmosphere and the water column.
NAME	**	string	NAME : atmospheric pressure	Surface property name
UNITS	**	string	UNITS : atm	Surface property units
DESCRIPTION	No description available.	String	DESCRIPTION : climatologic solar radiation	The property description is a character*132 where the user can store information about the property.
IN_TIME	CONSTANT	string	IN_TIME : VARIABLE	The surface property can be CONSTANT in time or VARIABLE (not active yet)
IN_SPACE	CONSTANT	string	IN_SPACE : FILE	The user can choose 3 types of initialization of the surface property values: CONSTANT - a constant value is admitted in all the domain BOXES - a constant value for each box is admitted FILE - the property values are read from a file (see Table 29)
DIMENSION	3D	string	DIMENSION : 2D	The user can define properties 1D scalar (ex: solar radiation) or 2D vectorial (ex: wind stress)
DEFAULTVALUE	0. or 0. 0.	Real or 2*real	DEFAULTVALUE : 2 2.1	Property default value. For the vectorial case two values must be define (X,Y)
RANDOM_COMPONENT	0	Real	RANDOM_COMPONENT : 1	Interval of a random uniform distribution centered in the surface property value. For example a random component of 1 means that the property value (P) will oscillate randomly between P-0.5 and P+0.5.
COLD_RELAX_PERIOD	0.	Real (s)	COLD_RELAX_PERIOD : 3600.	The user specify the period along which wants the relaxation have a linear growth
TIME_SERIE	0	integer	TIME_SERIE : 1	Checks out if the user pretends to write time series of this property (1) or not (0). This is valid only if at least one block

				with a time serie location is defined (see Table 49).
ALBEDO	0.0	Real	ALBEDO : 0.2	Only necessary if the property is "solar radiation". Reflection coefficient of the water surface. Varies between 0 (no reflection) and 1 (total reflection)
DEFINE_CDWIND	0	integer	DEFINE_CDWIND : 1	Checks if the user wants to specified a shear coefficient (1) or not (0). If the option is 0 then the model compute the shear coefficient function of the wind velocity using the Large and Pond (1981) formulation.
CDWIND	0.0015	Real	CDWIND : 0.0015	Shear coefficient use to compute the wind stress function of the velocity square.
MAIN_SOURCE	FILE	string	MAIN_SOURCE : MODEL	A surface property variable in time can be defined in a file (FILE) or function of other properties (MODEL)
FILE_FORMAT	EU_CENTER	string	FILE_FORMAT : MM5	A surface property variable in time can be defined in a file (FILE) using the follow formats - EU_CENTER (a binary format) - MM5 (HDF formar) - ASCII_COL (ASCII time série format). The last format is the some of the time series output. The first two are more complex it is necessary to see the code of module surface.
FILENAME	***		FILENAME : d:\AtmosphericMM5.txt	This keyword is used to give the model the filename where the values are defined or where the boxes structure is defined.
DATA_COLUMN	****	Integer	DATA_COLUMN	In case of scalar surface property is defined using a time serie then is necessary to define a column of the time serie file where the surface property is defined.
DATA_COLUMN_X	****	Integer	DATA_COLUMN_X	In case of the X component surface property is defined using a time serie then is necessary to define a column of the time serie file where the surface property is defined.
DATA_COLUMN_Y	****	Integer	DATA_COLUMN_Y	In case of the Y component surface property is defined using a time serie then is necessary to define a column of the time serie file where the surface property is defined.
GRID_FILENAME	***	String	GRID_FILENAME : d:\MM5Out.hdf	In the case of FILE_FORMAT : MM5 then is necessary to give a file name where the grid of MM5 file output is defined.

GRID_POINT	1	String	GRID_POINT : 2	The surface property in MM5 file is defined in the grid: 1 – center point 2 - cross point
STATISTICS	0	integer	STATISTICS : 1	Checks out if the user pretends the statistics of the hydrodynamic properties (1) or not (0).
STATISTICS_FILE	See Table 42	string	STATISTICS_FILE : d:\HydroStatistics.txt	The statistics definition file of the surface properties
RAMP	0	Integer	RAMP : 1	Check if the user wants to start with a surface property null and only after a specific period the total force is compute (1) or not (0)
RAMP_PERIOD_UNITS	1	String	RAMP_PERIOD_UNITS	The time units of the RAMP period. The options available are: 1 – Inertial Periods. The time units are inertial periods 2*pi/ f 2 - Seconds
RAMP_PERIOD	1.	Real	RAMP_PERIOD	The period after which the total flux associated with a specific surface property is compute.
BOXESVALUE	+	n*real	BOXESVALUE	The n boxes scalar property value is defined by this keyword. The first value corresponds to the first box and so one.
BOXESVALUE_X	+	n*real	BOXESVALUE_X	The n boxes component X property value is defined by this keyword. The first value corresponds to the first box and so one.
BOXESVALUE_Y	+	n*real	BOXESVALUE_Y	The n boxes component Y property value is defined by this keyword. The first value corresponds to the first box and so one.

* - A block do not have a default value

** - Property name and is units don't have a default value. The program stops when it is not specified the property name and units.

*** - When MAIN_SOURCE : FILE or IN_SPACE : FILE or BOXES a filename must be given or the model stops.

**** - FILE_FORMAT : ASCII_COL then this keyword must be defined

+ - IN_SPACE : BOXES the surface values per box must be given or the model stops.

Table 28 – An example of surface properties definition.

```
<beginproperty>
   NAME           atmospheric pressure
   UNITS          atm
   IN_TIME        CONSTANT
   IN_SPACE       CONSTANT
   DIMENSION      1
```

```
    DEFAULTVALUE      1e5
    RANDOM_COMPONENT  0
    TIME_SERIE        0
<endproperty>
<beginproperty>
NAME                    : wind velocity
UNITS                   : m/s
DESCRIPTION             : calculated wind velocity
MAIN_SOURCE             : FILE
FILE_FORMAT             : MM5
FILENAME                : Z:\PrestigeSpill\GeneralData\GaliciaMeteo\Dia11_30\Wind_Day11to30.hdf
GRID_FILENAME           : Z:\PrestigeSpill\GeneralData\GaliciaMeteo\Dia11_30\BatimWind11to30.dat
IN_TIME                 : VARIABLE
IN_SPACE                : VARIABLE
DIMENSION               : 2
DEFINE_CDWIND           : 1
OUTPUT_TIME             : 0.0   21600.
TIME_SERIE              : 1
<endproperty>

<beginproperty>
NAME                    : wind stress
UNITS                   : m/s
DESCRIPTION             : calculated wind velocity
MAIN_SOURCE             : MODEL
IN_TIME                 : VARIABLE
IN_SPACE                : VARIABLE
DIMENSION               : 2
DEFINE_CDWIND           : 1
!OUTPUT_TIME            : 0.0   21600.
TIME_SERIE              : 1
<endproperty>
<beginproperty>
NAME                    : sensible heat
UNITS                   : W/m^2
DESCRIPTION             : european center values
MAIN_SOURCE             : FILE
FILE_FORMAT             : EU_CENTER
FILENAME                : ..\data\Interpolated_Fields.Dat
IN_TIME                 : VARIABLE
IN_SPACE                : FILE
OUTPUT_TIME             : 0. 172800.
<endproperty>


<beginproperty>
NAME                    : latent heat
UNITS                   : W/m^2
DESCRIPTION             : european center values
MAIN_SOURCE             : FILE
FILE_FORMAT             : EU_CENTER
FILENAME                : ..\data\Interpolated_Fields.Dat
IN_TIME                 : VARIABLE
IN_SPACE                : FILE
OUTPUT_TIME             : 0. 172800.
<endproperty>


<beginproperty>
NAME                    : infrared radiation
UNITS                   : W/m^2
DESCRIPTION             : european center values
MAIN_SOURCE             : FILE
FILE_FORMAT             : EU_CENTER
FILENAME                : ..\data\Interpolated_Fields.Dat
IN_TIME                 : VARIABLE
IN_SPACE                : FILE
OUTPUT_TIME             : 0. 172800.
<endproperty>

<beginproperty>
NAME                    : solar radiation
UNITS                   : W/m^2
```

```
DESCRIPTION              : european center values
MAIN_SOURCE              : FILE
FILE_FORMAT              : EU_CENTER
ALBEDO                   : 0.05  ![%]
FILENAME                 : ..\data\Interpolated_Fields.Dat
IN_TIME                  : VARIABLE
IN_SPACE                 : FILE
OUTPUT_TIME              : 0. 172800.
STATISTICS               : 1
STATISTICS_FILE          : ..\data\SurfStatistic.dat
<endproperty>

<BeginTimeSerie>
LOCALIZATION_I           : 30
LOCALIZATION_J           : 31
LOCALIZATION_K           : 1
<EndTimeSerie>
```

Table 29 – File format to be used when IN_SPACE : File and IN_TIME : CONSTANT.

Input data file		FILENAME (see Table 27)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
<ValueBegin> <VauleEnd>	*	block	<ValueBegin> 1000 100 10 5 1000 12 10 2 <ValueEnd>	The reading sequence is the following for scalar property: do i=ILB,IUB do j=JLB,JUB SurfaceProp(i,j) enddo enddo For a vectorial property 2 values per line are read (X, Y)

Bottom

Table 30 – Keywords available in the hydrodynamic input data file available to control the bottom boundary condition.

Input data file		IN_DAD3D (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
BOTTOMWATERFLUX	0	integer	BOTTOMWATERFLUX : 1	hecks if the user want to consider the effect of the soil infiltration or consolidation (see Table 31)

Table 31 – Keywords that the user can use to define the bottom boundary condition in the input data file of the module bottom (BOT_DAT - see Figure 1).

Input data file		BOT_DAT (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
MANNING	0	integer	MANNING : 1	Checks if the user wants to define the drag coefficient from the Manning

				coefficient (1) or from the absolute rugosity using the log profile (0). The manning coefficient can only be used in 2D models.
RUGOSITY	*	Real (m)	0.022	Depending of the option taken (MANNING : 0 or 1) this value is absolute rugosity or a manning coefficient.
RUGOSITY_FILE	******.***	string	RUGOSITY_FILE : d:\rugosity.txt	The absolute rugosity or manning coefficient can be defined in a file (Table 33)
RUGOSITY_BOX	******.***	string	RUGOSITY_BOX : d:\boxRugosity.txt	The absolute rugosity or manning coefficient can be defined using boxes. This keyword is used to say to the model the filename where the boxes are defined
BOXES_VALUES	**	n*real	BOXES_VALUES : 0.003 0.002	The n boxes rugosity values are defined by this keyword. The first value corresponds to the first box and so one.

Table 32 – Example what can be input data of the module bottom (BOT_DAT - see Figure 1)

```
RUGOSITY           : 0.0025
```

Table 33 – File format to be used when the user wants to define the rugosity absolute coefficient or manning coefficient variable in space.

Input data file		RUGOSITY_FILE (see Table 31)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
<RugosityBegin> <RugosityEnd>	*	block	<RugosityBegin> 0.002 0.003 0.0034 0.0035 <RugosityEnd>	The reading sequence is the following for the rugosity coefficient: do i=ILB,IUB do j=JLB,JUB Rugosity(i,j) enddo enddo

Turbulence parameterisation

Hydrodynamic Input Data File

Table 34 – Keywords available in hydrodynamic input data file (IN_DAD3D - see Figure 1) to control the turbulence parametrization.

Input data file		IN_DAD3D (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
BIHARMONIC	0	integer	BIHARMONIC	Check if the user wants to compute the horizontal diffusion of momentum with a bi-harmonic formulation (1) or not (0).
BIHARMONIC_COEF	1e9	real	BIHARMONIC_ COEF	Horizontal diffusion ocefficent used when the bi-harmonic option is active.
SLIPPING_CONDITION	1	integer	SLIPPING_CON DITION : 0	Checks if the user wants to consider the slipping condition for horizontal diffusion (1) or not (0).

Turbulence Input Data File

Table 35 – Keywords used to control the diffusion of momentum in the horizontal direction. These keywords are defined in the input data file of the module turbulence (IN_TURB see Figure 1 or Table 1).

Input data file		IN_TURB (see Figure 1 or Table 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
Background_Viscosity	1.3e-6	Real	Background_Visco sity : 1e-8	Background viscosity/diffusivity. By default, it is equal to molecular diffusion.
MIXLENGTH_MAX	100.	Real	MIXLENGTH_M AX : 10.	Maximum allowed mixing length. Parameter used in the Nihoul and Leendertse (Nihoul, 1984) parameterizations
MODTURB	constant	string	MODTURB : turbulence_ equation	The options are: constant leendertsee (prandlt; L, Ri) backhaus (Ri) pacanowski (Ri) nihoul (prandlt, L, Ri) turbulence_ equation (k,ε)
MODVISH	constant	string	MODVISH	The options are: constant estuary (v, H)

				smagorinsky (Smagorinsky, 1963)
CONTINUOUS	0	Integer	CONTINUOUS : 0	Check if the user wants to continuum a past run (1) or not (0)
TIME_SERIE	0	integer	TIME_SERIE : 1	Checks out if the user pretends to write time series of this property (1) or not (0). This is valid only if at least one block with a time serie location is defined (see Table 49).
OUTPUT_TIME	*	(n)*real	OUTPUT_TIME : 3600. 7200. 1800.	The n-1 first values are considered specific outputs in time. These first values must be in ascending order, the values are given in seconds and the reference is the start date (0 seconds). The n (last) value is consider to be the output time interval from the n-1 output example: START : 1998 1 1 0 0 0 END : 1998 1 1 12 0 0 OUTPUT_TIME : 0. 7200. 14400. 14400. The result of this are the outputs in the follow dates: 1998 1 1 0 0 0 1998 1 1 2 0 0 1998 1 1 4 0 0 1998 1 1 8 0 0 1998 1 1 12 0 0
DT_OUTPUT_TIME	DT/2 (see Table 7)	Real (s)	DT_OUTPUT_TIME : 600.	Output interval for the time series. Defined out of the property blocks.
<BeginTimeSerie> LOCALIZATION_I LOCALIZATION_J LOCALIZATION_K <EndTimeSerie>	*	block	<BeginTimeSerie> LOCALIZATION_I : 27 LOCALIZATION_J : 43 LOCALIZATION_K : 1 <EndTimeSerie>	In each block the user can define the location of each time serie.
OUTPUT_TIDE	0	integer	OUTPUT_TIDE : 1	Checks out if the user pretends to write tidal information in HDF output (1) or not (0). This is valid if the tide is one of the forcing mechanisms
MLD	0	Integer	MLD : 1	Checks out if the user pretends to compute the mixed layer length (1) or not (0).
MLD_BOTTOM	0	integer	MLD_BOTTOM	Checks out if the user pretends to compute the bottom mixed layer length (1) or not (0). Valid only if MLD : 1 option is active.
MLD_Method			MLD_Method	The option available are : 1 – Turbulent kinetic energy (TKE) inferior to a minimum predefined; 2 – Richardson number (Ri) superior to a critical value predefined; 3 – Maximum value of Brunt-Vaisalla frequency (N)

TKE_MLD	1e-5	real	TKE_MLD : 1e-6	TKE limit used to compute the surface mixing length based on the TKE
RICH_MLD	0.5	real	RICH_MLD : 0.5	Ri used to compute the surface mixing length based on the Ri number
STATISTICS_MLD	0	integer	STATISTICS_MLD : 1	Checks out if the user pretends to do a statistics analysis of the surface mixing length (1) or not (0).
STATISTICS_MLD_FILE	*	string	STATISTICS_MLD_FILE : d:\StatInput.dat	The statistics analysis definition file of the surface mixing length (see Table 42)
VISCOSITY_H_FILE	******.***	string	VISCOSITY_H_FILE : d:\ViscH.txt	File of horizontal viscosities. The format is defined in Table 37. Valid only if " MODVISH : constant ".
VISCOSITY_H	**	real	VISCOSITY_H : 10.	Default horizontal viscosity.
VISH_REF	50.	real	VISH_REF	Horizontal viscosity used as the minimum viscosity possible when MODVISH : estuary or Smagorinsky.
HREF_VIS	10.	real	HREF_VIS : 5.	Water column reference thickness used in the option MODVISH : estuary
VREF_VIS	1	real	VREF_VIS : 0.5	Reference velocity used in the option MODVISH : estuary
HORCON	0.2	real	HORCON : 0.4	Limits : 0< HORCON<1. Coefficient use in the option MODVISH : smagorinsky.
PRANDTL_0	1.	real	PRANDTL_0 : 2.	Initial vertical Prandtl number. Used to compute the initial diffusivity.
VISCOSITY_V	***	real	VISCOSITY_V	Default vertical viscosity.
MIXLENGTH_V	10.	Real	MIXLENGTH_V : 5.	Default vertical mixing length. Used to compute the random trajectory of particle (Lagrangian Module)
CONST_MIXING_LENGTH_HORIZONTAL	NYQUIST * DX	real	CONST_MIXING_LENGTH_HORIZONTAL : 20.	Default horizontal mixing length. Used to compute the random trajectory of particle (Lagrangian Module)
NYQUIST	2	real	NYQUIST : 6.	By default the horizontal mixing length is consider equal to the spatial step plus the NYQUIST number that in theory represent the number of points to compute a wave. In practice this value is 4 or 5 basically the smaller eddies compute by the model have a diameter of 4 to 6 cells.

* - If the option STATISTICS_MLD : 1 is active then this keyword must be defined.

** - If the option MODVISH : constant is active then this keyword must be defined.

*** - If the option MODTURB : constant is active then this keyword must be defined.

Table 36 – An example of a input data file of the module turbulence.

```
!VISCOSITY_V          : 0.001
!VISCOSITY_H          : 5.0

VISH_REF           : 1
HORCON             : 0.04
MODVISH            : smagorinsky

MODTURB            : turbulence_equation

CONTINUOUS                       : 0

MLD        : 1
MLD_BOTTOM  : 1

TKE_MLD    : 1e-5
RICH_MLD   : 0.5

TIME_SERIE           : 0

OUTPUT_TIME          : 0 900.
```

Table 37 – File format to be used to define a field of horizontal viscosities constant in time and variable in space.

Input data file		VISCOSITY_H_FILE (see Table 35)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
<HorizontalViscosityBegin> <HorizontalViscosityEnd>	*	block	<HorizontalViscosityBegin> 10 5 12 2 <HorizontalViscosityEnd>	The reading sequence is the following for horizontal viscosity: do i=ILB,IUB do j=JLB,JUB ViscH(i,j) enddo enddo

When the option MODTURB : turbulent_equation is active then is used the model GOTM (http://www.gotm.net) to compute the evolution of vertical visocosity and diffusivity. In this case is necessary to define in the "nomfich.dat" the filename where the parameters specific of the GOTM model are defined. The Keyword is TURB_GOTM (see Table 1 and Figure 1). An example of this file is presented in Table 38.

Table 38 – An example of data file where parameters specific of the GOTM turbulence model are defined.

```
!-------------------------------------------------------------------------------
! The namelists 'turbulence','turb_parameters', 'keps', 'my', 'stabfunc',
! 'iw' and 'eobs' are all read from init_turbulence in the module
```

```
! turbulence.F90.
! They have to come in this order.
!-------------------------------------------------------------------------------

!-------------------------------------------------------------------------------
! General turbulence settings.
!
! turb_method=          0: Convective Adjustment
!                       1: Analytical eddy visc. and diff. profiles, not coded yet
!                       2: Turbulence Model calculating TKE, length scale, stab. func.
! tke_method=           How to calculate TKE.
!                       1= Algebraic equation.
!                       2= Dynamic equation for k-epsilon model.
!                       3= Dynamic equation for Mellor-Yamada model.
! len_scale_method=  How to calculate the lenght scale.
!                       1= Parabolic shape
!                       2= Triangle shape
!                       3= Xing and Davies [1995]
!                       4= Robert and Ouellet [1987]
!                       5= Blackadar (two boundaries) [1962]
!                       6= Bougeault and Andre [1986]
!                       7= Eifler and Schrimpf (ISPRAMIX) [1992]
!                       8= Dynamic dissipation rate equation
!                       9= Dynamic Mellor-Yamada kL equation
!
! stab_method=          How to calculate stability functions.
! Note that the given values for cm0,cmust,Prandtl0 are recommendations
! For values for ce3minus, see below.
!             1, Kantha and Clayson [1994],      full version, cm0 = 0.5544
!             2, Burchard and Baumert [1995],    full version, cm0 = 0.5900
!             3, Canuto et al. [2000] version A, full version, cm0 = 0.5270
!             4, Canuto et al. [2000] version B, full version, cm0 = 0.5540
!             5, Kantha and Clayson [1994],      quasi-eq. version, cm0 = 0.5544
!             6, Burchard and Baumert [1995],    quasi-eq. version, cm0 = 0.5900
!             7, Canuto et al. [2000] version A, quasi-eq. version, cm0 = 0.5270
!             8, Canuto et al. [2000] version B, quasi-eq. version, cm0 = 0.5540
!             9, Constant stability functions,   cm0 = cmust = 0.5477, Prandtl0=0.74
!             10, Munk and Anderson [1954],      cm0 = cmust = 0.5477, Prandtl0=0.74
!             11, Schumann and Gerz [1995],      cm0 = cmust = 0.5477, Prandtl0=0.74
!             12, Eifler and Schrimpf [1992],    cm0 = cmust = 0.5477, Prandtl0=0.74
!
! craig_banner= .true.: Craig and Banner wave breaking parameterisation
! length_lim=           apply length limitation or not
! k_min=   minimun TKE
! L_min=   minimum lengthscale
! eps_min=minimum dissipation
!-------------------------------------------------------------------------------
 &turbulence
  turb_method=                  2,
  tke_method=                   2,
  len_scale_method=   8,
  stab_method=                  3,
  craig_banner=                 .false.
  length_lim=                   .false.,
  k_min=              1.e-6,
  L_min=              0.01,
  eps_min=            1.e-12,
 /

!-------------------------------------------------------------------------------
!Empirical parameters used in turbulence modeling.
!
! kappa=    von Karman's constant.
! Prandtl0=The turbulent Prandtle number (constant)
! cm0=                  stab. func. for momentum for unstrat. equilibrium flow
!                       or if a "constant" stability function is used.
! cm_craig=             surface value for stability function for wave breaking,
!                       shold be set to cm0 except for stabfunc = 1, 2, 3, 4
! cw=                   proportionality factor for TKE injection
! cm0=                  stab. func. for momentum for unstrat. equilibrium flow
! galp=                 coef. for length limitation, should be 0.53
!-------------------------------------------------------------------------------
 &turb_parameters
```

```
 kappa=    0.4,
 Prandtl0=0.714,
 cm0=                  0.527,
 cm_craig=             0.73,
 cw=                   100.,
 galp=                 0.53,
/

!-------------------------------------------------------------------------------
! Empirical parameters used in the k-epsilon model.
!
! ce1=                 emp. coef. in diss. eq.
! ce2=                 emp. coef. in diss. eq.
! ce3minus=            ce3 for stable stratification
!          Recommended values for ce3minus
!          (steady-state Richardson number=0.25) are:
!          stab_method = 1 --> ce3minus = -0.404
!          stab_method = 2 --> ce3minus = -0.444
!          stab_method = 3 --> ce3minus = -0.629
!          stab_method = 4 --> ce3minus = -0.566
!          stab_method = 5 --> ce3minus = -0.404
!          stab_method = 6 --> ce3minus = -0.444
!          stab_method = 7 --> ce3minus = -0.629
!          stab_method = 8 --> ce3minus = -0.566
!          stab_method = 9 --> ce3minus = +0.499
!          stab_method =10 --> ce3minus = +0.035
!          stab_method =11 --> ce3minus = -0.368
!          stab_method =12 --> ce3minus = +0.239
! ce3plus=  ce3 for un-stable stratification
! sig_k=    Schmidt # for TKE eddy diffusivity
! flux_bdy=            flux boundary conditions
!-------------------------------------------------------------------------------
 &keps
 ce1=                  1.44,
 ce2=                  1.92,
 ce3minus=             -0.629,
 ce3plus=  1.0,
 sig_k=    1.,
 flux_bdy=             .true.,
 /

!-------------------------------------------------------------------------------
! Empirical parameters used by the Mellor-Yamada model.
!
! sl=eddy diffusivities of k and kL (sl=cl/sqrt(2))
! e1=coef. in MY kL equation
! e2=coef. in MY kL equation
! e3=coef. in MY kL equation
!          Recommended values for e3
!          (steady-state Richardson number=0.25) are:
!          stab_method = 1 --> ce3minus =  5.808
!          stab_method = 2 --> ce3minus =  5.888
!          stab_method = 3 --> ce3minus =  6.258
!          stab_method = 4 --> ce3minus =  6.132
!          stab_method = 5 --> ce3minus =  5.808
!          stab_method = 6 --> ce3minus =  5.888
!          stab_method = 7 --> ce3minus =  6.258
!          stab_method = 8 --> ce3minus =  6.132
!          (for motivation, see Burchard [2000], JPO)
! MY_length=           prescribed barotropic lengthscale in kL eq.
!                      1=parabolic
!                      2=triangular
!                      3=lin. from surface
!-------------------------------------------------------------------------------
 &my
 sl=                   0.2,
 e1=                   1.8,
 e2=                   1.33,
 e3=                   6.258,
 MY_length=            3,
 /

!-------------------------------------------------------------------------------
```

```
! Empirical parameters used for the stabillity function calculations.
!
! a1=                    coef. in Galperin QE SF
! a2=                    coef. in Galperin QE SF
! b1=                    coef. in Galperin QE SF
! b2=                    coef. in Galperin QE SF
! c2=                    0.0 for Galperin SF, 0.7 for Kantha & Clayson SF
! c3=                    0.0 for Galperin SF, 0.2 for Kantha & Clayson SF
! qesmooth=              smooth in unstable stratification (true/false)
! qeghmax=               max. value of parameter gh in qeSF
! qeghmin=               min. value of parameter gh in qeSF
! qeghcrit= critical value of gh to start smoothing
!-------------------------------------------------------------------------------
 &stabfunc
  a1=                    0.92,
  a2=                    0.74,
  b1=                    16.6,
  b2=                    10.1,
  c2=                    0.7,
  c3=                    0.2,
  qesmooth=              .true.,
  qeghmax=               0.0233,
  qeghmin=               -0.28,
  qeghcrit= 0.02,
 /

!-------------------------------------------------------------------------------
! Internal wave parameters.
! iw_model=              IW specification
!                        0=no IW, 2=Large et al. 1994
! alpha=     coeff. for Mellor IWmodel (0: no IW, 0.7 Mellor 1989)
!
! The following six empirical parameters are used for the
! Large et al. 1994 shear instability and internal wave breaking
! parameterisations (iw_model = 2, all viscosities are in m**2/s):
!
! klimiw=    critcal value of TKE
! rich_cr=   critical Richardson number for shear instability
! numshear=              background diffusivity for shear instability
! numiw=     background viscosity for internal wave breaking
! nuhiw=     background diffusivity for internal wave breaking
!-------------------------------------------------------------------------------
 &iw
  iw_model=              0,
  alpha=     0.0,
  klimiw=    1e-6,
  rich_cr=   0.7,
  numshear=              5.e-3,
  numiw=     1.e-4,
  nuhiw=     1.e-5,
 /
```

Output control

Table 39 – Keywords that the user can use to control the output.

Input data file		IN_DAD3D (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
RESIDUAL	0	integer	RESIDUAL : 1	Check if the user want to compute the residual values of the hydrodynamic properties (1) or not (0). In the affirmative case the model writes the residual values in the end of the output file of transient results (HDF format).
ENERGY	0	integer	ENERGY : 1	Check if the user wants to compute the potential and kinetic energy of the entire domain. In the affirmative case the results are written in ASCII format. The file name is defined with the keyword ENERGY (see Figure 1) in central data file where file names and paths are defined.
TIME_SERIE	0	integer	TIME_SERIE : 1	Checks if the user pretends to write time series (1) or not (0) (see Table 49).
<BeginTimeSerie> LOCALIZATION_I LOCALIZATION_J LOCALIZATION_K <EndTimeSerie>	*	block	<BeginTimeSerie> LOCALIZATION_I : 27 LOCALIZATION_J : 43 LOCALIZATION_K : 1 <EndTimeSerie>	In each block the user can define the location of each time serie.
DT_OUTPUT_TIME	DT/2 (see Table 7)	Real (s)	DT_OUTPUT_TIME : 600.	Output interval for the time series
MAX_BUFFER_SIZE	100000	Real (bytes)	MAX_BUFFER_SIZE : 1e3	The maximum Buffer Size is set here to 0.1Mb (for each property). This lets perform 25000 outputs to the buffer (considering each output of 4 bytes). Basically when the time serie output occupies in memory this value then the information is written to the file and the buffer is set to zero.
TIDE_PREVIEW	0	integer	TIDE_PREVIEW : 1	Checks if the user wants to now in advance all the outputs relatively to the high tide (the present GUI do not use this information)
OUTPUT_TIME	*	(n)*real	OUTPUT_TIME : 3600. 7200. 1800.	The n-1 first values are considered specific outputs in time. These first values must be in ascending order, the values are given in seconds and the reference is the start date (0 seconds). The n (last) value is consider to be the output time interval from the n-1 output example: START : 1998 1 1 0 0 0 END : 1998 1 1 12 0 0 OUTPUT_TIME : 0. 7200. 14400. 14400. The result of this are the outputs in the follow dates:

				1998 1 1 0 0 0 1998 1 1 2 0 0 1998 1 1 4 0 0 1998 1 1 8 0 0 1998 1 1 12 0 0
STATISTICS	0	integer	STATISTICS : 1	Checks out if the user pretends the statistics of the hydrodynamic properties (1) or not (0).
STATISTICS_FILE	*	string	STATISTICS_FILE : d:\HydroStatistics.txt	The statistics definition file of the hydrodynamic properties (see Table 42)
RECORDING	0	integer	RECORDING : 1	Checks if the user wants to record the hydrodynamic properties in binary format that can be used latter by the option 'Read_File' of the Keyword = EVOLUTION. By default the model do not record the hydrodynamic properties

* - If the STATISTICS option is active then the statistics input data filename must be defined.

Table 40 – An example on how is possible to control the hydrodynamic output.

```
RESIDUAL             : 1
ENERGY               : 1
RECORDING            : 0
 OUTPUT_TIME         : 0 900
TIME_SERIE           : 1
MAX_BUFFER_SIZE  : 1e4.
<BeginTimeSerie>
LOCALIZATION_I       : 27
LOCALIZATION_J       : 43
LOCALIZATION_K        : 1
<EndTimeSerie>
<BeginTimeSerie>
LOCALIZATION_I       : 17
LOCALIZATION_J       : 21
LOCALIZATION_K        : 1
<EndTimeSerie>
```

Hydrodynamic solution

Table 41 – Keywords used to do an output of an hydrodynamic solution. This solution can be used later as an input solution.

Input data file		IN_HYDRO_FILE (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
OUTPUT	0	Integer	OUTPUT : 1	Check if the user wants to write an hydrodynamic solution to a file in binary format.
OUT_FILE_VERSION	2	Integer	OUT_FILE_VERSION : 1	The user can choose from two options. One more old (1) and another more recent and more optimized (2). This last option should be used always. The old option was maintained only to allow the used of files written by model old versions.

TIME_INTEGRATION	0	Integer	TIME_INTEGRATION : 1	Check if the user wants to integrate is time the solution (1) or not (0).
DT_HYDROFILE	*	Real (seconds)	DT_HYDROFILE	If the TIME_INTEGRATION option is active then is necessary specified the time step along each the hydrodynamic properties are integrated in time
WINDOW	**	4*integer	WINDOW : 20 40 35 75	Lower, upper lines and lower, upper columns. Given in this order is possible to define a grid sub-domain. In this case only the hydrodynamic properties in this sub-domain are record.
SPACE_INTEGRATION	0	Integer	SPACE_INTEGRATION : 1	Check if the user wants to integrate is space the solution (1) or not (0). This integration is basically consists in creating new cells that result from merging the reference grid cells.
N_ITEGRATION_CELLS	***	Integer	N_ITEGRATION_CELLS : 3	If the SPACE_INTEGRATION option is active then the user must define the merge coefficient. For example if the value is 3 then the model will create cells that correspond 3x3 cells of the reference grid.
NEW_BATIM	****	String	NEW_BATIM	If the WINDOW or SPACE_INTEGRATION options are active a new bathymetry file is created and this keyword is used to define the name of the new file.

*- If not defined the model stops if the TIME_INTEGRATION option is active .
** - If the keyword WINDOW is written then the user must write four integer number in front: Imin, Imax, Jmin, Jmax;
*** - If not defined the model stops if the SPACE_INTEGRATION option is active .
**** - If not defined the model stops if the SPACE_INTEGRATION or WINDOW options are active .

```
OUTPUT                : 1
OUT_FIELD             :  OutHydroFile.bin
OUT_FILE_VERSION      : 2
DT_HYDROFILE          : 3600.
TIME_INTEGRATION      : 1
SPACE_INTEGRATION     : 1
N_ITEGRATION_CELLS    : 3
NEW_BATIM             : OutBatim.txt
WINDOW                :  20 40  30 75
```

Statistics Analysis

The user in the case of the hydrodynamic properties can perform several statistic analysis o the three velocity components. This analysis includes computing averages, standard deviations and frequencies. The user can do this analysis for the entire run, in daily and monthly bases.

Table 42 – Statistics input data file to control the type of statistics the user wants to do over some properties (ex: hydrodynamic, water properties, surface properties). Keywords use to define the statistics analysis.

Input data file		STATISTICS_FILE (see Table 39)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
METHOD_STATISTIC	1	integer	METHOD_ST ATISTIC : 3	The has the follow options: 1 – The statistics for 3D field are compute cell by cell 2 - The statistics for 3D field are integrated between specific layers 3 - The statistics for 2D field are compute cell by cell
GLOBAL_STATISTIC	0	integer	GLOBAL_STA TISTIC : 1	Checks if the user wants to output the statistics resulting of the global analysis of the entire run. The statistic analysis does not take in account previous runs.
DAILY_STATISTIC	0	integer	DAILY_STATI STIC : 1	Checks if the user wants to output the statistics relative to daily periods. In this case run must be greater than 1 day.
MONTHLY_STATISTIC	0	integer	MONTHLY_ST ATISTIC	Checks if the user wants to output the statistics relative to monthly periods. In this case run must be greater than 1 month.
<BeginClass> <EndClass>	*	block	<BeginClass> 0 .1 .1 .2 .3 .4 .4 2. <EndClass>	The user using this block can define intervals (classes) where a frequency analysis can be compute.
PERCENTILE	90.	Real (%)	PERCENTILE	The user can define a percentile and compute to each cell the correspondent class (see row above).
<beginlayer> <endlayer>	**	block	<beginlayer> LAYER_DEFI NITION : 1 MAX_DEPTH : 10000 MIN_DEPTH : 0 <endlayer>	For the option METHOD_STATISTIC : 3 is necessary to define the layers limit. This block is used to define one layer
LAYER_DEFINITION	2	integer	LAYER_DEFI NITION : 1	The layers can be define using depths (2) or the number of the grid layers (1)
MIN_LAYER	Bottom layer (Kmin)	Integer	MIN_LAYER	If the option LAYER_DEFINITION : 1 then the user must give the bottom limit layer
MAX_LAYER	Surface layer (Kmax)	Integer	MAX_LAYER	If the option LAYER_DEFINITION : 1 then the user must give the surface limit layer
MIN_DEPTH	0		MIN_DEPTH	If the option LAYER_DEFINITION

				: 2 then the user must give the bottom limit depth
MAX_DEPTH	10000		MAX_DEPTH	If the option LAYER_DEFINITION : 2 then the user must give the surface limit depth

Table 43 – An example of a file where is defined the type of statistic analysis of the hydrodynamic properties the user wants to output.

```
METHOD_STATISTIC  : 2
GLOBAL_STATISTIC  : 1
DAILY_STATISTIC   : 1
MONTHLY_STATISTIC : 1

<beginlayer>
LAYER_DEFINITION  : 1
MAX_DEPTH         : 10000
MIN_DEPTH         : 0
<endlayer>

<beginlayer>
LAYER_DEFINITION  : 1
MAX_DEPTH         : 10000
MIN_DEPTH         : 0
<endlayer>
<beginlayer>
LAYER_DEFINITION  : 1
MAX_DEPTH         : 10000
MIN_DEPTH         : 0
<endlayer>
<BeginClass>
-1 -.1
-.1 -.01
-.01 -1e-3
-1e-4 -1e-5
-1e-5 0
0 1e-5
1e-5 1e-4
1e-4 1e-3
1e-3 1e-2
1e-1 1
<EndClass>
```

Water properties evolution

Table 44 – Options available to define the time and spatial variability of the density field, important for the hydrodynamic if the baroclinic pressure effect is computed (BAROCLINIC : 1 see Table 13).

Input data file		DISPQUAL (see Figure 1)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
REFERENCE_DENSITY	1026.0	Real (kg/m^3)	REFERENCE_DENSITY : 1028	The default value for density
OUTPUT_TIME	*	(n)*real	OUTPUT_TIME : 3600. 7200. 1800.	The n-1 first values are considered specific outputs in time. These first values

				must be in ascending order, the values are given in seconds and the reference is the start date (0 seconds). The n (last) value is consider to be the output time interval from the n-1 output example: START : 1998 1 1 0 0 0 END : 1998 1 1 12 0 0 OUTPUT_TIME : 0. 7200. 14400. 14400. The result of this are the outputs in the follow dates: 1998 1 1 0 0 0 1998 1 1 2 0 0 1998 1 1 4 0 0 1998 1 1 8 0 0 1998 1 1 12 0 0 In the new version the output time is defined out of the block properties and there for equal for all properties.
DT_OUTPUT_TIME	DT/2 (see Table 7)	Real (s)	DT_OUTPUT_TIME : 600.	Output interval for the time series. Defined out of the property blocks.
<BeginTimeSerie> LOCALIZATION_I LOCALIZATION_J LOCALIZATION_K <EndTimeSerie>	*	block	<BeginTimeSerie> LOCALIZATION_I : 27 LOCALIZATION_J : 43 LOCALIZATION_K : 1 <EndTimeSerie>	In each block the user can define the location of each time serie.
<beginproperty> <endproperty>	*	block	<beginproperty> NAME : salinity UNITS : psu DESCRIPTION : sal DEFAULTVALUE : 36 INITIALIZATION_METHOD : CONSTANT ADVECTION_DIFFUSION : 0 <endproperty>	In each block the user can define the initial, the boundary conditions and the processes that condition the evolution of a specific property. The keywords that can be used in each block are described below. The evolution of density is only function of temperature and salinity using the UNESC polynomial
OUTPUT_HDF	0	integer	OUTPUT_HDF : 1	Checks out if the user pretends to write the transients results of this property (1) or not (0). This is valid only in OUTPUT_TIME option is active.
TIME_SERIE	0	integer	TIME_SERIE : 1	Checks out if the user pretends to write time series of this property (1) or not (0). This is valid only if at least one block with a time serie location is defined (see Table 49).
NAME	*	string	NAME : temperature	Name of a water property
UNITS	*		UNITS : ºC	Units use for a specific water property
DESCRIPTION	*	string	DESCRIPTION : The initial condition is a climatologic one	The property description is a character*132 where the user can store information about the property
DEFAULTVALUE	Temperature=11	real	DEFAULTVALUE : 2	The default value of a specific property

	Salinity=35 Others = 0.			
DEFAULTBOUNDARY	Temperature=11 Salinity=35 Others = 0.	real	DEFAULTBOUNDARY : 1	The default value of a specific water property imposed in the open boundary
OLD	0	integer	OLD : 1	This variable is a logic one is true if the property is old and the user wants to continue the run with results of a previous run (1) or not (0).
INITIALIZATION_METHOD	CONSTANT	string	INITIALIZATION_METHOD : FILE	The user has the follow option to initialize a property: CONSTANT - a constant value is admitted in all the domain BOXES - a constant value for each box is admitted FILE - the property values are read from a ASCII file LAYERS - For each layer a constant value is admitted
BOUNDARY_INITIALIZATION	INTERIOR	string	BOUNDARY_INITIALIZATION : EXTERIOR	Two processes were consider to initialize the boundary values: EXTERIOR - A value exterior to the domain is be imposed. For this option was only considered a constant value. INTERIOR - The boundary are admitted equal to the values given in the same cells during the domain initialization.
FILENAME	*	string	FILENAME : d:\temp.txt	The keyword FILENAME is used to give to the model the file name where the boxes structure is defined when the properties are initialized by boxes (see Table 52). It is also used to give to the model the file name where a property field is defined by file (see Table 46).
BOXESCONCENTRATION	**	n*real	BOXESCONCENTRATION : 2 3.2 4	The n boxes concentration is defined by this keyword. The order is the first value corresponds to the first box and so one.
LAYERSCONCENTRATION		n*real	LAYERSCONCENTRATION : 1.1 2.3 4	A constant value in each layer is considered. The n layers concentration is defined by this keyword. The order is the first value corresponds to the first layer and so one.
ADVECTION_DIFFUSION	1	integer	ADVECTION_DIFFUSION : 0	Check if the user wants to compute the effect of transport in the property evolution (1) or not (0).
DISCHARGES	0	integer	DISCHARGES : 1	Check if the user wants to compute the effect of discharges in the property evolution (1) or not (0).
SURFACE_FLUXES	0	integer	SURFACE_FLUXES : 1	Check if the user wants to compute the effect of surface fluxes in the

				property evolution (1) or not (0).
DATA_ASSIMILATION	0	integer	DATA_ASSIMILATI ON : 1	Check if the user wants to assimilate water properties define in the input data file of Data Assimilation module (1) or not (0)
DT_INTERVAL	DT/2 (see Table 7)	Real (s)	DT_INTERVAL : 600.	Time step evolution of each property.
ADV_DIF_NUM_STABILITY	0	integer	ADV_DIF_NUM_ST ABILITY	Check if the user wants to verify advection-diffusion numerical stability (1) or not (0)
SCHMIDT_NUMBER_H	1	real	SCHMIDT_NUMBER _H : 0.9	Schmidt number for the horizontal. Conversion number between the horizontal turbulent viscosity and the horizontal turbulent Diffusion. If the property is Heat the name of this number is not SCHMIDT but prandtl
SCHMIDT_COEF_V	1	real	SCHMIDT_COEF_V : 0.95	Vertical diffusivity of each property is calculated as SCHMIDT_COEF_V*TU RBULENTDIFFUSIVITY +SCHMIDT_BACKGRO UND_V
SCHMIDT_BACKGROUND_ V	1e-8	real	SCHMIDT_BACKGR OUND_V : 1e-5	See above cell.
ADVECTION_UP_DC	1	real	ADVECTION_UP_D C : 0	The advection scheme implemented is a hybrid one. This coefficient is limited by 0 and 1. 1 - advection computed using a Upwind discretization 0 - advection computed using a center differences discretization
ADVECTION_V_IMP_EXP	0	integer	ADVECTION_V_IM P_EXP : 1	Vertical advection computed using a : 0 - implicit discretization 1 - explicit discretization
DIFFUSION_V_IMP_EXP	0	integer	DIFFUSION_V_IMP_ EXP : 1	Vertical diffusion computed using a : 0 - implicit discretization 1 - explicit discretization
ADVECTION_H_IMP_EXP	0	integer	ADVECTION_H_IM P_EXP : 1	Horizontal advection computed using a : 0 - implicit discretization 1 - explicit discretization
NULLDIF	0	integer	NULLDIF : 1	When the horizontal water flux in a face is zero (no advection) then the horizontal diffusion flux is also zero (1) or not (0).
BOUNDARY_CONDITION	1	integer	BOUNDARY_COND ITION : 8	1 – Evolution due only conservative advection equation. This equation is compute in two directions normal to the boundary and in the vertical direction 2 –Value imposed 3 – Evolution due only to vertical mixing 4 – Null gradient 5 – SubModel 6 - A variant of option 1 plus a radiation equation to estimate the exterior value

				function of the internal variability. This exterior value is important in the inflow case. In option 1 the exterior value in the inflow case is considered equal to the initial boundary value 7 – A hybrid between option 1 and 4. When the flux is outward option 1 is valid when is inward the option 4 is valid 8 – Cyclic boundary
DECAY_TIME	0	Real (s)	DECAY_TIME : 1000.	This option is only valid when BOUNDARY_CONDITION : 1 0. - The exterior value is constant along time and equal to the initial boundary value Infinity - The exterior value in t is equal to the boundary value in t.
LW_EXTINCTION_TYPE	1	Integer	LW_EXTINCTION_TYPE : 1	Only the option constant extinction coefficient is implemented (1).
LW_EXTINCTION_COEF	1/3.	Real (m^{-1})	LW_EXTINCTION_COEF : 1/5	Extinction coefficient for the atmospheric radiation or long wave radiation
LW_PERCENTAGE	0.4	Real (%)	LW_PERCENTAGE : 0.3	Percentage of the total radiation that reaches the surface water is atmospheric radiation
SW_EXTINCTION_TYPE	1	Integer	SW_EXTINCTION_TYPE : 2	The option available are: 1:Constant 2:Parsons Ocean 3:Portela Tagus 4:Valdemar Estuary 5:Parsons+Portela 6:ASCIIFile (see Rosa, 2002)
SW_EXTINCTION_COEF	1/20.	Real (m^{-1})	SW_EXTINCTION_COEF : 1/5	Extinction coefficient for the solar radiation or short wave radiation
SW_PERCENTAGE	0.6	Real (%)	SW_PERCENTAGE : 0.7	Percentage of the total radiation that reaches the surface water is direct solar radiation

*- must be defined when the property is initialized by boxes or by file.

**- must be defined when the property is initialized by boxes.

Table 45 - An example of water properties definition.

```
REFERENCE_DENSITY       : 1026.72546

<beginproperty>
NAME                    : salinity
UNITS                   : psu
DESCRIPTION             : salinity
DEFAULTVALUE            : 36
INITIALIZATION_METHOD   : CONSTANT
ADVECTION_DIFFUSION     : 0
<endproperty>


<beginproperty>
NAME                    : temperature
UNITS                   : ºC
```

```
DESCRIPTION                  : temperature
DEFAULTVALUE                 : 15
INITIALIZATION_METHOD        : LAYERS
!LAYERSCONCENTRATION         : 8 8 8 9 9.25 9.5 9.75 10 12 14
LAYERSCONCENTRATION          : 15 18
!FILENAME                    : D:\Aplica\Mohid2000Testes\AjusteGeostrofico\GlobalData\boxes23layers50x50.dat
BOUNDARY_INITIALIZATION      : INTERIOR
ADVECTION_DIFFUSION          : 1
ADVECTION _V_IMP_EXP         : 0
ADVECTION _H_IMP_EXP         : 1
DIFUSION_V_IMP_EXP           : 0
OLD                          : 0
TIME_SERIE                   : 0
OUTPUT_TIME                  : 0 86400
DISCHARGES                   : 1
SURFACE_FLUXES               : 1
BOUNDARY_CONDITION           : 4
SCHMIDT_COEF_V               : 0
SCHMIDT_BACKGROUND_V         : 0
SCHMIDT_NUMBER_H             : 0
<endproperty>


TIME_SERIE                   : 0
DT_OUTPUT_TIME               : 300

<BeginTimeSerie>
!Figueira da Foz
LOCALIZATION_I               : 175
LOCALIZATION_J               : 147
LOCALIZATION_K               : 9
<EndTimeSerie>
```

Table 46 - File format used to define initial water properties fields variable in space.

Input data file		FILINAME (see Table 44)		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
<ConcentrationBegin> < ConcentrationEnd>	*	block	<ConcentrationBegin> 12 12 13 14.5 13 14.5 < ConcentrationEnd>	The water property value per cell. The reading sequence is the following: do i=ILB,IUB do j=JLB,JUB do k=KLB,KUB Property(i,j,k) enddo enddo enddo

* - If the option FILE is choose to initialize a water property then this file must be built.

Discharges Input

Table 47 – Options available to define a discharge input of mass or momentum in any cell of the domain.

Input data file	DISCHARGES see (Table 44) WATER_DISCHARGES and MOMENTUM_DISCHARGE see Table 13

KEYWORD	DEFAULT	TYPE	EXAMPLE	DESCRIPTION
<begindischarge> <enddischarge>		Block	<begindischarge> <enddischarge>	Block use to define a discharge properties (ex: flow, location)
NAME	*	String	NAME	Keyword used to give name to the discharge .
DESCRIPTION	No description was given	String	DESCRIPTION	Discharge description.
I_CELL	**	Integer	I_CELL : 21	Line where the discharge is locate
J_CELL	**	Integer	J_CELL : 12	Column where the discharge is locate
K_CELL	**	Integer	K_CELL : 3	Layer where the discharge is locate
ALTERNATIVE_LOCATIONS	0	Integer	ALTERNATIVE_LOCATIONS : 1	Searches for alternative locations. The model searchs another location when the discharge point is not a covered point.
DATA_BASE_FILE	***	String	DATA_BASE_FILE	Time serie file to each is possible to associate a discharge property evolution (ex: flow, temperature).
DEFAULT_FLOW_VALUE	0.	real	DEFAULT_FLOW_VALUE : 10.	Default flow value.
FLOW_COLUMN	***	integer	FLOW_COLUMN : 2	If this keyword is defined automatically the flow is considered variable in time. This keyword is used to give to the model the column where the flow is defined in the time serie (see DATA_BASE_FILE).
FLOW_OVER	0	Integer	FLOW_OVER : 1	Check if the user wants to compute a negative discharge function of the water level also known as spill flow (1) or not (0)
WEIR_LENGTH	***	Real (m)	WEIR_LENGTH : 5	Weir Length. Parameter need in the case of the option FLOW_OVER : 1 is active (spill flow).
WEIR_COEF	0.4		WEIR_COEF : 0.5	Weir coefficient. Parameter need in the case of the option FLOW_OVER : 1 is active (spill flow).
CREST_HEIGTH	***	Real (m)	CREST_HEIGTH : 0.1	Crest Height. Parameter need in the case of the option FLOW_OVER : 1 is active (spill flow).
DEFAULT_VELOCITY_VALUE	0. 0.	2*real (m/s)	DEFAULT_VELOCITY_VALUE : 1.2 0.2	Default velocity associated with the discharge. Important to compute momentum fluxes
U_COLUMN	***	integer	U_COLUMN : 3	If this keyword is defined automatically the velocity X is considered variable in time. This keyword is used to give to the model the column where the velocity X is defined in the time serie (see DATA_BASE_FILE).
V_COLUMN	***	integer	V_COLUMN : 4	If this keyword is defined automatically the velocity Y is considered variable in time. This keyword is used

				to give to the model the column where the velocity Y is defined in the time serie (see DATA_BASE_FILE).
<<beginproperty>> <<endproperty>>		Sub-block	<<beginproperty>> <<endproperty>>	Sub-block use to define a specific water property associated with the flow discharge (ex: temperature)
NAME	*	string	NAME : temperature	Keyword written in a sub-block to define a water property name (ex: salinity). If not defined the model stops.
UNITS	*	string	UNITS : ºC	Keyword written in a sub-block to define the units of a water property (ex: psy). If not defined the model stops.
DESCRIPTION	No description was given	String	DESCRIPTION	Water property description.
CONSTANT_CONC	1	integer	CONSTANT_CONC : 0	hecks if the property have constant value (1) or not (0).
DEFAULTVALUE	0.0	real	DEFAULTVALUE : 10.0	Water property default value.
TIME_SERIE_COLUMN	***	integer	TIME_SERIE_COLUMN	If this keyword is defined automatically this specific water property is considered variable in time. This keyword is used to give to the model the column where the water property associated with this sub-block is defined in the time serie (see DATA_BASE_FILE).

*- If this keyword is not defined in side the discharge block or the properties sub-blocks the model stops.

**- If this keyword is not defined in side the discharge block the model stops.

*** - If this keyword is defined a correct value must be given.

Table 48 – An example of a discharge input data file.

```
<begindischarge>
NAME                  : Rio Tejo
DESCRIPTION           : Tagus river characteristics
I_CELL                : 70
J_CELL                : 27
K_CELL                : 1
DISCHARGE_DEPTH       : 2.
DATA_BASE_FILE        : ..\Cenarios_Tejo\Referência_hidrodinâmica_Ano2.dat
FLOW_COLUMN           : 2
TIME_SERIE_COLUMN     : 1
!DEFAULT_FLOW_VALUE   : 140
<<beginproperty>>
NAME                  : temperature
UNITS                 : ºC
DESCRIPTION           : temperature in the tagus river
CONSTANT_CONC         : 0
TIME_SERIE_COLUMN     : 3
<<endproperty>>
<<beginproperty>>
NAME                  : salinity
UNITS                 : psu
DESCRIPTION           : salinity in the tagus river
```

```
CONSTANT_CONC          : 1
!TIME_SERIE_COLUMN     : 5
DEFAULTVALUE           : 0.001
<<endproperty>>

<begindischarge>
NAME                   : Sorraia
DESCRIPTION            : Sorraia's Discharges Characteristics
I_CELL                 : 51
J_CELL                 : 82
K_CELL                 : 1
DISCHARGE_DEPTH        : 2.
DEFAULT_FLOW_VALUE     : 39.5
ALTERNATIVE_LOCATIONS  : 1

<<beginproperty>>
NAME                   : temperature
UNITS                  : ºC
DESCRIPTION            : temperature in the Sorraia river
CONSTANT_CONC          : 1
DEFAULTVALUE           : 14.
<<endproperty>>
<<beginproperty>>
NAME                   : salinity
UNITS                  : psu
DESCRIPTION            : salinity in the Sorraia river
CONSTANT_CONC          : 1
!TIME_SERIE_COLUMN     : 5
DEFAULTVALUE           : 0.001
<<endproperty>>

<enddischarge>
```

Time Series

Input and Output

Output

Table 49 – Options available to define a time serie output.

Input data file		TIME_SERIE see Table 27, Table 35, Table 39, Table 44		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
MAX_BUFFER_SIZE	100000	Real (bytes)	MAX_BUFFER_SIZE : 1e3	The maximum Buffer Size is set here to 0.1Mb (for each property). This lets perform 25000 outputs to the buffer (considering each output of 4 bytes). Basically when the time serie output occupies in memory this value then the information is written to the file and the buffer is set to zero.

DT_OUTPUT_TIME	DT/2 (see Table 7)	Real (s)	DT_OUTPUT_TIME : 600.	Output interval for the time series
LOCALIZATION_I	*		LOCALIZATION_I	
LOCALIZATION_J	*		LOCALIZATION_J	
LOCALIZATION_K	*		LOCALIZATION_K	
<BeginTimeSerie> <EndTimeSerie>		block	<BeginTimeSerie> LOCALIZATION_I : 30 LOCALIZATION_J : 31 LOCALIZATION_K : 1 <EndTimeSerie>	Blcok use to define a time serie output location.
FIRST_OUTPUT_TIME	*	6*real	FIRST_OUTPUT_TIME : 2002 1 1 0 0 10	The date from which the time serie input starts.

*- For each block (<BeginTimeSerie> <EndTimeSerie>) is necessary to define this keyword.

The output time serie file name is defined using the location for example 30_31_1.ext . the file exytension (.ext) depends of the properties being output.

- Hydrodynamic properties : .srh
- Water properties : .srw
- Turbulence properties : .srt
- Surface properties : .srs

The number of columns in each time serie depends also from the type of properties. In all time series the first 7 columns are used to define time. The first column is the time seconds relatively to the first output (SERIE_INITIAL_DATA see Table 50). The other 6 are used to define the date: year, month, day, hour, minutes and seconds. The other columns depend of the type of properties:

- Hydrodynamic properties : 8 – west face velocity, 9-east face velocity, 10-south face velocity, 11 – north face velocity, 12 – bottom face velocity, 13 – top face velocity, 14 – water level, 15 – covered point (1) or not (0)
- Turbulence properties : 8 - eddy vertical viscosity and 9 - diffusivity, 10 - Brunt-Vaisalla and 11 – prandlt frequencies, 12 – surface mixed layer depth, 13 – TKE, 14 – dissipation rate of TKE, 15 - mixing length, 16 – Production, 17 – Buoyancy.
- Surface properties : variable
- Water properties : variable

For the case of surface and water properties the number of columns depend of the properties that the user wants to output in a time serie (see Table 27 and Table 44).

Table 50 – An example of a time serie output of surface properties.

```
Time Serie Results File
LOCALIZATION_I      : 30
LOCALIZATION_J      : 31
```

```
LOCALIZATION_K         : 10
 SERIE_INITIAL_DATA     : 1994.  7.  1.  0.  0.  0.
TIME_UNITS             : SECONDS
    Seconds   YY  MM  DD  HH  MM       SS   wind_stress_X       wind_stress_Y        solar_radiation
<BeginTimeSerie>
       30.00 1994   7   1   0   0  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
       90.00 1994   7   1   0   1  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
      150.00 1994   7   1   0   2  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
      210.00 1994   7   1   0   3  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
      270.00 1994   7   1   0   4  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
      330.00 1994   7   1   0   5  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
      390.00 1994   7   1   0   6  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
      450.00 1994   7   1   0   7  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
      510.00 1994   7   1   0   8  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
      570.00 1994   7   1   0   9  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
      630.00 1994   7   1   0  10  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
      690.00 1994   7   1   0  11  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
      750.00 1994   7   1   0  12  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
      810.00 1994   7   1   0  13  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
      870.00 1994   7   1   0  14  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
      930.00 1994   7   1   0  15  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
      990.00 1994   7   1   0  16  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
   172770.  1994   7   2  23 59  30.0000   0.000000000000E+000 -0.100000001490E+000  0.000000000000E+000
<EndTimeSerie>


<BeginResidual>
   172800.00 1994   7   3   0   0   0.0000   0.000000000000E+000 -0.100000001490E+000  0.333430816650E+003
<EndResidual>
```

Input

In the case of a time serie input the time is defined only by the first column. A initial date is defined (SERIE_INITIAL_DATA see Table 51) and to this date is add the value of the first columns. The untis of theses values can bedefined by the keyword TIME_UNITS (see Table 51).

Table 51 – Options available to define a time serie input.

Input data file		DATA_COLUMN_* see Table 27 COLUMN_* see Table 47		
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
TIME_UNITS	SECONDS	string	TIME_UNITS : DAYS	The options available are: • SECONDS • MINUTES • HOURS • DAYS • MONTHS
SERIE_INITIAL_DATA	*	6*real	SERIE_INITIAL_DATA : 2002 1 1 0 0 0	Time serie initial date.
TIME_CYCLE	0	integer	TIME_CYCLE : 1	Check if the user wants a time serie cyclic (1) or not (0). In the afirmative case the SERIE_INITIAL_DATA is not used. For example if the TIME_UNITS : MONTHS then is possible to define the follow time serie. <BeginTimeSerie>

				1 2 6 3 12 4 <EndTimeSerie> Now if any model asks for the value in column 2 in a specific date the model the column 2 to the date required. In this case if the a value is ask for the month 4 (March) a linear interpolation is made between the months 1 (Jan.) and 6 (June).

*- If the keyword SERIE_INITIAL_DATA is defined then a correct file name must be given.

Boxes definition

The boxes are used to associate values to areas. For doing that is necessary to now the box numeration system. The first layer (bottom layer) of the first polygon is box number one, the second layer of the same polygon is second box. If the first polygon only has to layers it means that the third box in the first layer of the second polygon and so one.

Table 52 – Options available to define boxes.

Input data file			DEADZONE_FILE see Table 17 FILENAME_PROP see Table 22 FILENAME see Table 27, Table 44 RUGOSITY_BOX see Table 31	
KEYWORD	**DEFAULT**	**TYPE**	**EXAMPLE**	**DESCRIPTION**
HMIN_BOX	MINIMUM DEPTH (see Table 5)	real	HMIN_BOX : 0.2	Minimum water column thickness above which the cells can be members of boxes.
CALC_EXTERNAL_FLUXES	0	integer	CALC_EXTERNAL_FLUXES : 1	Checks if the user wants to compute fluxes between the areas defined (boxes) and their exterior.
TYPE	1	integer	TYPE : 2	The options available are: 1 - grid coordinates 2 - metric ccordinates (Origin is the grid left corner 0,0) 3 - militar coordinates
<beginpolygon> <endpolygon>		block	<beginpolygon> <<beginvertix>> 12 14 14 23 15 34 <<endvertix>> <<beginverticallayer>>	This block is used to define one or more boxes associated with a horizontal polygon. The horizontal geometry of the polygon is defined with the sub-block

			1 12 13 20 <<endverticallayer>> <endpolygon>	<<beginvertix>>, <<endvertix>>. The vertical division in several blocks is defined with the sub-blocks <<beginverticallayer>>, <<endverticallayer>>.
<<beginvertix>> <<endvertix>>		Sub-block	<<beginvertix>> 12 14 14 23 15 34 <<endvertix>>	The horizontal polygons are defiend by pares of position values X, Y. The polygon can be defined clockwise or anti-clockwise.
<<beginverticallayer>> <<endverticallayer>>		Sub-block	<<beginverticallayer>> 1 12 13 20 <<endverticallayer>>	This is used to devide the horizontal polygon in several slices. Is used pares of layers values King, Ksup. The layers limits are considered to belong the box.
HLIMI	-1e16 1e16	2*real	HLIMI : 1 100	Depth limits outside of which the cells are not consider members of the box

*-

Table 53– An example of boxes definition file.

```
HMIN_BOX              : 0.100
CALC_EXTERNAL_FLUXES    : 0
TYPE                  : 1

 <beginpolygon>
 <<beginvertix>>
        30        115
        95        115
        96         23
        64         23
        48         70
        29        104
 <<endvertix>>
  <endpolygon>
```

Bibliography

Abbot M.B., Damsgaardand A., Rodenhuis G.S., System 21, Jupiter, a design system for two-dimensional nearly-horizontal flows, J. Hyd. Res. 1 (1973) 1-28.

Blumberg, A.F. and L.H. Kantha, 1985. Open boundary condition for circulation models. J. of Hydraulic Engineering, ASCE, 111, 237-2555.

Flather, R.A., 1976: A tidal model of the northwest European continental shelf. Mem. Soc. R. Sci. Liege, Ser. 6(10), 141-164.

Large, W.G. and S. Pond, 1981, Open ocean momentum flux measurements in moderate to strong winds, J. Phys. Ocean., 11:324-336.

Leendertse J., 1967. Aspects of a computational model for long water wave propagation, Memorandum RH-5299-RR Rand Corporation, Santa Monica, 1967.

Marchesiello, P., J. C. McWilliams e A. Shchepetkin (2001): Open boundary conditions for long-term integration of regional oceanic models. Ocean Modelling 3, 1-20, 2001.

Martinsen, Eivind A. e Harald Engedahl: Implementation and testing of a lateral boundary scheme as an open boundary condition in a barotropic ocean model, Coastal Engineering, 11, 603-627, 1987.

Nihoul, J. C. J. (1984) - A three-dimensional general marine circulation model in a remote sensing perspective. In Annales Geophysicae, 2, 4, 433-442

Oey, L. e P. Chen (1992). A Model Simulation of Circulation in the Northeast Atlantic Shelves and Seas. J. Geophys. Res., 97, 20,087-20,115.

Orlanski, I., A simple boundary condition for unbonded hyperbolic flows, J. Comput. Phys., 21, 251-269, 1976.

Smagorinsky, J (1963). General Circulation Experiment with the Primitive Equations, Monthly Weather Review, 91, No. 3, pp 99-164, 1963.

Trancoso, A. R. (2002). Modelling Macroalgae In Estuaries. Trabalho Final de Curso da Licenciatura em Engenharia do Ambiente, Instituto Superior Técnico, Universidade Técnica de Lisboa, 2002. http://194.65.82.105/dataserver/products/Thesis/TFC_RosaTrancoso.pdf