FlexSem User Guide

Janus Larsen

January 2025

Contents

1 Introduction

The modeling framework FlexSem is a fast, flexible and user friendly tool specifically targeted towards scientific and management challenges of the complex biogeochemical processes in coastal zone ecosystems. Please refer to the web site https://marweb.bios.au.dk/FlexSem/ for publications, examples of use, downloads and more.

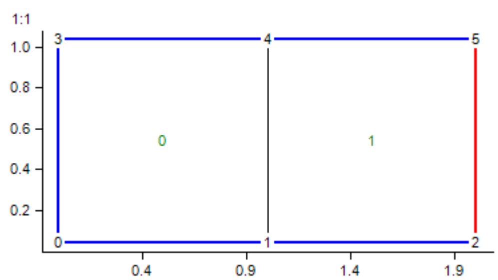
Questions? please contact Janus Larsen janus@bios.au.dk

2 Mesh

The computational mesh is defined in a tab seperated text file. The first line gives the number of nodes, the number of elements and optionally the number of Voronoi points (always equal to number of elements). All nodes and elements are indexed starting with zero.
The following [number of nodes] lines gives the x,y coordinates of the nodes and in the third column optionally the node type. Supported node types are: 0=internal, 99=land bnd, (>0<99)=open boundary.
Then follows one line per element, each line giving the node number of the nodes that makes up the element (polygon) in counter clockwise order (the connectivity).
If the number of voronoi points are given in the header, coordinates of voronoi points for each element is given in after the element connectivity. Notice that all vononoi points must be inside the corresponding element and the connection lines between the voronoi points perpendicular to the element faces in order for the solution to be numerically accurate.

Mesh example

```
6 2
0 0
1 0
2 0 1
0 1
1 1
2 1 1
0 1 4 3
1 2 5 4
```



3 Generic definitions

Generic ways of inputting setup information is defined in this section. These definitions apply to multiple tags in the setup files. See the tag descriptions in the module sections for information about where the generic defintions can be used. All items etc. are zero index in FlexSem.

3.1 Initial values

Initial values can be defined as a numeric constant or by refering to a file that contains an initial field.

`<InitValue>` Numeric constant value.

`<InitFilename>` Path and name of file with initial values. Alternative to numerical value.

`<InitFileFormat>` File format. "TEXT0D", "TEXT1D" or "EMB". Mandatory for input from file.

<InitItemNo> Item number of item with initial values. Mandatory for input from file.

Numeric example

```
<InitValue>0.01</InitValue>
```

File example

```
<InitValue>
    <InitFilename>sediInit.emb</InitFilename>
    <InitFileFormat>EMB</InitFileFormat>
    <InitItemNo>0</InitItemNo>
</InitValue>
```

3.2 Selecting cells

Selecting specific cells in a 3D model can be done by giving a:

- layer number
- comma separated list of layers numbers
- path and name of file with layer numbers
- comma separated list of cell numbers
- path and name of file with cell numbers
- coordinates of a polygon

<Layers> Define layer(s). ”surface”, ”bottom”, integer value or CSV list of integer values. <Layers> or <Cells> mandatory.
<Filename> Path and name of file with layer numbers.
<Cells> Define cell(s). Integer value or CSV list of integer values. Alternative to <Layers>.
<Filename> Path and name of file with cells numbers.
<Polygon> Define cells by a polygon. Alternative to ¡Filename¿.
<XCoor> CSV of polygon X coordinates.
<YCoor> CSV of polygon Y coordinates.

3.3 Passing constant values

Passing values constant in time to the model can be done by giving a:

- path and name of file with the values
- numeric
- comma separated list of numerics
- symbol of a constant (or a variable)

File example

```
<ConstantValues>
    <Filename>MyfileWithConstantValues.txt</Filename>
</ConstantValues>
```

Numeric example

```
<ConstantValues>3.14</ConstantValues>
```

Comma separated example

```
<ConstantValues>2,1.4142,6371</ConstantValues>
```

Symbol example

```
<ConstantValues>MySymbol</ConstantValues>
```

3.4 Defining a time interval

A time interval can be defined by a decimal number followed by a unit. No unit is seconds. Supported units are:

- 'y' : year
- 'q' : quater
- 'm' : month
- 'w' :week
- 'd' : day
- 'h' : hour
- 'M' : minute

3.5 Equation semantics

Equation definitions can be writting follwoing the normal math rules using symbols or numbers, e.g. <Y = 3.14*(x+b)> If statements must be written as if(criteria,value if true, value if false). In the criteria fortran notation must be used, i.e.

- '<' : .lt.
- '≤' : .le.
- '>' : .gt
- '≥' : .ge.

e.g. <theta_nn=if(nn.gt.0,1,0)>
The following other functions are supported: exp, log, ln, abs, cos, sin, tan, tanh, max, min
The following other operators are supported: +, -, *, \

Equation example

```
<Equation>
    <Definition>salt = if((x.lt.19).and.(d.gt.2),35,10)</Definition>
</Equation>
<Equation>
    <Definition>flow_P_A = p_AmT*f*volmus^(2/3)</Definition>
</Equation>
```

4 Modules

This sections contains a description of all FlexSem modules. All modules can run indpendently/are optional, unless otherwise stated. All date times must be given in the format "yyyy-MM-dd HH:mm:dd".

4.1 General

The purpose of the general section is to define the computationel domain in time and space. Mandatory.

<General> Definiton of model. Mandatory.
<Mesh> Mandatory.
<Filename> Path to mesh file. Mandatory.
<Format> Format of mesh file. "EMT" (text) or "EMB" (binary). Mandatory.
<Projection> Projection string. "UTM32" or "LONLAT". Optional, default is "UTM32".
<Bathymetry> Bathymetry. Mandatory.

<Value> Bathymetry values. Numeric (constant bathymetry), comma separated values or filename with one bathymetry value in each line.
<Filename> Bathymetry values. Filename with one bathymetry value in each line. Alternative to <Value>.
<DT> The timestep in seconds. Mandatory.
<NoTimesteps> Number of time steps. Default 0.
<NoLayers> Number of layers. Ignored if the layer thickness is given as comma seperated values.
<LayerThickness> Numeric. Sets all layers to given thickness. Mandatory.
<LayerThicknessCSV> Comma separated list of layer thicknesses (NB from bottom up). If LayerType=="SIGMA", the sum of the layerthicnesses must equal 1. Alternative to <LayerThickness>
<OffsetTopDepth> The z axis origin. Optional. Default is 0.
<LayerType> The Layer type. "SIGMA" (default) or "Z".
<StartDT> Start date and time. Default "1970-01-01 00:00:00".
<CheckBlowUp> Indicates if the engine should check all varaibles in each time step for blow up values. "True" or "False (default).
<LogLevel> Control the logging detail level. A number between 0 and 10. Optional. Default is 0 (least logging).
<RestartFilename> Filename of .emb file containing restart information (outputted by <Output><Format>RESTART</Format></Output>
<RestartTimestep> Timestep in <RestartFile> to start from.

Optionally the entire <General> section can be replaced by a reference to a file from which the general setup will be read. This will typically be a binary output file (.emb or .nc): <SetupFile>Myfile.emb</SetupFile>

Example

```
<General>
    <Mesh>
        <Filename>..\OneBox.mesh</Filename>
        <Format>EMT</Format>
    </Mesh>
    <Bathymetry>
        <Filename>..\OneBox.bathy</Filename>
    </Bathymetry>
    <NoLayers>1</NoLayers>
    <LayerThicknessCSV>1</LayerThicknessCSV>
    <LayerType>sigma</LayerType>
    <StartDT>2004-01-01 00:00:00</StartDT>
    <DT>3600</DT>
    <NoTimesteps>10</NoTimesteps>
</General>
```

4.2 SpecialValues

In this section symbols for model generated variables can be defined (each creates a new variable). These symbols can then be used as any other symbol, e.g. in an equation on in an output.

<SpecialValues>
<NoTimesteps> Symbol for the number of timesteps (=<NoTimesteps> in <General>).
<TimestepLengthSymbol> Symbol for the time step length (=<DT> in <General>).
<PISymbol> Symbol for Pi (=4*atan(1)=3.141593).
<TimestepNumberSymbol> Symbol for the time step number.
<ElementAreaSymbol> Symbol for the area all elements (2D).

<CellDepthSymbol> Symbol for the depth of all cells (3D).
<CellVolumeSymbol> Symbol for the volume of all cells (3D).
<Bathymetry> Symbol for the bathymetry values (2D).
<BottomLayerThickness> Symbol for thethickness of the bottom layer cells (2D).
<BottomLayerMask3D> Symbol for a mask that is 1 in all bottom cells, 0 in all others (3D).
<Distance2Cell> Distance to given cell.
<Symbol> Symbol for distance (3D). Mandatory.
<CellNumber> Integer cell number to the cell from which the distances are calculated. Mandatory.
<ElementCoordinates> Element coordiantes (cell center).
<XCoorSymbol> Symbol for the element X coordinate (2D). Mandatory.
<YCoorSymbol> Symbol for the element Y coordinate (2D). Mandatory.
<SedimentCellVolumeSymbol> Symbol for the volume of all cells in the sediment model.

Example

```
<SpecialValues>
    <TimestepLengthSymbol>myDT</TimestepLengthSymbol>
    <PISymbol>myPI</PISymbol>
    <TimestepNumberSymbol>myTS</TimestepNumberSymbol>
    <CellDepthSymbol>cd</CellDepthSymbol>
    <CellVolumeSymbol>cv</CellVolumeSymbol>
    <Distance2Cell>
        <Symbol>dist2cell0</Symbol>
        <CellNumber>0</CellNumber>
    </Distance2Cell>
    <ElementCoordinates>
        <XCoorSymbol>eXCoor</XCoorSymbol>
        <YCoorSymbol>eYCoor</YCoorSymbol>
    </ElementCoordinates>
</SpecialValues>
```

4.3 Constants

Definition of constants

<Constants>
<Constant> Optionally multiple.
<Symbol> Symbol of the constant (creates new 0D variable). Mandatory.
<Value> Numerical value of the constant. Mandatory.
<Mask> Setting up a constant mask (¡Value¿ in given cells, zero elsewhere). Optional.
<Dim> Dimension of the mask [0,1,2 or 3]. Default is 3.
<Cells> or <Layers> Define the cells that will get the value gien in <Value>. See section 3.2. Alternative to Cells.
<Dim> 2D or 3D Constant [2 or 3]. Optional alternative to <Mask>.

Example

```
<Constants>
    <Constant>
        <Symbol>gam</Symbol>
        <Value>0.8</Value>
    </Constant>
    <Constant>
        <Symbol>PMask</Symbol>
        <Value>1</Value>
```

```
        <Mask>
            <Dim>3</Dim>
            <Cells>
                <Polygon>
                    <XCoor>398.7,651.5,652.3,398.1</XCoor>
                    <YCoor>853.0,853.6,646.6,647.2</YCoor>
                </Polygon>
            </Cells>
        </Mask>
    </Constant>
    <Constant><Symbol>res</Symbol><Value>0</Value><Dim>2</Dim></Constant>
</Constants>
```

4.4 Inputs

Read data from files into variables. FlexSem will automatically interpolate data from "TEXT0D" and "TEXT1D" formatted files in time and space.

<Inputs>
  <Input> Optionally multiple.
    <Filename> Path and name of file to read data from. Mandatory.
    <Format> File format. "TEXT0D", "TEXT1D", "EMB" or "NC", Mandatory.
    <Item> . Defines the item to read. If only one item in file, optionally leave out this tag.
      <Symbol> Symbol to read data into (creates new variable). Mandatory.
      <ItemNumber> Item number to read (zero based). Mandatory.
      <Factor> Multiply input by this factor. Optional.

Example

```
<Inputs>
    <Input>
        <Filename>kattegatBndT_2.emb</Filename>
        <Format>EMB</Format>
        <ItemNumber>0</ItemNumber>
        <Symbol>bnd2t</Symbol>
    </Input>
    <Input>
        <Filename>..\data\HBMriver4_FlexSem.txt</Filename>
        <Format>TEXT0D</Format>
        <Item>
            <ItemNumber>0</ItemNumber>
            <Symbol>src4aa</Symbol>
        </Item>
        <Item>
            <ItemNumber>4</ItemNumber>
            <Symbol>src4oo</Symbol>
        </Item>
    </Input>
</Inputs>
```

4.5 HD

The hydrodynamic (HD) module of FlexSem implements a 3D semi-implicit, finite difference-finite volume, hydrostatic and nonhydrostatic solution to the Navier-Stokes equations on an unstructured computational mesh as outlined by Casulli and Zanolli 2002

Horizontal and vertical advection and diffusion of momentum is discretized using an eulerian second order Adam-Bashford approach following Finger et al. 2006

The model includes a Smagorinsky formulation (Smagorinsky 1963) of the turbulent viscosity, a semi-implicit advection diffusion scheme (Casulli and Zanolli 2002). Options to include clamped open boundary forcings, surface heat budget, Coriolis forces, sources as well as bottom, vertical wall and surface drag are also included.

The implementation is documented in Larsen et al. A versatile marine modelling tool applied to arctic, temperate and tropical waters.. See more at http://marweb.bios.au.dk/flexsem

`<HD>`

- `<WetDry>` Enable wetting and drying. Optional. Default False.
- `<ADType>` Advection-Diffusion scheme. 2=implicit advection-diffusion scheme following Casulli (2002), -99=No AD. Optional. Default 2.
- `<ImplicitnessFactor>` Impliciness factor. 0.5 ¡= factor ¡=1. Default 1.
- `<RampupSeconds>` Number of seconds to ramp up the HD model.
- `<FaceInterpolationRule>` Type of face heigh interpolation. 0=max, 1=min, 2=mean, 3=upwind. Default if 3.
- `<ViscosityInterpolationType>` Type of viscosity interpolation. 0=face, 1=elem mean. Default is 1.
- `<NumberADSubcycles>` Number of subcycles in each timestep for the AD scheme.
- `<Decoupled>` Decoupled hydrodynamics
- `<InputFilename>` Path and name of the file with velocities and fresh water fluxes for the decoupling. Mandatory if Decopuled.
- `<InitialTemperatureValue>` Initial temperature. See section 3.1
- `<InitialSalinityValue>` Initial salinity. See section 3.1
- `<NonHydrostatic>` Add the nonhydrostatic solution (corrector) to the hydrostatic (predictor) HD model.
- `<CGTolerance>` Tolerance for the Conjugate Gradient numerical solution. Default 1e-10.
- `<InitialWaterLevel>` Initial water level. See section 3.1
- `<PrescripeDensitySymbol>` Symbol of density. Optional, if not given density is calculated from temperature and salinity.
- `<DensityCoefficient>` Density coefficient. Default 1.
- `<DensityCalculationType>` Density calculation type. 0: from S,T and P (mason.gmu.edu/ bklinger/seawater.pdf), 1: DensityCoefficient*S, 2: DensityCoefficient*T.
- `<AirPressure>` Air pressure. Default not enabled.
- `<WindSpeedU>` Wind speed u component for the surface drag.
- `<WindSpeedV>` Wind speed v component for the surface drag.
- `<IceCover>` Ice cover symbol. Turn off surface drag when above threshold. Optional.
- `<IceCoverThreshold>` Ice cover threshold value. Optional. Default 0.5.
- `<BottomDrag>` Bottom drag coefficient. Default 0.005
- `<AtmosphericDrag>` Atmospheric drag coefficient. Default 0.
- `<AtmosphericDensity>` Default 1.2 $[kg/m^3]$.
- `<VerticalWallDrag>` Vertical wall drag coefficient. Default 0.005
- `<CoriolisF>` Prescribe constant coriolis frequency. If not given it will be calculated as 2*Omega*sin(latitude) for each face.
- `<SurfaceRoughnessLength>` Surface roughness length.
- `<BottomRoughnessLength>` Bottom roughness length.
- `<Turbulence>`
  - `<HorisontalType>` Horisontal viscosity model type. <0: no turbulent viscosity (default), 0: Smagorinsky.
  - `<VerticalType>` Vertical viscosity model type. <0: no turbulent viscosity (default), 0: k-epsilon.

<Advect> Boolean. If true, the vertical turbulence model state parameters are advected. Default false.
<VerticalBoundaryType> Boundary type for vertical viscosity model. 0: prescribed (Dirichlet), 1: flux (Neumann). Default is 1.
<HorisontalDiffusivity> Horisontal diffusivity. Optional. Default 0.
<VerticalDiffusivity> Vertical diffusivity. Optional. Default 0.
<VerticalLaminarViscosity> Optional. Default 1e-4.
<HorisontalLaminarViscosity> Optional. Default 0.1.
<SmagorinskyConstant> Constant for the Smagorinsky turbulent viscosity. default 0.01.
<VerticalViscositySymbol> Symbol for the vertical viscosity.
<VerticalDiffusivitySymbol> Symbol for the vertical diffusivity.

<PhysicalBoundaries>
<PhysicalBoundary>
<Number> Mandatory. Boundary number type. Must match number in .mesh file.
<Type> Physical boundary type. 'open': open boundary (inflow), 'waterlevel': specify h only, 'velocity': specify (u,v) only, 'waterlevelandvelocity': specify both h and (u,v).
<h> Boundary water level.
<u> Boundary velocity u component.
<v> Boundary velocity v component.

<CalculateCellCenteredVelocities> Calculate momentum conserving cell centered velocities [Perot 2000]. Adds variables 'u' and 'v'.
<USymbol> Optionally provide an alternative symbol for the cell centered east-west velocity component.
<VSymbol> Optionally provide an alternative symbol for the cell centered north-south velocity component.

<BottomShearStress> Calculate bottom shear stress. Adds 2D variable 'TauB', 'u' and 'v' (cell centered velocities). $TauB = u_*^2 \rho$, where ρ is the reference density and $u_* = \frac{u(z)}{k} ln(\frac{z}{z_0})$, where k is von karman constant (=0.41) and z_0 is the bottom roughness length.
<Symbol> Optionally provide an alternative symbol for the 2D bottom shear stress variable.
<ReferenceDensity> Optionally provide reference density. Default is 1025 kg/m3.

Example 1

```
<HD>
    <NonHydrostatic>false</NonHydrostatic>
    <ImplicitAdvection>true</ImplicitAdvection>
    <ImplicitnessFactor>0.55</ImplicitnessFactor>
    <RampupSeconds>720</RampupSeconds>
    <NumberADSubcycles>1</NumberADSubcycles>
    <HorisontalDiffusivity>10</HorisontalDiffusivity>
    <VerticalDiffusivity>0</VerticalDiffusivity>
    <CGTolerance>1e-5</CGTolerance>
    <VerticalLaminarViscosity>100</VerticalLaminarViscosity>
    <HorisontalLaminarViscosity>0.02</HorisontalLaminarViscosity>
    <ViscosityModelType>0</ViscosityModelType>
    <SmagorinskyConstant>1e-8</SmagorinskyConstant>
    <DensityCalculationType>0</DensityCalculationType>
    <BottomDrag>0.5</BottomDrag>
    <VerticalWallDrag>0.0</VerticalWallDrag>
    <CoriolisF>0.0001220354</CoriolisF> //2*Omega*sin(phi) = 2 * 7.2921e-5 *
         sin(56.8/180*pi) = 0.0001220354
    <CalculateCellCenteredVelocities>true</CalculateCellCenteredVelocities>
    <InitialSalinityValue><InitFilename>forcing\initts.emb</InitFilename><
        InitFileFormat>EMB</InitFileFormat><InitItemNo>0</InitItemNo></
        InitialSalinityValue>
```

```
    <InitialTemperatureValue><InitFilename>forcing\initts.emb</InitFilename>
        <InitFileFormat>EMB</InitFileFormat><InitItemNo>1</InitItemNo></
        InitialTemperatureValue>
    <InitialWaterLevel><InitFilename>forcing\inith.emb</InitFilename><
        InitFileFormat>EMB</InitFileFormat><InitItemNo>0</InitItemNo></
        InitialWaterLevel>

    <PhysicalBoundaries>
        <PhysicalBoundary>
            <Number>1</Number>
            <!--<Type>WaterLevel</Type>-->
            <!--<Type>Velocity</Type>-->
            <Type>WaterLevelandVelocity</Type>
            <u>bnd1u</u>
            <v>bnd1v</v>
            <w>0</w>
            <h>bnd1h</h>
        </PhysicalBoundary>
        <PhysicalBoundary>
            <Number>2</Number>
            <Type>WaterLevelandVelocity</Type>
            <u>bnd2u</u>
            <v>bnd2v</v>
            <w>0</w>
            <h>bnd2h</h>
        </PhysicalBoundary>
        <PhysicalBoundary>
            <Number>3</Number>
            <Type>WaterLevelandVelocity</Type>
            <u>bnd3u</u>
            <v>bnd3v</v>
            <w>0</w>
            <h>bnd3h</h>
        </PhysicalBoundary>
    </PhysicalBoundaries>

    <WindU>metU</WindU>
    <WindV>metV</WindV>
    <AtmosphericDensity>1</AtmosphericDensity>
    <AtmosphericDrag>1e-8</AtmosphericDrag>

</HD>
```

Example 2

```
<HD>
    <Decoupled>
        <InputFilename>FSsetupVel.emb</InputFilename>
        <NumberSubcycles>2</NumberSubcycles>
    </Decoupled>
    <HorisontalDiffusivity>0</HorisontalDiffusivity>
    <VerticalDiffusivity>0</VerticalDiffusivity>
    <CalculateCellCenteredVelocities>true</CalculateCellCenteredVelocities>
</HD>
```

4.6 HDLite

This module contains simplified formulations of estuarine and fjord dynamics including advection, horizontal and vertical mixing. It employs a time-dependent, combined boxmodel and 3-D model approach. Water mass exchange in the HDLite module is realized in the form of volume fluxes between the mesh cells, formulated as an exchange volume per time unit scaled by the time step of the model. Fluxes between mesh cells are automatically calculated from the computational mesh geometry. See Janus Larsen, Christian Mohn, Karen Timmermann, A novel model approach to bridge the gap between box models and classic 3D models in estuarine systems, Ecological Modelling, Volume 266, 24 September 2013, Pages 19-29, ISSN 0304-3800, http://dx.doi.org/10.1016/j.ecolmodel.2013.06.030. Alternative to <HD>

<HDLite>
- <CalcDens> If true, adds variable 'dens' and calculates density from salinity and temperature in each timestep. "True" or "False".
- <InitialSaltValue> Initial value for salinity. See section 3.1
- <Temperature>
  - <InitialValue> Initial value for water temperature. See section 3.1
  - <AirTemperature> Numerical value or symbol for air temperature.
- <HorisontalMixing> Numeric for constant horisontal mixing. Default: 0.
  - <Value> Horisontal mixing coefficient. Alternative to constant horisontal mixing.
  - <ConstantHM> If "True" do not scale horisontal mixing with cell volume. This makes the horisontal mixing in each time step propotional to the area on the cell interface overlap and there by more 'current like' (small overlap -¿ small exchange). If "false" the mixing scales with the total volume of the cell. Default: "False".
  - <WindSpeedCoef> Wind dependent mixing coefficent component.
  - <WindSpeed> Numeric or symbol for wind speed. Mandatory if <WindSpeedCoef>.
  - <MaxMixDirection> Wind direction of maximum mixing coefficent.
  - <WindDirection> Numeric or symbol for wind direction. Mandatory if <MaxMixDirection>.
- <Wind> Wind speed dependent vertical mixing. Default: No vertical mixing.
  - <Speed> Numeric or symbol for wind speed. Mandatory.
  - <AirDensity> Air density. Default: 1.293 kg/m3.
  - <CoriolisParameter> Coriolis parameter. Default: 0.0001.
  - <WaterDensity> Density of water. Default: 1000 kg/m3.
  - <AttenuationDepth> Attenuation depth of veritical mixing. Default: 10 m.

Example

```
<HDLite>
    <InitialSaltValue>
        <InitFilename>..\initSalt.emb</InitFilename>
        <InitItemNo>0</InitItemNo>
        <InitFileFormat>EMB</InitFileFormat>
    </InitialSaltValue>
    <Temperature>
        <InitialValue>0</InitialValue>
        <AirTemperature>airtemp</AirTemperature>
    </Temperature>
    <HorisontalMixing>
        <Value>2e-03</Value>
      <WindSpeed>ws</WindSpeed>
        <WindSpeedCoef>0</WindSpeedCoef>
        <WindDirection>wd</WindDirection>
        <MaxMixDirection>-15</MaxMixDirection>
    </HorisontalMixing>
```

```
    <Wind>
        <DragCoef>1e-07</DragCoef>
        <AirDensity>1.293</AirDensity>
        <Speed>ws</Speed>
        <CoriolisParameter>0.0001</CoriolisParameter>
        <WaterDensity>1000</WaterDensity>
        <AttenuationDepth>6</AttenuationDepth>
    </Wind>
</HDLite>
```

4.7 FlexSem

3D pelagic equation solver.

<FlexSem>
- <DisableGroupCSV> Comma separated list of group names to disable. Names must match "Group" attribute in <Variable> or <Equation>
- <Variables>
  - <Variable> Optionally multiple. Optionally group attribute: <Equation Group="MyGroup">.
    - <Symbol> Variable symbol. Mandatory.
    - <Advect> Indicates if the variable should be advected (by HD or HDLite). "True" or "False" (default).
    - <NotNegative> Indicates if FlexSem should truncate negative values to 0. "True" or "False" (default).
    - <InitValue> Initial value for the variable. See section 3.1
- <Equations>
  - <Equation> Optionally multiple. Optionally group attribute: <Variable Group="MyGroup">.
    - <Definition> Equation definition. Mandatory. See Equation semantics
    - <Description> Equation description. Does not affect the model.
  - <MappedOperation> Move a fraction of one variable in specific cells to another variable in optionally an other set of specific cells. Used to model a patchy process moving stuff from the pelagic to the sediment (or vica versa). Optionally multiple.
    - <From> Defines what to map from. Mandatory.
      - <Variable> Symbol of the variable to map from. Mandatory.
      - <Fraction> The fraction of the variable to move assuming that the variable represents a concentration. Must be same length as the number of given cells. See section 3.3 . Mandatory.
      - <Cells> Cells to map from. See section 3.2. <Layers> or <Cells> mandatory.
      - <Layers> Give the cells to map from as layers. See section 3.2. Alternative to Cells.
    - <To> Defines what to map to. Mandatory.
      - <Variable> Symbol of the variable to map to. Mandatory.
      - <Cells> Cells to map to. Length must equal <From> length. See section 3.2. <Layers> or <Cells> mandatory.
      - <Layers> Give the cells to map to as layers. See section 3.2. Alternative to Cells.
- <DumpSystemAsRCode> Write a file with the equation system as R code.
  - <CellNo> Cell number to use in equation system. Mandatory.
  - <FileName> Path and name of file to output code to. Mandatory.

Example

```
<DisableGroupCSV>B</DisableGroupCSV>
<FlexSem>
    <Variables>
        <Variable>
            <Symbol>A</Symbol>
```

```
            <Description>My A variable</Description>
            <InitValue>1</InitValue>
        </Variable>
        <Variable Group="B">
            <Symbol>B</Symbol>
            <Description>My z variable</Description>
            <InitValue>1</InitValue>
        </Variable>
        <Variable>
            <Symbol>C</Symbol>
            <Description>My C variable</Description>
            <InitValue>0</InitValue>
        </Variable>
    </Variables>
    <Equations>
        <Equation Group="B">
            <Definition>B=A*C</Definition>
            <Description>Test</Description>
        </Equation>
        <Equation>
            <MappedOperation>
                <From>
                    <Variable>A</Variable>
                    <Layers>Bottom</Layers>
                    <Fraction>0.1</Fraction>
                </From>
                <To>
                    <Variable>C</Variable>
                    <Cells>Bottom</Cells>
                </To>
            </MappedOperation>
        </Equation>
    </Equations>
</FlexSem>
```

4.8 EcoSediment

Benthic equation solver (sediment model) including multiple layers (3D), sedimentation, diffusion. The horisontal computation mesh is the same as the one defined in the <General> section 4.1.

<EcoSediment>
- <General> Defines the
  - <NoLayers> Number of layers. Mandatory.
  - <LayerThicknessCSV> Comma separated list of layer thicknesses (NB from bottom up). If LayerType==”SIGMA”, the sum of the layerthicnesses must equal 1. Mandatory.
  - <Bathymetry> Bathymetry. Mandatory.
    - <Value> Bathymetry values. Numeric (constant bathymetry), comma separated values or filename with one bathymetry value in each line.
    - <Filename> Bathymetry values. Filename with one bathymetry value in each line. Alternative to <Value>.
  - <Run> Indicates if the sediment module modifies the valures in each time step (model on/off). ”False” or ”True” (default).
- <PelagicVariables> Use values from bottom layer of a pelagic variable in all layers of the sediment. E.g. Temperature.
  - <Variable> Optionally multiple.
    - <Symbol> Symbol of new benthic variable. Mandatory.

<PelagicVariable> Symbol of pelagic variable to copy values from. Mandatory.
<Diffusion> Diffuse transport between bottom pelagic layer and surface benthic layer.
<DiffusionLength> The diffusion length to use. Default 0.1.
<DiffusionCoef> Default diffusion coefficient. Default 0.
<Variable> Optionally multiple.
<Symbol> Symbol of new benthic variable. Mandatory.
<InitValue> Initial value of benthic variable. If omitted, the benthic varible will inherit the pelagic variables initial value. See section 3.1
<NotNegative> Indicates if negative values are truncated to 0. "True" or "False" (default).
<PelagicVariable> Symbol of existing pelagic variable to diffuse with. Mandatory.
<DiffusionCoef> Diffusion coefficient for current variable which overrides the default diffusion coefficient.
<FluxSymbol> Add 2D variable with given symbol were the flux tranported across the pelagic/benthich interface is stored.
<Sedimentation> Enables sedimentation/vertical advection.
<ImplicitSedimentation> Use implicit scheme for sedimentation. If false use explicit scheme. Default true.
<PelagicVerticalVelocity> Default fall rate in the pelagic. NB positive upwards. See section 3.3
<BenthicVerticalVelocity> Default fall rate in the sediment. NB positive upwards. See section 3.3
<UsePelagicBottomVinTopSediment> If true, use pelagic bottom vertical velocity in top of sediment. This causes the pelagic to sedimentate into the surface sediment layer. "True" or "False" (default).
<Variable> Optionally multiple.
<PelagicVerticalVelocity> Fall rate in the pelagic for this variable only (overriding default value). NB positive upwards. See section 3.3
<BenthicVerticalVelocity> Fall rate in the sediment for this variable only (overriding default value). NB positive upwards. See section 3.3
<Symbol> Symbol of new benthic variable. Mandatory.
<InitValue> Initial value of benthic variable. See section 3.1
<NotNegative> Indicates if negative values are truncated to 0. "True" or "False" (default).
<PelagicVariable> Symbol of existing pelagic variable to enable sedimentation for. Mandatory.
<Variables> Same as for pelagic equation solver. See section 4.7.
<Equations> Same as for pelagic equation solver. See section 4.7.

Example

```
<EcoSediment>
    <General>
        <Run>true</Run>
        <NoLayers>2</NoLayers>
        <LayerThicknessCSV>0.5,0.5</LayerThicknessCSV>
        <Bathymetry>
            <Value>0.1</Value>
        </Bathymetry>
    </General>

    <PelagicVariables>
        <Variable>
            <Symbol>botV1</Symbol>
            <PelagicVariable>V1</PelagicVariable>
```

```
        </Variable>
    </PelagicVariables>

    <Diffusion>
        <DiffusionLength>0.1</DiffusionLength>
        <DiffusionCoef>1e-5</DiffusionCoef>
        <Variable>
            <DiffusionCoef>1e-8</DiffusionCoef>
            <Symbol>sediA</Symbol>
            <InitValue>0</InitValue>
            <PelagicVariable>A</PelagicVariable>
        </Variable>
    </Diffusion>

    <Sedimentation>
        <PelagicVerticalVelocity>-1e-5</PelagicVerticalVelocity>
        <BenthicVerticalVelocity>-1e-7</BenthicVerticalVelocity>
        <UsePelagicBottomVinTopSediment>true</UsePelagicBottomVinTopSediment
            >
        <Variable>
            <Symbol>sediC</Symbol>
            <InitValue>0</InitValue>
            <PelagicVariable>C</PelagicVariable>
        </Variable>
    </Sedimentation>

    <Variables>
        <Variable>
            <Symbol>sediV2</Symbol>
            <InitValue>0</InitValue>
        </Variable>
    </Variables>

    <Equations>
        <Equation>
            <Definition>sediV2=botV1-sediC</Definition>
            <Description></Description>
        </Equation>
    </Equations>
</EcoSediment>
```

4.9 Sources

Adds fresh watersources including concentrations of pelagic variables.

<Sources>
  <Source> Optionally multiple.
    <Symbol> Source symbol. Mandatory.
    <ElemNo> Element number to add source to. Mandatory.
    <Flux> Numeric or symbol that defines the fresh water flux. Mandatory for non-decoupled sources.
    <Description> Source description.
    <LayerNo> Layer number to add source to. Optional CSV to destribute source over multiple layers. Default is surface layer.
    <LayerFraction> When <LayerNo> is CSV, this must be CSV of same length with fractions for each layer. Optional. Default is normalized layer thicknesses.

<VirtualFlux> Virtual flux i.e. no water is added - only concentration. Optional. Default false.
<DynamicElementNumber> Symbol of dynamic element number, i.e. source moves in time. Optional.
<Variable> Optionally multiple.
<Symbol> Symbol of the pelagic variable to modify. Mandatory.
<Concentration> Numeric or symbol that defines the concentration of the pelagic variable in the fresh water. Mandatory.
<ConcentrationFactor> Numeric or symbol that defines a factor that the concentration is multiplied by. Optional. Default=1.

Example

```
<Source>
    <Symbol>s1</Symbol>
    <Description>Source 1 water flux [m3/s]</Description>
    <ElemNo>7</ElemNo>
    <Flux>fwi</Flux>
    <Variable>
        <Symbol>salt</Symbol>
        <Concentration>0</Concentration>
    </Variable>
</Source>
<Source>
    <Symbol>S1</Symbol>
    <Flux>Timedistribution1</Flux>
    <ElemNo>0</ElemNo>
    <LayerNo>0,1,2,3,4,5,6,7,8,9</LayerNo>
    <VirtualFlux>true</VirtualFlux>
    <DynamicElementNumber>S0ElemNo</DynamicElementNumber>
    <Variable>
        <Symbol>sediFrac1</Symbol>
        <Concentration>0.1244</Concentration>
    </Variable>
    <Variable>
        <Symbol>sediFrac2</Symbol>
        <Concentration>sfc2</Concentration>
    </Variable>
</Source>
```

4.10 Boundaries

Defines open boundary conditions.

<Boundaries>
<Boundary> Optionally multiply.
<Number> Boundary number. Must match an open boundary number in the mesh. Mandatory.
<ExchangeRate> Numeric. Boundary exchange rate [m3/s]. HDLite only.
<WindDirection> Numeric or symbol. Modifies the exchange rate as a function of wind direction. HDLite only.
<MaxExchangeDirection> Numeric. Direction of maximum exchange. HDLite only. Mandatory if <WindDirection> present.
<Variable> Optionally multiple. NB: all advected variables must be defined on all open boundaries.
<Symbol> Symbol of the pelagic variable to modify. Mandatory.

<Concentration> Numeric or symbol that defines the concentration of the pelagic variable on the boundary. if "upwind" values in the boundary cells are used (outflow). Mandatory.

Example

```
<Boundaries>
    <WindDirection>wd</WindDirection>
    <MaxExchangeDirection>0</MaxExchangeDirection>
    <ExchangeRate>300</ExchangeRate>
    <Boundary>
        <Number>1</Number>
        <Variable>
            <Symbol>salt</Symbol>
            <Concentration>saltbnd</Concentration>
        </Variable>
        <Variable>
            <Symbol>temp</Symbol>
            <Concentration>3.14</Concentration>
        </Variable>
        <Variable><Symbol>TRC</Symbol><Concentration>Upwind</Concentration><
           /Variable>
    </Boundary>
</Boundaries>
```

4.11 Surface

Adds surface exchanges e.g. heat exchange, oxygen, precipitation/evaporation, or modifies other pelagic state variables in the surface.

<Surface>

<IceCover> Ice cover symbol. Affects surface heat, oxygen and other surface exchanges. Optional.

<IceType> 0: no ice, 1:turn off exchange when IceCover larger than 0, 2: scale exchanges with iceCover. Optional. Default 0.

<HeatExchange> Optional. Add heat exchange.

<WaterTemperature> Mandatory. Numeric or symbol. Optional if <HD>, default is 'temp' (HD temperature).

<AirTemperature> Mandatory air temperature. Numeric or symbol.

<!--Choose between Fick's law, prescibed fluxes or heat flux model-->

<!-- Fick's law (simple conduction)-->

<ConductivityCoefficient> Mandatory heat conduction coefficient. 10 to 100 seems to be resonable (if 500 then water air)

<!-- Prescribed heat fluxes-->)

<ShortwaveHeatFlux> Mandatory shortwave radiation heat flux [W m^{-2}]

<LongwaveHeatFlux> Optional longwave radiation heat flux [W m^{-2}]. Default is 0.

<SensibleHeatFlux> Optiona lsensible heat flux [W m^{-2}]. Default is 0.

<LatentHeatFlux> Optional latent heat flux [W m^{-2}]. Default is 0.

<!-- Heat flux model-->)

<CloudCover> Mandatory cloud cover (0-1)

<WindU> Mandatory wind west-east velocity
<WindV> Mandatory wind south-north velocity
<RelativeHumidity> Relative humidity. RelativeHumidity, DewPointTemperature or SpecificHumidity mandatory.
<DewPointTemperature> Dew Point Temperature. Alternative to <RelativeHumidity>
<SpecificHumidity> Specific Humidity. Alternative to <RelativeHumidity>
<AirPressure> Optional air pressure [Pa]
<PenetrationDepth> Optional. Penetration depth of longwave, sensible and latent heat fluxes. Default is 1 m.
<AddHeatFluxesAsVariables> Adds 2D variables of calculated heat fluxes: 'HeatShortWave', 'HeatLongWave', 'HeatSensible' and 'HeatLatent'
<WaterEmissivity> Optional. Default 0.96
<LongwaveReflectivity> Optional. Portion of atmospheric longwave radiation reflected by water surface. Default 0.045
<WaterAlbedo> Optional. Default 0.06
<ExtinctionCoef> Optional. Default log(0.1) / 0.6=-3.837642 (90% absorbed in the top 0.6 m)
<ShortWaveExtinctionCoef> Optional. Default log(0.1) / 5.0=-0.460517 (90% absorbed in the top 5.0 m)
<BottomReflectivity> Optional. Proportion of heat reflected in bottom. Default 1.0 (all heat reflected back to water)
<SensibleHeatBulkCoef> Optional. Default 1.4e-3.
<EvaporationBulkCoef> Optional. Default 1.4e-3.
<MinimumWaterTemperature> Optional.

<Exchange> Optionally multiple.
<AtmosContribution> Numeric or symbol. Rate of loss/gain.
<PelagicSymbol> Symbol of pelagic variable to modify. Mandatory if <AtmosContribution> present.
<Precipitation> Numeric or symbol. Modify surface salinity by precipitation/evaporation.
<SaltSymbol> Salinity symbol. Mandatory if <Precipitation> present.
<Oxygen> Modifies surface oxygen following https://aslopubs.onlinelibrary.wiley.com/doi/10.4319/lom.2010.8.0628
<OxygenSymbol> Symbol of pelagic oxygen variable to modify. Mandatory.
<Salinity> Salinity symbol or numeric. Mandatory.
<Temperature> Water temperate symboll or numeric. Mandatory.
<WindSpeed> Wind speed symboll or numeric. Mandatory.
<UnitFactor> Oxygen saturation is calculated in ml O2/l. This is the conversion factor to the oxygen unit that your model uses. Default 44.661 which is the conversion factor between ml O2/l to mmol O2/m3 following ocean.ices.dk/Tools/UnitConversion.aspx.

Example

```
<Surface>
    <IceCover>icecov</IceCover>
    <IceType>2</IceType>
    <HeatExchange>
        <WaterTemperature>temp</WaterTemperature>
        <AirTemperature>airtemp</AirTemperature>
        <CloudCover>cloudcov</CloudCover>
        <WindU>wu</WindU>
        <WindV>wv</WindV>
        <SpecificHumidity>spechum</SpecificHumidity>
        <EvaporationBulkCoef>1e-4</EvaporationBulkCoef>
        <AddHeatFluxesAsVariables>true</AddHeatFluxesAsVariables>
    </HeatExchange>
```

```
    <MinimumWaterTemperature>-2</MinimumWaterTemperature>
    <Oxygen>
        <OxygenSymbol>O2</OxygenSymbol>
        <Salinity>salt</Salinity>
        <Temperature>temp</Temperature>
        <WindSpeed>ws</WindSpeed>
        <UnitFactor>44.61</UnitFactor>
    </Oxygen>
    <Exchange>
        <AtmosContribution>3.4e-6</AtmosContribution>
        <PelagicSymbol>NO3</PelagicSymbol>
    </Exchange>
    <Exchange>
        <Precipitation>precip</Precipitation>
        <SaltSymbol>salt</SaltSymbol>
    </Exchange>
</SurfaceExchanges>
```

4.12 EcoLight

Parameterization and calculation of light attenuation. The module adds the variable "I" to the model. I is calucated as I = I0*exp(-kd*d) , where d is the cell center depth, I0 = (1-k5)*v4 and kd = k1-k2*v1+k3*v3+k4*k6*v3 This non-intuitive formulation has historic origins.

<EcoLight>
- <IceCover> Ice cover symbol. Optional.
- <IceType> 0: no ice, 1:turn off light when ice¿0, 2: scale light with ice cover. Optional. Default 0.
- <v1> Variable symbol.
- <v2> Variable symbol.
- <v3> Variable symbol.
- <v4> Variable symbol. Mandatory.
- <k1> Numeric. Default 0.
- <k2> Numeric. Default 0.
- <k3> Numeric. Default 0.
- <k4> Numeric. Default 0.
- <k5> Numeric. Default 0.
- <k6> Numeric. Default 0.
- <AddkDasVariable> If "True" adds a new variable with symbol "kD" were kd values are stored.
- <kdSymbol> Symbol for new variable were kd values are stored.

Example

```
<EcoLight>
    <v1>salt</v1>
    <v2>C</v2>
    <v3>X</v3>
    <v4>glorad</v4>
    <k6>1</k6>
    <k1>1.5</k1>
    <k2>0.06</k2>
    <k3>0</k3>
    <k4>0.1</k4>
    <k5>0.55</k5>
    <kdSymbol>kd</kdSymbol>
</EcoLight>
```

4.12.1 Heat flux model

The temperature change in a cell caused by a heat flux Q is given by:

$$dT = \frac{Qdt}{dz\rho_w C_p} \qquad \text{(Hodges eq. 57)}$$

where Q is the heat flux [W m^{-2}], dt the timestep length [s], dz the cell thickness [m], ρ_w the water density [kg m^{-3}] and C_p the heat capacity of water = 4185.5 [J K^{-1} kg^{-1}]. The surface heat flux is calculated positive downwards (negative out of the water) ans is given by:

$$Q = Q_s + Q_l + Q_H + Q_L$$

where Q_s is the short wave radiation heat flux, Q_l the long wave heat radiation heat flux, Q_H the sensible heat flux and A_L the latent heat flux.

The long wave radiation heat flux is given by:

$$Q_l = -\epsilon_w \sigma T_w^4 + \epsilon_a \sigma T_a^4 (1 - R_{tlw})(1 + 0.7C_{cloud}^2) \qquad \text{(Hodges eq. 4)}$$

where T_w is the water temperature in Kelvin [K], T_a is the air temperature in Kelvin [K], ϵ_w the emissivity of water], ϵ_a the emissivity of air = $0.920 * 10^{-5} T_a^2$, σ is Stefan-Boltzmans constant = 5.669*10^{-8} W m^{-2} K^{-4}, R_{tlw} the total reflectivity of the water surface[1] and C_{cloud} the fractional cloud cover.

The sensible heat flux is given by:

$$Q_H = \rho_a C_{pa} C_H U_{wind} (T_a - T_w) \qquad \text{(Hodges eq. 29)}$$

where ρ_a is the air density =1.2 kg m^{-3}, C_{pa} the specific heat capacity of air = 1003 J kg^{-1} K^{-1}, C_H the bulk transfer coefficient for sensible heat = 1.4*10^{-3} and U_{wind} the wind speed in m s^{-1}

The latent heat flux is given by:

$$Q_L = LC_W U_{wind} \rho_a (q_a - q_w) \qquad \text{(Hodges eq. 22)}$$

where L is the latent heat of evaporation = 2.453*10^6 J Kg^{-1}, C_W the bulk transfer coefficient for evaporation = 1.4*10^{-3}, q_a the specific humidity in air and q_s the saturation humidity.

$$q_s = \frac{0.622}{P_a} e_s \qquad \text{(Hodges eq. 24)}$$

where P_is the air pressure [pa] and e_s, the saturation vapor pressure[2]:

$$e_s = exp(log(611.2) + \frac{17.62 * T_a}{243.12 + T_a}) \qquad (1)$$

$$q_a = \frac{0.622 e_s}{P_a} R_h \qquad (2)$$

where R_h is the relative humidity:

$$R_h = \frac{e}{e_s} \qquad (3)$$

[1] [Hodges 1998] uses 0.3, but others e.g. Josley et al 1997 https://agupubs.onlinelibrary.wiley.com/doi/pdf/10.1029/97JC02420 uses much smaller numbers: 0.045, which is default in FlexSem

[2] from www.npl.co.uk/reference/faqs/how-do-i-convert-between-units-of-dew-point-and-relative-humidity-(faq-thermal)

where e is the vapor pressure. If the dew point temperature is given instead of the relative humidity, the specific humidity can be calculated by using the dew point temperature in equation **??** to find the vapor pressure e and then calculating the specific humidity by:

$$q_a = \frac{0.622}{P_a} e \tag{4}$$

The sum of the heatfluxes from long wave radiation, sensible and latent heat is absorbed exponentially decreasing by depth until the penetration depth is reached. If the water is shallower than the penetration depth, the heat is reflected in the bottom and the remaining heat equally distributed in the water column. See Hodges 1998 for a detailed description.

The short wave radiation heat flux is given by:

$$Q_s = Q_{swsurf}(1 - 0.65C_{cloud}^2)(1 - R_{tsw}) \qquad \text{(Hodges eq. 42)}$$

where Q_{swsurf} is the clear sky incomming surface radiation and R_{tsw} the short wave surface reflectivity (albedo).

$$Q_{swsurf} = 0.75\frac{S_0}{d^2}cos(\theta) \qquad \text{(Sun eq. 7)}$$

where S_0 is the solar constant = 1367 W m^{-2}, d the earth-sun distance in astronomical units and θ the solar zenit angle.

$$d = 1 + 0.0167sin(2\pi\frac{DOY - 93.5}{365}) \qquad \text{(Sun eq. 3)}$$

where DOY is the day of year. The solar zenit angle is determined by:

$$cos(\theta) = sin(\delta)sin(\varphi) + cos(\delta)cos(\varphi)cos(\omega) \qquad \text{(Sun eq. 4b)}$$

where φ the latitude, ω the solar hour angle = $\pi\frac{t-12}{12}$, t = local time in hours and δ is the solar declination:

$$\delta = 0.4093sin(2\pi\frac{DOY + 284}{365}) \qquad \text{(Sun eq. 4b)}$$

Like the other heat fluxes, the short wave radiation heat flux is also absorbed exponentially decreasing by depth until the penetration depth is reached. However the penetration depth is typically greater (~ the secchi disk depth). If the bottom is reached the remaining flux is reflacted and absorbed exponentially decreasing upwards toward the surface. If the surface is reached before all the heat is absorbed, the remaining is equally distributed in the water column. See Hodges 1998 for a detailed description.

References:

Ben Hodges 1998 Heat budget and thermodynamics at a free surface: some theory and numerical implementation (revision 1.0c) ED 1300 BH

Sun, Zhigang; Gebremichael, Mekonnen; Wang, Qinxue; Wang, Junming; Sammis, Ted W.; Nickless, Alecia. 2013. "Evaluation of Clear-Sky Incoming Radiation Estimating Equations Typically Used in Remote Sensing Evapotranspiration Algorithms." Remote Sens. 5, no. 10: 4735-4752.

4.13 Sections

Calculate fluxes through user defined vertical sections. The flux is calculated positively to the right when walking along the section in the order that the nodes numbers are given.

<Sections>

- <Section> Define new section to calculate flux through. Optionally multiple.
  - <NodeNumberCSV> Comma separated list of node numbers that defines the section. The node numbers must be connected by a face.
  - <StartNode> Start node of section. Alternative to <NodeNumberCSV>
  - <EndNode> End node of section. Shortest path (Dijkstra) between <StartNode> and <EndNode> will be used. Mandatory if <StartNode>.
  - <FluxCalc> Calculate flux of specific variable through the section. Optionally multiple.
    - <PelagicSymbol> Symbol of pelagic variable to calculate flux for. Mandatory.
    - <Symbol> Symbol of new 0D variable to hold result of flux calculation. Mandatory if not <BruttoPositiveFluxSymbol> and <BruttoNegativeFluxSymbol>.
    - <BruttoPositiveFluxSymbol> Symbol of new 0D variable to hold postive part of flux calculation. Optional alternative to <Symbol>
    - <BruttoNegativeFluxSymbol> Symbol of new 0D variable to hold negative part of flux calculation. Optional alternative to <Symbol>

Example

```
<Sections>
    <Section>
        <NodeNumberCSV>142,300,264,41</NodeNumberCSV>
        <FluxCalc>
            <Symbol>saltflux</Symbol>
            <PelagicSymbol>salt</PelagicSymbol>
        </FluxCalc>
    </Section>
    <Section>
        <StartNode>1200</StartNode>
        <EndNode>1042</EndNode>
        <FluxCalc>
            <Symbol>NFlux1</Symbol>
            <PelagicSymbol>TN</PelagicSymbol>
            <BruttoPositiveFluxSymbol>NFlux1Pos</BruttoPositiveFluxSymbol>
            <BruttoNegativeFluxSymbol>NFlux1Neg</BruttoNegativeFluxSymbol>
        </FluxCalc>
    </Section>
</Sections>
```

4.14 AgentBasedModel

Add agent based model (ABM). Agents can have a model system which will be solved for each agent and can be advected in a Lagrangian way in the model.

<AgentBasedModel>

- <Agents> Add agents for model. Optionally multiple. Mandatory.
  - <MaxNumberAgents> Set the maximum number of agents in the model. Optional. default is the initial number of agents.
  - <Depth> Default initial depth for the agents. Optional. Default is 0.
  - <ActivateTimestep> Timestep or datetime when the agents will be activated. Optional. Default 0.
  - <OneInEachElementCenter> Add one agent in each element (2D) center. or
  - <OneInEachCellCenter> Add one agent in each cell (3D) center. or
  - <GridDx>
  - <GridDx> and <GridDy>Add agents in a grid given by ¡GridDx¿ and ¡GridDy¿.

<GridMinX> and <GridMaxX> and <GridMinY> and <GridMaxY> Boundaries of grid. Optional. Default is mesh area. or

<Filename> Add agents from a text file with columns: x,y, z (optional), number agents (optional). or

<Number> Number of agens to be added randomly distributed in polygons.

<DepthDistribution> Depth distribution. 0=surface, -1=random depths, 1=bottom

<ActivateFrom> Timestep or datetime to activate agents from.

<ActivateTo> Timestep or datetime to activate agents to.

<X> and <Y> CSV coordinates of polygons. Mandatory. Optionally multiple.

<GroupID> Assign these group IDs to the agents. Can be used in equations to add diffent initial values/behaviour.

<GroupIDDistribution>Distrubution of group IDs. sum(GroupIDDistribution)==1.

<IgnoreInitAgentsOutsideMesh> If true agents with initial positions outside mesh will be ignored. Optional. Default false.

<RandomSpeed> Agent velocities will be added with a random number between 0 and this value. Optional. Default 0.

<AgentBasedDiffusivity> Add diffusivity to the agent movement. ¡0 : no diffusion, 0:random, 1:from Hydrodynamics

<TimestepInterval> Integer interval of timesteps at which the agent bsae model is runned. (so 1 means every timestep).

<ABMDTSymbol> The symbol assigned to the ABM timestep length (dt*<TimestepInterval>). Optional. If not given no variables will be created.

<Variables> Same as for pelagic equation solver. See section 4.7.

<Equations> Same as for pelagic equation solver. See section 4.7.

<AgentPerElementSymbol> Sum up the number of agents per element in this 2D variable. Optional.

<AgentPerCellSymbol> Sum up the number of agents per cell in this 3D variable. Optional.

<Feedbacks> Let the agents affect eulerian 3D variables.

<Feedback> Optional multiple.

<Sink3DVariable> Symbol of the 3D varaible to be affected.

<ABMVariable> Symbol of the ABM varaible to affect the 3D variable.

<Boundaries>

<AllClosed> Make open boundaries behave like land boundaries (agents do not leave the domain).

<AgentDeactivateCriterion> Equation which can be evaluated as a boolean. Deactivate agents when true. Deactivated agents still exist, but their properties

Example

```
<AgentBasedModel>
    <TimestepInterval>1</TimestepInterval>
    <Agents>
        <Depth>0</Depth>
        <GridDx>0.005</GridDx>
        <GridDy>0.005</GridDy>
    </Agents>

    <ABMDTSymbol>abmdt</ABMDTSymbol>

    <RandomSpeed>0.001</RandomSpeed>

    <Variables>
        <Variable>
            <Symbol>age</Symbol>
            <InitValue>0</InitValue>
```

```
        </Variable>
    </Variables>

    <Equations>
        <Equation>
            <Definition>age = age + abmdt</Definition>
        </Equation>
    </Equations>

    <AgentDeactivateCriterion>age.gt.72000</AgentDeactivateCriterion>

    <AgentPerElementSymbol>abmPE</AgentPerElementSymbol>
    <AgentPerCellSymbol>abmPC</AgentPerCellSymbol>
</AgentBasedModel>
```

4.15 Outputs

Output variable values to files. Each file can only contain variable of the same dimenstion, i.e. you cannot output e.g. 2D and 3D variables to the same file.

<Outputs>
- <InputDTEqualsOutputDT> If true initial values are not outputted. Optional. Default false.
- <OutputFolder> All output Filenames will be concatenated with this string. Optional. Default empty string.
- <Output> New output. Optionally multiple.
  - <Filename> Output filename. Mandatory.
  - <Format> Format of output file. "EMT" (tab separated), "EMF" (fixed width), "EMB" (native binary), "NC (NetCdf), "RESTART" (output restart file), "DECOUPLED" (output decoupled file). Mandatory.
  - <TimestepFrom> The first time step to output. Default 0.
  - <TimestepTo> The last time step to output. Default is all time steps.
  - <TimestepInterval> The interval between outputted time steps. Default 1.
  - <Interval> The time between outputted time steps, see 3.4. Alternative to <TimestepInterval>
  - <CellNoCSV> Integer array of cell number to output. Default is to output all cells.
  - <BoundaryNumber> Output boundary file for this boundary number.
  - <FixedWidthFieldWidth> Width of the fields in fixed width format. Default 12.
  - <MaxFilesize> Maximum file size. If output is larger, the files are automatically split into multiple files. E.g. "1.9Gb"
  - <Variable> The symbol of the variable to output. Optionally multiple. Optional average attribute: <Variable Average="True"> to average values over <TimestepInterval> timesteps. Optional attribute: <Variable WaterColumnIntegration="True"> to integrate varaible over water column (Coverts 3D variable to 2D variable.

Example

```
</Outputs>
    <Output>
        <Filename>InOutAvg.txt</Filename>
        <TimestepFrom>0</TimestepFrom>
        <TimestepTo>-1</TimestepTo>
        <TimestepInterval>4</TimestepInterval>
        <Format>EMF</Format>
        <FixedWidthFieldWidth>14</FixedWidthFieldWidth>
        <Variable Average="true">ws</Variable>
```

```
        <Variable>ws</Variable>
        <Variable>wd</Variable>
    </Output>
    <Output>
        <Filename>LaholmsbuktenVelOneDay3D.emb</Filename>
        <Format>EMB</Format>
        <Variable>TRC</Variable>
    </Output>
    <Output>
        <Filename>LaholmsbuktenVelOneDay2D.emb</Filename>
        <Format>EMB</Format>
        <Variable>h</Variable>
    </Output>
</Outputs>
```