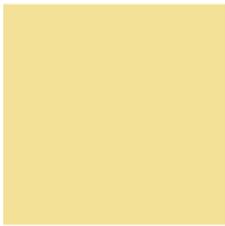


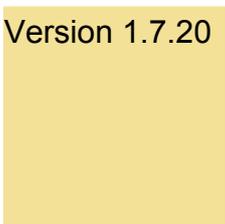
# Simulationsmodell Marina

Manual



smile consult GmbH  
Vahrenwalder Straße 7  
30165 Hannover  
[info@smileconsult.de](mailto:info@smileconsult.de)

Version 1.7.20 (1.3.2009)



# **1 Introduction**

The software package MARINA is a modeling system designed to simulate hydrodynamics and morphodynamics in rivers, estuaries and coastal areas. The numerical approximation of the transport-dominant, unsteady state governing equations is executed with stabilized finite triangular elements. A fully developed strategy for wetting and drying allows model simulation in complex river and estuary topographies.

Together with the pre- and post-processing tools JANET and DAVIT, the simulation system forms a complete process chain, from preparation of natural data to analysis and interpretation of model results.

XML format files are used for configuration of the different model modules as well as for steering numerical simulations.

## **1.1 Model components**

Model components MARINA2D and MARINA3D offer 2-dimensional and 3-dimensional capability depending on the requirements of the simulated processes.

All model components can be coupled with each other. The coupling takes place directly on module level.

### **1.1.1 MARINA2D**

MARINA2D is mainly based on the assumption of depth-integrated system variables and contains the model components CurrentModel2D, WaveHypModel and SedimentTransportModel2D. These models are complemented by components that describe depth-integrated salinity and heat transport (SaltTransportModel2D and HeatTransportModel2D).

### **1.1.2 MARINA3D**

Currently MARINA3D only consists of the 3-dimensional current model CurrentModel3D. Coupling with modules from MARINA2D is possible under particular conditions.

## 2 Theoretical foundations

The implemented mathematical model equations are based on the Navier-Stokes-Equations. Model equations for currents and waves can be derived based on decomposition in turbulent and wave fractions similar to the Reynolds decomposition. The assumption of depth integration results in the shallow water equations (CurrentModel2D) and --along with a period-averaged treatment-- results in the hyperbolic wave model (WaveHypModel).

### 2.1 CurrentModel2D

The current module CurrentModel2D is a 2-dimensional depth-integrated unsteady current model based on the depth-averaged Reynolds equations:

$$\frac{\partial u}{\partial t} = -u \frac{\partial u}{\partial x} - v \frac{\partial u}{\partial y} - g \frac{\partial \eta}{\partial z} + v \cdot 2\Omega \sin \varphi + \frac{1}{\rho(\eta + z_B)} (T_x - T_x^B + T_x^W)$$

$$\frac{\partial v}{\partial t} = -u \frac{\partial v}{\partial x} - v \frac{\partial v}{\partial y} - g \frac{\partial \eta}{\partial z} - u \cdot 2\Omega \sin \varphi + \frac{1}{\rho(\eta + z_B)} (T_y - T_y^B + T_y^W)$$

$$\frac{\partial \eta}{\partial t} = \frac{-\partial u(\eta + z_B)}{\partial x} - \frac{\partial v(\eta + z_B)}{\partial y}$$

where  $z_B$  is the average still water depth or bottom level;  $u$  and  $v$  are components of the current vector; and  $\eta$  is the deflection of the water surface from still water level.

The following effects are taken into account in the model:

- energy loss due to bottom friction (according to Manning-Strickler and rough bottoms)
- energy input due to wind (according to Smith and Banke (1975))
- coriolis effect
- wetting and drying
- turbulence modeling (according to Smagorinsky).

Additional terms are taken into account as the current model is coupled with other modules. This includes radiation stresses to describe wave-induced currents, among other effects.

### 2.1.1 Energy loss due to bottom friction

Energy loss in depth-integrated models is described by the bottom shear stress  $T^B$ . Bottom shear stress consists of two parts, the shear stress  $\bar{\tau}_B$  which is proportional to the square of the average velocity and a term  $\bar{\tau}_{BX}$  which considers the unevenness of the bottom.

$$T^B := \bar{\tau}_B + \bar{\tau}_{BX}$$

All equations describing bottom shear stress for depth-integrated models have in common that the velocity is included squared. But there is disagreement about the power of the water depth. In the depth-integrated current module, the Manning-Strickler approach is implemented, which is an approximation of Nikuradse's law:

$$\frac{\tau_B}{\rho} := \frac{g}{h^{1/3} k_{Str}^2} u^2$$

The Strickler coefficient has the awkward dimension  $m^{1/3} s^{-1/2}$ , which lacks a readily discernable physical meaning. The Strickler coefficient ranges from  $100 m^{1/3} s^{-1/2}$  for very smooth new steel to  $20 m^{1/3} s^{-1/2}$  for creeks with coarse rubble. A Strickler coefficient of  $48 m^{1/3} s^{-1/2}$  describes a sandy bottom.

Calculating the bottom shear stress with classic approaches is not sufficient for areas with steep bottom gradients. This can be corrected by taking an additional term into account:

$$\bar{\tau}_{BX} = \begin{pmatrix} \rho \bar{v}_t \left( 2 \frac{\partial \bar{u}}{\partial x} \frac{\partial z_B}{\partial x} + \left( \frac{\partial \bar{u}}{\partial y} + \frac{\partial \bar{v}}{\partial x} \right) \frac{\partial z_B}{\partial y} \right) \\ \rho \bar{v} \left( \left( \frac{\partial \bar{u}}{\partial y} + \frac{\partial \bar{v}}{\partial x} \right) \frac{\partial z_B}{\partial x} + 2 \frac{\partial \bar{u}}{\partial y} \frac{\partial z_B}{\partial y} \right) \end{pmatrix}$$

This extended shear stress component is also used to determine the critical shear stress in the sediment transport module.

### 2.1.2 Turbulence model

Turbulence is a 3-dimensional phenomenon and cannot be simulated directly by depth-averaged models. Different model concepts can be used to describe the effect of turbulence on depth averaged currents. In the depth averaged model a combination of the kinematic viscosity of water  $\nu_c$ , depth averaged turbulence  $\nu_E$

according to Elder, and a parameterization of eddies, which cannot be represented by the model grid.  $\nu_S$  according to Smagorinsky is used:

$$\nu_t = \nu_c + \nu_E + \nu_S$$

The default value for kinematic viscosity of water is  $\nu_c = 10^{-6} \text{ m}^2/\text{s}$ .

The Elder approach establishes a connection to water depth and bottom shear stress:

$$\nu_E = \nu_c \cdot u_x \cdot (\eta + z_B)$$

The friction velocity results from Manning's equation.

The oldest and best comprehended fine structure model is that of the meteorologist Smagorinsky (1963). It is based on the approach of eddy viscosity by Boussinesq and can be taken as an adaptation of the mixing length model from statistic turbulence modeling to the fine structure model.

$$\nu_S = (c_s \cdot \Delta)^2 \left[ 2 \cdot \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 + 2 \cdot \left( \frac{\partial v}{\partial y} \right)^2 \right]$$

where  $c_s$  is a constant and  $\Delta$  is the filter length (a measure for the grid width). Here, for the parameter  $c_s$  the kinematic viscosity of water is used again.

### 2.1.3 Energy input due to wind

The input of momentum from the atmosphere is described by the wind shear stress  $\vec{\tau}_{wind}$ :

$$\vec{\tau}_{wind} = \rho_{air} C_D \vec{v}_{10,wind} |\vec{v}_{10,wind}|$$

$C_D$  is calculated with constants proposed by Smith and Bank (1975):

$$C_D = \left( 0.63 + 0.066 |\vec{v}_{10,wind}| 10^{-3} \right)$$

### 2.1.4 Wetting and drying

In natural topographies, including those in river and coastal areas, the boundary between dry and wet regions is constantly changing due to varying water levels.

To classify dry and wet nodes, a minimum water level is defined. The minimum water level  $h_{dry}$  shows that a node can be considered as potentially dry. A node is completely dry, if the water level is lower than  $h_{dry} / 2$ .

An element is completely dry, if the water level at all nodes is lower than  $h_{dry} / 2$ . Elements with at least one node lower than  $h_{dry}$  are called partially dry and are treated separately. For partially dry elements, a horizontal water surface around the dry node is assumed, and the spatial derivative of the water surface used in the momentum equation will be adjusted accordingly.

## 2.2 WaveHypModel

The basic principle of the spacial wave model is a period averaged consideration based on linear wave theory and leads to the following time dependent differential equations:

$$\frac{\partial k_x}{\partial t} = -\frac{\partial \sigma}{\partial x} - C_{gy} \left( \frac{\partial k_x}{\partial y} - \frac{\partial k_y}{\partial x} \right)$$

$$\frac{\partial k_y}{\partial t} = -\frac{\partial \sigma}{\partial y} + C_{gx} \left( \frac{\partial k_x}{\partial y} - \frac{\partial k_y}{\partial x} \right)$$

$$\frac{\partial \sigma}{\partial t} = -C_{gx} \frac{\partial \sigma}{\partial x} - C_{gy} \frac{\partial \sigma}{\partial y} + f \frac{\partial h}{\partial t}$$

$$\frac{\partial a}{\partial t} = -\frac{1}{2a} \left( \frac{\partial (C_{gx} a^2)}{\partial x} + \frac{\partial (C_{gy} a^2)}{\partial y} \right) + \frac{\varepsilon_b}{\rho g a} + \frac{S_{source}}{\rho g a} + c_b a^2$$

with  $c_b =$  bottom friction coefficient (according to Bijker)

$\varepsilon_b =$  wave energy loss due to breaking

The calculation of energy loss due to a multitude of wave breaking criteria can be found in literature. When coupling the wave model with the current model, more terms are taken into account including refraction and shoaling due to currents.

## 2.3 SedimentTransportModel2D

In addition to the 2-dimensional shallow water equations, a transport equation for suspended material and the bottom continuity equation are solved to describe morphodynamic changes. The bottom continuity equation

$$\frac{\partial z_B}{\partial t} = -\frac{1}{1-n} \left( \frac{\partial q_x}{\partial x} - \frac{\partial q_y}{\partial y} \right)$$

describes the change of the river bottom by spatial derivations of the sediment transport rates  $q_x$  and  $q_y$  in the x- and y-direction. Here  $n$  is the porosity of the

naturally bedded sediment and  $z_B$  is the location of the river bed. The total transport rate  $q$  consists of bed load  $q_B$  and suspended load  $q_S$ :

$$q = q_B + q_S$$

### 2.3.1 Bed load determination

Bed load is the component of solids transport that involves particles being carried by the current velocity close to the bed. Above a specific critical velocity, more and more particles first reach a rolling motion, then eventually a jumping and hopping motion.

The total bed load transport  $q_B$  occurs as a quasi 2-dimensional phenomenon above the bed. It can be split between a component due to current  $q_{BC}$ , and a component due to gravity, which results from the bed slope (analogous to sand grains that roll down a slope):

$$\vec{q}_B = \vec{q}_{BC} + \beta \cdot \text{grad } z_B \cdot |q_{BC}|$$

The empirical value 0.2 is used for the parameter  $\beta$ .

#### 2.3.1.1 Bed load transport subjected to currents

Particles that do not lift from the bottom, but roll, slide, hop or glide along the bed in the direction of the current, are referred to as bed load, which is caused by currents. Significant factors for bed load include size and weight of the grain, current velocity and bed slope. Einstein defines the region in which bed load generally takes place, as a layer above the bottom with a thickness of twice the grain diameter.

Different transport approaches are implemented within Marina and are described in the following sections.

#### Van Rijn (1984)

One method implemented is the approach according to van Rijn [Sediment pick-up-function in J. of Hydraulic Engineering, Number 110(10), p. 1494-1502, 1984]. It takes the following factors into account:

- grain velocity
- height of jumps
- bed load concentration

Van Rijn follows the concept of Bagnold, which holds that bed load depends mainly on grain motion caused by current forces. The bed load rate  $q_{BC}$  is defined as the product of velocity, jumping height and bed load concentration.

$$q_{BC} := u_b \cdot \delta_b \cdot c_b$$

Combination and insertion of the three components results in a bed load rate of:

$$q_{BC} = 0.053 \cdot d_{50} \sqrt{\frac{\rho_s - \rho_w}{\rho_w}} \cdot g \cdot d_{50} \cdot D^{-0.3} \cdot T^{2.1}$$

$$\text{with } T = \frac{\tau_B - \tau_{cr}}{\tau_{cr}}$$

where  $\tau_B$  is the bottom shear stress caused by a combination of currents and bed slope, and  $\tau_{cr}$  is the critical bottom shear stress according to Shields.

As a general rule the van Rijn equation, based on experiments, is valid for grain diameters ranging from  $200 \mu\text{m} < d_{50} < 2000 \mu\text{m}$  and water depths greater than 0.1 m.

### **Meyer-Peter and Müller (1948)**

xxxx

### **EngelundHansen (1967)**

$$q_{BC} = 0.05 \left( \frac{\rho_w}{\rho_s - \rho_w} \right)^2 \frac{u^5}{\sqrt{g d_{50} C^3}}$$

where C is the Chezy-coefficient.

### **CamenenLarson (2005)**

xxxx

#### **2.3.1.2 Portion caused by secondary currents**

In addition to the main flow of a water body, secondary currents are generated in river bends. These currents lead to an additional loss of momentum and in general to higher erosion rates. The increase of bottom shear stress is described by:

$$\tau_{\text{secondary current } x} = \frac{7r}{\alpha} \frac{\rho g h}{(u^2 + v^2)} \left( u \frac{\partial \eta}{\partial y} - v \frac{\partial \eta}{\partial x} \right) (+v)$$

$$\tau_{\text{secondary current } y} = \frac{7r}{\alpha} \frac{\rho g h}{(u^2 + v^2)} \left( u \frac{\partial \eta}{\partial y} - v \frac{\partial \eta}{\partial x} \right) (-v)$$

where  $r$  is the friction coefficient according to Strickler

$$r = g / (k_{str}^2 h^{1/3})$$

The correction factor  $\alpha$  is stated in literature ranging from 0.75 for rough beds to 1.0 for very smooth beds. The default value for  $\alpha$  is set to 1.0.

### 2.3.2 Determination of suspended load transport

To determine suspended load transport, an additional transport equation is solved:

$$\frac{\partial C}{\partial t} = -u \frac{\partial C}{\partial x} - v \frac{\partial C}{\partial y} + \frac{1}{\rho(\eta + h)} T_c + S$$

Suspended load is calculated by multiplying the integrated sediment concentration over the depth by the current velocity:

$$q_s = \int_0^d \vec{u} \cdot C \, dz$$

Suspension and deposition is realized with the help of a source and sink term  $S$ . The formulation of the source and sink term  $S$  assumes that one can find a maximum theoretically possible sediment concentration for every flow situation at one unique time and position. This saturated sediment concentration  $C_{\max}$  is calculated based on several methods.

On the other hand, there already exists an actual concentration at the observed position due to water motion. If the actual sediment concentration is greater than the maximum, sediment deposition occurs, referred to as a sink:

$$S = w_s (C_{\max} - C)$$

But with an actual concentration less than the maximum, erosion occurs and  $S$  becomes a source:

$$S = \lambda_e (C_{\max} - C)$$

where  $\lambda_e$  is the adaptation time.

### Rossinsky and Debolsky (1980)

$$C_{\max} = \frac{8.9 \cdot 10^{-5} u^3}{g w_s d}$$

### Bagnold (1966)

Bagnold uses energy equilibrium to determine suspended load:

$$C_{\max} = \frac{1}{d} \frac{e_s (1 - e_b)}{(\rho_s - \rho_w) g (w_s / u - \tan(\beta))} \tau_b u$$

with

efficiency factors  $e_s \cong 0.02$  and  $e_b \cong 0.1$

bottom slope  $\beta$

#### 2.3.2.1 Settling velocity

The formulation of Oseen, which is valid for a wide range of grain diameters, is used to determine the settling velocity:

$$w_c = \sqrt{\frac{g}{18} \frac{w_c d^2}{\left(v + \frac{3}{16} w_c d\right)} \frac{\rho_s - \rho_w}{\rho_w}}$$

This implicit equation has to be solved iteratively, where the settling velocity according to Stokes is appropriate as an initial value:

$$w_{c_0} = \frac{g}{18 \cdot v} \frac{\rho_s - \rho_w}{\rho_w} d^2$$

For grain diameters up to 0.1 mm, the settling velocities calculated with the approaches of Stokes and Oseen are almost equal. Above this, Stokes' approach leaves its area of validity, and Oseen's approach should be used, which is more universally valid.

#### 2.3.3 Variable mean grain diameters

In general, next to a varying bottom layer a shift in sediment composition within the limit of the available material occurs due to erosion and deposition. Thus in areas of erosion, a so called plastering can be observed with coarse grain as the top layer. By contrast deposition areas are characterized more by fine material. This phenomenon is described by an empirical differential equation:

$$dd50/dt = \text{morphFactor} * \text{slope} * dz/dt * d50 * (1 - d50/\text{upperGrainSize}) * (1 - \text{lowerGrainSize}/50);$$

with default values of      slope = 1  
    upperGrainSize = d50\_init\*10  
    lowerGrainSize = d50\_init/10.

## 2.4 SaltTransportModel2D

Simulation of salt transport is done by the module SaltTransportModel2D coupled with CurrentModel2D. The module is based on the depth averaged transport equation for salinity concentration:

$$\frac{\partial C}{\partial t} = -U_x \frac{\partial C}{\partial x} - U_y \frac{\partial C}{\partial y} + \frac{1}{\rho(\eta + h)} T_c + S$$

When using the salt transport module along with CurrentModel2D, the terms for density induced transport are activated.

## 2.5 HeatTransportModel2D

Analogous to salinity transport, density variations due to changing temperatures are taken into account in the depth averaged current model. The depth averaged heat transport is described by:

$$\frac{\partial T}{\partial t} = -u_i \frac{\partial T}{\partial x_i} + \frac{\partial}{\partial x_i} \left( k \frac{\partial T}{\partial x_i} \right) + \frac{a \cdot (T_L - T)}{\eta + h} + \frac{I_{sur}}{c \cdot (\eta + h) \cdot \rho_w}$$

where  $T_L$  is the mean air temperature of the atmosphere,  $T$  is the depth averaged water temperature,  $\eta + h$  is the water depth and  $a$  the specific heat-transfer coefficient. Solar radiation is included with a solar radiation efficiency value  $I_{sur}$ .

## 2.6 CurrentModel3D

Incorporating the Navier-Stokes equations together with the Boussinesq approximation and taking into account the earth's rotation leads to the following system of unsteady partial differential equations:

$$\frac{\partial u}{\partial t} = -u \frac{\partial u}{\partial x} - v \frac{\partial u}{\partial y} - w \frac{\partial u}{\partial z} - \frac{1}{\rho} \frac{\partial p}{\partial x} + v \cdot 2\Omega \sin \varphi + (T_x - T_x^B + T_x^W)$$

$$\frac{\partial v}{\partial t} = -u \frac{\partial v}{\partial x} - v \frac{\partial v}{\partial y} - w \frac{\partial v}{\partial z} - \frac{1}{\rho} \frac{\partial p}{\partial y} - u \cdot 2\Omega \sin \varphi + (T_y - T_y^B + T_y^W)$$

$$\frac{\partial w}{\partial t} = -u \frac{\partial w}{\partial x} - v \frac{\partial w}{\partial y} - w \frac{\partial w}{\partial z} - \frac{1}{\rho} \frac{\partial p}{\partial w} + g + T_w$$

The equation for free surface motion results from the continuity equation

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

and the boundary condition

$$\frac{\partial \eta}{\partial t} = -\frac{\partial q_x}{\partial x} - \frac{\partial q_y}{\partial y}$$

where  $q_x$  and  $q_y$  are the discharges of all layers, which are calculated in layers.

The velocity  $w$  for the layer boundary can now be calculated for each node by summing up the divergences starting from the bottom.

The model is discretized by defining an arbitrary number of vertical layers with different thicknesses.

### 3 Numerical approximations

The hydro- and morphodynamical model equations are unsteady partial differential equations that are numerically approximated by stabilized finite triangular elements. The model equations that describe the physical processes are solved in MARINA with direct coupling. For the sub-processes (waves -  $W$ , currents -  $U$  and sediment transport -  $S$ ) the following differential operators can be introduced:

$$L_v \equiv A_{vi} \frac{\partial}{\partial x_i} - \frac{\partial}{\partial x_i} \left( K_{vij} \frac{\partial}{\partial x_j} \right)$$

Hence, the model equations can be written as

$$\frac{\partial W}{\partial t} + L_w W + Q_w = 0$$

$$\frac{\partial U}{\partial t} + L_U U + Q_U = 0$$

$$\frac{\partial S}{\partial t} + L_S S + Q_S = 0$$

As a general rule, the differential operators  $L_i$  as well as the source and sink terms  $Q_i$  are dependent on all unknowns of the system.

### 3.1 Semi-discrete stabilized finite element approximation

Let  $\Omega$  be an open bounded region in  $\mathbb{R}^n$  and  $\Gamma$  the boundary. The desired result is a time dependent vector function  $U: \Omega \rightarrow \mathbb{R}^m$ , which satisfies the equation

$$\frac{\partial U}{\partial t} + LU + Q = 0$$

Here  $L$  is a quasi-linear differential operator and  $Q$  is a source and sink term. It is assumed that all necessary initial values and boundary conditions required for a solution are given. The quasi-linear operator has the following form:

$$L \equiv A_i \frac{\partial}{\partial x_i} - \frac{\partial}{\partial x_i} \left( K_{ij} \frac{\partial}{\partial x_j} \right)$$

where  $A_i$  is the  $i^{\text{th}}$  transport matrix and  $K_{ij}$  are the diffusion matrices. Hence, the operator  $L$  can be understood as the sum of a transport operator  $L_{adv}$  and a diffusion operator  $L_{diff}$ :

$$L = L_{adv} + L_{diff}.$$

Each operator can be separated furthermore into its local components. For example, the transport operator has the form:

$$L_{adv} = \sum L_{adv_i} = \sum A_i \frac{\partial}{\partial x_i}.$$

To approximate the solution of the basic underlying equations with the finite elements method, the region  $\Omega$  will be discretized into  $n_{el}$  finite elements  $\Omega_e$ . Using this functional approach, the components of  $\Phi_i$  are defined to approximate the unknown solution  $U \cong \tilde{U} = \sum \Phi_i U^i$ . Here,  $U^i$  can be considered node values of the approximated function  $\tilde{U}$ .

The derivation of the semi-discrete stabilized finite element approximations is realized by combination of a standard Galerkin method and a method of least squares:

$$\int_{\Omega} \Phi_l \left( [\Phi_i]^T \left[ \frac{\partial U^i}{\partial t} \right] + L([\Phi_i]^T [U^i]) + Q \right) d\Omega$$

$$+ \sum_k \tau_k \int_{\Omega_k} L^T(\Phi_l) \left( [\Phi_i]^T \left[ \frac{\partial U^i}{\partial t} \right] + L([\Phi_i]^T [U^i]) + Q \right) d\Omega_k = 0$$

The first integral contains the Galerkin approximation, and in the second line the terms of the least squares stabilization can be found, which consist of a sum of integrals over the inside of the elements. This approximation is called a semi-discrete Galerkin-least-squares approximation (GLS).

The stabilization parameter  $\tau$  plays a major role in determining the quality of the numerical approximation. Based on the determination of the upwinding coefficient  $\tau$  (symbol?) in the one dimensional scalar case, the following rule based on the spectral radii of the quasi-linear differential operators is used:

$$\tau := \frac{\alpha_{opt} \cdot h_e}{2 \cdot \|L_{adv}\|}$$

According to the definition of the absolute value of a number or a vector respectively, the value of the quasi linear differential operator is determined as follows:

$$\|L_{adv}\| := \sqrt{\sum \rho(A_i)^2}$$

where  $\rho(A_i)$  are the spectral radii of the operator components.

$$\rho(A_i) := \max |\lambda_j(A_i)|$$

The parameter  $\alpha_{opt}$  is calculated for each element similarly to the one dimensional case

$$\alpha_{opt} := \coth(Pe) - \frac{1}{Pe}$$

with the element-Peclet number

$$Pe := \frac{\|L_{adv}\| \cdot h_e}{\|L_{diff}\|}$$

where  $h_e$  is a measure of the extension of elements. The element extension is calculated based on the transport direction.

### 3.2 Time integration

The integration of time is realized based on an adapted 2<sup>nd</sup> order Adam-Bashfort:

$$x(t + \Delta t) = x(t) + (3x'(t) - x'(t - \Delta t)) / 2 \cdot \Delta t$$

The classical Euler method is used especially at nodes with boundary conditions and in transition zones between dry and wet areas .

## **4 Model creation**

The numerical simulation tools within the model system MARINA require a computational grid consisting of triangular elements, which represents the topography of the model area. Furthermore the modeler has to define boundary conditions and values at the open model boundary, which steer the simulation.

In addition to the computational grid and the boundary conditions, the hydro- and morphodynamical simulation models need initial conditions, which define the initial state of the system parameters to be simulated throughout the model area.

As a rule, three alternative approaches to specify initial conditions are possible:

- presetting of constant values in the XML file
- in the case of restarting with an existing result, a result-set file can be imported
- the third possibility is to use an ASCII file

If no specifications of initial conditions are made in the Marina XML file, general algorithms are implemented to determine appropriate initial conditions.

Boundary conditions need to be specified at the borders of the model area. It is possible to provide implicit boundary conditions by using the characteristic values of the boundary. Time series data for specific nodes can also be defined.

The Marina XML file is used for model steering as well as for model configuration. The complete structure and an example file are available in the appendix.

### **4.1 Computational grid**

The preprocessor Janet can be used for building the computational grid. This tool provides an interactive, graphical user interface to accomplish this.

A computational grid must fulfill specific requirements. In addition providing a realistic reproduction of the area's depths and an adequate reproduction of structures such as jetties, a number of (geometric) quality characteristics need to be fulfilled to obtain good simulation results with a numerical model.

The quality characteristics of a structurally suitable grid are:

- element quality (equilateral triangles are preferred); an index for the element quality is the shape parameter; degenerated elements can be localized and edited with aid of a system analysis tool
- homogenous transitions of adjacent elements (element areas of adjacent elements are preferably equal)
- do not use large patches
- the grid should not contain over-determined elements (elements with solely boundary nodes)

Grid element edges, which can be in TicadSysDat- or JanetBinary-format, can be marked as impervious. In current models this designation leads to a no-slip boundary condition and, in the wave model, to a wave absorbent boundary condition. Such implicitly given boundary conditions can be overwritten with explicitly specified conditions. The specification of the grid is done in the block *<ControlParameter>* of the Marina XML file. The keyword is called *<system>* with the subordinate key *<TriangleMesh>* and requires the specification of the name *<FileName>* and the file type *<FileType>*.

## 4.2 Model steering

The model is steered by the *<ControlParameter>* block in the Marina XML file. Following elements can be specified in this block:

<i>&lt;BaseDir&gt;</i>	- determination of working directory
<i>&lt;NumberOfThreads&gt;</i>	- number of used processors
<i>&lt;SystemTriangleMesh&gt;</i>	- computational grid specification
<i>&lt;FileName&gt;</i>	- name of file
<i>&lt;FileType&gt;</i>	- type of file ( <i>JanetBinaryFile</i> , <i>SysDatFile</i> )
<i>&lt;TimeIntegrationScheme&gt;</i>	- time integration ( <i>Euler</i> , <i>Heun</i> )
<i>&lt;SimulationTime&gt;</i>	- solution time interval
<i>&lt;StartTime&gt;</i>	- start time of simulation
<i>&lt;EndTime&gt;</i>	- end time of simulation
<i>&lt;ResultTimeStep&gt;</i>	- time step in which the results are written on a file

## 4.3 Model configuration

The model configuration takes place in a second block named *<Configuration>* in the Marina XML file. This block defines which model modules will be used for the simulation. Each individual module needs further specifications, which are also given in this block.

### 4.3.1 CurrentModel2D

The parameters needed to steer the model module CurrentModel2D are given in an element called *<CurrentModel2D>* which is subordinate to the *<Configuration>* block.

The specification of boundary conditions is made in a boundary condition file. Water levels, velocities, and relative as well as absolute discharges can be defined. Name and type of the file are defined in the Marina XML element *<BoundaryCondition>*, which is located directly below the *<CurrentModel2D>* element in the hierarchy:

- <BoundaryCondition>* - specification of boundary conditions
  - <FileName>* - name of file
  - <FileType>* - type of file (*RndwerteBAW*)

The model module also contains elements to specify initial conditions (*<InitialCondition>*), bottom friction (*<BottomFriction>*) and the boundary where grid elements become dry (*<DryFallBound>*).

If e.g. bottom friction is specified in a model, this is done either by a given grid with Manning-Strickler values that vary spatially or by a given constant Manning-Strickler value. Alternative elements are available for specification of initial conditions (*<StartFromResultFile>*, *<StartWithWaterLevel>*, *<ConstantWaterLevel>*).

Available elements are listed in the following table.

Row 1 see german manual	Row 2 see german manual	Limit above which a node will be treated as dry
		Specification of boundary conditions File name Type: <i>RndwerteBAW</i>
		Specification of Manning-Strickler-values Grid with location variable Mann.-Str.-values File name Type: <i>JanetBinaryFile, SysDatFile</i> Constant Manning-Strickler-value

		Specification of initial conditions Read from results file File name Type: TicadSysErg Dataset Read from ASCII file File name Type: SysDatFile Constant water level
		File with weir definitions File name Type: <i>WeirsDat, Weirs2DXML</i>
		Result file File name Type: <i>Current3DErg, TicadSysErg</i>

### 4.3.2 WaveHypModel

Parameters for the WaveHypModel are defined in the element `<WaveHypModel>` which is subordinate to the element `<Configuration>`.

Available elements are listed in the following table.

Row 1 see german manual	Row 2 see german manual	Result file File name Type: TicadSysErg
		Specification of boundary conditions File name Type: RandnDat
		Specification of initial values Read from results file

		File name Type: TicadSysErg Dataset
		Specifies a spacial- and time-constant amplitude of the water surface relative to sea level that is positive in the upward direction

### 4.3.3 SedimentTransportModel2D

Sediment transport simulations always require a current model and therefore the element *<CurrentModel2D>* in the steering file. Additionally, process effects of the hyperbolic wave model *WaveHypModel* are taken into account.

Boundary conditions are implicitly specified by assuming that to borders with current boundary conditions always a sediment saturated water body is connected.

Available elements are listed in the following table:

Row 1 see german manual	Row 2 see german manual	Result file File name Type: TicadSysErg
		Maximum erosion depth with space-variable values File name Type: JanetBinaryFile, SysDatFile
		Average grain size (d50) File name Type: JanetBinaryFile, SysDatFile Constant grain size in the entire area
		Type: RossinskyDebolsky
		Type: vanRijn84
		Specification of boundary conditions File name

		Type: <i>RndwerteBAW</i>
		Specification of initial conditions Read from results file File name Type: TicadSysErg Dataset

#### 4.3.4 SaltTransportModel2D

Salt transport simulations always require a current model (<*CurrentModel2D*>).

Boundary conditions are given in a boundary condition file (rndwerte.baw). From this file only values with the boundary value type 4 (ZWEIG) are read and taken into account.

More information about the boundary condition file is available in appendix I.

Row 1 see german manual	Row 2 see german manual	Specification of boundary conditions File name Type: <i>RndwerteBAW</i>
		Specification of initial conditions Read from results file File name Type: TicadSysErg Dataset
		Result file File name Type: TicadSysErg

#### 4.3.5 HeatTransportModel2D

A heat transport model always requires a current model (<*CurrentModel2D*>).

Boundary conditions are given in a boundary condition file (rndwerte.baw). From this file only values with the boundary value type 6 (ZWEIG) are read and taken into account.

More information about the boundary condition file is available in appendix I.

Row 1 see german manual	Row 2 see german manual	Specification of boundary conditions File name Type: <i>RndwerteBAW</i>
		Specification of initial conditions Read from results file File name Type: TicadSysErg Dataset
		Result file File name Type: TicadSysErg

#### 4.3.6 CurrentModel3D

The model parameters of the module CurrentModel3D are similar to the model module CurrentModel2D.

But CurrentModel3D requires additional information about the depths in which the process parameters are simulated (<DepthLayer>).

Row 1 see german manual	Row 2 see german manual	Limit above which a node is considered dry
		Specification of boundary conditions File name Type: <i>RndwerteBAW</i> for depth averaged boundary conditions
		Specification of Manning-Strickler-values Grid with space-variable Manning-Strickler values File name Type: JanetBinaryFile, SysDatFile

		Constant Manning-Strickler value
		List of depths at which process parameter are simulated
		Specification of initial conditions Read from results file File name Type: Current3DErg, TicadSysErg Dataset Read from ASCII file File name Type: SysDatFile Constant water level
		Result file File name Type: TicadSysErg

## 5 Start of a simulation

The following files are the minimum required to start a simulation:

- model grid in depth coordinate system (i.e. positive coordinate axis points downward)
- boundary value file
- Marina XML file

The simulation is started in a special console (prompt) which is opened by using the batch file "marina\_shell.bat":

Graphic see german manual

In this shell, which is especially configured for Marina, the environment variables are already set, so a numerical simulation can be started immediately.

After entering the term "marinahelp" and pressing the return key the user will get information about starting a simulation.

Graphic see german manual

One way to start a simulation is by entering the command "marina -gui". This will open a file dialog box where the XML steering file can be chosen. After accepting the entry the simulation starts.

Alternatively a simulation can be started by using the console. In this manner the desired XML file is specified as a parameter in the command line (e.g. "marina Simulation.xml").

After the successful start of the simulation, the system is read from system.dat and the boundary conditions from rndwerte.baw. If other configuration parameters (e.g. wind, sediment, etc.) are used in the XML file, those are read as well from the simulation system. The actual state is displayed on the console during the preparation of the simulation. If the boundary value file is incorrect, the simulation will be canceled with an error message during reading the boundary value file. In this case the user has to correct the boundary value file and start the simulation again.

Graphic see german manual

After all files and parameters are read accurately the simulation starts, which is indicated on the console by the prompt *Start Simulation*.

Graphic see german manual

The calculated result sets are written into the result file specified in the XML file and can be visualized and interpreted with the postprocessor DAVIT.

Graphic see german manual

## 6 Appendix I: File descriptions

### 6.1 Sysdat format

System.dat is a text file and contains the computational grid in TICAD-format. In principle the system.dat file contains two parts, the set of node and element lists. Rows starting with *C* are comment lines and are ignored.

The node list contains the number of boundary and area nodes. Boundary nodes are the first nodes in the list and describe a polygon that is numbered counterclockwise. After that other nodes follow. The total number of nodes is the sum of all boundary and area nodes.

```
C Anzahl der Randknoten (number of boundary nodes)
```

```
35
```

```
C Anzahl der Gebietsknoten (number of area nodes)
```

```
151
```

After defining the number of nodes, then follows the list of nodes with each identifying number, x- and y-coordinate, and depth. Space characters or tabs are used to separate the values. Node numbers are used in the element list, then the x- and y-coordinates are read and saved in the result file as floating numbers. The Gauss-Krueger-coordinates (or UTM-coordinates) should be decreased by the highest non-relevant places (with the preprocessor Janet a coordinate transformation for all nodes of a grid can be accomplished in one step). The scalar value of the depth is defined as positive pointing downward relative to sea level.

```
C Koordinaten und Skalarwerte der Knoten (coordinates and  
scalar values of nodes)
```

```
C -----
```

```
C zuerst die Randknoten (Anzahl s.o.) (boundary nodes first  
(number see above))
```

```
C dann die Gebietsknoten (Anzahl s.o.) (then area nodes  
(number see above))
```

```

C -----
C   Nr. | x-Koord. | y-Koord. | Skalarwert (scalar value)
      ...      .....      ....      ...

```

The node list follows the block "element list", consisting of the number of elements and a list of the elements themselves.

Each element consists of three nodes numbered counterclockwise. The fourth value is an identifier which determines which edge of the triangle is closed. Closing of element edges can be used to define closed boundaries as well as for modeling structures e.g. sheet pile walls or groins and jetties in an area.

The identifier is realized by bit combination (summation of the numbers) (see Sysdat.java):

Bit no. 0 (1) stands for edge j-k, closed

bit no. 1 (2) stands for edges k-i, closed and

bit no. 2 (4) stands for edge i-j, closed.

```

C
-----
C Anzahl der Elemente: (number of elements)
      324
C Elementverzeichnis (element list)
C
-----
C Knoten i   knoten j   Knoten k   Kennung   Nr
      (node i node j node k identifier No)
      ....      ....      ....      ....      .....

```

## 6.2 Rndwerte format (boundary values)

Rndwerte.baw is an ASCII file that determines the boundary value steering of depth-integrated models within MARINA. This file format is based on the TICAD system, which was developed at the University of Hanover (Germany) and adjusted by the BAW-DH (Federal Waterways Engineering and Research Institute, Hamburg/Germany).

(see

[http://www.baw.de/vip/en/departments/department\\_k/publications/fkb/rndw/rndw-en.html](http://www.baw.de/vip/en/departments/department_k/publications/fkb/rndw/rndw-en.html))

Below, a complete boundary value file is given. The meaning of each line is subsequently described in detail with an example file. Lines starting with C are comment lines and will be ignored.

```
C Gesamtzahl Knoten fuer die Randwerte anzugeben sind
  (total number of nodes which require boundary values)
```

```
15
```

```
C Zeitintervall (sec), fuer das Randwerte aufbereitet
werden
```

```
(time interval (sec) for which boundary values are processed)
```

```
44700
```

```
C IDRB/ZWEIG/ITYP/INTER/INTPER/SUMMA/FAKTZ/FAKTH
```

```
C 5 3 2 8 0 0. 1. 1.
```

```
C Knotennummern mit identischen Randbedingungen
```

```
(node numbers with identical boundary conditions)
```

```
0 1 2 3 4
```

```
C Zeitreihe beginnt bei t in s
```

```
(time series starts at t in s)
```

```
.... ....
```

```
..... .....
```

```
C IDRB/ZWEIG/ITYP/INTER/INTPER/SUMMA/FAKTZ/FAKTH
```

```
C 10 3 2 1 0 0. 1. 1.
```

```
C Knotennummern mit identischen Randbedingungen
```

```
(node numbers with identical boundary conditions)
```

```
5 6 7 8 9
```

```
10 11 12 13 14
```

```
C Zeitreihe beginnt bei t in s
```

```
(time series starts at t in s)
```

```
.... ....
```

Each file starts with the number of distinct boundary value time series. Note that each node is counted several times, when time series with different boundary conditions (e.g.  $v_x$ ,  $v_y$  and  $h$ ) are defined for this node.

```
C Gesamtzahl Knoten fuer die Randwerte anzugeben sind
  (total number of nodes that require boundary values)
```

```
15
```

The second number of the boundary value file specifies the time interval in seconds for which boundary values are processed. This information must be present in the file, but will not be processed by MARINA, because the time information is implicitly calculated and taken into account from start and end times of the simulation.

```
C Zeitintervall (sec), fuer das Randwerte aufbereitet
  werden
```

```
(time interval (sec) for which boundary values are processed)
```

```
44700
```

The following blocks show examples in which nodes for time series are defined.

```
C IDRB/ZWEIG/ITYP/INTER/INTPER/SUMMA/FAKTZ/FAKTH
```

```
C 5 3 2 8 0 0. 1. 1.
```

```
C Knotennummern mit identischen Randbedingungen
```

```
(node numbers with identical boundary conditions)
```

```
0 1 2 3 4
```

```
C Zeitreihe beginnt bei t in s
```

```
(time series starts at t in s)
```

```
.... ....
```

In the following, the purpose of each block is explained with an example file, before going into a detailed explanation of the individual numbers. The total number (15) relates to the sum of all time series nodes of the different blocks (5+10).

```
C Gesamtzahl Knoten fuer die Randwerte anzugeben sind
  (total number of nodes which require boundary values)
```

```
15
```

C Zeitintervall (sec), fuer das Randwerte aufbereitet werden

(time interval (sec) for which boundary values are processed)

44700

C IDRB/ZWEIG/ITYP/INTER/INTPER/SUMMA/FAKTZ/FAKTH

C 5 3 2 8 0 0. 1. 1.

C Knotennummern mit identischen Randbedingungen

(node numbers with identical boundary conditions)

0 1 2 3 4

C Zeitreihe beginnt bei t in s

(time series starts at t in s)

.... ....

C IDRB/ZWEIG/ITYP/INTER/INTPER/SUMMA/FAKTZ/FAKTH

C 10 3 2 1 0 0. 1. 1.

C Knotennummern mit identischen Randbedingungen

(node numbers with identical boundary conditions)

5 6 7 8 9

10 11 12 13 14

C Zeitreihe beginnt bei t in s

(time series starts at t in s)

.... ....

Within a block used for time series definitions, the first number (10) stands for the sum of all specified node numbers (5-14).

C IDRB/ZWEIG/ITYP/INTER/INTPER/SUMMA/FAKTZ/FAKTH

C 10 3 2 1 0 0. 1. 1.

C Knotennummern mit identischen Randbedingungen

(node numbers with identical boundary conditions)

5 6 7 8 9

10 11 12 13 14

The fourth number (1) states how many time series entries are part of the first time series record.

```

C IDRB/ZWEIG/ITYP/INTER/INTPER/SUMMA/FAKTZ/FAKTH
C 10 3 2 1 0 0. 1. 1.
C Knotennummern mit identischen Randbedingungen
  (node numbers with identical boundary conditions)
      5 6 7 8 9
     10 11 12 13 14
C Zeitreihe beginnt bei t in s
  (time series starts at t in s)
      0.0 0.605
      44700.0 0.605

```

When generating a Rndwerte.dat file, one should be careful that the syntax described above is followed consistently, otherwise the simulation will not start.

After describing the syntax of each block, a detailed explanation of the time series definition follows.

```

C IDRB/ZWEIG/ITYP/INTER/INTPER/SUMMA/FAKTZ/FAKTH
C 5 3 2 8 0 0. 1. 1.

```

The first line provides the following information:

IDRB: number of nodes with identical boundary conditions / integer

ZWEIG: boundary condition type / integer

ITYP: interpolation type is always 2 / integer

INTER: number of specified intervals / integer

INPER: number of intervals to be generated / integer

(not used)

SUMMA: / double (not used)

FAKTZ: / double (not used)

FAKTH: / double (not used)

To separate the values space characters or Tabs can be used.

### **boundary condition type (ZWEIG) / integer:**

1. specific discharge ( $q_x$ ) in x-direction (currentModel2D)

2. specific discharge ( $q_y$ ) in y-direction (currentModel2D)
3. water level (positive upwards above sea level) (currentModel2D/3D)
4. salinity (SaltTransportModel2D)
5. water level (on inner grid nodes) (not used)
6. temperature (HeatTransportModel2D)
7. heat quantity (not used)
8. suspended matter concentration in [mg/l] (Sedimentmodel2D)
9. suspended matter quantity (not used)
10. weir height (U-spot) (not used)
11. weir height (V-spot) (not used)
12. velocity in x-direction in [m/s] (currentModel2D/3D)
13. velocity in y-direction in [m/s] (currentModel2D/3D)
14. discharge ( $Q_x$ ) in x-direction in [m<sup>3</sup>/s] (currentModel2D)
15. discharge ( $Q_y$ ) in y-direction in [m<sup>3</sup>/s] (currentModel2D)

All boundary condition types, ZWEIG 1-13, relate to nodes and can be defined for one or more nodes at the same time.

#### *Q-steering:*

The boundary condition types 14 and 25 are related to the cross section, where at least two nodes need to be defined to get an accurate evaluation. . Velocity boundary conditions are generated dynamically (iterative) at the defined nodes, so that the discharge through the cross section (limited by the bottom and the water surface) matches the given discharge in m<sup>3</sup>/s.

#### **Interpolation method for the time function (ITYP):**

1. sum of several cosine functions (not used)
2. linear interpolation between non equidistant nodes
3. cubic spline interpolation between non equidistant nodes (not used)

In the model system only the linear interpolation is implemented. If another number given, an error message while reading the file is shown and linear interpolation takes place internally.

In the next line, nodes are specified for which the following time series applies.

```
C Knotennummern mit identischen Randbedingungen
(node numbers with identical boundary conditions)
```

```
0 1 2 3 4
```

Thereafter the time series data follows (here the number is INTER+1)

```
C Zeitreihe beginnt bei t in s
(time series starts at t in s)
```

```
... ...
... ...
```

### 6.3 StricklerDat

To describe space-variable bottom friction in the depth-integrated and three dimensional current model, Strickler coefficients can be optionally defined. The strickler.dat file is an ASCII file with the same format as system.dat (TICAD-format). Instead of depth values in the nodes list, Strickler coefficients are given in the strickler.dat file.

The only check during reading this file is, if the number of nodes are consistent with the number of nodes in the system.dat file. The element list is not read again at this stage.

**Default value:** If no strickler.dat file is read, a constant Strickler coefficient of 48 is used in the entire area.

### 6.4 WaterLevelDat

An initial water level condition can be optionally given with an ASCII file as an initial condition for the three-dimensional depth-integrated current model. Waterlevel.dat is a text file with the same format as system.dat (TICAD-format). Instead of depth values in the nodes list, water level conditions are given in the waterlevel.dat file. The water level is defined as positive upwards related to sea level.

The only check during reading this file is, if the number of nodes are consistent with the number of nodes in the system.dat file. The element list is not read again at this stage.

## 6.5 WindDat

Wind.dat is a text file that contains time series data for wind at a height of 10 m. This file is used to generate meteorological model data to describe spatially-constant and time-variable wind fields over the area.

In the first line of the wind.dat file the number of nodes of the subsequent time series stands (different from the rndwerte.dat file, where the number of intervals is given). Thereafter the nodes of the wind time series follow with time,  $w_x$  and  $w_y$ . To separate the values space characters or Tabs can be used.

Note: Wind directions are oriented mathematically not meteorologically.

...  
...  
...

## 6.6 MaxErosion

To define erodible horizons in the sediment transport model, an optional ASCII file can be used. Maxerosion.dat is a text file with the same format as system.dat file (TICAD-format). Instead of depth values in the nodes list, erodible depths based on the depths in the computational grid are given. An erodible depth of zero means that erosion below the depth of the computational grid at this location is not allowed. Sedimentation with subsequent erosion above the depth  $z$  is possible.

**Default value:** If no maxerosion.dat is read, an erodible depth of 100 m is used in the entire area.

## 6.7 WaveRandnDat

The definition of boundary conditions for the wave model is done by an ASCII-format file called WaveRandnDat.

.....  
.....  
.....  
.....

The time series data for this file include wave height, direction and period. The wave direction is described in degrees and mathematical orientation.

## 6.8 Marina XML format

### 6.8.1 Introduction to XML

Before we go into detail about the Marina XML file, some basic concepts of the XML format are explained first.

Elements are a fundamental part of an XML document. They determine the structure of an XML document and can contain further elements. The element which stands highest in the hierarchy is called root element. In each document only one root element is allowed.

An element consists of an opening tag, optional content and a closing tag. The opening tag consists of a less than sign "<", the name of the element, which cannot contain a spaces and whose first character has to be an alphabetic character, and lastly a greater than sign ">".

The closing tag consists of a less than sign "<", a forward slash "/", the name of the element followed by a greater than sign ">".

```
<Projectname> my first project </Projectname>
```

It is possible to include comments in the XML file. In general they are ignored while the data is processed, like in any other programming language, and serve exclusively to provide remarks and notes. A comment consist of a less than sign "<", an exclamation mark "!", two minus signs "--" and the content of the comment. This if followed by two minus signs "--" and a greater than sign ">".

```
<!-- This is a comment -->
```

Comments are particularly convenient, if you do not want to use single model modules, without deleting the respective steering elements from the Marina XML file.

Structuring of the XML file is accomplished through so called kind elements, which are located between the opening and closing tag of an element.

### 6.8.2 Structure of the Marina XML file

A well shaped XML document for steering Marina consists of the root element *<Marina>* of the type "MarinaType". Because the root element is ranked highest in the hierarchy of the XML document, it has to be the first opened and last closed tag in the document.

By definition, the root element `<Marina>` has to contain the kind elements `<ControlParameter>` and `<Configuration>`. The block `<ControlParameter>` specifies parameters for steering the coupled models. The block `<Configuration>` serves the model configuration.

With the following listed optional kind elements, additional information can be stored in the XML file:

`<Description>` - additional description

`<Projectname>` - name of the project

`<Author>` - name of the author

`<Date>` - date

Example:

see [german manual](#)

...

....

...

A complete specification of all permitted elements and their hierarchy can be found in Appendix II.

## 7. Appendix II: Overview of XML elements

As described in section 6.8, an XML element consists of exactly one root element (here `<Marina>` of the type "MarinaType") and several kind elements with an arbitrary complex structure.

The structure of the Marina XML document is defined in a so-called XML scheme file (Marina.xsd). The XML scheme distinguishes between simple and complex data types. A type indicates in the following an abstract definition of the element structure.

Simple types are e.g. `string` or `int`. As opposed to simple types, the complex types offer the possibility to define kind elements of an element (example: `ControlParameterType`).

Kind elements can be combined in several ways. The valid values are defined in the scheme file and are listed in the following table in the middle column (cardinality):

- a kind element can appear 0 or 2x [0..1]

- a kind element has to appear exactly once [1..1]

If an element from a list has to be chosen exactly once, this is marked in the table with the word "choice".

## 7.1 Marina

Marina		
Kind element	Cardinality	Type
...	...	...

## 7.2 ControlParameter

...  
...

### 7.2.1 SystemTriangleMeshType

...

## 7.3 Configuration

...

### 7.3.1 CurrentModel2D

...

#### 7.3.1.1 *BoundaryCondition*

...

#### 7.3.1.2 *BottomFrictionTriangleMesh*

...

#### 7.3.1.3 *InitialCondition*

...

#### 7.3.1.4 *ResultFile*

...

....

....

### **7.3.3 CurrentModel3D**

...

...

The kind elements listed for CurrentModel3D conform to the kind elements of the respective hierarchy level of the element CurrentModel2D. A description of those kind elements (e.g. Current2DBoundarFile) can be found in chapter 7.3.1.x.

### **7.3.4 MeteorologyCondition**

No kind elements exist.

...

...

#### ***7.3.5.2 InitialCondition and ResultFile***

Description of kind elements see chapter 7.3.1.x.

...

...

#### ***7.3.6.2 InitialCondition and ResultFile***

Description of kind elements see chapter 7.3.1.x.

...

...

#### ***7.3.7.2 InitialCondition and ResultFile***

Description of kind elements see chapter 7.3.1.x.