This file contains the original source contributions of Dipl.-Ing. S. Rath (s.rath@tuhh.de) for the DAAD E-Learning Platform, based at http://134.28.87.45/typo3/index.php

This text contains editable formulas! Several configurations do not confirm the specifications of the E-Learning platform, but they serve as basic summary, e.g. to supply a PDF download. Partly, the images in this document are too small to serve detailed information, on the web they can be visualised in full extent.

**2D-Simulations**

Two-dimensional steady and unsteady flood and tide simulations describe the spatial motion of water bodies together with their flow pattern in coastal waters, estuaries, rivers and floodplains. These domains typically raise flow patterns being essentially 3D in nature. If a logarithmic velocity distribution can be assumed, 2D depth integrated models are a common simplification of the complex flow pattern. They permit efficient computations and faster feedback to engineering questions. Usually these domains cannot be represent using a 1D network model without a dramatic loss of information and accuracy. 2D models are the typical choice to model complex spatial distributions of the water surface level, that can be considered as response surface to its domain. Hence typical applications comprise amongst others flood extent modelling, considerations within the vicinity of structural encroachments, overtopping of a levee or the determination of risk due to flood in urban and industrial areas close to rivers. For that purpose, 2D models are frequently coupled wit 1D models, exemplarily serving to model flooding of river systems in conjunction with complex overland flow and to consider simultaneously piped urban flows.

*Animated GIF: Inundation extents in Bangladesh for dry and wet season (January / September). Spaceborne Landsat Imagery.*
1 Introduction

Hydrodynamic computations in river hydraulics use the Shallow Water Equations (SWE) to model open channel flow. The SWE are partial differential equations, being derived from the 3D Navier-Stokes equations (3D-NSE). Remember: partial differential equations (PDEs) involve at least two variables in space (boundary value problems) or time (initial value problems). The SWEs are commonly used to approximate the water depth (h) and the spatial pattern of the velocity field (u, v) and most commonly describe more than one time instance, but an entire hydrograph.

This approximation within the SWEs is a simplification, introducing time averaging and depth integration. The 2D model theory in river hydraulics assumes that vertical velocity components are negligible. Consequently, a hydrostatic pressure distribution is presumed. This simplification comes along with a reduction in computational costs afforded by the SWE approach. This is particularly important for ocean and estuary simulations, usually involving huge domains and a serious time span for a hydrodynamic simulation period, resulting in a large number of time steps. Calculations of complex flow situations such as caused by interactions with building structures or in urban areas show differences between 3D-NSE and SWE. They raise vertical velocity components that are generally not negligible.

Vast flood plains in global and local perspective in Bangladesh,

local flood phenomena in urban area for River Rhine in Germany.
1.1 Floodplain Processes

The variety of natural processes being encountered in a floodplain are exemplarily referred in the schematic figure of Anderson et al. below. These processes, if they are to be represented in a numerical model, require representations in terms of physical laws. If these are not available, empirical data may serve for the description of the responds pattern, e.g. erosion as consequence of intensive flood flow.

1.2 Roots of 2D Hydroinformatics

To introduce into the theory of numerical modelling and to motivate the reader, a concise genesis of the equations and fundamental ideas in provided by selected statements of important scientists.

Leonardo da Vinci [1452-1519]: ‘… First of all I shall make a few experiments and then demonstrate why bodies are forced to act in the manner they do … Nature begins by reasoning and ends by experience; but nevertheless, we must take the opposite route: we must begin with experiments and try out through it to discover the reason …’

Benedetto Castelli [1577-1644]: ‘… Sections of the same river discharge equal quantities of water in equal times, even if the section themselves are unequal. Given two sections of a river, the ratio of the quantity of water passing the first section is in proportion to that passing the second section …‘

René Descartes [1596-1650]: ‘… I assume that the movement which is once imposed on a given body is permanently retained, if it is no removed by some other course, i.e. whatever started to move in a vacuum continues to move indefinitely at the same velocity … ’
The Further development of hydrodynamic science is briefly tabularised referring to the most important contributions, that should be well known to the interested reader.

The fundamental basis for mathematical modelling in hydrodynamics date back to the 17th and 18th century.

| **Blaise Pascal [1623-1662]** | Pressure concepts in fluids  
|                               | Image Source:  
|                               | http://paedpsych.jk.unilinz.ac.at/INTERNET/ARBEITSBLAETTERORD/PHILOSOPHIEORD/PersonPhil1.html |
| **Christian Huygens [1629-1695]** | Pressure transmission equal in all directions  
|                               | Image Source:  
|                               | http://www.astromia.com/biografias/fotos/huygens.jpg |
| **Isaac Newton [1642-1727]** | Principle of centrifugal force  
|                               | Principle of energy conservation  
|                               | Image Source:  
| **G. W. v. Leibniz [1646-1717]** | 3 laws of motion  
|                               | Concise definition of mass, momentum, inertia and force  
|                               | Resistance between fluid and solid ~ velocity |
Definition of kinetic energy

Image Source:

http://www.cronologia.it/storia/biografie/leibniz.jpg
The first roots for mathematical modelling in hydrodynamics date back to the early 18th century.

Daniel Bernoulli [1700-1782]
Fluid static’s and dynamics


Leonard Euler [1707-1782]
Rigorously deriving the Bernoulli equations & their differential forms of continuity and acceleration for frictionless fluids

Today’s concepts for hydronumeric simulations have been raised in the 19th century:

| Research in Resistance Laws | Darcy, Weisbach, Poiseuille |
| Research in Viscosity       | Saint Vernant, Cauchy, Stokes |
| Research in Turbulence      | Reynolds, Prandtl, v. Kármán |

Today’s concepts for sediment transport mainly denote to following researchers:

| Albert Brahm:               | Relationship between bedflow velocity and 1/6 power of the immersed weight of bed material. |
| Paul Francois Dominique du Boys | Concept of tractive force and the relationship to bed-sediment transport. |
| Leo van Rijn                | Prediction for occurrences of dune and transition regions (1984) |
| Selim Yalin                 | Regime theory for alluvial channels |
The Shallow Water Equations are the basis for 2D hydroinformatics and based on the 3D Navier-Stokes (3D-NSE) equations, which describe compressible fluid motion as conservation of

mass,
momentum and
energy.

This description in terms of non-linear partial differential equations are the 3D-NSE equations. Their solution expresses the velocity distribution within a fluid body as function of space and time

\[ \vec{v} = f(\vec{s}, t) \]  

(1.1)

Today, mathematics does not provide a means for direct solution the 3D-NSE for river hydraulics and most industrial flows. Hence, it is required to approximate the equations to obtain numerical solutions. Established solution techniques for the 3D-NSE are:

Direct Numerical Simulation (DNS),
Large Eddy Simulation (LES),
Time-dependent Reynolds Averaged Navier Stokes equations (TRANS),
Reynolds Averaged Navier Stokes equations (RANS).

River Hydraulics assumes that water has constant density and is incompressible. Incompressible turbulent flows are governed by the conservation laws for mass and momentum, the Navier-Stokes equations.

Consequently, the mass conservation in the Navier-Stokes equations, i.e. the continuity equation, is simplified while introducing

\[ \frac{\partial \rho}{\partial t} + \frac{\partial \rho}{\partial x} + \frac{\partial \rho}{\partial y} + \frac{\partial \rho}{\partial z} = 0, \]  

(1.2)

while aiming to express continuity in an incompressible flow. Considering a control volume \(dV\) bounded by a surface \(dS\), the inlet mass flux through \(dS\) equals the outlet mass flux through \(dS\). This condition is expressed via

\[ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0. \]  

(1.3)

Generally, the NSE considers three forces acting on the water body, being
Weight: \( \tilde{F}_G = \Delta_m \tilde{g} \), \hspace{1cm} (1.4)

Pressure: \( \tilde{F}_p = -\nabla p \Delta V \) and \hspace{1cm} (1.5)

Friction: \( \tilde{F}_r = \eta \nabla^2 \tilde{v} \Delta V \). \hspace{1cm} (1.6)

The balance of these forces is described as

\[ \tilde{F}_{\text{total}} = \Delta_m \tilde{g} - \nabla p \Delta V + \eta \nabla^2 \tilde{v} \Delta V. \] \hspace{1cm} (1.7)

Using the acceleration to describe these forces

\[ \Delta_m \ddot{a} = \Delta_m \tilde{g} - \nabla p \Delta V + \eta \nabla^2 \tilde{v} \Delta V, \] \hspace{1cm} (1.8)

following vector form can be derived to express the 3D-NSE:

\[ \rho \frac{d\tilde{v}}{dt} = \rho \tilde{g} - \nabla p + \eta \nabla^2 \tilde{v}. \] \hspace{1cm} (1.9)

3D Navier-Stokes equations implicitly include mass conservation through the continuity equation. Energy conservation, known as first law of thermodynamics, constitutes that all forces acting on a water particle are balanced. The conservation of momentum, known as Newton’s second law, imposes a balance of the momentum of a water body with the external forces acting on it from outside.

The momentum for incompressible fluids as expressed in the Navier Stokes equations is referred for Cartesian coordinates in their dimensional, non-conservative form by

\[ \rho \left( \frac{\partial \tilde{u}}{\partial t} + u \frac{\partial \tilde{u}}{\partial x} + v \frac{\partial \tilde{u}}{\partial y} + w \frac{\partial \tilde{u}}{\partial z} \right) = \tilde{F}_x - \frac{\partial p}{\partial x} + \eta \left( \frac{\partial^2 \tilde{u}}{\partial x^2} + \frac{\partial^2 \tilde{u}}{\partial y^2} + \frac{\partial^2 \tilde{u}}{\partial z^2} \right), \hspace{1cm} (1.10) \]

\[ \rho \left( \frac{\partial \tilde{v}}{\partial t} + u \frac{\partial \tilde{v}}{\partial x} + v \frac{\partial \tilde{v}}{\partial y} + w \frac{\partial \tilde{v}}{\partial z} \right) = \tilde{F}_y - \frac{\partial p}{\partial y} + \eta \left( \frac{\partial^2 \tilde{v}}{\partial x^2} + \frac{\partial^2 \tilde{v}}{\partial y^2} + \frac{\partial^2 \tilde{v}}{\partial z^2} \right), \hspace{1cm} (1.11) \]

\[ \rho \left( \frac{\partial \tilde{w}}{\partial t} + u \frac{\partial \tilde{w}}{\partial x} + v \frac{\partial \tilde{w}}{\partial y} + w \frac{\partial \tilde{w}}{\partial z} \right) = \tilde{F}_z - \frac{\partial p}{\partial z} + \eta \left( \frac{\partial^2 \tilde{w}}{\partial x^2} + \frac{\partial^2 \tilde{w}}{\partial y^2} + \frac{\partial^2 \tilde{w}}{\partial z^2} \right). \hspace{1cm} (1.12) \]

where \( \eta \) is the kinematic viscosity and \( F \) summarizes the forces acting on the volume of fluid. This notation introduced the Stokes' hypothesis, assuming that viscous stresses are related to the rate of strain of a fluid element.
The Navier-Stokes equations are valid under the postulation that the density is constant or if the Boussinesq approximation applies. The Boussinesq approximation states that for small density variations may be approximated by a constant in all terms except the gravitational term.

Hydrodynamic simulations generally consider gravitation, tide and Coriolis effects to represent the force $F$ in equations (1.11) - (1.12).

The non-conservative character of this form denotes the fact that frictional force continually drain energy from the water body as it moves over the surface. This energy is dissipated as heat loss to the system. Friction is a non-conservative, energy dissipating force. Thermodynamics claims that energy being converted once into heat from being converted back to its original form. Considering the internal friction of the fluid, the last term on the right side of equations (1.11) - (1.12) express the difference between the 3D-NSE and the Euler equations. This term includes the dynamic viscosity $\eta$, often simply referred to as viscosity or as absolute viscosity. The International System of Units (SI) has had very little international impact on viscosity (Elert, 2005). The pascal second [Pa·s], referring to [kgm$^{-2}$s$^{-1}$], is hardly ever used in scientific and technical publications today. The most common unit of viscosity is the dyne second per square centimeter [dyne·s/cm$^2$], given the name poise [P] after the French physiologist Jean Louis Poiseuille. Ten poise equal one pascal second [Pa·s] making the centipoise [cP] and millipascal second [mPa·s] identical. The dynamic viscosity $\eta$ denotes the variation of density of water in dependency of temperature. For water and temperatures between 0 and 20°C Celsius it is expressed as empirical formula in centipoises (Weast, 1981)

$$\log \eta = \frac{1301}{998.333 + 8.155(T - 20) + 0.00585(T - 20)^2}$$ (1.13)

This dynamic viscosity relates to the fluid density and the kinematic viscosity $\nu$ via

$$\eta = \rho \nu.$$ (1.14)

The kinematic viscosity $\nu$ is a measure of the resistive flow of a fluid under the influence of gravity, frequently being measured using a capillary viscometer. It is expressed in SI unit of square meter per second [m$^2$/s]. This unit is that large that it is rarely used. A common unit of kinematic viscosity is the square centimeter per second [cm$^2$/s], denoting the CGS-unit system. This common unit is given the name stoke [St] after the English scientist George Stoke. As this unit is often considered as being too large, the most common unit can be considered as the square millimeter per second [mm$^2$/s] or centistoke [cSt].
2 Theory

Your motivation to follow the theoretical part of this course are reflected in two scenarios below: inland and coastal flood may affect our lives and belongings. As engineer you must understand the fundamental basics of a flood event before attempting to use simulation models. The challenges wait for you, now proceed with the following three parts. They will explain the Shallow Water Equations (SWEs), the need for initial and boundary conditions to describe flood scenarios and the methods being applied to solve the SWEs.
2.1 Depth-averaged Shallow Water Equations

The Navier-Stokes equations are used to model water flows in many technical and engineering applications. However, when considering engineering tasks in catchment management, we commonly face horizontal scales exceeding by far the vertical scale. Consequently, a depth-integrated formulation of the three-dimensional shallow-water equations is frequently considered to be sufficient. River engineering, flood risk assessment and hazard assessment using 2D hydronumeric simulations commonly rely on the shallow water equations (SWEs) and their depth-integrated, Reynolds averaged form. The central assumptions of the SWEs basically consider a logarithmic velocity profile and a hydrostatic pressure distribution. Hence, they neglect the vertical velocity and accelerating forces. Consequently they are incomplete in the description of secondary circulation, but generally applicable typical river systems (w>>d). Due to their incomplete representation of secondary circulations, the SWEs are to be used with caution at high sinuosity’s in meandering rivers.

Bankfull discharge and logarithmic velocity distribution
An intuitive formulation of the SWEs is provided in compact vector notation according to Nujić (2003) and Schwanenberg (2003):

$$\frac{\partial \mathbf{W}}{\partial t} + \frac{\partial \mathbf{f}}{\partial x} + \frac{\partial \mathbf{g}}{\partial y} \cdot \mathbf{S} = 0$$

where

$$\mathbf{W} = \begin{bmatrix} H \\ \mathbf{u}_h \\ \mathbf{v}_h \end{bmatrix}, \quad \mathbf{f} = \begin{bmatrix} \mathbf{u}_h \\ \mathbf{u}_h^2 \mathbf{h} + 0.5 \mathbf{g} \mathbf{h}^2 - \mathbf{v}_h \frac{\partial \mathbf{u}}{\partial x} \\ \mathbf{u}_h \mathbf{v}_h - \mathbf{v}_h \frac{\partial \mathbf{v}}{\partial x} \end{bmatrix}, \quad \mathbf{S} = \begin{bmatrix} 0 \\ \mathbf{g} h (I_{\text{fric},x} - I_{\text{bottom},x}) \\ \mathbf{g} h (I_{\text{fric},y} - I_{\text{bottom},y}) \end{bmatrix}, \quad \mathbf{g} = \begin{bmatrix} \mathbf{v}_h \\ \mathbf{u}_h \mathbf{v}_h - \mathbf{v}_h \frac{\partial \mathbf{u}}{\partial x} \\ \mathbf{v}_h^2 \mathbf{h} + 0.5 \mathbf{g} \mathbf{h}^2 - \mathbf{v}_h \frac{\partial \mathbf{v}}{\partial y} \end{bmatrix}$$

and

$$\mathbf{W} = \text{conservative terms for mass & momentum}$$

$$\mathbf{F}, \mathbf{G} = \text{flux terms for mass & momentum incl. diffusive terms}$$

$$\mathbf{S} = \text{source term incl. friction- and bottom slope}$$

For the SWEs, the friction slope is introduced via the Darcy-Weisbach resistance law:

$$I_{\text{fric, bottom}} = \frac{\lambda_{\text{bottom}}}{d_{\text{hy}}} Q^2 \frac{1}{A^2} \approx \frac{\lambda_{\text{bottom}} Q^2}{A^2} \frac{1}{8g}$$

Here, the resistance coefficient $\lambda$ is introduced according to Colebrook-White, as being referred by Marchi (1961):

$$\frac{1}{\sqrt{\lambda_{\text{bottom}}}} = -2.03 \cdot \log \left( \frac{2.51}{\text{f} \cdot \text{Re} \sqrt{\lambda}} + \frac{k_s}{\text{f} \cdot 14.84 \cdot n_{\text{hy}}} \right)$$

A more detailed formulation of the momentum equations in the SWEs represents the flow field while representing turbulent fluctuations via the Reynolds stresses within the friction terms. Generally, the Reynolds stresses are the dominant of all stress components.
Due to turbulent flow regimes in fluvial domains eddies occur. They raise small variations in the flow velocities and pressure. Regularly these variations are too little to be represented in numerical schemes unless the discrete resolution of the numerical domain is chosen very fine.

\[
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + \frac{g}{\rho} \frac{\partial z}{\partial y} = \frac{1}{h} \frac{\partial}{\partial x} \left( h \frac{\tau_{xu}}{\rho} \right) + \frac{1}{h} \frac{\partial}{\partial y} \left( h \frac{\tau_{yu}}{\rho} \right) - \frac{1}{h} \frac{\tau_{xh}}{\rho} + \frac{1}{h} \frac{\tau_{yh}}{\rho} + \frac{2\alpha n \sin \Phi}{h \rho}
\]

(1.15)

Due to turbulent flow regimes in fluvial domains eddies occur. They raise small variations in the flow velocities and pressure. Regularly these variations are too little to be represented in numerical schemes unless the discrete resolution of the numerical domain is chosen very fine.

A turbulent flow is characterized by following properties (Ferziger, 1999):

Turbulence processes are highly unsteady, e.g. the flow velocity at a given point is subject to great variance over time.

Turbulence is a three-dimensional phenomenon.

Vortices are an essential part of flow, and the interaction of vortices and the so-called vortex stretching are basic mechanisms increasing and widening turbulent structures.

Turbulence often multiplies the mixing processes due to general diffusion.
Due to intensive mixing processes, the contact between fluid volumes with small and large motion moment increases. Subsequently, the acting viscous forces come along with a loss of kinetic energy. This loss is translated into internal energy, i.e. heat energy. Hence, this process is irreversible and dissipative.

Research confirms the occurrence of so-called coherent structures. These reproducible and deterministic processes have a significant influence on the mixing processes.

When determining the total shear stress (1.16), time averaging (-) of the velocity fluctuations (') is introduced.

$$\tau_{ij} = \mu \frac{\partial u_i}{\partial x_j} - \rho u_i' u_j'$$ \hspace{0.5cm} (1.16)

Comparing this expression of the total shear stress with its representation in the Navier-Stokes equations reveals that additionally to the viscous part (left) another term is introduced. The right term results from the time-averaging. Generally it is the dominant part of the total shear stress. It is called Reynolds stress or apparent turbulent shear stress.

The SWE-system can only be closed if the Reynolds stresses are approximated as function of the velocity field of the main stream. Here, the eddy viscosity concept is introduced, providing the basis for many turbulence models used in practice. Stated by Boussinesq in 1877, this concept assumes in analogy to the viscous shear stress in laminar flows that the apparent turbulent shear stress is proportional to the velocity gradients of the main flow. This can be written as

$$-u'_i u'_j = \nu_T \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij}$$ \hspace{0.5cm} (1.17)

The Kronecker delta $\delta_{ij}$ ensures that the equation is also valid for normal stresses. With $k$ being the turbulent kinetic energy defined as

$$k = \frac{1}{2} \left( \overline{u'^2} + \overline{v'^2} + \overline{w'^2} \right)$$ \hspace{0.5cm} (1.18)

The so-called eddy viscosity $\nu_T$ is no material constant, but denotes a factor of proportionality, primarily depending on the degree of turbulence, i.e. $\nu_T$ varies inside the flow and depends on the flow state (sub-/supercritical).

The eddy viscosity concept is not a turbulence model in the original sense, it merely provides the basic framework for a hydronumeric model. Turbulence models are approximations, providing a means of calculating the eddy viscosity $\nu_T$. 

[SR, ABWB, TUHH]  Page 16 of 44
2.2 Initial and Boundary Conditions

Hydrodynamic simulations distinguish following boundary conditions to model the movement of water in river systems:

Initial boundary conditions,

Bottom boundary condition,

Free surface boundary condition,

Lateral boundary conditions,

Hydrodynamic boundary conditions.

From the mathematical point of view and the problem under consideration, the initial condition is formulated via

\[ f(x,t) = f_0(x), \quad \forall x \in \Omega, \quad t = 0 \]

and or boundary conditions need to be imposed to retrieve a solution within the model domain. Boundary conditions are often of the form

\[ a f + b \frac{\partial f}{\partial x} = c, \quad \forall x \in \partial \Omega, \quad \forall t \]

These boundary conditions are called Dirichlet condition if \( a=0 \), Neumann if \( b=0 \) or Robin if \( c=0 \).

This chapter focuses merely on the modelling of water and does not consider mixed transport processes, such as sediment transport, pollutant transport of biologic degradation rates, involving each further particular initial and boundary conditions. Other forms of boundary conditions contain the periodic condition or the outgoing-wave condition, if the domain is open.

2.2.1 Initial conditions

Initial conditions in river hydraulics characterise the physics of the river system at the very beginning of a simulation \( (t=0) \). Initial boundary condition for 2D models in river hydraulics introduce the water-surface elevation or the energy gradient downstream of the modelled river reach for a specified discharge, being supplied at the upstream model boundary.

Other hydrodynamic simulations, e.g. ocean tide simulations, impose initial boundary condition to the interface of ocean and land, assuming that the water level equals the ocean wave oscillation. This interface is referred for \( x=0 \) via
\[
\Phi = h(0,t) = H\left(1 + \alpha \cos \omega t\right) \quad (1.19)
\]

where \(\omega = \frac{2\pi}{T}\) is the wave frequency based on the period of the tide \(T\) and \(\alpha = \frac{A}{H}\) is a dimensionless amplitude parameter, representing the maximum tidal variation \(A\) to the average height of the water table \(H\).

### 2.2.2 Bottom boundary conditions

If not being coupled with groundwater models, river hydraulic models assume that no vertical flow occurs at the impermeable bottom of the bathymetry. The bottom friction is being modelled via the shear stress.

### 2.2.3 Free surface boundary conditions

Free surface boundary conditions within the domain may introduce wind shear stresses. Commonly negligible for small to medium size domains, huge coastal basins do significantly respond to the wind force. Following empirical approach might be used

\[
\begin{pmatrix}
\tau_{w,x}^+ \\
\tau_{w,x}^-
\end{pmatrix} = \begin{pmatrix}
-c_d \cdot \rho_{\text{air}} \cdot u_{\text{wind}} \cdot \cos \varphi \\
-c_d \cdot \rho_{\text{air}} \cdot u_{\text{wind}} \cdot \cos \varphi
\end{pmatrix} \quad (1.20)
\]

where the drag coefficient due to wind \(c_d\) is a function of the wind velocity, being reviewed from literature and software applications by Malcherek (2001). The model RMA2 generally offers several wind models, amongst them are the equations of Wu, Safaie, Ekman, Van-Dorn (Rath, 2001).

### 2.2.4 Lateral boundary conditions

Lateral boundary conditions are assumptions being imposed to model the behaviour of domain boundary segments without further specification of initial and boundary condition. For river systems, these are typically the model boundaries except inflow and outflow region. A typical assumption is that inundated lateral model boundaries do not suffer from any lack of mass, except when specified in terms of boundary conditions. Consequently, velocity vectors are modelled such that they follow the model boundary without exiting the domain, the so called slip condition.

Considering ocean tide simulations, a lateral boundary condition might assume that the tide influenced water level vanishes far away from the beach, as the water table fluctuation is only limited to regions near the coastline. Thus, for \(x \to \infty\) the gradient of the water level tends to zero.

The height of the free surface of a fluid above ground in a hydraulic system is a measure of the pressure or force exerted by the fluid within the considered domain. Whereas the free surface flow in
the internal domain commonly is unconstraint throughout the simulation, selected parts the domain boundaries are regularly used in 1D models to impose boundary conditions. These may impose boundary conditions, assigning a fixed head loss, e.g. via assigning that the energy loss equally the terrain slope.

2.2.5 Hydrodynamic boundary conditions

Hydrodynamic boundary conditions for hydrodynamic models are obtained from gauging stations, terrestrial and vessel based measurements of water levels and discharges. Boundary conditions for typical flow field simulations use an assignment of

- Measured (fixed) water levels \( h \) at a given time (‘h-Bound’),
- Discharge \( Q \) at a given time (‘Q-Bound’) or
- Water level as function of time (‘h-Q-Bound’).

A distinction for stage-discharge bounds, here referred to as ‘h-Q-Bounds’, should be considered. Typically, h-Q-Bounds are applied for river stream discharge. Alternatively, h-Q-Bounds are applied for dyke breaching scenarios, where the relation between water level and discharge needs to be described.

The stage discharge relation for river streams can be approximated for a closed and smooth description in hydrodynamic models via different regression equation, such as

- potential approximation: \( Q(t) = a \cdot h(t)^b \), \hspace{1cm} (1.21)
- exponential approximation: \( Q(t) = a \cdot e^{b\cdot h(t)} \), \hspace{1cm} (1.22)
- or polynomial approximation: \( Q(t) = a_0 + a_1 \cdot h(t) + ... + a_n h^n(t) \).\hspace{1cm} (1.23)

A typical non-breaching h-Q-Bound is illustrated below for River Stoer, located in the North German lowlands, showing a potential approximation of the measurements at the gauging station Willenschachen.
Potential approximation for non-breaching h-Q-Bound
The breaching $h$-$Q$-bound can be approximated by following relation (Vreugdenhil, 1989):

$$h - h_0 = \frac{q_n}{\sqrt{gH}}$$

where $h$ is the water level, $h_0$ is a reference elevation, $q_n$ is the discharge per unit width in a direction perpendicular to the downstream boundary and $H$ is the water depth.

A recent detailed study on boundary conditions for hydrodynamic models, their approximation, accuracy and pitfalls is given by Schmidt (2002).

### 2.2.6 Implementation in RMA2

To model the flow in a closed domain, boundary conditions must be applied, exemplarily demonstrated for the hydrodynamic model RMA2. Considering a short reach of River Stör in the North German Lowlands, merely an inflow and an outflow section must be described. For long river reaches, frequently tributaries must be considered, hence additional hydrodynamic boundary conditions must be adopted to the model describing these flow regimes.

Considering this simple scenario, following boundary conditions can be imposed for a steady discharge simulation for flood return periods of 10 and 20 years:

**Upstream boundary condition:** $Q_{10} = 40 \text{ m}^3/\text{s}$, $Q_{20} = 48 \text{ m}^3/\text{s}$,

**Downstream boundary condition:** $h_{10} = 2.85 \text{ m}^3/\text{s}$, $h_{20} = 2.95 \text{ m}^3/\text{s}$,
2.3 Solution Methods

The transfer of the depth-averaged Navier-Stokes equations from continuous to discrete notion is called discretisation. Discretisation strategies are essentially required to make a choice for a numerical method or commercial software product from hydroinformatics. Different classes of methods are applicable for numerical discretisation:

Finite Difference Methods, replacing the differential quotient by difference quotients based on the definition of local state variables at points of a regular grid.

Finite Volume Methods, using boundary and surface integration to express the differential equation for the considered finite domain at the nodes.

Finite Element Methods, using approximations while describing the state variables during integration over the elements of the discretized domain, being the method of weighted residuals or the Galerkin approach.

Spectral Methods, using approximation methods while representing the state variables in the coefficient-matrix of the equation system, called ‘spectra’.

Regarding numerical concepts it is worth to realize, that the collocation method fulfils the differential equation only at certain defined points. The Galerkin method uses the same function for approximation and spatial discretisation during integration of the domain.

Within these classes, several methods exist for different equations as well as there have been developed different variations of the same methods. They all include a description of spatial and temporal discretisation. This variety of methods can be ordered in a certain corridor by following simplifying scheme referring to accuracy and flexibility.
3 Data sources

3.1 Maps

See Container “Data Sources for Hydroinformatics”.

3.2 Topography

See Container “Data Sources for Hydroinformatics”.

3.3 Bathymetry

See Container “Data Sources for Hydroinformatics”.

3.4 Measurements of flow hydraulics (observed longitudinal water surface profiles, measured velocity profiles)

To solve 1D or 2D flow equations, boundary conditions for a river reach must be determined. Commonly, these are water levels and discharges. Manual field surveys for small streams conducting measurement of flow hydraulics are known as stream gaging. Within the whole cycle of measurements being required to set up a hydrodynamic model, the flow hydraulics are only one of several contributions, being referred in the graph below.

- historical rainfall & runoff data across catchment;
- historical river height & discharge information;
- catchment topography & land use;
- surveys of river, floodplain levels & cross sections;
- hydrologic processes (rainfall, runoff, infiltration, concentration etc);
- hydraulic processes (propagation, attenuation etc).

Manual measurement campaigns essentially involve cross-sectional measurements on discharge and mean velocity. They serve to perform and confirm hydraulic calculations and estimates for the equivalent sand roughness $k_s$ or likewise Mannings’s $n$. 
Detailed manual measurement campaigns cover local vertical velocity profiles. They serve to determine shear stress and roughness. The laws being required for that purpose assume a logarithmic distribution of the vertical velocity, hence long records of measurements must be conducted to eliminate noisy fluctuations in the measurement data set.

To obtain a measure of the mean velocity, Chow suggests an approximation of the mean vertical velocity via measuring the velocity at 0.6 of the depth. Where more reliable results are required, the average velocity at 0.2 and 0.8 of the water depth is taken. When the stream is covered with ice, the mean velocity is no longer close to the velocity at 0.6 depth but the average velocity at 0.2 and 0.8 depth still gives reliable results (CHOW, 1973).

Measurement of local near-bank velocity profiles are used to determine bank shear-stresses, and reveal information on the local flow patterns, such as cross-stream currents or helical flow.

Manual field surveys should roughly consider:

An average of 20 depth measurements per cross-section.

A series of mean velocity measurements for cross-sections requiring estimate the flow discharge to perform integration across the channel width.

A formal characterisation of the flow pattern across-stream and in the vertical.

Optionally two or three vertical velocity profiles near the channel-centre-line, if the shear stress on the river bottom should be determined.

Optionally at least one horizontal velocity profile near the outer bank wall, if bank shear-stresses need to be assessed.

Verifications of manual calculations and 1D simulations may benefit from cross-sectional estimates of the hydraulic radius, maximum depth, inundation extent (total width), surface slope and details on the velocity directions (flow pattern).

Measurement of flow velocities in flumes use devices called current meter. Their principle of operation uses the proportionality between the fluid velocity and the resulting angular velocity of the meter rotor. The velocity of water at that point is determined by placing a current meter at a point in a stream and counting the number of revolutions of the rotor during a measured interval of time. The sensitivity of the rotor limits the use of the flow meter to natural streams with negligible particle transport. Streams being too deep or too swift to wade require deploys of the current meter from a boat or bridge. Commonly, a sounding weight is attached below the current meter to keep it stationary in the water. This weight prevents damage to the meter when the assembly is lowered to the streambed.
A detailed review on literature, devices and fundamentals of measurements of the water discharge are referred by the USGS (see recommended links).

Technically advanced devices apply acoustic principles based on the Doppler effect. Exemplarily, these devices might probes such as the ADV (Acoustic Doppler Velocimeter) or ADCP (Acoustic Doppler Current Profiler), being available at the Department of River and Coastal Engineering at Hamburg University of Technology. These probes offer regular deployment on boats or other floating devices for stationary and real-time-measurements. In addition to regular velocity measurement, the measurement of turbulent structures has successfully been carried out at this department with its research vessel ‘Groppe’ and will be carried out in the next years on the vessel ‘Nekton’ (www.tuhh.de/wb).

ADCP deployment on catamaran via radio transmission
River bed roughness is an essential component within computations of open channel flow, being represented via resistance laws. The roughness of the bed material can be determined via average grain
size and determines the resistance towards the fluid motion. Generally, greater roughness results in higher water level and lower velocities. River engineering expresses the roughness of the bed material via the equivalent sand roughness according to Nikuradse. The so-called ks-parameter reflects a length scale, describing three attributes of the bed material:

Shape of the materials

Arrangement of the materials (pattern)

Distance between the materials (density)

As the equivalent sand roughness expresses a length scale, it is measure in metre. As the bed material usually reflects a minor resistance magnitude, it is frequently referred in millimetre. To illustrate this phenomena, a brick wall commonly refers to an equivalent sand roughness of 1-8mm, whereas forested ground is being referred with 160-320mmm. For more detail the table below reflects experience from a variety of measurements in different rivers (BWK, 1999).


<table>
<thead>
<tr>
<th>Equivalent Sand Roughness ks [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Plane river bed</strong></td>
</tr>
<tr>
<td>- sandy or from gravel</td>
</tr>
<tr>
<td>- gravel</td>
</tr>
<tr>
<td>- rough gravel</td>
</tr>
<tr>
<td>- heavy stones</td>
</tr>
<tr>
<td>- plastered floor</td>
</tr>
<tr>
<td><strong>Floodplains and banks with</strong></td>
</tr>
<tr>
<td>- farmlands</td>
</tr>
<tr>
<td>- farmland with crops</td>
</tr>
<tr>
<td>- forested ground</td>
</tr>
<tr>
<td>- stones 80/450 with grass</td>
</tr>
<tr>
<td>- stones with rough vegetation</td>
</tr>
<tr>
<td>- grass with shrubs</td>
</tr>
<tr>
<td>- shrubby vegetation</td>
</tr>
<tr>
<td>- grass and stone grid</td>
</tr>
<tr>
<td><strong>Walls made of</strong></td>
</tr>
<tr>
<td>- bricks</td>
</tr>
<tr>
<td>- concrete, smooth</td>
</tr>
<tr>
<td>- concrete, rough</td>
</tr>
<tr>
<td>- rubble</td>
</tr>
<tr>
<td>- natural stone wall, rough</td>
</tr>
<tr>
<td>- steel sheet piling, depending on profile depth</td>
</tr>
</tbody>
</table>
The roughness coefficient being presented is theoretically embedded into simulation models via the Colebrook-White’s Resistance law, commonly referred as

\[
\frac{1}{\sqrt{\lambda}} = -2.03 \cdot \log \left[ 2.51 \frac{k_s}{Re \cdot \sqrt{\lambda}} + 14.84 \cdot r_{hy} \right]
\]

with:

\( \lambda \) = Friction coefficient  
\( Re \) = Reynolds number  
\( r_{hy} \) = Hydraulic radius  
\( k_s \) = Equivalent sand roughness

The friction coefficient of Colebrook-White’s Resistance law reflects the roughness within the open channel section. Implemented into the flow equation of Darcy-Weisbach, the impact on the velocity,

\[
v = \frac{1}{\sqrt{\lambda}} \sqrt{8 \cdot g \cdot r_{hy} \cdot I_E}
\]

hence the impact on the water level in a 1D cross-section stationary flow \((Q = v \cdot A)\) is straightforward from the mathematical point of view.

Instead of \( k_s \), the impact of bottom roughness can also be included to Colebrook-White’s Resistance law via Manning’s roughness coefficient \( n \). As it does not reflect a physical scale and as it depends on the flow regime and water depth, its choice is more difficult. Hence, some internet portals offer guidelines for successful choices of Manning roughness coefficients linked to typical channel types and regime descriptions. An example for the USGS data base and British Conveyance Generator (See Recommended Links).

Nevertheless, floodplains with similar bed material will impose distinct resistance towards flow, if one of them is significantly covered with high vegetation.
Hydronumeric vegetation classification on floodplains (Pasche, 1984)

Hydraulic quantification of vegetations on banks and floodplain require distinct classifications, differentiating between small, medium and large vegetation elements. The height of small vegetations is substantially smaller than the flow depth. Consequently, the velocity distribution above this vegetation elements turns out according to a boundary layer flow. Middle and high vegetations vertically block great parts the entire flow depth and may even induce similar effects like piles in the case of trees. Flow velocities around these vegetation elements can not be described according to a logarithmic velocity distribution, in first order it can be assumed to be constant over the depth. Hence, distinct approaches are needed to describe the impact of different classes of vegetation elements on flow fields. Three mayor classes of vegetation are distinguished for hydraulic calculations:

- submerged vegetation elements \( h_p << h \)
- immersed elements \( h_p \sim h \) and
- isolated vegetation \( h_p > h \).

Here, only the friction coefficient of middle and high vegetations on immersed condition will be referred in detail. More detailed information on the fundamental of fluid mechanic are available in following online script (http://elbe.wb.tu-harburg.de/english_site/education/script/flu_mech/...

...skript_fundamentals-of-fluid-mechanics.pdf).

For middle and high vegetations in immersed condition, the friction factor is determined according to Pasche, introducing the form drag of the vegetation elements, projected on the base area \( a_x \cdot a_y \) of the vegetation elements.
**Definition scheme for roughness modelling of middle and high immersed vegetation**

\[ \lambda_p = \frac{4 \cdot h_p \cdot \frac{d_p}{a_x \cdot a_y}}{c_{WR} \cdot \cos(\alpha_{lat})} \]

\[ c_{WR} = 1.31 \cdot c_{WR_\infty} \left[ \frac{v_n}{V_{vor}} \right]^2 + \Delta c_W \]

with:
- \( h_p \) = vegetation height under water
- \( d_p \) = vegetation diameter
- \( \alpha_{lat} \) = lateral bank slope
- \( c_{WR} \) = form friction coefficient for a cylinder in group
- \( c_{WR_\infty} \) = form friction coefficient for a cylinder
- \( v_n \) = flow velocity in front of vegetation
- \( v_{vor} \) = velocity in the vegetation area
- \( \Delta c_W \) = wave-current form drag

Empirical estimates for the form friction factor \( c_{WR} \) and the relative initial flow velocity \( v_n/v_{vor} \) have been determined by Lindner (1982) and Pasche (1984). (LINK TO: [http://elbe.wb.tu-harburg.de/english_site/education/scripts/flu_mech/skript_fundamentals-of-fluid-mechanics.pdf](http://elbe.wb.tu-harburg.de/english_site/education/scripts/flu_mech/skript_fundamentals-of-fluid-mechanics.pdf)).

Denoting the fact that the distribution of the bottom roughness usually changes along the wetted perimeter of a cross-section, individual friction coefficients of separate sections can be linearly superimposed to a total friction coefficient according to
The figure below illustrates this equation for a cross section, being divided into three segments, each having a unique k\textsubscript{s} value. The separation of these segments is constructed perpendicular to the isotachs.

\[
\lambda_{\text{Total}} = \frac{\sum_{j=1}^{N} \lambda_j \cdot l_{u,j}}{l_{u,\text{Total}}}
\]

with:

- \( j = 1,2, \ldots, n \)
- \( N \) = number of roughness sections \( j \)
- \( \lambda \) = friction coefficient
- \( l_{u,j} \) = wetted perimeter for roughness section \( j \)
- \( l_{u,\text{Total}} \) = total wetted perimeter
- \( A_{\text{Total}} \) = sum of individual areas
- \( r_{\text{hy},\text{Total}} \)

\[ A_{\text{Total}} = \sum_{j=1}^{N} A_j \]

\[ l_{u,\text{Total}} = \frac{A_{\text{Total}}}{r_{\text{hy},\text{Total}}} \]

Einstein superimposition of cross-sectional roughnesses (BWK, 1999).

Following a fluvial domain is depicted by a near-infrared imagery, enhancing the classification of vegetation features, as well as a classification, revealing 14 distinct ground cover types.
3.6 Structures data (bridges, dikes, groynes, etc.)

See Container “Data Sources for Hydroinformatics” and 1D-Hydroinformatics.
4 Pre-processing methods for input data [SR]

Text is yet missing.

4.1 Data filtering

Remote sensing data commonly requires hydro-enhancement, as the collected data may contain surface objects (trees, bushes, etc.) requiring a certain amount of hydro-enforcement. Furthermore, hydronumeric software products are somewhat limited in their capacity to represent all individual measurements.

1D models use cross-sectional representations of the topography. They frequently apply moving average filters to reduce micro-roughness pattern and random and system noise of the remote sensing system. Likewise, they perform a reduction or abstraction of the measured profiles. The performance and smoothing introduced during cross-sectional filtering essentially depends on the filter window size.

2D models require spatial filters if the terrain data needs to be hydro-enforced. Spatial filters commonly perform initial gridding before applying roving filter kernels. These kernels, being clusters \( n \times n \) cells, evaluate each elevation cell consecutively while applying spatial criteria limiting acceptable terrain differences or slopes.

The DTM accuracy should be such that important properties of the flow field can be modelled in accordance with measurements of water levels, velocities or even sediment transport and deposition. Filtering algorithms holding these requirements, not only on the global basis of the available test point set, but also on a local basis, is still recognised as matter of research today.

4.2 Validation of input data

Any model input data requires quality assessment. This procedure is a critical aspect of model validation. Measurements can not simply be regarded as absolute values (IAHR, 1999). Hydraulic research performing simulations requires source data contributions from topography, bathymetry, surface cover and hydrography. Commonly, hydraulic research relies on the quality assessment, being carried out by data suppliers, being geographers or cadastral offices, hydrographical service providers, field engineers as well as water and shipping authorities. Contributions from the accuracy of individual source data contributions to hydronumeric simulations and subsequent risk analysis remains to be an active field of current research.
5 Simulation

Text is yet missing and needs to be written.

5.1 Model parameters (roughness coeff., eddy viscosity coeff.) [SR, DK]

The fundamentals of modelling of bottom shear stress originate from the Newtonm-Taylor law, being essentially derived from measurements. Hence, these fundamentals are of empirical nature. This law formulates a proportionality for the quadratic ‘characteristic’ flow velocity and the bottom shear stress. This empirical relation is only hold for the assumption that the flow is almost uniform. Furthermore, almost parallel streamlines are assumed with an approximate logarithmic velocity profile over the vertical axis. This assumption conforms with the basic requirements of 2D depth averaged hydrodynamic models, neglecting the vertical flow component, hence discarding the possibility for a deformation of the velocity profile due to vertical eddy formation.

Frequently, the model parameterisation uses resistance coefficients, roughness parameters or magnitudes for eddy viscosities from previous model applications. Likewise published values from literature are used. Ideally, these values are directly derived from measurements within the domain and a discharge used for model calibration.

For river systems, the type of floodplain has an impact of the flow pattern. Commonly, straight streams in regular divided cross-sections and meandering streams are distinguished. Since the flow pattern are dominated by the discharge and the geometry, a conscious understanding of the model parameterisation is required, if models with constant parameterisation are used. Algebraic turbulence models, e.g. Prandtl mixing length concept, or enhanced concepts considering the impact of secondary flow (dispersion) due to meandering compensate the problems for parameterisation choices. The dispersion term compensates the deviation of the depth and time averaged values from their real distribution in the flow field. Commonly, the dispersion term can be neglected for straight channels, whereas it significantly contributes for meandering flows. Models according Malcherek (2005) and Flokstra (1977) serve to model turbulent shear stresses.

5.2 Checking and analysis of results [SR, DK]

The resolution of a hydronumeric model defines the degree of detail being resolved during computation. Ultimately, it settles the appropriateness of the obtained result. With respect to hydroinformatics, this aspect denotes the necessity to represent the spatial structure of the flow, realized through discretisation of the hydrodynamic variables on a computational mesh.
Hydraulic models were typically assessed in terms of their ability to simulate the reach downstream hydrograph, usually achieved by calibration of the model friction parameters. Furthermore, the maximum inundation extent is a measure reflecting the internal model performance, reflecting topographic accuracy in a discretisation and the appropriateness of model calibration. This criterion should not be considered as a primary indicator for the accuracy of the solved equations.
6 Model Performance

Text is yet missing and needs to be written by [DK + MJ].
7 Introduction to modelling software

Text still needs to be written.

7.1 RMA2 in Kalypso-2D

RMA2 uses isoparametric quadrilaterals and triangles are for its finite element discretisation, i.e. it uses a parametric definition of both coordinate and displacement functions. Element boundaries may be either curved or straight. RMA2 uses Gaussian quadrature to evaluate the integral forms. The shape functions are bi-quadratic representations of the velocity components and bi-linear functions describing the water depth. The governing equations, i.e. the depth averaged shallow water equations, are time dependant. Time dependency is implemented using a modified Crank-Nicholson scheme. This implicit method requires the solution of a set of simultaneous equations, while for this problem the equations are non-linear. A Newton-Raphson Method is applied to derive a locally linear set of equations, solving for the corrections to an initial estimate of velocities and water depth. The initial assumption is commonly called lake-solution and assumes zero velocity and an overall wetted simulation domain.

Scheme for Lake Solution of a single River Branch

The hydraulic model RMA2 was originally developed by Norton, King and Ortlob, of Water Resource Engineers, for the Walla Walla District, Corps of Engineers, and delivered in 1973 (Donnel, 1997). Subsequent enhancements, particularly of the marsh porosity option, have been carried out by King and Roig at the University of California, Davis. Today, RMA2 is a internationally accepted hydraulic

The available RMA2 code contains several modifications and extensions introduced to the original source, amongst others at the roughness concept. The Darcy-Weisbach friction factor $\lambda$, the resistance-formula of Colebrook/White and the determination of the flow resistance of non-submerged vegetation has been included. Further more, two different concepts for mass-conservation are available. The first applies a conventional Galerkin weighted residual approach to solve finite element integrals. The second realizes a hybrid Control-Volume-FEM concept, solving the continuity for nodal control volumes while maintaining the Galerkin weighted element integration for the momentum equations, providing higher accuracy in mass balance.

Handling the equation system to be solved is a significant factor of calculation stability and simulation time. RMA2 improves its matrix solution providing an option to reorder the assembled sequence of elements to minimize the bandwidth of the matrix (Sloan, 1986). This strategy attempts to cluster non-zero terms along the diagonal of the matrix to exploit the banded nature of the resulting structure, restricting the number of required calculations. More recent ideas tend to solve element-by-element, storing the matrices in their elementary form without ever fully assembling the terms (Carey and Jiang, 1986), implemented for instance to Telemac-2D (Hervouet, 1992). Whereas problems in heat flow and structural analysis lead to symmetric stiffness matrices, fluid flow and its convective terms lead to non-symmetric matrices (Brooks and Hughes, 1982). The authors note that the solution for fluid flow may be corrupted by spurious node-to-node oscillations if rapid changes in the solution are enforced and conclude, stable solution require high FE-mesh resolutions simulating highly dynamic events. Galerkin-FEM simulations of overbank flow and topographic complexity combine to a problem approaching the limit of current computational resources (Bates et. al., 1998). Topographic complexity in the channel cross-section can generate large differences in flow velocity over short distances via the momentum exchange mechanism between channel and floodplains (Pasche, 1984, Knight & Shiono, 1990, Shiono & Knight, 1991). These rapid changes can lead to node-node oscillations, causing in extreme situations instabilities in the iterative numerical procedure whereby the solution ‘explodes’, i.e. the predicted variables grow without limit instead of gradually converging to a single value (Bates et. al., 1998).
Referred Literature


Referred Web-Resources

Lecture notes:

Derivation of Navier Stokes Equation:

Fluid properties: Viscosity
http://hypertextbook.com/physics/matter/viscosity/

Fluid properties: Dynamic Viscosity

USGS: Measurement of Water Discharge
http://wwwrcamnl.wr.usgs.gov/sws/fieldmethods/QMeas.tmp/

USGS: Surface Water Field Techniques - Verified Roughness Characteristics of Natural Channels
http://wwwrcamnl.wr.usgs.gov/sws/fieldmethods/Indirects/nvalues/

Approved 2D-Simulation Software for Risk & Hazard Modelling:
Referred Web-Portals

FEMA Portal – Delineation for flood hazard zones in the USA, New York, Wall Street


Environmental Agency Portal – Delineation for flood hazard zones in the Great Britain, Greater London

http://maps.environment-agency.gov.uk/wiybv/mapController
Referred Image Sources

NASA: Effects of Hurricane Floyd on the North Carolina Coast.
http://www.msfc.nasa.gov/NEWSROOM/camex/photos/NC_flood_19991105_m.jpg