ČESKÉ VYSOKÉ UČENÍ TECHNICKÉ V PRAZE

Fakulta stavební

FACULTY OF CIVIL ENGINEERING CTU IN PRAGUE

doc. Ing. Jaroslav Pollert, Ph.D.

# Mathematical and physical modelling in water management research

# Matematické a fyzikální modelování ve vodohospodářském výzkumu

# Summary

At present, no major structure in the water sector can be constructed without proper planning, which requires knowledge of the causes and consequences induced by hydraulic phenomena. Reliable analysis of flow structure allows for achieving optimal shape or more specifically higher reliability of the construction. Modern technology provides for a wide range of assessment of such works, but in some cases not enough and here modelling plays important role.

Models in hydraulic research can be divided into two basic groups - mathematical models and physical models. Physical modelling can be defined as displaying objects and phenomena in a dynamic physical state that enables systematic, rational and effective investigation. Relationship similarities can transmit information from model to prototype.

Mathematical modelling is intended for monitoring physical phenomena, if we are able represent phenomena properly as mathematically described. An important property of mathematical modelling is its ability to explore a wide range of cases that can occur in nature and in technical practice, and to find optimal solutions based on these investigations.

In certain cases, we can combine both approaches, i.e. mathematical and physical modelling and use their specific advantages for the overall solution to the problem. It is possible to interconnect, verify and calibrate models, thus providing a far more comprehensive view on the issue of water flow and reveal the problematic areas of construction. Linking these approaches to BIM is the next step, which puts knowledge of water flow into future projects and avoids conflicts with other elements of construction.

To explain this, further examples of studies carried out by the author are shown. In them are clear explanations of the advantages and disadvantages of each approach or their combined approach.

# Souhrn

V současné době se žádná významná stavba ve vodním hospodářství neobejde bez důkladného plánování, kde je nutná znalost příčin a následků vyvolaných hydraulickými jevy. Spolehlivá analýza struktury proudění umožňuje dosáhnout např. tvarové optimalizace objektů a zejména vyšší provozní spolehlivost stavby. Moderní technologie sice poskytují široké možnosti posouzení takovýchto děl, v některých případech však nestačí a zde je nezastupitelná role modelování.

Modely v hydraulickém výzkumu lze v zásadě rozdělit do dvou základních skupin – modely matematické a modely fyzikální. Fyzikální modelování lze definovat jako zobrazování objektů a jevů na nich probíhajících fyzikálními prostředky, které umožňují jejich systematické, racionální a efektivní zkoumání. Vztahy podobnosti umožňují přenášet poznatky z modelu na prototyp.

Matematické modelování je určeno pro sledování fyzikálních jevů, pokud jsme schopni daný jev dostatečně výstižně matematicky popsat. Důležitou vlastností matematického modelování je jeho obecnost, umožňující zkoumat širokou škálu případů, které se mohou v přírodě a v technické praxi vyskytnout, a na jejich základě hledat optimální řešení.

V určitých případech lze propojit oba přístupy, tj. matematické a fyzikální modelování a využívat i jejich jednotlivých výhod pro celkové řešení problému. Takto je možné modely vzájemně propojovat, verifikovat a kalibrovat, což poskytne daleko ucelenější pohled na problematiku proudění vody a odhalí problematická místa stavby. Propojení těchto přístupů do BIM je další krok, kterým vložíme poznatky z proudění vody do budoucích projektů a vyhneme se tak kolizím s ostatními prvky stavby.

Pro názornost jsou v další části uvedeny příklady výzkumů, které byly v této oblasti autorem provedeny. Na nich je názorně vidět výhody i nevýhody obou přístupů nebo jejich kombinací.

# Keywords

Hydraulics; Mathematical modelling; Physical modelling; Computational Fluid Dynamics (CFD)

# Klíčová slova

Hydraulika; Matematické modelování; Fyzikální modelování; Computational Fluid Dynamics (CFD)

# Contents

# Introduction

*"Measure what can be measured and make measurable what cannot be measured."*

*— Galileo Galilei*

Many of the engineering structures, such as spillways, turbines or different hydraulic structures, designed by engineers, are subjected to very complex flow phenomena for which a completely rigorous mathematical analysis is not always available. Design must frequently be made on the basis of inadequate information and on numerous assumptions based on the engineer´s judgement and experience. For these reasons, and because of the financial and safety risks involved, it has become customary in the design of many engineering structures to take advantage of the information obtained from model studies. They permit visual observations of the flow and make it possible to obtain certain numerical data.

## Physical modelling

Many engineers who have been involved with design and construction believe that using hydraulic models to design structures is the best procedure. There is a long history of hydraulic physical models of free surface water flows and there is no doubt that they can give a very accurate representation of water levels, flow directions, and velocities. Physical hydraulic models are commonly used during design stages to optimize a structure and to ensure a safe operation of the structure. Furthermore, they have an important role to assist non-engineers during the 'decision-making' process. A hydraulic model may help the decision-makers to visualize and to picture the flow field, before selecting a 'suitable' design. In civil engineering applications, a physical hydraulic model is usually a smaller-size representation of the prototype (i.e. the full-scale structure). In any case the model is investigated in a laboratory under controlled conditions.

Physical modelling is based on dimensional analysis. Dimensional analysis is a most useful tool in experimental fluid mechanics, allowing for the implicit formulation of criteria for dynamic similarity in a simple and direct manner. A physical problem with independent parameters $q_1$, $q_2$, $q_3$, .... $q_n$ can be reduced to a product of independent, dimensionless parameters. Turbulence causes the appearance in the flow of eddies with a wide range of length and time scales that interact in a dynamically complex way, with r as the minimum number of reference dimensions (length [L], mass [M] or time [T]) required to describe the dimensions of these n parameters. Similarity requires that each of these dimensionless parameters quantitatively agree between model and reality, as well as the force ratios such as Fr (Froude number), Re (Reynolds number), and similar dimensionless numbers.

A physical scale model is similar to its real-world prototype and involves no scale effects if it satisfies mechanical similarity according to the following three criteria:

1. geometric similarity
2. kinematic similarity
3. dynamic similarity

Geometric similarity requires similarity in shape, i.e. all length dimensions in the model are $M_l = M_p / M_m$ where the subscripts p and m refer to prototype (full-scale) and model parameters respectively and l describes characteristic length times shorter than of real-world prototype. Model lengths, areas and volumes therefore scale with M, $M^2$, $M^3$ respectively, in relation to the prototype. Kinematic similarity implies geometric similarity and in addition indicates a similarity of motion between model and prototype particles. It requires constant ratios of time, velocity, acceleration and discharge in the model and its prototype at all times. Dynamic similarity requires, in addition to geometric and kinematic similarities, that all force ratios in the two systems are identical.

In fluid dynamics, the most relevant forces are:

- Inertial force = mass x acceleration = $\rho\ l^2\ v^2$
- Gravitational force = mass x gravitational acceleration = $\rho\ l^3 g$
- Viscous force = dynamic viscosity x $\frac{\text{velocity}}{\text{distance}}$ x area = $\mu\ v\ l$

The parameters are fluid density $\rho$ , characteristic length l, characteristic velocity v, gravitational acceleration g and dynamic viscosity $\mu$ . Dynamic similarity requires constant ratios of all forces, namely (Inertial force)$_P$ / (Inertial force)$_M$ = (Gravitational force)$_P$ / (Gravitational force)$_M$ = ...= constant. A direct consequence is that the corresponding ratios among the various forces must be identical in the model and real-world prototype. The inertial force is normally the most relevant in fluid dynamics and is therefore included in all common force ratio combinations:

- Froude number $\mathrm{Fr}=\frac{\text{inertial forces}}{\text{gravitational forces}}=\frac{v^2}{g \cdot l}$
- Reynolds number $\mathrm{Re}=\frac{\text{inertial forces}}{\text{viscous forces}}=\frac{l\,.v}{\nu}$
    - where $\nu$ is the kinematic viscosity $\nu=\frac{\mu}{\rho}$.

In uniform equilibrium flows, the gravity force component counterbalances exactly the flow resistance and the flow conditions are deduced from the continuity and momentum equations. In practice, river models are scaled with a Froude similitude $Fr_M = Fr_P$ and viscous scale effects must be minimized. The model flow must be turbulent with the same relative roughness as for the prototype (usually for $Re_M > 5000$).

## Similitude and hydraulic model studies

A physical model is a scaled representation of a hydraulic flow situation. Both the boundary conditions (e.g. channel bed and sidewalls), the upstream flow conditions and the flow field must be scaled in an appropriate manner. One of the most powerful tools in fluid mechanics is dimensional or similarity analysis, which permits a wide generalization of experimental results.

Geometric similarity implies that the ratios of prototype characteristic lengths to model lengths are equal: Kinematic similarity implies that the ratios of prototype characteristic velocities to model velocities are the same. Dynamic similarity implies that the ratios of prototype forces to model forces are equal. In open channel flows, the presence of the free surface means that gravity effects are important. The Froude number is always significant. The Froude number is used generally for scaling free-surface flows, open channels and hydraulic structures.

The basic relevant parameters needed for any dimensional analysis may be grouped into the following groups:

- Fluid properties and physical constants. These consist of the density of water ρ ($kg/m^3$), the dynamic viscosity of water μ ($Ns/m^2$), the acceleration of gravity g ($m/s^2$), etc.
- Channel (or flow) geometry. These may consist of the characteristic length(s) l (m).
- Flow properties. These consist of the velocity (ies) v (m/s) and the pressure difference(s) p (Pa).

Taking into account all basic parameters, dimensional analysis yields:

$$f(\rho, \mu, g, l, v, \Delta p) = 0 \qquad (1)$$

There are basic parameters and the dimensions of these can be grouped into three categories:

- Mass (m), length (l) and time (t). The Buckingham Π -theorem [3]
- Implies that the quantities can be grouped into independent dimensionless parameters.

Traditionally model studies are performed using geometrically similar models. In a geometrically similar model, true dynamic similarity is achieved if and only if each dimensionless parameter (or Π terms) has the same value in both model and prototype:

$$Fr_{prototype} = Fr_{model} \qquad (2)$$

Scale effects will exist when one or more Π terms have different values in the model and prototype. In practice, hydraulic model tests are performed under controlled flow conditions. The use of the same fluid on both prototype and model prohibits simultaneously satisfying the Froude number scaling criteria. Scale effects may be defined as the distortions introduced by effects (e.g. viscosity and surface tension) other than the dominant one (e.g. gravity in free-surface flows). They take place when one or more parameters that are dimensionless differ between model and prototype.

In free surface flows (e.g. rivers and wave motion), gravity effects are predominant. A Froude number modelling is typically used when friction losses are small and the flow is highly turbulent: e.g. spillways overflow weirs and flow past bridge piers as well as could be used for whitewater courses modelling.

In hydraulic structures and for wave motion studies, the gravity effect is usually predominant in the prototype. The flow is turbulent and hence viscous and surface tension effects are negligible in the prototype if the flow velocity is reasonably small. In such cases, a Froude similitude must be selected.

The most economical strategy is:

- to choose a geometric-scale ratio such as to keep the model dimensions small,
- to ensure that the model Reynolds number is large enough to make the flow turbulent at the smallest test flows.

Before building a physical model, the engineers must have the appropriate topographic and hydrological field information. Then the type of model must be selected and a question arises:

'Which is the dominant effect: e.g. viscosity, gravity and surface tension?' In the general case, the engineer model investigator must choose a proper geometric scale. The selection procedure is an iterative process:

- Select the smallest geometric-scale ratio to fit within the constraints of the laboratory.
- For geometric scale, and for the similitude criterion (e.g. Froude or Reynolds), check if the discharge can be scaled properly in model, based upon the maximum model discharge $Q_m$max.
- Check if the flow resistance scaling is achievable in the model (is it possible to achieve required λ - Darcy-Weisbach or Manning – n friction coefficient).
- Check the model Reynolds number for the smallest test flow rate (if the prototype flow is turbulent, model flow conditions must be turbulent, i.e. typically $Re_m$> 5000).
- Choose the most convenient scale.

Physical hydraulic modelling is a design technique used by engineers to optimize the structure design, to ensure the safe operation of the structure and/or to facilitate the decision-making process. In practice, most hydraulic models are scaled with either a Froude or a Reynolds similitude: i.e. the selected dimensionless number is the same in model and prototype (i.e. full scale). The most common fluids are air and water. Free-surface flow modelling is most often performed with the same fluid (mainly water) in full scale and model.

Open channel structures; generally, have gravity forces and inertial forces that far outweigh viscous and turbulent shear forces. Thus, geometric similitude and equivalency of Froude number for the model number and prototype produce a good approximation to dynamic similitude.

Although the accuracy of Froude scale models is well established, their interpretation can be difficult. The main problem is that since all water levels, including heights of standing waves and other surface disturbances are reduced by the linear scale factor, this very fact makes observation difficult.

A further difficulty arises in that the water surfaces of the full-size channel will have much more "whitewater" than the model. In fact, the model has very little "whitewater", whereas the full size channel will be very impressive with large areas of whitewater. This because the white colour is produced by entrained air in the water surface and the amount of air entrained depends to a great extent on the absolute water velocity. Therefore, there will be many regions in the model where the water surface looks smooth and black, that in full-size will appear white and turbulent.

## Mathematical modelling

With the rapid increase in computer power in recent years, it is not surprising to note that three-dimensional (3-D) numerical modelling of flow has shifted from academic research to practical applications in order to reduce the cost and time of design of hydraulic structures. The emphasis of the most studies is on simulating the three-dimensional hydrodynamic processes because of the complexity of the flow behaviour. We call this science CFD - Computational Fluid Dynamics.

The governing equations used to describe flows are generally based on 3-D Reynolds averaged Navier-Stokes equations for incompressible and unsteady turbulent flows. If the vertical acceleration of the flow is negligible in comparison with the gravity and the vertical pressure gradient terms, then the hydrostatic pressure distribution assumption can be made. Since the reservoir to be studied in the current study has a relatively shallow water, the density of the water has also been assumed constant throughout the computational domain.

Turbulence causes the appearance in the flow of eddies with a wide range of length and time scales that interact in a dynamically complex way. Given the importance of the avoidance or promotion of turbulence in engineering applications, it is no surprise that a substantial amount of research effort is dedicated to the development of numerical methods to capture the important effects due to turbulence. The methods can be grouped into following categories: Turbulence models for Reynolds-averaged Navier-Stokes (RANS) equations: attention is focused on the mean flow and the effects of turbulence on mean flow properties. Prior to the application of numerical methods, the Navier-Stokes equations are time averaged. Extra terms appear in the time-averaged (or Reynolds-averaged) flow equations due to the interactions between various turbulent fluctuations. These extra terms are modelled with classical turbulence models: among the best known ones are the model and the Reynolds stress model. The computing resources required for reasonably accurate flow computations are modest, so this approach has been the mainstay of engineering flow calculations over the last three decades.
- Large eddy simulation: this is an intermediate form of turbulence calculations which tracks the behaviour of large eddies. The method involves space filtering of the unsteady Navier-Stokes equations prior to the computations, which passes the large eddies and rejects the smaller eddies. The effects on the resolved flow (mean flow and large eddies) due to the smallest, unresolved eddies are included by means of a so-called sub-grid scale model. Unsteady flow equations must be

solved, so the demands on computing resources in terms of storage and volume of calculations are large, but (at the time of writing) this technique is starting to address CFD problems with complex geometry. Direct numerical simulation (DNS): these simulations compute the mean flow and all turbulent velocity fluctuations. They can resolve the Kolmogorov length scales at which energy dissipation takes place and with time steps sufficiently small to resolve the period of the fastest fluctuations. These calculations are highly costly in terms of computing resources, so the method is not used for industrial flow computations. Enhancing the ability of the numerical model in simulating turbulence is considered in different CFD codes specifically.

## Multiphase flow and CFD

A large number of flows encountered in nature and technology are a mixture of phases. Physical phases of matter are gas, liquid, and solid, but the concept of phase in a multiphase flow system is applied in a broader sense. In multiphase flow, a phase can be defined as an identifiable class of material that has a particular inertial response to and interaction with the flow and the potential field in which it is immersed. In our case it is water – air, which leads to free surface modelling, or sludge – water which leads to miscible liquids. [1]

Multiphase flow can be classified by the following regimes, grouped into four categories:

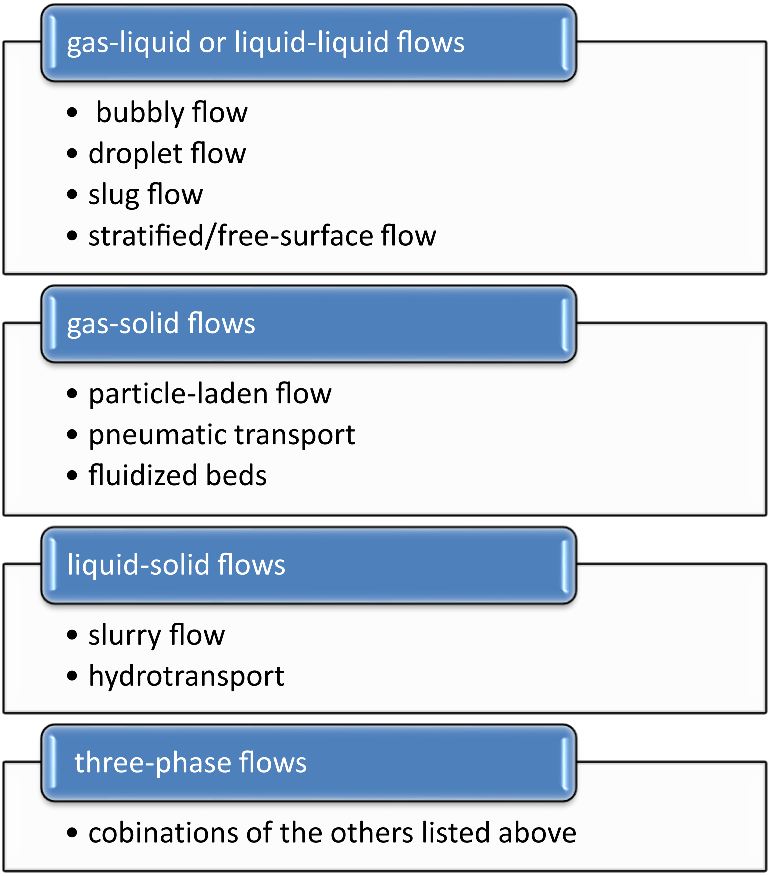


*Fig. 1 Multiphase flow categories*

Advances in computational fluid mechanics have provided the basis for further insight into the dynamics of multiphase flows. Currently there are two approaches for the numerical calculation of multiphase flows:

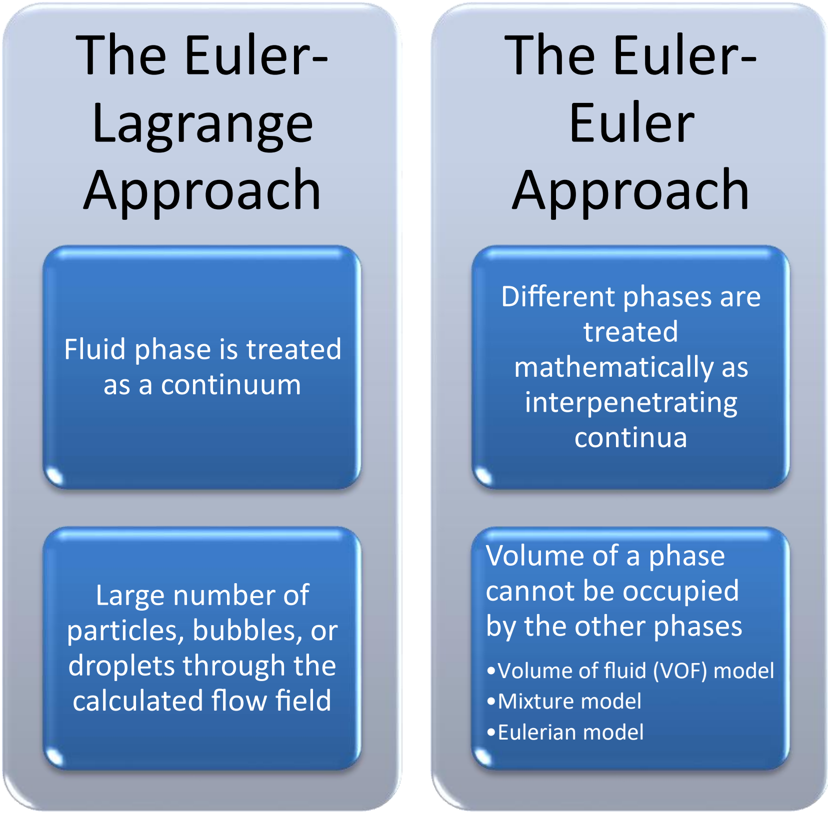


*Fig. 2 Multiphase flows approaches*

The first step in solving any multiphase problem is to determine which of the regimes (Fig. 1) best represents your flow. As a second step we are choosing the numerical calculation approach (Fig. 2). In our case are the most common model is the VOF (Volume of Fluid), which can describe the free surface – interface between liquid and gas. Other models like the Mixture or Eulerian model are used as well for describing mixing fluids with different physical variables.

# From physical problem to physical and mathematical modelling

Mathematical and physical models are considered with reference to some fundamental differences; the main advantages and disadvantages of each method are emphasized. Then, the possibilities are shown, which physical models offer today with the application of modern techniques, especially when they are used in combination with digital computers, as, for example, in the so-called "hybrid-static" (hybrid structural model analysis) [17].

There are many examples, which show the advantage of modelling. Both approaches (mathematical and physical) have their own place, but can be combined and/or replaced (Fig. 3). Usually, decisions for choosing one of these approaches depend on the difficulty of the system and the need for observation of the behaviour. The price (which means computational time vs price of the model) is also influential in decision-making.

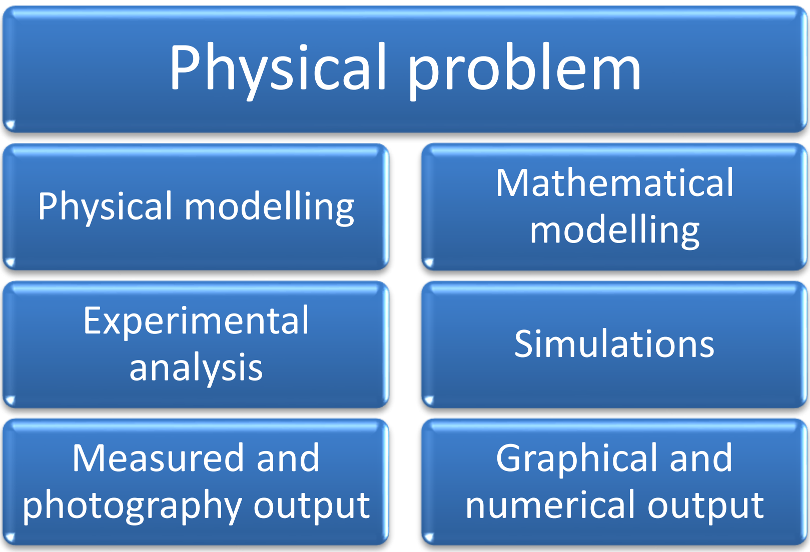


*Fig. 3 Basic processes in the modelling*

Both physical experiments and analytical/numerical simulations complement each other. Both the approaches have their own limitations, advantages and disadvantages.

## Physical modelling

These are usually very time consuming and expensive to set up. There are limitations on the extrapolation of the results obtained on comparing the scaled model of a problem to the actual prototype, but the experimentally observed data provides the closest possible approximation of the physical reality within the limits of experimental errors (Fig. 4).


*Fig. 4 Basic scaling model problems*

## Numerical Simulation

Mathematical modelling is based on a set of assumptions with regard to the variation of the problem variables, constitutive relations and material properties (Fig. 5). Numerical simulation process introduces additional approximation errors in the solution. Hence, results of any analytical or numerical study must be carefully validated against physical experiments to establish their practical usefulness. However, once validated, a numerical simulation can be easily performed on the full scale prototype, and thereby eliminate the need of extrapolation. [17]

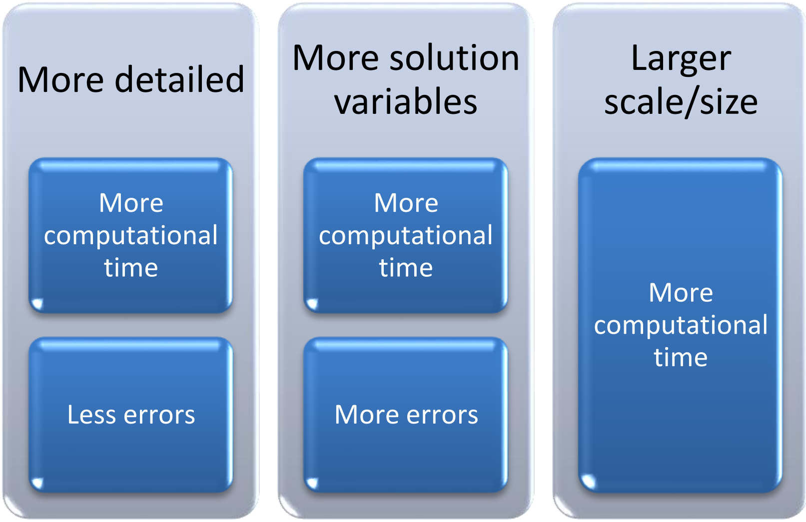


*Fig. 5 Basic mathematical model problems*

## Combination of the modelling techniques

The combination of these two techniques always brings the best results. It can eliminate weaknesses of both techniques and bring better results. We have two possibilities for how to combine, extend, or increase validity (Fig. 6).

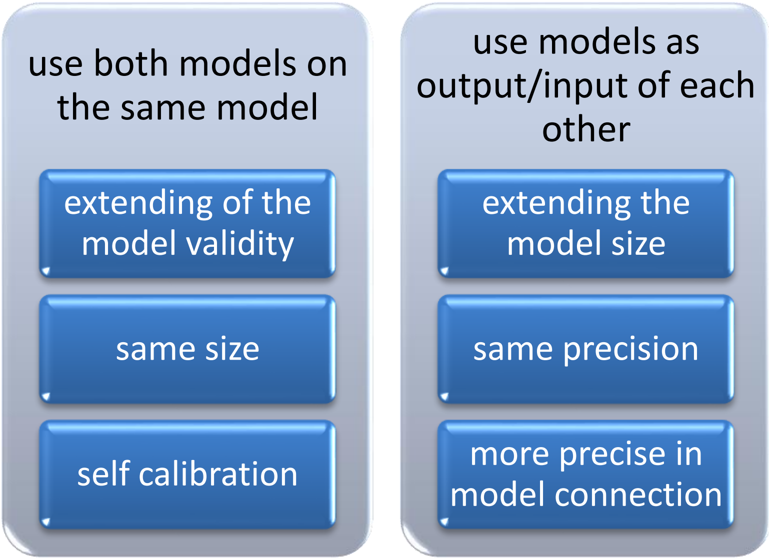


*Fig. 6 Basic model combining*

The next examples explain problems connected with using clear physical modelling, combinations of physical and mathematical modelling, as well as clear mathematical modelling to demonstrate advantages / disadvantages of each approach.

# Example 1: physical model - Whitewater Canoe Course in Rio de Janeiro for Olympic Games in 2016

This is the best example when only a physical model can give proper results. Results of this research cannot be achieved with a mathematical model. The main reason is that we are looking for the water flow behaviour according to changes in geometry.

The results of the physical modelling investigations for the proposed Whitewater Canoe Course in Rio de Janeiro for Olympic games in 2016 were an evaluation of the design in regards to overall validation, location-specific functional hydraulic performance, and identification of potential issues with solutions.

## What is expected of these water structures?

During design, itis necessary to evaluate the different expectations of future users before the design itself with respect to IOC (International Olympic Committee) and ICF (International Canoe Federation) regulations.

The expectation of these courses falls into broad categories. First, from the athletes' point of view, there is the expectation of an exciting, spectacular course. Secondly, from the promoters' point of view, is the expectation of a facility that is attractive to spectators and the public, who may never actually go on the water, but will often come to observe what is going on there. Thirdly, the course must be safe for all categories of users, ranging from elite to novice. Lastly, from the financier's point of view, the course has to be affordable, to both design and construct initially, but also to maintain in the future [2].

- **GENERAL USERS.** Users of the course break down into two categories: elite athletes, who would like to use the course for important races, such as National Championships, or World Championships, or training for events like these; and people with much less skill - let's say novices - who want a much less difficult course.

- **ELITE ATHLETES** - these people want a course with the following characteristics:
    - It should conform to the standards of the International Canoe Federation (ICF), the international governing body of the sport, in that the course should be from the minimum length 200 m and the maximum length 400 m measured from the start line to the finish line down the centre-line of the course.
    - Based on much experience of athletes, coaches and expert engineers, the width should be approximately 10 meters, with variations permissible in short sections.
    - Again, based on experience, the depth should be 0.6 m or more, to make it safe for eskimo rolling and playing. It should be noted, however, that the deeper the water, the more volume of water must be available from the reservoir, and this may be a limiting factor.
    - A rather severe drop in all or at least many sections. Experience has shown this to be approximately 3-5 m over the entire length of the course. These figures are based on what athletes would like, but also on the costs involved. Drops of natural rivers are usually found to be quite large, even as high as 20 m. If this is compared with the height difference on the course at Augsburg (about 4 m) it can be seen that relatively, small drops in height are very suitable for making artificial canoe slalom courses. This is because in the artificial course the water flow is designed, and hence efficient, while in the natural river it is inefficient.
    - A sufficient discharge through the course. Experience had shown this to be roughly from 10 – 20 $m^3/s$. The lower figure would be appropriate for a training course and the higher figure would be appropriate for a top international race.
- **NOVICES.** These people want a course with the following characteristics:
    - All the above hydraulic features, but on a smaller scale.
    - Long stretches of water deep enough in which to eskimo roll, because they will often go through many attempts.
    - Safe and quick escape from the course, should they capsize and fail to roll.

    - Positioning of obstacles, which permits a more direct passage down the river, unlike the elite athlete who would like a more twisting passage.
- **THE SPECTATOR.** In general, spectators want the following:
    - An exciting course to watch.
    - A comfortable place from which to watch.
    - To be able to see a lot of the course from one location.
    - Refreshments, rest rooms and other comforts.
    - Things to do besides just watch the race; small children who are easily bored may be present.
- **SAFETY.** This is included in some of the above descriptions, but the following points should be added:
    - Smooth contoured edges on all solid obstacles, such as river banks, boulders forming eddies, stoppers, etc.
    - Large enough obstacles to prevent "wrapping" a canoe around them.
    - No possibility of foot entrapment.
- **THE FINANCIER.** The financier simply needs to know that he can afford to build and maintain the facility. Thus, he needs to be able to raise the funds through governmental or commercial sources and he may want to design a program whereby users pay for using the course.

## The hydraulic characteristics of artificial whitewater courses

Basic geometry - the length of an artificial slalom course should conform to the standards of International Canoe Federation (ICF) from 200 – 400 m, the width of 10 m or more is required for putting gates allowing manoeuvring among them, minimum depth of about 0.6 m is needed, since canoeists would have difficulty in paddling in shallower water; depth of 0.6 m of water is preferred so that so paddlers can successfully perform an Eskimo roll; however, depths great enough should be minimized to reduce costs or necessary flow rate Q.

The second group of parameters, which apparently influence the quality and difficulty of the course, includes the discharge through the course and the bottom slope. Both of them are often limited; the discharge - by insufficient capacity of upstream reservoir) or by low discharge in the river during the sport season, the bottom slope - by configuration of the ground.

The energy losses through the open channel are attributed partly to the friction of riverbed, but there are also losses attributed to the formation and dissipation of high velocity jets, hydraulic jumps. etc. These latter types of losses, so called "obstruction losses" would be determined separately and added to the mean friction loss. The roughness of the course surface is a very important parameter for optimization of the whitewater course geometry and flow conditions.

## The creation of turbulent surface flow features

There are three ways of forming hydraulic features (waves, stoppers, upstream, etc.) in whitewater courses.

1. Permanent blocks, with rounded edges for safety, to construct and divert the water flow. These should not be undercut, and their upstream surfaces should be sloped to allow the current to push boats to the surface.
2. Moveable blocks temporarily secured in position so that the flow patterns within the course can be changed from time to time and for different standards of paddling skills.
3. Rapids created by high velocity water falling into slow-moving water areas. They have no obstacles in them to injure boats or swimmers. Most of the rapids at the upstream ends of the deep pools at Nottingham are of this nature.

## Design components and construction

Two independent physical models were constructed (Tab. 1) in scale 1:13. Each model was investigated separately. The models were referred to as the Competition Course and Training Course.

*Tab. 1 Parameters of the courses*

| | Competition course | Training course | Unit |
|---|---|---|---|
| **Length** | 250 | 210 | m |
| **Flow rate** | 12.0 | 10.5 | $m^3/s$ |
| **Max slope** | 2 | 1 | m |
| **Elevation** | 4.5 | 2.0 | m |

The model of the courses was built on perpendicular plywood ribs. Bottom and banks in sections were made from plywood with steel bottom. Steel bottom and banks had very good friction similarity with reality. Intermediate pools and their banks were designed from extruded polystyrene due to the detailed difficulty.

Obstacles were made from extruded polystyrene with Neodymium magnets on the bottom, which hold the obstacles at any place (because of steel bottom), and allow them to be moved very easily at any time during the water flow. This developed concept sped up the process of tuning the obstacles.

For recalculation between reality and model, Froude similitude was used (see Similitude and hydraulic model studies p. 9). In the following tables are the recalculation for each unit used (Tab. 2).

*Tab. 2 Scale similitude*

| | | Dimension | Scale of similarity | Model scale |
|---|---|---|---|---|
| **Length** | L | M | $M_L$ | 1 : 13 |
| **Area** | S | $m^2$ | $M_L^2$ | 1 : 169 |
| **Velocity** | v | m/s | $M_L^{1/2}$ | 1 : 3.605 |
| **Discharge** | Q | $m^3/s$ | $M_L^{5/2}$ | 1 : 609.3 |

## Competition Course

Competition Course was designed as course with high difficulties – for top whitewater users. It can be used for high sports events such as

Olympic Games, World Championships or World Cups. Difficulty is ranging between WW III – V (International Scale of River Difficulty).

*Fig. 7 Overall view to model*

The course was divided into a start pool, 11 sections or pools, and a lake. Each of the parts were described separately, with focus on singularities for achieving a wide range of water scenarios. Some examples of the sections demonstrate the transfer results from model to reality. For the results interpretation the “protocol” was developed. (Fig. 8, Fig. 9).

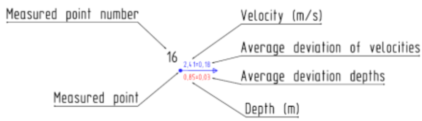


*Fig. 8 Measured point description*


*Fig. 9 Water behaviour legend*

# Stopper pool

*Fig. 10 Stopper pool*

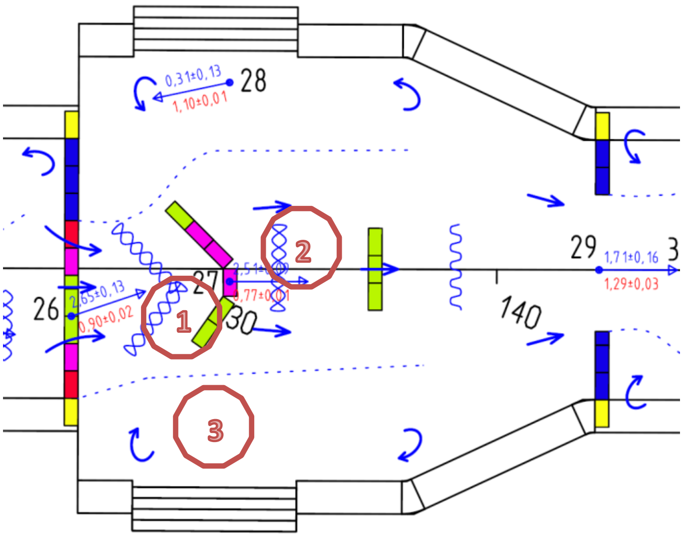


*Fig. 11 Stopper pool scheme*

As an example of modelling technique, we pick the stopper pool in the middle of the course. The pool has the first wave followed by the narrow stopper. On the bottom are recommended perpendicular V shaped configuration of movable obstacles with a perpendicular bar (sill), which helps wave and narrow stopper creation. The end of the pool finished with obstacles from both sides for establishing the water level in the pool. Maximum velocities are 2.34 m/s. Water depth is about 0.75 m, not lower than 0.65 m.

The best interpretation of the results is found in the video, but the pictures below show it as well.

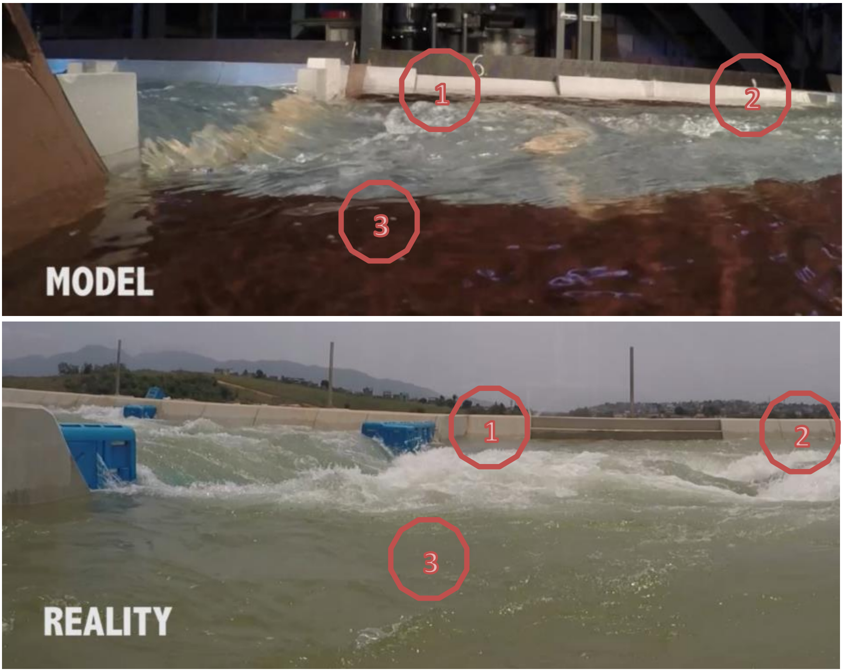


*Fig. 12 From model to reality*

For clear understanding of the above pictures (Fig. 11, Fig. 12) the number shows the important behaviour:

1. V shape stopper
2. Narrow stopper
3. Upstream

# Example 2: mathematical model – secondary clarifiers used at WWTP Prague

Purely mathematical modelling was used in following example. Knowledge about the fluid characteristics (water, sludge), which is important for sedimentation, was used from previous research. Afterwards the modelled inflow structure was built and results were confirmed.

Several possible structures and their modifications of the secondary clarifier in Prague WWTP [20] were tested in previous research. The most suitable influent for clarifiers in Prague, as well as other clarifiers in Czech, were selected from the type with circular horizontal baffles and plate separating the influent form sludge removal. Inspiration for such a construction was influent at Vienna's clarifier [12]. This influent was further modified for increasing the separation efficiency.

## Methodology

Simulations for optimization were carried out for average flow and for the maximum flow in the clarifier when its efficiency is significantly deteriorated. Results were compared with the current state.

To determine suitable solution were used next criteria:

- **Maximum velocity** – not excess of level of 0.6 m/s [28]). Higher velocities can cause deflocculating of biological flocs [7].
- **Minimizing the return currents** – they create dead spaces causing an increase of the maximum velocities of the flow. [11]
- **Uniform outflow field** – with no preferable flow in clarifier [9]

The final design was tested on[27]:

- Dead zones in the clarifier
- Shortcuts flow
- Overflowed sludge

## Mathematical model of the clarifier

Mathematical modelling by ANSYS Fluent carried out research of the influent structure by CFD. According to previous criteria, the inflow structure was changed (32 versions) and simulated (Fig. 13).

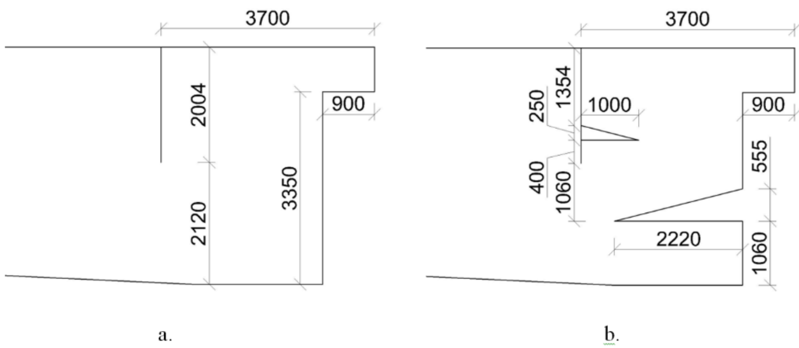


*Fig. 13 Schema of the old influent structure (a.) and new influent structure (b.)*

Testing of different variations were carried out at first in 2D. The most suitable design from the calculated results was chosen for testing in 3D with water at first and then finally in 3D with mixed liquid (water and sludge). 2D simulations were the fastest but not accurate for radial flow. The variables of flow were not corresponding to reality, but the simulations were sufficient for finding the most suitable geometry of influent structure, because there was a correlation between the 2D and 3D [20]. 3D simulation was done without sludge for testing function of the clarifier. The shortcut flow and dead zones in the clarifier were controlled. Finally, a simulation in 3D with sludge was done. Such a simulation gave real variables of the flow and real efficacy of the clarifier. Using such a procedure, it was possible to reduce the computation time to a minimum. [16]

2D investigation perfectly fulfils the criteria mentioned above and was compared with 3D and results. Using a simpler model, a correct solution was confirmed an Eulerian model for solving two mixed phases was used in simulations with sludge. Momentum for these phases was solved separately - pressure was shared. Parameters of sludge were set up by previous research according to experimental data and Dupont and Dahl equation [14].

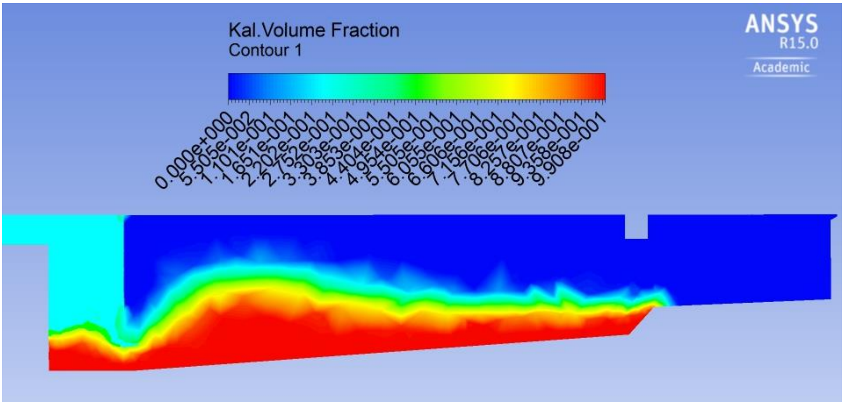


*Fig. 14 Sludge stratification in clarifier with old influent structure for inflow of 750 l/s*

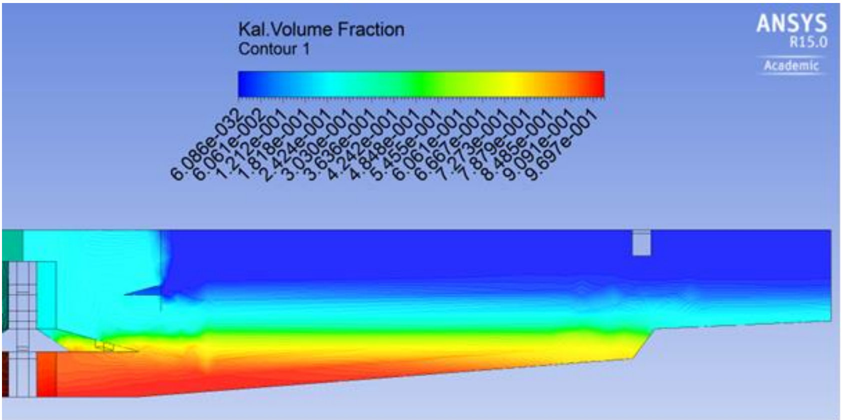


*Fig. 15 Sludge stratification in clarifier with new influent structure for inflow of 750 l/s*

## Model proved by measurements in situ

Measurements in situ lasted 5 months. Measurement equipment was installed in two independent clarifiers. The main goal of the measurements was to compare them (with and without innovation). The only difference between them was the designed inflow structure. The first clarifier called DN 1, was a reconstructed clarifier equipped with a new inflow structure. The second clarifier, called DN 3, was the original one. Both had the same water inflow and the same sludge outflow.

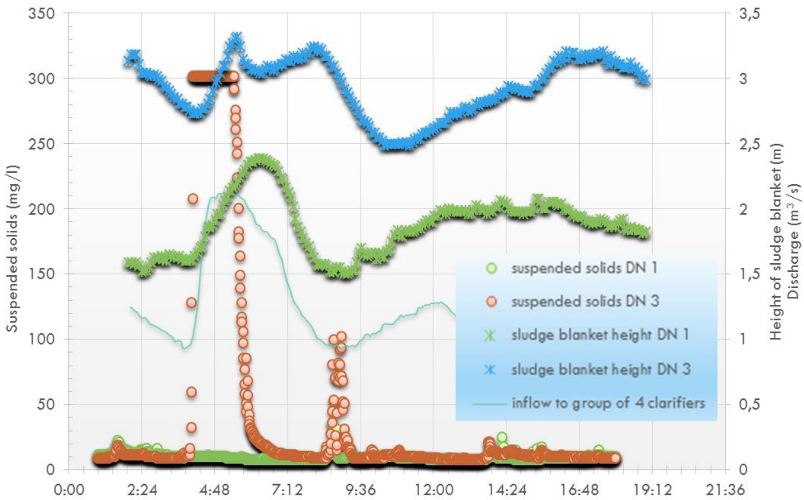


*Fig. 16 Inflow, overflowed suspended solids and height of sludge blanket*

Measurements showed the improvements in efficiency in the clarifier with the new influent structure. The sludge blanket height was nearly stable and on an acceptable level. Overflow of suspended solids were under 20 mg/l, which is the limit prescribed by the authority [24]. This limit was the break quite often found in the clarifier with old inflow structure.

When critical inflow was reached, sludge began rise out of control and could be washed out. In the reconstructed clarifier, there were values of inflow up to 950 l/s, but the critical rising of the sludge blanket level never occurred. As you can see (Fig. 16) the developed inflow structure can be used with 2,5x higher inflow than the old structure. The limit of the new structure was never higher than the limit.

New influent structure, according to measurements, has positive impact on the quality of the environment. The capacity of wastewater treatment increases. The new solution of influent reduces the overall input of pollution into the environment during high inflow to the WWTP.

# Future activities

Mathematical and physical modelling have a future. Both approaches have their place in the prediction of water flow structures. Good prediction can save a lot of money and help to avoid problems.

With physical modelling, it is possible to use more complicated flow structures, with many details and changes of details. Demonstration of the flow in the model is also one of the points for using this type of modelling.

The combination of mathematical and physical modelling is the best technique to achieve the best results, but is more expensive. We can use one approach to validate and increase the meaningful results of the second one or to extend it.

Pure mathematical modelling needs some previous knowledge of the material (for boundary conditions) and fluid behaviour (for which mathematical model to choose). Blind mathematical modelling can help with the design (when you are planning construction), but it has to come from previous simulations which are verified.

For the future, both approaches have a place in the water construction sector. We cannot omit either one, but the future is in the combination of both techniques that will give us the best and widest results.

In civil engineering, the future belongs to BIM (Building Information Modelling). Because BIM begins with 3D modelling. It stands to reason combine it with CFD. CFD is not part of the BIM yet, so for now we have to find some solutions for including it. Basic principles are shown on the next figure (Fig. 17). This technology was successfully tested on some projects. For optimization only the part with water flow is extracted, simulated and returned back to whole construction. If there is no construction conflict, the improved part is included in the whole construction. The advantage is the exchange of 3D geometry between CFD and BIM, which saves time and improve the precision.

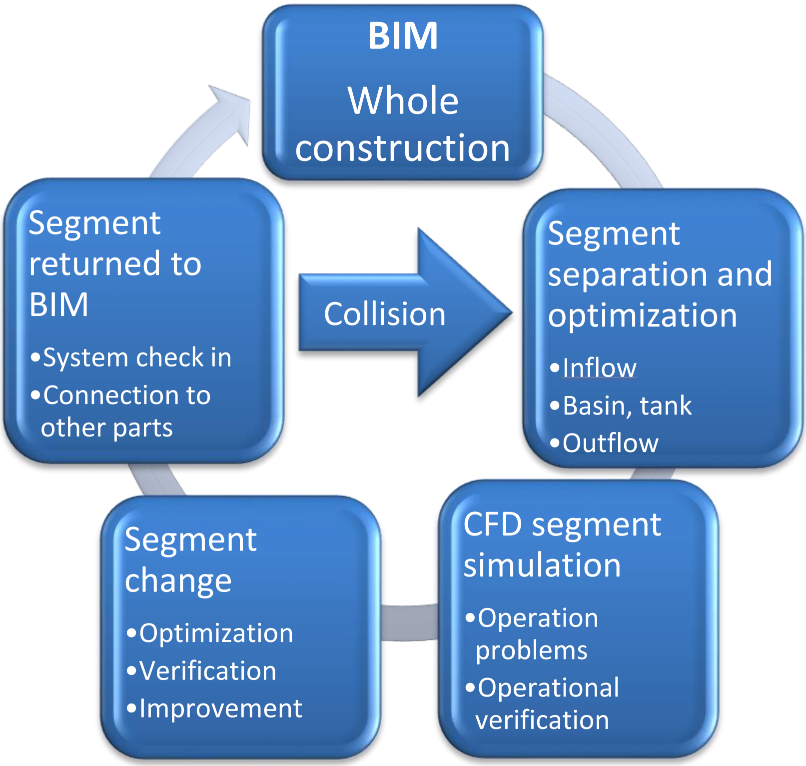


*Fig. 17 Basic principles of CFD implementation to BIM*

# References

[1] ANSYS user manual
[2] Bémová I., Pollert, J.:Návrhové charakteristiky umělých drah pro vodní slalom. (Design characteristics for slalom artificial courses), In Czech, Stavební obzor, roč.5, č.1, str. 18-21, ISSN 1210-4027, Praha 1996
[3] Buckingham, E. (1915). "The principle of similitude". Nature. 96 (2406): 396–397.
[4] Chanson, H. The Hydraulics of Open Channel Flow: an Introduction. Elsevier Publishing, ISBN 0 7506 5978 5, 2004
[5] ČSN 75 6401 (2014) - Wastewater treatment plant for a population equivalent (PE) of more than 500 (Czech)
[6] ČSN EN 872 (757349) - Jakost vod - Stanovení nerozpuštěných látek
[7] Das D., Keinath T., Parker D., Wahlberg E. (1993), Floc breakup in activated sludge plants, Water Environment Research.
[8] Duckworth, R.A. Mechanics of fluids (1977), Longman Group Ltd., ISBN 0-582-44139-0,
[9] Fitch E. B., Flow path effect on sedimentation (1956), Sewage and Industrial Wastes, Water Environment Federation, pp. 1-9.
[10] Harremoës, P. and Madsen, H. (1999) Fiction and reality in the modelling world – Balance between simplicity and complexity, calibration and identifiability, verification and falsification. Water Science and Technology. 39(9), 47-54.
[11] Imam E. (1981), Numerical Modelling of Rectangular Clarifiers, University of Windsor, Windsor, Ontario, Canada.
[12] Kainz H., Design and construction of the WWTP of Vienna (2008), Conference of Vodni Dnevi, Slovenia.
[13] Kainz H., Design and construction of the WWTP of Vienna (2008), Conference of Vodni Dnevi, Slovenia.
[14] Koníček Z., Pollert J., Kříž K. (2009), Feasibility Study of reconstruction of the old secondary clarifiers at WWTP, ČVUT Praha. (Czech)
[15] M. Lesieur, Akiva M. Yaglom, F. David (2000), New trends in turbulence, Les Houches. ISBN 978-3-540-45674-2.
[16] Maršík F. (1990), Numerical methods of continuum Mechanics, ÚT AV ČR. (Czech). Ebook available: http://www.it.cas.cz/files/u1771/Num_metody_mech_kontinua.pdf
[17] Mueller, R.K. Experimental Mechanics (1978) 18: 241. doi:10.1007/BF02324164

[18] Pollert J, R. Ugarelli, S. Saegrov, W. Schilling, V. Di Federico (2004). “The Hydraulic Capacity of Deteriorating Sewer Systems” (2004). IWA 2004, Marrakech, September 2004.
[19] Pollert J. jun., Pollert J. sen. (2016), Český hydraulický výzkum stavby pro olympijské hry Rio 2016, Stavebnictví 4/2016
[20] Pollert J., Chmátal P. (2012), Feasibility Study of reconstruction of the old secondary clarifiers at WWTP – Supplement of study, ČVUT Praha. (Czech)
[21] Pollert, J,. Vodní stavitelství a Olympijské hry v Sydney 2000 - Penrith Whitewater Stadium. (Water structures and Olympic Games 2000 - Penrith Whitewater Stadium ), In Czech, - In: Vodní hospodářství. 2001, roč. 51, č. 4, s. 90-91. ISSN 1211-0760.
[22] Pollert, J.: Artificial Whitewater courses from perpective of construction and using in capital of Czech Republic Prague. Proceedings of International conference „Las aquas bravas como recurso turístico“. Ayuntamiento Granada, Granada Spain, May 1999
[23] Punmia B. C., Jain K. A. (2005), Waste water engineering, Laxmi publications. ISBN 978-8170080916
[24] PVK, Quality of treated water from WWTP Praha. Retrieved from: www.pvk.cz
[25] Streeter, V.L., Wylie, E.B., Bedford, K.W.: Fluid Dynamics. Nith Edition, Mc Graw-Hill Book Company, Inc, ISBN 0 07 062537 9, 1998
[26] Streeter, V.L.: Hanbook of Fluid Dynamics. First Edition Mc Graw-Hill Book Company, Inc. , 1964
[27] Vesilind P. A. (2003), Wastewater treatment plant design, IWA Publisinhg. ISBN 978-1572782525
[28] Water environment federation (2005), Clarifier design, McGraw-Hill. ISBN 0-07-146416-6
[29] Zhou, S., Mak, B. W. L., Leong, E., Hon-Shing, K., Shu-kuen, I. and Tze-Kwan, L. (2010), Optimized centre-feed clarifier design for the Tai Po Sewage Treatment Works, Hong Kong. Water and Environment Journal, 24, pp. 140–146.

# Acknowledgements

This work was supported by the TAČR TE02000077 Smart Regions – Buildings and Settlements Information Modelling, Technology and Infrastructure for sustainable Development.

# Životopis

## Osobní údaje:

| | |
|---|---|
| Jméno: | Doc. Ing. Jaroslav Pollert, Ph.D. |
| Narozen: | 1. 5. 1971, Praha |
| Rodinný stav: | Ženatý, dvě děti – Jáchym (2004) a Jonáš (2007) |
| Adresa bydliště: | 182 00, Praha 8, Trojská 782/53 |
| Občanství: | Česká republika |
| Jazyky: | Anglicky, rusky |

## Vzdělání:

| | |
|---|---|
| 1977 - 1985 | Základní škola v Praze 8, Na Šutce 440 |
| 1985 - 1989 | Gymnázium v Praze 7, Nad štolou 1 |
| 1989 – 1995 | Inženýrské studium - Fakulta stavební ČVUT v Praze, obor „Vodní hospodářství a vodní stavby“ |
| 1996 - 2001 | Doktorandské studium - Fakulta stavební ČVUT v Praze, obor „Vodní hospodářství a vodní stavby“ |
| 8. 12. 2009 | Ukončeno habilitační řízení - docent |

## Zaměstnání:

| | |
|---|---|
| 1995 – dosud | Fakulta stavební ČVUT v Praze, Katedra zdravotního a ekologického inženýrství |

## Studijní pobyty:

| | |
|---|---|
| 1994: | 4 měsíční studijní pobyt The Ohio State University, Ohio, U.S.A., |
| 2004: | 4 měsíční pobyt na The Norwegian University of Science and Technology, Trondheim, Norsko |
| 2005: | 4 měsíční pobyt na The Norwegian University of Science and Technology, SINTEF, Trondheim, Norsko |

### *Odborné zaměření:*

Mechanika tekutin, proudění kapalin a suspenzí v otevřených a uzavřených profilech, matematické modelování proudění kapalin a suspenzí, fyzikální modelování proudění v otevřených korytech, odvodnění urbanizovaných území, čištění odpadních vod.

### *Ostatní aktivity:*

Sport – 4 x mistr světa ve vodním slalomu v kategorii 3xC2 muži (1995 – Nottingham, 1999 - La Seu d'Urgell, 2003 - Augsburg, 2006 – Praha), 2 místo ve světovém poháru ve vodním slalomu v kategorii C2 muži (2003), člen reprezentačního družstva vodního slalomu od 1994 – 2006, 2x 1. místo World Masters Games Turin 2013 (C1, C2)