5. HYDRAULIC MODELING IN HYDROPOWER DEVELOPMENT

5.1 GENERAL

Hydraulic modeling is an important tool with respect to development of an appropriate layout of the planned hydropower scheme. After the optimum layout and size of the high head hydropower plant has been determined with help of the economical tool of cost estimation, a preliminary design of all components of the project is completed.

As mentioned above design of hydraulic structures, such as weir, intake, bottom outlets, headrace canals etc. is based on one-dimensional approaches. This simplification of flow conditions is sufficiently accurate in special applications, i.e. design of headrace canal, weirs etc. However flow conditions at the intake, in bends of the river etc. are strongly influenced by three-dimensional effects, such as secondary flow, vortexes, fluctuating water levels and their consequences as deposition/erosion of sediments. Especially in high-head hydropower, the sediment transport of high mountainous areas is considerable and has an important impact on hydraulic structures, such as reservoir, intake, gravel and sand trap etc.

These detailed flow conditions cannot be considered in the calculations of hydropower components and have to be studied in detail by hydraulic modelling. By this procedure problem areas in the design should be found out and clarified before any action is taken to go ahead with the final design. For this reason, a hydraulic modelling is necessary to carry out.

There are two principal methods to solve hydraulic engineering problems. They are:

1. Numerical modelling
2. Physical modelling

Both methods should be applied parallel to optimize the design of the hydropower development and to utilize available resources to maximum extent. The input data are the same for both methods, procedures differ in the following aspects:

In numerical modelling governing equations of fluid flow in space are solved with mathematical algorithms. Today with the increasing capacities of computers the complex equations can be solved for the fluid dynamics and the transport of solids, such as sediment, and chemical components. Once the geometry of the river upstream the weir or dam is constructed in form of a mesh, the different alternatives of layout can be studied in detail. For instance the proper location of the intake can be investigated numerically. The preliminary design of the headrace canal and calculated velocities can be controlled. Numerical modelling can also be used for investigations of a single structure, for example the geometrical dimensions of the sandtrap can be gained by parameter studies at different sediment loads.

There are many applications for computational fluid dynamics, and the engineer gets a first impression about the flow conditions with different layouts. Especially in case of mountainous rivers, the flow is strongly determined by three-dimensional flow effects. For this reason, it is recommended to apply at least two-dimensional models, better three dimensional ones.

This kind of modelling technique is the first approach to modelling the planned hydro power plant, since it helps in the determination of a good layout, which can be studied with a physical model in detail.

Physical models are based on the similarity theory. Therefore observations made in the model are transferable by a scale to nature conditions. The construction of physical models in laboratory as well as in open fields is costly and time consuming. As the same as in numerical modelling, the geometry of the nature has to be rebuilt in the lab with help of a civil construction. The accuracy of the investigations is directly dependent on the accuracy of construction of the geometry. For this purpose manpower, engineering knowledge and material costs have to be made available. If the physical model is calibrated, results of the model can be transferred to nature. A study of different layouts is much more complicated in case of physical models, because for each small change the model has to be partly newly constructed.

Therefore it is recommended first to carry out some numerical modelling, which might reduce the number/design variations of alternatives and to optimise the remaining design proposals with help of a physical model. Using this approach, costs and time can be saved.

5.2 NUMERICAL MODELING

5.2.1 INTEGRATION OF GOVERNING EQUATIONS OF FLUID FLOW

In this context a short overview should be given about the different numerical models, their basics, and the fields of application. Like in other systems of differential equations, the governing equations describing fluid flow can be integrated as follows:

- direct integration methods, which can only be applied to extremely simplified cases
- numerical integration methods

The following paragraphs contain a brief description of the numerical integration methods most frequently used. Some more detailed information can be found in literature regarding computational fluid dynamics.

5.2.2 INTEGRATION METHODS

The best-known methods applied in numerical integration are:

Method of characteristics:

This method has received much attention in the early stages especially due to its suitability to manual solution of some hydraulic problems. It has been intensively used to study transient flow conditions in closed conduits and to analyse such problems as water hammer in pipes and penstocks, which will be discussed in the chapter “Design”.

It has also been applied to study hydraulic transients in fluid flow with open surface. There is minimal further development in this field due to the inherent problem of its application in solving complex problems.

Finite differences:

Finite differences have been intensively used, because the algorithms are amenable for implementation on digital computers. It has found many applications in hydraulic engineering, which vary from the solution of water profiles using simple difference schemes to solving more complex three-dimensional problems. The main drawback is with the discretization of two- and three-dimensional models and the representation of the boundary conditions.

For one-, two- and three-dimensional simulations a large variety of schemes have been applied to practical problems, however they are one of following categories:

1. Explicit, which sequentially solve the equations on the basis of the values of the previous time step and the known values of the present time step. The schemes are relatively easy to program but frequently have convergence problems.

2. Implicit, which solve a system of equations at every time step. The equations include the known values of the previous time step and the boundary conditions of the present step. The algorithms are numerically more intensive but present less convergence problems

Finite elements:

Some important contributions have been made in applying the finite element method to fluid flow problems. It has the advantage of allowing a more accurate representation of boundaries in two- and three-dimensional problems. The disadvantage is the intricate mathematical formulation, intensive computational effort and convergence problems. Applications have been reported for laminar flow and some problems in turbulent regime.

Finite volumes:
The method has been intensively applied in aeronautical engineering and has found extensive applications in hydraulic engineering (CFD-Computational Fluid Dynamics). The partial differential equations are transformed into total differential equations through an integration procedure. The space is divided into volumes, which may have different forms, allowing an easy representation of boundaries. Many integration algorithms are known, such as Euler, Predictor/Corrector, MacCormack, power-law, etc.

5.2.3 ONE-DIMENSIONAL METHODS

One-dimensional numerical models have a limited field of application. If more-dimensional flow effects are negligible, this model approach can be used. A typical application is the calculation of the water surface profile in a headrace channel. The required input can be divided into three groups, as already mentioned above:

- geometry / morphology
- water
- sediment

One-dimensional numerical models are normally used to simulate the water depth/elevation and mean velocity in open channels. With the help of the field measurements of the cross sectional profiles, the water levels and the corresponding discharge, the numerical model can be calibrated in such a way, that the flow conditions in nature and in the model match.
After the calibration of the model, the channel geometry for example can be changed in the numerical model. With the changing of the cross sections the water levels also differ from their original state. The result of the one-dimensional numerical simulation is the calculation of the water levels and the mean velocities. With this method several possibilities of different geometrical dimensions can be examined quickly. One of the most commonly used programs for calculating these parameters is the American program HEC-2.

There are also one-dimensional numerical models which include the effects of sediment transport. They are more complicated in handling because the discharge hydrograph has a great effect upon the sediment transport. With the help of these models, the American program HEC-6 for example, river sections can be examined with respect to sediment transport. It can be determined over a long period of time if a section is in stable conditions or not. This program should not be applied for steep mountain torrents with step pool systems in longitudinal profile. Big mountain rivers, such as Jhelum or Chitral can be simulated with one-dimensional approaches.

5.2.4 THREE-DIMENSIONAL MODELS

As mentioned above in case of steep mountain torrents, a one-dimensional simulation of the flow conditions is not sufficient, three-dimensional numerical models have to be applied then. The groups of input data are similar. The geometrical data in combination with the water surface defines a water body with three-dimensional elements. The elements are defined by the coordinates of the grid intersections.

To compute the flow conditions in the given volume the boundary conditions of the water body must be characterized. For example the water inflow and the velocity in all three directions have to be specified. For sediment transport calculations, the sediment concentration at the boundaries also must be specified. Shear stress calculations are based on roughness, which can be described as an equivalent particle diameter for all elements bordering the riverbed.

The three-dimensional model calculates, on the basis of these input data, the velocity in all three directions. The difference between the one- and three-dimensional models is that i.e. the velocity distribution, pressure distribution etc. is calculated in the three directions x, y, and z.

5.3 PHYSICAL MODELING

5.3.1 BASICS OF PHYSICAL MODELS

Physical models are small-scale reproductions in the laboratory. Physical models are defined as follows: “Any physical model for the simulation of flow process, flow states and events, which concern problems of hydraulic engineering or technical hydromechanics (Kobus).“ The decisive criterion of these models is the fact that the observations made on a small scale model must be transferable to natural conditions or exhibit a direct similarity relationship.

Since physical modelling can be considered as complex tool in hydraulics, only some aspects of basics should be discussed in this context. A comprehensive summary about different techniques with discussions of applied examples can be found in the literature of Kobus.

Usually in hydropower development flow conditions of the river upstream the planned site location are of special interest, for instance flow conditions in case of floods to the planned structures. Another important aspect is the movement of sediments in the river reach and the question how to avoid sediments entering into the intake, which causes abrasion at the runners. Both applications of physical hydropower modelling are so called long models of flow with a free water surface. One simulation can be carried out with a fixed bed, while the latter one has to be done with a moveable bed. Therefore physical models can be distinguished into

- models with fixed bed
- models with movable bed

Each model can be distinguished between the so called “short models“ and “long models“. Short models are in general models where the viscous forces are negligible in comparison to gravity and inertial forces. Overfalls and weirs are characteristic examples for this case. This kind of modelling can be applied, if detailed knowledge of one single structure regarding i.e. discharge coefficient is required.

In long models, however, long sections of the flow are rebuilt. In these models viscous forces, i.e. friction forces influence the flow conditions, they cannot be ignored. The slope of the water surface as well as energy losses have to be similar in the scale model and in prototype.

5.3.2 SIMILARITY MECHANICS AND FROUDE MODEL LAW

The fundamentals for the application of physical models are based on the similarity mechanics. “Similarity“ between nature and model implies geometrical, kinematic and dynamic similarity.
Geometrical similarity is achieved, if all geometrical lengths L_n in nature exhibit a constant ratio to the lengths L_m in the hydraulic model. This ratio is the length scale number L_r and is defined as:

$$L_r = \frac{L_n}{L_m} \qquad \textbf{(5.1)}$$

The same equations can also be derived for the kinematic similarity with the time variable t as well as the dynamic similarity with the parameter of the force F.
For free surface flow in open channels the Froude model law must be followed. In other words the geometrical similarity is achieved when the Froude number F is equal both in the scale model and in nature.

$$F_r = \frac{v_r}{\sqrt{g_r \cdot L_r}} = 1 \tag{5.2}$$

By combination with the acceleration of gravity, which is the same in the model and in the nature following scaling rules can be derived:

Lengths $$L_r \equiv \frac{L_n}{L_m} \tag{5.3}$$

Areas $$A_r = L_r^2 \tag{5.4}$$

Velocities $$v_r = \sqrt{L_r} \tag{5.5}$$

Times $$t_r = \frac{L_r}{v_r} = L_r^{\frac{1}{2}} \tag{5.6}$$

Discharges $$Q_r = v_r \cdot A_r = L_r^{\frac{5}{2}} \tag{5.7}$$

5.3.2.1 SIMILARITY REQUIREMENTS FOR FIXED BED SIMULATIONS

In the hydropower engineering long models with a free water surface are usually examined. A long section of the river or the channel is the subject of investigation. The following considerations explain the basics of long models with a fixed bed.

The discharge of a channel is characterized by following physical parameters:

- fluid parameters: ρ, ν
- flow conditions and channel: V, V_* resp. S_e , R_{hy} , g, k

The method of the dimensional analysis gives exactly four dimensionless parameters in these flow conditions:

the Froude number $$F = \frac{V}{\sqrt{g \cdot R_{hy}}} \tag{5.8}$$

the Reynolds number $$R = \frac{V \cdot R_{hy}}{\nu} \tag{5.9}$$

the slope of the energy line S_e **(5.10)**

and the relative roughness $$\frac{k}{R_{hy}} \tag{5.11}$$

It can be seen from the dimensionless parameters that the influence of viscosity and boundary roughness *k* is not negligible in this case.

This becomes important in case of small scale Freudian models, where as a consequence viscous forces have a greater significance than in nature. As long as the flow is in the hydraulically rough region in nature and in the model, this observation does not have any consequences. If the Reynolds number is large enough, changing this parameter does not cause any change in the loss coefficient λ. The loss coefficient λ is responsible for the energy losses and therefore for the energy gradient (h_v/L) in a Froude model. Having the same coefficient λ both in the scale model and in nature, the energy losses are equal and the water surface slope is the same in both cases. As a result the model gets hydraulically smoother than the prototype and the influence of viscous effects is not scaled correctly. In river hydraulics and natural flows the Reynolds number can be considered to be in the hydraulically rough region ($\cong 10^6$).

5.3.2.2 SIMILARITY REQUIREMENTS FOR MOVEABLE BED SIMULATIONS

In models of rivers with moveable bed the parameters of the soil material are added when compared to the flow conditions for a fixed bed. The density of the soil ρ_s and the diameter of the grain d_s are the additional physical values. Instead of the roughness *k* in the rigid-boundary channel the diameter of the grain is used. This substitution yields the following physical parameters:

- fluid parameters: ρ, ν
- soil material: ρ_s , d_s
- flow conditions and channel: V,V_* resp. S_e , R_{hy} , g, k

The dimensional analysis method gives exactly five dimensionless parameters:

froude number $F = \frac{V}{\sqrt{g \cdot R_{hy}}}$ **(5.12)**

sediment Reynolds number $R_* = \frac{\sqrt{gyS} \cdot d}{\nu} = \frac{V_* \cdot d_s}{\nu}$ **(5.13)**

critical shear stress $\tau_c^* = \frac{\tau}{(\gamma_s - \gamma_w) \cdot d_s} = \frac{V_*^2}{\Delta \cdot g \cdot d_s} = \left(\frac{\gamma_w}{\gamma_s - \gamma_w} \right) \cdot \frac{y}{d_s} \cdot S$ **(5.14)**

relative density $\Delta = \frac{\rho_s - \rho}{\rho}$ **(5.15)**

relative roughness $\frac{d_s}{R_{hy}}$ **(5.16)**

In general these models are much more complicated than models with a fixed bed. On one side the Froude model law has to be obtained but on the other side, the sediment transport of the river or the channel has to be simulated. Therefore the flow conditions are unsteady in the model, because the changing discharges have an important effect upon the formulation of the bed in the channel.

It is a well known fact that the sediment transport depends on the dimensionless numbers R^* and F^*, see Chapter three for details. If sediment transport processes have to be simulated in a physical model, some criteria for choosing the model scale have to be considered. The dimensionless parameters R^* and F^* have to be equal in model and in prototype. This leads to guide lines for simulation of sediment movements, where one parameter has to be chosen freely and the others are determined thereby. Moreover simulation guide lines depend on the matter of interest. If the suspended load movement shall be simulated, other methods have to be followed as in case of bed load. Special literature can be found concerning the simulation of sediment transport in the literature of Kobus.

5.4 CALIBRATION OF MODELS

5.4.1 CALIBRATION OF NUMERICAL MODELS

The aim of the calibration of one-,two-, and three-dimensional numerical modelling is to simulate exactly the flow conditions in nature.

The principle of a calibration is to use the input data which are described in the section on field measurement and to adopt the parameters of the numerical model to these flow conditions. The calibration is finished when all data of the numerical model are equal to the measured ones. If the geometry, the discharges with its water elevations and velocity components are the same, the effects of a new channel construction can be investigated.

If sediment transport was measured, the input and output of the sediment into the model must also be the same as in nature. The phenomena of erosion and sedimentation can be simulated in the same way.

Fitting the model to the natural data is achieved by changing the parameters of the model. One important parameter in this case is the Manning's *n* value. The aim of the said model is to investigate the water elevations in a long section of a channel.

The calibration of the numerical model is done in the following way:

If the channel is very long it should be divided into several calculation sections with its specific characteristical flow conditions. The flow is influenced by various factors, for example the slope, the surface roughness and the shape of the channel.

In each section several calculations of the water surface profile should be done with different Manning-coefficients until the calculated water level is equal to the measured values. The calibration is finished if the accuracy between the water levels in the model and in nature is acceptable. In case of different discharges in the channel, the calibration has to be done for each typical discharge separately.

5.4.2 CALIBRATION OF PHYSICAL MODELS

For the calibration of physical models, the same principle is used as in numerical models. The flow conditions in nature have to be simulated in the model.

Similar to the similarity requirements explained above, the calibration of physical models has to be distinguished between models with fixed bed and models with moveable bed. Because of the sediment transport, the handling of physical models and its calibration is much more complicated.

5.4.2.1 MODELS WITH FIXED BED

As already mentioned, the reproduction of the water surface and the energy losses is achieved by changing the roughness of the fixed bed model. If the flow conditions in the model are similar to the prototype the calibration is complete. After the calibration, the different problems can be investigated, e.g. changing the cross sections in a reach, changing the bed slope, etc. With the help of scale factors the conditions in the model can be applied at the prototype.

Here in this section some practical limitations of physical modelling will be discussed. To put the model into the available space of the laboratory is often a problem. The upper limits depend upon the prevailing conditions in the laboratory, whereas the lower limits are given by the similarity conditions of Froude model law.

One lower limit of the model size is the scaling of viscous effects, as explained above. The principal requirement is that the Reynolds number in the model must always be large enough to ensure turbulent flow conditions in the model.

5.4.2.2 MODELS WITH MOVEABLE BED

Besides the problem of the reproduction of the water surface and the energy losses, the practical calibration of physical models with a movable bed is much more complicated. It will be necessary to discuss the details of the calibration in another chapter. Here the main similarity requirements are given for the sediment simulation. They have to be applied in physical models with moveable bed:

The similitude as regards the beginning of the sediment transport is important. The transportation should start in both the model and prototype at a similar discharge.
Also the similitude as regards the transport capacity should be kept. This means that the ratio of the sediment capacity between the model and the prototype should be maintained constant for each discharge.

In effect there are two time scale numbers in this kind of model.

Table Of Content

Table of Figures

Table of Tables