Buletinul Ştiinţific al Universităţii POLITEHNICA Timişoara
Seria HIDROTEHNICĂ
TRANSACTIONS on HYDROTECHNICS
Volume 59(73), Issue 2, 2014

Numerical Limitations of 1D Hydraulic Models using MIKE11 or HEC-RAS software

Robert F. Beilicci[1] Erika Beilicci[1]

Abstract – MIKE 11 is a advanced hydroinformatic tools, professional engineering software package for simulation of one-dimensional flows in estuaries, rivers, irrigation systems, channels and other water bodies. MIKE 11 is a 1-dimensional river model. It was developed by DHI Water • Environment • Health, Denmark. The basic computational procedure of HEC-RAS for steady flow is based on the solution of the one-dimensional energy equation. Energy losses are evaluated by friction and contraction / expansion. The momentum equation may be used in situations where the water surface profile is rapidly varied. These situations include hydraulic jumps, hydraulics of bridges, and evaluating profiles at river confluences. For unsteady flow, HEC-RAS solves the full, dynamic, 1-D Saint Venant Equation using an implicit, finite difference method. The unsteady flow equation solver was adapted from Dr. Robert L. Barkau's UNET package. Fluid motion is controlled by the basic principles of conservation of mass, energy and momentum, which form the basis of fluid mechanics and hydraulic engineering. Complex flow situations must be solved using empirical approximations and numerical models, which are based on derivations of the basic principles (backwater equation, Navier-Stokes equation etc). All numerical models are required to make some form of approximation to solve these principles, and consequently all have their limitations.

Key words: Basic equations, numerical modelling, limitations

I. INTRODUCTION

The basic computational procedure of HEC-RAS for steady flow is based on the solution of the one-dimensional energy equation. Energy losses are evaluated by friction and contraction / expansion. The momentum equation may be used in situations where the water surface profile is rapidly varied. These situations include hydraulic jumps, hydraulics of bridges, and evaluating profiles at river confluences.

For unsteady flow, HEC-RAS solves the full, dynamic, 1-D Saint Venant Equation using an implicit, finite difference method. The unsteady flow equation solver was adapted from Dr. Robert L. Barkau's UNET package.

HEC-RAS is equipped to model a network of channels, a dendritic system or a single river reach. Certain simplifications must be made in order to model some complex flow situations using the HEC-RAS one-dimensional approach. It is capable of modeling subcritical, supercritical, and mixed flow regime flow along with the effects of bridges, culverts, weirs, and structures.

HEC-RAS is a computer program for modeling water flowing through systems of open channels and computing water surface profiles. HEC-RAS finds particular commercial application in floodplain management and flood insurance studies to evaluate floodway encroachments. Some of the additional uses are: bridge and culvert design and analysis, levee studies, and channel modification studies. It can be used for dam breach analysis, though other modeling methods are presently more widely accepted for this purpose.

HEC-RAS has merits, notably its support by the US Army Corps of Engineers, the future enhancements in progress, and its acceptance by many government agencies and private firms. It is in the public domain and peer-reviewed. The use of HEC-RAS includes extensive documentation, and scientists and engineers versed in hydraulic analysis should have little difficulty utilizing the software.

Users may find numerical instability problems during unsteady analyses, especially in steep and/or highly dynamic rivers and streams. It is often possible to use HEC-RAS to overcome instability issues on river problems. HEC-RAS is a 1-dimensional hydrodynamic model and will therefore not work well in environments that require multi-dimensional modeling. However, there are built-in features that can be used to approximate multi-dimensional hydraulics.

MIKE 11 is a advanced hydroinformatic tools, professional engineering software package for simulation of one-dimensional flows in estuaries, rivers, irrigation systems, channels and other water bodies.

MIKE 11 is a professional engineering software package for the simulation of flows, water quality and sediment transport in estuaries, rivers, irrigation systems, channels and other water bodies. MIKE 11 is a user-friendly, fully dynamic, one-dimensional modelling tool for the detailed analysis, design, management and operation of both simple and complex river and channel systems. With its exceptional flexibility, speed and user friendly environment, MIKE 11 provides a complete and

1 Politehnica University of Timisoara, Faculty of Civil Engineering, Department of Hydrotecnnics, Timisoara, Str. George Enescu, no. 1/A, Timisoara, Romania, www.hidro.upt.ro

effective design environment for engineering, water resources, water quality management and planning applications. The Hydrodynamic (HD) module is the nucleus of the MIKE 11 modelling system and forms the basis for most modules including Flood Forecasting, Advection-Dispersion, Water Quality and Non-cohesive sediment transport modules. The MIKE 11 HD module solves the vertically integrated equations for the conservation of continuity and momentum, i.e. the Saint Venant equations.

The MIKE 11 is an implicit finite difference model for one dimensional unsteady flow computation and can be applied to looped networks and quasi-two dimensional flow simulation on floodplains. The model has been designed to perform detailed modeling of rivers, including special treatment of floodplains, road overtopping, culverts, gate openings and weirs. MIKE 11 is capable of using kinematic, diffusive or fully dynamic, vertically integrated mass and momentum equations. Boundary types include Q-h relation, water level, discharge, wind field, dam break, and resistance factor. The water level boundary must be applied to either the upstream or downstream boundary condition in the model. The discharge boundary can be applied to either the upstream or downstream boundary condition, and can also be applied to the side tributary flow (lateral inflow). The lateral inflow is used to depict runoff. The Q-h relation boundary can only be applied to the downstream boundary. MIKE 11 is a modeling package for the simulation of surface runoff, flow, sediment transport, and water quality in rivers, channels, estuaries, and floodplains. The computational core of MIKE 11 is hydrodynamic simulation engine, and this is complemented by a wide range of additional modules and extensions covering almost all conceivable aspects of river modelling.

Fluid motion is controlled by three basic principles: conservation of mass, energy and momentum. Derivatives of these principles are commonly known as the continuity, energy and momentum equations. These principles are among the first taught in basic fluid mechanics, and they form the foundation of the field of hydraulic engineering. However, as situations become increasingly complex, we lose track of these essential principles. Basic equations are replaced by empirical approximations, and mathematical calculations with numerical models. Determining an equivalent surface roughness of a floodplain is far more difficult than estimating an equivalent roughness height or a Manning's roughness coefficient; solving a backwater equation for an irregular channel would be an arduous task without the assistance of a numerical model.

Numerical models come in a wide range of shapes and flavours – one, two or three dimensions, steady or unsteady flow conditions etc. All are based on derivations of the basic principles. All are required to make some form of numerical approximation to solve these principles. All have their limitations.

The objective of this paper is to promote a basic awareness of how numerical models operate and to draw attention to some of the more common limitations that are implicit to this operation, in the hope that this may encourage these models to be used in (and only in) the manner for which they are intended.

II. FLUID MECHANICS

Fluid mechanics is the study of fluids statics or dynamics. As a branch of mechanics, fluid flow is governed basic principles. However, application of these principles is far easier in theory than in practice due to the complexity of fluid flow. It is therefore necessary to make assumptions that simplify the application of the controlling equations, and use numerical modelling techniques to obtain solutions of complex problems. The principles of fluid mechanics can be found in many textbooks. Simple descriptions of the basic concepts and some of the more common simplifications, necessary for understanding how these principles are used by numerical modelling software, are provided below.

The mechanics of fluid flow is governed by three basic principles of conservation:

Mass – The Lomonosov-Lavoisier law states that the mass of a closed system remains constant. If a system is open then the rate of increase in the mass within the control volume is equal to the cumulative mass flowrate into the control volume:

$$\int_S m dS = \frac{dM}{dt} \tag{1}$$

If the fluid is incompressible then the equation can be simplified by replacing mass with volume. For steady-state conditions, this further simplifies to Q = constant.

Momentum – Newton's second law states that the rate of change of momentum of a body is proportional to the resultant force acting on the body and is in the same direction. For a control volume, this may be written in differential vector format as:

$$\frac{d}{dt}\left(m x \vec{v}\right) = \sum \vec{F} \tag{2}$$

For a Newtonian fluid and assuming constant density and viscosity, the equation of motion may be written in the x-direction as:

$$\begin{aligned}\rho\left(\frac{\partial v_x}{\partial t} + v_x\frac{\partial v_x}{\partial x} + v_y\frac{\partial v_x}{\partial y} + v_z\frac{\partial v_x}{\partial z}\right) = \\ \rho g_x - \frac{\partial P}{\partial x} + \mu\left(\frac{\partial^2 v_x}{\partial x^2} + \frac{\partial^2 v_x}{\partial y^2} + \frac{\partial^2 v_x}{\partial z^2}\right)\end{aligned} \tag{3}$$

Similar equations derived for the y and z directions are collectively known as the Navier-Stokes equations.

Energy – The first law of thermodynamics states that the net energy supplied to a system is equal to the increase in energy of the system and the energy that leaves the system as work is done:

$$\frac{dE}{dt} = \frac{dQ_h}{dt} - \frac{dW}{dt} \tag{4}$$

The 'Backwater Equation' is a derivative of the Energy Equation for steady flow along a streamline:

$$\frac{\partial H}{\partial S} = -S_f = -f\frac{1}{D_H}\frac{v^2}{2g} \qquad (5)$$

III. 1D NUMERICAL MODELS

Computational fluid dynamics can be defined as a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyse problems involving fluid flows. The term "CFD model" is commonly used to refer to a high-order numerical model capable of solving complex flow situations with relatively few simplifications.

In reality, all numerical models are CFD models (even a simple spreadsheet solution of the backwater equation). There are generally considered to be two methods of analysing fluid motion: by describing the detailed flow pattern at every point in the flow field (small scale or differential analysis), or by examining a finite region and determining the gross effects of and on the region (finite or control-volume analysis).

The complexity of real fluid flow makes it impossible to solve the governing equations without making some form of simplifying approximation, even with the use of complex models and fast computers. Common practices include: simplification of the spatial and geometric properties, assumption of steady or quasi-steady flow conditions, neglect of fluid properties that would have negligible influence in the circumstances being investigated, and use of empirical formulae to approximate flow characteristics.

Hydraulic models may be categorized by the spatial and temporal simplifications that the model employs. Each category has associated with it a number of fluid property and dynamic assumptions.

One-dimensional models assume that the flow is in one direction only, and there is no direct modelling of changes in flow distribution, cross-section shape, flow direction, or other two- and three-dimensional properties of the flow. The channel geometry is typically represented as a series of cross-sections at fixed (but not necessarily uniform) intervals. Although often considered to be relatively simplistic by modern standards, one-dimensional modelling remains a useful and valid tool in many situations. One-dimensional hydraulic models may be categorized as steady or unsteady. While these appear, superficially, to be similar and share many of the same limitations, the basic hydraulic principles to numerically solve these two situations are very different. Steady-state numerical models are in most cases based on a derivative of the 'backwater' (or Energy) equation, while unsteady models are based on a derivative of the Saint Venant (or Momentum) equation. Each solution has its advantages and disadvantages, and neither is appropriate in all situations. The derivation and implicit limitations of these solutions are described in the sections below.

One-dimensional models make a number of approximations in line with their simplistic nature. Some are so obvious that they (hopefully) cannot be missed, while others are not so well recognised. Flow properties must be calculated based on characteristic properties of the cross-section (eg hydraulic diameter, average velocity). Some software packages try to provide greater flexibility by dividing each cross-section into sub-areas (such as the main channel, left and right overbanks), then applying various weighting factors to the flow distribution between the sub-areas and the travel distance of each component. More complex software packages can simulate quasi-2D situations as a series of inter- linked channels, however the definition of flow path and length is inflexible. Even with these abilities, one-dimensional modeling is only appropriate for modelling well-defined and constant flowpaths; the model cannot match the flexibility of two- and three-dimensional modelling necessary for representing complex channel/floodplain interactions.

A less obvious simplification common to many numerical models (HEC-RAS, MIKE 11) is to assume that the grade of the channel is small, nominally less than 1:10, and therefore the sine and cosine of the channel slope can be assumed equal to zero and unity respectively.

Rather than using a physically derived coefficient, such as the Darcy friction factor f, most numerical models estimate friction losses using an empirical approximation such as Manning's coefficient. While Manning's equation has been in use for over 120 years, it is perhaps this universal acceptance that has lead to evident ignorance about the limitations of the equation. A popular misconception is that Manning's roughness coefficient is a dimensionless constant, whereas in reality it has units (s/m1/3) and is dependent upon the hydraulic radius. Care must therefore be taken, not only in the estimation of appropriate roughness coefficients, but to realise that although a model has been calibrated for one particular discharge, the performance may be different for other flow conditions. Additionally, because there is no direct modelling of two- and three-dimensional flow effects, the roughness coefficient must account for the contribution of these aspects to hydraulic losses in the channel.

Numerical models usually solve the **backwater equation** between adjacent cross-sections using an iterative procedure called the standard step method, where the backwater equation is integrated as:

$$\left(d + z_0 + \frac{\alpha v^2}{2g}\right)_2 - \left(d + z_0 + \frac{\alpha v^2}{2g}\right)_1 = LS_f + C\left|\frac{\alpha v_2^2}{2g} - \frac{\alpha v_1^2}{2g}\right| \qquad (6)$$

where the losses are separated into friction and contraction/expansion losses with Sf a representative friction slope between the two sections, L is the distance between the sections, C is a contraction or expansion coefficient, d is the flow depth, z0 is the invert level.

The primary assumption of the integrated backwater equation used in steady-state numerical modelling is that the flow is gradually varied (Henderson 1966). This implies that changes along the channel, such as cross-section shape, invert level, flow depth and pressure distribution, are relatively small over short distances. The backwater equation has questionable or no accuracy in: areas of rapid acceleration or deceleration, where the assumption of a hydrostatic pressure distribution is no longer valid, areas of large turbulence and/or energy loss, and areas of large change in cross - section property where the assumption that the representative friction slope and contraction/expansion losses can be estimated by some combination of the section properties at each end.

Unlike steady-state modelling, which uses a solution of the continuity and energy equations, unsteady modelling is based on a solution of the continuity and momentum equations. The derivation of these equations into a format suitable for one-dimensional modeling is complex but fairly well documented. The vertically integrated equations of continuity and momentum, commonly known as the **Saint Venant equations**, may be presented as:

$$\begin{aligned} &\frac{\partial Q}{\partial x}+\frac{\partial A}{\partial t}-q=0 \\ &\frac{\partial Q}{\partial t}+\frac{\partial Qv}{\partial x}+gA\left(\frac{\partial z}{\partial x}+S_f\right)=0 \end{aligned} \tag{7}$$

where Q is the discharge, q is the lateral inflow (per unit length), A is the flow area, z is the free- surface elevation and v is the velocity.

While the backwater equation is based on a steady-state differential form of the Energy equation, the Saint Venant equation based solutions can model unsteady flow conditions. Many software packages, including HEC-RAS and MIKE 11, adopt an algorithm that cannot accommodate two boundary conditions at the same boundary. As a consequence they cannot model supercritical flow, for which both discharge and water level are controlled by the upstream boundary. Instead, supercritical flow conditions are ‘solved’ by suppressing the convective acceleration as the Froude number increases.

III. CONCLUSIONS

The study of hydraulics and fluid mechanics is founded on the three basic principles of conservation of mass, energy and momentum. Real-life situations are frequently too complex to solve without the aid of numerical models. There is a tendency among some engineers to discard the basic principles taught at university and blindly assume that the results produced by the model are correct. Regardless of the complexity of models and despite the claims of their developers, all numerical models are required to make approximations. These may be related to geometric limitations, numerical simplification, or the use of empirical correlations. Some are obvious: one-dimensional models must average properties over the two remaining directions. It is the less obvious and poorly advertised approximations that pose the greatest threat to the novice user. Some of these, such as the inability of one-dimensional unsteady models to simulate supercritical flow can cause significant inaccuracy in the model predictions.

REFERENCES

[1] Henderson, F.M. (1966). Open Channel Flow. MacMillan Company, New York, USA.

[2] Liggett, J.A. (1975). Unsteady Flow in Open Channels WRP Publ. Vol. 1 Fort Collins, USA, pp. 29-62. Liggett, J.A. (1994). Fluid Mechanics. McGraw-Hill, New York, USA.

[3] MIKE by DHI (2014) MIKE 11 A Modelling System for Rivers and Channels Reference Manual

[4] USACE (20138). HEC-RAS River Analysis System Hydraulic Reference Manual, v4.0, California, USA