Mathematical modelling techniques for flood propagation in urban areas

F. ALCRUDO Universidad de Zaragoza, SPAIN
alcrudo@posta.unizar.es

SUMMARY

This document presents a description of mathematical modelling techniques for flood propagation in urban areas as developed and implemented in computational models within Work Package 3 (WP3) of IMPACT project. The corresponding deliverable is D3.1.1. The approach adopted within IMPACT project to deal with flood propagation in urban areas is a local one. This means that the aim is to represent water depth and velocity in the vicinity and around buildings as opposed to computing an averaged or general flow picture.

1 INTRODUCTION

The effects of flooding are most striking in heavily populated or urban areas. This drives the need for efficient prediction of flood propagation in such environments. During the last decade, an enormous amount of work on mathematical modelling has been performed. The advent of more capable computing machines has paved the way to the use of mathematical models in all aspects of engineering, including hydraulics and also, more specifically, flood propagation. Flood propagation modelling can be defined as the art of quantitatively describing the characteristics and evolution of the flow that is set up when a large amount of water moves along the earth surface in an uncontrolled way. The sizes and scales of terrestrial floods can span several orders of magnitude as the affected surface areas do. Their nature and origin can also be very varied, ranging from slow, reservoir-filling like inundations due for instance to intense, long lasting rains, to extreme, short, violent floods that can follow the failure of a dam or other control structure.

When speaking of modelling techniques for urban flooding, the aim is to develop a mathematical framework numerically implemented into a computer code that can describe the flow behaviour in the area occupied by a city or a group of dwellings. Within IMPACT project work the goal has been to implement such technology into open field flood propagation models. This is to say that the resulting computational tool can model the propagation of the flood in open and urban areas in the same way in a straightforward manner.

The use of computational models in flood propagation dates back to the sixties, although it has been during the last decade that, thanks to the availability of high performance computers, an explosion of publications reporting the development and use of flood propagation models has occurred. Stringent regulations regarding hazard and risk mitigation and management affecting dam owners, basin authorities, land use planning bodies etc… that have been slowly but constantly enforced by states worldwide for many years now, have much to do with this
situation too. Progress in flood propagation modelling technology is directly related to the following issues:

- Understanding the flow processes relative to the problem
- Formulation of appropriate mathematical laws describing it
- Development and tuning numerical techniques to solve them
- Validation of model output against representative experimental data and real life observations

Very important to every day engineering practice are model interfacing with other engineering tools, such as topography and visualization software, data acquisition and geographical information systems (GIS), that can considerably simplify and speed up problem set up and analysis time. Most of the models commercially in use are particularly strong on these issues.

Validation of the output obtained from computer simulations is one of the most important steps in model development in order to quantify the uncertainty associated with its predictions. Every effort should be made to confront model results with real life data. Since it is difficult to retrieve well documented extreme flood events, resorting to laboratory scaled down experiments is a useful means to gain knowledge in model performance. The amount of output provided by present day models can be overwhelming and is usually analyzed by means of sophisticated graphical software that can easily lead the engineer to think that displayed information is inherently correct. A strong knowledge of a given model basic assumptions and limitations, sound judgment as well as a critical attitude is always advisable.

It can be said that the flow characteristics of a flood are relatively well understood from a fluid dynamics standpoint, which is not yet the case when the phenomenon is coupled with erosion and deposition of material, debris flow etc...

The fundamental mathematical laws that govern the phenomenon, the Navier-Stokes equations, are well known. However their solution is practically impossible for the spatial and temporal scales of any real case leading to the need of simplified descriptions such as the Shallow Water model which is now the most widely spread despite its many shortcomings.

2 MATHEMATICAL MODELS OF FLOOD PROPAGATION

2.1 Navier-Stokes models

The propagation of a flood over a portion of the earth crust is just a three dimensional (3-D) time dependent, incompressible, fluid dynamics problem with a free surface. Putting aside the erosion and deposition effects which are the subject of a separate branch of study, the flow can be considered single phase. The well known Navier-Stokes (NS) equations (in 3-D) perfectly describe the dynamics of a portion of fluid (Landau et al. 1959). However the flow is turbulent and of geographical size, and the cascade of length and time scales present is huge what impairs any attempt to solve the 3-D NS equations by any means.

In order to circumvent the problem of turbulence the NS equations can be averaged in time (Spurk 1997) to obtain the so called Reynolds-Averaged Navier-Stokes equations (RANS) that describe the mean flow. The effects of the turbulent fluctuations on the mean flow are
taken care of by means of turbulence models, i.e. formulations whereby the stresses due to turbulence are related to the mean flow variables (Wilcox 1993). It can be said that there are currently dozens of turbulence models in use, each adapted to a particular fluid dynamics situation. The RANS equations are of wide use in industrial fluid mechanics and aerodynamics but are still too complex to be applied to describe flood propagation, mainly due to the resolution that would be needed to make such a procedure meaningful. Furthermore, since turbulence models are developed to be well suited to a specific situation, those currently available may not even make sense in a flood propagation scenario for which, to our knowledge, no turbulence characteristics have been studied.

Further to the problem of turbulence the NS and RANS based models have the added difficulty of the air-water interface movement. The free surface moves with the velocity of the fluid particles located at the boundary, and therefore its position is one of the unknowns that must be solved for during the computational procedure. The problem lies in that the equations of motion only apply to the space occupied by the fluid which is not known a priori. Several techniques have been developed to circumvent these difficulties, mostly relying on an iterative procedure. Among them the Volume of Fluid (VOF) (Hirt and Nichols 1981), Marker in Cell (MAC) (Welch et al. 1966) methods have gained a reputation of accuracy and robustness and are now widely used but their application to flooding problems has not yet been possible due to the extraordinary computational power needed. It is true that simulations of propagating and breaking waves as well as dam break flows have been simulated (Tome and McKee 1994, Lemos 1996, Stansby et al. 1998, Mohapatra et al. 1999, Maronnier et al. 1999, Zwart et al. 1999, Dimas et al. 2000). However, these simulations are limited to idealised, two dimensional (in the vertical plane 2-DV) cases with no practical interest or to limited size industrial applications such as tank sloshing. Furthermore, in order to simplify the problem only laminar flows are considered or either the diffusive terms are dropped from the NS equations thus solving the in-viscid or Euler equations. Fully three dimensional simulations are usually limited to slow or steady flow (Casulli and Stelling 1998, Sinha et al. 1998, Ye and McCorquodale 1998) what is not the case in flooding scenarios or applied only to solve local flow effects (Stelling and Busnelli 2001).

Given their demand on computational power it is unlikely that NS or even Euler (inviscid) models can be applied to practical flood propagation calculations in urban areas in the near future.

2.2 Shallow Water models

The usual approach in order to simplify the mathematical description is a depth averaging procedure of the NS equations that leads to the Shallow Water equations (SWE) model (Benqué et al. 1982). Alternatively, the SWE can be derived from mass and momentum conservation in the plane of motion (the earth surface) (Cunge et al. 1980 and others). The depth averaging procedure eliminates from scratch the free surface location problem which is now simply placed as the depth above the bottom surface. In attention to their importance in flood propagation modelling the 2-D SWE are reproduced below:

\[
\frac{\partial U}{\partial t} + \frac{\partial}{\partial x}(F + F_d) + \frac{\partial}{\partial y}(G + G_d) = H + I
\]

(1)
where $\mathbf{U}$ is the variables vector, $\mathbf{F}$ and $\mathbf{F}_d$ and $\mathbf{G}$ and $\mathbf{G}_d$ are the convective and diffusive fluxes vectors in the $x$-direction and $y$-direction respectively (in the plane of movement). $\mathbf{H}$ is the friction and slope source term vector and $\mathbf{I}$ the infiltration source vector. The expressions for vectors $\mathbf{U}$, $\mathbf{F}$ and $\mathbf{G}$ above read:

$$
\mathbf{U} = \begin{pmatrix} h \\ h_u \\ h_v \\ h_v \end{pmatrix}, \quad
\mathbf{F} = \begin{pmatrix} h_u \\ h_u^2 + gh^2/2 \\ h_u v \\ h_v^2 + gh^2/2 \end{pmatrix}, \quad
\mathbf{G} = \begin{pmatrix} h_v \\ h_v u \\ h_u v \\ h_v^2 + gh^2/2 \end{pmatrix}
$$

(2)

where $h$ is water depth, $u$ and $v$ the averaged (over the depth and time in a turbulent flow) cartesian velocity components and $g$ the acceleration of gravity. The corresponding expressions for the diffusive fluxes $\mathbf{F}_d$, $\mathbf{G}_d$ can be written:

$$
\mathbf{F}_d = \begin{pmatrix} 0 \\ -\varepsilon h \frac{\partial u}{\partial x} \\ -\varepsilon h \frac{\partial v}{\partial x} \\ -\varepsilon h \frac{\partial v}{\partial y} \end{pmatrix}, \quad
\mathbf{G}_d = \begin{pmatrix} 0 \\ -\varepsilon h \frac{\partial u}{\partial y} \\ -\varepsilon h \frac{\partial v}{\partial y} \end{pmatrix}
$$

(3)

where $\varepsilon$ is a kinematic viscosity coefficient that accounts for the fluid kinematic viscosity, the turbulent eddy viscosity and the apparent viscosity due to velocity fluctuations about the vertical average (see below). Finally $\mathbf{H}$ and $\mathbf{I}$ read:

$$
\mathbf{H} = \begin{pmatrix} 0 \\ gh(S_{ox} - S_{fx}) \\ gh(S_{oy} - S_{fy}) \end{pmatrix}, \quad
\mathbf{I} = \begin{pmatrix} -i_r \\ -1/2 \cdot \mathbf{u} \cdot i_r \\ -1/2 \cdot \mathbf{v} \cdot i_r \end{pmatrix}
$$

(4)

Here $S_{ox}$ and $S_{oy}$ are the bed slopes in the two cartesian directions which are assumed small:

$$
S_{ox} = -\frac{\partial z_B}{\partial x}, \quad S_{oy} = -\frac{\partial z_B}{\partial y}
$$

(5)

where $z_B$ is the bottom surface function and $S_{fx}$, $S_{fy}$ are the friction slopes, usually represented by means of an empirical formula, in this case Manning’s:

$$
S_{fx} = -\frac{n^2 u \sqrt{u^2 + v^2}}{h^{4/3}}, \quad S_{fy} = -\frac{n^2 v \sqrt{u^2 + v^2}}{h^{4/3}}
$$

(5)

Coriolis forces effects and wind stresses have not been included because of their minimal influence on flood flow. Finally $i_r$ is the infiltration rate ($dh_{infiltrated}/dt$) into the ground/sinks that can be estimated with empirical formulae (Mahmood and Yevjevich 1975). Although infiltration effects are usually associated with irrigation (Hauke 2002), they could be of
appreciable importance in flooding problems (Fiedler and Ramirez 2000) when the residence
time of water over porous unsaturated soil is large or when sinks are present as happens with
sewer networks in urban flooding (Haider 2001).

Figure 1: The Shallow Water description

Mathematically speaking the SWE description reduces the problem from a 3-D free surface
NS/Euler incompressible (velocity divergence free) problem to a 2-D compressible one where
the depth of the water layer plays the role of the density (Landau and Lifshitz 1959). A further
simplification is obtained by considering only one-dimensional (1-D) movement of the water
body when the flood propagates in a well delimited, narrow valley (see for instance Soares
2002) that can be coupled to a 2-D formulation downstream when the floodplain is attained to
or when several branches come together (Soares 2002). The compressibility introduces new
difficulties in the solution procedure but these are by far offset by the simplifications brought
about with the SWE.

However the SWE are not a true mathematical representation of the movement of water over
the earth surface. Putting aside errors originated from a numerical integration of the SWE, the
following known shortcomings of this approximation stand:

i) Vertical velocities are neglected/not considered (therefore vertical accelerations are identically zero)
ii) The pressure field is assumed hydrostatic
iii) The bottom slope is assumed small (such that the sinus of the slope angle can be approximated to the angle itself)
iv) A uniform horizontal velocity field is assumed across the water layer
v) Turbulence effects are usually ignored
vi) Friction formulae are usually taken from uniform flow conditions
Points i) and ii) are intimately related, and iii) in turn appears also, but not only, as a consequence of them. A real flood event implies the movement of water in the vertical direction too and restrictions i) to iii) altogether make the SWE lack fundamental physical properties which are well represented within the NS model. It is likely that situations in which vertical movement is substantial be poorly represented in a SWE simulation. A method proposed by Stansby and Zhou (1998) consist of correcting for non-hydrostatic pressure distribution when slope is large or separation in the vertical plane is expected but in the context of a 2-D vertical RANS model. Also an ingenious idea within the SWE context that has been put to work recently (Zhou et al. 2002) considers the movement of the layer of fluid in a horizontal plane over a succession of a piecewise flat discontinuous bed.

It is clear that a boundary layer must extend from the bed to the free surface and the fact that the horizontal velocity field is considered uniform across the depth of the water layer (point iv) may also be responsible for important deviations between model predictions and real observations. When deriving the SWE by depth averaging the NS equations, the dispersion effects due to non-uniformity of the velocity profile can be quantified and are usually approximated as a diffusion of momentum:

$$\frac{\partial}{\partial x} \int_{\text{free surface}} \hat{u} \hat{v} dz + \frac{\partial}{\partial y} \int_{\text{free surface}} \hat{u} \hat{v} dz = D \cdot \text{div} (\text{grad} \hat{u})$$

(6)

where \(\hat{u}\) and \(\hat{v}\) are the deviation of the velocity components with respect to the depth-averaged values \(u\) and \(v\). \(D\) is a diffusion coefficient with dimensions of a kinematic viscosity. The effect is analogous to the viscous and turbulent stresses. However as pointed out by Benqué et al. (1982), the equivalent kinematic viscosity, \(D\), may be more than an order of magnitude higher than the kinematic viscosity of the fluid thus competing with the turbulence effects. Despite this fact the dispersion effects due to non-uniformity of the velocity profile are to, our knowledge, not usually taken into account or not reflected in the technical literature. Estimation of the diffusion coefficient can be a difficult task but may be worth a bit of attention in flood propagation models, at least to learn about its relative importance compared to the rest of competing effects. An interesting alternative to assuming a uniform velocity profile is the use of several shallow water layers, an approach that is frequently used in stratified ocean models but that so far has not found application in flood propagation. In the work reported here a uniform vertical velocity profile has been considered and hence dispersion effects have not been taken into account.

The turbulent contribution to the momentum equations in hydraulics (point v) has received more attention (Rodi 1988). For instance Stansby and Zhou (1998) apply the k-e turbulence model in the 2-D vertical plane, but this is not so frequent in SWE models (an example is Nadaoka and Yagi 1998 for river flow). When this is the case, tidal studies in coast lines and harbors as well as circulation in lakes or dispersion studies include some form of turbulence model (Yulistiayanto et al. 1998) while these are always present in atmospheric SWE simulations (for instance, Gelb and Gleeson 2001). However when it comes to flood propagation turbulence modelling is not considered an important matter. In some cases a constant eddy viscosity coefficient is used which seems to play more the role of a tuning parameter rather than a characterization of the turbulence characteristics. In the course of this work turbulence effects have not yet been considered.
Altogether it seems that the importance of diffusion (either due to turbulence or to dispersion of the velocity profile) has not been evaluated in SWE models of flood propagation.

Another issue that deserves attention regards the friction laws with bottom (point vi). Traditionally empirical formulae of the Manning or Chézy type, scaling with the square of the depth averaged velocities are assumed. This has been experimentally verified for uniform flow but nothing can be said in a highly unsteady situation. It is well known that the friction law significantly diverges from the uniform flow form in unsteady and oscillatory pipe flow (Wylie and Streeter, 1993) and hence a similar effect can be expected in free surface flows. Furthermore the friction coefficients are almost universally considered constant along the flow path disregarding variation in bed roughness and topography (see Nujic in CADAM 1999, Aronica et al. 1998 for variable friction coefficient applications). Slinn et al. (1998, 2000) have used a linear friction term in SWE simulations of alongshore currents near beaches. Nadaoka and Yagi 1998 take into account the effects of vegetation in the generation of horizontal shear in shallow flows, an issue linked to differential or non uniform bottom friction.

Finally down the ladder of mathematical complexity, one finds also kinematic models or even flat pond models in which only conservation of mass is considered and the flow is thought of as locally steady between connected ponds. Although extremely simple this method which completely disregards the dynamic effects of a flood has been successfully used to model practical cases (Estrela 1999, Horritt and Bates 2001).

Despite its many shortcomings, it can be said that most mathematical models of flood propagation currently in use are based upon the SWE and it seems that it will continue to be so for a few more years at least.

3 NUMERICAL SOLUTION OF THE SWE

The numerical solution of a given set of differential equations is defined by the discretization strategy, the mesh used and the numerical scheme implemented. In turn, for evolutionary problems the numerical scheme is usually made up of a separate time and space integration (except for combined space-time integrations such as, for instance, the Lax-Wendroff method). These issues are briefly described below for the case of the SWE.

3.1 Discretization strategies

The SWE system being of hyperbolic character in time it represents an evolutionary problem in the form of propagating waves. Therefore a time marching procedure starting with a given initial condition in space, supplemented with boundary conditions along the time path is the proper mathematical conceptual treatment. This is usually accomplished by the method of lines whereby first a space like discretization is made and the resulting Ordinary Differential Equation (ODE) is then solved in time.

The spatial discretization can be made with one of the following approaches: Finite Difference, Finite Volume and Finite Element methods. The use of the former is progressively decaying due to being geometrically less flexible than the other two. Nevertheless there are several successful packages both free and commercial that are based using finite differences (for example FLWDWAIV, Mike21). In order to enhance the flexibility
of finite difference discretizations, coordinate transformations can be used, but these complicates the calculation procedure and still is difficult for practical flooding applications. It is generally used in conjunction with Cartesian grids. Despite these facts, the finite difference method is used overwhelmingly in time discretizations.

The finite volume formulation is now the most widespread modelling strategy within the SWE approximation. The domain under study is divided into a certain number of non overlapping finite volumes and the SWE cast in integral form are applied individually to each one of them. The finite volume method can be applied to both structured and unstructured meshes and any geometrical shape of the elementary volumes including the mixed type. This procedure guarantees (a priori) the conservation of the mechanical properties like mass and momentum, is extremely flexible, conceptually simple and considerably intuitive, what explains its popularity. It must be said that finite volumes and finite differences are equivalent in 1-D and, depending on the type of discretization and grid used, also in higher dimensions and are often applied mixed (Wang and Liu 2001). Finite volume schemes can be cast in cell-centred or cell-vertex formulations, and there is evidence that the latter can maintain higher order of accuracy with distorted grids.

The finite element method relies upon a variational formulation of the motion equations. Its main advantage stems from its rigorous mathematical foundation that allows a posteriori error estimation. However it is conceptually more difficult and much fewer authors choose this option. Furthermore some finite element formulations can be proved to be equivalent to the finite volume method. The program TELEMAC developed at EDF (Hervouet and Petitjean 1999) is a good example of one of the earliest successful applications of the finite element method to flood propagation, although the software has lately evolved towards blending with finite volumes. The way of finite element discretizations into flood propagation models is most hindered by the difficulty to include approximate Riemann solvers and shock capturing operators in as a natural way as finite volumes do.

During IMPACT work, the finite volume discretization technique has been chosen for its flexibility and ease of implementation in complex geometries and to practical problems.

3.2 Mesh configurations

Closely related to the strategies described above is the meshing procedure adopted. The finite difference method is usually linked to structured, Cartesian lay outs. If boundary conforming is needed it is usually achieved by means of mappings. The finite element and finite volume methods can be applied to both structured and unstructured grids (usually made up of triangles). The latter allows a more flexible geometric treatment, but the indexing and bookkeeping tasks required result in a larger memory and cpu time overhead in comparison to the former. Furthermore all interpolation needed in the definition of interface fluxes is considerably more complicated (Sleigh et al. 1998). However the unstructured grid approach easily allows mesh motion and refining in areas of the domain where higher flow resolution is needed. This can be done a priori which is more common (Sleigh 1998, Soares 1999) or during the flow solution process what poses more difficulties.

A relatively new trend in this field is the use of Q-tree or Quad-tree grids that share properties of both structured and unstructured meshes. The initial lay out is cartesian and a grid refining procedure progressively divides and subdivides only some of the cells to achieve the required resolution of the flow field or its boundaries. Thus the Q-tree approach is intrinsically associated with mesh refining. Special interpolation-extrapolation operators are needed. Usually this approach, that uses cartesian cells, is supplemented with specific treatments at the boundaries to deal with irregular geometries and even moving boundaries what can be
very interesting to model land slide induced floods (Causon et al. 2000 and 2001). Within IMPACT project work both structured (Cartesian and non-cartesian) and unstructured grids have been used.

### 3.3 Numerical schemes

The approximate evaluation of the different terms of the SWE needed for any discretization strategy and mesh configuration is discussed here. The SWE represent a system of nonlinear hyperbolic differential equations (that changes to parabolic-hyperbolic if diffusion terms are present) and a proper numerical treatment strongly influences the quality of the computed solutions.

Most numerical schemes in use today perform a separate spatial-temporal discretization, whereby the spatial derivative terms are firstly discretized and then the resulting ordinary differential equation (ODE) is integrated in time. The use of formally second order accurate operators in both space and time is presently quite common, but not universal particularly in practical applications.

Among the important issues in flood propagation stands the location and propagation of wave fronts. Hence the importance of the shock capturing capabilities of the methods in order to accurately represent the celerity and intensity of the water fronts in flooding simulations. Shock capturing algorithms are based upon a conservative discretization of the conservation equations aided with some sort of artificial dissipation. The latter can be linear as in the early methods or more recently non-linear leading to the so called high resolution schemes that can capture discontinuities (water fronts) without the characteristic wiggles of earlier linear methods.

In the work reported here the scheme used is of the high resolution type, based upon Roe’s Riemann solver and flux limiters and cast in finite volume formulation as explained in what follows. Firstly the differential equations are written in conservative integral form:

\[
\frac{\partial}{\partial t} \int_U U dV + \int_S (F \cdot n_x + G \cdot n_y) \cdot dS = \int_V (H + I) dV \tag{7}
\]

where \(dV\) represents an elementary surface area and \(dS\) an elementary line. The integral form is then applied to every one of the finite volumes into which the domain of integration has been discretized, as is shown in the figure below. If the average value of the vector variables \(U\) over cell \((i,j)\) is \(U_{i,j}\), defined by the average:

\[
U_{i,j} = \frac{1}{V_{i,j}} \int_{V_{i,j}} U dV \tag{8}
\]

where \(V_{i,j}\) is the surface area of the cell. Then a discrete version of the integral equation valid in the given cell is:

\[
\frac{\partial}{\partial t} U_{i,j} + \frac{1}{V_{i,j}} \int_{S_{i,j}} (F \cdot n_x + G \cdot n_y) dS = \frac{1}{V_{i,j}} \int_{V_{i,j}} (H + I) dV \tag{9}
\]

where \(n_x\) and \(n_y\) are the components of the normal vector to the side of the cell as shown in the figure.
Figure 2: Finite volume discretization

A further discretization of the flux terms is performed as:

\[ \sum \int_{S_{ij}} (F \cdot n_x + G \cdot n_y) \cdot dS \approx \sum_{k=1}^{4} (F^* \cdot n_x + G^* \cdot n_y)_{wk} dS_{wk} \]

\( w_k = (i \pm 1/2, j \pm 1/2) \quad k=1,2,3,4 \)

where now \( F^* \) and \( G^* \) are the components of the discretized or numerical flux vector, and \( w_k \) makes reference to the k-th wall or side of the cell. In a quadrilateral cell there are 4 sides but the number of sides can vary according to the mesh morphology.

The particular characteristics of the scheme depend mainly on the definition of the flux approximation (or numerical flux function), \( F^* \) and \( G^* \). In the course of this work the flux vector splitting technique of Roe has been used for the computation. Using the shorthand:

\[ (F G^* \cdot n)_{wk} = (F^* \cdot n_x + G^* \cdot n_y)_{wk} \]

(11)

gives rise to the following expression for the numerical flux:

\[ (F G^* \cdot n)_{wk} = \frac{1}{2} \left[ (F G^*_{R} \cdot n)_{wk} + (F G^*_{L} \cdot n)_{wk} - \left| \tilde{J}_{RL_{wk}} \right| (U_R - U_L) \right] \]

(12)

The quantities \( F G \) and \( \tilde{J} \) with sub indexes R (right) and L (left) are computed using variables \( U_R \) and \( U_L \) to the right and left of the corresponding cell side \( w_k \). \( U_R \) and \( U_L \) can be taken as the values at the neighbouring cells (first order interpolation) or by means of a MUSCL procedure (second order interpolation). \( \tilde{J}_{RL_{wk}} \) is the matrix whose eigenvalues are the modulus of the eigenvalues of matrix \( \tilde{J}_{RL_{wk}} \), which is the Roe approximate matrix of the
system flux jacobian $J_{wk}$ of the exact flux function, $(FG \cdot n)_{wk}$ evaluated at cell side $w_k$. More explicitly:

$$J_{wk} = \frac{\partial (FG \cdot n)_{wk}}{\partial U} = \frac{\partial F}{\partial U} \cdot n_x + \frac{\partial G}{\partial U} \cdot n_y = A \cdot n_x + B \cdot n_y$$  \hspace{1cm} (13)

which is the standard Roe’ Riemann solver procedure and formulae.

Once the spatial discretization has been performed, the resulting ODE must be integrated in time. As regards time integration, the vast majority of models use explicit methods, mostly Runge-Kutta, that combine well with formally second (or higher) order spatial discretizations. A time step restriction of the CFL type must be respected and usually this is more stringent than what is needed for time accuracy. This is done in present work by means of a semi explicit two step Runge-Kutta procedure (explicit for the fluxes and implicit for the sources) as follows:

$$\frac{U_{ij}^{n+1/2} - U_{ij}^n}{\delta t} \left[ 1 - \theta \cdot \delta t \cdot \left( \frac{\partial H}{\partial U} \right)_n \right] = \left[ -\frac{1}{V_{ij}} \sum_{k=1}^{4} (FG \cdot n)_{wk} \cdot dS_{wk} + H_{ij} \right]^{n+1/2}$$  \hspace{1cm} (14)

$$\frac{U_{ij}^{n+1} - U_{ij}^{n}}{\delta t} \left[ 1 - \theta \cdot \delta t \cdot \left( \frac{\partial H}{\partial U} \right)_{n+1/2} \right] = \left[ -\frac{1}{V_{ij}} \sum_{k=1}^{4} (FG \cdot n)_{wk} \cdot dS_{wk} + H_{ij} \right]^{n+1/2}$$  \hspace{1cm} (15)

and the parameter $\theta$ controls how much implicit the computation is: $\theta=1$ corresponds to an Euler implicit scheme and $\theta=0$ to totally explicit one. In most practical situations $\theta$ must be equal or greater than 1/2.

Special treatment of the discretization of the bottom slope source terms are needed in order to balance flux and source contribution in a still water situation. There are several appropriate source term discretizations to fix this problem. Some regard an upwinding of the source term, others a splitting between pressure and convective forces (with asymmetrical treatments) or a flux lateralisation that work well to solve the problem. These techniques, however, will not be discussed here since they are subject of a separate task.
4   SHALLOW WATER MODELS FOR URBAN AREAS

There are several modelling techniques that can be embedded into open field flood propagation models to deal with urban inundation. The choice depends on the type of representation required. If only a rough estimation of water levels and velocities at a general scale is needed, then a simple coarse mesh calculation with a high friction coefficient representing the overall increase in flow resistance due to the presence of buildings and urban structures is enough. This simplest approach consists of representing urban environments as areas of reduced conveyance by simply ascribing a high bed friction coefficient. Manning’s roughness coefficients as high as 0.5 have been reportedly used to account for the presence of a city. During Impact project work evidence suggests that friction coefficient figures are subject to scale effects that depend on building density, scale of flooded area and ratio of flow depth to building height.

However, this approach, which is very frequently used in urban flood modelling, does not provide a local description of the flow, but rather an averaged picture of the phenomenon. Even if the mesh resolution is high, down to the order of one metre or even less, the computed flow features are not fully physically meaningful because the local effects of buildings are not present. In fact when non uniformities in the flow variables in the urban area arise they are not due to the local presence of buildings but to other causes (bottom slope, flow evolution or interactions etc…). The friction approach is successful in simulating roughly the area covered by the flood and, if friction coefficients are adequately tuned, the flood duration. In many circumstances this information is enough and can provide very valuable help in emergency planning and risk mitigation studies. An excellent example can be found in ENEL-PIS work for the UE RESCDAM project (RESCDAM 2000, 2001).

A step further is brought by the concepts of Urban Porosity and Transmissivity that are used to represent the effect that the area subject to flooding is only a fraction of the total surface area, hence affecting the mass conservation equation. This approach has been successfully used by Braschi and Gallatti (1989) who proposed the method and many others thereafter (Hervouet et al. 2001). A disadvantage is the lack of momentum interchange between the flow and the buildings. This can be arranged via the friction term in the momentum equation by artificially increasing the roughness in the area where buildings are present as explained in the previous paragraphs (Testa et al. 1998).

The increased resistance and the urban porosity approaches both provide an averaged view of the city-flood interaction and hence no local effects can be told from the output of such type of simulation. The fluid dynamics information provided can have the appearance of being local because the flow variables (water depth and velocity vector) are given for every grid point of the domain. However they represent averaged or smeared values of those quantities over a certain surface area somehow related to the grid spacing and the building density.

These techniques are devised for two-dimensional models and are best suited for modelling large areas where it is impractical or plainly impossible to seek high resolution of topographic and flow features due to problem size (although they could eventually be applied within one-dimensional models this does not seem natural). The increased resistance and urban porosity approaches are successful in simulating roughly the area covered by the flood and the flood duration provided friction coefficients are adequately tuned. In many circumstances this
information is enough and can provide very valuable help in emergency planning and risk mitigation studies.

There is frequently a need to know not only whether a given area is going to be flooded or not but also what local inundation conditions are likely to occur. If a more detailed description of the flood is envisaged, then the presence of buildings must be explicitly introduced in the computational model. Such level of detail is needed if a quantitative measure of the threat posed by the flood is required, or simply if a precise assessment of the flood effect and its interaction with the city is sought.

Within IMPACT project, work has focused around techniques that can provide insight into local flow conditions, at the cost of higher computational cost or problem size reduction. In particular four strategies have been considered:

1. A one-dimensional (1-D) treatment of the city area whereby it can be represented as a channel network (Tanguy et al. 2001).
2. Local friction based representation of buildings and obstacles to flow in a two-dimensional (2-D) approach.
4. Vertical walls (detailed 2-D meshing and resolution of the streets and city areas, incorporating buildings as solid walls).

### 4.1 One dimensional (1-D) channel network representation

The first alternative is capable of providing local flow information at low computational cost, although problem set up may require considerable work and expertise for network layout and data management. Problems may arise at junctions, in particular if these are numerous, or in wide areas where some flow features can be lost because the flow is markedly 2-D. Another difficulty of 1-D channel models lies in its interaction with larger area models. If the urban flood is the result of flooding of surrounding terrain, appropriate coupling between the urban network and the outer flood plain model is needed. In this case the Shallow Water equations are reduced to one dimension:

\[
\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}}{\partial x} = \mathbf{H} + \mathbf{I}
\]  

(16)

where \( \mathbf{U} \) is the variables vector and \( \mathbf{F} \) the convective fluxes vector in the direction of movement. Usually no diffusion is considered in 1-D models. \( \mathbf{H} \) is the friction and slope source term vector and \( \mathbf{I} \) the infiltration source vector. The expressions for vectors \( \mathbf{U} \), \( \mathbf{F} \), and \( \mathbf{H} \) above read:

\[
\mathbf{U} = \begin{pmatrix} A \\ Q \end{pmatrix}, \quad \mathbf{F} = \begin{pmatrix} Q \\ Q^2/A + gl_1 \end{pmatrix}, \quad \mathbf{H} = \begin{pmatrix} 0 \\ gA(S_{ox} - S_{fx}) + gl_2 \end{pmatrix}
\]  

(17)

where \( A \) is the normal section surface area, \( Q = Au \) is the discharge and \( g \) the acceleration of gravity. The friction and bed slopes are analogous to their 2-D counterparts. \( I_1 \) is the pressure term that reads:

\[
I_1 = \int_0^h \rho g (h - \eta) \, d\eta
\]  

(18)
in complete analogy with the 2-D formulation. The term $I_2$ represents lateral force reactions due to cross section variations:

$$I_2 = \int_0^h \rho g \sigma(\eta)(h - \eta) \, d\eta$$

(19)

where $\sigma(\eta)$ is the cross section width at elevation $\eta$ above the minimum.

The differential equations listed above are solved numerically with the aid of the methods described in previous section with given initial and boundary conditions. Hence the definition of the numerical fluxes and time integration schemes are the same save for the dimensional simplification. The computational procedure solves the equations for every reach and boundary conditions are applied at the ends of the reach. In the external nodes of the net, the boundary conditions are imposed according to the flow state in that point and the information of the flood. At the ends connecting two or more reaches the boundary condition comes from the compatibility conditions at the node. The specific feature of the 1-D representation is that of the coupling of the different reaches (streets). Usually conservation of mass an equality of total head are imposed according to the formulae below:

$$0 = \sum_{j=1}^{N_k} Q_j s_j \quad k = 1 \ldots NN$$

(20)

where $Q_j$ is the flow of the reach, $s_j$ is the sign of the flow (entering or leaving the junction node) and $N_k$ is the number of reaches joining at node $k$ and $NN$ is the total number of nodes connecting two or more reaches. As regards the head equations:

$$H_i = H_m$$

(21)

for all reaches $l, m$, connecting at the junction as figure 3 below shows.

![Figure 3: Sketch showing the coupling between different reaches](image-url)

It is interesting to note that the application of the channel network representation to real cases requires coupling between the external 2-D propagation model outside the city and the inner
1-D model. Coupling between 1-D and 2-D flood propagation models is the subject of a separate work.

Figure 3: An example 1-D mesh with coloured with bottom elevation

4.2 Local friction based representation of buildings

Representation of buildings as local areas with increased friction coefficient within a 2-D simulation can provide the needed resolution to capture local flow effects with little extra cost. This approach is easy to set up, for local friction can be treated as another field variable and provides reasonably accurate results. The approach can be sketched in figure 4 below:

Figure 4: Different friction representations of a group of buildings (left picture, in black): a) As an area of overall increased friction (centre, in red). b) As a region with local spots of high friction (right, in red).

As the picture makes clear, introducing structure into the friction distribution can greatly enhance resolution and, hopefully, accuracy of model output. As a practical example, figure 5 shows a view of the bed friction distribution in the city centre corresponding to a practical computation performed during IMPACT project. Streets, squares etc… can be easily distinguished, even if a little blurry.
Figure 5: Coloured representation of the Manning roughness in a downtown area embedded into a larger domain in a practical computation.

One of the difficulties of this approach lies in the determination of appropriate Manning roughness or friction level that leads to a practical blocking of the flow in the area occupied by a building or a given obstruction.

The table below shows estimates of global Manning’s roughness coefficient for different terrain coverage. These are the values that would be used within approach corresponding to the central picture of figure 4, i.e. taking the urban area flooded as a continuum.

<table>
<thead>
<tr>
<th>LAND USE</th>
<th>Manning Roughness</th>
</tr>
</thead>
<tbody>
<tr>
<td>Natural reach</td>
<td>0.035</td>
</tr>
<tr>
<td>Crop land</td>
<td>0.06</td>
</tr>
<tr>
<td>Thin Forest</td>
<td>0.08</td>
</tr>
<tr>
<td>Medium Forest</td>
<td>0.1</td>
</tr>
<tr>
<td>Dense Forest</td>
<td>0.15</td>
</tr>
<tr>
<td>Urban Areas</td>
<td>0.4</td>
</tr>
</tbody>
</table>

Table 1: Manning roughness coefficients

However the roughness to be ascribed to represent a single building would presumably be considerably higher than the value assigned to an urban area in the table. During Impact project work evidence suggests that friction coefficient figures are subject to scale effects that depend on building density, scale of flooded area and ratio of flow depth to building height.

In order to get insight into the appropriate values for simulation of building impedance a study was carried out in a simple configuration of buildings making up an imaginary city as shown in figure 5 below. This corresponds to earlier runs on one of the benchmarking
configurations of IMPACT project. Flow corresponds to a flood wave and is from left to right, and different values of Manning’s roughness coefficient were used until saturation was reached.

Figure 6: Imaginary city composed of quadrangular buildings represented as local bed friction

Results of the study can be seen in the following figures that compare water depth evolution computed with the model for different values of the bed friction coefficient at different positions located amidst the imaginary buildings. The computation cover an area of about three squared meters, and the blocks representing the city are squares of 15cm side. The mesh is made of about 10000 points (159x63) and the sides of the domain are rigid walls. Only some of the gauging points chosen are shown here.

Figure 7
Figure 8

Figure 9
The sequence of friction coefficients is 0.016 (baseline friction in the modelled area), 0.1, 0.25, 0.5, 1.0, and 2.0. As can be seen from the sequence of plots the computed history of water depth approaches the observed values as Manning’s roughness is increased until a saturation situation is reached at a value between 0.5 and 1.0. A closer exam shows that too high friction coefficients lead to slight instabilities in model output. A rather moderate value of 0.025 is apparently enough in this example to reproduce the observed behaviour.

Despite the encouraging results of this and other tests performed, it is difficult to estimate a priori in a practical situation the correct friction level required to represent different constructions. Furthermore, it has been found during the study that the friction value depends on the intensity and characteristics of the flood. Since friction formulae have always a strong dependence (usually parabolic) on water velocity, lower values of the friction coefficient will be needed to model buildings in a fast stream flood than on a slowly varying inundation. Another problem lies in that buildings turn practically into local water storage tanks which is not the case in reality. Some sort of urban porosity treatment would be needed to offset this effect. On the other hand, coupling the urban area model with the surrounding terrain flood propagation model is straightforward.

Despite its many shortcomings, the local friction approach for urban flood modelling has proven useful yet simple to implement in current flood propagation models.

4.3 Bottom elevation technique (2-D topography building representation)

The topography based approach involves placing buildings upon the bottom representation within a 2-D calculation scheme. This can be easily done after the meshing stage of problem set up by rising the grid points that fall within a building area to the roof top elevation. Often some sort of mesh adaptation will be needed to accurately represent the city.

Although buildings protrude vertically from the surface up, the discrete slope representation is not infinite as the method leads to think, but is extremely steep and depends on cell size. If the discretization is coarse (mesh size of the order of building height), the slope is of order 1, whereas if the discretization is very fine (mesh size much smaller than building height) the slope can reach several orders of magnitude.

This indeed violates one of the SWE model assumptions, namely that the bed slope is so gentle that the sinus and the tangent can be replaced by the angle itself. This is never the case in this situation, even in the most favourable conditions (coarse grids). This fact, however, does not cause the model to be invalid for the following reason: Putting aside complex three dimensional effects that are well known (such as the saddle vortex etc …) and that were never intended to reproduce, what a vertical obstacle placed in a stream causes is the stagnation of the fluid around it, exactly in the same way as an internal boundary (what in fact is). Hence the flow can be considered as shallow all around the building as is sketched in Figure 12.

The problem here regards the numerical representation of such giant bottom slopes that translate into very large contributions of the source terms and can, in some cases, destabilize or even destroy the computation. The details of the numerical treatment of source terms are left aside in this document since it is the subject of a different task. Nevertheless there are two
strategies to cope numerically with this situation: a) Devise a numerical integration scheme that is robust enough to withstand such severe source term forcing in the vicinity of buildings; b) modify the numerical scheme locally whenever such a situation is encountered.

\[ H = h + \frac{v^2}{2g} \]

Figure 12: Sketch showing flow stagnation in front of abrupt bottom elevation (building).

The first strategy has proved to work well with the numerical method described in section 3.3 as long as the time integration is kept fully implicit. The second method works also well with little extra computational cost and can be easily implemented into the basic numerical scheme.

There are several options of implementing the needed logic. Either to check for extremely large slopes (\( S_0 > 1 \)) and then set source terms and mass flux to zero and momentum flux to only the pressure contribution, which is the simplest but not exempt from error (water depths are of the order of building elevation) or check not only for large slopes but also for normal stagnation depth of the flow larger than building height. If stagnation depth of normal incoming flow is larger than building height then the building is likely to be overtopped and clearly a loss of significance of the SWE model would occur.

Any of the two numerical strategies produces the same results, namely when water reaches such adverse bed slope its momentum is abruptly reduced and if flow head is lower than building elevation then stagnates. In case flow head is large enough, water can overtop the building and it is submerged.

Figures 13 and 14 show the imaginary city made up of simple quadrilateral buildings used in previous section for model testing. The first figure corresponds to a coarse mesh representation (slope of the order of 1), and the second to a finer mesh one (\( S_0 > 1 \)). Finally Figure 15 shows a sequence of a flooding simulation on the same imaginary city by means of the bottom elevation technique.
Figure 13: Example of group of buildings represented as bottom elevation on coarse mesh.

Figure 14: Example of group of buildings represented as bottom elevation on finer mesh.
Figure 15: Sequence of a flooding simulation test with the bottom elevation technique.
4.4 Vertical walls

We refer here to a solution of the problem by means of a precise 2-D meshing procedure that resolves the streets and city areas, incorporating buildings as solid walls. Put in other words, a careful 2-D meshing of the area subject to flooding, excluding buildings from the computational domain. Depending on the computational model architecture it can be done by blocking grid cells occupied by buildings or by meshing them around so that buildings are treated as impervious zones. This is indeed the most accurate city representation that can be obtained from a SWE simulation. The two dimensional flood propagation model is then run in the void area as usual.

Figure 16 left, below, shows the flooding simulation of the imaginary city used for development and testing of the techniques described in this report. The plot is coloured by the Froude number of the flow. As can be seen in this example the detailed meshing technique can provide a very detailed picture of the flow. Supercritical regions can be clearly seen around the corners of the buildings as well as dead water zones in their wakes.

This method can theoretically provide the highest accuracy because the assumptions of the underlying model equations (SWE) are less likely to be violated and the topography is more accurately represented. However, in practical cases, the meshing procedure can be extremely complex, particularly if structured grids are used. As an example, Figure 16 right, below shows a mesh used in a practical urban flood simulation, where it can be seen the heavy stretching of the mesh required to represent a complex street pattern in a populated area.

![Figure 16: Two examples of computations using the vertical walls technique.](image)

The problem of grid distortion can be somewhat alleviated by using unstructured meshes, but yet, if the town area is embedded into a larger flooded region and the computational cost must be kept within reasonable limits quite a high degree of stretching is nevertheless needed. Overall it can be said that the method is the best suited urban flood modelling save for the burden that mesh set up means in practical complex situations.
5 PERFORMANCE OF THE DIFFERENT TECHNIQUES

In this section a brief comparison of the performance of the different techniques explained is presented. The simulated flood corresponds to the imaginary city made up of several quadrilateral shaped buildings that has been used to test the different modelling techniques and their implementation into flood propagation models as shown in previous sections. A sketch is shown in Figure 17 below showing the building lay out (green squares) and the gauging locations (black crossed circles). The comparison is not exhaustive since it represents a small sample of flooding conditions and geometrical lay outs on very simple topographic data (practically flat bottom throughout).

![Figure 17: Sketch of the imaginary city used for testing of urban flood modelling techniques.](image)

Furthermore the imaginary city corresponds in fact to a scale down model and hence the dimensions of buildings are considerably reduced with respect to actual ones. Nevertheless the comparison of water depth evolution data at several probe positions located around and between the buildings can provide a valuable measure of what can be expected from the different modelling techniques developed and implemented during the course of IMPACT project work. A more thorough comparison and benchmarking have been performed under a task 3.1.3 of WP3 work package and the reader is referred to the corresponding report for further details. Also a validation of modelling techniques for urban flooding within a case study application against real life data has been conducted under task 3.1.4 of WP3 work package.

The set of Figures 18 to 25 shows a comparison of water depth elevation at the different gauge positions as predicted by the different modelling techniques. As can be seen, error in water depth is within twenty per cent of observed values for all the gauges and techniques used, save perhaps the one dimensional approach that clearly can not reproduce two dimensional effects and hence its predictions deviate form those of other methods and observed results when these are dominant. Another effect that can be told of the plots is the fact that the bottom friction technique usually under predicts water depth figures. This can perhaps be attributable to the fact that, as previously discussed, building surface area is turn into storage area in this representation, and hence actual water flowing in the streets is reduced.
Figure 18: Water depth history at gauging point 3.

Figure 19: Water depth history at gauging point 4.
Figure 20: Water depth history at gauging point 5.

Figure 21: Water depth history at gauging point 6.
Figure 22: Water depth history at gauging point 7.

Figure 23: Water depth history at gauging point 8.
Figure 24: Water depth history at gauging point 9.

Figure 25: Water depth history at gauging point 10.
6  CONCLUSIONS

This document describes the modelling techniques developed, implemented and tested during work on Work Package 3 (WP3) of IMPACT project. These are the one dimensional, bed friction, bottom elevation and vertical walls methods. The first one solves the one dimensional SWE whereas the other three solve the two dimensional SWE, and hence are more computer intensive. However, part of the advantage of the 1-D approach is lost because special arrangements must be made to couple the 1-D urban area to the wider area flood model (flood plain or valley model) which can be laborious, particularly if many streets have a connection with the wider area model.

Preliminary tests performed during model development are very encouraging and have shown that any of the four techniques listed can provide a reasonable picture of the inundation of a city. The 1-D approach fails when the flow is markedly two dimensional, such as in squares or plazas, as well as in complex junctions as it could be expected. The bed friction approach is the most straightforward to implement and run, but suffers from the strongest deficiencies: The value of the saturation roughness adequate to represent buildings can vary from one situation to another as it is subject to complex scale effects that have not yet been determined. Furthermore if the flow velocity is small the technique can completely fail, and finally there is a storage effect in the area occupied by buildings, although this effect is in some cases realistic.

The bottom elevation technique is also straightforward to implement and requires little extra programming if the underlying flood propagation model is robust and accurate enough. In the tests run during this work, the base flood model was based upon a high resolution semi implicit scheme that withstood well the abrupt source term rocking, but other simpler or less stable methods can give rise to stability or accuracy problems. In case the urban area modelled is complex, the bottom elevation method can lead to very fine meshes, if all streets and buildings are to be adequately represented. Finally the vertical walls method (a detailed meshing of the city) probably provides the most accurate results, but the set up and computational effort required can be considerably higher.

During the course of IMPACT project work a critical assessment of these modelling methods has been performed by means of benchmarking and comparison with laboratory data as well as the case study exercise.

ACKNOWLEDGEMENTS

The authors wish to acknowledge the financial support offered by the European commission for the IMPACT project under the fifth framework programme (1998-2002), environment and sustainable development thematic programme, for which Karen Fabbri was the EC project officer.

The contents of this paper are based mainly upon data and information produced within WP3 (Flood Propagation) of the project. The overall contribution made by the IMPACT project team is also recognised. IMPACT team member organisations comprise:
REFERENCES


Spurk, H.S., (1997), Fluid Mechanics, Springer-Verlag, Berlin, Germany


Wilcox, D.C., (1993), Turbulence modeling for CFD, DCW Industries Inc., La Cañada, USA.


