Enhancing river model set-up for 2-D dynamic flood modelling

Pierfranco Costabile* , Francesco Macchione

University of Calabria, Department of Environmental and Chemical Engineering, LAMPI (Laboratorio di Modellistica numerica per la Protezione Idraulica del Territorio), via P. Bucci, cubo 42/B, 87036 Rende, CS, Italy

ARTICLE INFO

Article history:
Received 7 August 2014
Received in revised form 9 January 2015
Accepted 11 January 2015
Available online

Keywords:
Accuracy
Flood mapping
Case studies
2-D modelling
Hydrologic/hydraulic modelling
River model

ABSTRACT

Flood hazard mapping is a topic of increasing interest involving several aspects in which a series of progress steps have occurred in recent years. Among these, a valuable advance has been performed in solving 2-D shallow water equations in complex topographies and in the use of high resolution topographic data. However, reliable predictions of flood-prone areas are not simply related to these two important aspects. A key element is the accurate set up of the river model. This is primarily related to the representation of the topography but also requires particular attention to the insertion of man-made structures and hydrological data within the computational domain. There is the need to use procedures able to 1) obtain a reliable computational domain, characterized by a total number of elements feasible for a common computing machine, starting from the huge amount of data provided by a LIDAR survey, 2) deal with river reach that receives significant lateral inflows, 3) insert bridges, buildings, weirs and all the structures that can interact with the flow dynamics. All these issues have large effects on the modelled water levels and flow velocities but there are very few papers in the literature on these topics in the framework of the 2-D modelling. So, in this work, attention is focused on the techniques to deal with the above-mentioned issues, showing their importance in flood mapping using two actual case studies in Southern Italy. In particular, the simulations showed in this paper highlight the presence of backwater effects, sudden and numerous changes in the flow regime, induced by the detailed river model, that underline the importance of using 2-D fully dynamic unsteady flow equations for flood mapping.

© 2015 Elsevier Ltd. All rights reserved.

1. Introduction

In the last few decades, the numerical modelling of flood events has been significantly enhanced due to the development of reliable numerical methods, computing power and innovative topographic survey techniques. This amount of progress has progressively encouraged the use of 2-D flood simulations, not only in the academic context but also in technical studies, replacing 1-D approaches that, despite their efficiency and the potential for their improvement in compound channels (see, for example, Helmiö, 2005; Proust et al., 2010; Costabile and Macchione, 2012) present conceptual problems when applied to overbank flows (Horritt and Bates, 2002; Tayefi et al., 2007; Costabile et al., 2015a).

Paradoxically, the significant improvements related to the flood simulation processes have raised worries in the literature because of the tendency of giving too much reliability to them. In particular, there is a well-founded concern that “sophisticated high-resolution models might be dangerous from this viewpoint as the false sense of confidence derived from their spuriously precise results might lead to making the wrong decisions” (Dottori et al., 2013).

Indeed, flood hazard assessments are affected by several sources of uncertainties which have significant consequences on the simulations accuracy. In particular, uncertainty concerns the hydrological data, the hydraulic parameters, calibration and validation data, the governing equations describing the physical processes, the way to take into account man-made structures interacting with the flow and so on (among the most recent ones see Merwade et al., 2008b; Di Baldassarre and Montanari, 2009; Bales and Wagner, 2009; Di Baldassarre et al., 2010; Hall et al., 2011; Stephens et al., 2012; Warmink et al., 2011; Brandimarte and Kebede Woldeyes, 2013; Grimaldi et al., 2013; Domenech et al., 2013; Dottori et al., 2013; Jung and Merwade, 2015).

In the uncertainty assessment, nowadays there is a tendency to overcome the deterministic approach by the development of probabilistic ones. The difference between these two approaches

* Corresponding author. Tel.: +39 0984 496557; fax: +39 0984 496558.
E-mail addresses: pierfranco.costabile@unical.it (P. Costabile), francesco.macchione@unical.it (F. Macchione).
http://dx.doi.org/10.1016/j.envsoft.2015.01.009
1364-8152/© 2015 Elsevier Ltd. All rights reserved.
can be summarized in the following way (Di Baldassarre et al., 2010). Advanced deterministic models consist of three steps: development of a 2-D fully-dynamic physically-based hydraulic model, model calibration using historical flood data, reorganization of the simulation results aimed at flood hazard mapping in a GIS environment. As regards the probabilistic approach, the authors identify three steps: construction of flood inundation models, sensitivity analysis of the model using historical flood data and use of the multiple behavioural models to perform ensemble simulation using an uncertain synthetic design event as hydrological input. According to Di Baldassarre et al. (2010), a fully dynamic 2-D model is not necessarily required in a probabilistic approach because the latter is not based on the assumption that the hydraulic model represents the physical behaviour of both the channel and flood-plain flow.

Indeed, reduced-complexity approaches are often sufficient to provide accurate results with respect to inundation extent, when compared to the more complex schemes (Horritt and Bates, 2001, 2002), even though there is some evidence that reduced complexity approaches tend to overestimate the inundation extent at coarser grids compared to the fully dynamic wave equations (Falter et al., 2013). This consideration is of course true in those situations in which the analysis is limited to the flood extent mapping and attention is focused on the probability of a given cell to be wet or dry. However, in those studies for which the hydraulic variables are used for hazard assessment throughout the flooded area, a more accurate approach should be required. In particular, information on the propagation of the flood wave, water depths and velocities and the rate at which the water level rises is very important for emergency planners in charge of evacuation and to estimate the potential of loss of life (Jonkman et al., 2008; Gómez-Valentín et al., 2009; Gómez et al., 2011; Xia et al., 2011; Russo et al., 2013). Moreover, further analyses are required to evaluate other important parameters necessary for assessing the flood hazard. For example in steep upstream areas and next to dyke breach locations, flow velocity is a very important factor for flood damage (de Moel et al., 2009; Qi and Altinakar, 2011). For this reason, accurate and local assessments of flood hazard in each point of the domain should require the use of 2-D fully-dynamic models (Ernst et al., 2010; Balica et al., 2013). It should be added that this not only applies to the deterministic approaches but also to the probabilistic ones, if the goal is to assess the probability of occurrence of hazard parameters related to the hydrodynamic variables.

Obviously, the computational times related to the use of 2-D fully dynamic modelling can be time demanding, even though a significant reduction is expected over the next years due to the increasing availability of parallel computing technique in flood analyses (see, for example, Neal et al., 2010; Yu, 2010; Kalyanapu et al., 2011; Vacondio et al., 2014).

The main purpose of this paper is to give a contribution on three important aspects related to the application of the 2-D modelling for flood hazard assessment, for which there are very few studies in the literature. In particular, the attention will be focused on:

1) correct representation of the flood-prone areas topography. The purpose is to get a reliable computational domain, characterized by a total number of elements feasible for a common computing machine, starting from the huge amount of data provided by a LIDAR survey;
2) interaction between hydrologic and hydraulic models;
3) insertion of bridges, buildings, weirs and all the structures that can interact with the flow dynamics.

The correct topographical representation is a key aspect as it brings the model closer to reality allowing water volumes and river conveyance to be correctly modelled (see, for example, Horritt and Bates, 2001; Sanders, 2007). Fewtrell et al. (2008) concluded that the model resolution has to be set up to the characteristic scale of buildings size and street width in order to obtain accurate predictions of flooding. Generally speaking, creating topographic representation of river systems is a challenging task because of issues associated with interpolating river bathymetry and then integrating this bathymetry with surrounding topography (Merwade et al., 2008a). For example, Cook and Merwade (2009) showed that the flood inundation area reduces not only with improved horizontal resolution and with vertical accuracy in the topographic data but also by incorporating river bathymetry in topography data. A more detailed geometry has a significant impact on the hydraulic modelling results, not only concerning the flood extent but also, above all, regarding the distribution of flow velocities and bottom shear stresses. The latter are equally important for estimating the risk potential in hazard zone mapping (Mandlburger et al., 2009). The great importance played by the micro-scale topographic and blockage effects, has led researchers and modellers to use high-resolution input data for flood simulation in urban areas and floodplains with human settlements, where the majority of at-risk assets are located. Airborne remote sensing such as LIDAR provides high quality digital terrain models, reducing the uncertainties in topography for numerical flood modelling. The direct use of the LIDAR survey as a computational grid is not possible due to the huge amount of data. Therefore, there is the need to use suitable procedures to obtain a reliable computational domain characterized by a total number of elements feasible for a common computing machine. Several papers claim that the computational grid is obtained by a LIDAR survey but, very often, the procedure used to do that is not clearly explained (see, for example, Bates et al., 2003). Mandlburger et al. (2009) presented a method for the generation of a hydraulic grid. In that work, the main purpose is the development of a DTM thinning approach, based on adaptive triangular irregular network (TIN) refinement, which allows an effective compression of the point data while preserving the most relevant features. Apart from the techniques used for getting the computational grid, particular attention has to be paid also to the so-called “quality” of the generated grid, checking some main parameters such as: the angle criterion, aspect ratio and expansion ratio (Fezinger and Peric, 2002).

The second issue analyzed in this paper concerns the interaction between hydrologic and hydraulic models. Almost all the studies in the literature show applications of numerical models for the propagation of just one upstream hydrograph. However, in practical studies, a river reach may have several lateral inflows and, therefore, the treatment of the tributaries requires particular attention. An accurate approach is the analysis of this problem at the basin scale, using a physically-based distributed rainfall-runoff model, but in practical cases the tributaries discharge hydrographs should be computed by a hydrological method. If the goal of the flood analysis is the study of flooded areas induced just by the main river, the separation between hydrologic and hydraulic models require to face us the problem of the choice of the boundary cross-section in which to insert the hydrological data. In particular, if the boundary cross-section is too close to the main river then the backwater effect induced by the latter can influence the boundary condition imposed on the tributary side. On the other hand, if the input section is too far from the main river, there is also the problem of the hydraulic simulation of the stretch of the tributary between the input section and the main river. This problem is not trivial because the selected section may have important consequences on flood extent predictions. In this paper, we will discuss this aspect and highlight the approach we have used in two real cases.
Another key aspect related to the interaction between hydrologic and hydraulic models is the numerical treatment of discharges flowing from the hill slopes and the minor tributaries whose contribution, globally, cannot be neglected but, likewise, they cannot be considered one by one in practical studies. In the literature, there is a recent paper dealing with this issue within 1-D modelling (Lerat et al., 2012) and it is, as far as we know, completely lacking in 2-D analysis. To face this problem, a specific procedure is discussed here.

The third aspect analyzed here is the numerical treatment of the man-made structures with particular reference to the bridges. These are often taken into account by an increase of the roughness coefficient or using formulas aimed at a global estimation of the backwater effect (Pappenberger et al., 2006; Brandimarte and Kebede Woldeyes, 2013). Moreover, in several cases, the bridge effects are analyzed using a 1-D approach. Indeed, these methods should be used carefully because the interaction between bridge and flood flow might not be properly simulated. In particular, local effects and backwater phenomena, eventually varying across the section, may be underestimated or neglected also in those situations in which the use of 1-D modelling seems to be an obvious choice (for example, in almost rectilinear and confined channels). In the two cases discussed here, we show how the accurate geometrical representation of the bridge can highlight these issues.

Therefore, in this work, attention is focused on the techniques to deal with the above-mentioned issues showing their importance for flood-prone areas mapping. For this reason, two actual case studies in Southern Italy have been considered for practical applications. The paper is organized as follows: after a brief presentation of the numerical model used in this study (Section 2) together with its validation (Section 3), two case studies are presented (Section 4). Section 5 is dedicated to the set up of the river model, discussing the procedures for building a reliable computational domain, for the insertion of buildings, bridges and hydrological data. Section 6 is devoted to the discussion of the influence of a detailed river model on the results. Finally, in the last section the conclusions of the work will be presented.

2. Mathematical and numerical model

As already stated in the introduction, there are a lot of papers dealing with the flood propagation models to be used in practical studies for flood hazard mapping. In our study, we have used a fully dynamic wave modelling because it is the most physically based approach for flood propagation and, therefore, it is able to manage situations in which numerous transitions to supercritical flows and numerical shocks occur. The occurrence of these complicated hydraulic phenomena will be discussed in Section 6.

The mathematical model is based on the 2D shallow water equations (SWE) that can be expressed in the following form:

\[
\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} = S
\]

where:

\[
U = \begin{pmatrix} h \\ h u \\ h v \\ q \end{pmatrix}, \quad F = \begin{pmatrix} hu \\ hu^2 + gh^2/2 \\ hv \\ hu \end{pmatrix}, \quad G = \begin{pmatrix} hv \\ hv \\ hu \end{pmatrix}, \quad S = \begin{pmatrix} gh(S_{0x} - S_{fy}) \\ gh(S_{0y} - S_{fy}) \end{pmatrix}
\]

in which: \( t \) is time; \( x, y \) are the horizontal coordinates; \( h \) is the water depth; \( u, v \) are the depth-averaged flow velocities in \( x \)- and \( y \)-directions, respectively; \( g \) is the gravitational acceleration; \( S_{0x}, S_{0y} \) are the bed slopes in \( x \)- and \( y \)-directions; \( S_{fy}, S_{fy} \) are the friction slopes in \( x \)- and \( y \)-directions; \( q \) is a lateral inflow.

For the numerical integration of system (1), in this paper the finite volume methodology (FVM) has been used, according to which Equation (1) for the \( i \)th control volume is rewritten in the following form:

\[
\int_{\Omega_i} \frac{\partial U}{\partial t} d\Omega + \int_{\partial \Omega_i} \mathbf{F} \cdot d\Gamma = \int_{\partial \Omega_i} \mathbf{S} \cdot d\Gamma
\]

\[
\mathbf{E} = [\mathbf{F}, \mathbf{G}] \text{ is the tensor of fluxes in the } x \text{ and } y \text{ directions and } \Omega_i \text{ is the area of the } i \text{th control volume. Applying the divergence theorem to the second integral, the following equation holds:}
\]

\[
\int_{\Omega_i} \frac{\partial U}{\partial t} d\Omega + \oint_{\partial \Omega_i} \mathbf{E} \cdot \mathbf{n} d\Gamma = \int_{\partial \Omega_i} \mathbf{S} \cdot d\Gamma
\]

where \( d\Gamma \) is the boundary of the \( i \)th control volume and \( \mathbf{n} \) is the unit outward normal vector to the boundary.

Denoting by \( U_i \) the average value of the flow variables over the control volume \( \Omega_i \) at a given time, Equation (7) may be discretized as:

\[
U^{i+1} = U^i - \frac{dt}{\Omega_i} \sum_{j=1}^{k} (E^i \cdot n_{ij} d\Gamma_{ij}) + dS_i
\]

in which \( i \) and \( j \) refer, respectively, to the \( i \)th cell and the \( j \)th edge of the cell; \( k \) is the total number of the cell edges; \( n_{ij} \) and \( d\Gamma_{ij} \) are the unit outward normal vector and the length of the \( j \)th edge respectively; \( E \) is the numerical flux through the edge which may be computed by an appropriate Riemann solver.

Apart from the computation of numerical fluxes, the numerical integration of the shallow water equations in complex topographies requires further specific algorithms to obtain stable and reliable results. Attention, in particular, has to be paid to the numerical treatment of the bottom slope, of the friction slope and of the wet–dry fronts. A detailed presentation of the numerical techniques implemented in the model used in this paper goes beyond the purpose of the paper. Therefore only the main features of the numerical model are mentioned herein. On the basis of the authors’ experience of the performances of several first and second order upwind and central numerical schemes (see for instance Macchione and Morelli, 2003; Costabile et al., 2012), the numerical scheme is based on the following key aspects:

a) an unstructured grid, based on irregular triangular elements (TIN), has been used to obtain the computational domain. This kind of grid allows one to modify the density of the grid points accordingly to the topographic features and the expected hydraulic situations. Its high degree of flexibility and adaptability provides an accurate geometrical description of the site even in the presence of a significant topographical gradient or when the hydraulic variables are expected to change very rapidly (hydraulic jump, shock wave and so on);

b) the first order Roe scheme (Roe, 1981) has been used for the computation of the numerical fluxes. This scheme is a quite consolidated solver in the literature (see Glaister, 1992; Bermudez and Vazquez, 1994; just to cite the earlier works);

c) the bottom slope has been computed starting from the equation of the plane \( Z \) containing the three vertices of the
triangles that cover the computation domain. The derivatives of \( Z \) over \( x \) and \( y \) give the bottom slopes along the two spatial directions;
d) a semi-implicit treatment of the friction slope has been used in order to improve the stability of the numerical code;
e) a specific algorithm, inspired by what has been proposed by Sleigh et al. (1998), has been used to manage the wet–dry fronts. Moreover, the reconstruction of the free surface in partially wetted cells proposed by Begnudelli and Sanders (2007) has been implemented.

3. Preliminary assessment of model performances

Since the numerical model used in this paper has been developed by the authors, an in-depth validation is required prior to justify its application in actual case studies. A good practice for the choice of a mathematical-numerical model is the analysis of its performance using specific test cases that should be representative of the different flow behaviors that may occur, even locally, during a flood event. For this reason, four tests are discussed in what follows.

3.1. Simulation of a dam break in a straight channel with a triangular bump in the bed

This case refers to the simulation of an experimental test of a dam break wave propagation into a channel with a triangular bump in the bed (Fig. 1a). Due to the triangle-shape sill, the phenomenon reproduced in this test is useful to check the model performance in rivers with weirs as those considered in our study. The facility was formed by a 38 m long and 0.75 m wide channel. The gate location was 15.5 m upstream of the channel and the initial water depth above it was 0.75 m. A complete and detailed description can be found in Soares-Frazão (2007). The situation analyzed here refers to the most complex condition in which the channel is dry upstream of the bump but not downstream since a wall is located at the end of the channel. This is a very demanding test for a numerical method because of the flow dynamics involving multiple wave interactions and backwater effects. As one may observe in Fig. 1b–d and 1e, the numerical results are in good agreement with the experimental data in all the gauge sites considered.

3.2. Simulation of a flood event on a physical model of the Toce river

This test, carried out within the CADAM project (Soares-Frazão and Testa, 1999), consists of an extreme flood event on a physical model, reproducing a 5 km reach of the Toce River valley located in the Northern Italian Alps. The physical model was very detailed since not only the main channel and the large floodplains but also a number of singularities were reproduced (see Fig. 2a): a lateral reservoir, a small barrage, bridges and buildings. The results shown in this paper refer to the most catastrophic situation in which the magnitude of the upstream hydrograph (identified by the authors using the code HY2) was such that it induced not only the overtopping but also the filling up of the reservoir by the incoming water. Therefore this test is useful to check the model performance in complex geometries in which lateral inundations or dike overtopping, discussed in the simulations on the Crati river, occur. All the hydrographs recorded by gauging are used for validation of the model. The results highlight a generally good agreement between simulated and observed hydrographs. For the sake of brevity, only

Fig. 1. Layout of dam break flow in a straight channel with a triangular bump (a) and comparison between simulated hydrographs (solid lines) and experimental data (points) at different gauge sites: G4 (b), G10 (c), G13 (d), G15 (e).
six water level hydrographs are shown in what follows (see Fig. 2b–g). These results show the same accuracy as that obtained from all other authors who have simulated the same test (see, for example, Caleffi et al., 2003).

3.3. Simulation of model city flooding experiments

This test is devoted to the simulation of flow behaviour in urban like environments with reference to the experimental data of the “Model city flooding experiment”, carried out during the IMPACT project (Testa et al., 2007). The test simulated in this paper refers to the model city arranged in a staggered manner in which concrete blocks, representing the buildings in an urban area, were placed in a checker board configuration (depicted in blue (in web version) in Fig. 3a) and two masonry walls were placed parallel to the TOCE model main axis. In particular, the numerical results shown here refer to the test identified by the code “1b” in Testa et al. (2007). In the numerical computation, the presence of the buildings was modelled as an interior boundary condition (solid wall). Therefore, this test is useful to check the model performance in urban areas and, more in general, allows the analysis of the model suitability in representing the influence on flow propagation induced by non-submergible obstacles (buildings, bridge piers and so on). The computational grid is shown in the Fig. 3b, in which the grid refinement near and among the buildings may be appreciated. Fig. 3 highlights a generally good agreement between simulated and observed hydrographs in terms of peak values, time to peak and recession limb. A satisfactory result was also obtained with regards to the arrival time and the rising limb of the hydrograph.

3.4. Malpasset dam-break simulation

The Malpasset dam, located in a narrow gorge of the Reyran river (France), failed after an intense rainfall in 1959 and the catastrophic flood resulted in the deaths of 421 people and considerable property damage in the inundated region downstream. A physical model of 1/400 scale was built in 1964 by Laboratoire National d’Hydraulique of EDF in order to obtain experimental data for analyzing the event. In particular, the front arrival time and the maximum water depth were measured at 9 observation points in the physical model (named S6–S14, see Fig. 4). Therefore this test is very useful to check the model ability in reproducing the front arrival time that, as is well-known, is strictly related with the flow velocity. The model was calibrated against field data (water marks and propagation times of the flood wave) and the Manning’s friction coefficient $n$ was estimated to be in the range 0.025–0.033 s/m$^{1/3}$ ($K = 33–40$ m$^{1/3}$/s), as suggested by EDF (Hervouet and Petitjean, 1999). The computational grid was composed of 98,282 cells and 49,424 nodes. Triangles side is variable ranging from 10 m to 200 m. Two simulations have been carried out using different configurations for the Manning coefficient. In the first one (Sim-1), a constant value of 0.0303 s/m$^{1/3}$ (Strickler’s coefficient $K = 33$ m$^{1/3}$/s) has been assumed throughout the domain while, in the second one (Sim-2), 0.0286 s/m$^{1/3}$ ($K = 35$ m$^{1/3}$/s) and 0.04 s/m$^{1/3}$ ($K = 25$ m$^{1/3}$/s) have been assumed, respectively downstream from the dam and in the reservoir.

The snapshots of flow depths simulated in the Sim-2 at different instants of time are presented in Fig. 4. Fig. 5 shows the comparison between model results and physical model observations for
maximum depth and flood propagation time at nine observation points. The simulated front arrival times are in close agreement with the experimental data. At the last point (S14), the model slightly overpredicted the flood front propagation time. On the other hand, the results are similar to those reported in Singh et al. (2011) in which a second order well-balanced explicit central-upwind scheme has been used. Moreover, in that study, two more detailed square meshes, having 121,000 cells (side 30 m) and 484,000 cells (side 15 m), respectively, have been used. As stated in that paper, this discrepancy can probably be attributed, at least

Fig. 3. Plan view of the physical model (a) and comparison between simulated and observed water depth hydrographs at different gauge sites: P4 (b), P5 (c), P7 (d), P8 (e), P9 (f), P10 (g).

Fig. 4. Malpasset dam break: simulated water depths in different instants of time.
partially, to the uncertainty related to Manning’s $n$ and it can be reduced using a finer mesh results. Our model over predicted the water level at station S9 but, also in this case, this effect is probably due to the coarse representation of the domain (Singh et al., 2011). Some slight differences exist between the simulated front arrival times according to Sim-1 and Sim-2 while the simulated maximum water depths are practically the same.

The maximum water level predicted by the proposed model matches with the physical model and with the results of second order models (Valiani et al., 2002; Singh et al., 2011) up to a reasonable accuracy. Therefore the model is capable of capturing the hydrodynamic behaviour of flood flows to a high accuracy level in terms of both maximum water levels and front velocity.

4. Study areas and data sets

The Region of Calabria is a narrow peninsula stretching between the Tyrrhenian and the Ionian seas. Due to the morphological configuration of the land, the basins have a very limited extent and high slopes so that the river network response to rainfall events, which may be very intense due to orographic effects, is very rapid.

Two case studies, representative of small-medium size basin in Mediterranean countries, are discussed in what follows. The first application concerns the most important river in Calabria: the Crati river (see Fig. 6a). This case study is complex for a number of reasons. The Crati river runs through the old town of Cosenza (Fig. 6a, c). The basin area, upstream of the confluence with the Busento.
river, is approximately 130 km\(^2\). This river reach is characterized by a number of drop structures and groynes. Here, masonry dykes, whose faces are almost vertical, provide the defense of the urban area both on the left and right sides of the Crati river (Fig. 6c). At the end of the old town, the Busento river (catchment area equal to 68 km\(^2\)), which in turns receives the water of the lassa stream (basin area equal to 72 km\(^2\)), flows into the Crati river, from the left, and three bridges (see Fig. 6d–f), whose piers are placed in the river bed, are located near or just downstream from the confluence. The total length of the considered river reach is about 10 km. The width of the main channel is variable ranging from 20 m, in the old town of Cosenza, to 90 m, downstream from the confluence with the Busento river, up to 150 m at the end of the domain.

The second application refers to the Corace river (see Fig. 7a). The river reach considered in the simulations is 13 km long and ends at the mouth, where the river flows into the Ionian sea. The basin area size, upstream of the beginning of the river reach is approximately equal to 180 km\(^2\). The river reach is characterized by the presence of several bridges, with several piers placed in the riverbed (Fig. 7b–e). The downstream part of the river interacts with roads and railways, as usually happens in most of the coastal areas located in Southern Italy, and a number of small houses are present behind the railway. Furthermore, the mouth of the Corace river is very close to the Catanzaro Lido and Roccelletta coastal towns. This is a typical situation of the small-medium size basin in South of Italy.

The topographical survey of the areas of interest was carried out by integrating aerial and terrestrial techniques. An innovative airborne LIDAR sensors (RIEGL LMS-Q560) was used to cover the case studies, providing a sampling density of four points over 1 m\(^2\). Different kinds of DEM (digital surface model, digital terrain model with and without buildings and other objects surmounting the ground etc.) were produced at the end of this step, and rearranged according to two regular grids whose size was 0.5 m and 2 m. Moreover, acquisition of digital images was provided in order to generate a number of orthophotos of the areas and, consequently, to recognize objects and to set distributed values of the Manning coefficient. Airborne LIDAR survey was integrated and completed by a terrestrial survey for the description of both the topography under bridges and all the man-made features that may influence, even locally, the flow dynamics. As is well-known, airborne LIDAR data cannot recognize elements under the bridge and if data are provided in these areas, they consist in an interpolation of the ground level around the bridge itself. Airborne and terrestrial LIDAR were provided using coordinates in the same reference systems. The procedure followed consists in deleting the fictitious airborne LIDAR data in these areas, merging the terrestrial LIDAR data for the bathymetry and structural details such as piers. The merging process was carried out using the SMS© (Surface water Modelling System) software (Aquaveo, 2010). Some details on this software are discussed in the next section.

For instance in Fig. 7f, a LIDAR overview of a sequence of four bridges located in the Corace river is shown.

5. Procedures for obtaining the river model

In this section, attention is mainly focused on two aspects: 1) procedures to obtain a suitable computational grid and 2) techniques to introduce all the man-made structures and hydrological data within the computational domain. The discussion of these aspects, in flood analysis of actual rivers, is often neglected in the literature probably because they are considered much less
important than the mathematical model. In our experience, the reliability of the hydraulic simulation aimed at quantifying the flood hazard, strongly depends on both of these issues and for this reason we provide a detailed description of the relative procedures in what follows.

5.1. Representation of the case studies: setting up the computational domain

The direct use of the LIDAR survey as computational grid is not possible due to the huge amount of data. So there is the need to use suitable procedures to obtain a reliable computational domain characterized by a total number of elements feasible for a common computing machine. In our experience, the use of an automatic procedure for unstructured grid generation processes can lead to a bad representation of the actual topography. The procedure proposed herein is divided into three steps.

5.1.1. Grid generation process

The nodal generation process was carried out using SMS© (Surface-water Modeling System) that is one of the most important software for simulating surface water models. This program also has a mesh generation system, in a GIS environment. We have used SMS just for building the computational domain and, therefore, no simulation of flood propagation were carried out using this software. SMS software generates nodes inside polygons and connects them into elements. In particular, we have chosen the so-called “paving method” that uses an advancing front technique to fill the polygon with elements. Further details can be found in the technical manual. So a triangular irregular network (TIN) can be obtained by 1) building polygons, 2) specifying the nodal density along the polygons and finally 3) interpolating the bathymetry (that in our case was the surveyed points re-arranged according to a 0.5 m regular grid). The main idea used for building polygons was to overlap the lines composing the polygons and the most “significant” contour lines of the area. Some polygon lines were organized in such a way as to follow the plan development and the cross section of the river. In this way, the computational element will follow the main river axis, reducing errors induced by an arbitrary mesh on the direction of the flow propagation. The next step is to reproduce the most important topographical features using a suitable detail. From a practical point of view, focalizing attention on small river reaches (200–300 m along the river axis), it may be convenient to highlight the contour levels every 1–2 m from the mean thalweg level to 10 m above. In the steep areas, the density of the grid points will be higher than that of the flat areas in order to reproduce the most significant topographical gradients. The distance between two consecutive nodes, belonging to the lines described above, may be set in such a way to not overcome, more or less, 1.5–2 times the distance (measured in plan and perpendicularly to the lines themselves) between two contour levels.

5.1.2. Checking the grid quality

Apart from the criteria used for the generation of the computational domain, it is important to check the grid quality prior to running the hydraulic simulation. In brief, common drawbacks encountered when an automatic procedure is used for the generation of an unstructured grid are represented by the generation of a number of triangular elements with a very small interior angle and the presence of adjacent triangular elements each with very different areas. These situations are not suitable for integrating the differential equations of the mathematical model. Sometimes it is possible to use semi-automatic tools to solve them. However, the use of such a tool is not recommended especially in the presence of banks or road embankments because of the possible smoothing of the topography. The grid correction process has to be carried out by hand, providing a suitable modification of the spatial position of the nodes, in order to solve the problem, or by the addition of other grid nodes. This operation is laborious but is fundamental for preserving the actual topography and, consequently, for achieving realistic simulations.

5.1.3. Checking the topographical suitability of the grid

The final step of the grid analysis consists in checking the “topographical correctness” of the grid itself that means an accurate comparison between cross-sections obtained by using the original surveyed points and those extracted by the developed computational grid. In our mind, the “accuracy” of a grid is simply related to its suitability in reproducing the most important topographical features of the site and, in particular, the most relevant elements within a cross-section that can play a significant role in the inundation dynamics: thalweg position and level, banks positions and elevations, cross-section area, extent of floodplain and so on. The grid can be considered satisfying if it reasonably reproduces these features. The idea is to check these features in several cross-sections (comparing sections) so that we can consider the grid very accurate in those sections, at least. Our approach is different from what has been proposed by Mandlburger et al. (2009) in which the accuracy of the grid is related to a maximum height tolerance.

In order to assess the “topographical correctness” of the grid, the maximum spacing between two consecutive comparisons sections is 50 m while the minimum one is up to 2–5 m depending on the structures or singularities that are located in the river reach. In some cases, further modifications in nodal positions were necessary to better represent the river model. For instance, in Fig. 8, a comparison between the surveyed (LIDAR) and re-constructed (TIN) cross-section is shown. Despite the reasonably good reproduction of the cross-section, it was important to provide further improvements to better describe the thalweg position and bank levels.

5.2. Representation of the case studies: insertion of man-made structures, hydrological data and variable roughness

5.2.1. Insertion of bridges, buildings and weirs

Very recently, the inclusion of bridges in the computational domain is becoming a more and more discussed topic in the literature and it is also treated in technical governmental documents (see for example Zevenbergen et al., 2012). For example, Ratia et al. (2014) proposed two representations of bridges to couple with the 2D shallow water equations. One of this considers also the head losses caused by bridges. In particular, an empirical formulation for total head loss caused by bridges which covers all flow regimes (free water surface, partially submerged flow, and fully submerged flow) is considered as an extra source term in the momentum equation. This model was verified on laboratory tests as well as on a case study of real flood. Costabile et al. (2014, 2015b) highlighted the occurrence of significant 2D local effects even in situations for which 1-D approaches should seemingly be recommended for practical studies. In particular, simulations with and without bridges highlighted that talking about a unique effect played by the bridge on flow dynamics can be misleading, even for a bridge perpendicular to the main flow direction, since the effects of the piers need to be analyzed individually.

In this work, we show that an accurate refinement of the computational domain of the 2-D model is a suitable technique to take into account local effects, varying also across the section, of the flood flow. A first situation is represented by almost prismatic piers with constant horizontal section along the vertical axis. In this case, the inclusion of the piers in the computational domain was easily
done in such a way that the triangular elements, that fall in the area occupied in plant by the piers themselves, can be automatically recognized as “no submergible” elements. That was made possible by setting a specific feature to the above mentioned elements in such a way to distinguish an “interior” element, in which the 2-D SWE have to be solved, from a “no submergible” element that works as an internal boundary condition (solid wall). For instance, looking at Fig. 9a and b, the no submergible elements are highlighted (in yellow) while the interior elements have no colour. The size variation of cells around the piers depends on the shape and dimension of the piers themselves. As a rough indication, the grid size around piers is 5–15 times less than that used upstream.

The inclusion of the bridge shown in Fig. 9c was more complex because the piers are founded on bases emerging from the river bed that induce a significant narrowing of the cross-section. In order to provide a suitable description of the flow behaviour near the bridge, the bases were reproduced by a sudden topographic variation over which a number of no submergible cells, reproducing the pier obstruction, were placed (see Fig. 9d). In this way, the bases work as an obstacle for the shallower water levels and, eventually, they may be submerged for the higher ones. It is important to observe that the topography under the top of the bridges was reproduced ignoring the airborne LIDAR data and using a terrestrial LIDAR survey. In this context, particular attention was paid to the correct merging of the two sources of data.

Buildings can be described in different ways but the best modelling strategy is still object of debate (see for example Schubert and Sanders, 2012). In this work, it was taken into account by using two different techniques according to the specific features of the area and to their importance. Isolated buildings that fall in rural areas, especially if not very close to the river, were considered by the equivalent roughness techniques (see for example Costanzo and Macchione, 2006). In contrast, the urban area near the river was modelled by means of interior boundary conditions (solid wall).
in the triangular elements falling in the area occupied by the buildings in order to simulate the flow propagation through the very narrow alleys among them. The minimum side of the triangular element used in these situations is 1.5–2 m.

Finally, weirs can be taken into account by a local grid refinement in order to provide a suitable description of the topographic discontinuity associated to the weirs themselves. Also in this case, the size variation of cells around a weir depends on the situation considered. As a rough indication, the grid size around weirs is 6–10 times less than that used upstream. Looking at the confluence between the Busento and Crati, all the devices to take into account the topographical features mentioned above are highlighted in Fig. 10. In particular, the red lines show the locations of the several weirs in the river reach and some grid details, near weirs, bridges and buildings are highlighted as well.

5.2.2. Insertion of hydrological data

The uncoupling of different models aimed at the generation and propagation of floods give rise the problems of the insertion of hydrological data in hydraulic models. Theoretically, the availability of just one model for simulating both phenomena should be the most suitable approach but, practically, this goal is far from being reached up till now. Inserting of hydrologic computations in flood propagation models is a relative easy task when the river reach is characterized by just one hydrograph in the upstream boundary section. However, when the river receives other hydrographs from other tributaries, the problem arises of how to treat these situations hydraulically. The problem can be generated not only by the most significant tributaries of the main river but also by several minor tributaries whose contributions, globally, cannot be neglected but, likewise, cannot be considered one by one in practical studies.

Both these situations involve different problems related to the hydraulic treatment of the two sources of discharge.

5.2.2.1. Insertion of main tributaries. Regarding the most important tributaries, the 2-D approach offers an opportunity to insert them as physical upstream boundary conditions in the hydraulic model. It should be noted that the operation is different from what is commonly performed using a 1-D model in which just one discharge can be used as upstream boundary conditions and the remaining tributaries have to be treated as source terms in the mass continuity equation. However, our approach requires us to cope with the problem related to the choice of cross-section of the tributary for imposing the boundary condition. In other words, there is a problem of adequately defining the 2D model domain a priori because a given area could be flooded: a) only by the main river, or b) only by the tributary or c) by the combining effect of both together. In our view, the 2-D computational domain has to cover, at least, the neighboring areas of the tributary potentially involved by the combining effect of the main river and the tributary. From a hydraulic point of view, this means: 1. to extend the computational domain up to a distance for which the backwater effect induced by the main river cannot play any influence in the tributary flow; 2. to choose an upstream boundary cross-section, on the tributary side, for which the tributary discharge can flow without producing overflow; in particular, the number of cells in

Fig. 10. Details of the computational grid and insertion of weirs, buildings and bridges in the computational domain close to the confluence between Crati and Busento Rivers. Red lines represent the weirs location. (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)
which a redistribution of the total discharge has to be provided can be determined by looking at the computational cells that are in the deepest part of the cross-section.

5.2.2.2. Insertion of minor tributaries and other sources of discharges. It is also possible to take into account the runoff volumes that flow over slopes and act as a nearly uniformly distributed inflow in the main river. In this case, one possible strategy is to consider a single contribution from small tributaries grouped together. These situations can be considered as a single basin that works as a lateral inlet in the main river. The insertion of these hydrological data is completely different from the previous ones treated as upstream boundary conditions. We included them as a source term in the mass continuity equation.

First of all, in particular, the river reach of the main river delimited by the initial and final cross-sections in which a sub-basin conveys its discharge has to be highlighted. Along the thalweg line belonging to that river reach, a number of computational cells (cells that contain the thalweg location or very close to it) were selected and a particular feature was assigned to each one in order to recognize them automatically. It is important to distinguish those cells because, for them, the source term of the mass continuity equation is not zero as it is for the remaining ones. In particular, for each cell, the source term is equal to the specific discharge (i.e. discharge for an area equal to one) computed as the ratio between the total discharge and the total area of the previously selected cells. In Fig. 11, an example of the procedure presented above is shown with reference to the Corace case study.

5.2.3. Insertion of variable roughness

The numerical model can take into account roughness variations both in the longitudinal and transversal directions. For the cases analyzed here, data of real events are not available and, consequently, the model cannot be calibrated. The approach used in this paper to face this aspect has been already explained in a previous work within a comparative study between 1-D and 2-D models (Costabile et al., 2015) and, therefore, only some key elements are recalled here. In order to define homogeneous roughness areas, the water course is divided into homogeneous river reaches in terms of bottom slope, bottom irregularities, sinuosity, obstacles. Then, the procedure proposed by Arcement and Schneider (1989) is used to estimate the roughness coefficients as a function of land cover, by means of 1-D steady state calculation imposing the mean value of the discharge hydrographs. The vegetation density and the vegetation height, located in the aerial photographs, were checked using terrestrial surveys. Using the well-known Cowan formula (Chow, 1959), the cross-section flow resistance in the 1-D model has been corrected in order to obtain the reach resistance that takes into account different kinds of energy losses due to hydraulic singularities occurring in the flow field, the channel obstructions and the degree of meandering. Apart from the modification due to the density of vegetation, no other corrections made in the 1-D calculation have been transferred in the 2-D model because the river sinuosity, the presence of narrowing sections and the bottom irregularities are inherently represented in the grid. Furthermore, no additional coefficients need to be introduced in the cells around the man-made structures due to the above-described method for bridge piers and buildings. The values of the Manning coefficient, in a medium part of the domain, are shown in Fig. 12.

6. Results and discussion on the influence of a detailed river model on flood mapping

For the Crati River case study, the hydrological data were put into the domain in such a way as to provide three localized inlets upstream from the confluence (Crati, Iassa and Busento rivers whose peak values are 491, 255 and 275 m³/s) and another two important left tributaries located in the downstream part of the river reach not shown here. Two “distributed” tributaries were also considered, one of these starts to contribute just downstream from the confluence between the Crati and Busento. 104165 triangular elements and 53,634 nodes formed the computational domain. The mean area of the elements is about 37 m² ranging from a minimum value of 0.1 m² to a maximum value of 1500 m².

In Fig. 13a, the maximum water depths associated to the most critical scenario related to T = 500 years are shown. According to the model prediction, some portions of the urban area adjacent to the Crati river may be flooded, especially near to the bend highlighted in the same figure. It is interesting to observe the high detail of the simulation that even provides the advancing of the front wave through the narrow alleys between the buildings. The contour map highlights a very complicated hydraulic phenomenon characterized by significant water depth variations, along the river axis of both the Crati and the Busento rivers, induced by the presence of several weirs, whose locations are highlighted by the white lines in Fig. 13a. The bridge piers, located close to the arrows in Fig. 13, seem to heavily influence the water levels generating significant backwater effects. The presence of both the confluence with the Busento river and the weirs make the hydraulic phenomenon very hard to simulate without a shock capturing scheme used to solve the fully

![Fig. 11. Example of insertion in the river model of localized and distributed hydrological data.](image)
dynamic wave equations. For a given instant of time, the surface water levels and Froude number profile in the reach of the Crati river, highlighted with a white line in Fig. 13a (delimited by letters A and B), are shown in Fig. 13b. In particular, the numerous transitions from supercritical flows to subcritical flow and numerical shocks may be noted. A map of the Froude numbers computed in different instants of time is shown in Fig. 14. The simulation highlights the presence of wide areas in which supercritical flow exists (blue areas) (in web version), especially downstream from the confluence (see areas near letter A). In both the Busento and Crati rivers, flow is supercritical when approaching the confluence (see areas near letter B₁ and B₂, respectively). The model simulates the transition...
from supercritical to subcritical flow crossing the weirs without numerical problems.

Finally, looking at the flow inundation near the bend, upstream from the confluence, it may be noted that supercritical flow condition characterizes the flow dynamics during the inundation process (see areas near letter C). According to the simulations presented here, supercritical flow occurs in several parts of the domain. In these processes, a significant role is played by the bridges. For instance, we have carried out another simulation in which the last bridge downstream from the confluence (located close to the white arrow in Fig. 13a) is not modeled (that is no boundary condition has been implemented in the cells covering the piers set in the river bed). The comparison between the simulated energy $E$ and the water surface $Y$ with and without piers is shown in Fig. 15, in which the backwater effect induced by the bridge is evident (Fig. 15a). The river reach analyzed is limited by the letters C and D in Fig. 13b. The flow approaching the bridge is supercritical (Fig. 15b) but the piers provoke a variation in the flow regime just upstream (near the section 2150 m), which is not present when neglecting the piers themselves. Then supercritical flow is restored. The simulation carried out without the bridge highlights supercritical flow along the river axis throughout the analyzed reach. The maximum water surface levels simulated with or without piers, the bridge soffit level and piers in the cross-section just upstream the bridge are represented in Fig. 15c. The maximum surface levels show significant transversal variations with local water rise close to the piers. Due to this phenomenon, the maximum water level is, locally, very close to the bridge soffit level. Therefore the bridge schematization used in this paper enables us not only to simulate backwater effect but also further local water rises, induced by the interaction with the piers, which can be important in flood hazard assessment.

It is interesting to discuss the influence of the individual steps taken to set-up the computational domain, described in Section 5. In particular, our intention here was to analyze the effect of the grid variations described in Section 5.1.3 (checking the topographical suitability of the grid) that represent the most important step in the grid modifications process to be carried out after the initial grid generation. For this reason two simulations are compared in what follows. They differ for the computational domain used since the first one refers to the grid after all the modifications described in Section 5 (namely, checked grid) while the second one considers the grid after the first step only (namely, unchecked grid). With reference to the areas upstream of the confluence between Crati and Busento river, highlighted in Figs. 10 and 16 represents the out-of-bank flooded areas with water depth lower than a given threshold value. This kind of graphs are commonly used in flood damages estimations (i.e. de Moel and Aerts, 2011; Kourgialas and Karatzas, 2012; Seifert et al., 2013). For example, according to the “checked grid” simulation, the extent of out-of-bank areas flooded by water depths lower than 3 m is 20 hm$^2$. In a similar way, we can say that to have 20 hm$^2$ flooded area the water depth should be up to 3 m (point A) while according to the unchecked grid it would be sufficient about 2.3 m (point B). This means that using the “unchecked grid simulation” would lead to an underestimation of the water depths for a given out-of-bank water level or an overestimation of the flooded areas for a given water depth.

Fig. 14. Froude number maps at different instants of time: 13 (a), 15 (b), 17 (c), 19 (d) hours after the beginning of the event.
For the Corace river case study, the hydrological data were put into the domain in such a way as to provide, respectively, four “main” and four “minor” tributaries. The peak discharge value, in the upstream section is equal to 875 m³/s. In this case, the downstream boundary condition consists of a constant water level, due to the presence of the sea. 113,657 triangular elements and 58,375 nodes formed the computational domain, globally. The mean area of the elements is about 70 m² ranging from a minimum value of 0.17 m², for elements near the piers of the bridge and the weirs, to a maximum value of 1300 m² in area located very far from the river or never reached by the flow.

In Fig. 17a, the maximum water levels near the mouth, related to $T = 500$ years, are shown. Looking at the water contour levels one may see that the model simulates very complicated hydraulic phenomena near the position where the three bridges are located (see also Figs. 7d and 9b) due to the interaction between the flow and the bridge piers. The railway embankment is not overtopped by the flow but an important area behind the railway itself, where a number of houses are present, is flooded. This fact can be explained by the situation depicted in Fig. 16b in which the product between the water depths and velocity (unit discharge), at a given instant of time (close to time to peak), are shown. In particular, Fig. 17b describes the hydraulic phenomenon near the confluence between the Corace River and the last main tributary considered in this study (Fiumarella stream) that is located close to the interruption of the railway embankment. Due to this sudden interruption, the vector lines show that some water volumes flowing in the Corace River...
river go away from the main river axis and interact with discharge flowing through the Fiumarella stream, which in turn, is forced to flow behind the railway embankment and cannot enter into the Corace watercourse. These water volumes, that start to run in a parallel way behind the railway, then provoke the inundation of the area described above. The representation of all these phenomena was made possible by the choice of locating the inlet of the Fiumarella stream as localized in flow far from the confluence with the Corace river according the suggestions provided in Section 5.2.2. It is evident that the use of a different inlet configuration or locating the inlet of the Fiumarella river near the confluence would result in a different simulation in terms of floodable areas neglecting all the inundation behind the railway. The location of a tributary inlet is thus very important for the correct flood-prone areas delimitation.

The values of the maximum discharge increase from upstream to downstream due to the effects of the lateral discharges. The latter, in the case analyzed here, are not particularly important if compared with the discharge flowing in the river reach and for this reason they have limited influence on the results. However, it has been important to focus on this problem from both a methodological and numerical point of view. In particular, from a numerical point of view, it is important to observe that neither the sudden increases of the discharges due to the "localized" inflow nor the "distributed" discharges caused numerical oscillations.

Another interesting aspect of this case study is the effect induced by the detailed reconstruction of the piers founded on the bases emerging from the river bed of the bridge shown in Fig. 9c and d. Fig. 18a shows a comparison between the maximum water levels simulated using, respectively, a computational grid in which the bases are reproduced (see Fig. 9d) or not. In the latter case, the presence of the bridges is taken into account only by the non submergible cells representing the plan area occupied by the piers. Fig. 18a highlights that the backwater effect is much more important, as expected, in the first case with differences of up to 60–70 cm between the two simulations. The river reach influenced by a detailed reconstruction of the bridge is about 500 m upstream from the bridge location while no particular effects may be observed downstream. The increase of flooded area is up to 20% (see Fig. 18b).

7. Brief discussion on model performances

It might be interesting to highlight that the five-step procedure proposed by Bennett et al. (2013) for evaluating the performance of our model has been faced in this paper. In particular, the first step ("Assessment of the model’s aim, scale and scope") has been taken into account in the Introduction section for stating the model purpose. Moreover, the lack of observed data for the actual rivers analyzed here did not allow us the calibration and the quantitative evaluation of the performances suggested in Step number 2 ("Check the data"). For this reason, the visual analysis has been the preferred approach for the model performance analysis proposed in Step number 3 ("Visual performance analysis"). The model validation has been performed using the experimental tests discussed in Section 3, for which quantitative data are available. However, since the experimental hydrographs are characterized by significant dispersions in the data (see Figs. 1–3), the visual approach seemed to be again the best choice and for this reason the metrics suggested in step 4 have not been applied here. In doing so, it was possible to provide a visual comparison between our results and other simulations, provided by other authors who used the visual performance method as well.

8. Conclusive remarks

The main aspect analyzed here is the suitable representation of the river model and its influence on the 2-D flood simulations. In particular, attention is focused on three issues: 1) procedures to
obtain a suitable computational grid, 2) techniques to introduce all the man-made structures and 3) inserting of hydrological data within the computational domain. The contribution on these aspects is summarized as follows:

- **Setting up the computational domain**: the triangular irregular elements grid can represent complex domains but, in our experience, procedures that automatically locate grid points, starting from a high resolution LiDAR, may provide poor results in terms of thalweg position and bank level. The procedure proposed herein is divided into three steps. It is based on an “automatic” step, to obtain a raw version of the computational grid, completed by two other steps in which corrections are provided by hand. The effect of the grid variations described in Section 5.1.3 (checking the topographical suitability of the grid), that represent the most important step in the grid modifications process to be carried out after the initial grid generation, has been highlighted in the paper. In particular, the use of “unchecked grid” simulation would lead to an underestimation of the water depths for a given out-of-bank flooded area or an overestimation of the flooded areas for a given water depth.

- **Inserting man-made structures that should interact with flow dynamics**: in situations similar to those analyzed in this paper, it is fundamental to take into account the presence of bridges, buildings and weirs. Near the bridges, the numerical results highlighted that an accurate geometric description of the piers coupled with a local grid refinement allowed the simulation of varying effects across the section and, therefore, the computation of a more detailed backwater upstream. These effects would not be described if the bridges were been described without considering the bridge geometry or with simplified methods. To improve the hydraulic description, a LiDAR survey has to be coupled with a terrestrial survey, in order to reproduce not only the local topography under the bridge but also the emerging bases and all the structures that could interact with the flow. This method requires a lot of data and manual work but it is seems necessary to avoid an underestimation of the backwater effects. However, it should be observed that the effect induced by a detailed bridge reconstruction discussed in this paper should be evaluated considering all the sources of uncertainty that may affect the estimation of backwater profile such as the design flood hydrograph and roughness values (Brandimarte and Kebede Woldeyes, 2013). Also for the weirs particular attention should be paid to the grid generation processes providing a local grid refinement, in such a way to approximate the weirs to a high topographic gradient.

- **Inserting hydrological data**: the main river receives significant lateral inflows in both of the case studies analyzed here. Therefore flood propagation analysis cannot be organized using just one discharge hydrograph imposed as the upstream boundary condition because there are a lot of inflows approaching the main river in different ways and from different points. In this paper, we have distinguished the main tributaries, that represent boundary conditions of the model, from the minor tributaries and other sources of discharges (small tributaries or lateral inlet in the main river due to the overland flowing over slopes), that are considered as a source term in the mass continuity equation. No problems related to model stability have been observed during the simulations. The analysis of the results has highlighted that the location of the upstream cross-section of a tributary has to be carried out carefully because it can induce significant variation in the prediction of flooded area extent. This observation allows us to underline the significant degree of uncertainty related to the connection between a hydrological model to calculate lateral inflows and a hydraulic model to calculate water levels along the river reach. Moreover, the effect due to the lateral inflow should be compared to the upstream inflow and considering all the uncertainty typically associated to river discharge data, which are quite inaccurate during flood conditions (Di Baldassarre and Montanari, 2009). A possible way to overcome these problems could be the use, at the catchment scale, of the shallow water equations modified in such a way to receive rainfall as input and linked with a specific module for the computation of the infiltration losses. Though this is a quite ambitious strategy, there are some recent papers in the literature that are moving in this direction (da Paz et al., 2011; Caviedes-Voullieme et al., 2012; Costabile et al., 2013; Paiva et al., 2013; Werner et al., 2013).

The accurate setting up of the river model influences the results of the 2-D model and provides a detailed description of the hydraulic behaviour of the flow dynamics. In particular, the analysis of flow regime highlighted significant longitudinal and transversal variations of the Froude number even for a channelized river. The simulated hydraulic phenomenon is characterized by backwater effects and sudden numerous changes in the flow regime, induced by the interaction between the flow and bridges, weirs and buildings. Supercritical flow occurs in several parts of the domain. In situations where high Froude numbers are to be expected, according to recent findings on this argument, simplified models (like diffusive or inertial models) should be used carefully because the disparity with a fully dynamic model in terms of water depths and flood propagation time (de Almeida and Bates, 2013). In all cases, it would be wise to check the performance of a simplified model using challenging tests (Macchione, 1994) like those shown in this paper.

The procedures and the techniques discussed in this paper require further analyses, comparisons and applications for an in-depth evaluation of their importance, limits and benefits. Moreover, we are aware that the issues discussed in this paper cannot enclose all the aspects that play a significant role in the flood dynamic. Indeed, in addition to what has been analyzed here, that usually are somehow treated in other papers, there are a lot of phenomena that can result as essential, especially in torrential rivers like those presented in this paper. In particular, we are making reference to river bed and bank erosion and deposition phenomena, transport of sediments, debris and woods. All these effects can play more dramatic effects of those described in our paper and in others like this. The study of these phenomena still requires a strong efforts by the scientific community and underlines the importance to deepen the knowledge of the processes within the fully-dynamic physical based modelling. For these reasons, the numerical results showed in this paper only have a methodological value for the processes considered here but do not exhaust the analysis of the hydraulic hazard for areas analyzed here.

In any case, looking at the practical purposes of this paper, several ideas proposed here may be useful not only for research reasons.

From a professional point of view, the approach proposed for setting up the computational domain leads to the awareness that the use of automatic procedures may be a source of errors in reproducing key elements of a cross-section (i.e. thalweg positions and bank level). Checking the “topographical correctness”, as described in Section 5.1.3, has to be a fundamental step in the computational grid development. Moreover, coupling airborne and terrestrial LiDAR has to become a common practice, especially in those situations in which the river reach is characterized by several structures that can interact with the flow dynamics (weirs, bridges and so on). In our opinion this is one of the key aspects in flood hazard mapping despite the fact that a detailed model set-up is
very time-consuming in both setting and run times and high-resolution data are not available for every region around the world. Finally, useful suggestions have been provided for the distributed discharges may be easily implemented in commercial software.

Acknowledgements

The authors want to thank the anonymous Reviewers for their constructive comments, that greatly contributed to improve the paper.

This work was supported in part by the European Union and the Region of Calabria in the funding project POR Calabria 2000–2006 Asse 1, Misura 14, Azione 1.4.c “Studio e sperimentazione di metodologie e tecniche per la mitigazione del rischio idrogeologico” Lotto 8 “Metodologie di individuazione delle aree soggette a rischio idraulico di eonsodazione”.

References


Asse 1, Misura 1.4, Azione 1.4.c, 2006. Metodologie di individuazione delle aree soggette a rischio idraulico di eonsodazione: “Studio e sperimentazione di metodologie e tecniche per la mitigazione del rischio idrogeologico” Lotto 8 “Metodologie di individuazione delle aree soggette a rischio idraulico di eonsodazione”.


