RESCDAM
DEVELOPMENT OF RESCUE ACTIONS BASED ON
DAM-BREAK FLOOD ANALYSIS

Final report: contribution of Enel.Hydro – Polo Idraulico e Strutturale

Index
1. Introduction ............................................................................................................. 2
2. FLOOD2D model .................................................................................................... 2
   2.1 The modelling method ...................................................................................... 2
   2.2 Modeling of hydraulic singularities .................................................................... 4
3. Seinajoki case study ............................................................................................... 5
   3.1 Topographical data ........................................................................................... 5
   3.2 Flood scenarios ................................................................................................ 5
   3.3 Computational figures ....................................................................................... 7
   3.4 Results .............................................................................................................. 8
   3.5 The effects of the topography and the treatment of urban areas ................... 12
References ............................................................................................................... 17
1. Introduction

This document describes the contribution of Enel.Hydro Polo Idraulico e Strutturale to the RESCDAM project. The main task has been the numerical simulation of the flood wave following the possible breaking process of the Kyrkosjärvi dam, upstream the city of Seinäjoki, Western Finland.

Chapter 2 describes the modelling method and the numerical code FLOOD2D developed and used by Enel.Hydro. In chapter 3 the inputs and the assumptions which have been used for six different scenarios are presented in a deep.

2. FLOOD2D model

2.1 The modelling method

Enel.Hydro Ricerca Polo Idraulico e Strutturale has developed a computer program based on the integration of de Saint Venant equations for two-dimensional flow. The model neglects the convective terms in the momentum conservation equations. The scheme to discretize the simplified equations in time is as follows:

\[
\frac{h^{n+1} - h^n}{\Delta t} + \left( \frac{\partial q_x}{\partial x} \right)^{n+1} + \left( \frac{\partial q_y}{\partial y} \right)^{n+1} = 0
\]

\[
\frac{q_x^{n+1} - q_x^n}{\Delta t} + gh\left( \frac{\partial h}{\partial x} \right)^{n+1} + gh\left( C_f q_x^{n+1} - S_x \right) = 0
\]

\[
\frac{q_y^{n+1} - q_y^n}{\Delta t} + gh\left( \frac{\partial h}{\partial y} \right)^{n+1} + gh\left( C_f q_y^{n+1} - S_y \right) = 0
\]

where \( h \) = water depth, \( \Delta t \) = time interval, \( g \) = gravitational acceleration, \( q \) = specific flow rate, \( x,y \) = spatial coordinates, \( C_f \) = resistance coefficient; \( S_x, S_y \) = ground slope components in the \( x \) and \( y \) directions. The parameter “\( \theta \)” can take values from 0 to 1. 

\( \theta = 0 \) gives an explicit model in terms of water depth gradients, which is easy to set up, but has a limited time integration step; \( \theta = 1 \) gives a completely implicit model, which is much more complicated to implement, but, however, allows large time steps.

The space derivatives of the specific flow rates which appear in the continuity equation are calculated at time “\( n+1 \)”, since the model is stable for all values of \( \theta \) only in this case, as can be seen from the analysis shown in (Molinaro et al., 1992).

For the spatial discretization the grid shown in Figure 1 is used.
Figure 1: Difference grid in x,y space

From the above equations the following expressions for the flow rate \( q_x \) and \( q_y \) at time “n+1” are obtained:

\[
q_{x}^{n+1} = -\frac{gh^u \Delta t}{1 + gh^u C^u \Delta t} \left( \frac{\partial h}{\partial x} \right)^{n+\theta} + \frac{gh^u S_x \Delta t + q_x^n}{1 + gh^u C_x \Delta t}
\]

\[
q_{y}^{n+1} = -\frac{gh^u \Delta t}{1 + gh^u C^u \Delta t} \left( \frac{\partial h}{\partial y} \right)^{n+\theta} + \frac{gh^u S_y \Delta t + q_y^n}{1 + gh^u C_y \Delta t}
\]

The equations are solved for \( \theta \neq 0 \) with an iterative method.

The two-dimensional model usually requires only the natural ground topography and estimated Manning’s friction factor for data input. The area of the surface of study is viewed as a set of adjoining rectangular shaped prismatic cells resulting in an cartesian grid. In cases where the area is represented with simplex triangle elements an irregular triangular grid is obtained. At each time step the normal components of velocities through “walls of interface” of adjoining cells are determined from the equilibrium equations, the related flows are computed and the storage volume in each cell is determined, this leads to a new water surface configuration for next computational interval.

The interpolation process of the depths of the water passing through the walls of interface of the cells is crucial in the prediction of the propagation of flood waves. In the developed of 2D models, Enel.Hydro Polo Idraulico e Strutturale has adopted various criteria in order to simulate incipient flood processes, embankment overflows or simple water passage on sloping ground.
2.2 Modeling of hydraulic singularities

The term “hydraulic singularity” refers to a point of the area of study where the gradually varied flow equations will no longer be used. The effect of singularities is expressed in terms of the discharge associated to head-loss between the upstream cell and the downstream cell of a singular point.

The flow through the singularities is simulated using a one-dimensional flow approximation. The singularities are easily accommodated between the cell sides and the set of equations that describe the flow through bridges, culverts, barrages and highway embankments are solved (Molinaro & Pacheco, 1994).

**Weir or roadway embankment**

The flow over a roadway embankment is simulated by dividing the singularity into sections called weir segments. Each weir segment is associated to a cell side and described by a discharge coefficient and the crest elevation of the weir. When flow over the weir segment is affected by tailwater, the discharge coefficient is corrected automatically using different relationships presented in the literature.

**Barrage**

Flow through a barrage is treated similarly. Weir segments and numerous orifices may be added together. The total discharge \( Q \) is distributed over each elementary singularity. In the case where the flow over both upstream and downstream cells are subcritical, the computation progresses upstream. The downstream and upstream levels are known. If the upstream flow regime is supercritical the singularity is a control section and for each of the weir segments a free flow expression is adopted.

**Culvert**

Flow through a culvert is computed as being controlled by characteristics of the outlet of the culvert. The hydraulic capacity is calculated with the sum of a liner friction loss and a Borda type head-loss at the downstream end.

**Bridge**

One-dimensional flow through a bridge is calculated using the equation of momentum and the total drag force exerted on bridge piers by the flow. Each bridge is described by a drag coefficient and the physical characteristics of the bridge. Both normal free-surface flow and pressure flow conditions are simulated. “Pressure flow” meaning the water being in contact with the top of the bridge arch.

Particular attention is paid to the continuity between the different flow conditions. In fact, the upstream hydraulic level becomes lower than the elevation of the upper
edge and the orifice acts as a sill. The continuity of the formulation for every flow condition (sill to orifice or viceversa), is very important in order to achieve convergence during the iterative solution process.

3. Seinajoki case study

3.1 Topographical data

The following two sets of topographical data were used:
1. Rectangular mesh # 10m, without buildings
2. Rectangular mesh # 10m, with buildings

Given the specific characteristics of the FLOOD2D code, no interpolation of the rectangular grid was required: as a consequence a total number of 446961 grid points were used. Much work has been devoted to the editing of the input files since FLOOD2D requires specific formats.

The location and dimensions of the breach were provided by FEI: in particular the breach width varies from 160 m (MW) to 170 m (HW) to 200 m (We).

3.2 Flood scenarios

The following six scenarios were used for the numerical computations:

**RUN 1:** MQ base-flow + breach hydrograph, constant Manning “n=0.06” in the entire modelling area.

**RUN 2:** HQ1/100 base-flow + breach hydrograph, constant Manning “n=0.06” in the entire modelling area.

**RUN 3:** MQ base-flow + breach hydrograph, Manning “n” varying in the area of the entire modelling area according to landforms and vegetation.

**RUN 4:** HQ1/100 base-flow + breach hydrograph, other conditions as in RUN 3.

**RUN 5:** Conditions of RUN 3 modified according to the partner’s choice of modelling buildings (Enel.Hydro: geometry).

**RUN 6:** Conditions of RUN 4 modified according to the partner’s choice of modelling buildings (Enel.Hydro: geometry).

Table 1 and Figure 2 shows the inflow conditions for all the runs.
Concerning the downstream boundary conditions, the rating curves provided by FEI were used: a \( q(h) \) function on each cell of the north boundary of the Seinäjoki river was set with reference to a specific location \( (X=2439960 \quad Y=6966863) \).

Some particular attention was paid to the railroad embankment and the relative boundary effect. With reference to Figure 2, a specific boundary condition for the case of the overtopping of the railroad was used, after having modified the topography of the entire area by setting the elevation of 44.2 m on the yellow line.
The initial conditions for all the runs are a steady state flow in the rivers (see table 1); this condition is given by a preliminary computation. The following values for the roughness coefficients (Manning-\(n\)) were used for runs 3, 4, 5 and 6:

0.055 1 = an area covered by water
0.025 2 = road
0.035 3 = cultivated area, field or hydraulically equal area
0.070 4 = cutted forest, wetland etc.
0.140 5 = forest
0.120 6 = buildings

3.3 Computational figures

All the simulations were performed with the Unix version of the FLOOD2D code. The following average data for a typical complete run (\(n=0.060\)) were obtained:
- Total duration (flooding time): approx. 8 hours
- Total duration (CPU time): 39 h 20 min (We), 32 h 10 m (MW)
Maximum error on water volume: 0.4%
Max number of wetted cells: 68512 (We), 55439 (MW)
Wave front arrival time at downstream boundary: 2 h (We), 2 h 30 m (MW)
Time step: 1 sec

3.4 Results

The following output arrangements were provided for each node or element:

* Time difference between result values 5 min
* Spatial co-ordinates x, y, z (ground elevation)
* Flow velocities Vx, Vy
* Water elevation or/and water depth m

As an example of the obtained results, figures from 4 to 7 show the wave propagation in four different time intervals for RUN 1, while figures from 8 to 11 are pertinent to RUN 6.
Figure 5

Figure 6
Figure 7

Figure 8
Figure 9

Figure 10
3.5 The effects of the topography and the treatment of urban areas

The two-dimensional model usually requires only the natural ground topography and estimated Manning’s friction factor for data input. The area of the surface of study is viewed as a set of adjoining rectangular shaped prismatic cells resulting in a cartesian grid. In cases where the area is represented with simplex triangle elements an irregular triangular grid is obtained. At each time step the normal components of velocities through “walls of interface” of adjoining cells are determined from the equilibrium equations, the related flows are computed and the storage volume in each cell is determined, this leads to a new water surface configuration for next computational interval.

The interpolation process of the depths of the water passing through the walls of interface of the cells is crucial in the prediction of the propagation of flood waves. In the development of FLOOD2D model, Enel.Hydro Polo Idraulico e Strutturale has adopted various criteria in order to simulate incipient flood processes, embankment overflows or simple water passage on sloping ground.
The bed elevation between two adjacent cells is computed considering the dimensions of ground irregularities according to the following procedure.

A control parameter “VMRH” is fixed, which represents the upper boundary of the ratio between the absolute value of the difference of the water depths at two adjacent cells “HO” and their mean value “HN” (see Fig. 12).

For the grid model, both the average elevation associated with each cell and the water surface configuration at each time step are known. Then, a value of “DHHN” is computed between two adjacent cells, its value $\text{DHHN}=(\text{DH}/\text{HN})_{\text{real}}$ is compared with the value of the control parameter “VMRH”. The following cases are possible:

CASE 1) $\text{DHHN} \geq \text{VMRH}$

The difference between water depths is large due to the extreme irregularity of the ground; the bottom elevation at the interface is computed as:

$$\text{ZFFON} = \text{ZFMAX} \quad (5)$$
$$\text{ZFFON} = \max [ \text{ZF}(i,j) , \text{ZF}(i-1,j) ] \quad (6)$$

CASE 2) $\text{DHHN} < \text{VMRH}$

The water depth does not change much from cell to cell and the bottom elevation is computed according to the following expression:

$$\text{ZFFON} = \left[ 1 - \frac{\text{DHHN}}{\text{VMRH}} \right] \times \text{ZFMED} + \left[ \frac{\text{DHHN}}{\text{VMRH}} \right] \times \text{ZFMAX} \quad (7)$$

The equation above ensures the continuity of the bottom elevation when a transition occurs between case 1 and case 2.

![Diagram of the bed elevation procedure](figure12.png)

Figure 12: Diagram of the bed elevation procedure
With reference to the above scheme, if $H$ is the water depth in the center of a cell, then the mean water depth is:

$$HMED = 0,5 \times [H_{i-1} + H_i]$$

While the average free surface elevation is:

$$Z = ZFMED + HMED$$

The water depth to be considered between two adjacent cells is:

$$HN = Z - ZFFON$$

and if the value of $HN$ is less than or equal to zero the discharge is set to zero.

In the case of flow over a step the possible cases and the corresponding correct choices of the water depth are illustrated in Figure 13.

---

**Figure 13: Possible flows over a sudden change of bed level**
In FLOOD2D the computation of water flow through urban areas can be also performed by assuming that only part of the total built-up area is available for water storage or water flow. In this case the concepts of “Urban Porosity” and “Transmissivity” of a built-up area were used, with clear analogy to flow in a porous fractured medium (Braschi and Gallati, 1989).

Considering a particular cell of the grid of computation over an inhabited zone, the total area of the cell that can be flooded is generally reduced by the buildings whose area is not available for water storage.

Figure 14 shows the buildings inside a computational cell area drawn in black, and the effective area “Ae” defined as:

\[
Ae = \eta_L D_x D_y + \eta_L D_y (1 - \eta_L) D_x
\]

where \(\eta_L\) = ratio between the “free length” and the total length (\(\Delta x\) or \(\Delta y\)) of each cell side; \(\Delta x, \Delta y\) = length of the cell side.

![Figure 14: Modelling of urban areas](image)

From the above relationship it is possible to evaluate the ratio “\(\eta_A\)” between the effective area and the total area as a function of “\(\eta_L\)”:

\[
\eta_A = \frac{Ae}{D_x D_y} = \eta_L + \eta_L (1 - \eta_L) = \eta_L (2 - \eta_L)
\]

The new parameters “\(\eta_A\)” and “\(\eta_L\)” may be referred to as the “aerial porosity” and the “linear porosity” of a built-up area. The parameter “\(\eta_L\)” can be defined as a function of “i,j” indexes.

The continuity equation should be modified considering the effective area of a cell and the “free length” of each side; following this line the general discretized form of this equation becomes, with reference to a generic “i,j” cell:
\[
\eta_L (2 - \eta_L) \frac{dh}{dt} + \frac{\bar{\nu}_{L,j+1} q_{x,j+1} - \bar{\nu}_{L,j} q_{x,j}}{\Delta x_j} + \frac{\bar{\nu}_{L,j+1} q_{y,j+1} - \bar{\nu}_{L,j} q_{y,j}}{\Delta y_j} = 0
\]

where:

\[
\eta_{L,i} = \min (\eta_{L,i}; \eta_{L,i+1})
\]

\[
\eta_{L,j} = \min (\eta_{L,j}; \eta_{L,j+1})
\]

The above expression is implicitly based on the hypothesis that water cannot get into buildings as the term for the rate of storage into buildings is smaller than the term representing the exchange of water between adjacent cells.
References


