PERIODICA POLYTECHNICA SER. CIVIL. ENG. VOL. 38, NO. 2, PP. 187-203 (1994)

NUMERICAL MODELLING OF FLOW PATTERNS TO ASSIST THE REVITALISATION OF SECONDARY RIVER BRANCHES IN GEMENC AREA

János JÓZSA, Csaba GÁSPÁR and Sándor SZÉL

VITUKI Consult Rt
H–1095 Budapest,
Kvassay J. út 1.

Received: March 31, 1994

Abstract

Depth-averaged numerical modelling of steady-state horizontal flow patterns carried out in the framework of the revitalisation project of the secondary branches in Gemenc Protected Landscape Area is presented. The feasibility of different variants is investigated in order to increase the water exchange in the secondary branches without creating unfavorable hydraulic and related sediment transport as well as navigation conditions. The applied mathematical model is based on the usual shallow water assumptions. The numerical solution is performed using finite differences and multigrid iteration technique. The model is able to reproduce recirculation zones and the horizontal velocity distribution in an acceptable manner.

Keywords: Gemenc revitalisation, Danube, numerical modelling, depth-averaged flow patterns, multigrid method.

1. Introduction

Recently great national and international attention has been paid to the Protected Landscape Area of Gemenc (see e. g. MARCHAND, 1993; ZSUFFA, 1993). To improve the water management of the largest flood plain forest in Europe, the investigations have focused on the revitalisation of the water system including the opening and/or dredging of some of the secondary river branches. In fact, since most of these branches are closed in low flow period, they have got rather unfavorable water exchange resulting occasionally in poor water quality. An obvious solution to improve the flushing of such branches would be to open them by removing cross-dams and by dredging. It would, however, significantly change the flow and silting conditions in the branches as well as in the vicinity of their bifurcation and confluence with the main Danube channel. Moreover, at high conveyance capacity of a secondary branch its effect, either favorable or not, may not be negligible on the flow in the main channel. High side-discharges can generate large recirculation zones and streamline curvatures in the region downstream of the confluence, which can then worsen the existing navigation conditions. To compensate such effects in the impact area, additional river training interventions may be needed.

To analyse the hydraulic feasibility of the above mentioned revitalisation works, field measurements, conventional scale modelling as well as advanced numerical modelling have been applied. In fact, in a number of recent fluvial projects numerical flow models have been proven a reasonable, although simplified alternative to laboratory scale models. It is particularly so in the preliminary and feasibility phases of the projects, in which a number of variants have to be investigated in a fast and relatively cheap way. Detailed studies by means of rather costly scale models are performed then for the pre-selected, most promising variants only. The present paper reports some of the hydraulic investigations of the planned conditions carried out by numerical tools, namely by depth-averaged flow modelling. In the investigations the stress was on calculating and evaluating steady-state flow patterns in prevailing discharge and topographic conditions. The studies included three secondary branches (*Fig. 1*) out of which the results for Vén-Duna will be analysed more in details.

2. Model Formulation

As to the mathematical basis, the most general description of free surface flows can be done in time dependent three-dimensional form. However, the special character of river flows, moreover, the required practical accuracy make it possible to simplify the complete description to some reasonable extent. The simplifications are based on order of magnitude analysis and permit the approximate numerical solution even in personal computer environment.

The most widely used simplification in river hydraulics is to handle the processes in their cross-sectional integral-averaged form. This leads to the well-known one-dimensional continuity and de Saint-Venant equations, which are the basis of most of the river flood wave propagation models. While the one-dimensional approach can be applied in long river reaches, the investigation of horizontal flow patterns in shorter stretches, at confluences as well as in the vicinity of river training works require an at least two-dimensional approach. As is well known, in fluvial hydraulics the so-called shallow water conditions generally apply, i. e. the depth is very small compared to the horizontal dimensions, currents are nearly horizontal, dominated usually by bottom friction, thus the primary flows can be described by depth-averaged velocities. In fact, it is the two-dimensional depth-averaged modelling that has become the most widely used tool of the practicing river engineers to analyse the above mentioned phenomena.

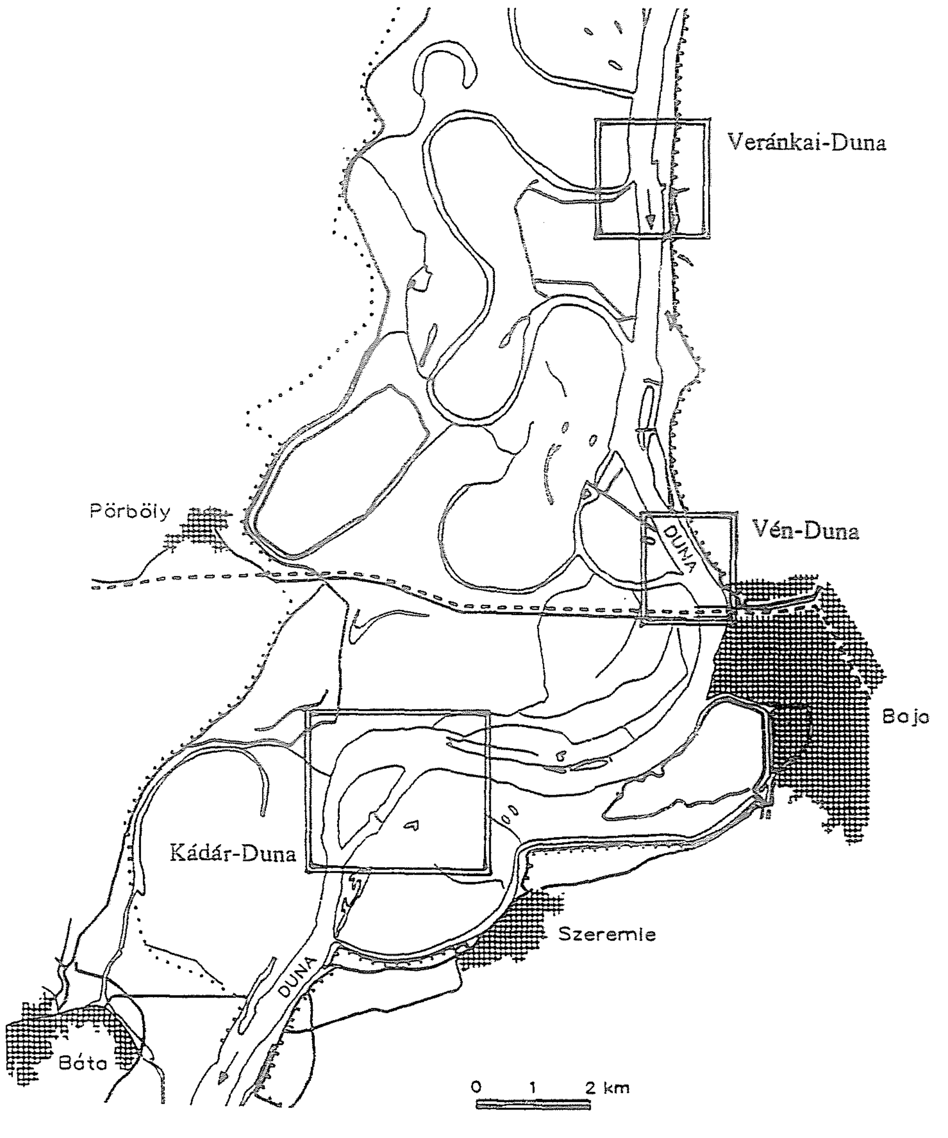

Fig. 1. General layout of Gemenc Protected Landscape Area showing the three case study sites

It is, of course, desirable to develop three-dimensional models as well, but the storage requirements and computing times of such schemes for large applications are still beyond the reach of present day computers. That is why three-dimensional models have so far been applied mainly to near-field problems only.

In general an analytical solution to the governing equations supplied with the appropriate initial and boundary conditions cannot be found. That is why a number of numerical solution techniques have been developed and are available to set up an approximate discrete solution to fluid flow problems. Some of these methods have been already implemented to the shallow water equations, whereas some innovative ones, although they present very favorable numerical features, have not been adapted and applied yet. From the point of view of the flow problem to be solved, an efficient and robust model should meet the following requirements:

- Appropriately refined discretised description of the flow domain including the shore line geometry and the depth conditions,
- negligible numerical error introduced by the finite approximation of the derivatives in the governing equations, and
- fast solution of the resulting system of algebraic equations to make it possible to carry out the evaluation of a great number of different hydraulic variants in reasonable computing time.

Usually all of the above mentioned features are simultaneously not the attributes of the existing techniques, thus the task of a numerical modeller is to find a reasonable compromise by trying to set up a nearly optimal combination of various techniques.

In the following sections the model formulation including the interpretation of the governing equations and the main principles of the applied numerical method is briefly described. Starting from the most general, time dependent three-dimensional Navier-Stokes equations, the shallow water equations are obtained by carrying out Reynolds-type time averaging, vertical integral averaging assuming hydrostatic conditions, as well as a simple closure of the resulting system of differential equations adopting the Reynolds-analogy with constant eddy viscosity concept. After having set up the mathematical model, a brief overview of the applicability of the available numerical solution techniques is given. Since the main objective of the investigations is to determine the steady-state flow patterns in various geometrical and hydraulic conditions, when selecting the method to be implemented, fast iterative solution methods are preferred to conventional time marching techniques. One of such techniques is the so-called multigrid method, providing practically all the features required for establishing an efficient, robust model.

2.1 Governing Equations

As is well known, the most general space-time description of incompressible fluid flows is given by the following Navier-Stokes equations:

$$\text{div } \mathbf{v} = 0\,, \tag{1}$$

$$\frac{\partial \mathbf{v}}{\partial t} + (\nu\nabla)\mathbf{v} - \nu\Delta\mathbf{v} + \frac{1}{\rho}\nabla P = 0\,. \tag{2}$$

Eqs. (1–2) express in a vector form the conservation of volume and momentum of the flow, respectively, and are valid both in laminar and turbulent conditions. However, in turbulent regime the fluctuation of the flow features over a wide spectrum makes it hardly possible to directly use *Eqs.* (1–2) for practical problems. The first step toward simplifying the basic equations is to carry out the so-called Reynolds-type time-averaging, which results in the Reynolds equations including turbulent stresses. Furthermore, in shallow water flows in which the depth is small compared to the typical horizontal length scale, a

$$P = \rho g(\eta - z)$$

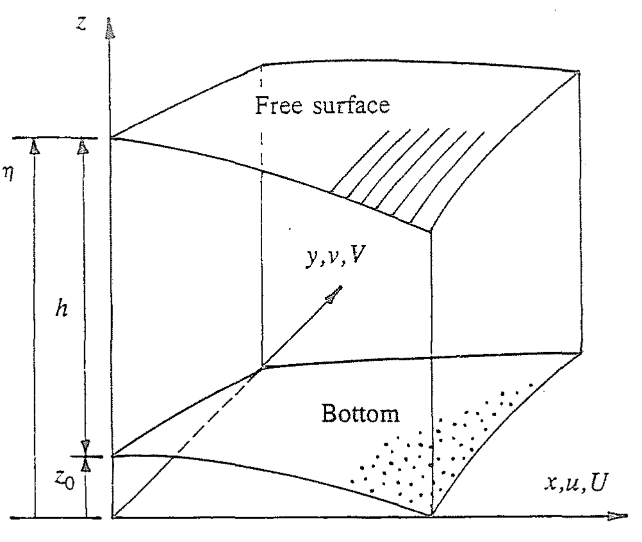

Fig. 2. Definition sketch

hydrostatic pressure distribution can be assumed, which implies that vertical fluid accelerations are negligible compared to gravity. Moreover, if the flow is nearly horizontal, then in order to obtain the shallow water equations, the spatial equations can be integrated over the total depth, from the bottom level, $z = z_0$, to the free surface, $z = \eta$, as defined in *Fig. 2*, and all parameters are expressed in terms of the water depth, $h = \eta - z_0$, and the following fluxes:

$$p = \int_{z_0}^{\eta} u\mathrm{d}z\,, \qquad q = \int_{z_0}^{\eta} v\mathrm{d}z\,,$$

which are divided by the local depth to determine the

$$U = \frac{p}{h}\,, \qquad V = \frac{q}{h}$$

depth-averaged velocities.

Integrating the Reynolds equations and applying the Leibnitz rule with appropriate boundary conditions at the free surface and at the bottom yield the shallow water equations which, however, contain the so-called effective stresses (FLOKSTRA, 1977) expressing the deviatoric fluid stresses and terms resulting from integration over depth. As to the effective stresses, the closure of the system of differential equations can be done in various ways. The most advanced versions to date treat this problem by introducing equations for the transport of kinetic energy and for its dissipation rate. However, in depth-averaged context the derivation of the model is rather heuristic (MCGUIRK – RODI, 1978 RODI, 1980), moreover, in many practical cases the depth-averaged shallow water flows seem to be dominated mainly by the bottom friction. Consequently, according to the required accuracy of feasibility studies, the simplest closure procedure is implemented here by the overall adoption of the Reynolds analogy and introducing a uniform effective viscosity coefficient ν_e, which yields the shallow water equations as

$$\frac{\partial h}{\partial t} + \frac{\partial p}{\partial x} + \frac{\partial q}{\partial y} = 0\,, \tag{3}$$

$$\underbrace{\frac{\partial p}{\partial t}}_{1} + \underbrace{\frac{\partial}{\partial x}\left(\frac{p^2}{h}\right)}_{2} + \underbrace{\frac{\partial}{\partial y}\left(\frac{pq}{h}\right)}_{3} + \underbrace{gh\left(\frac{\partial h}{\partial x} + \frac{\partial z_0}{\partial x}\right)}_{4} - \underbrace{\nu_e\left(\frac{\partial^2 p}{\partial x^2} + \frac{\partial^2 p}{\partial y^2}\right)}_{5} + \underbrace{\frac{\tau_{bx}}{\rho}}_{6} = 0 \tag{4}$$

$$\frac{\partial q}{\partial t} + \frac{\partial}{\partial y}\left(\frac{q^2}{h}\right) + \frac{\partial}{\partial x}\left(\frac{pq}{h}\right) + gh\left(\frac{\partial h}{\partial y} + \frac{\partial z_0}{\partial y}\right) - \nu_e\left(\frac{\partial^2 q}{\partial x^2} + \frac{\partial^2 q}{\partial y^2}\right) + \frac{\tau_{by}}{\rho} = 0\,. \tag{5}$$

The terms in the momentum equations (4–5) represent the following effects:
(1) Local acceleration.
(2–3) Advective acceleration.
(4) Gravity force assuming hydrostatic conditions.
(5) Horizontal momentum exchange.
(6) Dissipation due to bottom friction.

In river flows the bottom shear stress components follow a quadratic law based on the Manning formula, thus they can be expressed as

$$\tau_{bx} = \frac{\rho g}{k^2 h^{\frac{7}{3}}}\sqrt{p^2 + q^2}\,p\,,$$

$$\tau_{by} = \frac{\rho g}{k^2 h^{\frac{7}{3}}}\sqrt{p^2 + q^2}\,q\,.$$

In steady-state conditions the time derivatives of the water depth and the fluxes become zero in *Eqs.* (3–5), expressing the fact that in such cases there is no local acceleration and free surface variation.

2.2 Numerical Solution

Since the investigations focused only on the steady-state flow patterns, the numerical model was based on the steady-state version of *Eqs.* (3–5), although the solution could have been achieved by using the unsteady equations as well. In fact, unsteady models are often applied to determine steady-state solutions by carrying out time-dependent calculation with steady-state boundary conditions until the system reaches its hydraulic equilibrium. Nevertheless, this is usually not the fastest way to converge to the steady-state, that is why to speed up the calculations advanced iterative techniques are preferred.

In addition to the fast convergence, in the present practical case an important requirement for the applied numerical method is the sufficiently fine discrete description of the generally irregular, curved river shorelines and depth variations. It can be achieved essentially in two different ways:

– applying high resolution Cartesian grids, or
– curvilinear, boundary-fitted grids.

Most of the commonly used Cartesian systems apply an orthogonal, equidistant finite difference grid. On such a grid the space derivatives of *Eqs.* (3–5) can be approximated by finite differences using neighbouring nodal values in a simple way, however, the boundaries, especially the ones that do not coincide with a grid line, are represented in the conventional staircase way.

Using curvilinear systems, the grid is generated in such a way that it fits the boundaries of the flow domain, which requires the transformation of the governing equations to the same system (see e. g. PAVLOVIC – KAPOR – DJURIC; 1984, HÄUSER et al., 1985). It is because of this reason that the finite difference schemes in this system become much more complicated than in a Cartesian one, furthermore, in case of multiply connected domains the generation and handling of boundary-fitted grids are far from being straightforward.

As to the finite element methods, their inherent character is to provide boundary-fitted meshes with arbitrary internal refinement (CONNOR – BREBBIA, 1976). On the other hand, the discrete solution of the flow problem on such an unstructured mesh is very costly, since the band width of the system of equations to be solved is generally large.

To preserve simplicity, the numerical model applied in the present study was established using Cartesian finite differences. Since the staircase-like description of the irregular boundaries requires fine spatial resolution, a special, fast iterative method, the so-called multigrid technique was implemented (BRANDT, 1984; STÜBEN – TROTTENBERG, 1984). The main feature of this method is that instead of using one single fine grid, it uses a series of subsequently coarsening grids, and the solutions on the coarser grids are then used in the finer grid approximations. An important element of the procedure is to find a good enough solver or rather a smoother on a single grid. In our case the idea of the SIMPLE pressure correction algorithm due to PATANKAR (1980) was used. In a multigrid solution of the two-dimensional Navier-Stokes equations this algorithm was proven very efficient (SIVALOGANATHAN – SHAW, 1988). In free surface conditions where the pressure is replaced by the water depth and the position of the free surface generally changes during the iteration, the algorithm can also be used as is shown by GÁSPÁR – JÓZSA (1992). Derivatives in the model are approximated by central differences, except the advective terms, which are replaced by appropriate upwind schemes.

For flexible pre- and post-processing the model is supplied with easy-to-use computer graphical tools. These tools considerably facilitate the handling of the geometry of the flow domain represented on the finite difference grid, and make it possible to set up various hydraulic configurations and boundary conditions. The output flow patterns can be displayed either as velocity fields or can be visualised by means of particle tracking techniques. The model has been already applied both for river Danube (GÁSPÁR – JÓZSA, 1992) and Tisza (JÓZSA et al., 1992).

3. Applications

The above mentioned numerical model was used in three regions in Gemenc Protected Landscape Area. In all cases the task was to investigate the modification of the flow conditions in the vicinity of the bifurcations and confluences in the planned situations. The main goal was to find a compromise to improve the water exchange of the secondary branches while creating no significant new hydraulic conflicts such as the undesirable silting of some places or the worsening of navigational conditions in the main channel.

In the applications an appropriate grid cell size was chosen to represent sufficiently both the shoreline and depth irregularities. The model parameters, first of all the smoothness coefficient k, were calibrated against measured water levels. At shoreline-type boundaries the prescribed condition was, of course, flow parallel to the solid wall. At upstream open boundaries usually the steady-state discharge, or more exactly the flux distribution was given. At the downstream boundary constant water level related to the discharge was imposed. Since the model uses a so-called staggered grid system, the steady-state fluxes are calculated in the middle of cell faces, whereas water depths are interpreted in cell centres. Depth-averaged velocities can then be determined by some interpolation.

3.1 Veránkai-Duna

The influence of opening the Veránkai-Duna secondary river branch in the bifurcation region was investigated from the point of view of the flow and transport conditions. Steady-state flow patterns with a number of secondary branch discharge values were calculated. Simultaneously, in situ tracer measurements were also carried out providing calibration data for the model. Tracking particles in the calculated flow field in conditions similar to the ones existing in the field experiment, reasonable agreement between measured and calculated trajectories was found. Once the model having been justified, flow patterns were evaluated also by applying various planned topographic modifications.

3.2 Kádár-Duna

In this case the modelled area included the whole secondary river branch together with the bifurcation and the confluence, making it possible to directly model the opening of the cross-dam in Kádár-Duna. First the prevailing flow fields in the existing conditions were calculated, showing a large recirculation zone at the confluence, which probably plays an important role in the siltation of this zone. The calculations showed that by opening the cross-dam in the secondary branch the throughflow suppresses the recirculation zone, improving the sedimentation conditions. Furthermore, higher side-discharges influence a rather large area downstream in the main channel, including an inflexion point not far from the confluence. In the past severe ice jam formation was detected in this river reach. Apart from trying to reduce the extent of low velocity siltation zones, it is the subject of future numerical investigations to analyse the effect of the opening of the secondary branch on the ice and navigation conditions in this reach.

3.3 Vén-Duna

In the framework of an extensive feasibility study (ZSUFFA, 1993), detailed investigations were carried out to analyse the opening of Vén-Duna secondary branch, which is situated near Baja, just upstream of the only bridge of the region (see *Fig. 1*). To update the existing topographic information, first a survey resulting in the isodepth map seen in *Fig. 3* was conducted. To cover the study area, on the finest level a 80×41 grid with 20×20 m cell size was defined. All the calculations were carried out with the typical low flow discharge, $Q = 1067\ \mathrm{m}^3/\mathrm{s}$. As to the model parameters, the calibration against measured water level resulted in $k = 40\ \mathrm{m}^{\frac{1}{3}}/\mathrm{s}$ and $\nu_e = 1\ \mathrm{m}^2/\mathrm{s}$.

Without modifying the existing topographic conditions, steady-state flow fields were calculated with a range of side discharges, from 0 to 200 m^3/s. The flow patterns with side-discharge 100 and 200 m^3/s, visualised by selected particle trajectories, are given in *Fig. 4* and *Fig. 5*, respectively. What can be seen in these figures is the strengthening of the intrusion of Vén-Duna flow to the main stream by increasing side-discharge, together with the formation of a large recirculating eddy. In the existing topography, at 200 m^3/s, it seems to reach an undesirable impact even in the downstream part of the domain, at the bridge piers. Unluckily, the groin at the downstream edge of the confluence further enhances this ef-

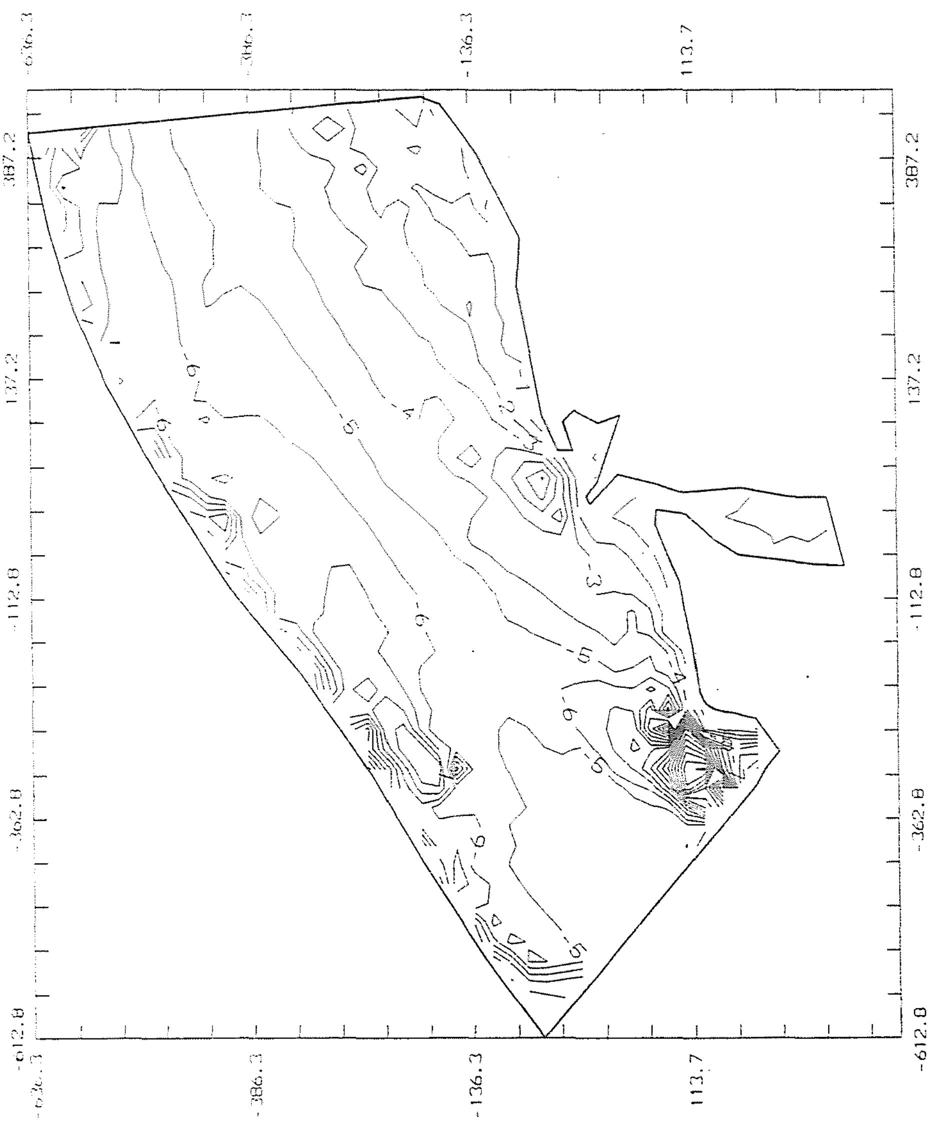

Fig. 3. **Depth contours of Vén-Duna confluence area based on recent survey**

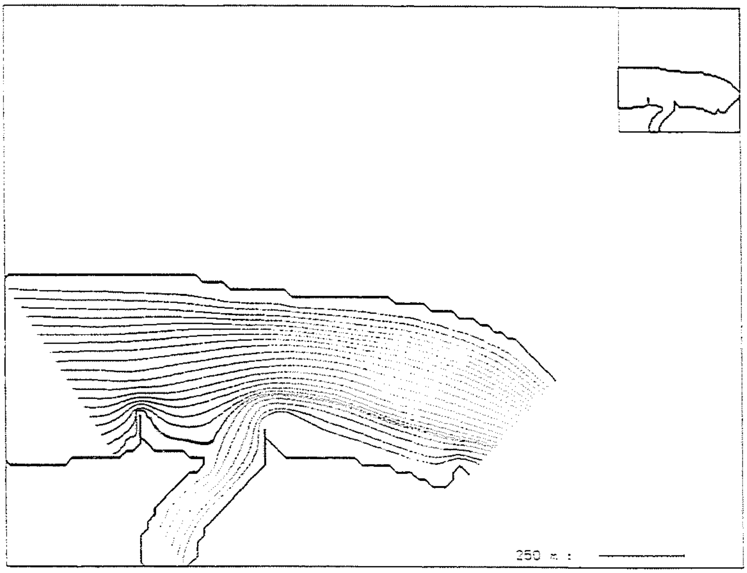

Fig. 4. Calculated flow pattern at Vén-Duna confluence in the existing topographic conditions. Discharge in the main channel: 967 m^3/s. Discharge in the secondary branch: 100 m^3/s

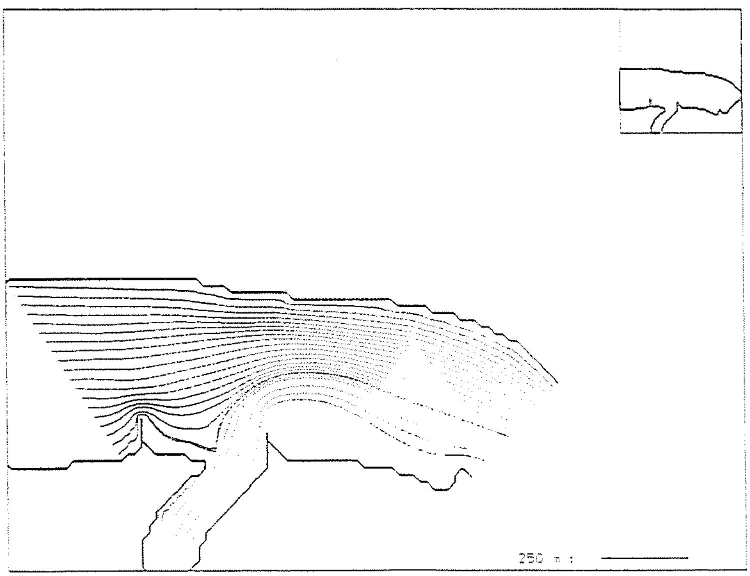

Fig. 5. Calculated flow pattern at Vén-Duna confluence in the existing topographic conditions. Discharge in the main channel: 867 m^3/s. Discharge in the secondary branch: 200 m^3/s

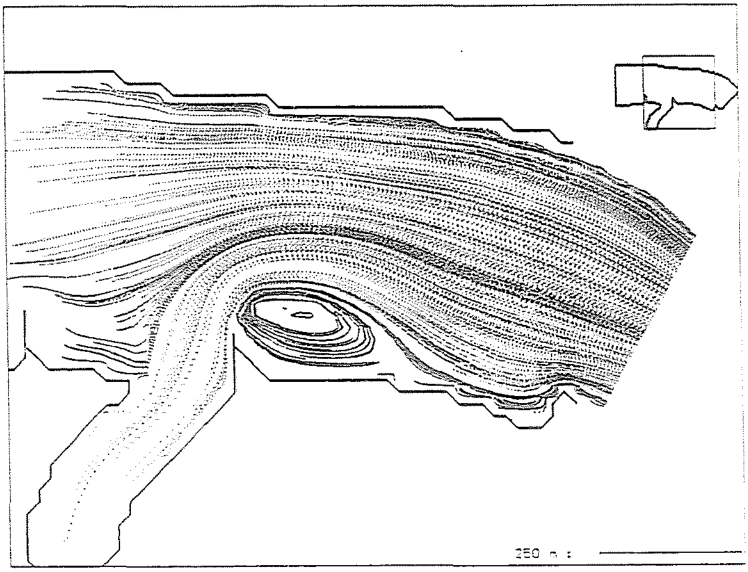

Fig. 6. Calculated flow pattern at Vén-Duna confluence in the existing topographic conditions. Discharge in the main channel: 867 m^3/s. Discharge in the secondary branch: 200 m^3/s

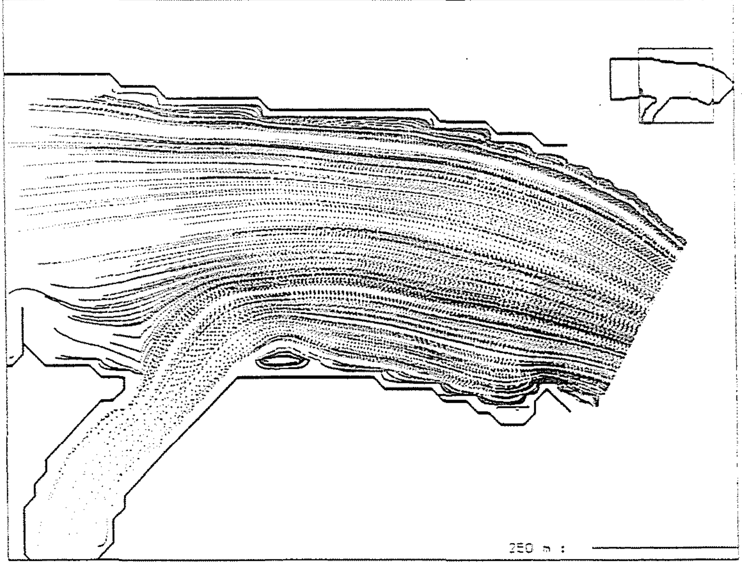

Fig. 7. Calculated flow pattern at Vén-Duna confluence with groin removal. Discharge in the main channel: 867 m^3/s. Discharge in the secondary branch: 200 m^3/s

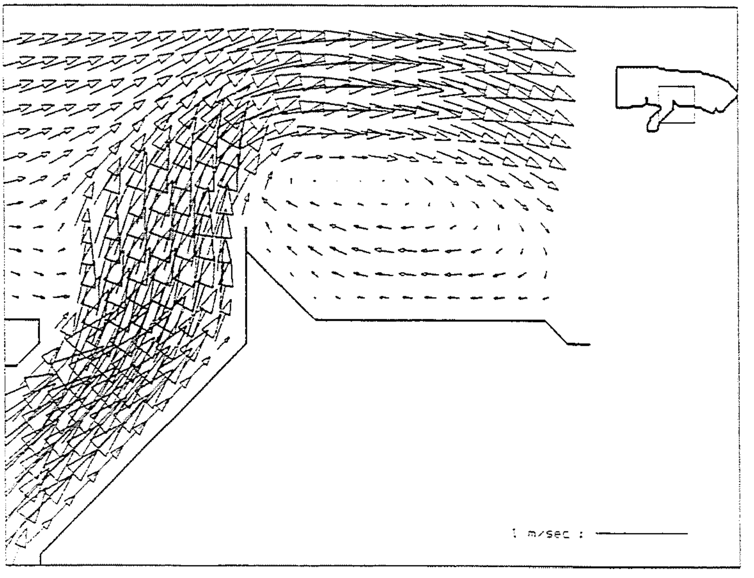

Fig. 8. Calculated depth-averaged velocity field at Vén-Duna confluence in the existing topographic conditions. Discharge in the main channel: 867 m^3/s. Discharge in the secondary branch: 200 m^3/s

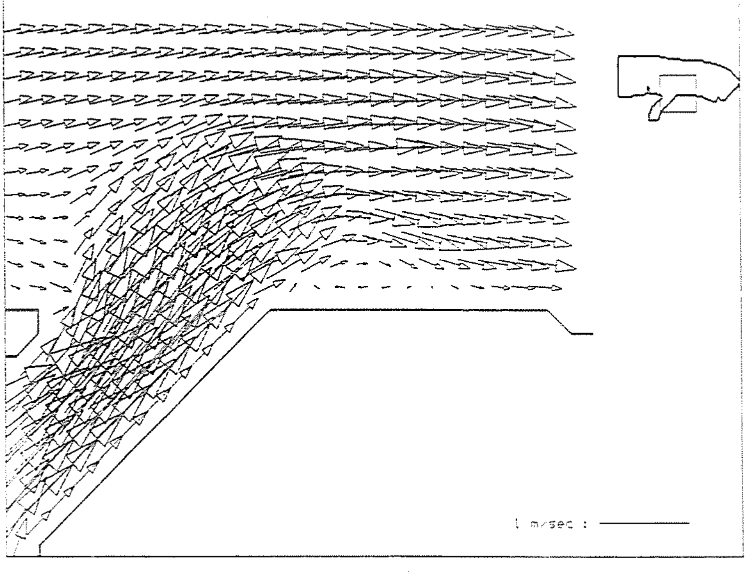

Fig. 9. Calculated depth-averaged velocity field at Vén-Duna confluence with groin removal. Discharge in the main channel: 867 m^3/s. Discharge in the secondary branch: 200 m^3/s

fect. Although the water exchange of the secondary branch would be very efficient, this may be unacceptable from navigational point of view.

In order to reduce the intrusion rate of the side discharge 200 m^3/s as well as the extent of the recirculation zone, a number of variants with the removal of the above mentioned groin were evaluated. In *Fig. 6* the flow pattern in the existing conditions, visualised by randomly seeded particle trajectories, can be seen. Comparing it to the situation without the groin, displayed in *Fig. 7*, the favorable modification of the flow condition can be clearly seen all over the downstream region. Zooming in on the recirculation zone, the fine structure of the velocity field is shown in *Fig. 8* and *Fig. 9*.

4. Conclusions

The application of a flexible, relatively inexpensive depth-averaged numerical flow model in the feasibility phase of the revitalisation project of the secondary river branches in Gemenc Protected Landscape Area was proven efficient. The calculated flow patterns gave an insight into the primary effect of the modification made in the hydraulic and/or topographic conditions of the study areas. Based on this information, the most promising variants can be selected and investigated more in details by means of much more costly scale models or three-dimensional numerical models.

In the present applications the river bed was considered fixed, although it is in most cases mobile due to suspended sediment and bed-load transport. To investigate the combined fluid-sediment fluvial system, sediment transport models have to be coupled with the existing flow model. In the study areas depth-averaged sediment models may be appropriate, however, at regions with strong secondary currents three-dimensional flow and transport models would certainly provide more realistic information.

Simultaneously with model developments, carrying out in situ flow velocity measurements and tracer experiments is of great importance to explore more in details the fluvial processes, furthermore, to collect data for model calibration and verification.

5. Notations

5.1 Greek Letters

η	– free surface water level	m
ν	– kinematic viscosity coefficient of water	m^2/s
ν_e	– effective viscosity coefficient	m^2/s

ρ	– water density	kg/m^3
τ_{bx}, τ_{by}	– horizontal bottom shear stress components	N/m^2

5.2 Latin Letters

g	– acceleration due to gravity	m/s^2
h	– water depth	m
k	– Manning-type smoothness coefficient	$m^{1/3}/s$
P	– pressure	N/m^2
p, q	– horizontal fluxes	m^2/s
Q	– discharge	m^3/s
t	– time	s
u, v	– horizontal velocity components	m/s
U, V	– depth-averaged horizontal velocity components	m/s
$\mathbf{v}$	– spatial velocity vector	m/s
x, y	– horizontal Cartesian coordinates	m
z	– vertical Cartesian coordinate	m
z_0	– bottom level	m

References

BRANDT, A. (1984): Multigrid Techniques: Guide with Applications to Fluid Dynamics. *GMD-Studien,* Nr. 85, Bonn.

CONNOR, J. J. – BREBBIA, C. A. (1976): Finite Element Techniques for Fluid Flow. Newnes-Butterworths.

FLOKSTRA, C. (1977): The Closure Problem for Depth-Averaged Two-Dimensional Flow. *Proc. XVII. IAHR Congress*, Baden-Baden.

GÁSPÁR, C. – JÓZSA, J. (1992): Kétdimenziós áramlási modell a dömösi gázlót és a nagymarosi építési területet magában foglaló Duna-szakaszra (Two-dimensional flow model for the Danube stretch including Dömös shallows and Nagymaros coffer dam). *VITUKI Report*, No. 7612/2/2164, Budapest. (In Hungarian).

HÄUSER, J. – PAAP, H. G. – EPPEL, D. – MUELLER, A. (1985): Solution of the Shallow Water Equations for Complex Flow Domains via Boundary-fitted Coordinates. *Int. J. Num. Meth. in Fluids*, Vol. 5.

JÓZSA, J. – GÁSPÁR, C. – SZÉL, S. – BAKONYI, P. (1992): Folyószabályozási művek és folyóba épített műtárgyak áramlási hatásának vizsgálata numerikus modellezéssel (Investigation of the Hydraulic Effect of River Training Works and Hydraulic Structures by means of Numerical Modelling). *Proc. X. National Congress of the Hungarian Hydrological Society*, Szeged. (In Hungarian).

MARCHAND, M. (ed.) (1993): Flood Plain Rehabilitation Gemenc. Final Report, Delft Hydraulics – RIZA – VITUKI.

MCGUIRK, J. J. – RODI, W. (1978): A Depth-Averaged Mathematical Model for the Near Field of Side Discharges into Open Channel Flow. *J. Fluid Mech.*, Vol. 86.

PATANKAR, S. V. (1980): Numerical Heat Transfer and Fluid Flow. Hemisphere Publishing Corporation.

PAVLOVIC, R. N. – KAPOR, R. – DJURIC, M. (1984): Application of the Body Fitted Coordinates for the Calculation of Elliptic Flows. *Proc. Hydrosoft'84*, Portoroz, Yugoslavia, Elsevier, Amsterdam.

RODI, W. (1980): Turbulence Models and their Application in Hydraulics. A State-of-the-art Review, IAHR Book Publication, Delft.

SIVALOGANATHAN, S. – SHAW, G. J. (1988): A Multigrid Method for Recirculating Flows. *Int. J. Num. Meth. in Fluids*, Vol. 8.

STÜBEN, K. – TROTTENBERG, U. (1984): Multigrid Methods: Fundamental Algorithms, Model Problem Analysis and Applications. *GMB-Studien*, Nr. 96, Birlinghoven.

ZSUFFA, I. (ed.) (1993): A Vén-Duna és a Nyéki-Holt-Duna vízforgalmának természetvédelmi rekonstrukciója (Landscape Protection Reinstatement of Water Exchange Conditions in Vén-Duna and Nyéki-Holt-Duna Secondary Branches). Feasibility Study, PMMF, Baja. (In Hungarian).