Software for modeling spatial and temporal evolution of river-type systems

Galina MARUSIC[1], Ion SANDU[2], Vasile MORARU[1],

Violeta VASILACHE[3], Ana-Monica CREȚU[3], Constantin FILOTE[3], Calin CIUFUDEAN[3]

Technical University of Moldova[1], Stefan cel Mare Blvd., nr.168, MD-2004, Chisinau, Republic of Moldova
„Al.I. Cuza" Univeristy of Iaşi [2], ARHEOINVEST Interdisciplinary Platform, Blvd. Carol I, nr. 22, Corp G Demisol ,700506, Iasi, Romania
"Stefan cel Mare" University of Suceava[3], str.Universitatii nr.13, RO-720229 Suceava
galinamarusic@yahoo.com, sandu_io3@yahoo.com, moraru@mail.utm.md,violetaren@yahoo.com, cretu_monika@yahoo.com, filote.constantin@gmail.com, ciufudean.calin@gmail.com

Abstract - **The paper discusses the problem of dynamic modeling of river - type systems using software packages. iThink, Simulink, Surface-water Modeling System (SMS) are analyzed. The research results are applied to modeling hydrodynamics and pollutants transport on a sector of the river Prut in the Ungheni town.**

Index Terms **- continuity equation, Navier-Stokes equations, two-dimensional dispersion of pollutants, software packages, Surface-water Modeling System (SMS)**

I. INTRODUCTION

The modeling of spatial-temporal evolution of river-type systems has a very important issue for determining and predicting water quality. Using the software significantly increases awareness of the attributes that determine water quality parameters [1].

Transformations that occur in an aquifer system can be described using equations system, which consists of the equations of motion of the fluid's incompressible Navier-Stokes, the continuity equation, the vector form which is as follows:

$$\frac{\partial \vec{v}}{\partial t} = -(\vec{v}\cdot\nabla)\vec{v} + \nu\Delta\vec{v} - \frac{1}{\rho}\nabla p + \vec{f}$$

$$\nabla\cdot\vec{v} = 0$$

where ∇ - Hamilton operator, Δ - Laplace operator, t - time, ν - the coefficient of viscosity, ρ - density, p - pressure, $\vec{v} = (v^1,\ldots,v^n)$ - the speed vector field, $\vec{f}$ - the field vector of mass forces [2]

Mathematical models can be solved using several dynamic modeling software packages, eg *Extend*, *iThink*, *Simulink*, which presents a graphical flow chart and allows the modeler to assemble a flow diagram of the modeled system with icons. The icons are assembled by the modeler in an orderly way for building the mathematical model. The active flow diagram can be simulated based on the mathematical model system.

II. A SHORT DESCRIPTION OF THE SOFTWARE PACKAGES

Extend package [3] has a graphical flow, can work with discrete or continuous variables, linear and nonlinear functions. It contains a library program as icons, which can have one or more inputs, perform calculations and produce an output. The model is built in the form of a diagram block, icons interconnect in an orderly way.

Key features of the package are used to simulate dynamic animation, customizable GUI, sensitivity analysis and optimization. Automatic dimensional consistency is not maintained.

iThink [3] also has a graphical flow. Only four basic blocks are used: stocks, flows, converters and connectors. Stock blocks are represented by rectangles in the diagram and flow values are static accumulators to keep records in any moment of time. Converters can receive one or more inputs and generate output from making a calculation. The calculation is included in the dialog box in the form of an algebraic equation. *IThink* package includes animation, tracking, customizable GUI, sensitivity analysis and optimization features. Automatic dimensional consistency is not maintained.

Simulink [3] is another flow diagram based on a simulation package for modeling dynamic systems. It is based on mathematical equations MATLAB package. It supports linear and nonlinear systems modeling in continuous time or discrete or is a hybrid of the two.

The essential differences between the packages listed are:

- *IThink* package is more economical in terms of number of icons needed for a simulation model;
- Converters of *iThink* can accept any number of inputs, while the corresponding icons and Simulink Extend accept a limited number of input parameters [3]

The benefits of reviewed packages are: providing a graphical interface flow simulation, animation, sensitivity analysis, while the disadvantages are: automatic dimensional consistency is not maintained and does not manage the whole process of modeling river-type systems.

It is very appropiate to use *Surface-water Modeling System* to solve water quality problems (SMS), which was designed by the specialists of Aquaveo USA company. SMS is a software package for efficient management of the entire process of modeling surface water: from importing data and topographic and hydrodynamic import up to visualizing and analyzing solutions. It contains a wide range of numerical models for hydrodynamic applications ncluding rivers, rural and urban flooding, wave modeling, tracking dynamics and physical properties of water particles, determination and analysis of pollutant dispersion [4].

II. CASE STUDY - APPLYING SMS TO DETERMINE THE HYDRODYNAMICS AND POLLUTANT DISPERSION IN A SECTOR OF THE RIVER PRUT IN UNGHENI TOWN

The numerical modeling of the hydrodynamics and bi-dimensional pollutant dispersion was performed using the system *Surface-water Modeling System* (SMS), which calculates the solution of Navier-Stokes equation system as Reynolds [4].

It is necessary to know the hydrodynamics of the studied sector for determining the dispersion of the chemical pollutants [5]. Therefore, the first step is to determine the evolution of the field speed and changes in depths in all the studied sectors of the river using the SMS software package RMA2 , then using the resulting hydrodynamic RMA2, the second stage is to evolve the field of concentrations using RMA4 module.

The digital image of the Prut River in the town Ungheni was imported into SMS from www.wikimapia.org , which was digitized directly in the SMS through manual creation of objects with parameters such as points, arcs and polygons. On each arc, segments, there were created equal segments by redistributing points at a distance.

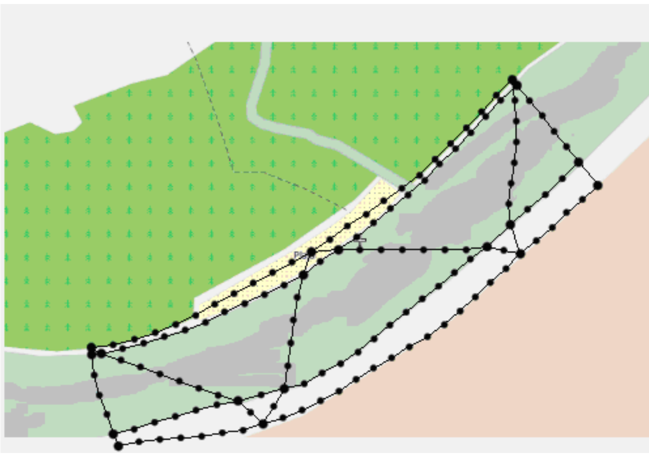

Fig 1. 2D representation of the domain geometry

There were determined three specific areas: the river, left bank and right bank (Figure 2)

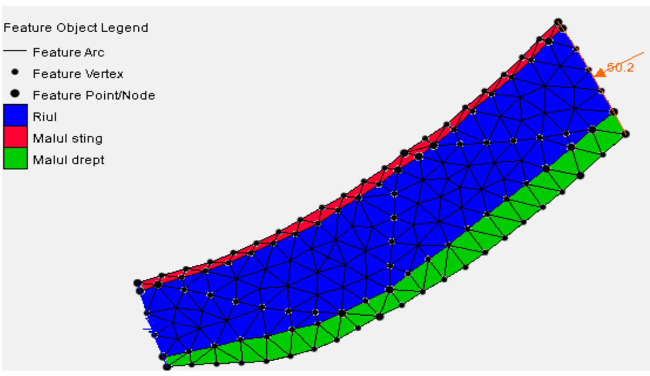

Fig.2. 2D profile obtained in SMS

The first phase – Hydrodynamics determination

The hydrodynamics of rivers is a turbulent flow, which is characterized by variation in time and space of local velocities, irregular trajectory and power lines intersecting the water particles [6]. Physicist O. Reynolds, who demonstrated the existance of turbulent flow regime, proposed to add to the system of Navier-Stokes Equation additional terms that are more uniform stress due to turbulence [6]. For water flow modeling on the Prut River, the system of Navier-Stokes equations as Reynolds (1) and (2) were used together with the continuity equation (3), which completely describe the dynamics of water in rivers in the turbulent regime :

$$h\frac{\partial u}{\partial t}+hu\frac{\partial u}{\partial x}+hv\frac{\partial u}{\partial y}-\frac{h}{\rho}\left(E_{xx}\frac{\partial^2 u}{\partial x^2}+E_{xy}\frac{\partial^2 u}{\partial y^2}\right)+gh\left(\frac{\partial H}{\partial x}+\frac{\partial h}{\partial x}\right)+$$
$$+\frac{gun^2}{\left(h^{1/6}\right)^2}\times\left(u^2+v^2\right)^{1/2}-\varsigma V_a^2\sin\psi+2h\omega v\sin\phi=0 \qquad (1)$$

$$h\frac{\partial v}{\partial t}+hu\frac{\partial v}{\partial x}+hv\frac{\partial v}{\partial y}-\frac{h}{\rho}\left(E_{yx}\frac{\partial^2 v}{\partial x^2}+E_{yy}\frac{\partial^2 v}{\partial y^2}\right)+gh\left(\frac{\partial H}{\partial y}+\frac{\partial h}{\partial y}\right)+$$
$$+\frac{gvn^2}{\left(h^{1/6}\right)^2}\times\left(u^2+v^2\right)^{1/2}-\varsigma V_a^2\sin\omega+2h\omega v\sin\phi=0 \qquad (2)$$

$$\frac{\partial h}{\partial t}+h\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}\right)+u\frac{\partial h}{\partial x}+v\frac{\partial h}{\partial y}=0 \qquad (3)$$

where h - water depth (m), u - local velocity in the x direction (m/s), v - speed local y direction (m/s), t - time (s), ρ - density of water (kg/m^3), g - gravitational acceleration (m/s^2), E - coefficients of turbulent viscosity (Pa.s or kg / m/s), which is calculated automatically, using the Peclet number

$$Pe=\frac{\rho U dx}{E}$$

where $U=\sqrt{u^2+v^2}$ is the velocity, and dx - length element in the direction of flow.

In (1) and (2) H is a geodesic rate (m), n - Manning's roughness coefficient, ζ - empirical coefficient on friction with air, Va - wind speed (m/s), ψ – wind direction (degrees counterclockwise from the positive x-axis), ω - angular velocity of rotation of the Earth (rad/s), ϕ - latitude of the place [7, 8].

The numerical model of water flow was developed using RMA2 program that uses Navier-Stokes equations of Reynolds after the Cartesian coordinates x and y by (1), (2) together with the continuity equation (3) for incompressible fluids in turbulent motion with free surface. The system of equations solution was determined using the finite element method. The interpolation functions are quadratic for speeds and linear for depths [6,8].

The turbulence effects are taken into account by using the coefficients of turbulent viscosity, which is also a means of ensuring the numerical stability of the solution.

The program determines the average local field vertical velocity u and v, and depth h.

Because the water particles velocity depends on the flow and water level the following boundary conditions were established:

- for the arc group at the inflow (top) cross sections, the constant flowrate Q = 50,2 m^3/s was assigned;
- for the arc group at the outflow cross sections the constant geodetic rate H = 4,6 m was assigned.

For the modeling were used:

- simulations for the real river section of length 700 m and width 76-65 m;
- constant water density (1,000 kg/m^3);
- Peclet number equal to 20. The recommended values are between 15-40.

There were determined in all the studied domain finite geometry depths h and local velocities in the directions *x* and *y* (including resultant speed *U*). Results are presented in Figures 3, 4, 5 and 6.

The image in Figure 3 presents different colors, which forms a series of values of the depths of the points of the river studied.

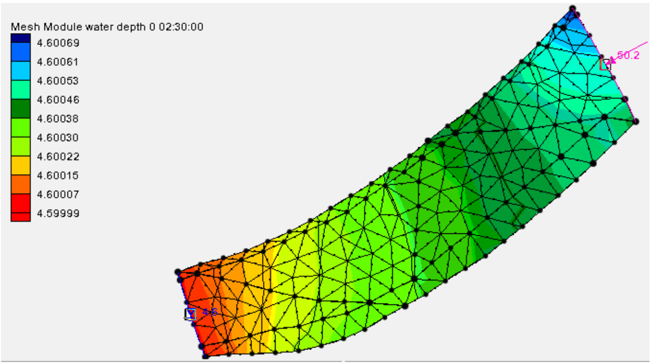

Figure 3. Variation in the depths of the sector studied

The resulting velocity field is shown in Figure 4. From the analysis it was found that the velocity values range from 0.0492 m/s up to 0.1482 m/s.

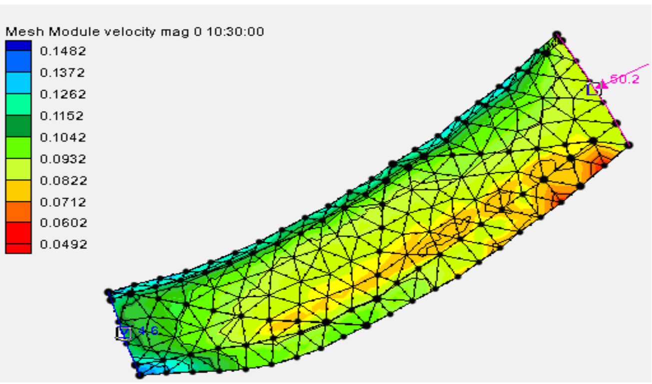

Figure 4. Distribution of the resulting velocity vectors

Phase II – Development determination of the field of the concentrations of pollutants

Dispersion of pollutants is the result of a simultaneous molecular diffusion phenomenon polluting substance with convection - advection, because of a speed field when the pollution occurs. Pollutant concentration in water is influenced by the environment porous nature, flow regime (the velocity field) and the nature of pollutant [9]

To determine the pollutant dispersion the advection-dispersion equation (ADE) is used, which is a partial differential equation obtained by applying mass balance to a unit volume of mass in the river. Pollutant dispersion model developed in this work is based on two-dimensional form of the ADE, applied to the turbulent flow regime [10]:

$$h\left(\frac{\partial c}{\partial t}+u\frac{\partial c}{\partial x}+v\frac{\partial c}{\partial y}-\frac{\partial}{\partial x}D_x\frac{\partial c}{\partial x}-\frac{\partial}{\partial y}D_y\frac{\partial c}{\partial y}-\sigma+kc+\frac{R(c)}{h}\right)=0$$

where h - water depth (m), c - concentration of pollutant (mg/l), t - time (s), u - velocity in the x direction (m/s), v - velocity in y direction (m/s), and D - coefficients of turbulent diffusion in the x and y, k - decay constant (s-1), σ - the term local source of pollutant (unit of concentration/s), *R(c)* - precipitation/evaporation (unit measure of concentration × m/s).

To determine the turbulent diffusion coefficients two methods are used: direct, in which each element receives the respective values of these coefficients, or automatically, using the Peclet number, which is given by:

$$Pe=\frac{Udx}{D}$$

where U - the mean velocity, dx - element length in the direction of flow, D - turbulent diffusion coefficient [11].

Samples of copper compounds were used for modeling, which were taken in August 2011 from two points of the river sector analyzed:

1. c_1 = 0,016 mg/l. 2. c_2 = 0,003 mg/l.

In August vine plantations near the river Prut were treated with copper sulphate. Because of heavy rainfalls, amounts of copper sulphate reached the river. As a result, the concentration of copper compounds in the river significantly exceeded the maximum allowable concentration value. In some places it has been exceeded by about 16 times.

Pollutant dispersion analysis was performed in dynamic conditions, which allows estimation of pollutant evolution at different time intervals.

Field evolution in concentrations of copper compounds in space and time is shown in Figures 5,6,7,8 and 9. Numerical simulation was performed for 12 hours.

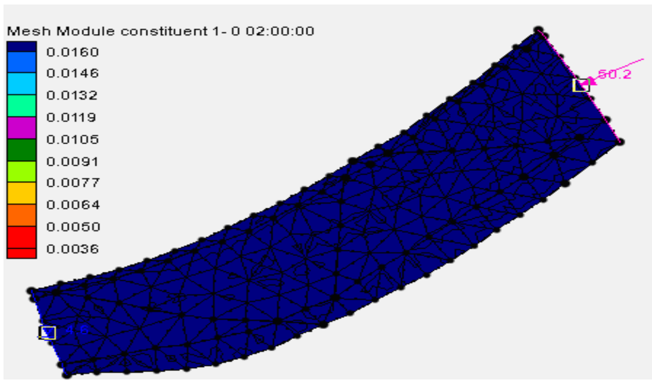

Fig.5. Pollutant dispersion after 2 hours

After 2 hours of the confluence of water, the maximum concentration of the pollutant remains.

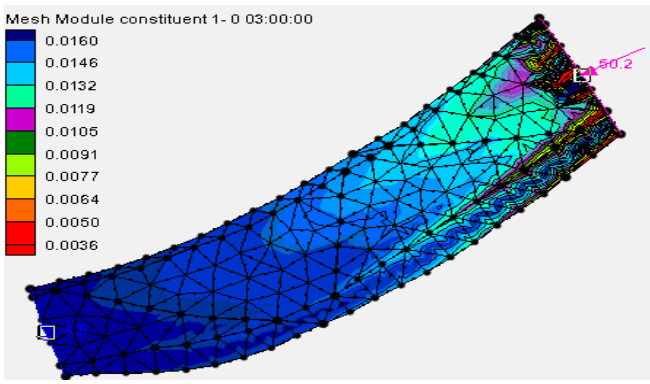

Fig.6. Field distribution of pollutant concentrations after 3 hours

After 3 hours the pollutant concentration decreased on parts of the sector studied.

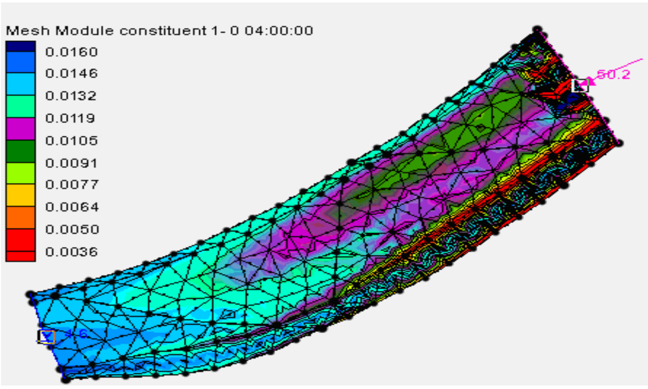

Fig.7. Field distribution of pollutant concentrations after 4 hours

The data presented shows that after 4 hours the pollutant concentration in some sectors has declined substantially.

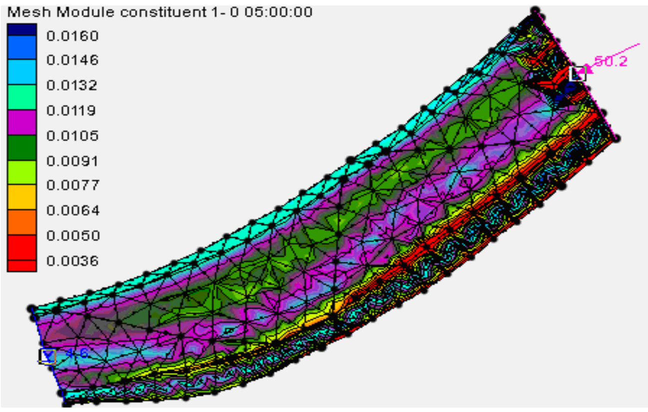

Fig.8. Field distribution of pollutant concentrations after 5 hours

The pollutant concentration decreased throughout the studied area after 5:00, the highest concentration of the pollutant while keeping to the sides.

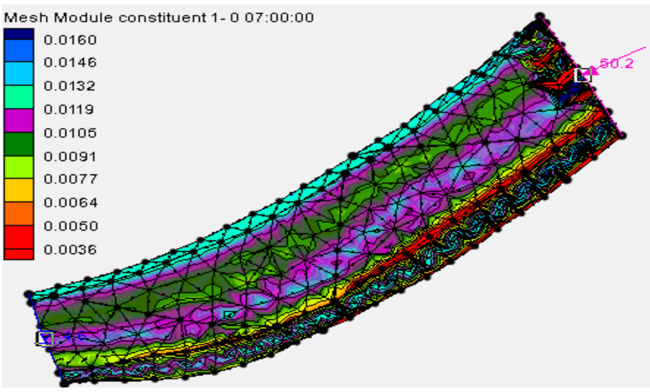

Fig.10. Distribution of pollutant concentration field after 7 hours

After 7 hours a steady transport of pollutants has been set.

The variation of depth and pollutant dispersion in the area of confluence are presented in Figures 11 and 12.

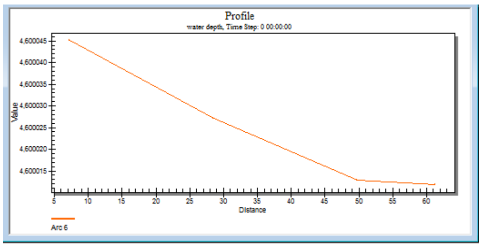

Fig.11. Depth in junction area

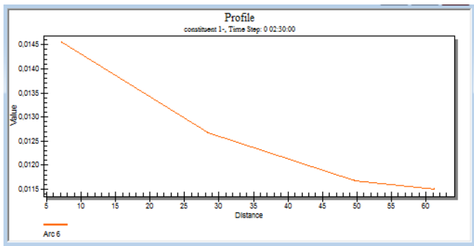

Fig.12. Dispersion of copper compounds in the area of confluence

III. CONCLUSION

A study on software packages used for dynamic modeling of river-type systems has been established. The advantages and disadvantages were determined.

The software Surface-water Modeling System was applied for determining the spatio-temporal evolution of a sector of the river Prut in the Ungheni town.

There were determined the hydrodynamics and field concentrations of the pollutant in time and space in any finite element of the sector considered.

The results are useful to accurately assess the degree of water quality and water quality parameters for optimization.

REFERENCES

[1] R.S. Lozba-Stirbuleac, C.R. Giurma-Handley, I. Giurma, Water Quality Characterization Of The Prut River, ENVIRONMENTAL ENGINEERING AND MANAGEMENT JOURNAL, 10, 3, 411-419, MAR 2011

[2] Iu. Florescu, Fluids mechanics, Bacau University press, 2007

[3] N. Nirmala Khandan, Modeling Tools for Environmental Engineers and Scientists, CRC Press, 2001

[4] V.Beşliu, C.Ciufudean, C.Filote, G.Marusic, V.Moraru, B.Ştefănescu, Mathematical modeling of hydrodynamics and pollutant dispersion in rivers, Proceedings of the 7th International Conference on Microelectronics and Computer Science ICMCS – 2011. Chişinău: The Technical University of Moldova, vol.I, p. 160 - 165, 2011

[5] K. Modenesi, L.T. Furlan, E. Tomaz, A CFD model for pollutant dispersion in rivers, BRAZILIAN JOURNAL OF CHEMICAL ENGINEERING, 21, 4, 557-568, OCT-DEC 2004

[6] C. Leupi, M. Altinakar, Finite element modelling of free-surface flows with non-hydrostatic pressure and k-epsilon turbulence model, INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN FLUIDS, 49, 2, 149-170, SEP 20, 2005

[7] *** „SMS Tutorials", SMS v.10.1.11, AquaVeo, 2011

[8] *** „Surface Water Modeling System - RMA2", US Army Engineer Research and Development Center, 2009

[9] H.W. Fang, G.Q. Wang, Three-dimensional mathematical model of suspended-sediment transport, JOURNAL OF HYDRAULIC ENGINEERING-ASCE, 126, 8, 578-592, AUG 2000

[10] S.A. Socolofsky and G.H. Jirka, Special Topics in Mixing and Transport Processes in the Environment (5th Ed.), A&M University, College Station, Texas, 2005

[11] *** „Surface Water Modeling System - RMA4", US Army Engineer Research and Development Center, 2009