Velocity Profile over Spillway by Finite Volume Solution of Slopping Depth Averaged Flow

SAEED-REZA SABBAGH-YAZDI
Civil Engineering Department,
KN Toosi University of Technology,
No.1346 Valiasr Street, 19697- Tehran, IRAN

NIKOS E. MASTORAKIS
Military Insitutes of University Education (ASEI)
Hellenic Naval Academy
Terma Chatzikyriakou 18539,
Piraues, GREECE

MOHAMMAD ZOUNEMAT-KERMANI
Civil Engineering Department,
KN Toosi University of Technology,
No.1346 Valiasr Street, 19697- Tehran, IRAN

Abstract: A depth averaged numerical model is developed to solve super critical free surface flows in canals with steep slope in certain direction is introduced in the present paper. The model consists of depth averaged equations for flow on surface with steep slope toward one of the main horizontal axes of Cartesian coordinates. The depth-averaged form of the flow equations converted to discrete form for unstructured meshes using the cell vertex finite volume method. Therefore, the horizontal mesh is firstly transformed into inclined coordinate system. Then the discrete formulations are solved on the transformed mesh. Having computed the velocity vector parallel to the bed surface and flow depth perpendicular to the bed surface, the model computes three components of the velocity in Cartesian coordinate system as well as the water depths parallel to the z axe in vertical direction. The developed model is utilized for simulation of various flow regimes in prismatic canals, which have two different constant bed slopes in certain direction. The numerical results present the ability of the model to simulate very high speed super-critical flows in very steep bed slopes. Finally, application of the developed model on simulation of sub and super critical flow from reservoir to steep chutes and ski jump flip bucket spillways of SEYMAREH in KHOSESTAN province of IRAN produced encouraging results by of computation of three dimensional velocity patterns and shock waves.

Key words: Computing Velocity Profile, Numerical Simulation, Depth Averaged Flow, Steep Slope Chute Spillway.

1 Introductions

The chute spillways mostly have steep slopes in a certain direction. In the chute spillways, the velocity field of the super critical flow forms parallel to the bed surface. Although the negligible velocity component normal to channel bed validates the hydrostatic pressure distribution in flow depth, the set of common shallow water equations (SWE) is not suitable for simulating most of the real world spillway flow cases because of mild bed slope assumption for derivation of this mathematical model. Therefore, casting these equations for steep slopes may help overcoming the problem. Such a modified set of shallow water equations can be developed by rewriting the depth averaged equation in a slopping frame of reference using flow depth normal to the bed surface and two velocity components parallel to the bed surface.

In the present work, a finite volume model suitable for the triangular unstructured mesh is used for solving the mathematical model. Proper artificial viscosity terms are added to the formulations to stabilize the explicit solution procedure in the convection-dominated flow regions. Formulations of these additional terms are designed for solving on unstructured meshes in such a way that not only damp out unwanted numerical oscillations but also preserve the accuracy of the solution. In addition, proper numerical techniques are adopted for increasing the efficiency of the computation on unstructured meshes.

Having solved the depth averaged velocity field parallel to the bed surface, the velocity profile in flow depth can be computed using empirical relations suggested for super critical flow chutes.

2 Literature Review

Among the various mathematical models, the set of shallow water equations (SWE) for simulating various types of flow regimes has been received great attentions for modeling super-critical flows.

Hence, many numerical workers focused on solving SWE for super-critical flow [1,2,3,4]. For example Jimenez et al were simulated super-critical flows using SWE (Shallow Water Equations) on structured grids. In their model, finite difference scheme was utilized considering mild channel slope and bed friction effects [5].

Some numerical workers used shallow water equations as a mathematical model for simulation of spillway flows. For example, Unami et al, used SWE for flow over a spillway on unstructured triangular domain. They utilized finite element and finite difference methods for discretizing the governing equations [6].

3 Mathematical Model

The shallow water equations have a wide application for solving many types of two-dimensional flow problems. The main assumption for using shallow water equations is hydrostatic distribution of pressure, which means there is no significant velocity component in vertical direction. The standard shallow water equations assume the channel bed is horizontal or its slope is mild. Such a depth average mathematical model best suits the flow characteristics of super critical flow on a mild slope [7].

In this work, a version of shallow water equations is introduced for two dimensional depth averaged flow on an inclined surface with assumptions hydrostatic distribution of pressure figure 1. This assumption implies negligible velocity component in normal to the flow plane. In the model, common assumptions for practical open channel water flows, like incompressibility, negligible wind stresses and earth rotation effects are considered as well.

The mathematical model is cast for solving supercritical flow in a chute canal with a slope α in a certain direction (in which the bottom elevation variation plays important role in forming flow patterns). In this model the set of depth averaged equations is modified to describe super-critical flow in a coordinate system with an axe normal and two axes of x* ($x^* = x / Cos\alpha$) and y parallel to the bed surface.

The set of governing equations contains an equation of continuity and two equations of motion in *y* and *x** directions as follow.

$$\frac{\partial}{\partial t}\left(h^* u^*\right) + \frac{\partial}{\partial x^*}\left(u^* h^* u^*\right) + \frac{\partial}{\partial y}\left(v h^* u^*\right) + \frac{\partial}{\partial x^*}\left(h^* \frac{g h^* / Cos\alpha}{2}\right) = -\frac{\tau_x^*}{\rho} - g h^* \frac{\partial Z_b}{\partial x^*} \quad \text{(1-a)}$$

$$\frac{\partial}{\partial t}\left(h^* v\right) + \frac{\partial}{\partial x^*}\left(u^* h^* v\right) + \frac{\partial}{\partial y}\left(v h^* v\right) + \frac{\partial}{\partial y}\left(h^* \frac{g h^* / Cos\alpha}{2}\right) = -\frac{\tau_y^*}{\rho} \quad \text{(1-b)}$$

Where, t is time, $u*$ is velocity component along x* (in x-z plane) which has α angle x ($u^* = u / Cos\alpha$), while v is the velocity component along y and h* is the flow depth normal to the bed surface ($h^* = hCos\alpha$). Here, Zb is bed elevation, g is the gravity acceleration. The global dissipative forces can be defined as,

$$-\frac{\tau_x^*}{\rho} = C_f u^* |U| \quad , \quad -\frac{\tau_y^*}{\rho} = C_f v |U| \qquad (2)$$

$$U = \sqrt{u^{*2} + v^2}$$

Where, Cf represents the effective global dissipative coefficient0.

The effects of gravitational force to the water body in x/ direction is introduced using the local bed slope (bed elevation gradient) in terms of: $W \ Sin\alpha = gh^* \, \partial Z_b / \partial x^*$, in the above equations. This term plays an important role in formation of flow patterns.

Using $\eta = Z_b + h^* / Cos\alpha$, the above mentioned depth averaged equations of motion is reduced to the following from [11].

$$\frac{\partial}{\partial t}\left(h^* u^*\right) + \frac{\partial}{\partial x^*}\left(h^* u^* u^*\right) + \frac{\partial}{\partial y}\left(h^* v u^*\right) + gh^* \frac{\partial}{\partial x^*}(\eta) = -\frac{\tau_x^*}{\rho} \qquad (3\text{-a})$$

$$\frac{\partial}{\partial t}\left(h^* v\right) + \frac{\partial}{\partial x^*}\left(h^* u^* v\right) + \frac{\partial}{\partial y}\left(h^* v v\right) + gh^* \frac{\partial}{\partial y}(\eta) = -\frac{\tau_y^*}{\rho} \qquad (3\text{-b})$$

Having computed the velocity vector parallel to the bed surface and flow depth perpendicular to the bed surface numerically, three components of the velocity in Cartesian coordinate system as well as the water depths parallel to the z axe in vertical direction can be computed as ,

$$h^* = h / Cos\alpha \quad , \quad u = u^* Cos\alpha \quad , \quad v = v \quad ,$$
$$w = u^* Sin\alpha \qquad (4)$$

Note that, by omitting the superscripts * from the above equations of continuity and motion, will make them similar to the standard form of shallow water equations (SWE), and hence similar numerical procedure can be applied for the solution [8]. Therefore, similar equations can be allied for the sub-critical parts of the solution domain (in which the water surface elevation variation plays important role in forming flow patterns), by considering; $Cos\alpha = 1$ and $Sin\alpha = 0$.

4 Numerical Solution Algorithm

The cell vertex finite volume method is applied for discretization of governing equations. Hence, the above partial differential equations can be solved on triangular unstructured meshes, which generated applying Delaunay triangulation method in a coupled manner using the algorithm developed for the compressible flow problems [9].

In this method, the domain is divided into triangular sub-domains (control volumes), which is formed by triangles meeting every computational node, and then the governing equations are integrated over each sub-domain. (3) The equations of continuity and the motions are integrated over each control volume. Application of the Green's theorem to the integrated equation in general form result is:

$$\int_{\Omega} \frac{\partial W}{\partial t} d\Omega + \oint_{\Gamma} \left(Fdy - Gdx\right) = \int_{\Omega} S \, d\Omega \qquad (5)$$

Where Ω_i and Γ are the area and boundaries of the control volume, respectively. W represents time dependent terms of above equations while, F and G represent x and y fluxes, respectively. S is the sink/source term of the equation. Its value equals to zero for the continuity equation and equal to global forces for two equations of motion. If nodal values of dependent variables at each triangle vertex are taken as the unknowns at the central node of the control volume, the discrete explicit form of the equation is evaluated by conversion of the boundary integral into the summation over m edges of the control volume, as

$$W^{n+1} = W^n - \frac{\Delta t^n}{\Omega_i}\left[\sum_{j=1}^{m}(\bar{F}\Delta y - \bar{G}\Delta x)\right]^n + S^n \quad (6)$$

Where W^{n+1} is the value of W^n to be computed after Δt. The parameters $\bar{F}$ and $\bar{G}$ are the average values of the fluxes in x and y spatial derivatives in each edge at the boundary edges of the control volume [10]. Note that, Δx and Δy should be computed using transformed coordinate in x* direction in sloping supercritical parts of the solution domain.

Explicit solution of the convective equations where the global dissipative terms are negligible, some numerical oscillations grow particularly near the high gradient regions. These numerical noises disturb the solution procedure in the cases with small physical dissipation mechanisms. For the flow problems with gradual changes in dependent variables (flows with no shock waves), the fourth order term (Biharmonic operator) produces enough dissipations to damp out the numerical oscillations and stabilize the explicit solution procedure [9].

Time marching of the explicit computations (Δt) should be proportional to the speed of wave propagation of the applied convective equations. This speed can be computed using, λ the maximum Eigen values of Jacobin matrix of homogonous form of the set of governing equations. Since we are dealing with unstructured meshes, the size of control volumes varies over the computational domain. Therefore, every control volume has its own time step, Δt_i. Hence, the speed of explicit computations is limited to the minimum $(\Delta t)_{\min}$ in the unsteady flow field. Although the values of the time step, Δt_i for every control volume vary during the stages of the numerical solution, it may approach to certain values when the computations converge to the steady state conditions.

5 Imposing Velocity Profile

Several logarithmic relations could be applied to obtain velocity distribution in flow depth. Prandtl-Von Karman is known as one of the most important relations for attaining this purpose, as [12].

$$v = 2.5V^* \ln\frac{y}{y_0} \quad (7)$$

In the above relation v is flow velocity in y depth and v* stands for shear velocity in which can be calculated from the following relation:

$$V^* = \sqrt{gRS} = \sqrt{\frac{\tau_0}{\rho}} \quad (8)$$

For turbulent flow over smooth surfaces, if the bed surface is hydraulically smooth, y_0 can be obtained using,

$$y_0 = \frac{m\upsilon}{u_*} \quad (9)$$

In the above relation, m is a constant coefficient equal to 1/9 for smooth surfaces. Using above mentioned relation for y_0, velocity magnitude in flow depths can be calculated to the following relation [13]

$$v = 2.5V^* \ln\frac{9yu_*}{\upsilon} \quad (10)$$

For turbulent flow over rough bed, y_0 depends on bed roughness and can be calculated by following relation (11).

$$y_0 = mk_s \quad (11)$$

Here, k_s is the bed surface roughness in terms of equivalent sand roughness and the value for m is suggested as $1/30$. Consequently, the velocity profile could be obtained from the following equations.

$$v = 2.5V^* \ln \frac{30y}{k_s} \tag{12}$$

Another formulation suggested by many researchers is prandtlel $1/n$ power law. The relation can be describe as;

$$\frac{v}{V_{\max}} = \left(\frac{y}{\delta}\right)^{\frac{1}{n}}, \qquad 6 \langle n \langle 11 \tag{13}$$

Where $V_{\max}$ is equivalent of maximum velocity at the outer region (higher than boundary layer) and δ prescribes boundary layer thickness. Since the boundary layer thickness is equal to the flow depth in most of the chute spillway length, the assumption of $\delta = y_{\max}$ is valid for present application. In order to relate the maximum velocity $V_{\max}$ (at water surface $y_{\max}$) the unit discharge in chute could be utilized.

$$q = v_{mean} y_I = \frac{V_I}{y_I^{1/n}} \int_0^{y_{\max}} y^{1/n} dy \tag{14}$$

By integrating the above relation, the relation between depth averaged (mean velocity) v_{mean} and maximum velocity at water surface is [14]:

$$V_{\max} = \frac{n+1}{n} v_{mean} \tag{15}$$

Using above relation, the computed depth average velocities and flow depth can be used for calculation of velocity profile at every point of the flow field.

6 Initial and Boundary Conditions

At the solid boundary nodes slipping condition are applied by imposing zero to normal velocity components to the wall. Having computed tangential velocity at wall nodes, wall resistance may be computed and applied to boundary nodes. The effective surface for wall global friction stresses may be computed multiplying the flow depth by contribution length of two wall boundary edges connected to each boundary node. Various types of boundary conditions are specified in the numerical model for flow and wall boundary conditions figure 2.

Inflow and outflow boundaries can be manually defined or automatically using normal vector and velocity vector at the boundary nodes figure 2. In the present model, different flow conditions at inflow and outflow boundaries are imposed. For sub-critical flows, at the inflow boundary, velocity components are imposed and water depth is extrapolated from the interior points and at the outflow boundary, and velocity components are imposed form the inside domain. For super-critical flows, at the inflow boundary, water depth and velocity components are imposed and at the outflow boundary, all of the variables are extrapolated from the interior points. When mixed flow forms in the channel, the boundary conditions of the flow boundaries have to be defined concerning flow regime at each boundary [15].

Proper initial condition may help accelerating the solution procedure. As can be seen in figure 2, in this work constant water elevation is considered for sub-critical part of the flow domain (where the depth averaged flow equations are to be solved in horizontal coordinate) and constant water depth is considered for super-critical part of the flow domain (where the depth averaged flow equations are to be solved in sloping coordinate).

7 SEYMAREH Spillway Simulation

In order to present the performance of the model to solve the real world engineering problems, the model is applied to simulate flow from the dam reservoir over the ogee spillway and two chute canals ending to the flip bucket of the SEYMAREH project, which is constructed at south west of Iran. Some measurements are available for the experimental laboratory model test data [16].

The unstructured triangular mesh which contains 5871 nodes and 16681 edges generated using Delauney triangulation method is converted to a three dimensional surface by assigning vertical elevation to each grid point [17]. This mesh is similar to that was used for previous numerical simulation by solving standard shallow water equations [15].

Sub-critical boundary condition at upstream (dam reservoir) and super-critical boundary condition at downstream (bucket end) are considered.

The computed water surface elevations and flow depths at the entire computational domain are shown in figure 3. The interesting point is the computation of two inclined shock waves starting from the middle part of the chute (from the point that the thickness of the dividing wall decreases, and hence, the width of the two chute bay decrease) and continue to the buckets. This fact proves that developed model can compute expected shock waves, which are expected to appear due to change in chute width.

The computed velocity vectors and stream lines, which show the flow direction, are plotted in figure 4 and figure 5. As can be seen, these vector present horizontal flow in sub-critical part of the domain (reservoir) and parallel to the bed surface flow in supercritical part of the flow domain (chute and bucket). This fact proves that developed model can compute the velocity patterns of chute spillways in a three dimensional manner. More details about velocity profile (computed on nodal points of the five layers) can be seen in figure 6.

8 Conclusion

Using an inclined coordinate system, the horizontal velocity components are transformed to the velocity components parallel to the bed surface. The depth averaged equations derived for flow in coordinate system mapped on sloping bed surface are very similar to the standard shallow water equations for flow in horizontal plane. Therefore, unstructured finite volume flow solver for solving sub-critical flow problems is easily adapted for the solution of above mentioned developed mathematical model for super-critical flow on steep chute canals. Using this technique, not only horizontal component of the flow velocity are computed correctly, but also vertical velocity component appeared in the simulation results. Consequently the computed velocity vectors are parallel to the bed surface at the entire supercritical part of the flow domain. Combining the developed numerical solver for super critical flow on steep slopes with the standard shallow water equation solver suitable for sub-critical flow, provided the ability of simulating mixed sub and super critical flows. Therefore, numerical simulation of low speed flow in reservoir and high speed super critical slopping flow in steep chute and ski jump flip bucket spillway is performed successfully by computation of three dimensional velocity patterns and shock waves. Having solved the depth averaged velocities parallel to the bed for supper critical flow in chute spillways and water depth, the velocity profile normal to the bed surface at nodal points of the arbitrary layers is computed by application of empirical formulations suggested by the experimental research workers.

9 References

[1] Berger، R. C. & Ltockstill، L., “Finite-Element Model for High Velocity Channels، Journal of Hydraulic Engineering”، Vol. 121، No. 10، (1995) PP. 710-716

[2] Molls، T. & Chaudhry،M. H.,”Depth Averaged Open Channel Flow Model”, Journal of Hydraulic Engineering، Vol.121, No. 6, (1995), PP.453-465

[3] Marchant، M.J. & Weatherill, 1995, “A Streamline Based Approach to the Adaptation of Unstructured Grids for Viscous Flow Simulation”, ICFD Conference، Oxford، U.K.، (1995)

[4] Kruger, S. & Burgisser, M. & Rutschmann,P., , Advances in Calculating Supercritical Flows in Spillway Contractions، Hydroinformatics'98، Babovic & Larson (eds) © 1998 Balkema،

Rotterdam، ISBN 90 5410983, (1998), PP. 163-169
[5] Jimenez، O. F. & Chaudhry، M. H., “Computation of Supercritical Free-Surface Flows”، Journal of Hydraulic Engineering، Vol/ 114، No.4، (1988) PP. 377-395
[6] Unami, K., Kawachi T., Babar M. & Itagaki H., "Two-Dimensional Numerical Model of Spillway Flow", Journal of Hydraulic Engineering, ASCE, Vol. 125, No. 4, (1999), PP. 369-375
[7], Abbot, A.B., “Computational Hydraulics, Pitman Publishing Limited”. (1980)
[8] Vreugdenhil, C.B., "Numerical Methods for Shallow Water Flow", Kluwer Academic Publisher. (1994)
[9] Jameson, A, Schmidt, W. & Turkel E., “Numerical Solution of the Euler Equations by Finite Volume Method Using Runge–Kutta Time Stepping Schemes”, AIAA (1981), Paper, 81-1259
[10] Weatherill, N.P. & Hassan, O. & Marcum D.L., “Calculation of Steady Compressible Flow-Fields with the Finite Elements Method”, 31 Aerospace Sciences Meeting and Exhibit, AIAA-93-0341, (1999)
[11] Sabbagh-Yazdi S.R. , “Spillway Flow Modeling by Finite Volume Solution of Slopping Depth Averaged Equations on Triangular Mesh; Application to KAROUN-4 ", Proceedings of the 10th WSEAS International Confenrence on APPLIED MATHEMATICS, Dallas, Texas, USA, November 1-3, (2006)
[12] Chow، V. T., Open-Channel Hydraulics, Mc.Grow Hill Book, PP.444-448 (1973)
[13] Dake, Jonas M.K., “Essential of Engineering Hydraulic”, The MAC Millan Press Ltd,PP.9, 103, (1983)
[14] Chanson H.,Brattberg T.,”Experimental study of the air-water shear flow in hydraulic jump”, International Journal of Multiphase Flow 26 (2000)
[15] Sabbagh-Yazdi S.R. & Mohamad Zadeh Qomi, M. , , "Using 2D Unstructured Mesh for Numerical Simulation of Free Surface Flow in Meandering Canal", Proceeding of Shallow Flow, TU Delft, Netherlands, (2003)
[16] Water Research Center, “Physical Model of Seymareh Dam” , Project Report, Tehran, Iran (2002)
[17] Thompson Joe F. & Soni, Bharat K. & Weatherill, Nigel P., “Hand book of grid generation”, CRC Press (1999)

Figures:

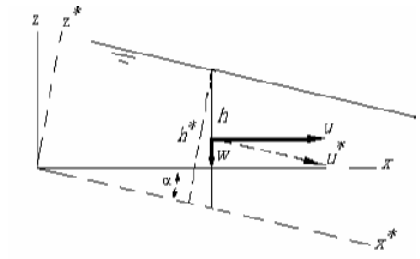


Fig.1, Transformed velocity and depth

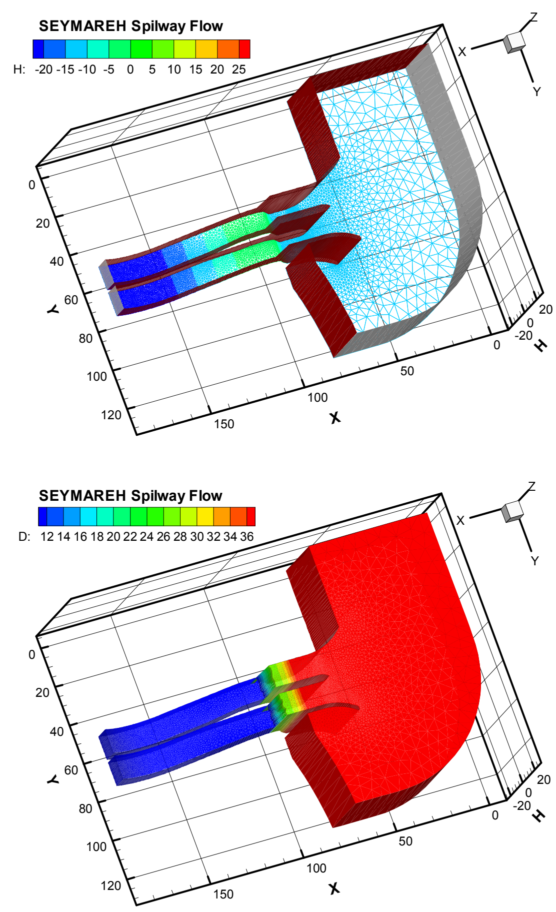
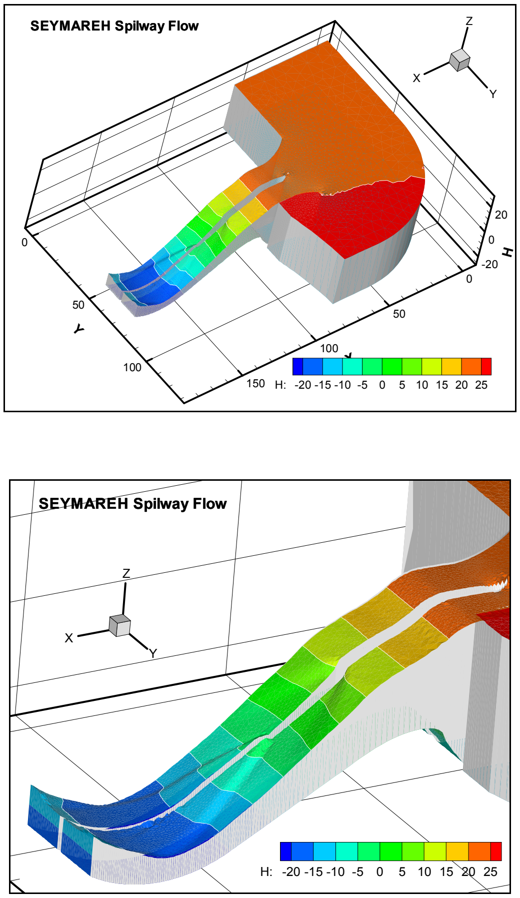


Fig.2, up; boundary conditions (Wall: red, Flow: grey), down; initial conditions (Depth)



Fi.3, computed water surface elevation , general view (up) close view (down)

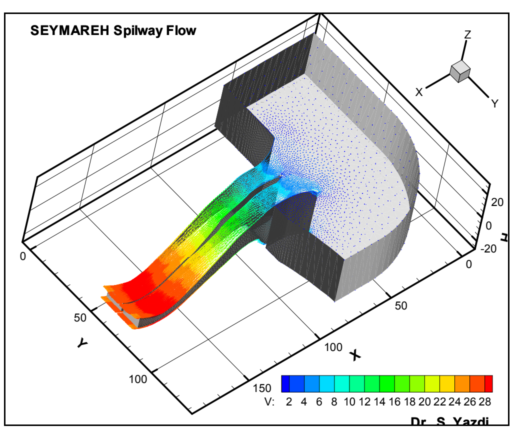


Fig.4, Computed velocity vectors

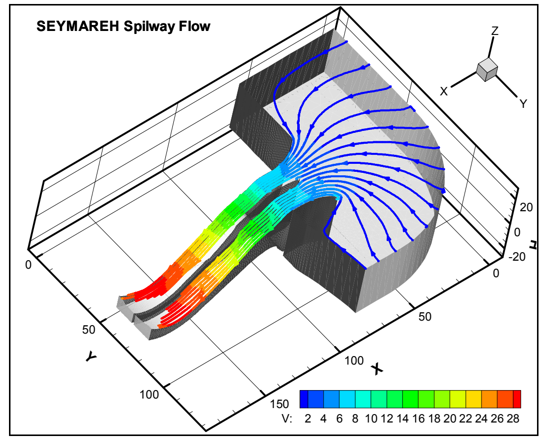


Fig.5, Computed stream lines, general view

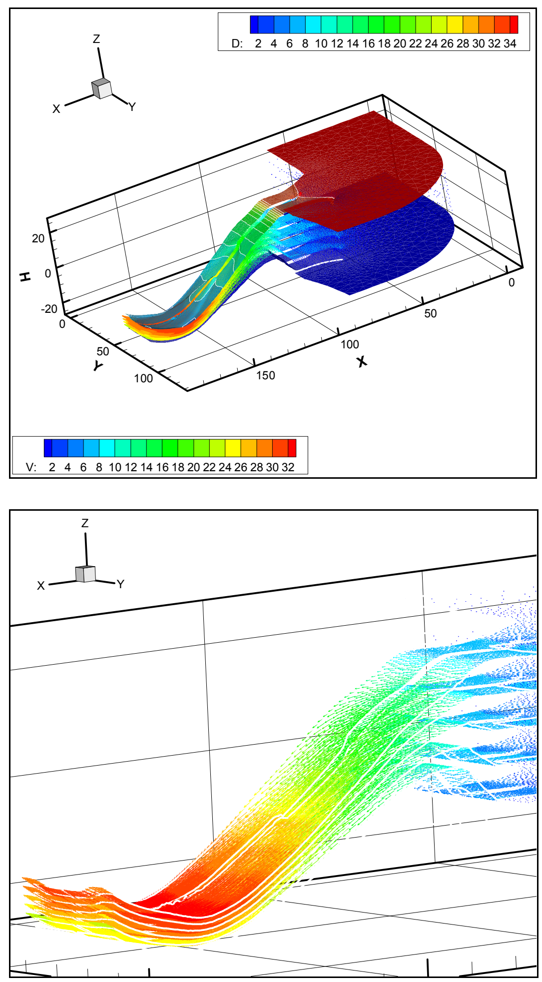


Fig.6, Computed vertical velocity profile, general view (up) close view (down)