STATIONARY LEVEL SET METHOD FOR MODELLING SHARP INTERFACES IN GROUNDWATER FLOW

N. H. Sharif[†]**, N-E Wiberg*[†],

[†]*Department of Structural Mechanics,*
Chalmers University of Technology,
Sven Hultins gata 8, SE-412 96,
Eklandagatan 80,
Gothenburg, Sweden
e-mail: nahidh@sm.chalmers.se

**Finite Element Center,*
Eklandagatan 80, SE-412 96,
Gothenburg, Sweden

Key words: level set method, free surface, finite element method, porous medium, vanishing viscosity method

Abstract

We present a new numerical procedure for modelling steady motion of interfaces in groundwater flow. A modified version of the level set method utilizing a technique from the method of vanishing viscosity is developed and solved together with the time independent Navier-Stokes equation. Simulation for steady non-linear seepage flow through porous media is performed. The discretized-coupled system is solved over a fixed mesh domain. The resulted numerical scheme allows tackling the front motion faster and simply with less computation efforts. The Streamline/Upwind technique is used to stabilize the solved-coupled system of FE-equations over unstructured 2D and 3D FE-meshes. Examples on the application of the technique for evolving sharp interface under a steady state of seepage flow in embankment dam are investigated.

1. Introduction

The level set method was recognized as a simple and versatile technique for computing and modelling motion of interfaces in many areas where multiphase flow are studied [1,8]. The main idea of using this technique for capturing sharp interfaces is to avoid the sequences of remeshing the flow domain and to use the one-phase PDE equation to model two-phase flow [4,5]. These reductions in the number of the needed computation efforts make the level set method competitive with other tracking methods and provide the technique with simple tools to relate unbounded number of phase dependent variables to the interface motion and its position. The level set method is constructed on the base of adding one extra variable and one extra PDE of the hamiltonian-Jacobi type to the governing flow equation [8]. The PDE has characteristics of initial value problems and it is simply derived from the definition of the location of the interface. The location of the interface is defined by the zero level set (at least Lipschitz continuos) of a smooth function $f(\boldsymbol{x}, t)$, i.e. $G_{interface}(t) = \{\overline{\boldsymbol{x}} \in W: f(\overline{\boldsymbol{x}}, t) = 0\}$ where

$\overline{\boldsymbol{x}}$ denotes the geometric position of the interface. The level set function has the following properties

$$f(\overline{\boldsymbol{x}}, t) \;\rightarrow \begin{cases} < 0 \;\text{ for } & \overline{\boldsymbol{x}} \in W_1, \\ = 0 \;\text{ for } & \overline{\boldsymbol{x}} \in G_{interface}, \\ > 0 \;\text{ for } & \overline{\boldsymbol{x}} \in W_2 \end{cases} \tag{1}$$

Where W_1 and W_2 are the domains of the fluid phases. Hence, the interface can be found by capturing the zero level set at later time. The motion of the interface is localized by advecting the level set values of $f(\boldsymbol{x}, t)$ with the velocity field $\boldsymbol{u}$ [6]**.** The advection equation is

$$\frac{\P\, f}{\P\, t} + \boldsymbol{u} \cdot \nabla f \;=\; 0. \qquad\qquad \text{on } W \qquad \forall t \in (0, T) \tag{2}$$

where the velocity is arbitrary elsewhere. Actually this is the main difference between the level set equation and the fast marching method where a stationary boundary value problem is solved to find the arrival time of the interface that advecting by a positive velocity field ($\boldsymbol{u}$ > 0) [8].

Generally, problems with sharp interface usually preserve discontinuous initial profile at the interface or in the vicinity of the interface. The discontinuity characteristic gives arise to numerical instability and reduce the accuracy of the solution for the final location of the interface. Furthermore, if the dependent phase variables are extremely different, the coefficients of the produced element matrix exhibit large jumps, which conserve strong discontinuous characteristics across the interface. However, numerical formulations towards simulating one phase flow with an external interface (Free surface) are well addressed in the literature. The one phase model flow based on the fact that the significant difference between the phase dependent variables make use to neglect the effect of the weaker represented phase e.g. air- water system. However the extend of the location of the external free surface is still to be accounted for explicitly. Hence modification algorithm of the mesh during each iteration and time steps is required, which is not promising technique in case of complex geometry. We refer to [2, 10] for the application of moving mesh technique.

The main issue in this study is to develop a technique that can handle moving boundary problems with fast numerical scheme and to produce accurate sharp interface on a fixed mesh domain. Recently, we simulated 2D and 3D steady seepage flow problems in Rock-fill dams by utilizing the Interface Capturing technique (**ICT**) [6, 7]. The **ICT** is based on the implementation of the Initial Value Problem (**IVB**) of the level set method. The implementation has shown powerful tools to model unbounded number of phases by a unified phase model. Furthermore, problems involve sharp interface motion that accompanied with complicated geometry can simply extend from one-dimensional to two- and three-dimensional flow problems. However, using technique derived from the level set method (**IVB**) stipulate a certain stable numerical scheme and solution algorithm. Recently we addressed a special iterative solver to avoid numerical instability [6, 7]. The main idea was to normalize the **IVB** solution of the front by using an artificial time step then to solve the system by a block like iteration approach. The algorithm used produces a stable and accurate solution, at the same time the convergence order required definitely large number of iterations and time steps to reach the tolerance error.

In this article we have investigated a new solution spectrum. A numerical scheme for a coupled system of the time independent Navier-Stokes equation and the stationary advection equation is formulated. The scheme implies the regularization of the advection equation. The regularization is arisen of a technique derived from the vanishing viscosity method. Then the SUPG- and the PSPG-type weighting function are used to provide accuracy and stability for the solved system. We also incorporated into the new method a sort of sharpening algorithm for the front [6]. Our experience of this technique has shown that the new numerical scheme is very fast and required less number of iteration to reach the preset tolerance and certainly for three-dimensional flow problems.

In Section 2 we review the formulation of the stationary level set method and stability of the solution, The Governing flow equation for porous medium is presented in Section 3. We provide details about the finite element formulations and solution algorithm for the coupled system in Section 4. Numerical examples are presented in Section 5 and we end with the conclusion in Section 6.

2. Stationary version of level set method

The numerical scheme implies the regularization of the advection equation (**IVB**):

$$f_t + \boldsymbol{u} \cdot \nabla f = 0. \qquad \text{in } \Re^n \times t \in (0, T) \tag{3}$$

The theoretical justifications for non-oscillatory discretization of the stationary level set method arise from the theory of artificial viscosity solutions for the **IVB**. The viscosity solution [3] has been applied to a range of Hamilton-Jacobi type PDE, which gives accurate computer simulations and unique solutions [9].

Our approach was to consider the approximation of the Hamilton-Jacobi PDE for $e > 0$:

$$f_t^e + \boldsymbol{u} \cdot \nabla f - e\Delta f^e = 0. \qquad \text{in } \Re^n \times t \in (0, T) \tag{4}$$

The idea is that (3) involves a fully nonlinear first – order PDE, while (4) is an initial – value problem for a quasilinear parabolic PDE, which turns out to have a smooth solution. The term $e\Delta$ in Eq. (4) is regularizing the Hamilton-Jacobi type PDE. Then it is proved that, where $\varepsilon \to 0$ the solution of the level set function f^e of Eq. (4) will converge to some sort of week solution of Eq. (3). This technique is the method of vanishing viscosity.

We refer furthermore to [3] for more details on the existence and the uniqueness and the motivation for the definition of the viscosity solution. The algorithm adopted here is to solve Eq. (4) for steady state together with the time independent Navier-Stokes equation and the compressibility constraint. The SUPG- and PSPG-type weighting function permits the usage of the same interpolation function over all the unknowns and provides reasonable accuracy to the solution [11].

Nevertheless, the motivation behind the development of the stationary version of the level set equation is that equation (3) describe the motion of the fluid interface in $\Re^n$, the solution of this equation has the characteristics of developing discontinuities. The discontinuities known as shock wave across the fluid interface where the fluid can undergoes different negative and positive topological changings. Actually the shocks can arise due to the defined initial profile of the fluid interface on the finite element field. Particularly these shocks are functions of the **IVB**. Now the artificial viscous term added to the stationary level set equation performs as a an artificial viscosity term,

$$\boldsymbol{u}\cdot\nabla f - e\Delta f^{e} = 0. \qquad \text{in } \Re^{n} \cup t = 0 \tag{5}$$

The Laplacian of the viscous term acts as a smoothing term and prevent the development of these undesired shocks. However, as $\boldsymbol{e} > 0$ it is simply to show that the solution will remain smooth for all time [0, T]. Actually, this technique is known nowadays as the perturbation theory.

On the other hand the smoothing of the initially discontinuous field of the level set function is simply performed through some built-in interpolation in the numerical formulation. Noting that the smoothing has nothing to do with the shock or artificial diffusion. Actually the sharpening and smoothing technique [6] is just a consequence of interpolation for the discontinuous function by the specified continuous finite element field. Furthermore, it is convenient to keep the level set function constant in each fluid phase, which is consistence with mathematical formulation. However, any initial profile can be used for the solution as well as the interface defined by the zero level set function.

3. Governing equations

The simulations based on solving the unified time independent Navier-Stokes equations of the incompressible fluids. The non-Darcian flow equation $((B + A\cdot\|\boldsymbol{u}\|^{m})\boldsymbol{u})$ is adopted and incorporated in the momentum equation instead of the convection term ($\boldsymbol{u}\cdot\nabla\boldsymbol{u}$). The power m is ranged between one and two. In the present formulation the power set to one to assemble a quadratic inertial term. We refer more to [6] and [7] for the motivation of the incorporation of the quadratic term in our formulation and particularly for turbulent flow in porous medium. The momentum conservation equations and the incompressibility constraint can be written in the following vector form:

$$\begin{aligned} &\frac{n_e(f)}{r(f)}\left(\boldsymbol{A}+\boldsymbol{B}\|\boldsymbol{u}\|^{m}\right)\boldsymbol{u} = n_e(f)\,\boldsymbol{f} - \frac{n_e(f)}{r(f)}\nabla p + \frac{n_e(f)}{r(f)} m\,(j)(\nabla^{2}\boldsymbol{u}) && \text{on } \boldsymbol{O} \cup (t=0) \\ &\nabla\cdot\boldsymbol{u} = 0 && \text{on } \boldsymbol{O} \cup (t=0) \end{aligned} \tag{6}$$

where W denote the space domain, while the time domain t is overall equal to zero. The symbols $\boldsymbol{u}$, $\boldsymbol{p}$ and $\boldsymbol{f}$ are the velocity vector, the pressure and the body force due to gravity. The symbols r, m, and n_e are the density, the dynamic viscosity and the porosity of the porous medium. Whereas $\boldsymbol{A} = m/k_i$ and $\boldsymbol{B} = r\,\hat{c}/\sqrt{k_i}$ are the Forchheimer parameters. The symbols k_i, $\hat{c}$, represent the intrinsic permeability of porous medium and the constant of the inertial effect, respectively [6].

We model in this paper a problem with a liquid-gas interaction. Hence, we have fluid l and fluid g with densities r_l and r_g and viscosities m_l and m_g. The gas here is the atmospheric air that has a sufficiently low density. The free surface function f *(Level Set function)* serves as a marker identifying the water and the gas domain.

We combined the two-phase variables in the flow domains by the following

$$x(f) = \begin{cases} x_l & f > 0 \\ 0.5 \cdot (x_l + x_g) & f = 0, \qquad x = r, m, n_e \\ x_g & f < 0 \end{cases} \tag{7}$$

where the evolution of the sharp interface is governed by the stationary level set equation (5) in Section 2. Actually the equations (7) and (5) constitute the law for the fluid system. The accuracy of modeling this law is depending on the accuracy of locating and prorogating the front between the level set functions of the flow domains.

The system of equation above is completed with an appropriate set of boundary conditions, an initial guess for the velocity field with divergence-free conditions and the initial guess of the level set profile in the flow domains.

4. Numerical formulation

We consider in this section the stabilized FE-formulation for the time independent incompressible Navier-Stokes equations and the stationary level set equation. From Section 3, we have Eq. (6) and Eq. (5) with the boundary conditions $\boldsymbol{u} = \bar{\boldsymbol{u}}$ and $s \cdot \boldsymbol{n} = \bar{s}$ on G**.** Whereas G is the boundary of the domain W. Then, if u^h, p^h and f^h are the discretized solution belonging to the appropriate solution space, the stabilized Galerkin formulations of (6) and (5) is to find u^h, p^h and f^h such that,

$$\int_\Omega \boldsymbol{w}^h \cdot \left(\frac{n_e(f)}{r(f)} \left[\left(\boldsymbol{A} + \boldsymbol{B} \|\boldsymbol{u}\|^m \right) \boldsymbol{u} \right]^h \right) dW + \int_\Omega \boldsymbol{w}^h \cdot \left(\frac{n_e(f)}{r(f)} \cdot Ñp \right) dW - \int_\Omega \boldsymbol{w}^h \cdot \left(n_e(f) \cdot \boldsymbol{f} \right) dW - \int_\Omega \boldsymbol{w}^h \cdot \left(\frac{n_e(f)}{r(f)} \cdot m(f) \nabla^2 \boldsymbol{u} \right) dW \tag{8}$$

$$\int_\Omega \boldsymbol{q}^h Ñ \cdot \boldsymbol{u}^h dW + \sum_{n=1}^{\text{nelem}} \int_\Omega \left(\frac{t_{PSPG}}{r(f)} Ñ\boldsymbol{q}^h \right) \cdot \left[-\frac{n_e(f)}{r(f)} \cdot Ñp^h \right] dW = \int_\Gamma \boldsymbol{w}^h \cdot \boldsymbol{G}^h dG,$$

$$\left[\int_\Omega ?^h \left(\boldsymbol{u} \cdot Ñ f^e \right)^h dW - \int_\Omega ?^h \left(e \, \mathrm{D} f^e \right)^h dW \right]_{e \to 0} + \sum_{n=1}^{\text{nelem}} \int_\Omega t_{SUPG} \, Ñ ?^h \left(Ñ f^h \right) dW = 0, \tag{9}$$

for all weighting functions $\boldsymbol{q}^h$, $\boldsymbol{w}^h$ and y^h belonging to the appropriate specified function spaces. While $\boldsymbol{G}^h$ is the boundary conditions associated with the momentum equation. The terms t_{SUPG}, t_{PSPG} are the stabilization parameters that depend on the mesh size and the velocity field [7]. The final coupled system of finite element equations becomes,

$$\begin{pmatrix} 2\mathbf{K}_{ii}+\mathbf{K}_{jj}+\mathbf{B}_i(u_i)+\mathbf{K}_{ij}+\mathbf{A}_i(u_i) & -\mathbf{C}_{\mathrm{i}} & \mathbf{0} \\ -\mathbf{C}_i^T & \mathbf{W}_{\mathrm{i}} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{D}_i(u_i)+\mathbf{E}_{ii}+\sum \mathbf{S}_i \end{pmatrix}\begin{pmatrix}\tilde{\mathbf{u}}_i\\ \tilde{\mathbf{p}} \\ \tilde{\mathrm{F}}\end{pmatrix} = \begin{pmatrix}\mathbf{F}_i \\ \mathbf{0}\\ \mathbf{0}\end{pmatrix} \tag{10}$$

where the indexes i,j = 1, 2, 3 and $\tilde{\mathbf{u}}_i, \tilde{\mathbf{p}}, \tilde{\mathrm{F}}$ are the nodal values. The coefficient matrices in the system (10) has the following properties,

$$\mathbf{K}_{ij} = \int_{\Omega} m(f) \cdot \frac{n_e(f)}{r(f)} \cdot \frac{\partial j}{\partial x_j} \cdot \frac{\partial j^T}{\partial x_i} d\Omega \tag{11}$$

$$\mathbf{C}_i = \int_{\Omega} y^{\mathrm{T}} \cdot n_e(f) \cdot \frac{\partial j}{\partial x_i} d\Omega \tag{12}$$

$$\mathbf{W}_i = \int_{\Omega} t_{PSPG} \frac{1}{r(f)} \left(\frac{\partial y}{\partial x_i} \right) \cdot \frac{\partial j^T}{\partial x_i} d\Omega \tag{13}$$

$$\mathbf{A}_{\mathrm{i}}(u_j) = \int_{\Omega} A \cdot \frac{n_e(f)}{r(f)} \cdot j \cdot \left\| u_j \right\| u_j \; d\Omega \tag{14}$$

$$\mathbf{B}_{\mathrm{i}}(u_j) = \int_{\Omega} B \cdot \frac{n_e(f)}{r(f)} \cdot j \cdot u_j \; d\Omega \tag{15}$$

$$\mathbf{D}_{\mathrm{i}}(u_{\mathrm{j}}) = \int_{\Omega} j \quad u_j \frac{\partial J^{\mathrm{T}}}{\partial x_j} d\mathrm{O} \tag{16}$$

$$\mathbf{S}_i = \int_{\Omega} t_{SUPG} \cdot \frac{\partial J}{\partial x_i} d\Omega \cdot \int_{\Omega} \frac{\partial J^{\mathrm{T}}}{\partial x_j} d\mathrm{O} \tag{17}$$

$$\mathbf{E}_{ii} = \left[e \int_{\Omega} \frac{\partial J}{\partial x_i} \cdot \frac{\partial J^{\mathrm{T}}}{\partial x_j} d\mathrm{O} \right]_{e \to 0} \tag{18}$$

$$\mathbf{F}_{\mathrm{i}} = \int_{G} s_{\mathrm{i}} j \quad d\mathrm{G} + \int_{W} n_e(f) \cdot \; f_{\mathrm{i}} \cdot j \quad d\mathrm{O} \tag{19}$$

Where in each element the velocity, pressure and f fields are approximated by,

$$u_i(x) = j^{\,T} \tilde{\mathbf{u}}_i(x), \qquad p(x) = y^{\,T} \tilde{\mathbf{p}}(x), \qquad f(x) = J^T \tilde{\mathrm{F}}(x) \tag{20}$$

Where j, y and J are appropriate basis functions for the approximation. In the implementation below we chose $j = y = J$. We observe that the equation (10) is non-linear and unsymmetric.

We further notice that equation (10) is coupled implicitly to the other field of equation by the non-linear terms $\mathbf{D}_i(u_i)+\mathbf{E}_{ii}+\sum\mathbf{S}_i$ in the diagonal. While it is related implicitly to the system of equations above by the density r, viscosity m and porosity n_e of the flow domain. This will be accounted for in the numerical solution with the regularization, which motivate the formulation of the viscosity solution. .

Normally, equation (5) usually stipulate special numerical scheme to avoid nonphysical oscillations in the vicinity of the fluid interface. However, we are not interesting in the accuracy of the level set function, but we make use the sign of f. Hence, the main point in present problem is to keep the position of the zero level set of f accurate enough for the velocity field and the pressure computations. This is achieved by the stabilization of the finite element formulation.

Since the system is nonlinear, we have to solve it iteratively by a coupled solver. We notice that the regularization is changed adaptively through the iterative solution. Hence, the term $e\Delta$ is turned to be very small in effect when the solution is converged. The solution algorithm is described in the Fig. 1.

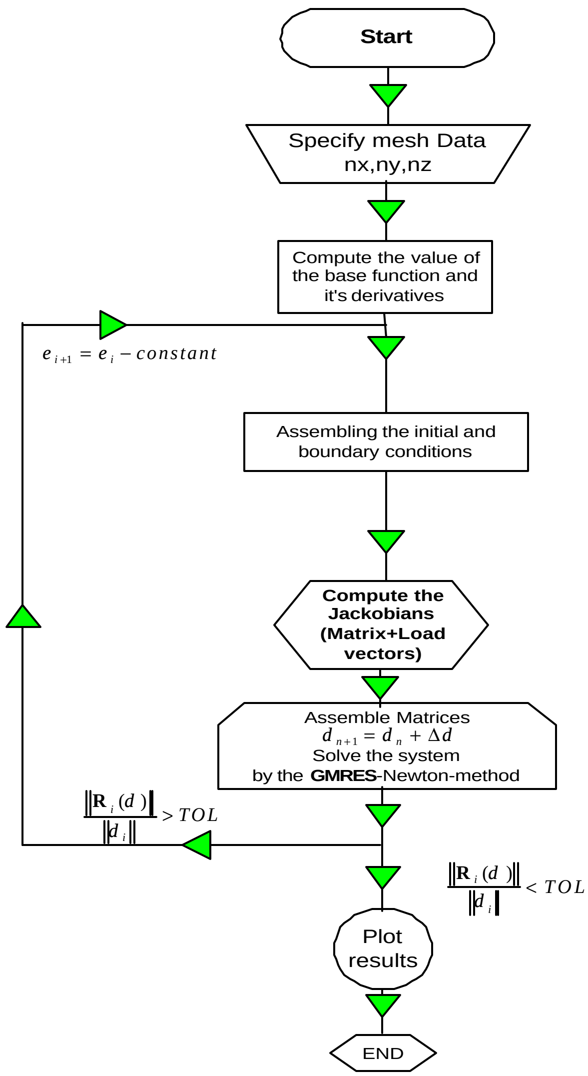


Fig. 1. Flow chart of the solution algorithm

The numerical scheme is implemented in a C++ code based on the **PETSC** [8] libraries. The implementation makes use heavily of the Object-Oriented Programming technique (OOP). We create a Non-linear solver based Newton-Raphson method with the GMRES diagonal precondition technique [11] for the coupled system. The solver is recognized as a class in C++. We can reuse the code for modeling problems that handle two-phase flows we also have the intention to generalize the implementation of the stationary level set method for tracking sharp interfaces in many industrial applications. Using the **PETSC** libraries, it is simply to implement the general Non-linear solver for the coupled system for higher dimensional problem. Hence, the solver can be simply used for 2D and 3D problems.

5. Numerical examples

2D Embankment dam with sloping sides: The 2D numerical example implemented in our code is shown in Fig. 2. The dimensions of the dam and the hydrostatic water levels applied at upstream and downstream boundaries are specified in meters. The flow domain is homogenous with isotropic characteristics. No heterogeneous feature is assumed. The 2D computations are performed for a triangulation FE -mesh with a linear interpolation over all the unknowns. The initial guess of the interface profile is taken as a straight line between the upstream and the downstream hydrostatic pressure. The first computation is performed with coarse triangulated mesh with 3-nodes (see Fig. 4a), then we refined the mesh globally in the flow domain (see Fig. 4b,c). The ratio of the densities and the viscosities are (1000) and (100), respectively. The porosity ratio is 1.28 and the permeability tensor is 0.002 m/s. The flow is subjected to the gravity acceleration (g = 9.81m/s^2). The zero level set is interpolated on the triangulation to produce the correct position of the interface at steady state and the exit critical point at the seepage face. Fig. 4 show the FE-SLS (Stationary Level Set Method) solutions of the flow domain.

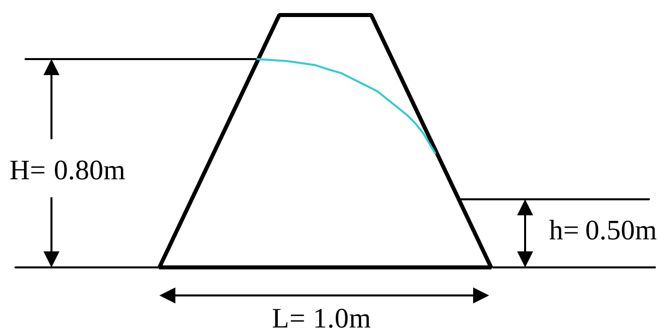
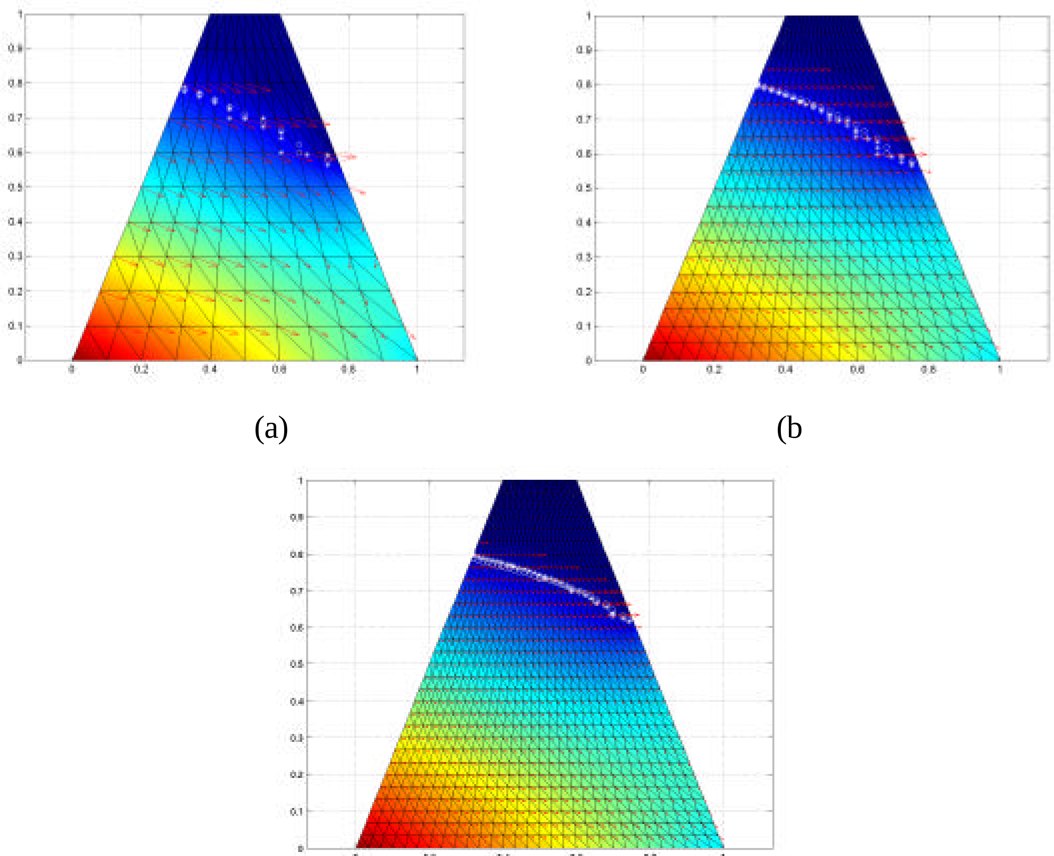


Fig. 3. free surface flow problem in earth-fill dam with sloping sides



(a) (b

(c)

Fig. 4. FE-SLS solutions for the pressure distribution, the velocity field, the interface with the zero level set

The element size h is reduced from 0.1 to 0.05 then 0.0033 in the simulations, which resulted in mesh of 121, 441 and 961 nodes, and 200, 800, 1800 triangulated linear elements, respectively. We notice further that the convergence rate is nonlinear.

We notice that the refined mesh produced sharper presentation of the zero level set and the convergence order is greater than coarse mesh. Furthermore, the number of iteration required to converge to the prescribed tolerance is reduced on finer mesh.

3D Embankment dam with sloping side: We extend the 2D problem to a 3D-flow problem. This simply performed by adding a new dimension to the stationary level set equation in our C++ classes. An additional set of boundary condition is prescribed to compensate with investigated flow problem. The same fluid properties and porous medium characteristics are specified as previously in 2D simulations. The three-dimensional domain is discretized by tri-linear tetrahedron mesh. A polynomial plane hydrostatic pressure prescribed on the upstream and downstream boundaries. The discretized domain resulted into 6000 (4-nodes) elements of 1331 nodes. The size of the meshes is varied between 0.1 and 0.0025. The order of convergence is also of second degree. The number of degree of freedom computed is 6655. The flow domain is shown in Fig. 5.

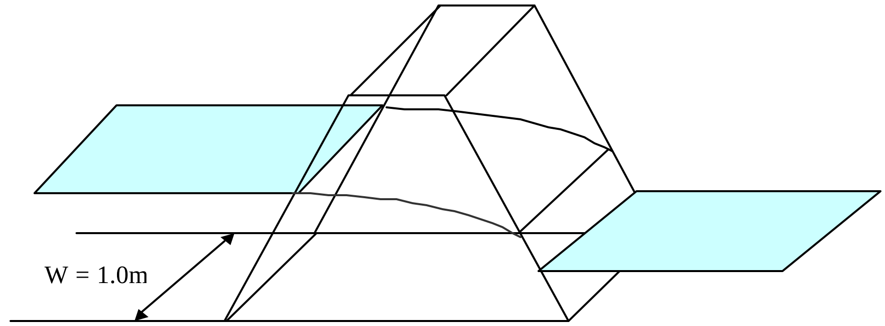
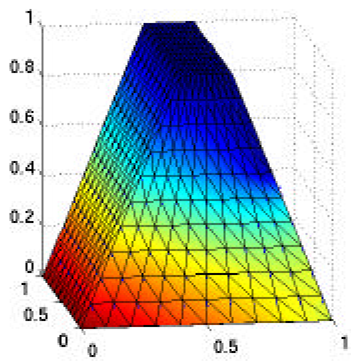
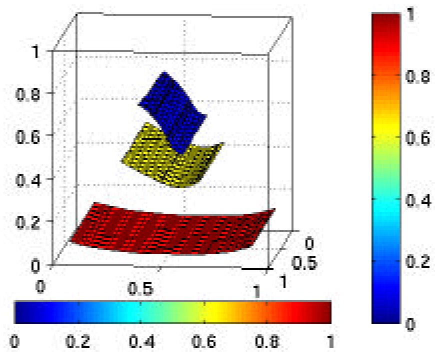


Fig. 5. 3D flow problem in embankment dam with sloping sides

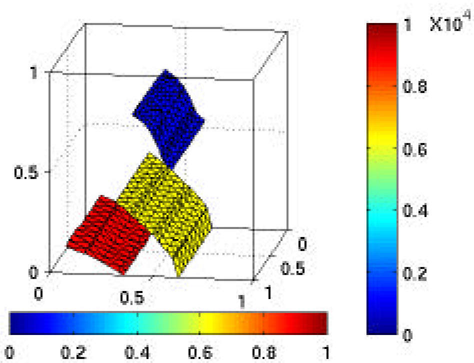
The FE-SLSM solution for the 3D problem is shown in Fig.6.



3D embankment dam



Surface of the Level set function



Pressure distribution surfaces

Fig. 6. FE-SLS solutions, the velocity field, surfaces of the level set, pressure distribution surfaces

6. Conclusions

We have developed a new and faster numerical framework for capturing sharp interfaces in groundwater flow. The numerical method implements a modified version of the level set method. This modified version is developed utilizing a technique from the vanishing viscosity method. The modified level set equation is a stationary version for capturing steady motion of sharp interfaces. The stationary version solved together with the conservation momentum equation for incompressible time independent equations over a fixed mesh domain. The mesh is refined globally in the flow domain to improve the accuracy of the fluid interface solution. Our C++ code used heavily the object-oriented programming techniques. This technique provides a very fast simulator for problems involve motion in two-phase domain. The technique is tested through simulations of 2D and 3D nonlinear seepage flows in embankment dam. These test examples associated with available quantitative and qualitative feature, and treated well in the literature. We have also discussed numerical instabilities and the nonphysical oscillations that arise in these flow problems.

The scope of the present numerical method is to make a faster numerical framework in comparison with recently solved time dependent version of the level set method [6, 7]. The stationary version is simply implemented and can be used for possible industrial applications. The method can easily generalized for more complicated mathematical models.

Acknowledgements

The Swedish foundations, LARS-ERIK LUNDBERGS AB and the J. GUST RICHERT, VBB group, Sweden, Stockholm, have sponsored this work.

References

[1] J. A. Sethian, Level Set Methods, Evolving Interface in Geometry, Fluid Mechanics, Computer Vision, and Material Science, Cambridge University Press (1996).
[2] KY Chung, N. Kikuchi, "Adaptive methods to solve free boundary problems of flow through porous media, *Int. Jour. Num. Analyt. Meth. Geomech.* Vol. 11, 17-31, (1987)".
[3] L. C. Evans, "Partial Differential Equations", American Mathematical Socity, Vol. 19, (1998).
[4] M. Sussman, P. Smareka and S. Osher, “A Level Set Approach for Computing Incompressible Two-Phase Flows”, *J. Comp. Phys.*, 114, 146-168, (1994).
[5] M. Sussman, E. Fatemi, P. Smareka and S. Osher, “A Improved Level Set Method for Incompressible Two-Phase Flows”, *J. Computers and Fluids*, 212-223 (1996).¨
[6] N. H. Sharif, N.-E. Wiberg, "Free-Surface Flow Predictions by an Interface Capturing FE-Technique” to appear in the International Journal for Computational Civil and Structural Engineering (IJCCSE).
[7] N. H. Sharif, N.-E. Wiberg, ”Free Surface Flow through Rock-Fill Dams Analyzed by FEM with Level Set Approach" to appear in Journal of Computer Modeling and Simulation in Engineering (C.M.S.E).
[8] PETSC WWW home page. http://www-fp.mcs.anl.gov/petsc/, 2000.
[9] S. Osher and R. P. Fedkiw, "Level Set Method: An overview and some Recent Results" Recent UCLA Computational and Applied Mathematics Reports, Department of Mathematics, University of California, Los Angeles, California, February 2000
[10] T. J. Barth and J. A. Sethian, “Numerical Schemes for the Hamilton-Jacobi and Level Set Equations on Triangulated Domains”, *J. Comp. Phys.*, 145, 1-40, (1998)
[11] T. Tezduyar, S. Aliabadi, M. Behr, “Enhanced-Discretization Interface-Capturing Technique (EDICT) for Computation of Unsteady Flows with Interfaces”, *Comp. Methods Appl. Mech. Eng.*, 155, pp. 235-248, (1998).

[12] Y. H. Tomas, “Numerical Solution to Free Boundary Problems”, in *Acta Numerica* 1995, Cambridge University Press, pp. 335-415 (1995).
[13] Y. Saad and M. H. Schultz, “GMRES: A Generalized Minimal Residual Algorithm for Solving Non-symmetric Linear Systems”, *J. Soc. Idust. Appl. Math.* Vol. 7, No. 3, July (1986).