Reprint from: Proceedings of the 7-th International Conference on Estuarine and Coastal Modeling, St. Petersburg, FL, November 2001, p. 628-642.

Evaluation of the UnTRIM Model for 3-D Tidal Circulation

Ralph T. Cheng
U. S. Geological Survey
Menlo Park, California 94025

Vincenzo Casulli
Department of Civil Engineering
University of Trento, ITALY

Abstract

A family of numerical models, known as the TRIM models, shares the same modeling philosophy for solving the shallow water equations. A characteristic analysis of the shallow water equations points out that the numerical instability is controlled by the gravity wave terms in the momentum equations and by the transport terms in the continuity equation. A semi-implicit finite-difference scheme has been formulated so that these terms and the vertical diffusion terms are treated implicitly and the remaining terms explicitly to control the numerical stability and the computations are carried out over a uniform finite-difference computational mesh without invoking horizontal or vertical coordinate transformations. An unstructured grid version of TRIM model is introduced, or UnTRIM (pronounces as “you trim”), which preserves these basic numerical properties and modeling philosophy, only the computations are carried out over an unstructured orthogonal grid. The unstructured grid offers the flexibilities in representing complex study areas so that fine grid resolution can be placed in regions of interest, and coarse grids are used to cover the remaining domain. Thus, the computational efforts are concentrated in areas of importance, and an overall computational saving can be achieved because the total number of grid-points is dramatically reduced. To use this modeling approach, an unstructured grid mesh must be generated to properly reflect the properties of the domain of the investigation. The new modeling flexibility in grid structure is accompanied by new challenges associated with issues of grid generation. To take full advantage of this new model flexibility, the model grid generation should be guided by insights into the physics of the problems; and the insights needed may require a higher degree of modeling skill.

Introduction

Based on a characteristic analysis of the shallow water equations, a semi-implicit finite-difference method of solution was proposed by Casulli (1990). The terms that affect the numerical stability are treated implicitly, and the remaining terms are treated explicitly. This formulation not only controls the numerical instability, it also improves the computational efficiency. Since 1990, following the concept of the semi-implicit finite-difference scheme, a family of TRIM (Tidal, Residual, and Intertidal Mudflat) models has been systematically developed by Casulli and his associates for solving 2D and 3D shallow water equations with or without invoking hydrostatic approximations (Casulli and Cheng, 1992; Cheng et al, 1993;

Casulli and Cattani, 1994; Cheng and Casulli, 1996; Casulli, 1997; Casulli and Stelling, 1998; Gross et al, 1998; Casulli, 1999a). Another unique feature of this family of models is that the flow field is solved in the original physical plane without invoking any coordinate transformation. The TRIM family of models has proven to be computationally efficient, and the computational efficiency is not compromised due to the fact that the numerical solutions are computed in the physical space using a uniform computational grid.

In contrast, because of the demand on computing resources is commonly quite high for most 3-D circulation models, some 3-D models introduce an orthogonal coordinate transformation on the x-y plane and a σ-coordinate transformation in the vertical to alleviate computing demands. The computations are performed in the transformed coordinate system (e.g., Blumberg and Mellor, 1987; Hamrick and Wu, 1997). Although the orthogonal coordinate transformation does have some advantages for simulating flows in complex domains, the orthogonal curvilinear transformation is cumbersome. There could be numerical errors admitted in the computed results inherited from the extra terms stemming from the transformations. The impacts due to these errors are hard to evaluate as some of the mathematical terms do not associate directly with any physical meaning. In order to deal with the usually complex basin geometry, the semi-implicit numerical algorithm of TRIM has been extended to an unstructured orthogonal grid. This new model is referred to as UnTRIM and has been reported previously by Casulli and Zanolli (1998) and Casulli (1999b). This new model preserves the advantages of the semi-implicit finite-difference formulation for numerical stability and robustness, and the use of unstructured grid allows boundary fitting and arbitrary local grid refinements to meet the needs of resolving fine spatial resolution in some numerical modeling tasks.

This paper presents a review and an evaluation of the UnTRIM model and discusses issues associated with unstructured computational grids with respect to its robustness and computational efficiency. Finally, the UnTRIM model is applied to San Francisco Bay to test its practicality in solving full-scale realistic problems.

Summary of the Numerical Algorithm for UnTRIM

The governing equations for three-dimensional, baroclinic, environmental flows and transport of conservative scalar variables in an estuary include the conservation equations of mass and momentum, conservation equations for scalar variables, an equation of state, and a kinematic free-surface equation. The estuarine system is assumed to be sufficiently large so that a Coriolis acceleration term (constant coefficient) is included in the momentum equations. To simplify the governing equations, the water is assumed to be incompressible; the pressure is assumed to be hydrostatic; and the Boussinesq approximation applies. In Cartesian coordinates, the governing equations are the continuity equation,

$$Div(\vec{U}) = 0 \quad , \qquad (1)$$

the kinematic free-surface equation (integrated continuity equation),

$$\frac{\partial \eta}{\partial t} + \nabla \bullet \left[\int_{-h}^{\eta} \vec{V} \, dz \right] = 0 \quad , \qquad (2)$$

where Div ( ) is divergence in three-dimension; $\vec{U}$ is the three-dimensional velocity vector; $\vec{V}$ is the horizontal velocity vector; and $\nabla \bullet ( )$ denotes the divergence on the horizontal plane. The horizontal momentum equation in $\vec{N}_j$ direction is,

$$\frac{DV_j}{Dt} - f(\nabla \times \vec{V}) \bullet \vec{N}_j = \frac{\partial}{\partial z}(\nu_v \frac{\partial}{\partial z} V_j) + \nu_h \nabla^2 V_j - g\frac{\partial \eta}{\partial N_j} - \frac{g}{\rho_o}\frac{\partial}{\partial N_j}\int_z^{\eta}(\rho - \rho_o)dz' \quad (3)$$

where $\vec{N}_j$ is a vector on the horizontal plane; $\frac{\partial ()}{\partial N_j} = \vec{N}_j \bullet \nabla( )$ is the gradient in $\vec{N}_j$ direction; V_j is the velocity component in $\vec{N}_j$ direction; $\frac{D}{Dt}$ is the substantial derivative; and $\nabla \times ( )$ is the cross-product on the x-y plane. The transport equation for salt and conservative solutes, C_i, is

$$\frac{D}{Dt} C_i = \frac{\partial}{\partial z}(K_v \frac{\partial}{\partial z} C_i) + K_h \nabla^2 C_i \quad ; \quad (4)$$

and an equation of state showing that the water density is a function of salinity and temperature,

$$\rho = \rho_o [1 + \alpha s + \beta(T - T_o)^2] \quad ; \quad (5)$$

where $\alpha = 7.8 \times 10^{-4}$ and $\beta = 7 \times 10^{-6}$, and
(u, v, w) are (x, y, z) velocity components;
η is the free-surface elevation measured from a reference datum;
ρ and ρ_o are density and a reference density;
f is Coriolis parameter;
ν_v and ν_h are vertical and horizontal eddy viscosity;
K_v and K_h are vertical and horizontal eddy diffusivity;
C_i are s, T, and conservative solutes, i = 1,2,3,…..;
s is salinity in practical salinity unit (psu);
T and T_o are temperature and a reference temperature in ^{0}C.

For three-dimensional barotropic flows (constant density), the solute transport equation is un-coupled from the momentum equations. The governing system of equations can be solved efficiently by a semi-implicit finite-difference method over a regular computational mesh as discussed by Casulli and Cheng (1992) and Casulli and Cattani (1994). For baroclinic flows, the transport equations are coupled with the momentum equations through the density gradient terms. In this case, the baroclinic forcing terms (density gradients) are solved explicitly in the momentum equations, and the solutions of the transport variables are solved lagged one time-step. The numerical scheme is subject to a weak Courant-Friedrich-Lewy (CFL) stability condition due to the explicit treatment of the transport equation, and the baroclinic pressure terms in the momentum equations. It is also subject to a weak stability condition due to the explicit treatment of the horizontal diffusion in the momentum equations. An equilibrium turbulence closure is used in the model. A non-negative bottom friction coefficient is specified by, typically, the Manning-Chezy formula, or fitting to a bottom turbulent boundary layer.

Orthogonal unstructured grids

One of the salient characteristics the TRIM family of models is that the numerical solutions are computed in the physical space without any coordinate transformation in the x-, y-, or z-directions. In the numerical algorithm, the stability properties of the governing partial differential equations are controlled by using the semi-implicit finite-difference schemes (Casulli, 1990; Casulli and Cheng, 1992), the resulting numerical algorithm is robust and computationally efficient. In lieu of a curvilinear coordinate transformation, traditional finite-difference schemes resort to refining the rectangular finite-difference mesh when a complicated domain is encountered in order to resolve the flow distributions in narrow and confined regions. Unless a sub-domain formulation is considered, it is necessary to use the same refined finite-difference mesh for the entire domain of the model. The resulting fine resolution grids in broad and open regions are unnecessary, and the fine computational mesh also consumes a large portion of computing resources, which cannot be justified. Therefore, it is logical to extend the semi-implicit finite-difference method in solving the shallow water equations to an unstructured grid (Casulli and Zanolli, 1998; Casulli, 1999b; Casulli and Walters, 2000) in which fine grid resolutions are used in complex regions, and relatively coarse grids are used in broad and open areas. The combined use of semi-implicit algorithm for stability and an unstructured grid for flexibility in solving the shallow water equations comprises the essence of the UnTRIM model.

<u>Issues of Unstructured Grid</u>

The numerical algorithm of UnTRIM is fundamentally the same as TRIM3D (Casulli and Cheng, 1992; Casulli and Cattani, 1994), except the finite-difference treatment of the governing partial differential equations is performed over an unstructured grid mesh. Before discretizing the governing equations, the horizontal domain (x, y) is covered by a set of non-overlapping convex polygons. Each side of a polygon is either a boundary line or a side of an adjacent polygon. Moreover, it is assumed that within each polygon there exists a point (hereafter called

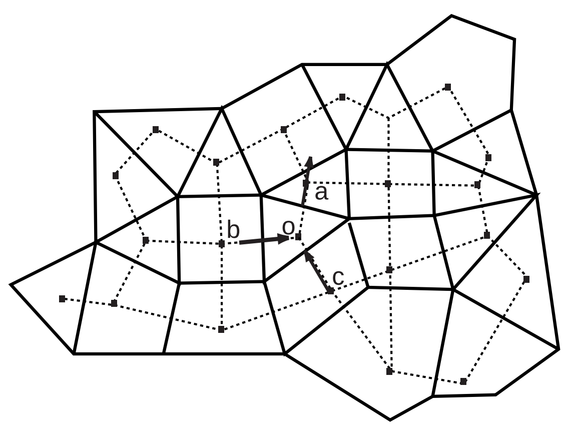


Figure 1: Orthogonal unstructured grid

center) such that the segment joining the centers of two adjacent polygons and the side shared by the two polygons have a non-empty intersection and are orthogonal to each other (see Figure 1). One such grid is called an <u>unstructured orthogonal grid</u> (Casulli and Zanolli, 1998; Casulli and Walters, 2000). The center of a polygon does not necessarily coincide with its geometrical center. The special cases of unstructured orthogonal grids include, of course, the rectangular finite-difference grids, as well as a grid of uniform equilateral triangles. In these particular cases the center of each polygon can be identified with its geometrical center. Another example of an unstructured orthogonal grid is a set of Delaunay triangles where the triangulation includes only acute triangles (Rebay, 1993).

In an unstructured grid representation of a domain (x, y), there are N_p number of polygons; each polygon has an arbitrary number of sides. For each polygon, the x-y coordinates of the vertices that define the polygon must be given. In addition, every side of the polygon is designated a unique number. The connectivities between polygon and vertices, and the connectivities between polygons and sides must be defined for an unstructured grid mesh. For example, the two polygons that share the j-th side of the grid are identified by two indices identifying the left polygon, i(j, 1), and the right polygon, i(j, 2). The positive direction for the normal velocity on the face (hereafter called face velocity) on the j-th side is defined to be from polygon i(j, 1) to polygon i(j, 2) given in the connectivity relations. The distance between the centers of two adjacent polygons that share the j-th side must be non-zero. In a three-dimensional space, the system is extended in the vertical direction by layers of horizontal surfaces (z-planes). Thus, the polygons on the horizontal planes become a stack of prisms whose thickness is related to the prescribed layer thickness. The water surface elevation is assumed to be constant within each polygon, and is defined at the center of the polygon. The velocity component normal to each face of a prism is assumed to be constant over the face. The true velocity is defined at each vertex in the middle of each layer. Spatial distribution of velocity is obtained by interpolations. Finally, the water depth of the basin is specified and assumed constant on the sides of polygons.

Numerical Approximation

A semi-implicit scheme defined above is used in order to obtain an efficient numerical algorithm whose stability is independent from the free-surface gravity wave, wind stress, vertical viscosity and bottom friction. Consider a typical polygon, Figure 1, the momentum equation, Eq.(3), is finite-differenced in the normal direction of each vertical face along oa, ob, and oc directions. The momentum equation relates the gradient of water surface elevation between adjoining polygons and the face velocity on the common face between these polygons. As stated previously, the wind stress, the vertical mixing and the bottom friction are discretized implicitly for numerical stability.

An explicit finite-difference operator is used to account for the contributions from the discretization of the advection and horizontal dispersion terms. A particular form for this operator can be given in several ways, such as by using an Eulerian-Lagrangian scheme (Casulli and Cheng, 1992). For stability, the implicitness factor θ has to be chosen in the range $\frac{1}{2} \leq \theta \leq 1$ (Casulli and Cattani, 1994). Along the vertical direction, a simple finite-difference discretization, not necessarily uniform, is adopted. The vertical space increment is defined as the distance between two consecutive level surfaces. In general, the vertical thickness of the top and bottom layers can vary depending on the spatial location and the thickness of the top layer can also vary with time. The vertical space increment is allowed to vanish. In fact this is how the wetting and drying of polygons are accomplished.

The free-surface equation, Eq.(2), is discretized implicitly by the θ-method (Casulli and Cattani, 1994; Casulli and Walters, 2000), and only face velocities are needed to complete the finite volume balance of total volume in the polygon. At the center of each polygon, by substituting the finite-differenced momentum equations on all faces of a polygon into the continuity equation (finite-volume method), the resultant matrix equation governs the water surface elevation distribution for the entire domain. This matrix equation is strongly diagonally dominant, symmetric and positive definite; thus its unique solution can be efficiently determined by preconditioned conjugate gradient iterations until the residual norm becomes smaller than a given tolerance (Golub and van Loan, 1996). Once the free-surface for the next time level has been calculated, the normal velocities on the faces of the prisms are calculated by back substitution. The details of the finite-difference equations are not reproduced here and readers are referred to Casulli and Walters (2000). If baroclinic flows are considered, the transport variables are solved explicitly using the velocity field obtained for the next time level. In summary, this numerical algorithm is a combination of the finite-volume method along with a semi-implicit consideration of the terms that control the numerical stability.

Examples of Unstructured Grid

Conversion of existing model grids

Although polygons of any number of sides can be used, the present version of the numerical model code accepts mixed 3- or 4-sided polygons. If the computational domain is covered by uniform rectangles or equilateral triangles then the space difference is second order. Since the traditional TRIM3D model (Casulli and Cheng, 1992) uses a regular mesh, that model grid is a special case of an unstructured grid acceptable by UnTRIM. In a regular finite-difference mesh, the connectivities between the computational nodes, sides, and polygons have a systematic pattern, and they need not be defined explicitly. In contrast, for an unstructured grid model, the definitions of the node locations, polygon numbers and side numbers must be supplied to the model. Even for a regular grid mesh, the connectivities between polygons, sides and nodes must be precisely defined. This, however, can be achieved by a preprocessor program that converts the regular finite-difference model depths to an UnTRIM model input file. One such example is the UnTRIM grid converted from a finite-difference grid prepared for a TRIM model application in San Francisco Bay (Cheng and Smith, 1998, 2000), Figure 2.

One of the characteristics for all finite-difference models is that the shoreline boundaries are approximated by staircase like boxes; and obviously, in some situations, this approach is not a good representation of the boundary. With exactly the same mesh, the computing resources requirements for TRIM and UnTRIM are similar as expected since the numerical algorithms in these two models are virtually identical. Therefore, in this case, the UnTRIM model supercedes previous TRIM models in all aspects. Similarly, a uniform finite-difference model grid prepared for other models can be easily converted to an UnTRIM model grid with an appropriate conversion program.

(a) (b) (c)

Figure 2. (a) San Francisco Bay Model domain, (b) Suisun Bay region [the zoomed in region shown in (a)] where darker colors represent deeper water, and (c) the four-sided polygons used in the model.

The only requirement on the computational grids for UnTRIM is that the grids are locally orthogonal. It would be useful to develop a model interface that could convert a general orthogonal curvilinear model grid to an unstructured model grid for input to the UnTRIM model. The capability of UnTRIM to use essentially the same orthogonal curvilinear grids in simulations

(a) (b)

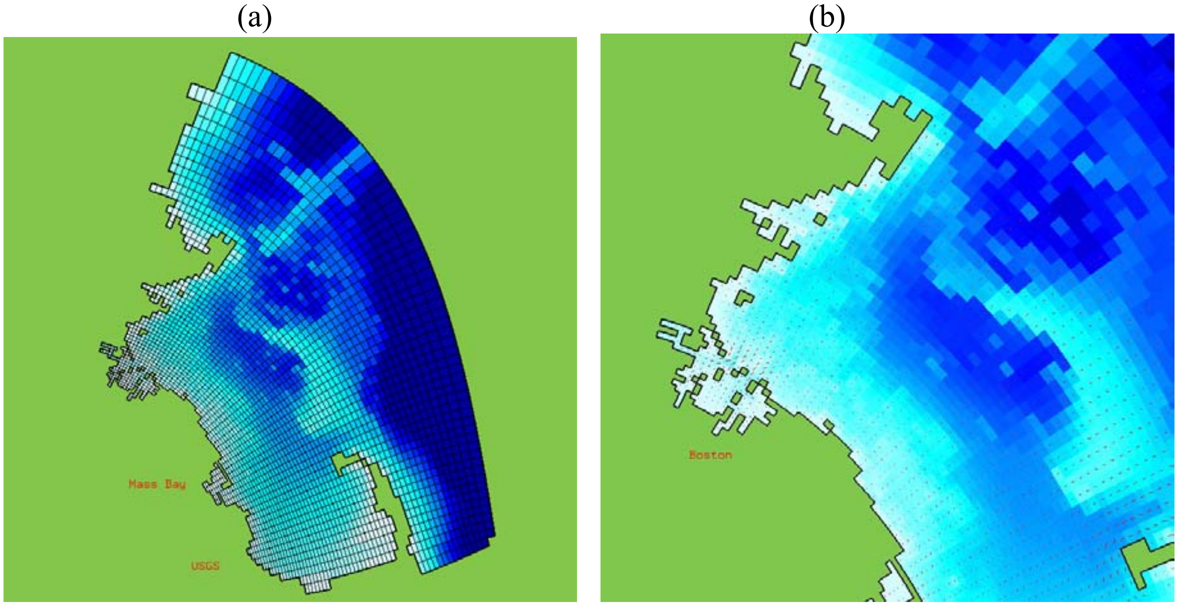


Figure 3. (a) The converted 'unstructured' orthogonal curvilinear grid of Massachusetts Bay, (b) A snap shot of the simulated tidal current pattern near Boston Harbor.

is of great interest and importance. This model grid conversion would allow future comparisons of model simulations of the same scenario by UnTRIM and by other models using an orthogonal curvilinear grid. This proposition is investigated using the Massachusetts Bay model studied by

Blumberg et al (1993) and Signell et al (2000) as an example. The converted UnTRIM grid is shown in Figure 3.

A simulation of the tidal circulation in Massachusetts Bay represented by this grid was made. The numerical model has 3002 polygons and 6304 sides on the top layer. If 25 vertical layers are used with their layer thickness varies from 4 x 1 m, 3 x 2 m, 5 x 4 m, 1 x 6 m, 1 x 10 m, 2 x 12 m, 4 x 15 m, and 5 x 20 m, the resulting 3-dimensional grid has a total of 84045 faces; on each face a normal velocity is calculated. The smallest length of the side is about 30 m, and the largest side is about 8000 m. The simulation was run on a Pentium-4 (1.7 GHz CPU) personal computer. For a 15 day simulation using Δt = 900 second, and an M_2 tide of 1.2 m amplitude specified on all open boundaries, the total CPU time required for this simulation is about 1600 second. The purpose of this simulation is to evaluate the computational efficiency of UnTRIM, and no direct comparison was made with results from previous studies (Signell et al., 2000).

Generating Unstructured Grid for UnTRIM

Obviously, for UnTRIM, the model grid definition file is very different when compared to the regular mesh, finite-difference models. Some aspects of the unstructured model grid are similar to those used in finite-element applications, therefore the literature in finite-element grid generation might be applicable and useful for grid generation in UnTRIM applications. For the purposes of generating input files to an UnTRIM model, a commercial product for mesh generation, "Argus"[1], has been adopted. This package is designed for mesh generations in connection with general finite element computations. The outputs from "Argus" cannot be used directly in UnTRIM, because substantial additional information is needed by UnTRIM.

For any hydrodynamic modeling study of flows and circulation in estuaries or coastal seas, a proper computational mesh representing the basin is essential. The water depths either at the nodes or on the side of computational cells must be extracted from a bathymetry data file in which the water depth z is given as a function of (x,y), or $z = -h(x,y)$. The domain of interest is defined by the definition of shoreline contours. Based on the given shoreline contours and bathymetry data, a set of either triangular or quadrilateral polygons can be generated quite easily using "Argus." There are numerous functions built-in "Argus" allowing users to control the desirable mesh properties (at least to some extent) to optimize the grid size and mesh distributions. The mesh density and grid distribution can also be controlled within "Argus" by user specified control functions to determine the mesh density. However, there is no guarantee that the generated meshes are orthogonal. The basic grid-mesh file generated by "Argus" includes the definition of nodal locations, definition of polygons, connectivities of polygons, and the water depths at nodes or at the center of each side. The basic grid-mesh file is treated as an interim file for building the input grid-file to UnTRIM. A model grid interface program (fortran based) has been developed to generate UnTRIM model input file that meets the requirements of the model. If the study considers an area that is very large and/or complex, the grid generation project can be broken up into many sub-regions. An appropriate grid-mesh for each sub-region

[1] Any use of trade, product, or firm name is for descriptive purposes only and does not imply endorsement by the U. S. Geological Survey.

is generated first as an independent task. The model grid interface program also has the capability of melting (combining) grid-meshes of sub-regions into an UnTRIM grid-file for the overall project. During the process of combining grid-mesh of sub-regions, the triangles with their centers outside of the triangle are identified. Since the UnTRIM model code accepts the combination of 3- and 4-sided polygons, attempts are made to ‘melt’ two triangles whose centers are outside or nearly outside into a 4-sided polygon for which its center is well defined within the polygon. While melting two 3-sided triangles into a 4-sided polygon may not always be possible, there remain a handful of triangles that may not meet the orthogonality conditions. Roughly, there are less than 0.1% such triangles, and most of them are boundary polygons that have less impact on the computed results. If necessary, it will be possible to go back to “Argus” to locate these polygons, and then to correct the grid geometry in these conditions. Finally, the interface program reorders the polygon numbers so that the open boundary polygons will appear in a sequential order at the beginning of the input grid-file.

The model mesh for Suisun Bay, a part of the San Francisco Bay estuary, California consists of 3279 nodes, 4911 polygons, and 8199 sides on the top layer. Twenty-five (25) vertical layers are specified with the layer thickness varying from 1 m in the top 20 layers, 2 m for the next 5 layers (Figure 4a). The resulting grid has a total of 62K faces; on each face a normal velocity is calculated. The smallest length of the side is about 45 m, and the largest side is about 800 m. In addition, the fine triangular mesh within the narrow channels can be replaced by long and narrow rectangular channel polygons (Figure 4b), with the same 25 vertical layers, the total number of face velocities is reduced to about 49K, or approximately 20% fewer velocities. For an identical simulation, the required CPU time is proportionally reduced by about 20% because there are 20% fewer faces due to the introduction of channel polygons. Some simulated tidal current patterns near a tidal channel are shown in Figure 5. The reduction in the total number of velocity points implies savings in CPU time.

(a) (b)

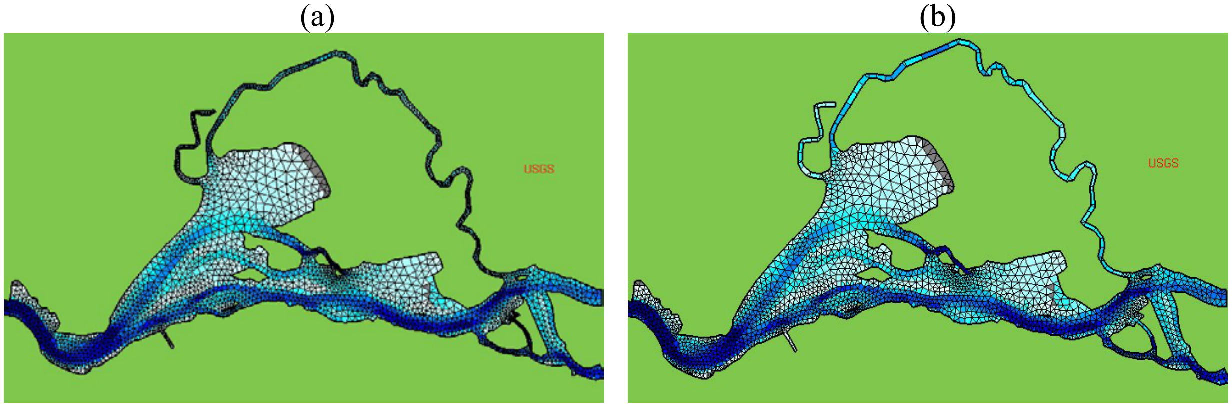


Figure 4. (a) An unstructured grid representing Suisun Bay, a part of San Francisco Bay, California. (b) The typical two rows of polygons in narrow channels are replaced by long and narrow rectangular polygons where the flow is expected to be bi-directional.

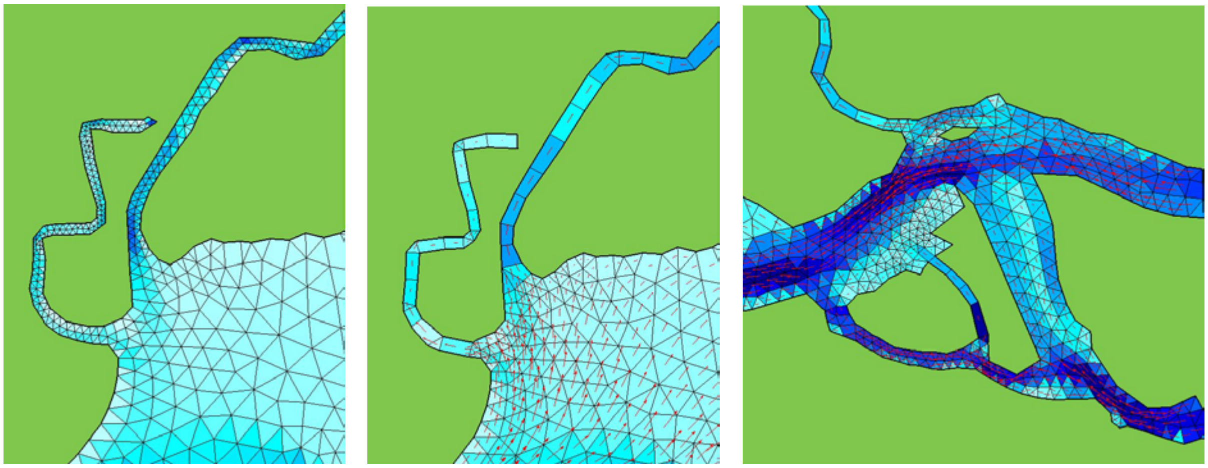


Figure 5. Simulated tidal current patterns near a tidal channel, the reduction in the total number of velocity points implies savings in CPU time.

Validation of UnTRIM

As the first step in evaluating and validating a new model, the simplest situations are considered. Specifically, in a tidal basin, the initial condition is set so that the water in the basin is stationary and at open boundaries, zero forcing is specified. In a long time-dependent simulation, the water body should remain stationary. Furthermore, if a stable stratification is present and no forcing is specified on open boundaries, again the water body should remain stationary. These exercises were performed using UnTRIM, and indeed, in a 15 days simulation, no detectable motions were found in the water body.

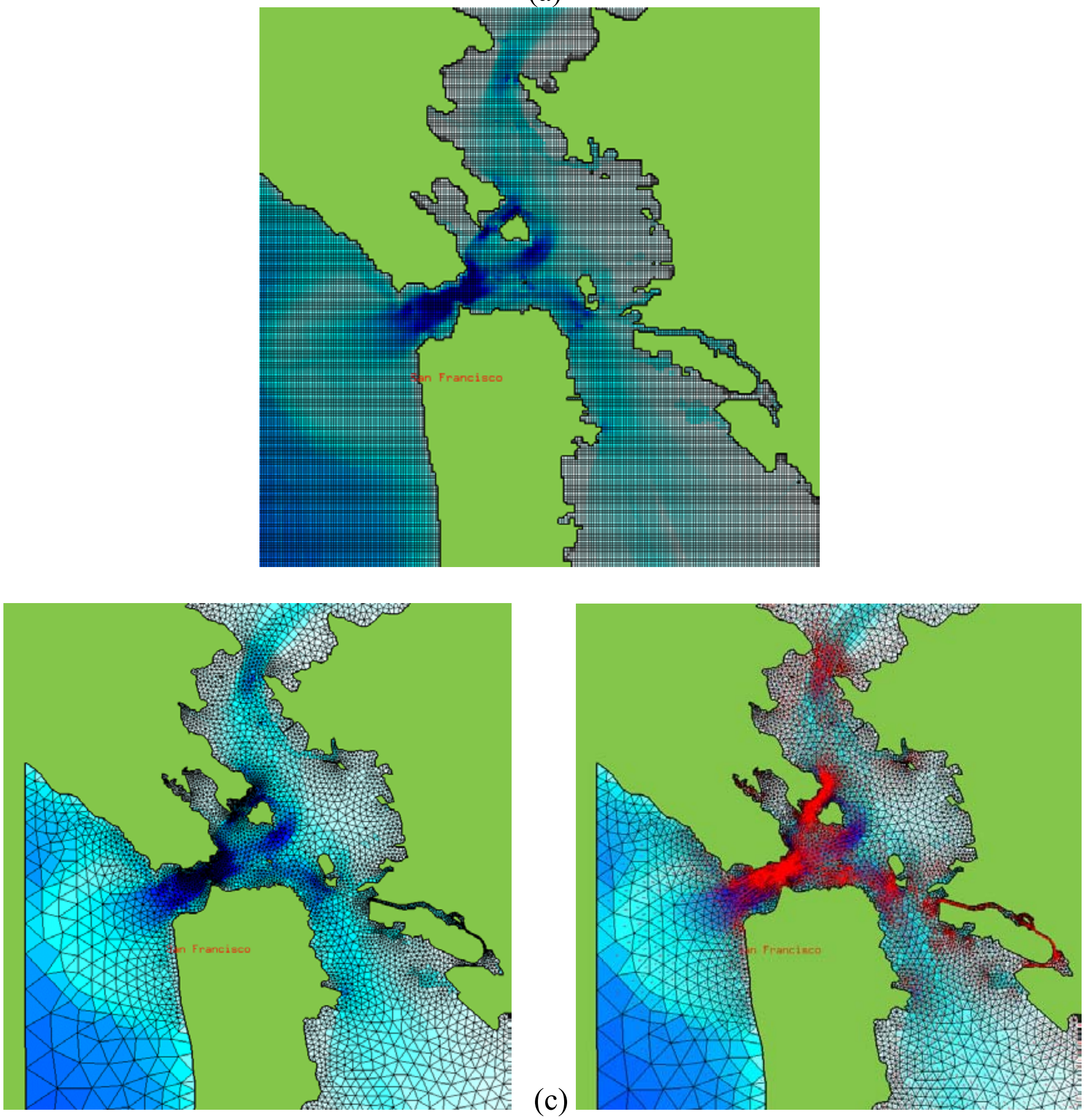
Comparisons between TRIM classic and UnTRIM

The next example considers a structured finite-difference grid mesh for the entire San Francisco Bay using $\Delta x = \Delta y = 200$ m, which is converted to an 'unstructured' grid resulting in 48506 nodes, 45841 polygons, 94374 sides on the top layer of a 42 layers model (Cheng and Smith, 1998, 2000). In a finite-difference grid, once a grid size is chosen, the same grid size is used for the entire domain.

Figure 6 (a) shows the finite difference grid for San Francisco Bay, California. In this study, although the interest is on tidal circulation in the bay, one must take the open boundary conditions some 15 kilometers west from the coast to properly simulate the processes near the Golden Gate region, the narrow entrance to the bay. An unnecessarily fine grid is used in the open ocean where the focus of interest is not there. The total number of face velocities is about 1.160 million. For a 72 hours simulation with $\Delta t = 180$ sec, it requires 5.26 hours CPU time on a 1.7 GHz PC, or a simulation time and CPU time ratio R = 13.7.

If an unstructured grid is used to represent the same area of the bay, very fine grids are used in areas of interest and in areas where high velocity and velocity gradients are expected. In other areas, the grid sizes are gradually increased from very fine grids to very coarse grids, for example in the Pacific Ocean. Figure 6 (b) shows such a grid, that gives 12682 nodes, 20126 polygons and 32827 sides on the top layer of a 42 layers, and a total of 295 K faces for the model. For 72 hours simulation with the same time step (180 seconds), it requires 1.33 hours

(a)



(b) (c)

Figure 6 (a) A 200 m finite-difference mesh of San Francisco Bay, California, (b) An unstructured grid-mesh in which very fine grids are used in areas of interest and in areas where high velocity gradients are expected, relatively coarse grids are used in other areas, (c) A snap shot of the simulated velocity distribution near the entrance of San Francisco Bay, California.

CPU time (R= 54) on a 1.7 GHz PC. In these simulations, the required CPU time is roughly proportional to the total number of face velocities. A qualitative snap shot of the tidal current distribution near the entrance region of San Francisco Bay is shown in Figure 6(c). Time-series records of these two identical simulations using structured and unstructured grids are compared in Figure 7. Although some small differences exist, possibly due to the difference in grid resolution, general agreements are achieved between these two cases. In fact, the side lengths of the polygons in the region near Golden Gate is on the order of 50 m and 100 m for the unstructured grid, while the finite difference grid has a side length of 200 m.

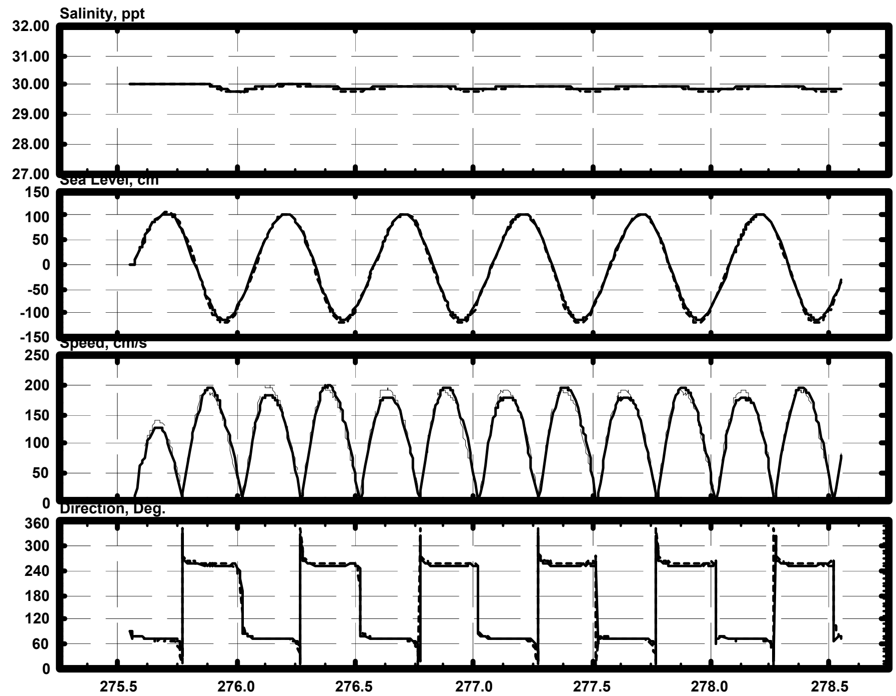


Figure 7. The time series of five days simulation using the structured and unstructured grids, recorded at a station near the entrance region of San Francisco Bay. The panels from the top is the depth-averaged salinity, sea-level, depth averaged tidal speed, and tidal current direction. Clearly these results are nearly identical, the slight difference is possibly due to the difference in grid resolution.

Discussion and Conclusion

The numerical algorithm used in the UnTRIM model is relatively simple, yet general and robust. It not only preserves the numerical properties of the semi-implicit finite-difference formulation and the desirable numerical stability and robustness, the unstructured grid used in the sense of a finite-volume method allows an arbitrary local grid refinement to meet the needs of resolving fine spatial resolution in complex regions while keeping relatively coarse spatial resolution in areas of less importance. From a practical point of view, UnTRIM has achieved both robustness in numerical properties and flexibilities in physical space that are important in numerical modeling of complicated environmental hydrodynamic problems. The mathematical formulation suggests that the unstructured polygons can be of any number of sides. In the present computer code, mixed 3- and 4-sided polygons are used. The essential properties of the algorithm concerning mass conservation, numerical accuracy, stability and generality are summarized below.

a) **Mass Conservation** -- In the present scheme the local and global mass conservation is guaranteed because the finite volume form is used in discretizing the incompressibility condition (mass conservation equation) and the free-surface equation.

b) **Reduction to Uniform Mesh** -- If the horizontal polygons are uniform rectangles, this algorithm is identical to the semi-implicit finite-difference scheme presented by Casulli and Cheng (1992) and Casulli and Cattani (1994). The highest numerical accuracy is obtained when a uniform grid, such as equilateral triangles or uniform rectangles, is used. In these cases, the discretization error for the gravity wave terms is second order in space.

c) **Numerical Stability Properties** -- For barotropic flows, the stability analysis of the semi-implicit finite difference method has been carried out by Casulli and Cattani (1994) on a uniform rectangular grid and under the assumptions that the governing differential equations are linear, with constant coefficients and defined on an infinite horizontal domain, or with periodic boundary conditions on a finite domain. The analysis shows that the method is stable in the von Neumann sense if a θ-method is used with $½ \leq \theta \leq 1$ and if the operator used to discretize the advection and horizontal friction terms is itself stable. Computational results on several test cases have indicated that no additional stability restrictions are required when a non-uniform unstructured mesh is used. The stability of the present algorithm is independent of the celerity, wind stress, vertical viscosity and bottom friction. It does depend on the discretization of the advection and horizontal friction terms. When an Eulerian--Lagrangian method is used for the explicit terms, a mild limitation on the time step depends on the horizontal viscosity coefficient and on the smallest polygon size. This method becomes unconditionally stable when the horizontal friction terms are neglected.

d) **Baroclinic Flows** -- When the baroclinic flows are considered, the solute transport equation is coupled to the momentum equations through the baroclinic forcing terms. When the semi-implicit finite-difference scheme is used to solve the governing system of equations, the baroclinic forcing terms are treated explicitly. More precisely, the solute transport equation is solved lagging by one time step. This treatment of the baroclinic forcing introduces a mild numerical stability constraint due to the presence of internal waves. A Courant-Friedrich-Lewy (C-F-L) stability condition based on internal wave speed limits the size of time-step in the integration. Properties of internal wave depend upon the degree of stratification in the water column. It is reasonable to estimate that the internal wave speed is lower than the surface gravity wave speed (celerity) by a factor $(\Delta\rho/\rho)^{1/2}$, thus, the internal wave C-F-L stability limit exists, and clearly, this stability constrain is not desirable. Fortunately, the stability condition is an order of magnitude less stringent than the stability condition due to gravity waves.

e) **Degeneration to 2D and 1D Domains** -- The structure of the numerical algorithm is such that if only one vertical layer is specified, the numerical model is reduced to 2D. If a long and narrow polygon is used to represent a channel, this section of the model could be a vertical 2D model, or an 1D model if only one layer is used.

f) **Flooding and Drying of Sub-regions** -- The present algorithm allows for the simulation of flooding and drying of low lying areas in a straightforward manner since the finite volume scheme is used for discretizing the mass conservation equation.

g) **New Flexibilities and New Challenges** – The unstructured grids used in the computation has the obvious advantages that allows the grids to be boundary fitting and allows an arbitrary local grid refinement to meet the needs of resolving fine spatial resolution in some numerical modeling tasks. Although methods and literature exist to assist model grid generation, the process of model grid generation can be a complicated task. The new modeling flexibility is facing new challenges associated with issues of grid generation. In order to take full advantage of these new model flexibilities, the model grid generation should be guided by insights into the physics of the problems; and the needed insights may require a higher degree of modeling skill.

References

Blumberg. A. F., and Mellor, G. L., 1987, A Description of a Three Dimensional Coastal Ocean Circulation Model, N. S. Heaps (ed.), Three Dimensional Coastal Ocean Circulation Models, Coastal and Estuarine Sciences 4, AGU, Washington, DC, 1-16.

Blumberg, A.F., R.P. Signell and H.L. Jenter, 1993, Modeling Transport Processes in the Coastal Ocean, Journal of Environmental Engineering, Volume 1, p. 31-52.

Casulli, V., 1990, Semi-implicit Finite-Difference Methods for the Two-dimensional Shallow Water Equations, J. Comput. Phys., Vol. 86, p. 56-74.

Casulli, V., and Cheng, R. T., 1992, Semi-Implicit Finite Difference Methods for Three-Dimensional Shallow Water Flow, Int. Jour. for Numerical Methods in Fluids, 15, 629-648.

Casulli, V., and E. Cattani, 1994, Stability, Accuracy and Efficiency of a Semi-implicit Method for Three-Dimensional Shallow Water Flow, Computers & Mathematics with Applications, 27, 99-112.

Casulli, V., 1997, Numerical simulation of three-dimensional free surface flow in isopycnal co-ordinates, Inter. J. for Num. Methods in Fluids, Vol. 25, p. 645-658.

Casulli, V. and G. S. Stelling, 1998, Numerical Simulation of 3D quasi-hydrostatic free-surface flows, ASCE, J. Hydra. Eng., Vol. 124, p. 678-686.

Casulli, V. and P. Zanolli, 1998, A Three-dimensional Semi-implicit Algorithm for Environmental Flows on Unstructured Grids, Proc. of Conf. On Num. Methods for Fluid Dynamics, University of Oxford.

Casulli, V., 1999a, A semi-implicit finite difference method for non-hydrostatic, free-surface flows, Inter. J. for Num. Methods in Fluids, Vol. 30, p. 425-440.

Casulli, V., 1999b, A semi-implicit numerical method for non-hydrostatic free surface flows on unstructured grid, Proceedings, Inter. Workshop on Numerical Modelling of Hydrodynamic Systems, Zaragoza, Spain, June 1999, p. 175-193.

Casulli, V., and R. A. Walters, 2000, An unstructured grid, three-dimensional model based on the shallow water equations, Inter. J. for Num. Methods in Fluids, Vol. 32, p. 331-348.

Cheng, R. T., V. Casulli, and J. W. Gartner, 1993, Tidal, residual, intertidal mudflat (TRIM) model and its applications to San Francisco Bay, California, Estuarine, Coastal, and Shelf Science, Vol. 36, p. 235-280.

Cheng, R. T. and V. Casulli, 1996, Modeling the Periodic Stratification and Gravitational Circulation in San Francisco Bay, in Proceedings of 4-th Inter. Conf.on Estuarine and Coastal Modeling, Spaulding and Cheng (Eds.), ASCE, San Diego, CA, October 1995, p.240-254.

Cheng, Ralph T. and R. E. Smith, 1998, A Nowcast Model for Tides and Tidal Currents in San Francisco Bay, California, Ocean Community Conf. '98, Marine Technology Society, Baltimore, Nov. 15-19, p. 537-543.

Cheng, R. T., and R. E. Smith, 2000, San Francisco Bay Marine Nowcast System, in the proceedings of 2000 Inter. Conf. on Hydroscience and Engineering, Seoul, Korea, September 25-29, 2000.

Gross, E. S., and V. Casulli, L. Bonaventura, and J. R. Koseff, 1998, A semi-implicit method for vertical transport in multidimensional models, Inter. J. for Num. Methods in Fluids, Vol. 28, p. 157-186.

Golub, G. H., and van Loan, C. F., 1996, Matrix Computations, 3rd edn., J. Hopkins, London.

Hamrick, J. M., and Wu, T. S., 1997, Computational design and optimization of the EFDC/HEM3D surface water hydrodynamic and eutrophication models, Computational Methods for Next Generation Environmental Models, G. Delic, and M. F. Wheeler, Eds., Society for Industrial and Applied Mathematics, Philadelphia, p. 143-156.

Signell, R.P., Jenter, H.L., and Blumberg, A.F., 2000, Predicting the physical effects of relocating Boston's sewage outfall, J. Estuarine, Coastal and Shelf Sci., 50, 59-72.

Rebay, S., 1993, Efficient Unstructured Mesh Generation by Means of Delaunay Triangulation and Bowyer--Watson Algorithm, Jour. of Computational Physics, 106, 125-138.