HYBRID FINITE ELEMENT / VOLUME METHOD TO SOLVE SHALLOW WATER EQUATIONS FOR STORM SURGE MODELING

Shahrouz Aliabadi[1] and Muhammad Akbar[2]
[1]Director and Professor, [2]Senior Research Associate,
Northrop Grumman Center for HPC,
School of Engineering, Jackson State University
1400 Lynch St , Jackson, MS 39217.

ABSTRACT

A hybrid numerical scheme based on finite element and finite volume methods is developed to solve shallow water equations (SWEs). The SWEs are derived by depth-averaging the Reynolds equations for a column of fluid with mass and momentum conservation. In SWEs, it is assumed that vertical motions are negligible and that pressure is hydrostatic. The dimensions in the horizontal plane are by far larger than the vertical dimension. Therefore, it is reasonable to assume that flow is homogeneous along the vertical axis. In the present study, we use a highly accurate segregated implicit projection method to solve the 2D SWEs on staggered unstructured meshes. Our hybrid numerical scheme is truly a segregated method with primitive variables stored and solved for at both node and element centers.

In the past recent years, we introduced a series of hybrid methods to solve incompressible low and high Reynolds number flows for single and two fluid flow problems. The hybrid finite volume/element solver is aimed to take advantage of the merits of both the FV and the FE methods and avoid their shortcomings. For example, highly-stretched elements (also known as high-aspect-ratio elements) are commonly used inside the boundary layer for high Reynolds number flows to resolve the boundary layer and reduce the number of elements. The stabilization parameters in the stabilized FE based flow solvers are related to the characteristic element length which is not well defined for high-aspect-ratio mesh elements. Due to this, it is very difficult to control the numerical dissipation of stabilized finite element solvers. By contrast, the finite volume flow solver is very insensitive to the aspect ratio of the mesh elements. For this reason, the finite volume method is used to solve the momentum equation. On the other hand, the classic Galerkin FE method is very suitable for elliptic typed equations like the pressure Poisson equation emerging from the segregated approach. Therefore, the combination of the FV method and the FE method perform well in the incompressible flow solvers based on the pressure projection method.

The present work extends the application of the above mentioned hybrid method to SWEs. Our hybrid shallow water flow solver is called CaMEL^{SWE} (<u>C</u>omputation <u>a</u>nd <u>M</u>odeling <u>E</u>ngineering <u>L</u>aboratory). The governing SWEs equations are written in non-conservation form. All the traditional shallow water equation terms including hurricane induced wind stress and pressure, bottom friction, Coriolis effects, and tidal forcing conditions are retained in this model. An intermediate velocity field is first obtained by solving the momentum equations with the matrix-free implicit cell-centered finite volume method. The nonlinear wave equation is solved by the node-based Galerkin finite element method with linear interpolation functions in space. This staggered-mesh scheme is distinct from other conventional approaches in that the velocity components and auxiliary variables are stored at cell centers and vertices, respectively.

The mesh used for this case study consists of 254,565 nodes and 492,179 elements, which covers the entire Gulf of Mexico and about half of the Atlantic Ocean. The mesh is displayed in Figure 1. The domain is spread deep in the ocean for better tidal forcing implementation. Zero-flux boundary conditions are applied on the land and island boundaries, and tidal conditions in the open ocean boundary. The time varying tidal condition should propagate from the open ocean to inside the domain. For the zero-flux boundary condition in the land and islands, the normal velocity component of the boundary nodes are set to zero and the tangential velocity component are kept non-zero. The mesh has the bathymetric depth information at every node.

The domain consists of patches where ground elevation can be higher than the sea level. As such, some nodes may be wet and dry in the course of simulation. A wet-dry algorithm is implemented to deal with this phenomenon. A node is assumed to be wet if its water depth is higher than a threshold value, typically 0.01~0.02m. An element is assumed to be wet if all of its nodes are wet. Similarly, an element is dry if all nodes are dry. An element having one or two wet nodes and the average element water height exceeding the threshold value (0.01~0.02m) is considered to be mixed. The graphical representation of the wet and dry condition is displayed in Figure 2. The mixed elements are treated in the same way as the wet elements. However, in the mixed elements there are one or two dry nodes. At the integration points of those mixed elements, a small water depth of 0.01~0.02m is assumed for the dry nodes to avoid any numerical inconsistencies. The nodal water depth or elevation is not modified. In the present solution technique, only the wet and mixed elements in the domain are solved, which expedites the simulation.

To enhance the stability of the hybrid method around discontinuities, we introduce a new shock capturing technique which will act only around sharp interfaces without sacrificing the accuracy elsewhere. The shock capturing adds numerical diffusion to the finite element formulation. However, the coefficient of numerical diffusion is residual-based which satisfies consistency criteria. In addition, the shock capturing adds nonlinearities to the formulation which is a desirable feature to control the over shoot and undershoot around discontinuities.

Several test problems are tested to validate the robustness and applicability of the numerical method used in CaMEL^{SWE} solver. Hurricane Katrina (2005) has been a popular benchmark case for many storm surge and other relevant models in the recent past. The storm surge of Katrina is simulated using CaMEL^{SWE} and the results are compared with the observed data. In addition, the ADCIRC model results are used for model-to-model comparison. Figure 3 compares the maximum elevation of water (MEOW) profiles simulated by CaMEL^{SWE} and ADCIRC for hurricane Katrina. Although the solution techniques of CaMEL^{SWE} and ADCIRC are different, the comparison of MEOW is very encouraging. Figure 4 shows the levee overtopping phenomena is the New Orleans area due to hurricane Katrina. The overtopped locations are highlighted by white ovals.

Since the CaMEL^{SWE} solver is implicit, a large time step can be used without accuracy and stability issues. Time steps of the order of minutes were successfully used in this solver, whereas maximum 5s can used in ADCIRC model with the current mesh due to stability restrictions. Usually the wet-dry component of ADCIRC is argued to be the bottle-neck for accuracy and hindrance for using a larger time step. On the other hand, the wet-dry algorithm used in CaMEL^{SWE} model appears to have little effect on the accuracy with the current mesh. A comparative MEOW is presented in Figure 5 for two different time step results from

CaMEL^{SWE}. This feature in particular is very important in the event of an actual hurricane when faster-than-real-time storm surge prediction is needed to setup the emergency response. Therefore, it is a very suitable model to predict the hurricane induced storm surges.

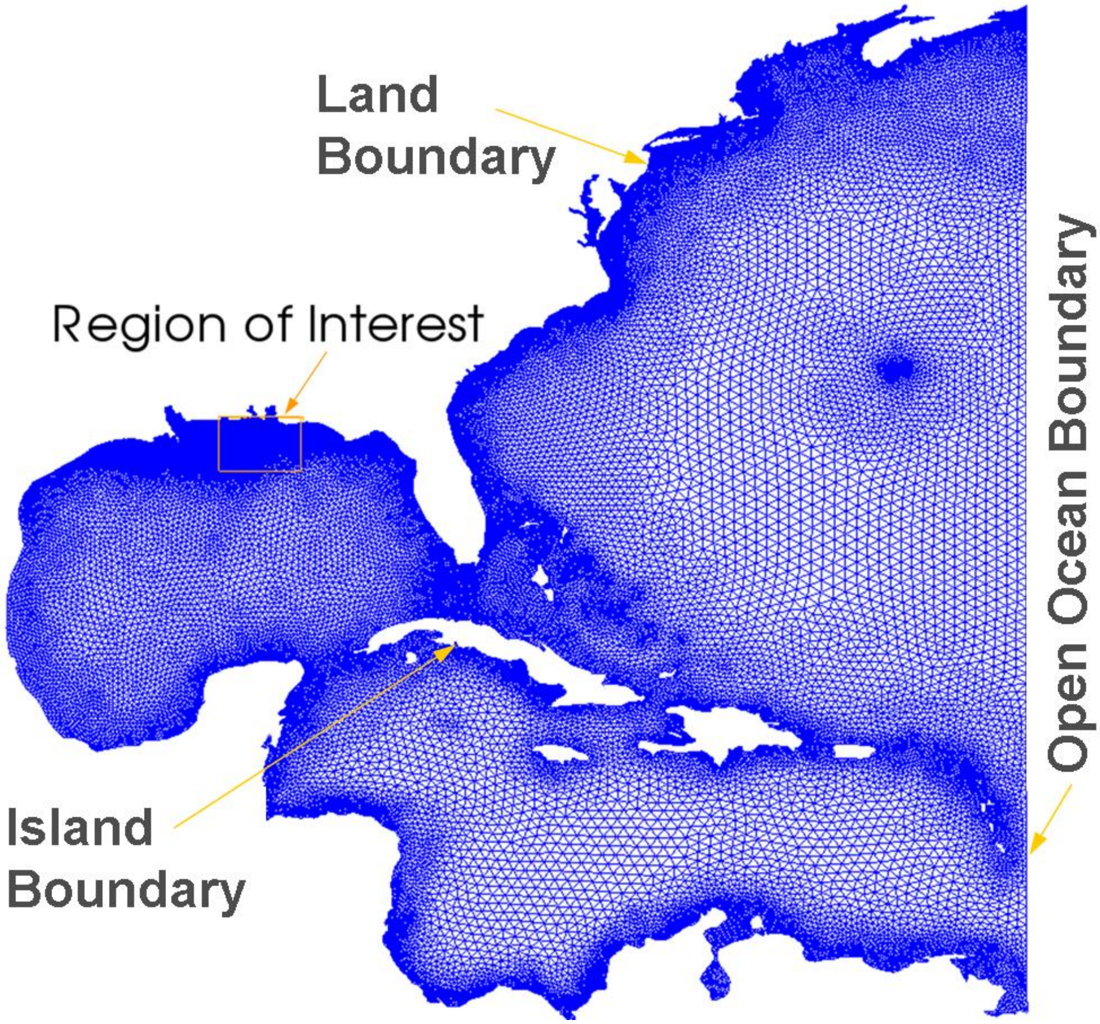

Figure 1: Computational domain used in the study.

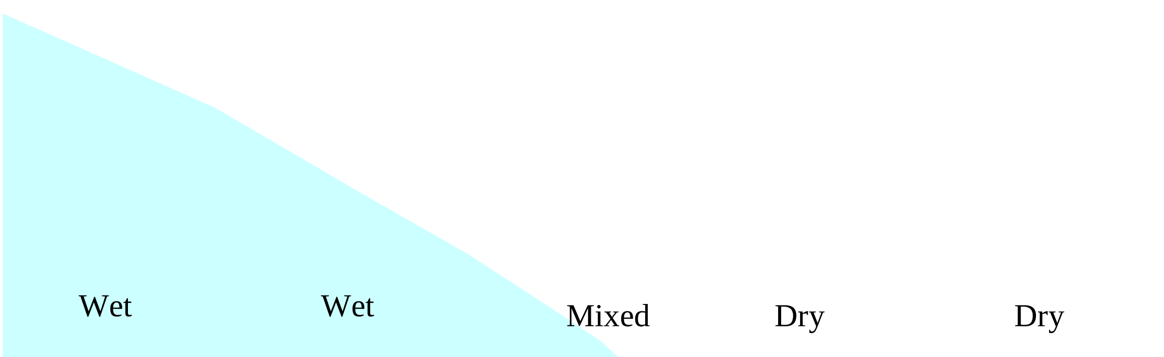

Figure 2: Wet and dry phenomena in the CaMEL^{SWE} model.

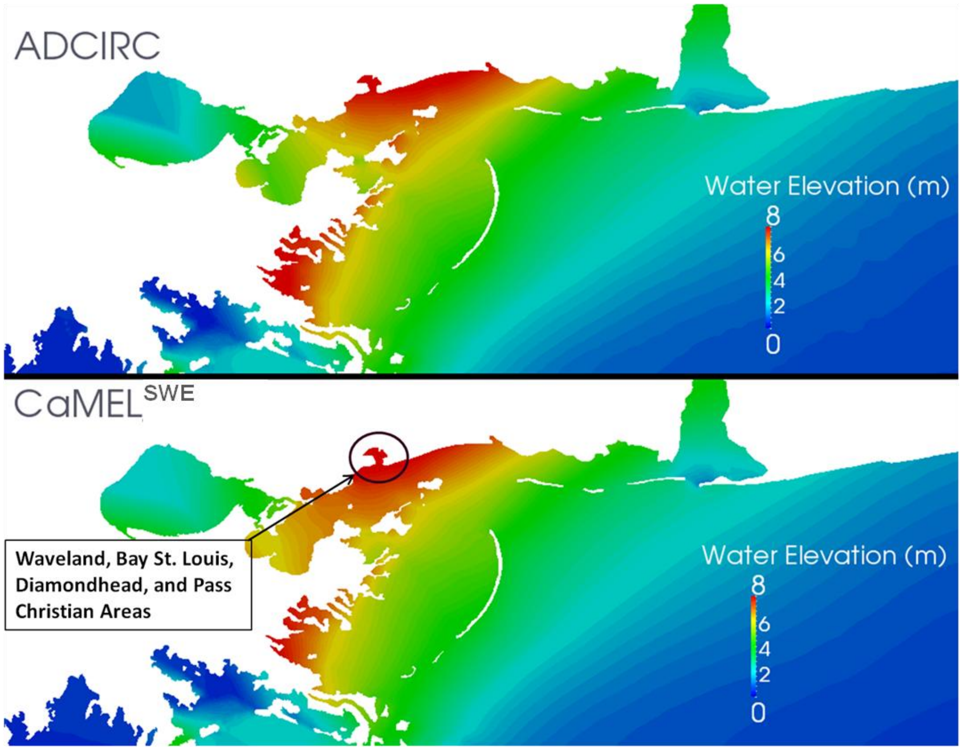

Figure 3: Simulated maximum elevation of water (MEOW) comparison for ADCIRC and CaMEL^{SWE}.

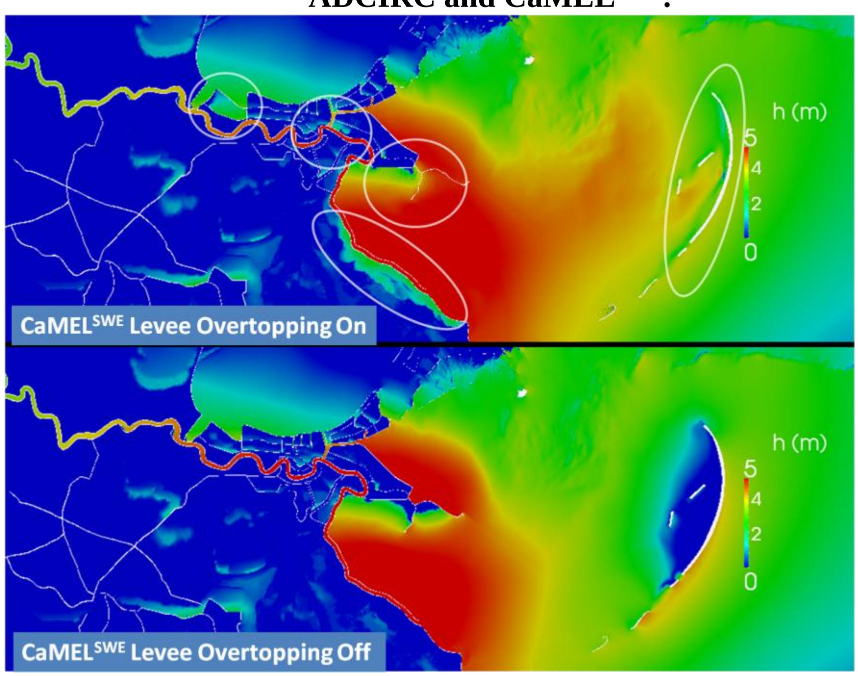

Figure 4: Simulated results of New Orleans area levee and barrier overtopping using CaMEL^{SWE} solver (Overtopped locations are pointed out by white ovals).

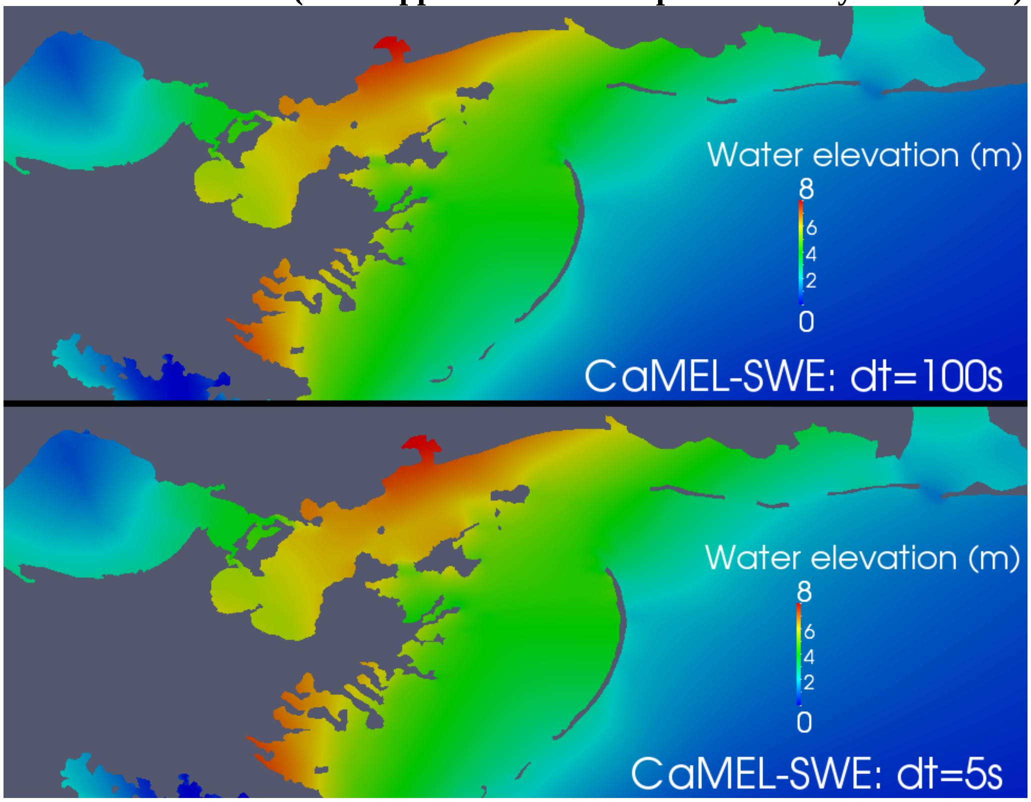

Figure 5: Timestep effect in CaMEL-SWE model.