Computational Fluid Dynamics Simulation of Open-Channel Flows Over Bridge-Decks Under Various Flooding Conditions

S. PATIL, M. KOSTIC and P. MAJUMDAR
Department of Mechanical Engineering
Northern Illinois University
DeKalb, IL 60115
U.S.A.
Email: kostic@niu.edu; Web: http://www.kostic.niu.edu

Abstract: - The bridge 'hydraulics analysis and design' could be substantially enhanced using advanced commercial Computational Fluid Dynamics (CFD) software and powerful parallel computing resources. The objective of the present study is to validate STAR-CD commercial CFD software for the bridge hydraulics research. This study simulates limited scaled experimental data conducted elsewhere for bridge flooding in open channel flow with Froude number, Fr=0.22, and dimensionless inundation height ration, h^*=1.5. Two approaches are used to simulate two-dimensional open channel turbulent flow with six different turbulence models. The first approach uses transient VOF methodology for two-phase, open channel free surface, and the other approach uses steady state method with free surface as a fixed slip-flat-top wall. Drag and lift coefficients are computed and compared with limited experimental data. The transient VOF simulations, which are found to be slow, unstable and over-sensitive to its computational parameters, result in larger discrepancies for drag and lift coefficients than the second, steady-state method with the fixed slip-flat-top wall, the latter gives faster, more stable solutions than transient VOF simulations, and shows better agreement with available experimental results. The k-ε Renormalization Group (RNG) is identified as the best turbulence model for our simulation, still predicting 27 % higher drag coefficient than limited experimental results with unknown accuracy. Further improvements can be done using the non-linear eddy viscosity turbulence models or Large Eddy Simulation (LES), as well as three-dimensional computational models with more realistic boundary conditions and more optimized simulation parameters.

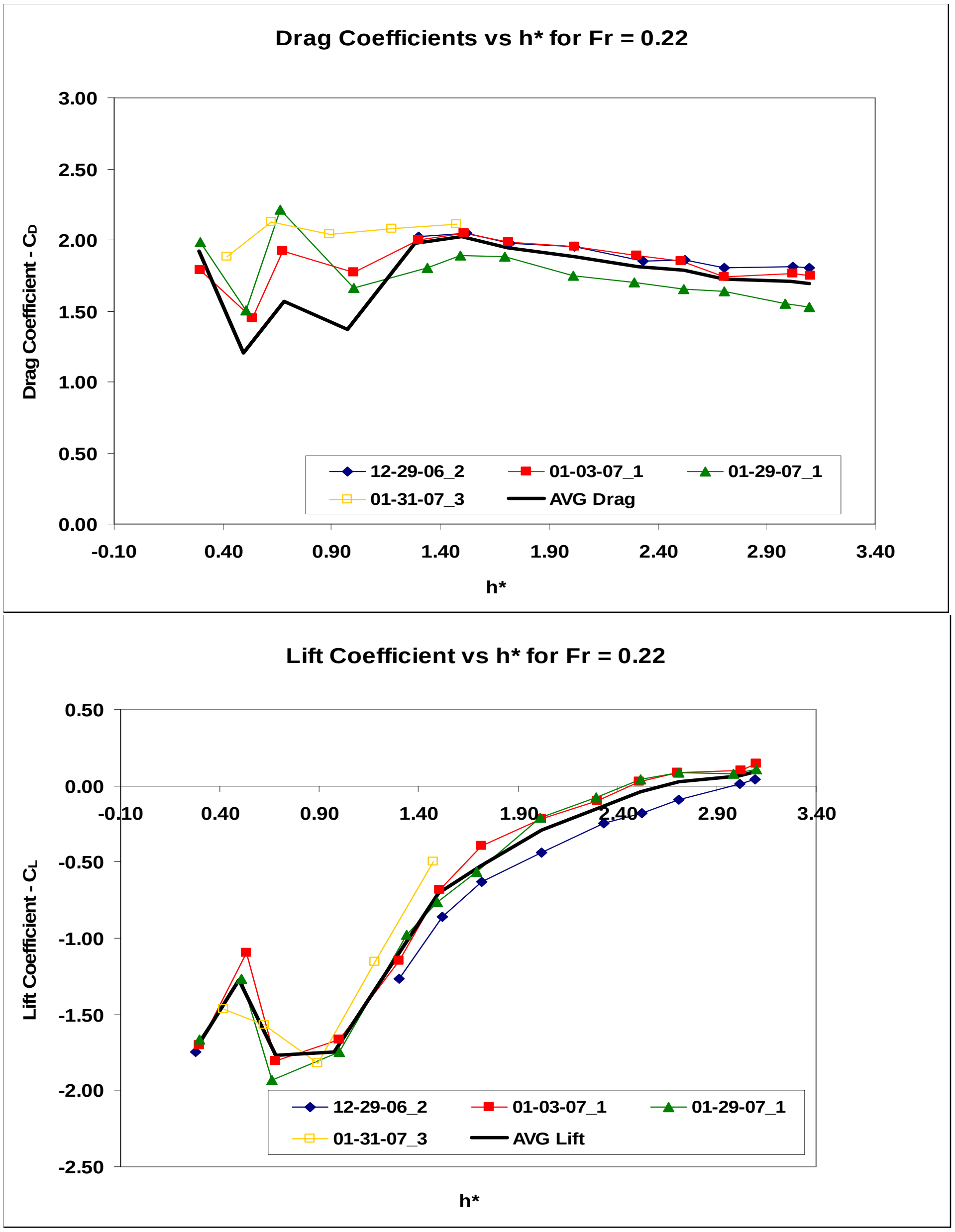
Key-Words:-Bridge deck, CFD simulation, Computational fluid dynamics, Flooding flows, Turbulence modeling, VOF modeling.

1. Introduction

Bridges are crucial constituents of any nation's transportation system. As large amount of funding is involved in bridge construction, the bridge structures should be analyzed for potential failures from different perspectives. This includes effect of water flow around bridge structures, including effects of flooding and scouring on bridge stability. Nature of water flow and geometry of the channel, the size, shape, and orientation of the bridge piers and abutments, will have effects on flow around the bridge. Scaled experiments are conducted for bridge flow stability analysis in terms of flow field parameters, and measurement of drag and lift forces. In such analysis, the channel geometry, flow depth, velocity and bed roughness have their effects on stresses and thus forces on a bridge. Related experimental studies are restricted to few design variations and limited flooding conditions as excessive time and cost are tied up with such testing. These real time scaled experiments can be complemented by Computational Fluid Dynamics (CFD) simulation, which uses numerical methods to solve fluid flow problems simulating real life applications. Advanced CFD software with powerful parallel computers allow for relatively fast and inexpensive parametric studies over wide range of parameters.

Real time experiments for open channel turbulent flows were conducted by several researchers and such experimental set ups were simulated, to compare simulation outcomes with experimental results. Ramamurthy, Qu, and Vo [1] found good agreement between existing experimental and computational results, for free surface flow simulation using the VOF method. They applied a three-dimensional, two-equation k-ε turbulence model for flow simulation in a trapezoidal channel. Maronnier, Picasso and Rappaz [2] performed numerical study for two-dimensional free surface flows with the VOF method, for several cases. With the PISO algorithm, numerical results agreed well with experimental ones. Koshizuka, Tamako and Oka [3] presented a method for transient incompressible viscous flow with fluid fragmentation of free surfaces. Simulation of fluid fragmentation for collapse of liquid column against an obstacle was carried out. A good agreement was found between numerical simulation and experimental data.

In present work, the STAR-CD commercial CFD software is used to simulate scaled experiments conducted by Turner Fairbank Highway Research Center (TFHRC) at their own laboratories. These experimental data consist of drag and lift coefficients as a function of various Froude numbers, Fr, and different flooding height ratios h^*. Two-dimensional (2-D) computational model is developed in STAR-CD to represent the bridge in scaled experiments. A base case of Froude number, $Fr = 0.22$ and flooding height ratio $h^*=1.5$ is simulated using two different methods. First method is the transient Volume of Fluid (VOF) methodology, which models two-phase flow with free surface interface. Another method simulates the same base case as a single-phase closed channel, steady-state flow with slip-top-wall at the top surface instead of the free surface interface. Non-uniform velocity profile and turbulence quantities at the inlet are also used to calculate the flow field, and drag and lift coefficients of the bridge using six linear eddy viscosity turbulence models. Drag and lift coefficients from simulations are compared with experimental results. Optimum computational parameters are identified through parametric study for the time step, mesh density, and convergence criteria. The suitable turbulence model is identified in terms of accuracy and computational requirements. This model is used to simulate the base case with Froude number, $Fr = 0.22$ having thirteen different flooding height ratios h^*.


Drag Coefficients vs h* for Fr = 0.22

Lift Coefficient vs h* for Fr = 0.22

Fig. 1: Experimental drag (top) and lift coefficient (bottom) vs. h^* (for Fr=0.22)

2. Experimental Data

Scaled experiments are carried out by TFHRC for open channel turbulent flow over a six-girder bridge. Experiments are conducted with various flow conditions, the *Fr* number from 0.12 to 0.40 corresponding to flow average velocities of 0.20 *m/s* to 0.65 *m/s*, respectively, and with thirteen different inundation (flooding) height ratios h^*, from 0.3 to 3. Drag and lift forces are measured for different Froude numbers and different flooding height ratios. The case of Froude number, $Fr = 0.22$ with upstream average velocity 0.35 *m/s* is repeated four different times to measure average drag and lift coefficients, see Fig. 1. The lift force is determined by excluding the effect of buoyancy force in vertical direction.

The dimensionless flooding height ratio h^* is defined as ratio of water height from the bottom of the bridge deck to the height of the deck itself. It is an indication of how much the bridge deck is flooded: $h^*<1$ case means bridge deck is partially flooded, $h^*=1$ means bridge deck is just fully flooded, and $h^*>1$ corresponds to a case where the bridge deck is over-flooded h^* times its deck height.

3. Modeling and Simulation with STAR-CD Commercial CFD Software

The STAR-CD (Simulation of Turbulent flow in Arbitrary Regions-Computational Dynamics) is a CFD analysis software, which uses finite volume code to solve fluid flow governing equations for steady state or transient problems. Since only 3-D computational model could be simulated in STAR-CD, a single- or double-cell thick layer in the 3rd dimension with the symmetry boundary conditions on opposite boundary planes is used for two-dimensional (2-D) simulation. The structural hexahedral (six rectangular faces) mesh is used to subdivide the current computational domain. In the first method air-water domains are simulated as two-phase problem using transient volume-of-fraction (VOF) methodology. The VOF method uses a volume of fraction variable to capture air-water, free-surface interface. In another method, only water domain is simulated as closed duct steady-state flow using the slip-wall at the top surface instead of the open channel free-surface, called here as *slip-top-wall* method. The boundary and other operating conditions are the same or similar as in the VOF free-surface modeling.

3.1. Transient Volume-of-Fraction (VOF) Simulation

Turbulent free surface flow is modeled using the VOF methodology with bridge deck geometry and operating conditions similar to the available experimental testing. The computational domain is long enough so that inlet and outlet boundary conditions do not have significant effect on flow around the bridge. The whole computational domain is discretized with non-uniform mesh and is two cells thick in Z-direction as explained above. The mesh is denser near the bridge and at the free surface. The initial free surface position is defined by assigning separate two-phase, a light (air) and heavy (water) fluid domains, see Fig. 2. The boundary conditions assigned to the computational model are also presented in Fig. 2.

The turbulence kinetic energy and turbulence dissipation rate at the inlet are taken as specified by our collaborators in Argonne National Laboratory.

Experimental data are simulated in STAR-CD using transient VOF methodology with time step of 0.01 *sec*. The transient VOF simulation was sensitive to its computational parameters (particularly the computational time steps), which has an effect on the final simulation outcome. Sensitivity of VOF transient simulation is studied for different time steps, channel dimensions and boundary conditions.

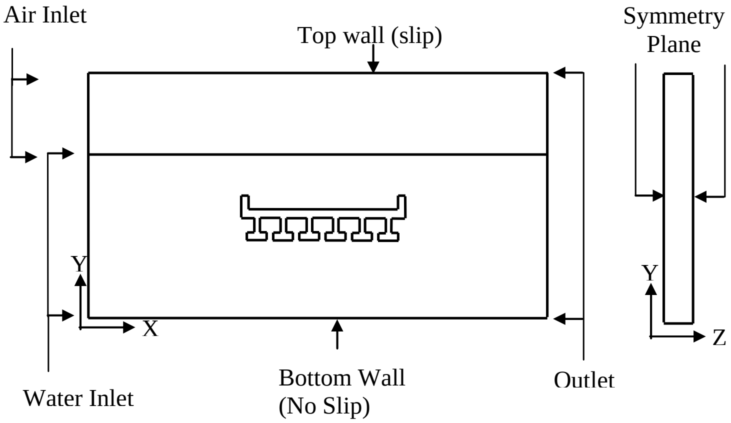


Fig. 2: Boundary conditions for VOF methodology

Open channel turbulent flow is simulated as a transient case to compare simulation outcome with

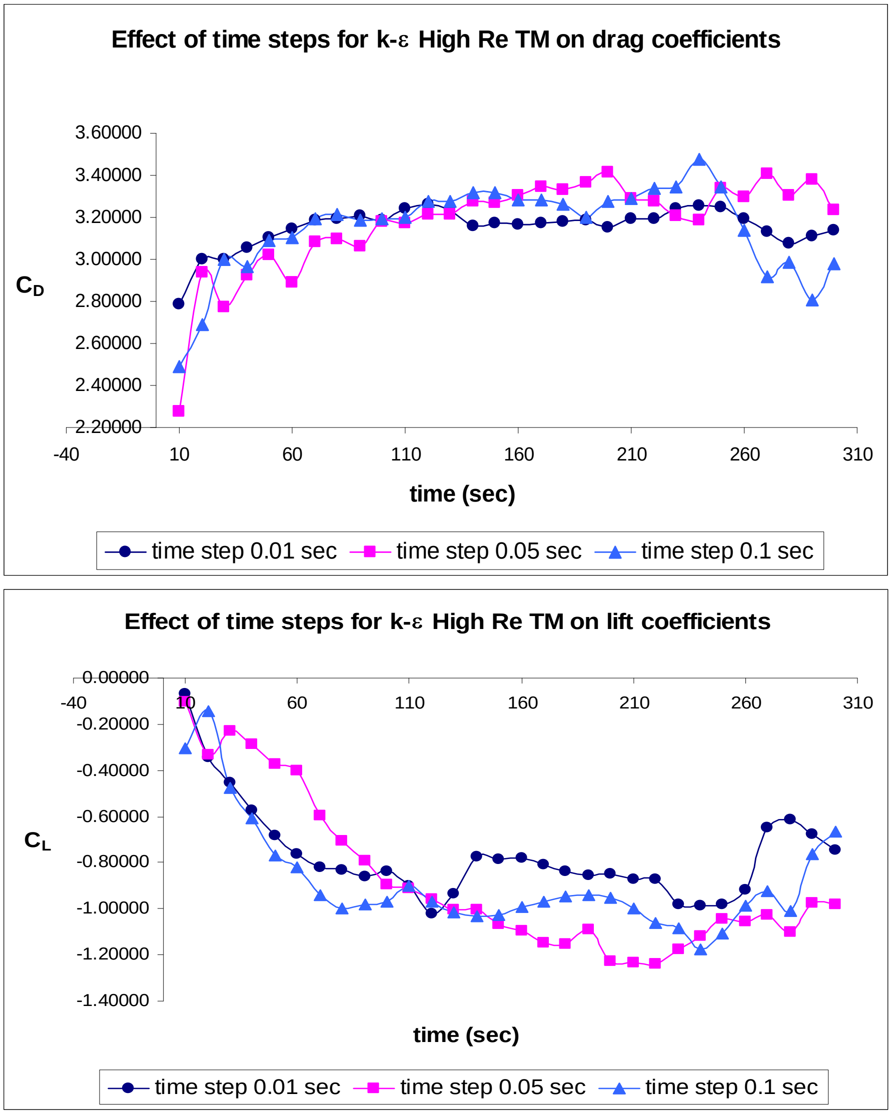


Fig. 3: Effect of time steps for *k-ε* High-Reynolds turbulence model on drag (top) and lift coefficients (bottom)

experimental results, with a hope that the transient case comes to a steady-state solution. The simulation is said to be in steady state when the free-surface profile is either steady or quasi steady with some fluctuations. Open channel flow is simulated with different time steps of 0.01, 0.05, and 0.1 *sec* for the standard case of *k-ε* High-Reynolds turbulence model. The free-surface development and drag and lift force coefficients are calculated for real (or live) flow time (not to be confused with computational time) up to 300 *sec*. However, the simulation run time is much longer than the transient VOF live time (so simulation cannot be run in live-time). The smaller the time step used for simulation, the more time it takes to run transient VOF simulation. The effect of time steps on the drag and lift coefficients are presented in Figures 3.

The results for the time steps of 0.05 *sec* and 0.1 *sec* are alternating around 0.01 *sec* time step. It is demonstrated that the time step of 0.01 sec is suitable for desired accuracy of our simulation. The free-surface development and flow around bridge is sensitive even for time step of 0.01 *sec*. Stability for free surface and flow around bridge need to be studied for smaller time steps and for longer simulation times than in this initial study. The VOF simulation is computationally expensive in terms of time and cost.

The upstream and downstream lengths of the computational model are chosen in such way that the inlet and outlet boundary conditions do not significantly influence free surface development, flow around the bridge, and forces on the bridge. The influence of channel dimensions on drag and lift coefficients is studied for different downstream channel lengths and under-bridge water depths. Decrease in exit length promote the simulation instability since the outlet boundary conditions have a convincing influence on flow around the bridge, and in turn on drag and lift coefficients. Decrease in under-bridge water depth by maintaining the same flooding height ratio, *h**, has a noteworthy effect on flow around the bridge..

Table 1: Comparison of drag and lift coefficients using VOF simulation and experimental results

Turbulence model	$C_{D\ avg}$	$C_{D\ exp}$	$C_{L\ avg}$	$C_{L\ exp}$
k-ε High Re	3.17	1.98	-0.83	-1.04
k-ε RNG	2.77	2.02	-1.39	-0.73
k-ω STD High Re	4.69	1.99	-0.55	-1.00
k-ω STD Low Re	10.91	1.97	-0.29	-0.60
k-ω SST High Re	3.03	1.98	-1.15	-1.10
k-ω SST Low Re	4.03	1.96	-0.91	-1.07

3.2. Influence of Different Turbulence Models on Simulation Results

Outcome of the transient VOF simulation strongly depends upon the turbulence models used, since turbulence near the bridge influence flow around the bridge and in turn influence the drag and lift coefficients. For six different turbulence models used in the simulations, the free-surface development was different, in addition to being unstable and slow.

For a fixed inlet h^* value of 1.5, the average h^* over the computational domain varied, with different turbulence model used, from 1.25 to 1.45, giving an average value of 1.34 with standard deviation of 0.08 (when the anomalous behavior of k-ω STD Low-Re is rejected as an outlier). All simulations, using family of k-ε turbulence models, k-ε High-Reynolds and k-ε RNG, are converging to steady state, and predicting drag coefficients close to each other. Similarly, all the k-ω turbulence models are converging to steady state, with k-ω SST High-Reynolds coming closer to experimental results. Also, all turbulence models are oscillating towards steady state, with k-ω SST High-Reynolds predicting closest to experimental lift coefficients. Still, final results are unstable and vary with different simulation parameters.

Table 2: Computational parameters for transient VOF and steady-state slip-top-wall method

Computational parameters	***VOF method***	***Slip-top-wall method***
Inlet velocity, V [m/s]	0.35	0.35
Turbulent kinetic energy k $[m^2/s^2]$	0.00125	0.00125
Turbulent dissipation rate ε $[m^2/s^3]$	0.000175	0.000175
Solution method	Transient VOF	Steady State
Solver method	Algebraic Multigrid (AMG)	Algebraic Multigrid (AMG)
Solution algorithm	SIMPLE	SIMPLE
Relaxation factor	0.3 for pressure; 0.7 for momentum, turbulence	0.3 for pressure; 0.7 for momentum, turbulence
Differencing scheme	MARS	UD
Convergence criteria	10^{-2}	10^{-6}
Time step, Δt *[s]*	0.01	N/A

3.3. Steady-State Slip-Top-Wall Simulation and Comparison with Transient VOF Simulation

In order to overcome the simulation instability of VOF method, the water domain is simulated with a fixed slip-top wall, instead of the free-surface, thus resulting with the single-phase steady state simulation, called here as *slip-top-wall* method. This steady state approach is in many ways concept-wise similar to the transient VOF simulation, but provides a simplified and faster way to calculate flow parameters and stresses around the bridge. The boundary conditions and relevant computational parameters are the same or similar as in the VOF free-surface modeling, see Fig. 4 and Table 2.

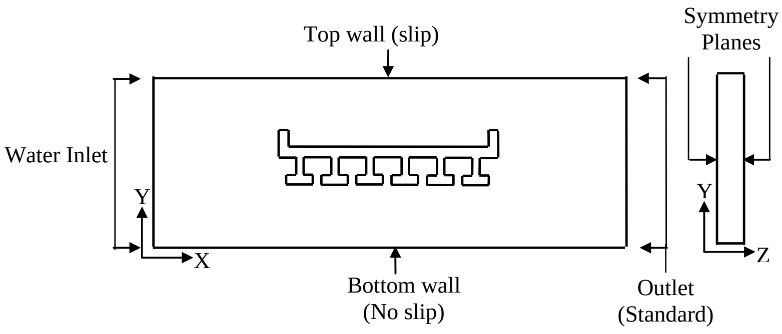


Fig. 4: Boundary conditions for steady-state slip-top-wall method.

The results of simulating steady-state slip-top-wall models as compared to VOF, are fast, stable, more accurate and converged within a small tolerance limit of 10^{-6}. These slip-top-wall models show better agreement with experimental drag and lift coefficients than transient VOF simulations. The slip-top-wall models are analyzed for mesh sensitivity, influence of convergence criteria and effect of inlet turbulence parameters on final simulation outcome.

The base case with Fr=0.22 and h^*=1.5 is simulated with-steady state model for different turbulence models. The comparison with each other and experimental results, between the VOF and steady-state, is presented in Fig. 5. Drag and lift force coefficients for VOF simulations are obtained after averaging over 200 s to 300 s. The turbulence intensity and mixing length for both VOF and steady state

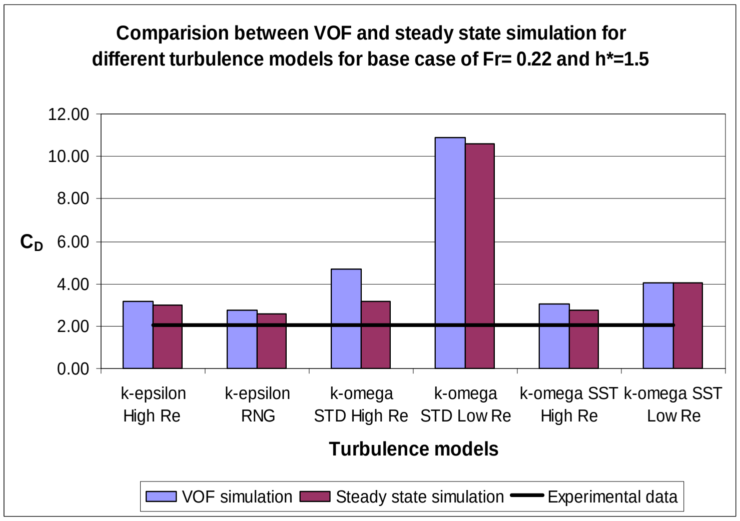


Fig. 5: Comparison of drag coefficients for different turbulence models, between transient VOF and steady-state simulation, for *Fr* = 0.22 and *h**=1.5. NOTE: The k- ω STD Low-Re case should be rejected as an outlier.

simulation is 8.25 % and 41.5 *mm*, respectively. Out of these turbulence models, the *k-ε* RNG model using the steady-state slip-top-wall method is predicting the experimental results better than the other models, still 27% higher drag and 54 % lower lift coefficient, the latter due to extremely small reference value comparable to uncertainty of measurements and simulation. The *k-ε* RNG turbulence model is selected as a base model to simulate experimental results with different *h**, see next section.

3.4. Comparison Between Simulation and Experimental Results for different Flooding Inundation Ratios

The CFD simulations are run with *k-ε* RNG turbulence model for thirteen different flooding height ratios with *h**= 0.3 to 3, see Fig. 6. The simulations (for *Fr*=0.22) predict higher drag coefficients and lower lift coefficients for wide range of submerged deck (*h**>1); however for smaller values of dimensionless inundation ratio, when the bridge deck is partially submerged (*h**<1), it is not the case. Due to instability of free-surface flow for *h**<1, the fluctuation of experimental results makes them unreliable. However, the steady-state slip-top-wall simulation results are stable for *h**<1 since the unstable free surface is modeled as a fixed slip-top-wall. The simulation results show interesting trends, but further investigations are needed to make the modeling and simulation more realistic.

4. Conclusion

The simulations of water flow over bridge using STAR-CD, a commercial CFD software, have been performed to compare and validate the drag and lift force coefficients with limited experimental data. In this initial work, only 2-dimensional simulations have been carried out with the turbulence closure models using the Reynolds Averaged Navier-Stokes (RANS) equations. The obtained average results have been up to 50% bigger than the limited experimental data available, but also unstable and varied widely with different simulation parameters.

The VOF free-surface simulation results, demonstrate rather slow and unstable transient development of the free surface with some initial fragmentations and interactions between the two water-air phases.

The simulation results have been strongly dependent on the turbulence models used. Not only that the drag and lift force coefficients varied significantly, but also the free-surface development and resulting average inundation dimensionless ratios, *h**, have been varying with the turbulence model used, with some models resulting in anomalous slope of the free-surface and increase of the *h** value downstream.

It is observed that transient VOF free-surface simulations are much more unstable than usual steady-state single-phase simulations, and thus VOF may amplify solution instability and compromise accuracy with regard to other modeling and solution parameters, namely spatial and time discretization size and non-uniformity transitions, and especially turbulence modeling. Further refinements and optimization of non-uniform mesh and time steps, as well as under-relaxation and other solution parameters should be tested and optimized in order to improve solution stability and accuracy. Furthermore, 3-D simulations and the Large Eddy Simulations (LES) or even Direct Numerical Simulations (DNS) may be employed in future, since more powerful computing recourses are becoming available.

Acknowledgments:

The authors like to acknowledge support by Dean Promod Vohra, College of Engineering and Engineering Technology of Northern Illinois University (NIU), and Dr. David P. Weber of Argonne National Laboratory (ANL); and especially the contributions by Dr. Tanju Sofu, and Dr. Steven A. Lottes of ANL, as well as financial support by U.S. Department of Transportation (USDOT) and computational support by ANL's Transportation Research and Analysis Computing Center (TRACC).

References:

[1] Ramamurthy A.S., Qu, Junying, Vo, D., VOF model for simulation of a free overfall in trapezoidal channel, *Journal of Irrigation and Drainage Engineering*, Vol.132, No.4, 2006, pp 425-428.

[2] Maronnier V., Picasso, M., Rappaz, J. (1999) Numerical simulation of free surface flows, *Journal of Computational Physics*, Vol. 155, 1999, pp 439-455.

[3] Koshizuka, S., Tamako, H., Oka, Y., A particle method for incompressible viscous flow with fluid fragmentation, *Computational Fluids Dynamics Journal*, Vol.4, 1995, pp 29-46.

[4] Patil, S., *Computational Fluid Dynamics for Open Channel Flow over Bridge Decks under Various Flood Conditions*, M. S. Thesis, Northern Illinois University, 2008

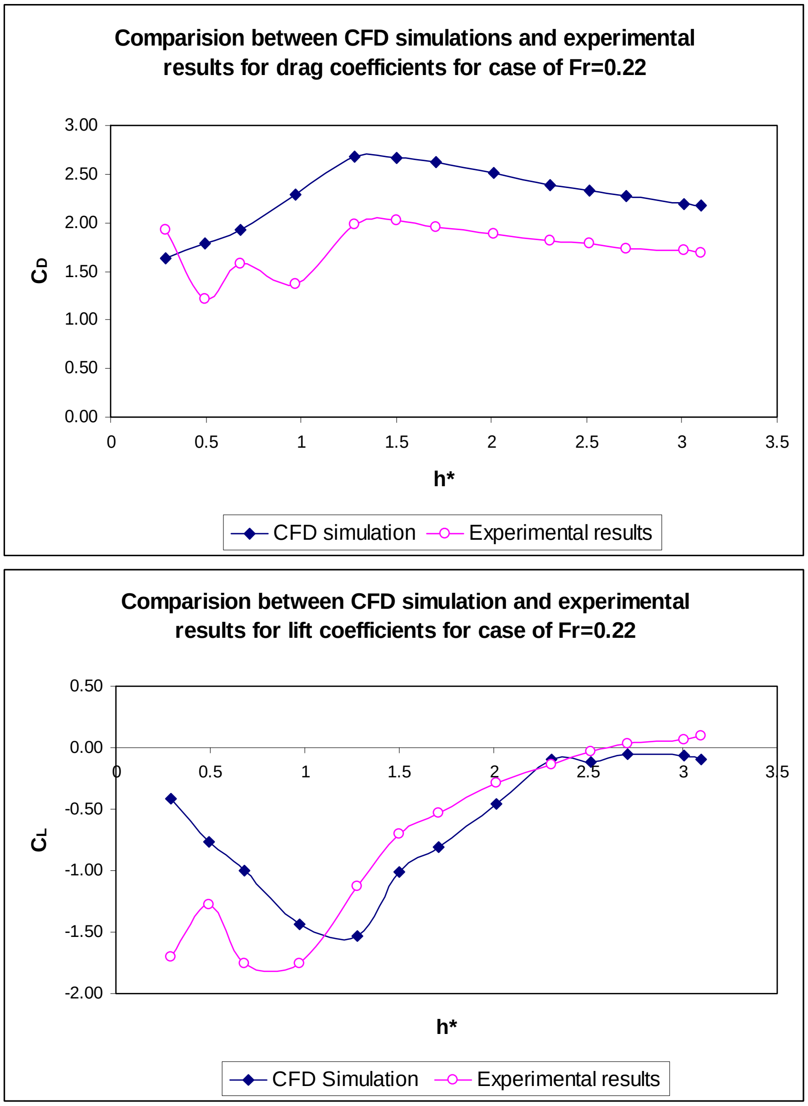


Fig. 6: Comparison of drag (top) and lift coefficients (bottom) between CFD simulation (k-ε RNG) and experimental data (for Fr=0.22)