Report as of FY2006 for 2006KY60B: "Feasibility of using 3D CFD Models in Simulating Hydrodynamics in Dam Design/Rehabilitation"

Publications

- Conference Proceedings:
  - Shao, Z.S. and S. A. Yost, 2007, Toward Using a Three-Dimensional Numerical Model for Simulating Hydrodynamics Near a Dam for Constructing the Rating Curve, in Proceedings of the Kentucky Water Resources Annual Symposium, Kentucky Water Resources Research Institute, Lexington, Kentucky, p 9-10.

Report Follows

Problem and Research Objectives

Lock and Dam 9, located on the Kentucky River just downstream from the Valley View Ferry on KY 169, was constructed by the United States Army Corps of Engineers (USACE) between 1902 and 1907 (Figure 1). It provides water storage for local water suppliers and maintains a pool for recreational and navigational use along the river. The Kentucky River Authority recently decided to stabilize and renovate Lock and Dam No. 9 to secure the structure against failure and major leak losses, and to add storage capacity. The existing dam consists of three main components: the main dam, the auxiliary dam and the navigation lock chamber (Figure 1). A concrete-filled, cellular sheet pile structure was proposed for the renovated dam. In the proposed design, eight circular cells positioned across the river will be connected with "arc cells." It is important for the new structure to maintain the same hydraulic characteristics (discharge-stage relationship, permanent pool elevation) as the existing dam. Therefore, a hydraulic study of the proposed design is important to make sure that the new construction will not change the flow pattern on the existing river system.

Figure 1 2001 Aerial Photo of Lock & Dam 9 (Source: FMSM Engineers Inc.)

Traditional approaches for studying hydrodynamics near structures involve field measurements and setting up laboratory physical models. However, laboratory models poorly satisfy hydraulic similarity with the original physical structure because some dominant non-dimensional parameters can not be represented well in a laboratory model. Also, lab models do not provide much flexibility. Numerical models are more flexible and can be used to simulate several possible scenarios without much extra effort. FLUENT is a standard industry computational fluid dynamics code used in a wide range of flow simulations. It has been used for applications ranging from inkjet printers to aerospace. However previous applications of FLUENT in water resources have been fairly limited and application of FLUENT to complicated large scale hydraulic structures (such as a dam) has not been widely studied. The objective of this study was to develop a three-dimensional numerical model using FLUENT to simulate flow near the dam site and ultimately to compare the rating curves of pre- and post-construction conditions.

Methodology

In this study, a three-dimensional numerical model was developed for the renovated Lock & Dam 9 post-construction conditions using the FLUENT program. The procedures used to develop the model are described below.

a. Simplify Geometry - The geometry of the post-construction condition of Lock & Dam 9 was simplified and input in the mesh generator - Gambit. Geometries of the existing dam and the proposed design were obtained from FMSM Engineers Inc. The total study section length was about 1,000 feet. The side slope of the river bank was treated as uniform along the study section. The model included deposited sediments near the river bed, but the geometry of the deposit was simplified as a flat bottom
b. Mesh Generation. A mesh generator – GAMBIT, embedded as a preprocessor in FLUENT software, was used to generate a three-dimensional mesh for the study area. The total number of resulting nodes was 480,000 and the total number of cells was 1.3 million. The computational domain upstream from the cellular dam was meshed with a uniform hexahedral grid while the rest of the dam was meshed with a non-uniform tetrahedral grid (Figure 2).
c. Unsteady Model. An unsteady state simulation was chosen for the overall calculation. The time step size varied between 0.1second to 0.5 second.
d. Turbulent Model. A k-ε model was selected to simulate the turbulence near the dam (k-ε is a standard turbulent model widely used in industry). With different treatment and calibration of wall function, the k-ε model simulates turbulence with large Reynolds number very well.
e. Free Surface Simulation. A Volume of Fraction (VOF) scheme was chosen to simulate the free surface feature of the simulated flow. VOF can handle long waves, short waves, and breaking waves.
f. Appropriate boundary conditions. A given mass flow rate was specified as the inflow boundary condition upstream of the computational domain. A fixed depth was also specified in FLUENT at the upstream boundary to initiate the calculation.

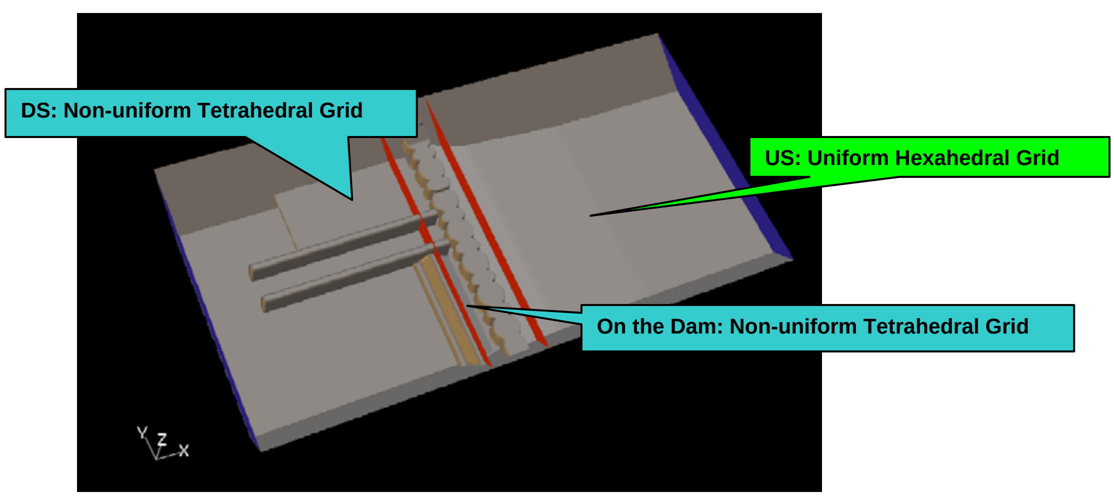

Figure 2 Computational Grids generated in Gambit

Principal Findings and Significance

Using commercial software for analyzing hydrodynamics around a dam requires a learning curve for both the software and the grid generation. FLUENT was a particularly complex package with many attributes that needed to be understood so that the model could be used for the chosen application. While the investigators found that this tool is well worth the learning effort, consulting companies who are driven by tight time-lines may never schedule the time or commit sufficient resources to fully utilize the numerical tool. It would likely require $50,000 to $80,000 to generate a final rating curve comparison. Of this, 75% of the resources would be a one time upfront cost to learn the details of the commercial software. After a full year of intense learning and investigation, well beyond the actual duration of the project, we are just now obtaining meaningful preliminary results. With an appropriate numerical grid developed compatible with FLUENT, simulation of flow across the dam is now possible. Figures 3 and 4 show the propagation of the free surface along the channel at time = *90s* and *290s*. These results are not steady-state, but steady-state conditions will be needed in order to generate a point for the stage discharge rating curve.

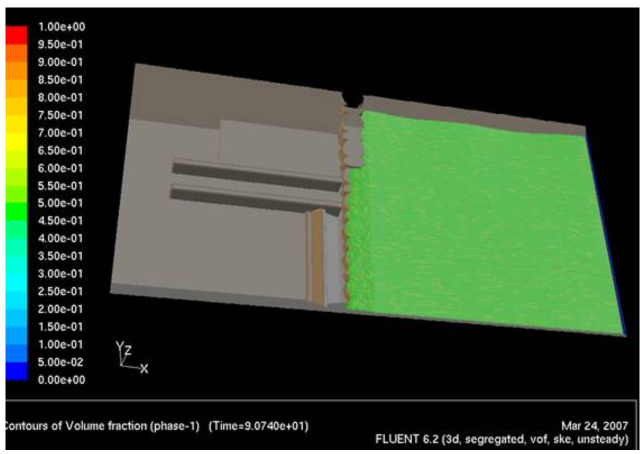

Figure 3 Free Surface at t=90s

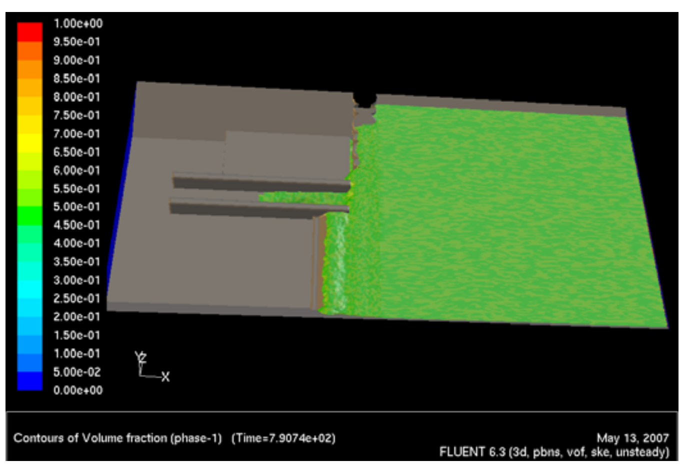

Figure 4 Free Surface at t=290s

Figure 5 shows the dynamic pressure contour at time = *290s*. Even though the results are unsteady, the pressure around the dam is nowhere near hydrostatic. While anyone familiar in fluid mechanics would know this to be true, many consulting companies still use simple models (ie, HEC-RAS) to attempt to generate stage-discharge information. The results from those studies are, at best, suspect because of the complex dynamics involved. The investigators hope to compare the steady state results of the FLUENT model to that of HEC-RAS to gain understanding of the errors due to the oversimplification.

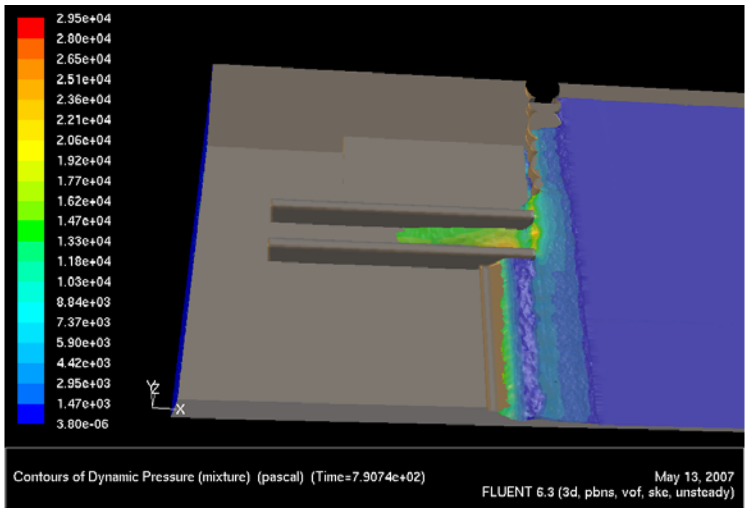

Figure 5 Dynamic Pressure at t = 790s

Figure 6 shows the sobering reality of using advanced numerical tools. While the HEC-RAS model's computational time is a fraction of the actual simulation time, it is just the opposite for models like FLUENT where the computational time is orders of magnitude greater than the simulation time. Parallel processing is an absolute necessity for making these advanced models useful for practicing engineers. Fortunately, the University of Kentucky has a supercomputing center that ranks in the top 10 public facilities in the country, and the top 200 in the world.

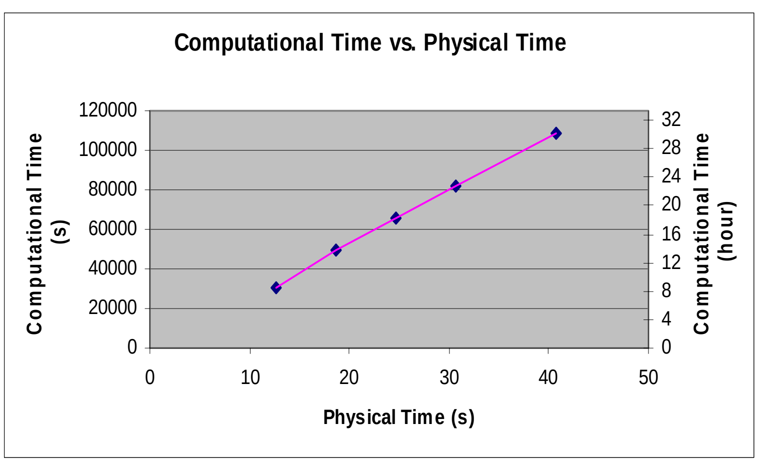

Figure 6 Computation time vs. Physical