

Three Dimensional Numerical Simulation of Turbulent Flow Over Spillways

Latif Bouhadji

ASL-AQFlow Inc., Sidney, British Columbia, Canada

Email: *lbouhadji@aslenv.com*

ABSTRACT

Turbulent flows over a spillway structure are investigated using computational fluid dynamics (CFD). Simulations are carried out to validate two and three dimensional CFD models in these structures. The numerical results are compared to available experimental data published by the US Army Corps of Engineers (USACE).

1 INTRODUCTION

Computational Fluid Dynamics, commonly known by the acronym 'CFD', is a branch of Fluid Mechanics that resolves numerically, fluid flow problems. The physical laws governing a fluid flow problem are represented by a system of partial differential equations regrouping the continuity equation, the Navier-Stokes equations and any additional conservation equations. The numerical analysis resolves these equations by accurate and complex numerical schemes. A program or code, where the numerical algorithm is implemented, is then solved on a computer. The faster the computer, the faster are the computations. Nowadays, most of CFD codes use parallel computation in order to resolve a flow problem faster by 'sharing' the calculation and the memory required among several computers. As the performance-to-cost ratio of computers has increased at a spectacular rate in the last decade and shows no sign of slowing down, CFD is considered more often as a key industrial tool. The main attraction in using Computational Fluid Dynamics, resides in its ability to investigate physical fluid systems and provide a large amount of data more cost effectively with more flexibility and more rapidly than with experimental procedures. CFD is able to overcome many difficulties that the physical models (if available) encounter to measure flow quantities and phenomena in inaccessible flow regions or due to disturbances caused by the instrument

and/or by the experimental environment. In this paper, the CFD technique is applied to investigate turbulent flow over a spillway structure (Ogee profile) under increased water head conditions. Note that the CFD technique has been applied to investigate several spillway structures in Australia (see ref [1]).

2 NUMERICAL IMPLEMENTATION

The numerical simulations are performed using the unstructured, parallel solver CFX 5.7 and the mesh is generated by using ICEM-CFD hexa. Both codes are from Ansys Inc. The governing differential equations are integrated over control volumes defined by the grid, such that the relevant quantity (mass, momentum, energy) is conserved in a discrete sense for each control volume. The diffusive and advective fluxes and the source terms in the volume integrals are then discretized using various techniques. The discretization method must be selected to ensure both adequate accuracy and numerical stability. For the advection terms, CFX 5.7 provides three different schemes: a first order Upwind Differencing Scheme (UDS), a numerical advection correction scheme and a high resolution scheme. The first order UDS is very robust (numerically stable) but suffers from numerical diffusion and is usually used as a first step to get an initial fluid flow solution before applying a higher resolution scheme. In the numerical advection correction scheme, one can specify a blend factor between 0 and 1 to fix a level of accuracy. A blend factor of 0 is equivalent to the first order advection scheme and a blend factor of 1 uses second order differencing which is more accurate but less robust. One can start a complex simulation by using a blend factor of 0 and gradually increase it towards 1. Usually a blend factor of 0.75 is sufficient. The high resolution scheme computes the blend factor throughout the domain based on the local solution field. In flow regions where gradients are low, the

blend factor will be close to 1. In flow regions where gradients are steep, the blend factor will be close to 0 to maintain robustness. The high resolution scheme was selected for the present study as a first step and a blend factor of 1 was selected in a final calculation. No difference has been found between using a blend factor of 1 and the high resolution scheme (in regions of interest). All computations were performed on ASL's parallel computing facility.

3 TURBULENCE MODELLING

Resolution of the instantaneous fluctuating flowfield in turbulent flows is not feasible for complex flows. Engineering methods implemented in CFD rely on the numerical solution of the Reynolds-averaged Navier-Stokes (RANS) equations in conjunction with turbulence models of varying degrees of complexity, ranging from algebraic eddy viscosity to Reynolds stress models. In the eddy viscosity models, such as the basic $k - \epsilon$, the RNG $k - \epsilon$ or the $k - \omega$ models, the Reynolds stresses are linearly related to the mean velocity gradients in a fashion similar to the relationship between the stress and strain tensors in laminar Newtonian flows. In Reynolds stress turbulence models (RSM), the eddy viscosity hypothesis is not invoked. Instead, a transport equation is defined for each component of the Reynolds stress tensor. This model provides a conceptually more correct representation of turbulence characteristics such as anisotropy and the effect of extra strains, but is computationally intensive and difficult to converge in complex configurations. As a result of the substantially lower computational effort required, the $k - \epsilon$ model is still one of the most commonly used turbulence models for the solution of practical engineering flows. There is, however, a large amount of evidence that though the $k - \epsilon$ model reproduces qualitatively many of the important flow features, it is not totally satisfactory in some complex flow situations, particularly those involving flow separation. In this work, the shear stress transport (SST) based $k - \omega$ turbulence model has been used. This model is designed to give a highly accurate prediction of flow separation under adverse pressure gradients. All solid walls are treated with the scalable wall functions.

4 THE MULTI-PHASE MODEL

CFX 5 provides two multi-phase models to simulate multiple fluid streams, bubbles, droplets, solid particles and free surface flows: the Eulerian-Eulerian multiphase model and the Lagrangian Partical Tracking multiphase model. The Eulerian-Eulerian model

has two sub-models: the homogeneous model and the inter-fluid transfer model or inhomogeneous model. In the inhomogeneous model, each fluid has its own flow field. The fluids interact via interphase transfer terms and there is one solution field for each separate phase. The homogeneous model, which is used in this study, may be seen as a limiting case of the Eulerian-Eulerian multiphase flow in which the interphase transfer rate is very large. This results in all fluids sharing a common flow field and this stays valid in a flow under gravity, where the phases have completely stratified, for example a free surface flow where the interface is well defined. In this case, volume fractions of the phases are equal to one or zero everywhere, except at the phase boundary and a single velocity field can thus be used.

5 RESULTS

Figures 1 to 3 show the topology of the Ogee profile with type 2 piers [ref 2, 3] and a close up of the mesh used. Symmetry boundary conditions are used at the front and back of the computational plane. The design head H_d is equal to 10 meters. Three different cases are studied with an upstream water head H equal to 5, 10 and 13.3 meters. The results of a two dimensional case (Ogee profile with no piers) are compared to available data [ref 2] in Figures 4 and 5. The upper nappe and crest pressure profiles show a good agreement with the USACE measurements. Figure 6 shows the distribution of the velocity vectors at a particular upstream head equal to 13.3 meters.

A three dimensional study, where the piers are included, with an upstream water head H equal to 5 meters was carried out. Figure 7 shows the velocity field at the center bay section and the velocity vector field at the crest longitudinal plane. Figure 8 shows the water volume fraction along the center bay, the pier and at the crest cross section. The red color corresponds to a water volume fraction equal to 1 (water), the blue color corresponds to a water volume fraction equal to 0 (air). The interface air/water is clearly noticeable and displays a different evolution along the center bay, pier and crest cross section. A three dimensional representation of the nappe surface, shown in figure 9, corroborate these observations. The nappe profiles at the center bay and along type 2 piers, displayed in Figure 10, show again a very good agreement with the experimental data. The crest pressure profiles along the center bay and pier are also displayed in the same figure. In Figure 11, some of the results for the $H/H_d = 1$ 3D case are displayed. The pressure field on the spillway surface and at the center bay section is shown. It is interesting to notice that the lowest pressure field

is located around the bottom of the pier at the front, halfway to the crest longitudinal section. The comparison of the nappe profile at center bay and along the piers with the USACE data is shown in the same figure. Figure 12 shows the results for the last 3D case studied: $H/H_d = 1.33$. The velocity vectors at the crest longitudinal section and the nappe profiles at the center bay and along the pier are displayed. The agreement with the experimental data is once again very good.

6 CONCLUSION

Two and three dimensional models of a Spillway structure were tested by CFD. The numerical results were compared to USACE experimental data and showed a very good agreement. The simulations demonstrated the strong potential of CFD to provide reliable results for spillway flow measurement projects.

7 REFERENCES

- [1]-D. Ho, K. Boyes, S. Donohoo and B. Cooper, *Numerical Flow Analysis For Spillways*, 43rd ANCOLD Conference, Hobart, Tasmania, 24-29 October 2003.
- [2]-US Army Corps of Engineers, *Hydraulic Design of Spillways*, Published by the Dept. of the Army, Washington, DC 20314-1000, January 1990.
- [3]-US Dept. of The Interior, *Design of Small Dams*, Water Resources Technical Publication, 2nd Edt, 1973.

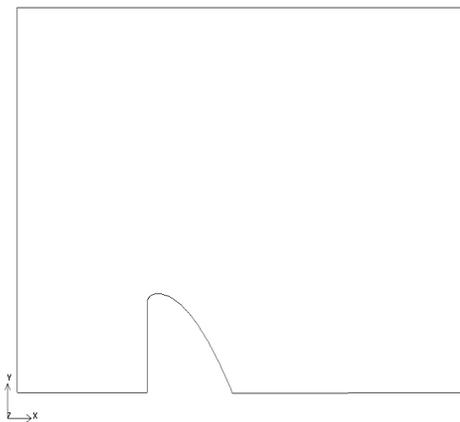


Figure 1: The two dimensional computational domain of the Ogee Spillway with no piers.

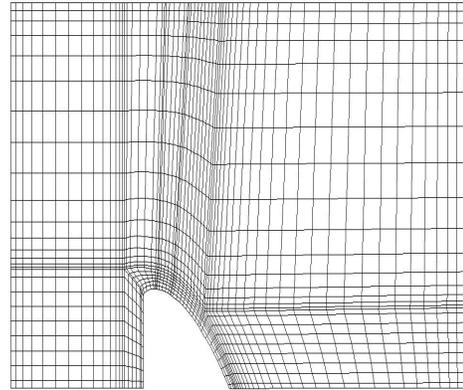


Figure 2: 2D coarse mesh.

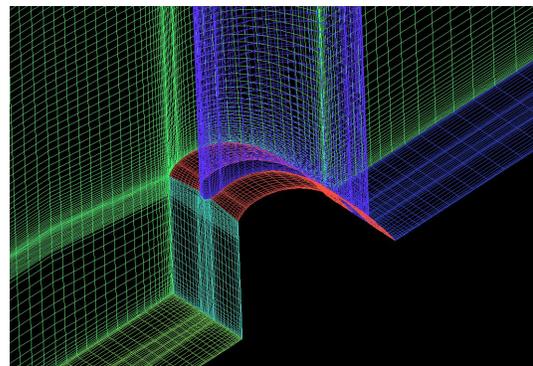
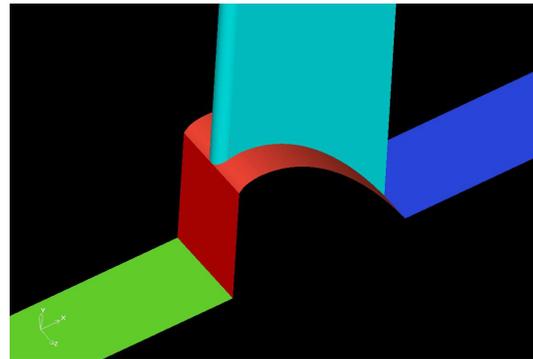


Figure 3: Topology and Mesh of the 3D Ogee Spillway with type 2 Piers.

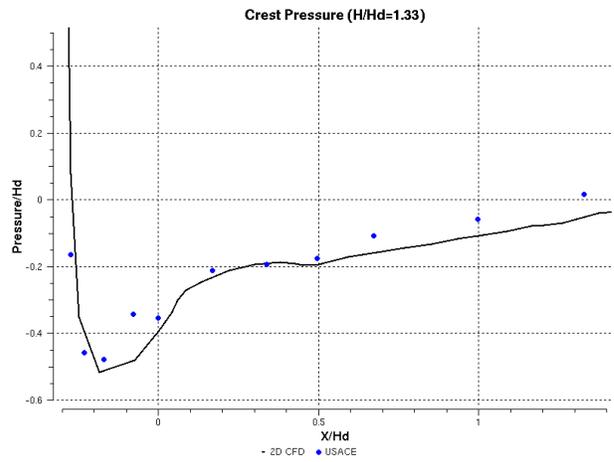
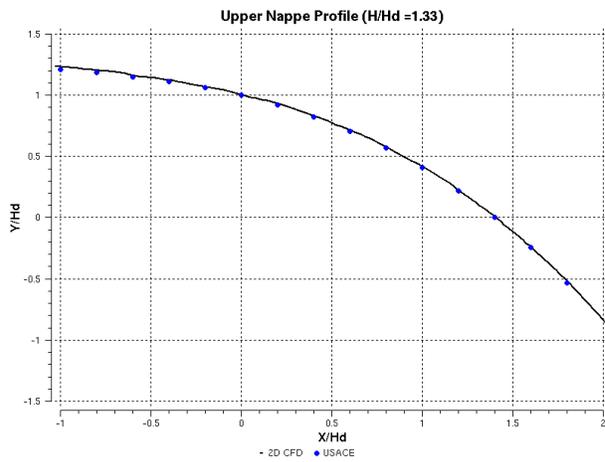
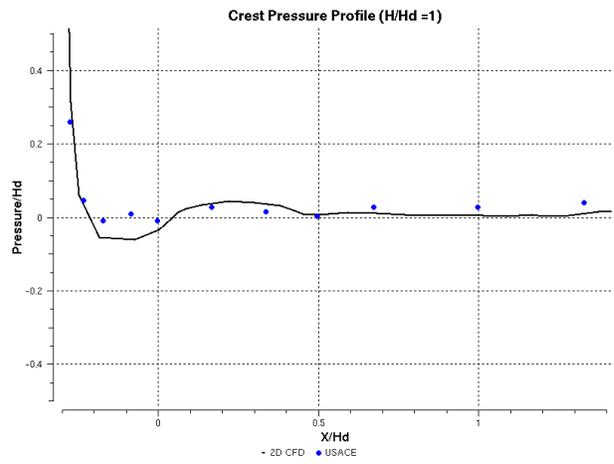
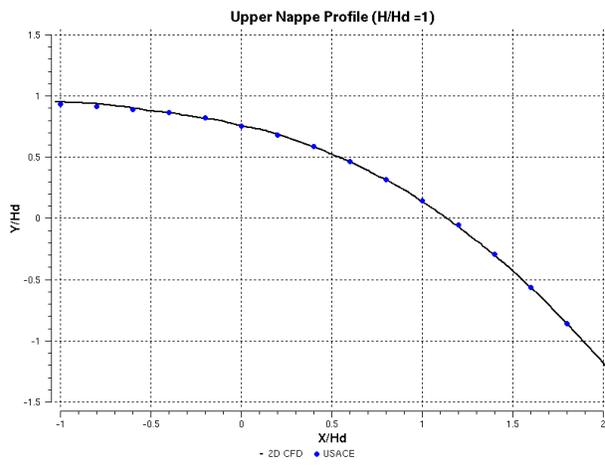
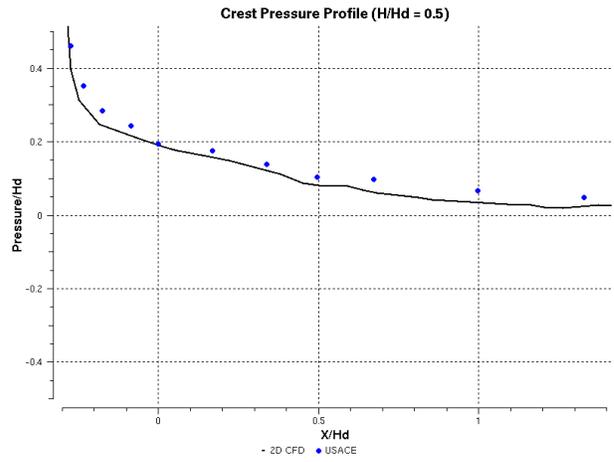
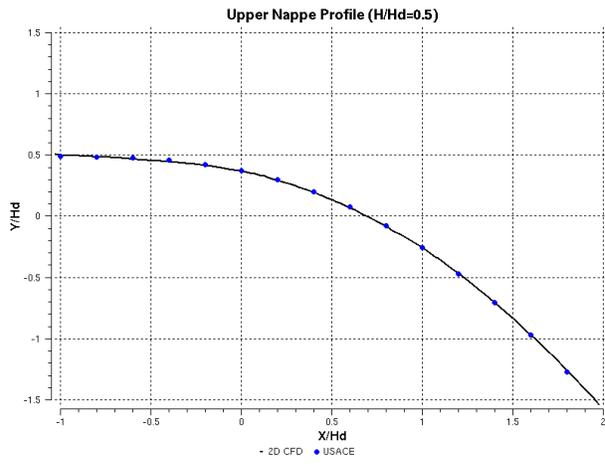


Figure 4: Two Dimensional case (no piers). Comparison of the Upper Nappe Profile for three different heads.

Figure 5: Comparison of the Crest pressure Profile for three different heads. Two Dimensional case (no piers).

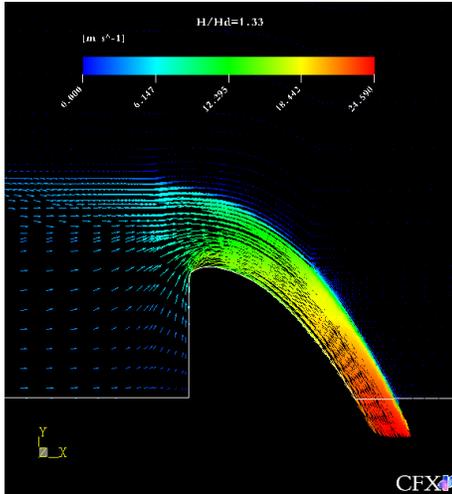


Figure 6: $H/H_d = 1.33$ 2D case. Velocity Vectors.

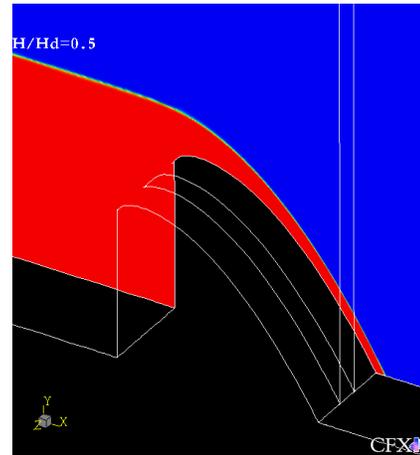


Figure 8: $H/H_d = 0.5$ 3D case. Water Volume Fraction at the center of bay, along piers and at the crest cross section. The red color corresponds to a water volume fraction equal to 1 (water), the blue color corresponds to a water volume fraction equal to 0 (air).

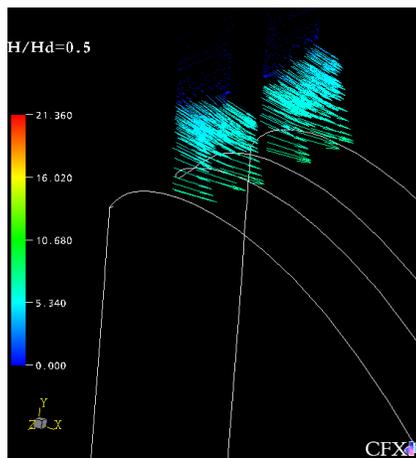
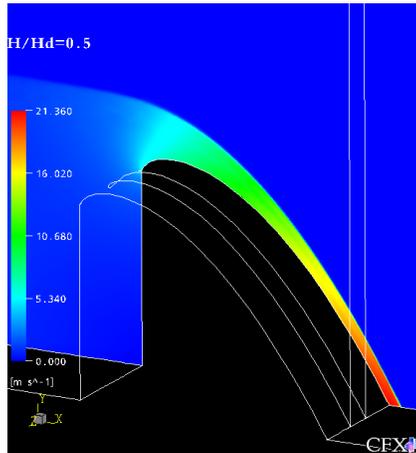


Figure 7: $H/H_d = 0.5$ 3D case. Velocity field at the center bay and Velocity vectors at the crest of the spillway.

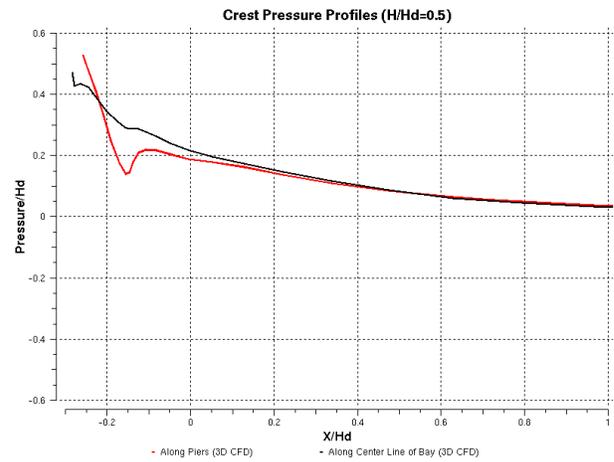
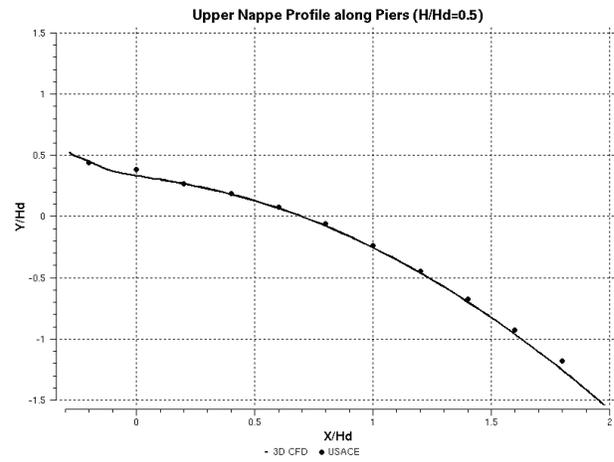
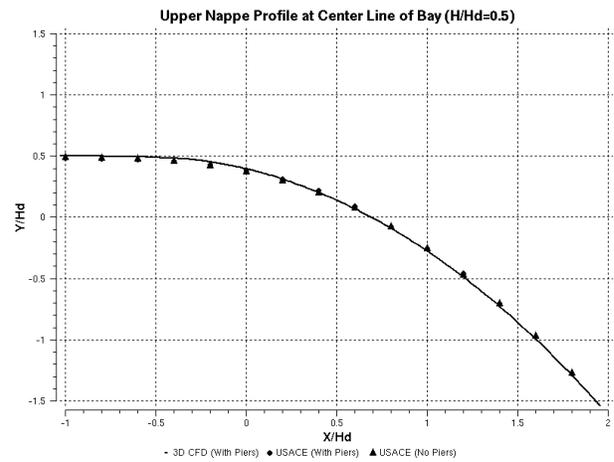
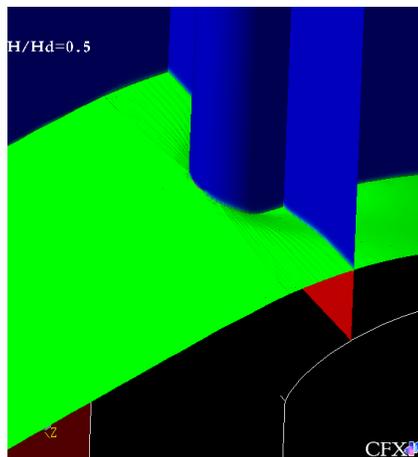
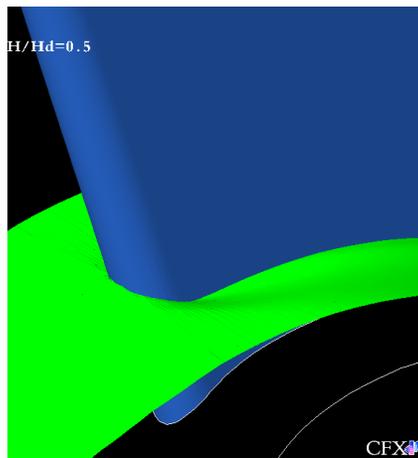
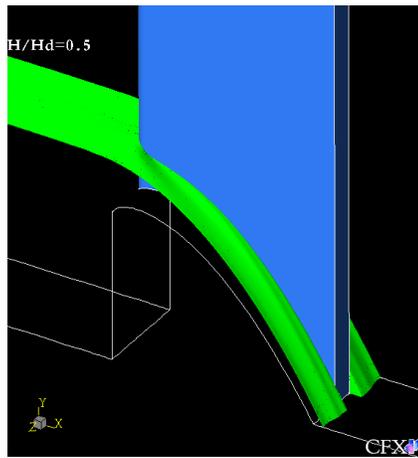


Figure 9: $H/H_d = 0.5$ 3D case. Upper nappe profile along center bay and piers.

Figure 10: $H/H_d = 0.5$ 3D case. Comparison of the Upper Nappe Profiles with USACE data at the center line of bay (top), along piers (middle). Crest Pressure profiles along the center line of bay and pier (bottom).

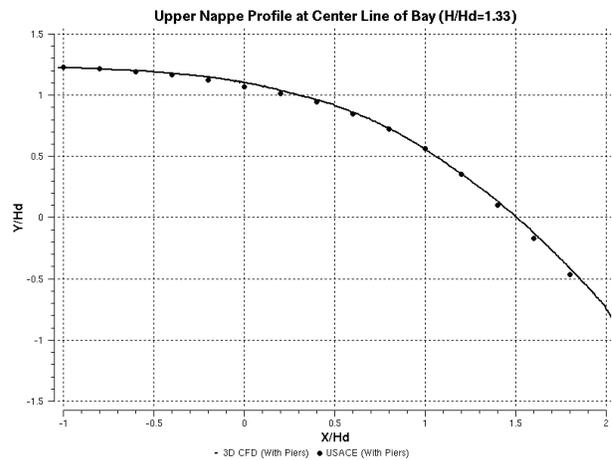
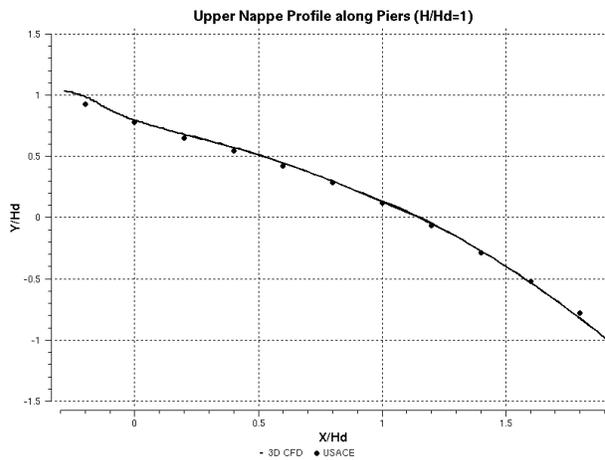
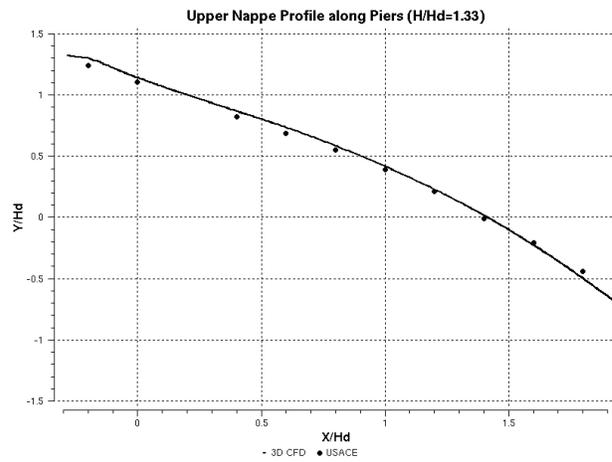
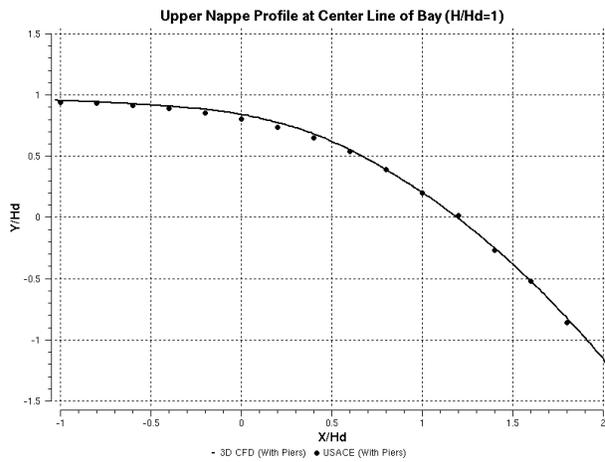
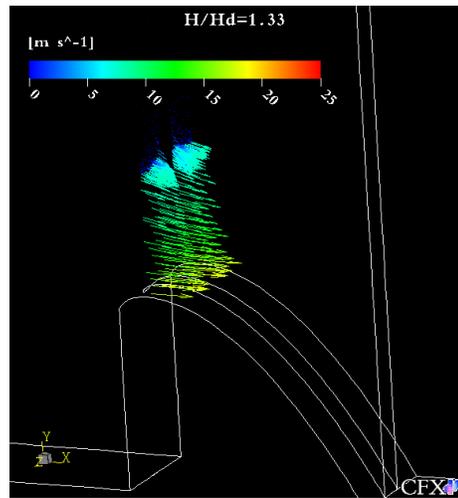
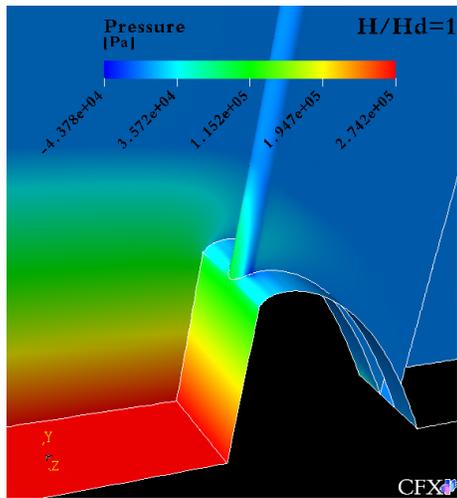


Figure 11: $H/H_d = 1$ 3D case. Pressure field at selected planes (top). Comparison of the Upper Nappe Profiles with USACE data at the center line of bay (middle) and along the pier (bottom).

Figure 12: $H/H_d = 1.33$ 3D case. Velocity Vectors at the crest cross section (top). Comparison of the Upper Nappe Profiles with USACE data at the center line of bay (middle) and along the pier (bottom).