By John E. Richardson, Ph.D., P.E., and Karel Pryl

Computer Simulation

Helps Prague Modernize and Expand Sewer System

Streamlines show the movement of water through the combined sewer overflow from right to left. The streamlines are colored by the lateral velocity of the fluid. The color map ranges from v = –0.02 m^3/s (blue) to v = 0.02 m^3/s (red). Large-scale turbulence (i.e., eddies) is observed in the throttle chamber downstream of the overflow.

Computer simulation is playing a critical role in helping the City of Prague in the Czech Republic modernize and expand its sewer system. In developing a master plan for the sewer system, the Prague Water Board needed to understand the performance of a combined sewer overflow (CSO) with a throttle chamber that appeared to stand in the way of increasing the capacity of the system. Building and testing a physical model of the CSO would have cost at least $100,000. Simulating the CSO with computational fluid dynamics (CFD) required only a fraction of the physical model cost and generated accurate performance-rating curves that showed the flow through the structure under various conditions. These curves were incorporated into a network model of the entire sewer system and used to plan Prague's future sewer requirements.

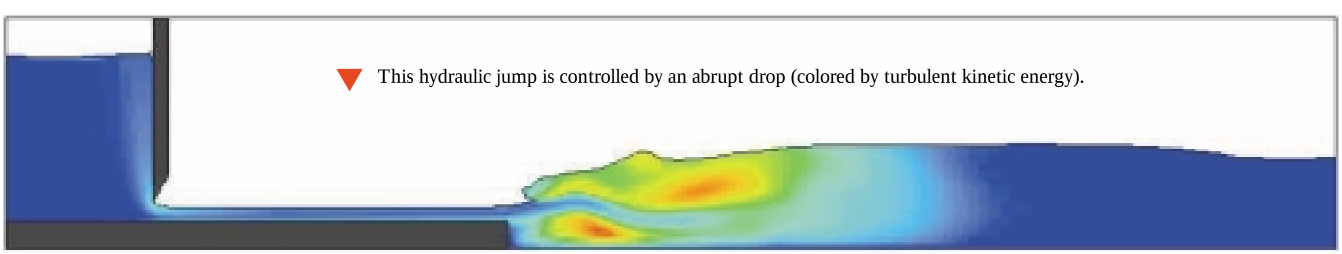
The Prague Water Board is in the process of designing and expanding its sewer system to meet the needs of the new millennium. Like most sewer plans, this one is being developed with a one-dimensional network model that predicts flows and depths throughout the system. This type of model works well for the majority of the sewer but cannot describe what happens in certain critical areas such as where the flow passes through a CSO structure.

A CSO basically is a hydraulic control structure in the sewer system designed to divert excess flows (usually to a river) caused by rainfall events. Wastewater treatment plants typically do not have the capacity to treat the very large flows that occur during rainfall events. One of the largest CSOs in the Prague sewer system has an exceptionally long overflow weir that is more than 50 feet in length. The behavior of this CSO is complicated by the fact that it flows under a gate into a secondary downstream throttle chamber designed as a flow limiter.

Need For Empirical Equation

The engineers working on the master plan needed to understand the behavior of this structure to accurately model the overall sewer system. This typically is accomplished by developing an empirical equation that determines the flow split between the system discharge flow rate to the treatment plant and the overflow to the river under various input flow rates. While standard equations are available to model structures of this type, Prague Water Board engineers were worried that these equations did not account for the unique throttle chamber. Using the wrong equations could be disastrous. Tens of millions of dollars of capital improvements were at stake.

Consulting engineers with Hydroinform A.S., a leading Prague consulting company in the field of water management, environmental sciences and related fields developed the network model. The engineers felt that CFD would be the ideal tool to develop an accurate empirical equation at a reasonable cost. CFD involves the solution of the governing equations for fluid flow at thousands of discrete points on a computational grid in the flow domain (i.e., the CSO structure). When properly validated, a CFD analysis allows engineers to determine the direction and speed of flow at any point in the flow domain. Recent improvements in CFD software and the continuing improvement in the power of personal computers make it possible to perform analyses for a small fraction of the cost of building a physical

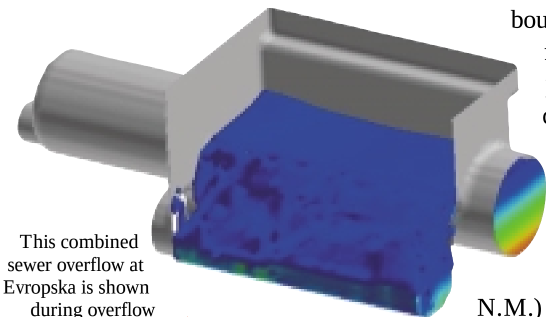


This hydraulic jump is controlled by an abrupt drop (colored by turbulent kinetic energy).

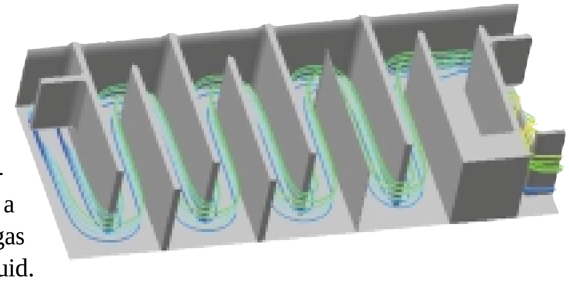
model. Unlike a physical model, the geometry of the CFD model can be changed quickly on the computer and re-analyzed to explore different options in project design or operation conditions.

CFD Simulation

Hydroinform contracted with Earth Tech, a consulting engineering firm that specializes in total water management, environmental, waste remediation, engineering, construction and transportation, to perform the CFD analysis. An important advance in CFD technology has made it more relevant to sewer system analysis. The software now can accurately model free fluid services. A free surface is an excellent approximation of the interface between a liquid and a gas when the gas applies a pressure only on the liquid. The most advanced method of modeling free surfaces is called the volume of fluid (VOF) technique. The VOF method has three parts: locating the surface, tracking the surface as a sharp interface through a grid and applying a boundary conditions at the interface. Some commercial CFD programs have claimed a VOF capability while implementing only one or two of the VOF parts. Earth Tech engineers selected a software package (FLOW-3D from Flow Science, Inc., Los Alamos, N.M.) that fully implements the VOF method on a personal computer.



This combined sewer overflow at Evropska is shown during overflow conditions.



These streamlines of flow in the clearwell are colored by elevation.

The engineers developed a computational model of the CSO and throttle chamber consisting of 18,900 cells, including 84 divisions in the direction of flow, 15 in the vertical direction and 15 laterally. Short portions of the sewer upstream and downstream of the CSO structure were included in the simulation as well. The flow in the virtual sewer was controlled by the sewer velocity and depth boundary conditions set upstream, continuative boundary conditions downstream, atmospheric pressure boundary conditions at the overflow and atmospheric pressure and zero shear boundary conditions at the fluid free surface.

Properties of water were used for the fluid and an eddy viscosity approximation was used for turbulence. Several runs were performed with other turbulence models but their influence on the solution was minimal. These runs were performed on a 400 MHz Pentium II personal computer with 128 MB RAM. The fluid in all of the simulations was initially at rest. Fluid velocities at the upstream boundary were increased linearly to match the flow rate for a given scenario. Simulations were performed until steady flow conditions were met. Some initial simulation runs were performed under normal operating conditions to validate the accuracy of the model. The flow depth and speeds predicted by the model were compared to physical measurements taken downstream of the overflow. The CFD results matched the gauge data within one percent.

Developing the Equation

Earth Tech engineers then performed a series of runs with steadily increasing upstream flow rates to gauge the performance of the CSO under high flow conditions. The simulation results confirmed the initial judgment of the Prague Water Board engineers that something unusual was happening in the throttle chamber. The graphical simulation results showed the flow passing out of the CSO and under the weir gate and then expanding into the throttle chamber. Large-scale eddies were set up by this flow expansion in the throttle chamber. This expansion choked off further flow of water. Once the gate was opened to a relatively low position, further opening of the gate did not affect the flow rate because it was controlled by the turbulence in the throttle chamber. The engineers performed a series of analysis runs while varying the upstream flow rate and the position of the gate. They used these runs to develop an empirical equation that defined the flow in the throttle chamber based on the two independent variables (upstream flow rate and gate position). This equation was drastically different from the standard weir flow formulas and validated the judgment of Prague Water Board engineers that using the standard formulas could have led to major errors.

The empirically developed hydraulic performance characteristics of the CSO were used in a one-dimensional model produced by Hydroinform based on the "Mouse" network simulation package. This model was used to predict the flow depth and velocity everywhere in the sewer system based on conditions specified by the engineers. The model was used to

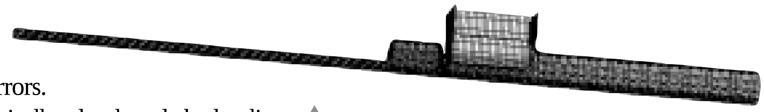
- Determine which parts of the present sewer system are most vulnerable to backups,
- Change overall flow rates to simulate different growth scenarios for the city and determine what type of capital improvements would be required to handle them, and
- Simulate the performance of different alternative improvements to the system (e.g., retention ponds) to determine how much they will improve overall performance.

Possible Solutions

Earth Tech engineers also modified their original model to investigate several ways to increase flow rates through the CSO. This process was expedited by the fact that CFD analysis not only provides numerical data but also generates graphical output such as velocity vectors superimposed on the domain that frequently show the cause of a particular problem. In this case, the eddy currents were clearly causing the flow bottleneck in the throttle chamber, so the engineers focused on methods of reducing them.

The simplest approach was to remove the throttle chamber and the analysis results showed that this would greatly increase the flow rate. However, engineers recognized that this approach might be too expensive and looked for a less costly alternative. After evaluating a number of different alternatives, they finally determined that adding a series of vanes parallel to the flow through the throttle chamber would substantially increase the flow rate by reducing the eddy currents at a relatively low cost.

At one time or another, every hydraulic engineer has wished for a way to investigate complicated fluid flows without spending time at the end of a mop. Experiments in a laboratory can be tough to set up, costly to carry out and difficult to replicate. On the other hand, CFD simulations of hydraulic systems can be set up in minutes and solutions to most problems can be obtained in a matter of hours. The computer solutions are just as accurate when compared to theoretical or experimental work, even when discontinuous or rapidly varied flows are involved.



▲ This computational model depicts a portion of the City of Prague's sewer system.

About the Authors:
John E. Richardson, Ph.D., P.E., is a senior water resources engineer with Earth Tech, Concord, N.H.

Karel Pryl is a senior water resources engineer with Hydroinform A.S. in the Czech Republic.

For more information on this subject, circle 871 on the reader service card.