adaptive RESEARCH — Fluids Engineering Services

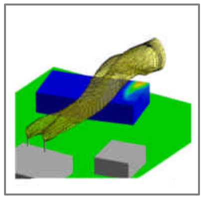
a Division of Simunet Corporation

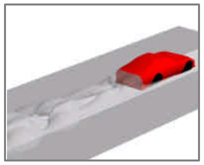
CFD and Environmental Engineering

Computational Fluid Dynamics (CFD) provides an accurate method for performing detailed flow analyses in both air and water environments. CFD has been effectively applied to many areas in environmental engineering including: contaminant or chemical dispersion, sedimentation/erosion processes, agricultural systems (e.g., greenhouse design/performance), wind engineering, HVAC system optimization, and many more.

Dispersion Flows

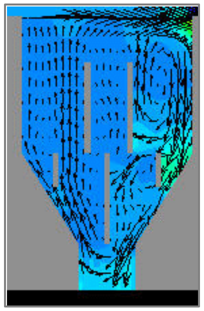
Among the wide range of possible applications in environmental engineering, CFD can play a significant role in the detection of pollutants and smoke propagation from fires. For example, CFD simulations have been used to predict concentration of toxins in urban settings under various topologic and atmospheric conditions. Similarly, numerical techniques have proven to be crucial for the optimization of ventilation systems in buildings or tunnels to guaranty people's safety in case of fire and smoke propagation.





Sedimentation/Erosion Processes

<u>CFD2000 Example Application</u>: "CFD calculations of flow in a straight flume for sediment erodibility testing. The simulations allowed for improved post-processing of the erosion data collected and better understanding of scour pit formation that is sometimes found in the flume's test section.

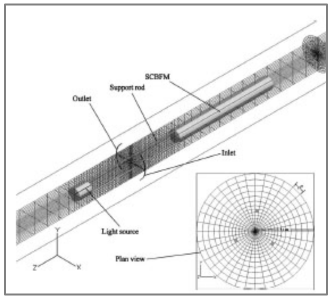


Knowledge of the erodibility of cohesive sediments (the rate of erosion as a function of hydraulic conditions) is necessary for conducting rate and transport studies of particle-bound contaminants and for developing sediment budgets. Unfortunately, cohesive sediment erodibility is not easily predictable based on environmental data (McCave 1984; Aberle et al. 2004)." *

Journal of Waterway, Port, Coastal, and Ocean Engineering, Nov/Dec 2006 457.

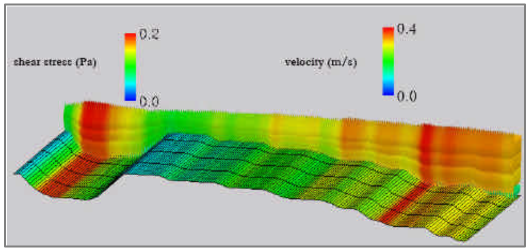
Sedimentation/Erosion Processes (continued)

<u>CFD2000 Example Application</u>: "This paper reports on experiments and simulations of subsurface flow from a slotted acrylic tube deployed in a sand-tank flow chamber for two different purposes. In the first instance, the slotted tube is used to represent a single fracture intersected by an uncased well. In the second instance, the slotted tube is used to represent a multislot well screen within a porous medium. In both cases, the scanning colloidal borescope flowmeter (SCBFM) measures ground water velocity within the well by imaging colloids traveling through a well to measure their speed and direction. Measurements are compared against model simulations."*



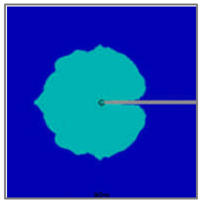
** Vol. 44, No. 3—GROUND WATER—May–June 2006 (pages 394–405).*

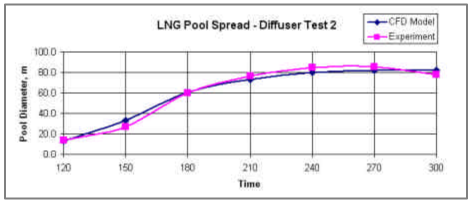
<u>Example Application</u>: SNL used CFD2000 to model local erosion from the stream bed and river banks of the Pecos River. The Figure below is an example of the CFD2000 moving grid erosion model (shear stress showing erosion potential).



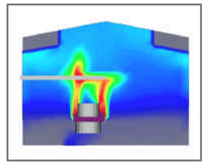
Free Surface Flows

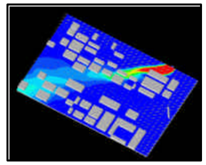
<u>Example Application</u>: SNL has recently (2010) conducted a large scale experiment to examine transient LNG pool spreading behavior on water. CFD2000 simulations were compared to experimental data, with the results comparing well.





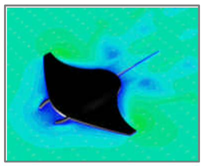
More Environmental CFD Applications



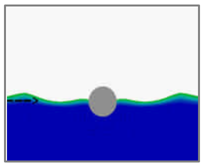


Typical environmental applications for which CFD can be used to perform detailed flow analysis include flow in urban settings or buildings. Accurate knowledge of flow structures and concentrations is often critical when predicting the dispersion of smoke and pollutants in atmospheric or ambient air conditions.

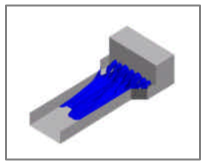
Other Applications



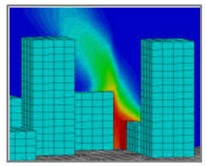
Marine



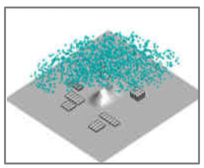
Free-surface



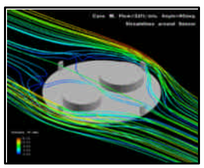
Channels



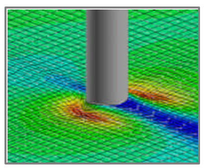
Fire



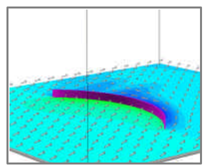
Particle Flow



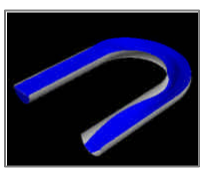
Smoke Detection



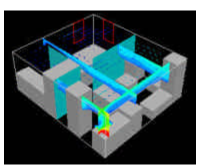
Erosion



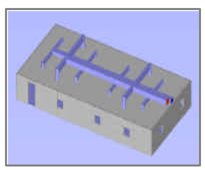
Wind Engr



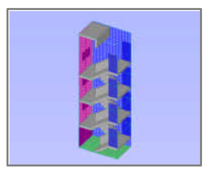
Tube Flow



Room Flow



HVAC



Buildings

CFD2000 Software

Adaptive Research utilizes the CFD2000 software system to perform engineering contract work. CFD2000 is an integrated program providing the necessary tools for simulating real engineering applications. The system includes modules for geometry creation and mesh generation, a Navier-Stokes equation solver, and advanced scientific visualization.

Computational Fluid Dynamics (CFD)

What Is CFD?

Computational Fluid Dynamics is a powerful engineering tool for simulating all types of fluid flows. CFD simulations produce detailed descriptions of flow characteristics including values for velocities, pressures, and other variables. Thermal characteristics and other advanced physics, like chemical reactions, particle flow, free-surface flow, can also be simulated.

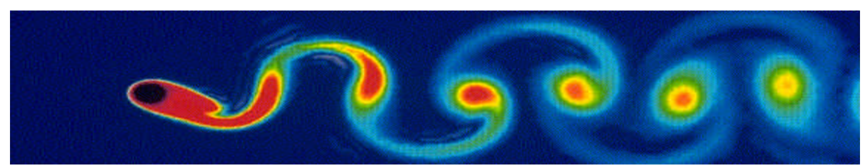
Why use CFD?

Computational Fluid Dynamics can significantly reduce design and development time, provide detailed information otherwise not available from physical experiments, and quickly simulate a wide range of flow conditions. The results are improved designs, lower risk, and shorter time to market for a product or process.

How does CFD work?

CFD simulations involve solving numerically the fundamental laws of physics, called governing equations, which mathematically describe the properties and motion of fluids. The CFD modeling process occurs in three fundamental steps.

1. Geometric/Mesh Model Creation
2. Numerically Solving Navier-Stokes Equations
3. Solution Analysis Visualization



Contact Adaptive Research

To learn more about engineering services or the CFD2000 software offered by Adaptive Research, please call, e-mail, or visit our website.

Fax: 808-356-0128
E-mail: arsales@adaptive-research.com
Web: www.adaptive-research.com