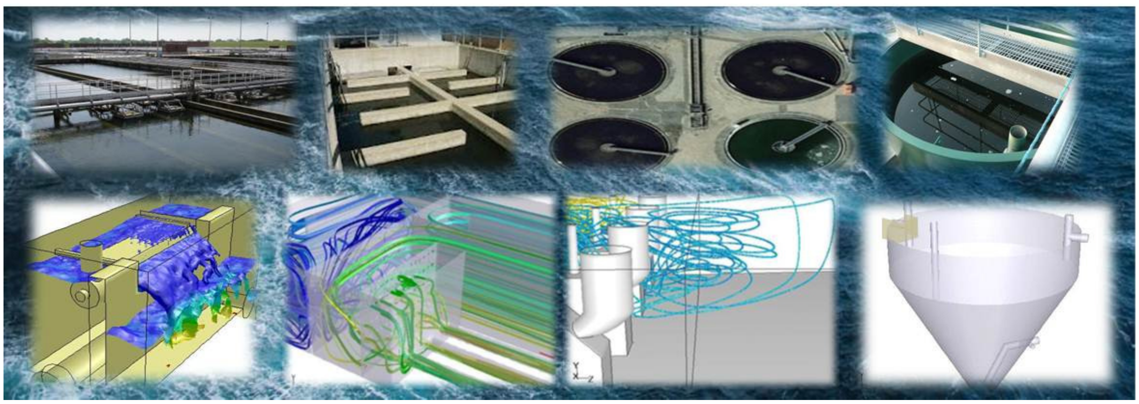
# CFD USE IN WATER – INSIGHT, FORESIGHT AND EFFICIENCY

**Prashant Huddar P.E. (California), Lead Structural Engineer, MWH ResourceNet, Pune, India**

**Yashodhan Dhopavkar, Environmental Engineer, MWH ResourceNet, Pune, India**

**The CFD simulation capability is combined with many years' experience, ensures process plant designs and water systems are accurately predicted. This in turn produces truly optimized unit performance and avoids costly design mistakes.**



Computational Fluid Dynamics (CFD) is a powerful numerical modelling technique, which is increasingly used to simulate the flow field within process plant and water systems. Typical modelling projects include clarifiers, contact tanks, mixing tanks, pumping station, combined sewer overflow (CSO), reservoirs, pipe loop, thermal plume model and digesters. CFD can provide significant benefits to clients by:

- Full flow field visualization for informed development of designs
- Performance optimization
- Rapid assessment of changes in geometry
- Reducing or eliminating the requirements for physical model tests
- Integrating with other methods and experience to provide the fullest set of information on which to base design decisions

CFD is becoming particularly useful in optimising the design and function of process unit, which are becoming increasingly important in meeting efficiency and environmental demands including conserving water and chemicals.

- ## Solution using CFD in Pumping station

The Computational Fluid Dynamics (CFD) model is used to investigate hydraulic performance of the pump station wet well. And potential vortex formation for different flow scenario.

**Key Benefits:**

- Easy to model no. of scenarios vs. physical modeling
- Pre-swirl rotation can easily identify
- Full scale modeling

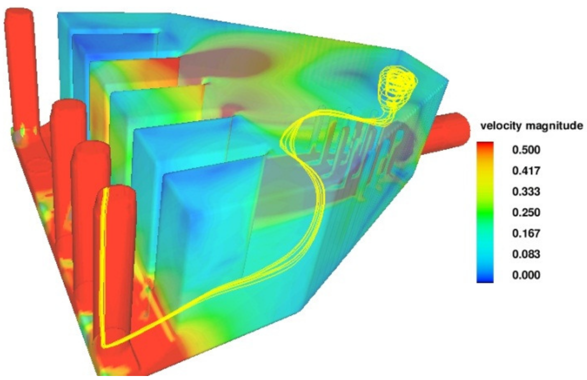



Figure 1 Streamlines inside Pumping Station

- ## Solution using CFD in Reservoir

CFD analysis was done to study the performance of the reservoir with all simulating hydraulic conditions inside the reservoir.
The parameters to monitor were residence time distribution, parallel path, channelling and mean concentration.

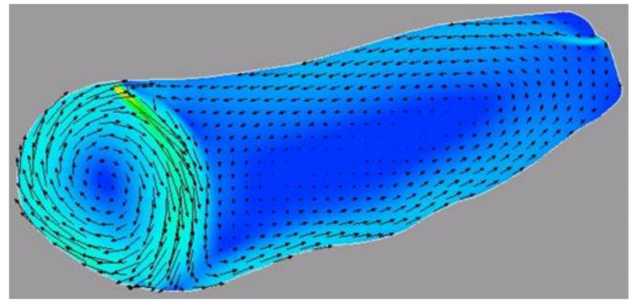


Figure 2 Velocity Contours in a Reservoir

**Key Benefits:**

- Investigate the internal hydraulics
- Analysis of the flow pattern in the reservoir with different arrangement of inlet and outlet arrangement.

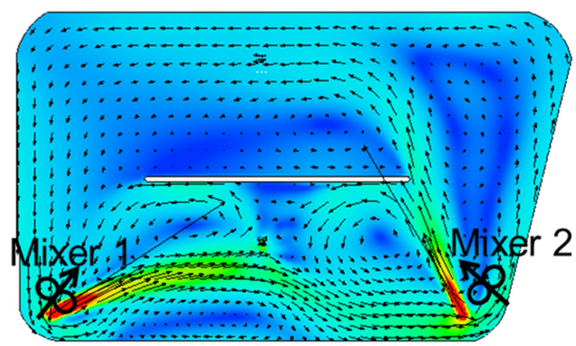



Figure 3 Velocity Contours in Big Canyon Reservoir

- ## Solution using CFD in contact tank

CFD study was completed for the proposed tank it had very good plug flow characteristics and that the contact time and hydraulic efficiency requirements were met.

**Key Benefits:**

- RTD
- Mean age of water simulated with extra user define function using scalar equation

## • Solution using CFD in CSO

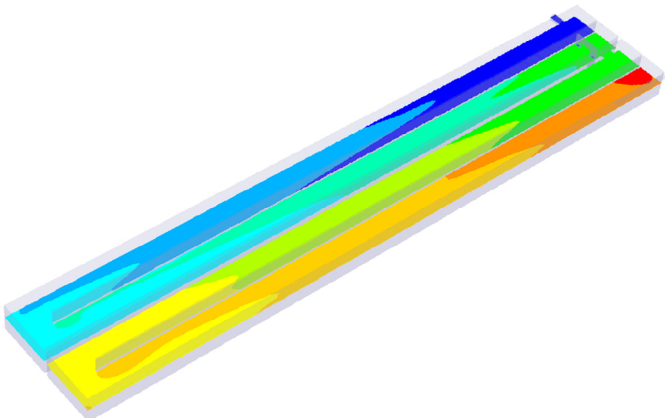


Figure 4 Mean Water Age in Frankley Contact Tank

The CFD modelling is used to simulate flow conditions inside the CSO chamber and simulate solids loading through inlet and as a screenings return. To investigate performance of high level pipe as a passage to the screenings.

**Key Benefits :**

- Optimizing the dimension
- Improved solid loading efficiency

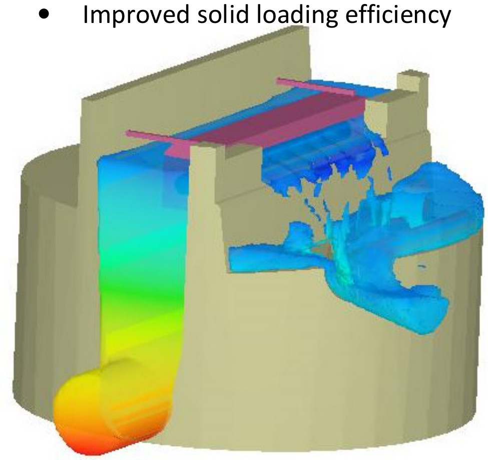


Figure 6 Surface Velocity Contours on CSO

## • Solution using CFD Clarifier

CFD analysis was done to examine the hydraulic performance of the settling tank. The main objectives are to check, that the outlet concentration is not at objectionable limits for the rated discharge as indicated by the pollution control board.

**Key Benefits:**

- Stream line and sludge concentration contours
- Computational fluid mixing is a very powerful tool when experimentation is
- either not possible or too expensive

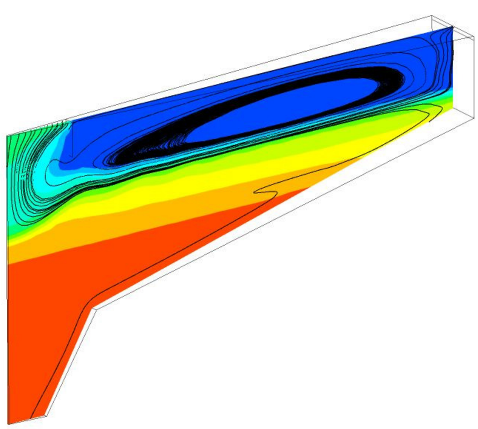


Figure 5 Sludge Concentration Contours in Millbrook tank

## • Solution using CFD Pipe loop

Residence time distribution, hydraulic performance analysis and improvement chlorine dissipation, short-circuiting and formation of dead zones are important parameter considerations for the pipe loop.

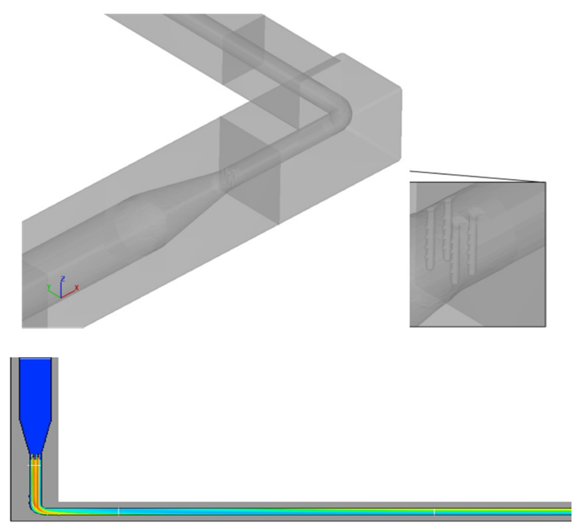


Figure 7 Mixing Contours in Pipe loop