



Discipline: Fluid flow, FSI, incompressible flow

Keywords: CFD Simulation, Computational Fluid Dynamics, pressure drop, fluid pressure, incompressible flow, volumetric flow rate, Femap, geometry preparation, CAD de-featuring, CFD validation

Main Graphic:



Caption: Relief Gate – Pressure Drop and Fluid Force Investigation.

CFD Investigation of Pressure Drop and Fluid Force through a Relief Gate in Water Storage Reservoir

Analysis Type: CFD, Incompressible Flow, Fixed Volumetric Flow

This CFD investigation was performed during the design process of a water control gate for a large water reservoir system in the northwest. A general volumetric flow rate for the gate was given and the designers needed to be sure that the pressure drop across the gate did not exceed project allowables. Given the constricted nature of the relief gate, a CFD analysis was the only logical approach. In addition to the pressure drop requirements, it was important to determine the CFD deriveforce on the gate for the structural design of the system. Our CFD services allowed our client to quickly assess whether or not their design would work.

Starting with CAD provided by the client, the geometry was split along the symmetry plane, the area of concern was isolated and small details were removed to facilitate a clean CFD analysis. As CFD consultants to this project, our goal was to ensure a high-accuracy CFD analysis in the minimum amount of time - this means clean CFD-type geometry. (see Figure 1).

Although geometry of the structure was provided, it was up to the CFD analyst to create a clean and efficient solid to represent the computation fluid dynamics fluid volume (see Figure 2).

With CFD boundary conditions set on the inlet, outlet, and symmetry plane, the CFD simulation was ready to run. Although setup was quite straightforward, significant time was dedicated to interrogating the CFD results from the simulation. This is normal in CFD consulting and the generation of full-field descriptive images is one of our strengths as CFD consultants. Once the CFD model was checked and there was confidence in the results, the client was provided with pressure drop and force values as well as contours of fluid velocity and static pressure (see Figure 3).

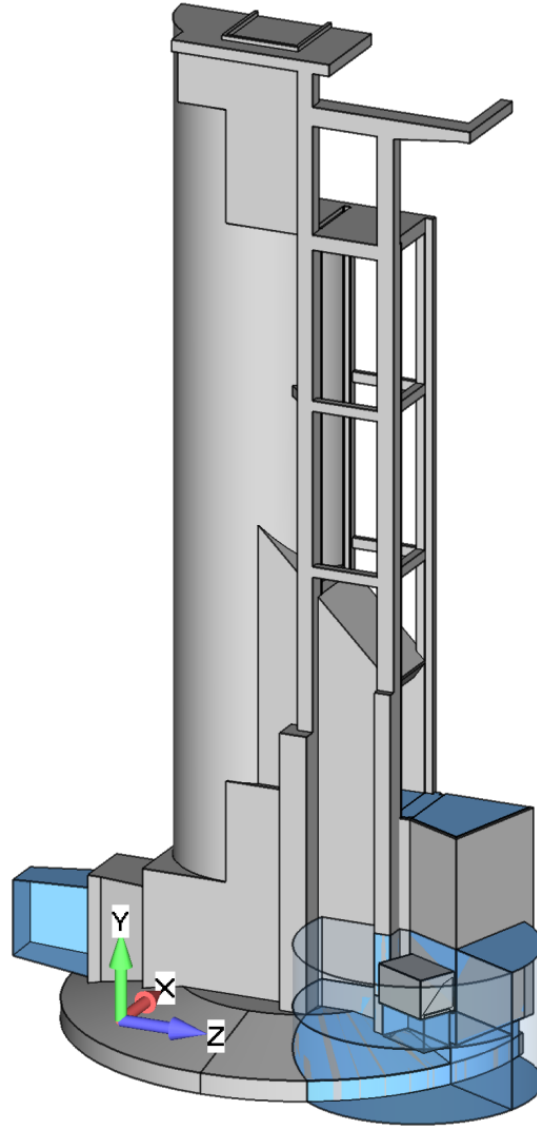


Figure 1: The **CFD** idealized geometry of the dewatering system with relief gate. As CFD consultants our job is to ensure that the geometry provides a fast numerical CFD simulation and time up-front in geometry preparation often saves our clients many hours downstream in CFD solution time.

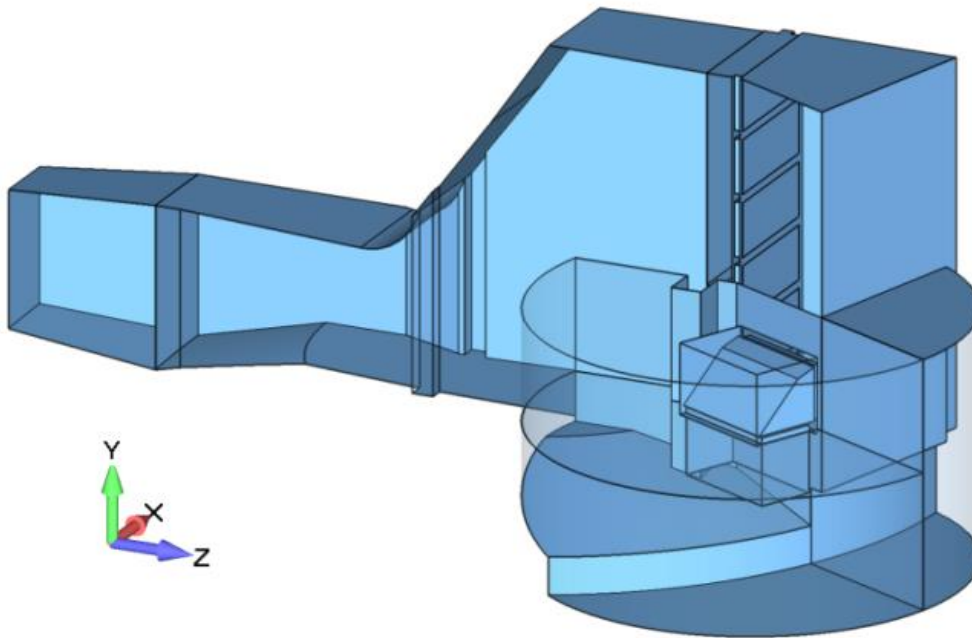


Figure 2: The CFD fluid volume used for the CFD simulation

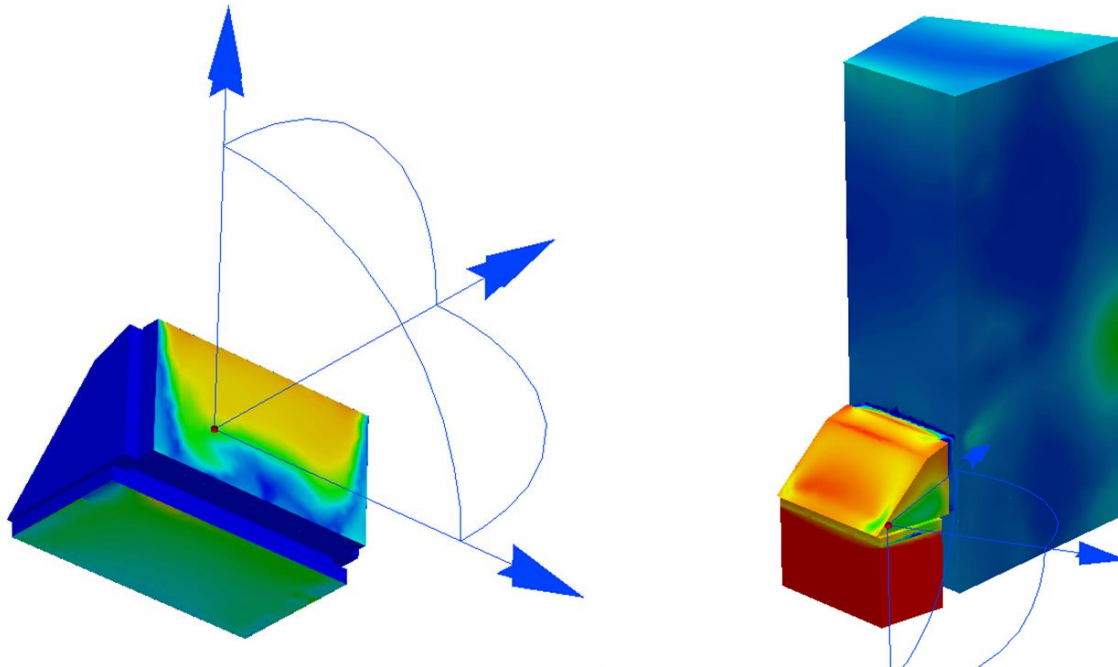


Figure 3: In the image on the left, the CFD calculated velocity magnitude is contoured over the relief gate fluid volume geometry. In the image on the right, CFD simulation provides the static pressure is contoured over the relief gate fluid volume geometry as well as the adjacent fluid volumes.

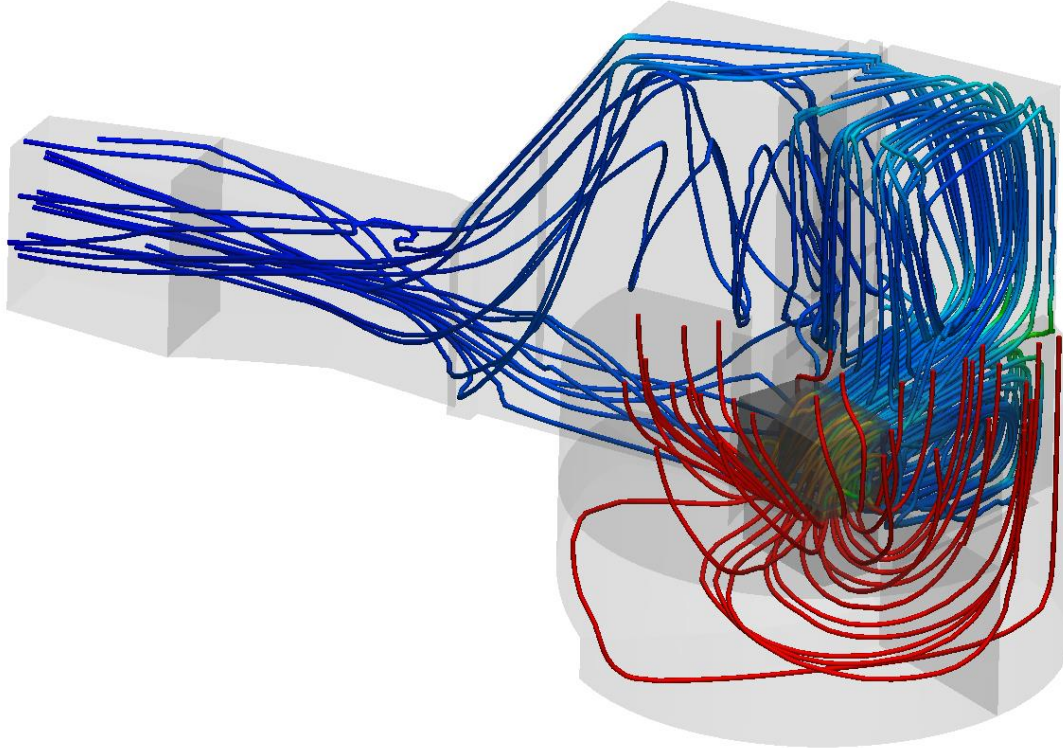


Figure 4: CFD particle tracing illustrates the flow path through the structure

Summary: CFD results from this simulation were used to successfully update their design and provide confidence that the final structure would perform as intended. The structure has been built and is in operation. We were later notified that this design won an award for its efficiency. As CFD simulation engineers it was nice to see our teamwork recognized.