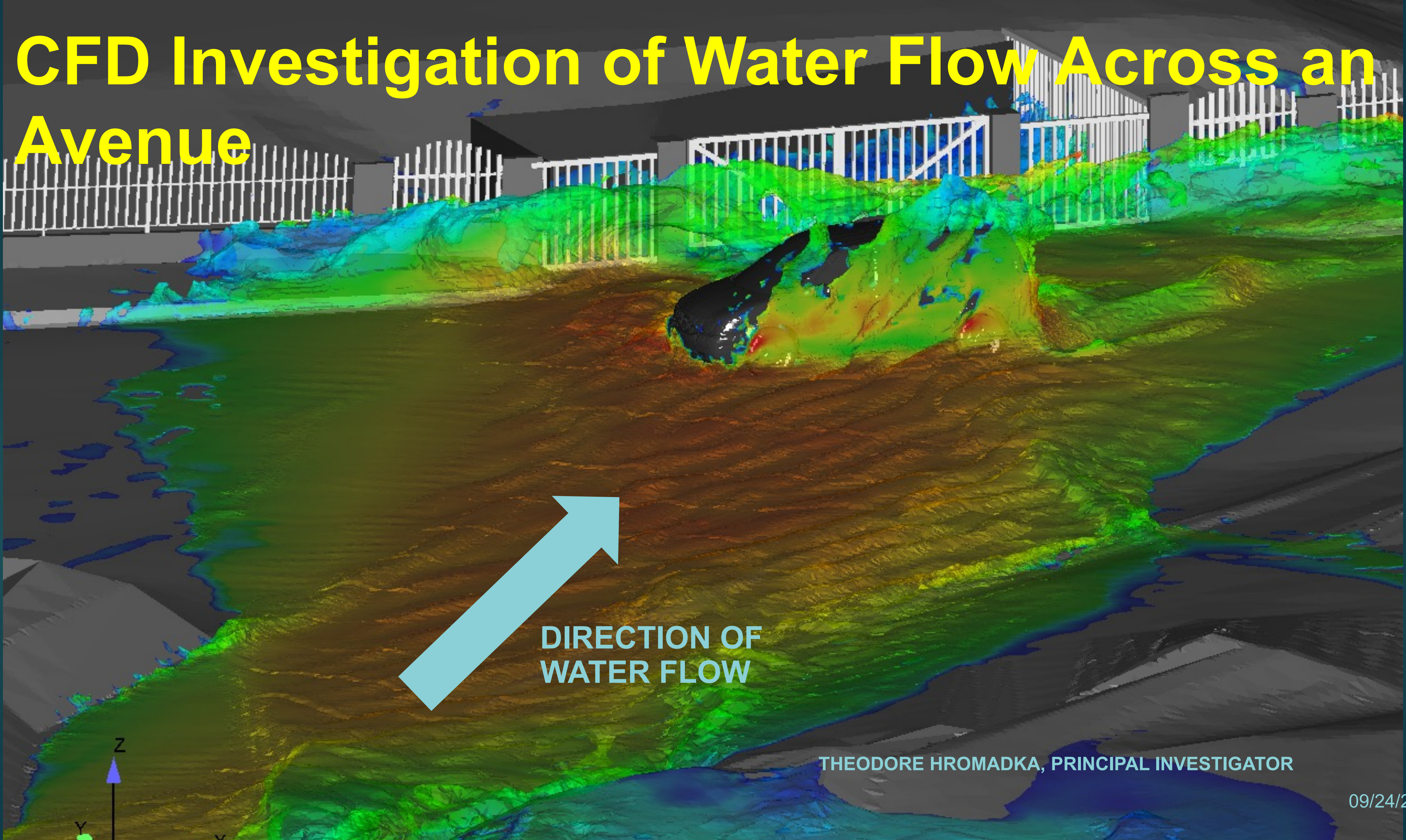
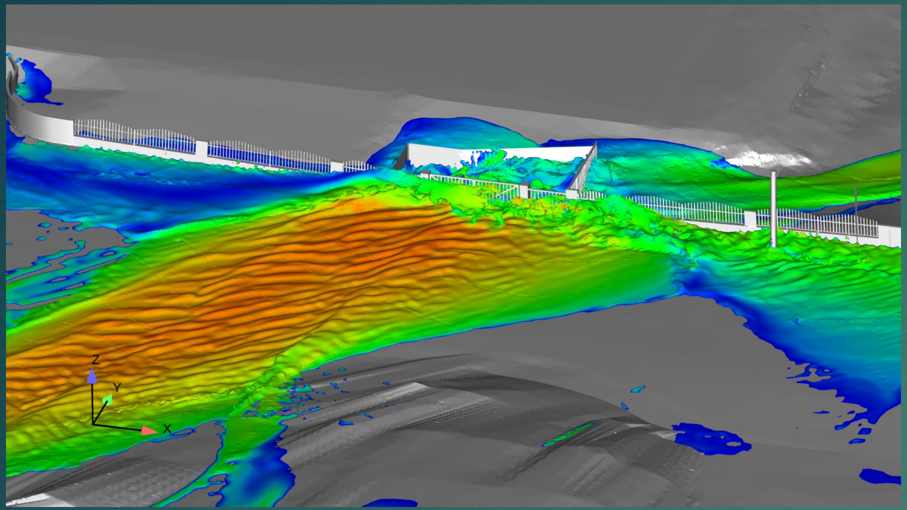
CFD Investigation of Water Flow Across an Avenue



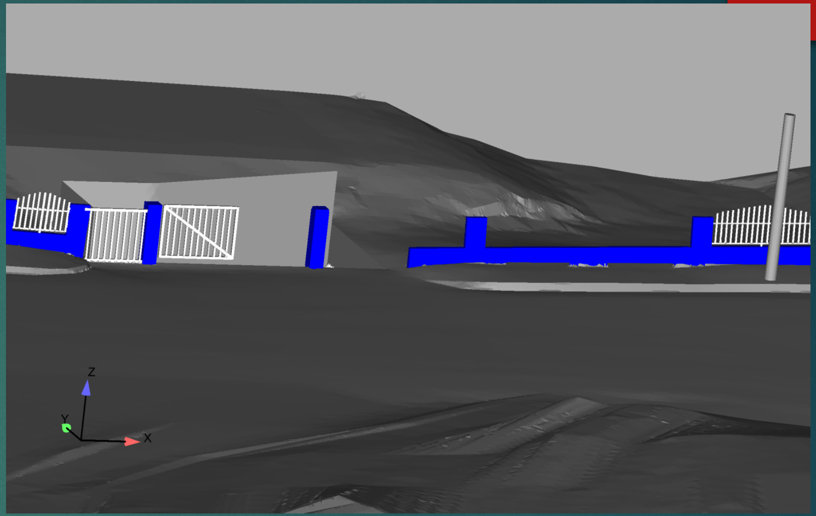
THEODORE HROMADKA, PRINCIPAL INVESTIGATOR

09/24/2019

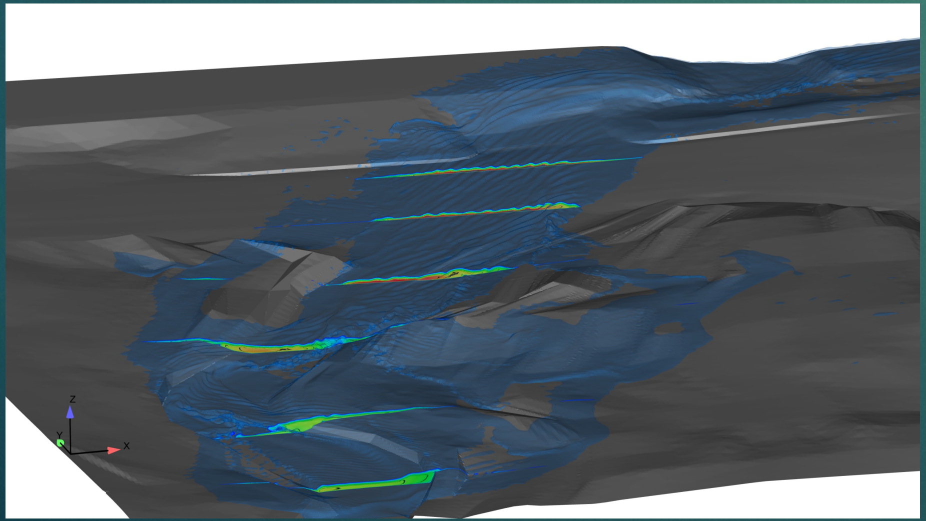
Baseline – In Situ Conditions



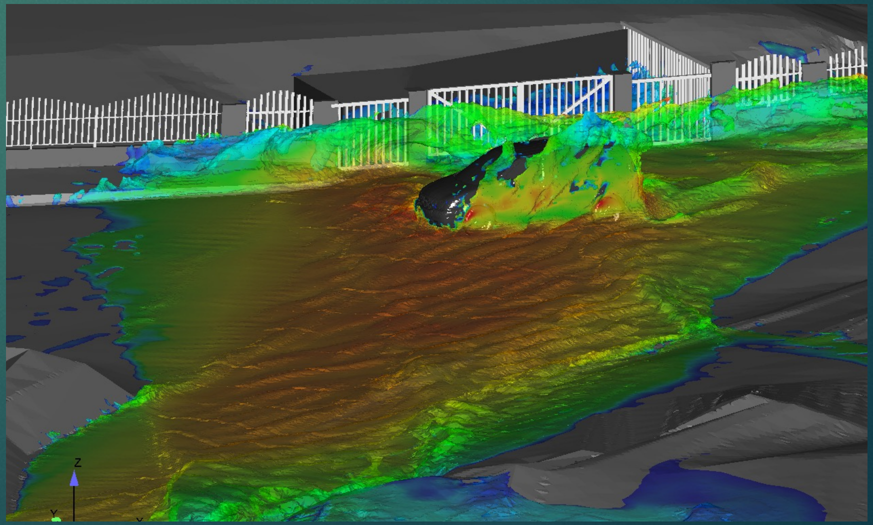
Baseline with Damaged Infrastructure – Wall/Fence modified to reflect the conditions after the storm event



Natural Conditions – Wall/Fence/Drop Inlet removed to reflect the natural conditions of the area surrounding the Avenue



Baseline with Vehicle – A vehicle model representing a minivan was placed in the westbound lane on the Avenue



Software

- In this study OpenFOAM an open source code was used as CFD tool.
- OpenFOAM solver solves mathematical equations known as **Navier-Stokes** equations.
- OpenFOAM solvers are based on Finite Volume Method for discretization
  - Domain is discretized into a finite set of control volumes (called cell)
  - General conservation of equation for mass, momentum, energy , etc. are discretized into algebraic equations.

$$\underbrace{\frac{\partial \rho \mathbf{U}}{\partial t}}_{1} + \underbrace{\nabla \cdot \rho \mathbf{U}\mathbf{U}}_{2} + \underbrace{\nabla \cdot \rho \mathbf{R}}_{3} = \underbrace{-\nabla p}_{4}$$

1. Local rate of change of $\rho \mathbf{U}$
2. Convective rate of change of $\rho \mathbf{U}$
3. Viscous dissipation (laminar + turbulent)
4. Pressure gradient

```
solve
(
    fvm::ddt(rho, U)
  + fvm::div(phi, U)
  + turbulence->divDevRhoReff(U)
 ==
  - fvc::grad(p)
);
```

  - All equations are solved to render flow field.

Software

- CFD software: OpenFOAM
  - Open source CFD software, licensed under the General Public License (GNU)
  - Developed and Maintained by OpenFOAM Foundation /OpenCFD Ltd, UK
  - OpenFOAM is one of the top 3 most used CFD software in the world.
  - Trusted by industry, R&D centers and universities.
  - https://openfoam.org

- Post Processing software: paraview
  - ParaView is an open-source multiple-platform application for interactive, scientific visualization. It has a client–server architecture to facilitate remote visualization of datasets and generate detailed models to maintain interactive frame rates for large datasets.
  - Developed and maintained by Sandia National Laboratory, Kitware Inc,
  - https://www.paraview.org

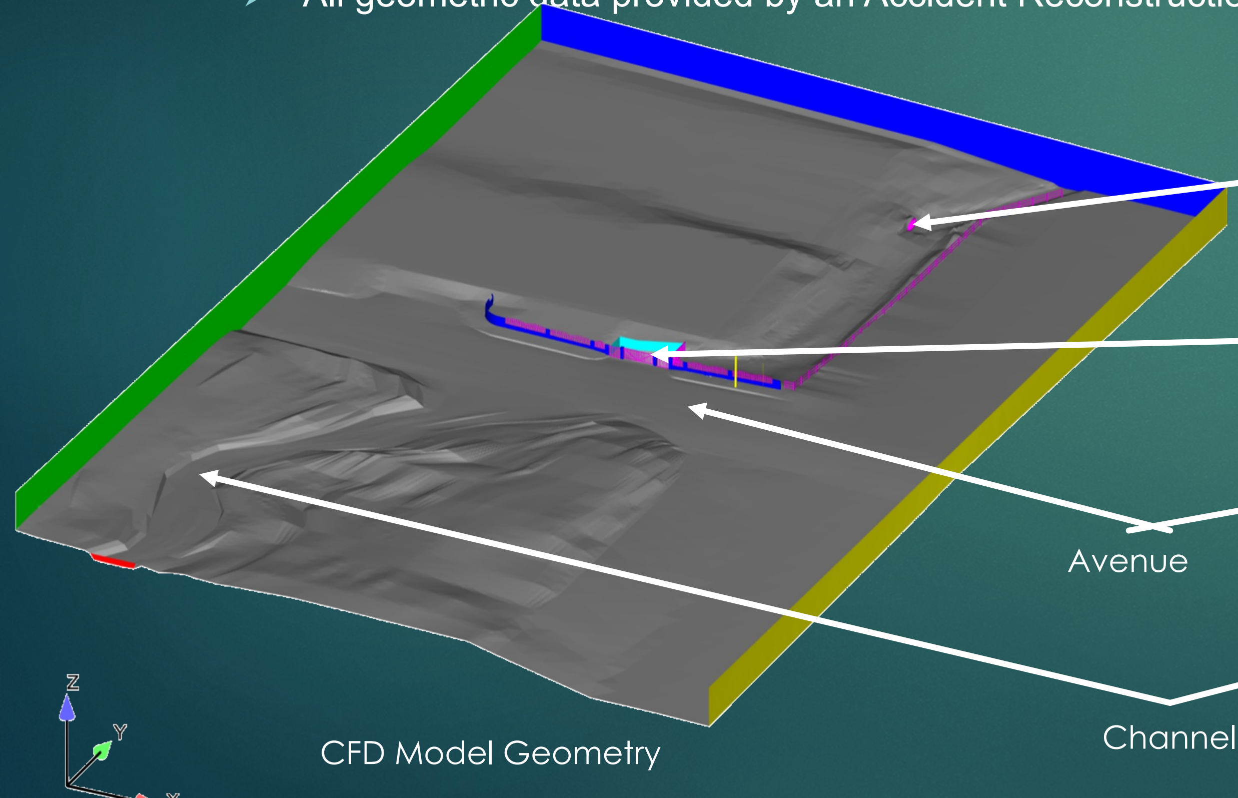
Computational Hardware

- Usually a High Performance Computing Cluster (HPCC) is required to perform a CFD simulation, specially when the domain size is very large and physics are complex. For the model set up and post-processing, the following computer hardware specification was used:
  - 128 GB RAM
  - 32 cores
  - 5 TB storage hard drive
  - 8 GB High resolution graphic card
- The simulation was performed on the cluster using 128 core processor.

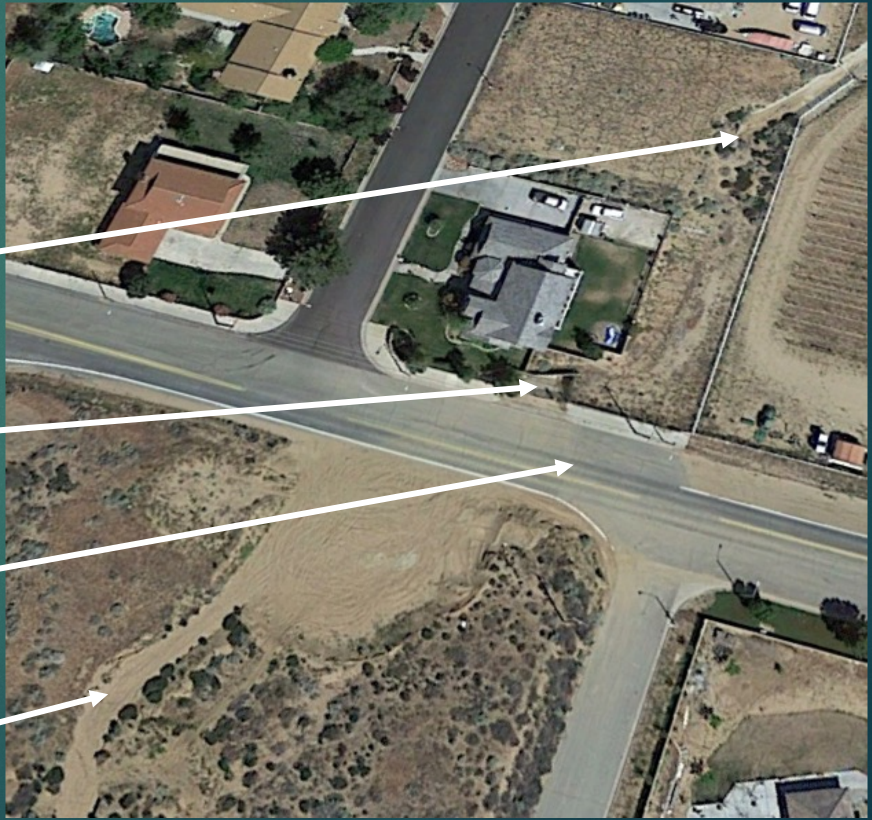
The Study Geometry

- Geometry preparation: Surface creation from topography data
  - Topography point data was used to create

    the CAD model (gate, ground, ..).
  - Side and top surfaces were added to create a closed volume domain.
  - The CAD model was meshed
  - All geometric data provided by an Accident Reconstruction Expert

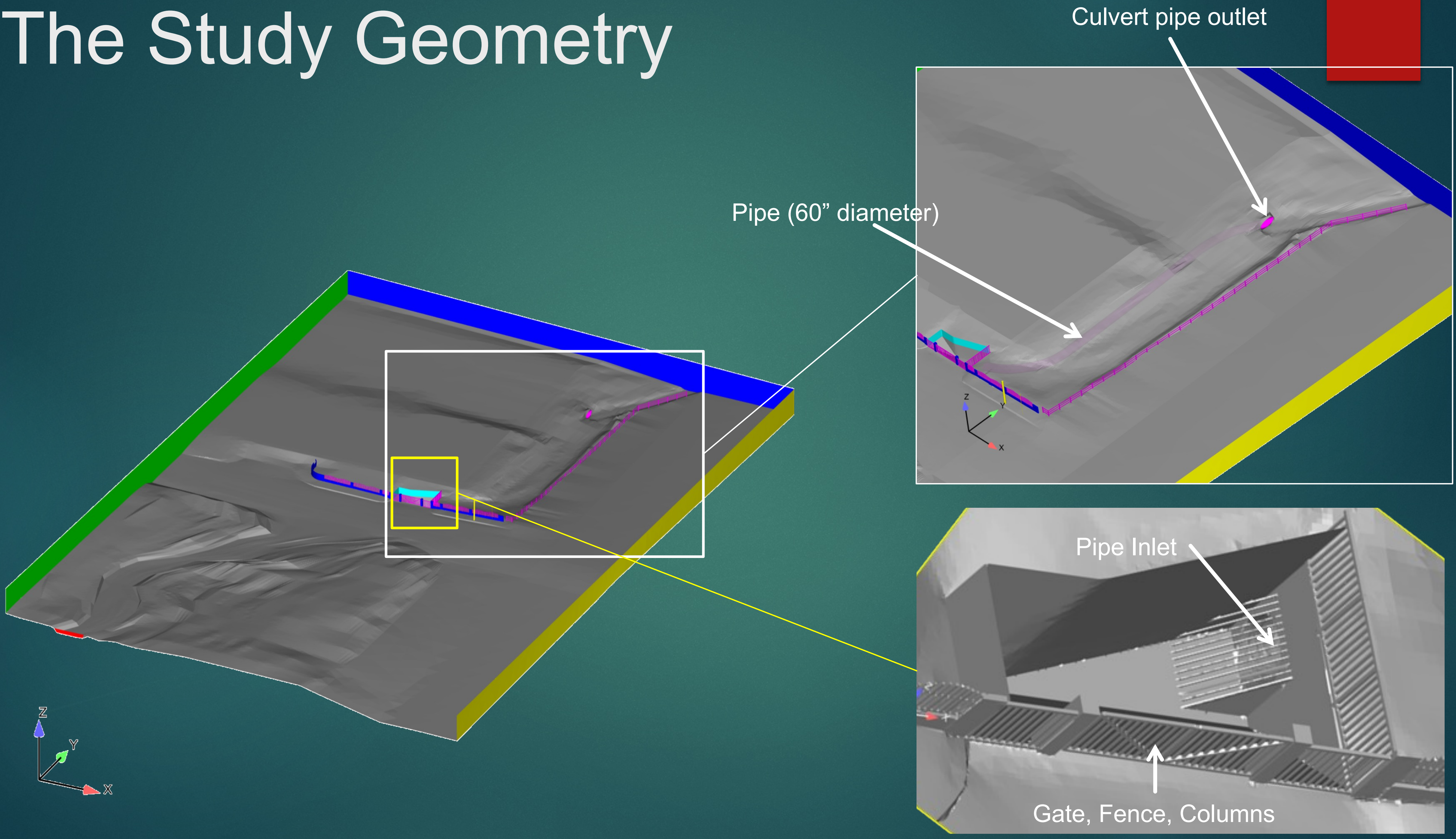


CFD Model Geometry



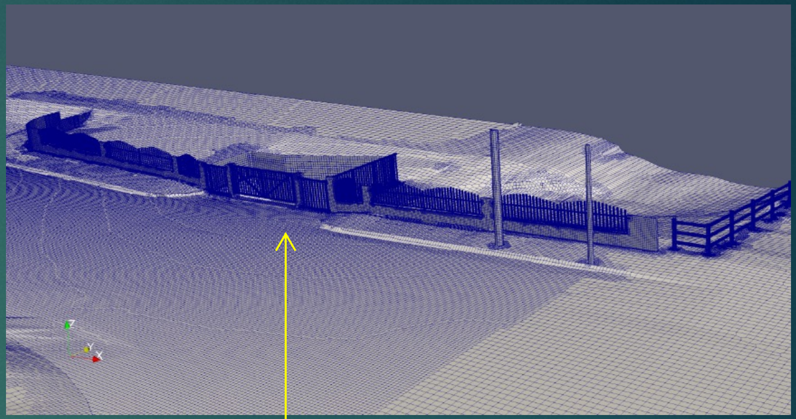
Source: Google Earth 4/11/2015

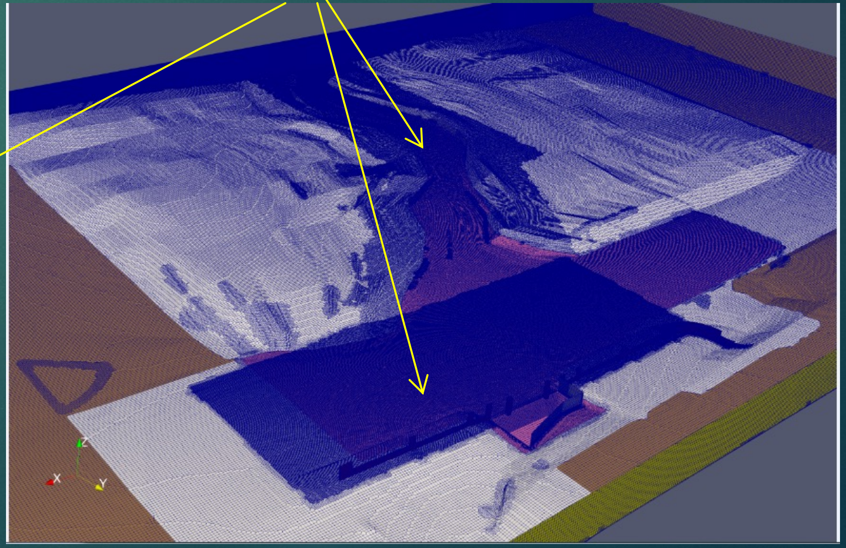
The Study Geometry

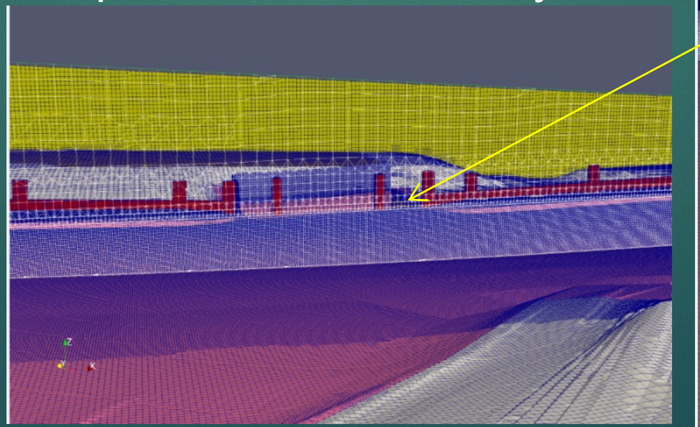


Computational Meshing

- Mesh Generation (domain space discretization)
  - The computational cleaned CAD geometry was used to construct the fluid domain.
  - Local surfaces and volume regions refinement were defined in order to capture all geometry details.
    - Channel, gate, bars, gate, street, spill way, culvert pipe,….
  - Proper surface and volume cell size distributions were applied to capture the geometry and physics precisely
  - 3 prism layers were generated to capture the flow velocity profile at the walls.

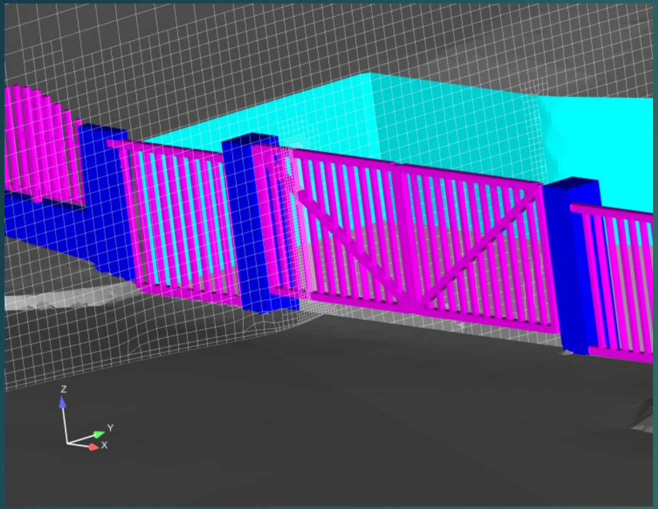


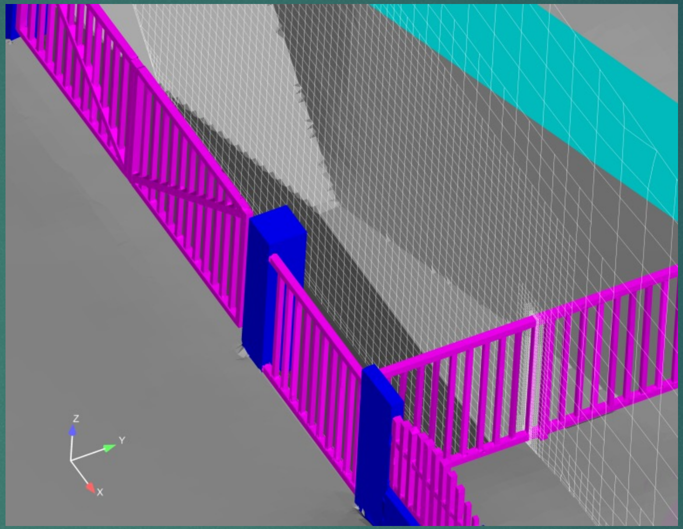


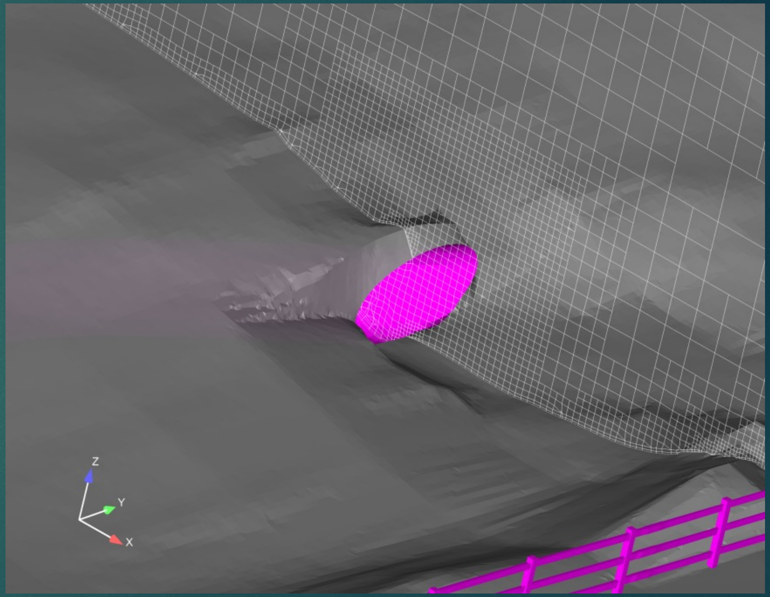


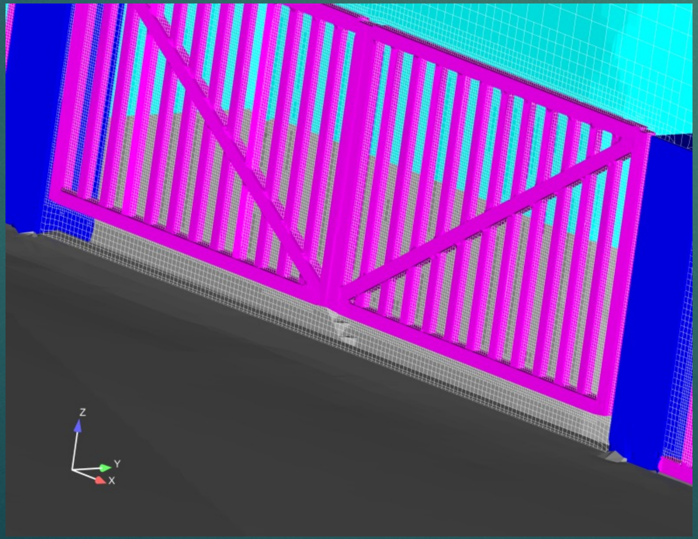
Computational Meshing

Hexahedral dominant cell type was used to construct the mesh



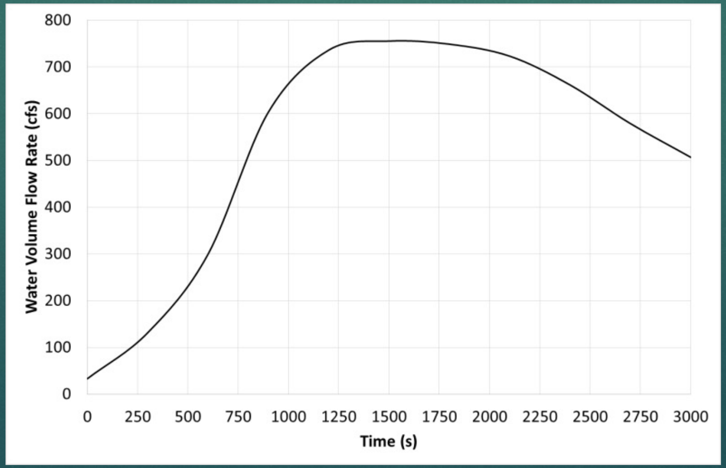




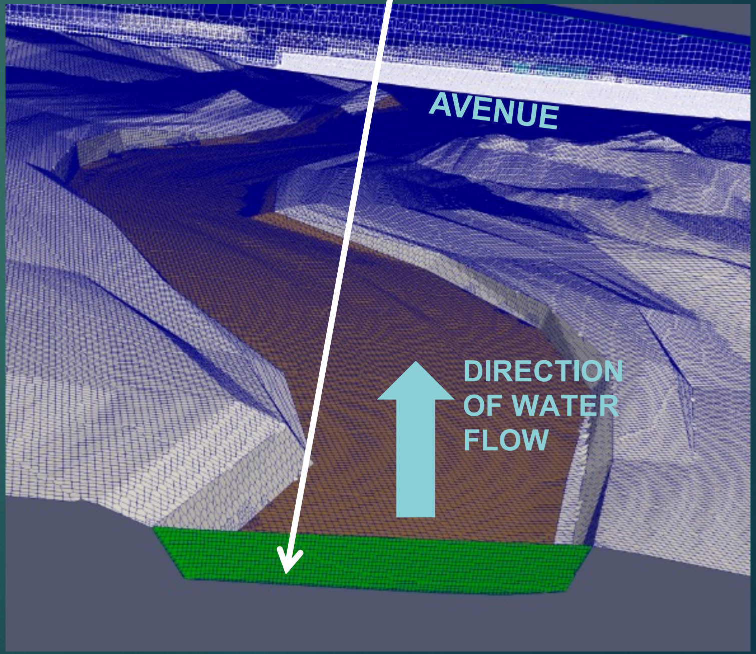


Boundary Conditions

- Boundary conditions set up
  - A variable volume flow rate was assigned at channel inlet. This volume flow rate was input from the output of a Unit Hydrograph Model which considers the Watershed and Precipitation of the actual storm event
    - The hydrology and hydrometeorology analysis results are detailed in a separate report
  - The Natural Conditions and Baseline with Vehicle simulations were performed from 1200s to 1800s of the volume flow rate data to capture the peak flow rate only

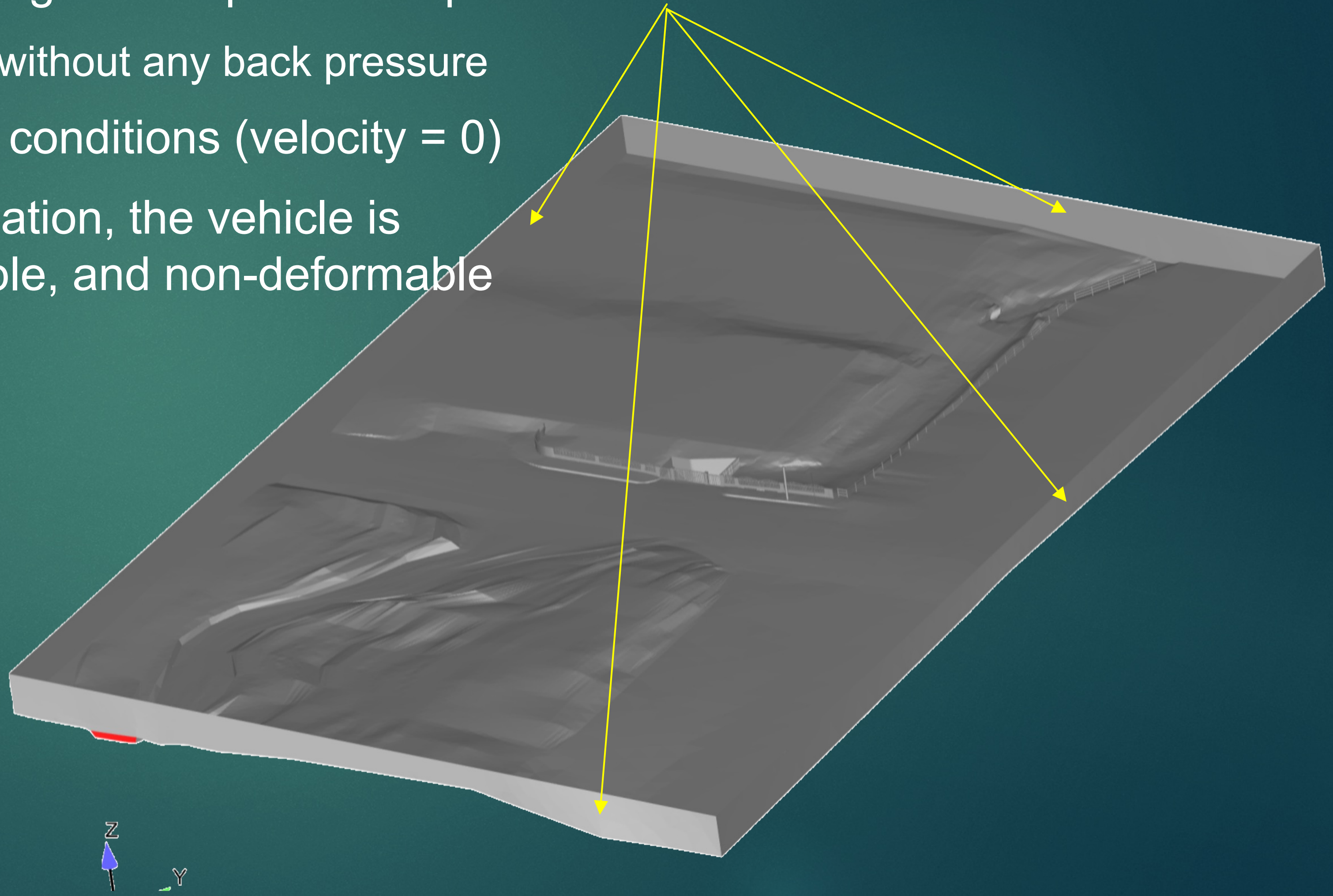


- Inlet
  - Variable Height Flow Rate inlet Velocity was assigned at inlet
  - This boundary condition provides a velocity boundary condition for multiphase flow based on a user-specified volumetric flow rate.
  - Water height and velocity will be adjusted based on the upstream resistance to maintain the specified volume flow rate



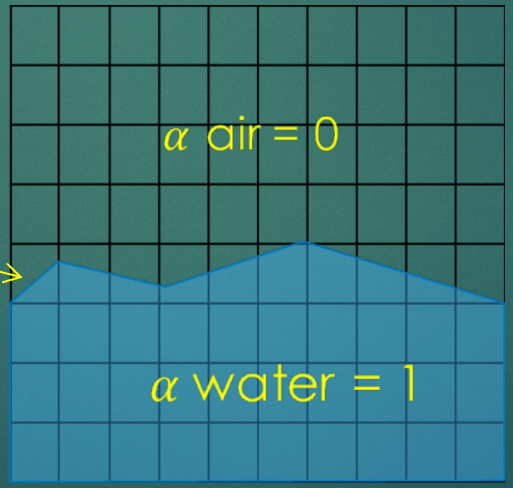
Boundary Conditions

- All sides and top patches were assigned to open atmosphere
  - Water can exit the domain freely without any back pressure
- All walls were assigned to no-slip conditions (velocity = 0)
- In the Baseline with Vehicle simulation, the vehicle is considered stationary, non-movable, and non-deformable



Solver and Model setup

- Volume of Fluid (VOF) method was used to capture and track the interface between air and water.
- VOF:
  - Solves 2 fluids of different phases, water and air
  - Captures the interface between the water and air
  - Each phase is described by fraction α $(Alpha)$ that occupies the local fluid volume
    - Phase 1 : α water = 1
    - Phase 2 : α air = 0
    - interface : $0 < \alpha < 1$



Solver and Model setup

- Volume of Fluid (VOF)
- Uses specie (scalar) transport equation to determine the relative volume fraction of two phases, or phase fraction α , in each computational cell.

$$\alpha = \alpha(\mathbf{x}, t)$$

- Physical properties are calculated as weighted averages based on the phase fraction

$$\begin{aligned} \mu(\mathbf{x}, t) &= \mu_{\text{water}}\alpha + \mu_{\text{air}}(1-\alpha) \qquad \mu = \sum_{i=1}^{N}(\alpha_i \mu_i) \\ \rho(\mathbf{x}, t) &= \rho_{\text{water}}\alpha + \rho_{\text{air}}(1-\alpha) \qquad \rho = \sum_{i=1}(\alpha_i \rho_i) \end{aligned}$$

- Governing equation

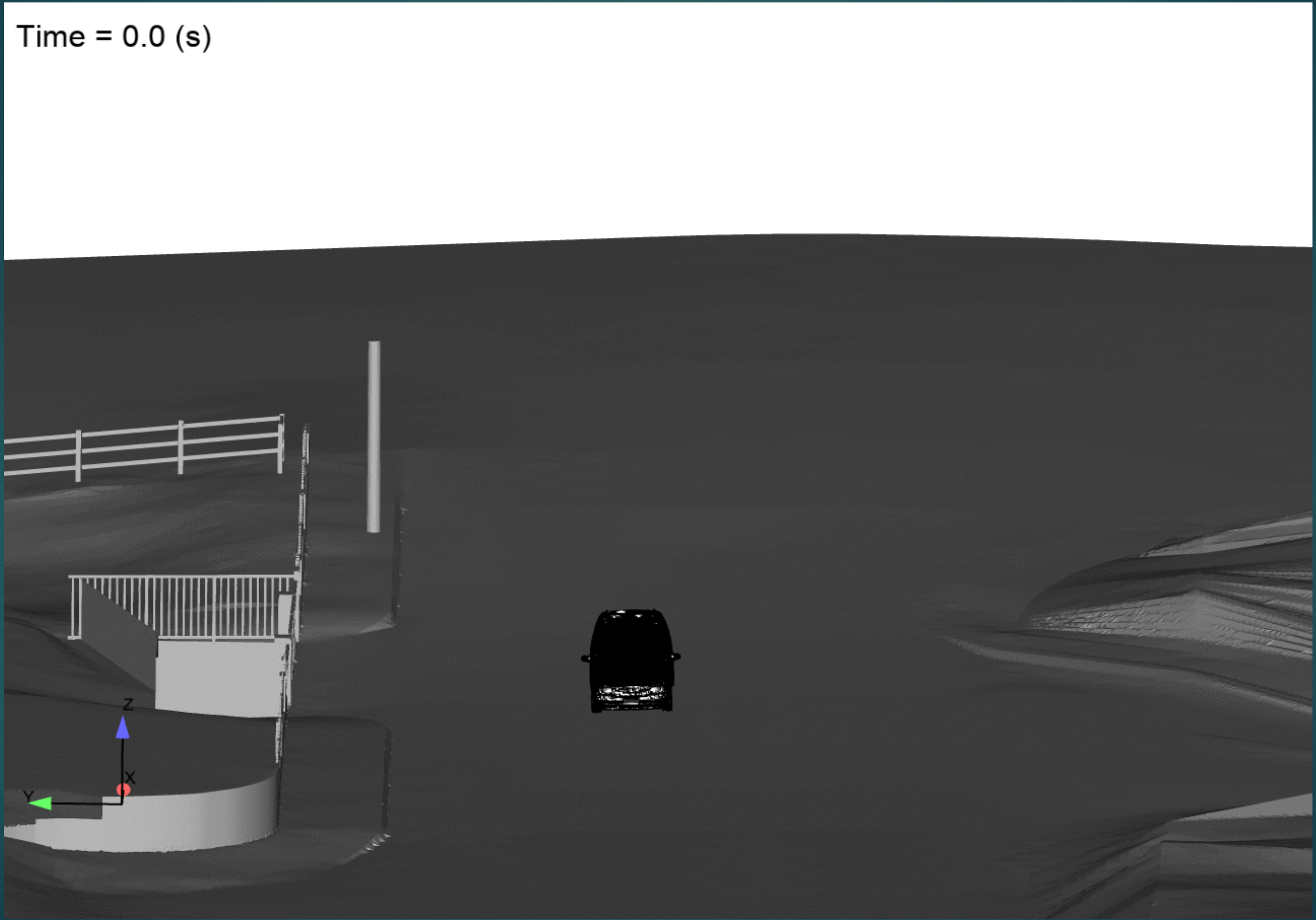
$$\frac{\partial \alpha_i}{\partial t} + \nabla \cdot \mathbf{U}\alpha_i = 0$$

- Interface between the phases is not explicitly computed, but rather emerges as a property of phase fraction field

Baseline with Vehicle
Animation: Water interface

Note: time = 0 sec. is equivalent to 1200s of the velocity profile at the inlet

Baseline with Vehicle
Animation: Water interface



Note: time = 0 sec. is equivalent to 1200s of the velocity profile at the inlet