Transportation Pooled Fund Program TPF-5(552) Quarterly Progress Report

Quarter 2, April – June 2025

prepared by
M. Sitek, S. Lottes, H. Ley

Nuclear Science & Engineering Division, Argonne National Laboratory

July 2025

TRANSPORTATION POOLED FUND PROGRAM
QUARTERLY PROGRESS REPORT

Lead Agency (FHWA or State DOT): _FHWA______________________________________

INSTRUCTIONS:
Project Managers and/or research project investigators should complete a quarterly progress report for each calendar quarter during which the projects are active. Please provide a project schedule status of the research activities tied to each task that is defined in the proposal; a percentage completion of each task; a concise discussion (2 or 3 sentences) of the status, including accomplishments and problems encountered, if any. List all tasks, even if no work was done during this period.

Transportation Pooled Fund Program Project # *(i.e., SPR-2(XXX), SPR-3(XXX) or TPF-5(XXX)*	**Transportation Pooled Fund Program - Report Period:**
TPF-5(552)	☐Quarter 1 (January 1 – March 31) ☒Quarter 2 (April 1 – June 30) ☐Quarter 3 (July 1 – September 30) ☐Quarter 4 (October 1 – December 31)

Project Title:
High Performance Computational Fluid Dynamics (CFD) Modeling Services for Highway Hydraulics

Name of Project Manager(s):	**Phone Number:**	**E-Mail**
James Pagenkopf	*(202) 493-7080*	*james.pagenkopf@dot.gov*
Lead Agency Project ID:	**Other Project ID (i.e., contract #):**	**Project Start Date:**
Original Project End Date:	**Current Project End Date:**	**Number of Extensions:**

Project schedule status:

☒ On schedule ☐ On revised schedule ☐ Ahead of schedule ☐ Behind schedule

Overall Project Statistics:

Total Project Budget	**Total Cost to Date for Project**	**Percentage of Work Completed to Date**

Quarterly Project Statistics:

Total Project Expenses and Percentage This Quarter	**Total Amount of Funds Expended This Quarter**	**Total Percentage of Time Used to Date**

Project Description:

The Federal Highway Administration established an Inter-Agency Agreement (IAA) with the Department of Energy's (DOE) Argonne National Laboratory (ANL) Transportation Analysis Research Computing Center (TRACC) to get access and support for High Performance Computational Fluid Dynamics (CFD) modeling for highway hydraulics research conducted at the Turner-Fairbank Highway Research Center (TFHRC) Hydraulics Laboratory. TRACC was established in October 2006 to serve as a high-performance computing center for use by U.S. Department of Transportation (USDOT) research teams, including those from Argonne and their university partners. The objective of this cooperative project is to:

- Provide research and analysis for a variety of highway hydraulics projects managed or coordinated by State DOTs.
- Provide and maintain a high-performance Computational Fluid Dynamics (CFD) computing environment for application to highway hydraulics infrastructure and related projects.
- Support and seek to broaden the use of CFD among State Department of Transportation employees.

The work includes:

- Computational Mechanics Research on a Variety of Projects: The TRACC scientific staff in the computational mechanics focus area will perform research, analysis, and parametric computations as required for projects managed or coordinated by State DOTs.
- Computational Mechanics Research Support: The TRACC support team consisting of highly qualified engineers in the CFD focus areas will provide guidance to users of CFD software on an as needed or periodic basis determined by the State DOTs.
- Computing Support: The TRACC team will use the TRACC clusters for work done on projects; The TRACC system administrator will maintain the clusters and work closely with the Argonne system administrator's community; The TRACC system administrator will also install the latest versions of the STAR-CCM+ and OpenFOAM CFD software and other software that may be required for accomplishing projects.

Progress this Quarter:
(Includes meetings, work plan status, contract status, significant progress, etc.)

1: Computational Mechanics Research on a Variety of Projects

1.1. CFD Modeling of Infiltration from Ditches into Roadway Embankments

Road ditches are built along roadways to prevent accumulation of rain on the pavement. They intercept rainwater from the pavement as well as from the adjacent land. Water can move freely in a ditch away from the road surface, thus making the road safer for the vehicular traffic. South Carolina DOT expressed interest in studying infiltration of water from ditches into roadway embankments. As part of the current design criteria, it is recommended that the road sub-grades be one foot above the design high-water level to protect the pavement from water infiltrating soil and subgrade under the road from the ditches. The distance between the water surface and bottom of the subbase is referred to as "freeboard". This requirement is put in place to reduce the risk of pavement damage. With changes in roadway design standards, larger shoulders and flatter cross-slopes have moved ditches further out. The additional embankment width may warrant a smaller freeboard without risking negative effects to the subbase. Computational fluid dynamics (CFD) modeling was selected as a method of testing how water infiltrates from the ditch into the embankment and how far the percolating water can travel into the embankment, with the goal to assess if the freeboard and horizontal distance between the road and the ditch can be reduced. A typical cross section through a road, embankment and ditch is shown in Figure 1. Main elements of the design, such as travel lanes and shoulder were marked in the figure, along with the freeboard as defined earlier.

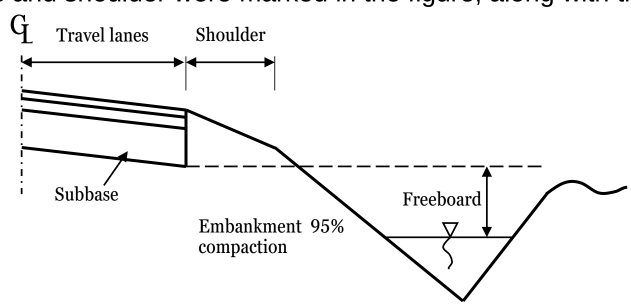

Figure 1. Typical section of a road with marked freeboard - clearance between the water free surface and bottom of the road subbase.

Road ditches are drainage elements with an almost constant cross-section and a longitudinal grade. During rain events, which uniformly fill up the ditch, it can be assumed that the geometry and flow conditions are the same in each cross-section. Moreover, water infiltration and movement in soil is mostly parallel to the cross-section. Therefore, it was decided to represent a thin section of a full-size ditch with elements: (1) soil extending up to the subbase on one side and beyond the ditch on the other, (2) soil extending downwards by a distance that exceeds the extent of the percolating waterfront; and (3) a volume of air above it, big enough to capture the free water surface in the ditch.

A Eulerian Multiphase model with two phases: water and air, and Volume of Fluid approach to compute the interface between the two phases are selected for modeling the open channel flow. The unsteady Reynolds-Averaged Navier-Stokes solver is used with a two-layer realizable k-epsilon turbulence model to solve for the flow field. Water migration through the soil is solved with the use of a porous media model.

Capillary rise is the upward movement of water between soil particles due to a combination of the cohesive and adhesive forces between water and particle surfaces. In fine soil, the upward movement of water is slow but covers a long distance. On the other hand, in coarse soil, the upward movement of water is quick, but covers only a short distance. An attempt was made to include the capillary rise into the CFD soil model.

An experimental set-up used in laboratories to study fluid properties such as viscosity and surface tension involves an empty capillary tube whose lower end is immersed in water was used as a reference case with a known solution. During the experiment, a concave free surface forms inside the tube. Depending on its inner diameter, water rises to a particular height. Water rises due to the adhesion forces between water and the solid, until the forces are balanced by the gravity force. Jurin's law [1] is commonly used to calculate the height of water rise in a capillary tube:

$$h = \frac{2\sigma \cos\theta}{\rho g r}$$

where: σ is surface tension, $\sigma = 0.072$ N/m, θ is contact angle, $\theta = 60$ deg for water, ρ is liquid density, $\rho = 998$ kg/m^3, g is gravity acceleration, $g = 9.81$ m/s, and r is tube radius, $r = 0.25$ mm.

The results of the computations are presented in Figure 2. On the left side, the figure shows the final water level in the tube, a close-up view of the meniscus, and on the right side, the plot of water level vs. simulated time compared with the capillary rise calculated using Jurin's law. The computational solution is comparable with the hand calculations as CFD converges to 29.07 mm, and Jurin's law gives 29.42 mm of capillary rise.

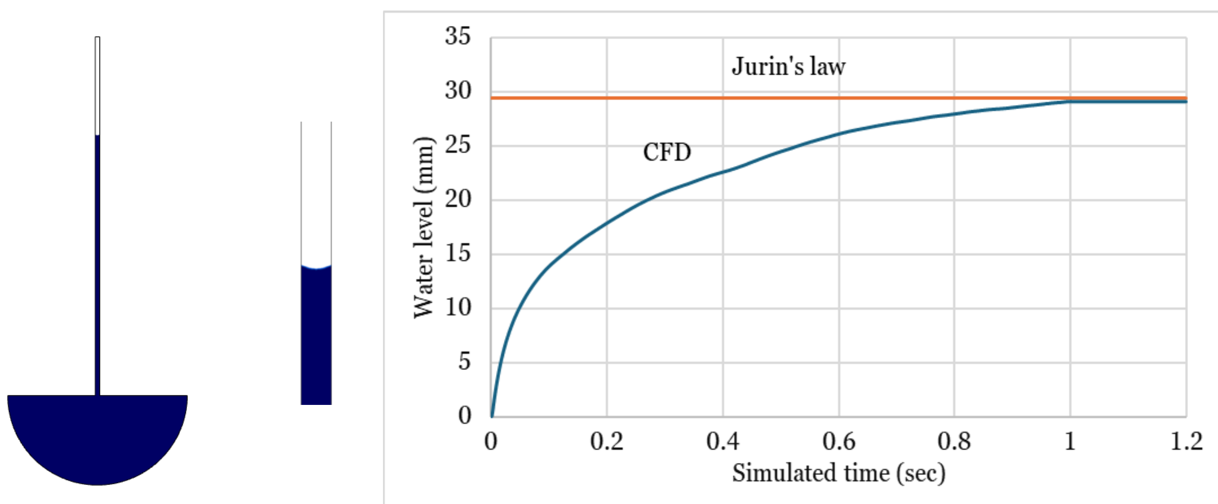

Figure 2. Capillary rise of water in a thin tube. The water level at the end of the simulation, a close-up view of the meniscus, and a plot of the rise of water level in time compared to the Jurin's law.

Capillary rise can be represented using the surface tension model only when the fluid moves between wall boundaries, like the walls of a tube. If this approach is to be used to model capillary rise in soil, then the space between the separate sediment grains needs to be meshed out and the grains' surfaces should be wall boundaries. This approach would lead to a mesh made of many millions of volume cells, which is not feasible to solve with the available resources.

An approach to calculate the capillary rise in porous media was found in [2], where the tube radius is replaced in the previous equation with an “effective radius”

$$r_{eff} = \frac{2(1-\phi)}{\phi\rho\bar{A}},$$

where ϕ is solid volume fraction, ρ is solid density, $\bar{A}$ is specific surface area of the solid fraction. Assuming that the grain shape is close to spherical, the specific surface is calculated from $\bar{A} = \frac{6}{\rho d_{50}}$. For the water surface to rise by h, the pressure applied to the water surface should be

$$p_c = \rho g h = \frac{6\phi\sigma\cos\theta}{(1-\phi)d_{50}}.$$

However, it is much simpler to apply this pressure to the outlet boundary at the top of the tube and ignore the relatively small pressure drop in the column of air above the water surface.

Figure 3 presents a CFD model of a short pipe and aspherical volume underneath it, filled with porous media. The saturated hydraulic conductivity is 0.004, porosity is 0.35, which results in mean grain diameter 0.73 mm. At time zero, the empty space is filled with water, and the tube is filled with air. The boundary conditions marked with grey are no-slip walls, and with orange are pressure outlets. The pressure assigned at the surface of the half-sphere is the hydrostatic pressure. At the top of the tube, a negative (pointing upwards) pressure calculated with the above equation is applied. During the simulation, the water surface moves upward up to 0.111 m. Jurin’s law gives 0.112 m.

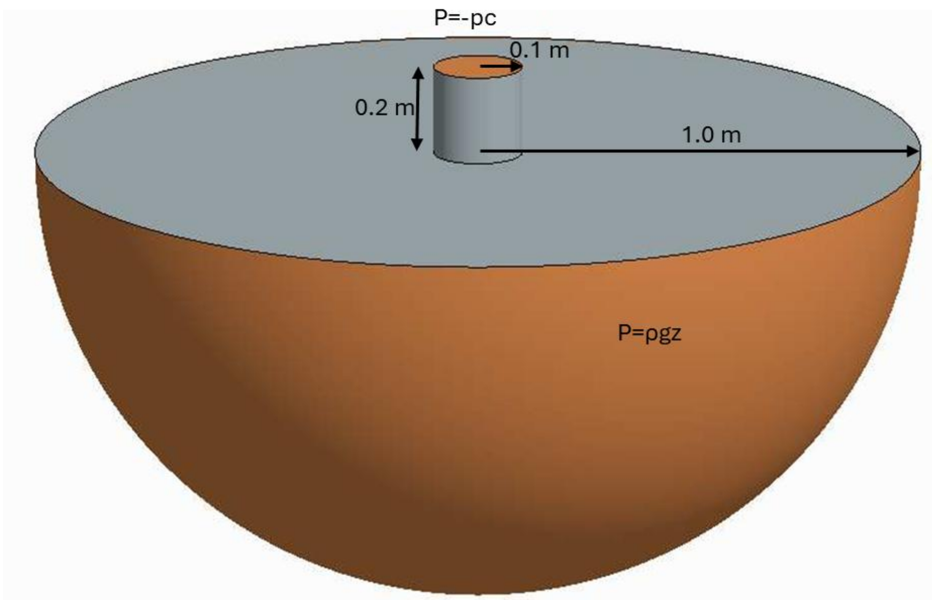

Figure 3. Model used to simulate capillary rise in porous media.

Figure 4 shows a contour plot of the volume fraction of water in the tube at the end of the simulation, and a point plot along the centerline with highlighted location of the water surface (water VOF=0.5) above which the soil is not fully saturated. The shape of the curve compares well with similar results reported in literature [3].

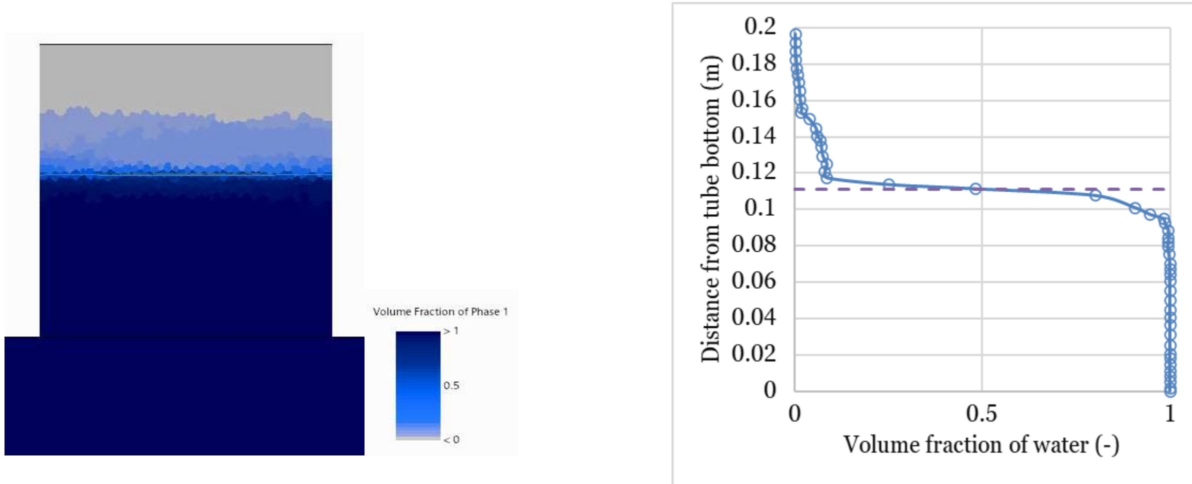

Figure 4. Volume fraction of water in a tube filled with porous medium at steady state, (a) contour plot on a plane section through the computational domain, (b) point plot along the tube centerline.

The findings of this test case will be incorporated into the full-scale models of the roadside ditches and the impact of the capillary rise on the water movement in soil will be evaluated.

References:
[1] Jurin J., An account of some experiments shown before the Royal Society; with an enquiry into the cause of some of the ascent and suspension of water in capillary tubes, Philosophical Transactions of the Royal Society of London, 30, 1718
[2] White, L.R., Capillary rise in powders, Journal of Colloid and Interface Science, vol 90, no 2, 1982
[3] Hird R., Bolton M.D., Clarification of capillary rise in dry sand, Engineering Geology 230, 2017

1.2. CFD modeling of an Ohio DOT Noise Wall Drainage Window for on Grade and Sag Conditions

Noise wall barriers are installed along highways to reduce noise pollution from vehicular traffic in residential areas. To allow for proper drainage, openings with appropriate spacing need to be included at the bottom of the barriers. This study is meant to examine varying window lengths and tapers to the outside, to maximize the hydraulic efficiency and to prevent potential clogging of debris under the barrier section.

A literature review was performed at the beginning of this study. Data useful in validating a CFD model of this flow system was found in a technical report prepared for Florida DOT [4]. The report presents findings of a study which covered experimental measurements of the discharge characteristics of rectangular drainage inlets in temporary barriers. Based on the information provided in the report, the experimental set up was represented in CFD software in a similar manner as in previous studies [5]. The model covers a section of a pavement with varying cross and longitudinal slopes, the opening in the barrier, and the volume above the pavement and under the barrier. Water enters the domain through a vertical surface at one end of the section through an inlet with defined velocity and depth which is calculated from the Manning equation. There are two outlets in the domain, the opening in the wall and the end of the domain downstream of the barrier inlet. Figure 5 shows a view of the computational solution of the flow on a pavement in an example test case. Two quantities of interest are: the ratio of the flow rate intercepted by the window to the total flow, which is the window hydraulic efficiency, and the water depth at the upstream edge of the opening.

Figure 6 shows a comparison of experimental and computational results for (a) the efficiency of the window vs. total flow, and (b) the intercepted flow rate vs. water depth at the window for longitudinal slope 1% and cross-slopes 2% and 6%. In all tested cases, the window interception estimated computationally is higher than obtained in the experiment. Also, the water depth at the upstream edge of the window is greater. Nevertheless, the deviation from the experimental results is relatively low; it exceeds 10% for flows lower than 0.4 cfs, but the two lines converge within a few percent for higher flow rates.

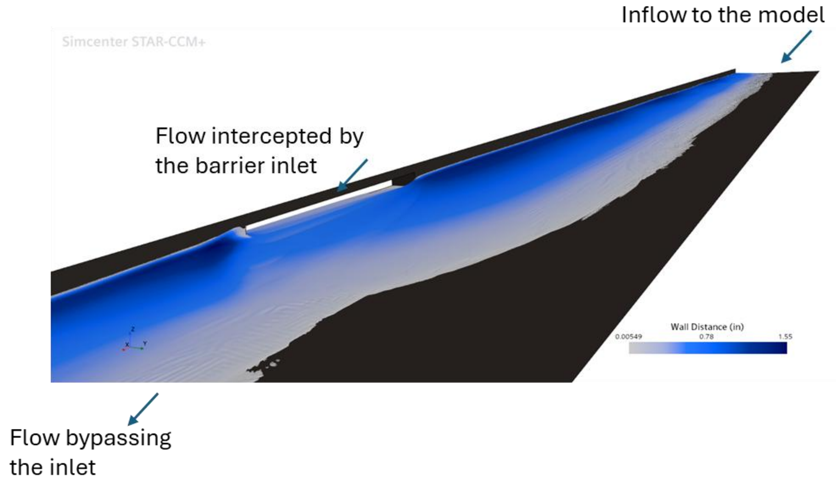

Figure 5. Flow pattern in an example CFD test case.

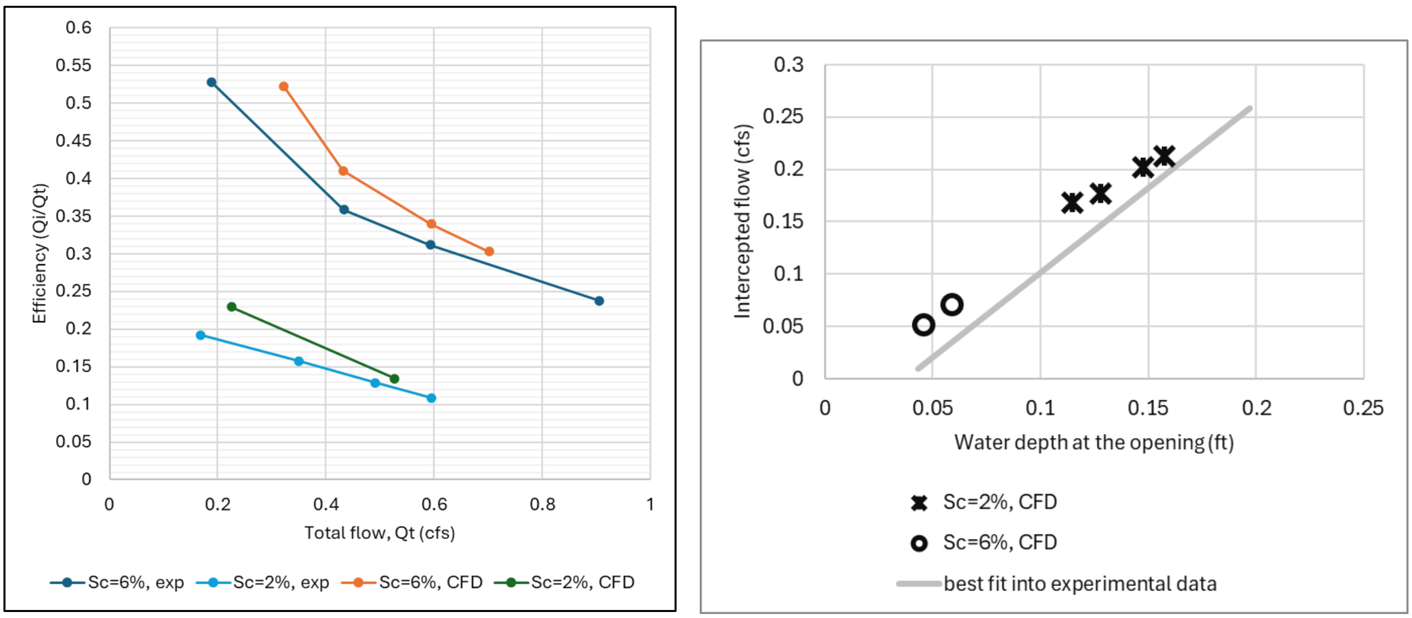

Figure 6. Comparison of experimental and computational results (a) efficiency of the window vs. total flow, and (b) intercepted flow rate vs. water depth at the window.

The findings of this preliminary study show that computational models can simulate water flow on pavement and the flow interception by slotted inlets with good accuracy when compared to physical tests.

References:

[4] Kranc S.C., et al. Experimental Investigation and Analysis of Flow Under Barrier Walls, Report No. BC-353-27, 2003
[5] Sitek M., Lottes S., Computational Analysis of Hydraulic Capacity of Ohio DOT Inlet No. 3 Single Slope Barrier in On-Grade and Sag Locations, ANL-24/73, 2024

Anticipated work in the next quarter:

1: Computational Mechanics Research on a Variety of Projects

- hydraulic analysis of catch basins on grade and in sump
- infiltration of water from roadside ditches

2: Computational Mechanics Research Support

This work will continue.

Task 3: Computing Support

This work will continue.

Circumstance affecting project or budget.

(Please describe any challenges encountered or anticipated that might affect the completion of the project within the time, and fiscal constraints set forth in the agreement, along with recommended solutions to those problems).

None.