Transportation Pooled Fund Program TPF-5(446) Quarterly Progress Report

Quarter 2, April – June 2024

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

Nuclear Science & Engineering Division, Argonne National Laboratory

July 2024

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)* *TPF-5(446)*	**Transportation Pooled Fund Program - Report Period:** ☐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): *Kornel Kerenyi*	**Phone Number:** *(202) 493-3142*	**E-Mail** *kornel.kerenyi@fhwa.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 for DI 36×36, DI 112, DI 125, CB 9, and CB 18 Roadway Drainage Inlets

The state DOTs have been reviewing their catch basin inlet spacing design approaches to assess if the hydraulic efficiency of the system meets the stormwater runoff requirements. Computational Fluid Dynamics (CFD) has been used for many years for various applications and, with advances in the computational power available to researchers, it has become a useful computational tool that can successfully supplement physical testing of existing and new designs of roadway drainage.

South Carolina DOT has expressed interest in modeling multiple inlets used for sump application. These are: grated drop inlets: DI 24×36, DI 112, DI 125, curb opening inlet CB 18, and CB 9, which has curb openings on all 4 sides. Table 1 presents the inlet types and their applications.

Table 1. Inlet types and their application

Inlet type	Application
CB type 9 MH	Ditch sections outside of the clear zone, low area behind curb and gutter sections, and yard drains where debris is not an issue
CB type 18	Primary CB for sags in curb and gutter sections
DI type 112	Ditch sections, grassed medians, and low areas located on controlled access and divided highways
DI type 125	Ditch sections, grassed medians, and low areas located on controlled access and divided highways near traffic, and where traffic is possible
DI 24" x 36"	Low areas within clear zones, yard drains where debris is not an issue, and low area with pedestrian traffic

In the ditch applications, the front slope of 6:1 and 4:1 and back slope of 2:1 were used. For the ditch-type flow on the back side of a fill, the front slopes were 2:1 and back slopes were 2:1, 6:1, 10:1, and 20:1. It was assumed that the modeled section of a ditch has no longitudinal slope. In this case, all water accumulated in the modeled section drains only through the catch basin. The geometry of the DIs and CBs is symmetric with respect to a plane perpendicular to the roadway, and therefore the flow into the drains is symmetric. For this reason, only half of the model was simulated. Figure 1 presents a sketch of an example cross section of a ditch and roadway. Table 2 summarizes the front and back slope combinations analyzed in ditch sections.

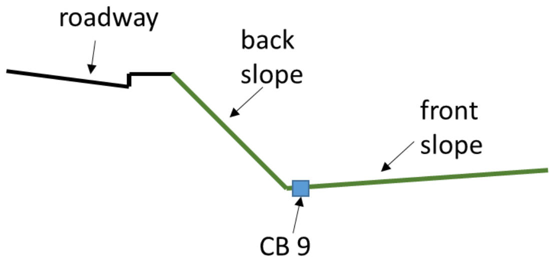


Figure 1 Sketch of a roadway and ditch cross-section.

Table 2 Slope combinations to be analyzed for ditch applications.

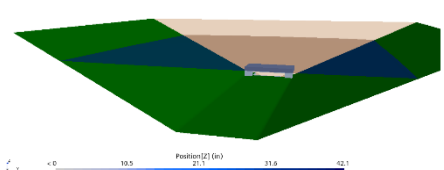
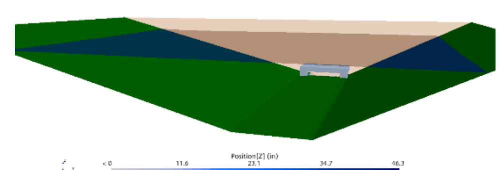
Case #	Front slope	Back slope
1	6:1	2:1
2	4:1	
3	2:1	2:1
4		6:1
5		10:1
6		20:1

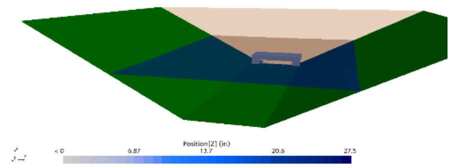
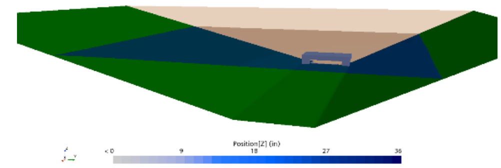
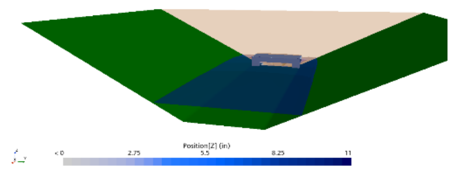
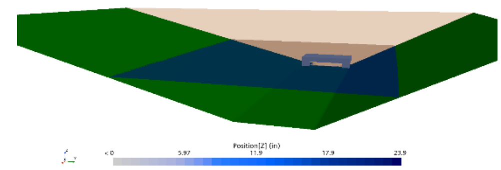
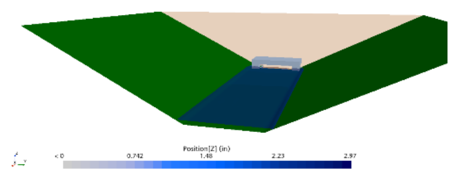
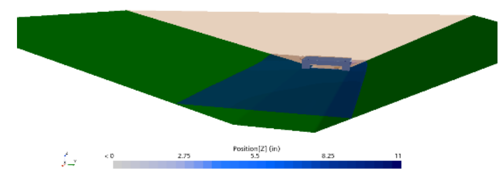
The three-dimensional CFD models were built using Argonne's High-Performance Computing cluster. The Unsteady Reynolds Averaged Navier-Stokes Solver with k-ω turbulence model combined with the Eulerian two-phase model with air for the gas phase, and water for liquid phase, together with the Volume of Fluid physics model, were selected to model the free surface water flow. Detailed information on the formulation and usage can be found in the Simcenter STAR-CCM+ User's Manual.

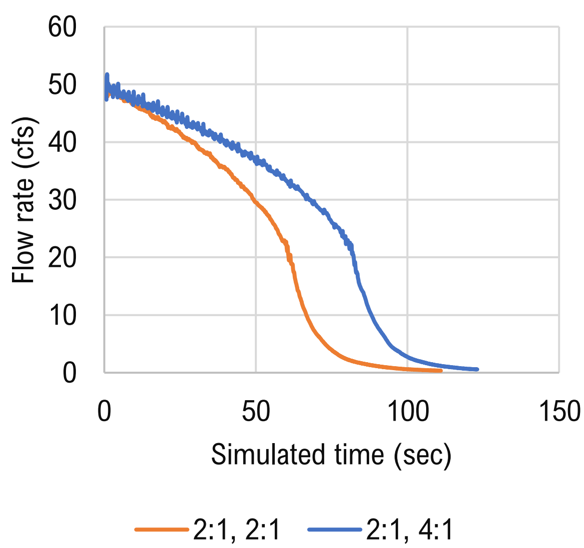
First simulations in the study were performed for the CB type 9, which has 5" by 3' openings on each of its four sides. The geometry of the computational domain represents a section of a ditch as well as the inlet and catch basin underneath. Due to the double symmetry of the CB-9 geometry, half of the model was analyzed, which makes the simulations less computationally intensive but does not introduce any errors to the solution. Two cross-sections of the ditch were presented here: 2:1 - 2:1 and 4:1 - 2:1, which corresponds to the front and back slopes, respectively. Both simulations were initialized with 4.5 feet of water above the bottom of the ditch. The surface roughness of the ditch was initially assumed to be 3 mm, which represents a uniform earth channel. In planned research, the flow resistance coming from the grass, weeds etc. will be accounted for in the model.

Table 3 presents the water surface in the ditch at various times during the simulations for two ditch geometries, and Figure 2 shows the plots of flow rate and water depth vs. simulated time. Note that the water surface drops faster for the slope combination 2:1 and 2:1 than 2:1 and 4:1. The main reason seems to be the smaller initial volume of water in the computational domain, which will be verified in upcoming tests. Despite this difference, the relationship between the flow rate through the catch basin to ponding water depth is very similar, as presented in Figure 3. At water depths below ~0.8 ft, the flow is in weir regime, and above ~0.8 ft it is in orifice regime.

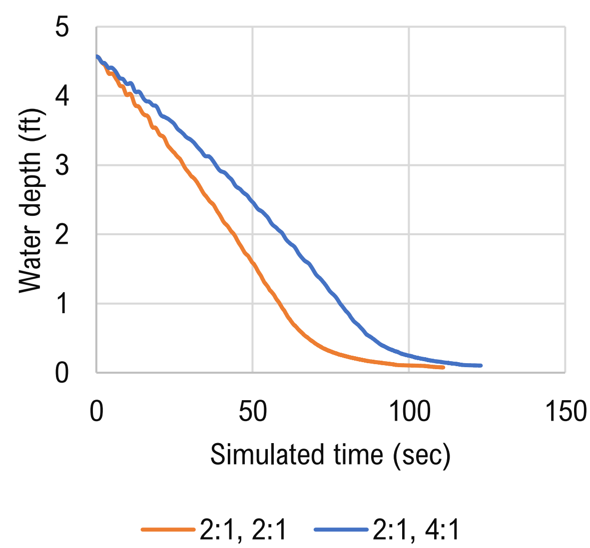
Table 3 Water surface in the ditch at various times during the simulations.

Time	Slopes 2:1 and 2:1	Slopes 4:1 and 2:1
20 sec		

40 sec		
60 sec		
80 sec		



(a)



(b)

Figure 2 Change of the (a) flow rate and (b) water depth during the simulation for two ditch geometries.

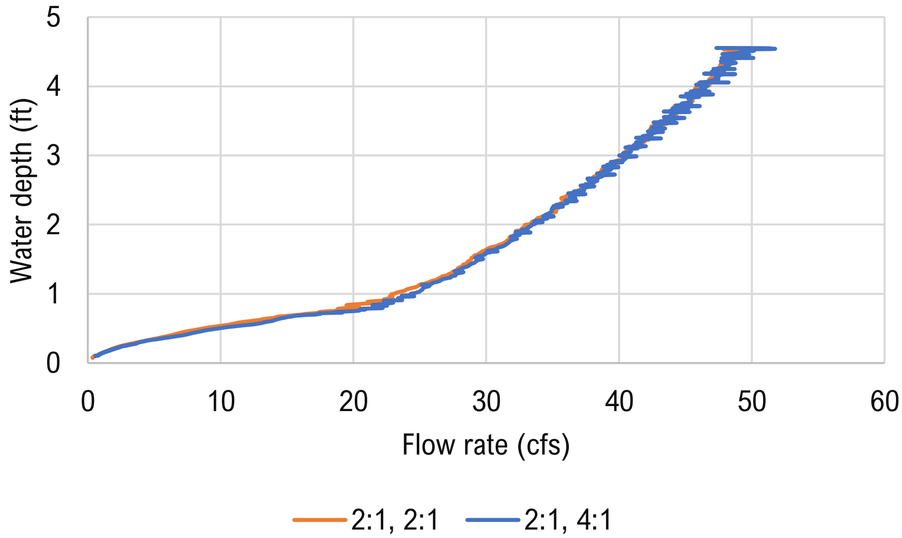


Figure 3 Water depth dependency on the flow rate through the CB-9

Anticipated work next quarter:

1: Computational Mechanics Research on a Variety of Projects

- hydraulic analysis of catch basins on grade and in sump
- analysis of water film thickness on pavements (hydroplaning water film thickness and speed)
- 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.