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

Quarter 1, January – March 2024

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

Nuclear Science & Engineering Division, Argonne National Laboratory

April 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)*	**Transportation Pooled Fund Program - Report Period:**
TPF-5(446)	☒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 and Analysis of the Inlet No. 3 Single Slope Barrier Hydraulic Efficiency

Stormwater runoff from streets and highways is typically captured by drainage structures strategically placed when roadways use curb or curb and gutter systems. These structures include catch basins with grates, inlets, or combination inlets that collect and discharge storm water runoff to buried pipe conveyance systems. The performance of these drainage structures is measured in terms of hydraulic efficiency, which is defined as the percentage of flow captured by the basin as compared to the total flow draining to the structure. Understanding the performance of these drainage structures allows for a safe and economical design which prevents flooding along Ohio roadways.

The hydraulic efficiency for an on-grade Inlet No. 3 Single Slope Barrier, a drainage structure used by Ohio DOT, was analyzed with the use of physical and computational modeling. The full-scale physical model of a section of the pavement with the barrier inlet and grate was built at the Iowa Institute of Hydraulic Research (IIHR), University of Iowa. The test channel modeled a range of flows and longitudinal and cross slopes to simulate typical hydraulic conditions encountered on Ohio highways.

The three-dimensional computational fluid dynamics (CFD) simulations of the tests were performed at Argonne National Laboratory with the use of Siemens' Simcenter STAR-CCM+ software on a high-performance computing cluster.

An example test case represents the following scenario:

- Pavement cross slope 4%,
- Longitudinal slope 1%,
- Flow rate 7.9 cfs (223.7 kg/s),
- Manning coefficient n=0.014, which corresponds to roughness height ~ 2 mm.

Figure 1 presents the mass flow rate through the inlet to the domain, window, grate and outlet (bypass flow). The inflow to the computational domain is kept constant throughout the run. The window and grate interception and the bypass flow are recorded during the run. The simulation is assumed to be finished when a steady state is reached, meaning the flow incoming and outgoing to and from the domain is in balance. The steady-state solution is given in Table 1. The combined efficiency of the grate and the window is 81.7%, which compares very well with the physical test result 88.6%.

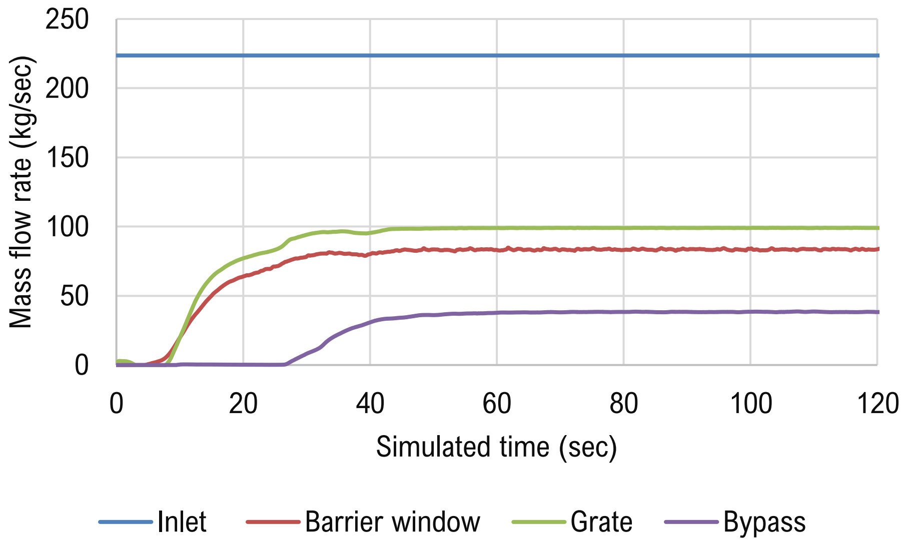

Figure 1. Mass flow rate vs. simulated time during the simulation.

Table 1. Mass flow rate at the end of the simulation.

	Inlet	Barrier window	Grate	Bypass
MFR (kg/s)	223.7	83.7	99.1	38.3
Percent of inflow	-	37.4%	44.3%	17.1%

The following figures are side-by-side comparisons of the physical test and CFD simulations and show a good comparison of the flow details:

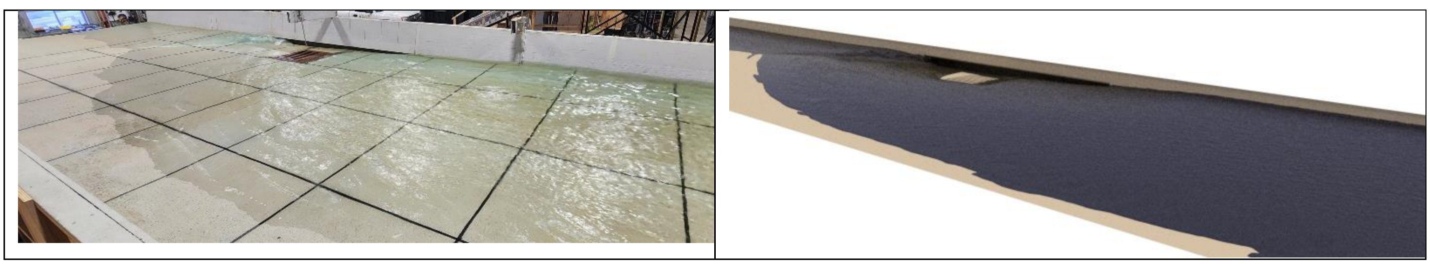

Figure 2. Standing wave at the downstream edge of the barrier inlet.

Figure 3. A similar portion of the grate is overtopped.

Figure 4. Standing wave at the pavement section downstream of the headbox.

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.