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

Quarter 3, July – September 2024

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

Nuclear Science & Engineering Division, Argonne National Laboratory

October 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.)

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.

An important part of this study is modeling the impact of vegetation, such as grass, on the flow in a roadside ditch. After a literature review, it was decided to adopt the approach of adding a sink term to the momentum equation, which accounts for the cumulative drag force from the grass stems. The grass stems in this case are represented by vertical cylinders of the same height and diameter, distributed on the surface of the channel.

The drag force of the grass stems is equal to:

$$\mathbf{F_D} = -\frac{1}{2}\left(\frac{1}{1-\phi}\right)\rho C_D \mathrm{a}|\mathbf{u}|,$$

where according to [Sonnenvald, 2019], the drag coefficient of the cylinders is

$$C_D = 2\left(\frac{\alpha_0}{Re} + \alpha_1\right),$$

and the coefficients α_0 and α_1 are estimated from physical tests:

$$\alpha_0 = 6475d + 32,\ \alpha_1 = 17d + 3.2\phi + 0.5,$$

(the formula for C_{Di} is valid for $\phi \leq 0.4$ and $d \leq 0.025$m), Re is the cylinder Reynolds number, $Re = \frac{|\mathbf{u}|d}{\nu}$, ν is water kinematic viscosity, ϕ is solid volume fraction, d is diameter of a cylinder, a is frontal area of the cylinders (cylinder area perpendicular to the direction of flow per unit volume, m^2/m^3), ρ is water density, and $\mathbf{u}$ is the interstitial velocity.

Per HEC-15, section 4.1 Grass lining properties [HEC-15, 2005], the number of grass stems in a given area for a good grass lining is about 2,000 to 4,000 stems/m^2 (200 to 400 stems/ft^2). For agricultural ditches, grass heights can reach 0.3 m (1.0 ft) to over 1.0 m (3.3 ft). The grass heights, h, are kept much lower near a roadway for safety reasons and are typically in the range of 0.075 m (0.25 ft) to 0.225 m (0.75 ft).

The frontal area for an array of cylinders as shown in Figure 2 is:

$$a = nhd,$$

and the solid volume fraction is equal to the fraction of base area covered by cylinders:

$$\phi = \frac{n\pi d^2}{4}.$$

where n is the number of stems per unit bed or ground area.

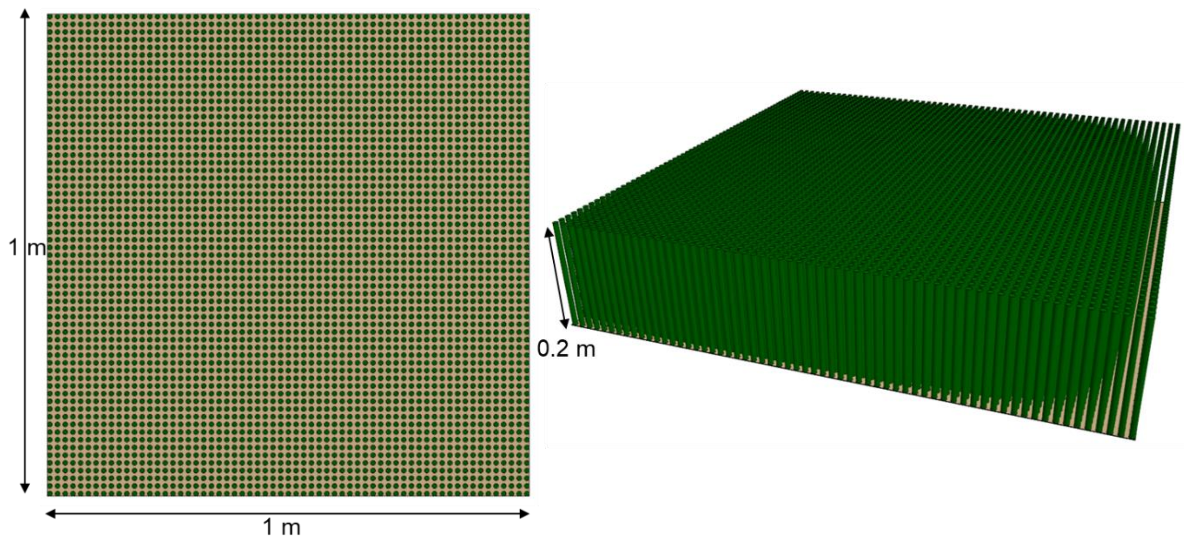


Figure 2. Sketch of 4000 0.2-m-tall rigid cylinders in a uniform grid.

In order to evaluate the approach, an example of flow in a trapezoidal channel was considered with varying stem height and number. The stems are 5 mm in diameter, the height is 5 cm and 10 cm, and the number of stems in a 1m-by-1m square is 2000 and 4000. The flow depth is 0.5 m and flow rate is 1 m^3/s.

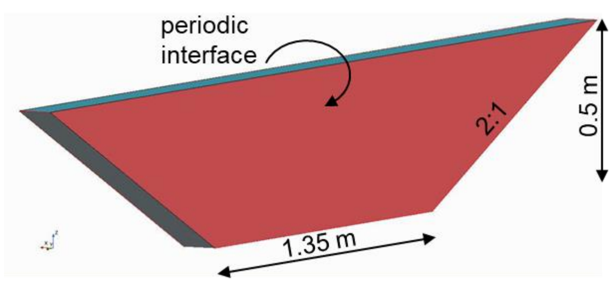


Figure 3. Model of a thin strip of a roadside ditch.

The CFD model geometry represents a thin strip of a long channel perpendicular to the main flow direction, as shown in Figure 3. The three-dimensional CFD models were built using the Unsteady Reynolds Averaged Navier-Stokes Solver with k-ω turbulence model. The top surface is a symmetry boundary. A periodic interface with assigned mass flow rate was used to simulate the flow. The hexahedral mesh is refined in the vertical direction with 2-mm-thick layers of cells. The target cell dimensions on the horizontal plane is 1 cm by 1 cm. The simulation is run until a steady state is achieved.

Figures 4 show the velocity and shear stress contour plots across the channel with a smooth surface of the channel and a channel with vegetation. The maximum velocity magnitude location is on the centerline of the channel, close to the top of the domain. The maximum value increases with increasing resistance of the vegetation which changes with the number and height of the stems. The flow going through the vegetation layer is slower and therefore the flow above it needs to speed up to conserve the mass flow rate. The distribution of the shear stress in the fluid and on the channel surface also differs. In the case of a smooth surface, the maximum shear occurs on the bottom surface of the channel. Introduction of the drag from the bed of cylinders shifts the maximum to the interface between the vegetation and section with unit porosity.

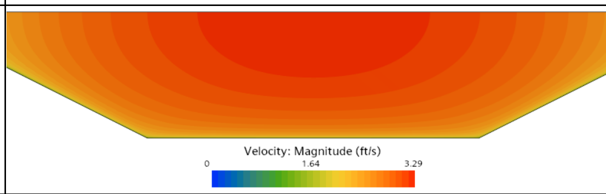
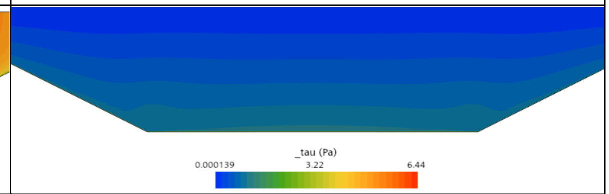
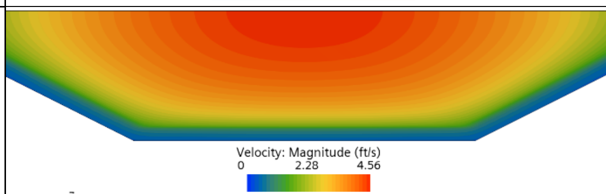
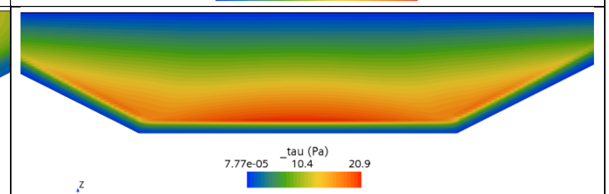
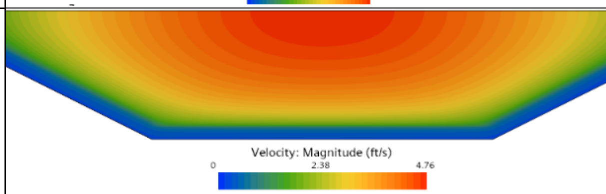
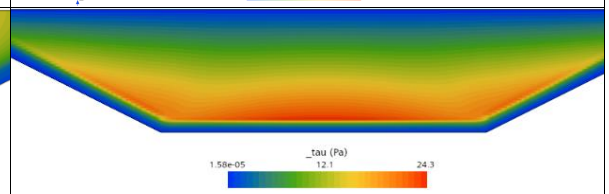
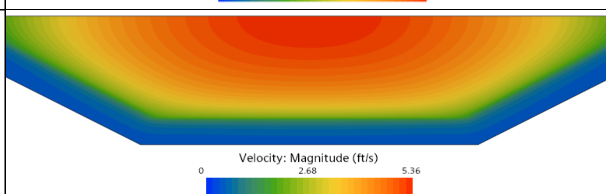
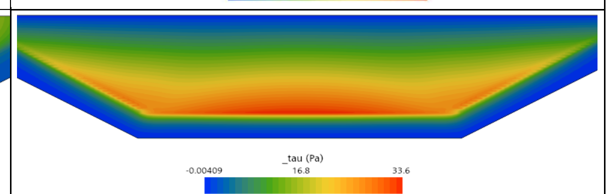
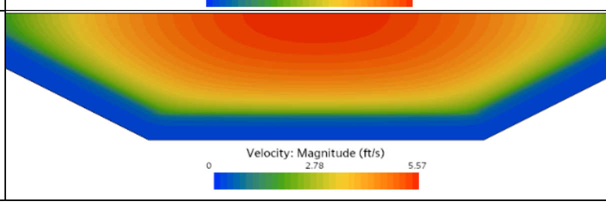
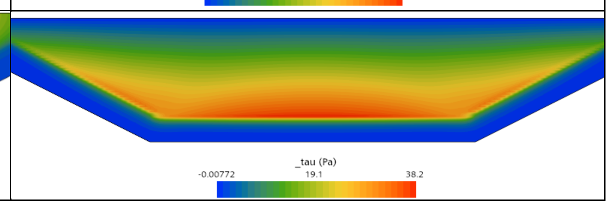
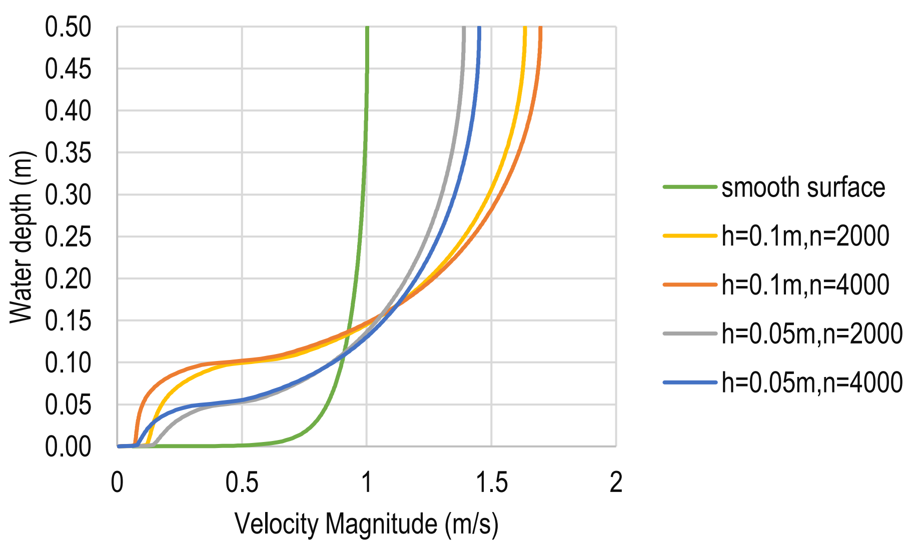
Stem Height (m)	Number of Stems	Velocity Magnitude	Shear Stress Magnitude
0	0		
0.05	2000		
0.05	4000		
0.1	2000		
0.1	4000		

Figure 4. Velocity and shear stress magnitude contour plots on a cross-section through a ditch.



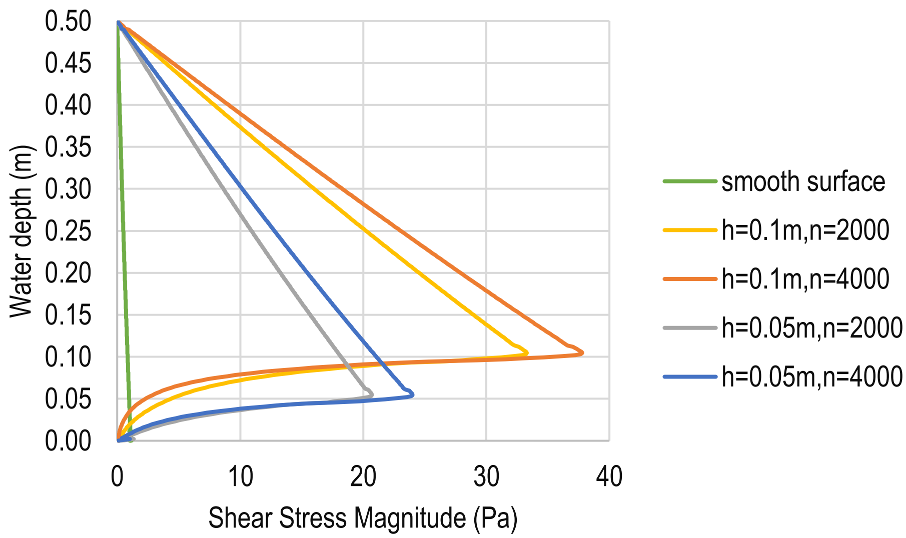


Figure 5. Velocity and shear stress profiles along the vertical centerline.

Figure 5 shows the velocity and shear stress vertical profiles on the centerline. In the smooth wall model, the velocity profile follows the log-law profile. When the porous media is introduced to represent a bed of cylinders, the velocity profile changes within the porous section as compared to the section above. The profiles change slope at the intersection of the two sections and the change is more pronounced with increasing solid volume fraction in the porous medium. For a smooth wall boundary, the shear distribution is close to linear with the maximum at the bed. The addition of the sink term shifts the max to the intersection between the two sections and the maximum magnitude increases significantly.

The approach presented in this report will be further studied and utilized in the modeling of the hydraulic efficiency of catch basins.

References:
[HEC 15, 2005] Hydraulic Engineering Circular No. 15, Third Edition, FHWA-NHI-05-114, 2005

[Sonnenvald, 2019] Fred Sonnenwald, Virginia Stovin & Ian Guymer, Estimating drag coefficient for arrays of rigid cylinders representing emergent vegetation, Journal of Hydraulic Research, 57:4, 591-597, 2019

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.