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

Quarter 1, January - March 2021

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

Nuclear Science & Engineering Division, Argonne National Laboratory

March 2021

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 current 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 OpenFOAM [1] and STAR-CCM+ [2] CFD software and other software that may be required for accomplishing projects.

Progress this Quarter:

1: Computational Mechanics Research on a Variety of Projects

1.1: CFD Modeling of Transition Flow Condition for Grate Inlets in Sag Locations

This study is focused on investigating the flow conditions over grate inlets in sag locations with emphasis on distinguishing three types of flow: weir flow, orifice flow, and the transition between them. A relationship between the ponding depth and the hydraulic capacity of several inlets was established with the use of the computational fluid dynamics software STAR-CCM+ and compared with the charts available in HEC 22 [3].

When there is a pool with an almost level, unbroken surface above the grate, flow is in the orifice type regime. In the weir flow regime, the grate is partially overtopped, and the surface is broken and dips down to the grate. When the flow is in an intermediate state, and the type of the flow is not clearly identifiable, it is said to be in a transition regime.

HEC 22 provides orifice flow and weir flow equations to calculate the relation between flow rate and water depth. Alternatively, Chart 9 A for SI and 9 B for English units can be used. The chart provides two sets of line plots on a log-log scale. One set is for weir flow equation with grate perimeter varying among lines, and the other set is for orifice flow equation with open area of the grate varying among lines. When the flow regime is not clear, HEC 22 recommends drawing a curve between the orifice flow and weir flow lines for the grate of interest to obtain the depth versus flow rate values.

The derivation of the weir flow equation leads to:

$$Q = C_w L d^{1.5}, \tag{1}$$

where C_w is a coefficient that includes effects of viscosity, surface tension, ratio of the weir to approach channel dimensions, nature of weir crest, velocity distribution in the approach, and roughness of the approach and weir, L is weir length, and d is water depth producing the discharge Q.

The orifice flow equation is derived from Bernoulli equation along a streamline from water surface above the grate to water discharge point at the grate with the assumption that the starting point of a streamline is on a still water surface above the drain and the end point of the streamline at the elevation of the grate with both end points at atmospheric pressure. The capacity of a grate is then:

$$Q = C_0 A_0 \sqrt{2gd}, \tag{2}$$

where C_0 is a coefficient that accounts for the orifice shape, friction losses, etc., A_0 is the open area of the grate, g is gravitational acceleration, and d is the mean water depth above the grate.

The orifice coefficient accounts for a variety of losses and depends on the shape of the orifice. In HEC 22, factor C_0 is constant and equal to 0.67. The open area, A_0, is calculated as the area of the grate minus the frame multiplied by the open area ratio that was established in experiments for several grate types.

An alternative procedure is proposed to use the total area of the grate minus the frame and let the losses and variations for different grate geometry be accounted for via the orifice coefficient. This would require providing orifice coefficient values for various types of grates, which can be accomplished via experiments combined with CFD modeling.

Figure 1 shows a modified HEC 22, Chart 9B, which combines plots of the weir and orifice flow equations for grates of various types and sizes. The modification extended the weir and orifice flow lines all the way across the chart.

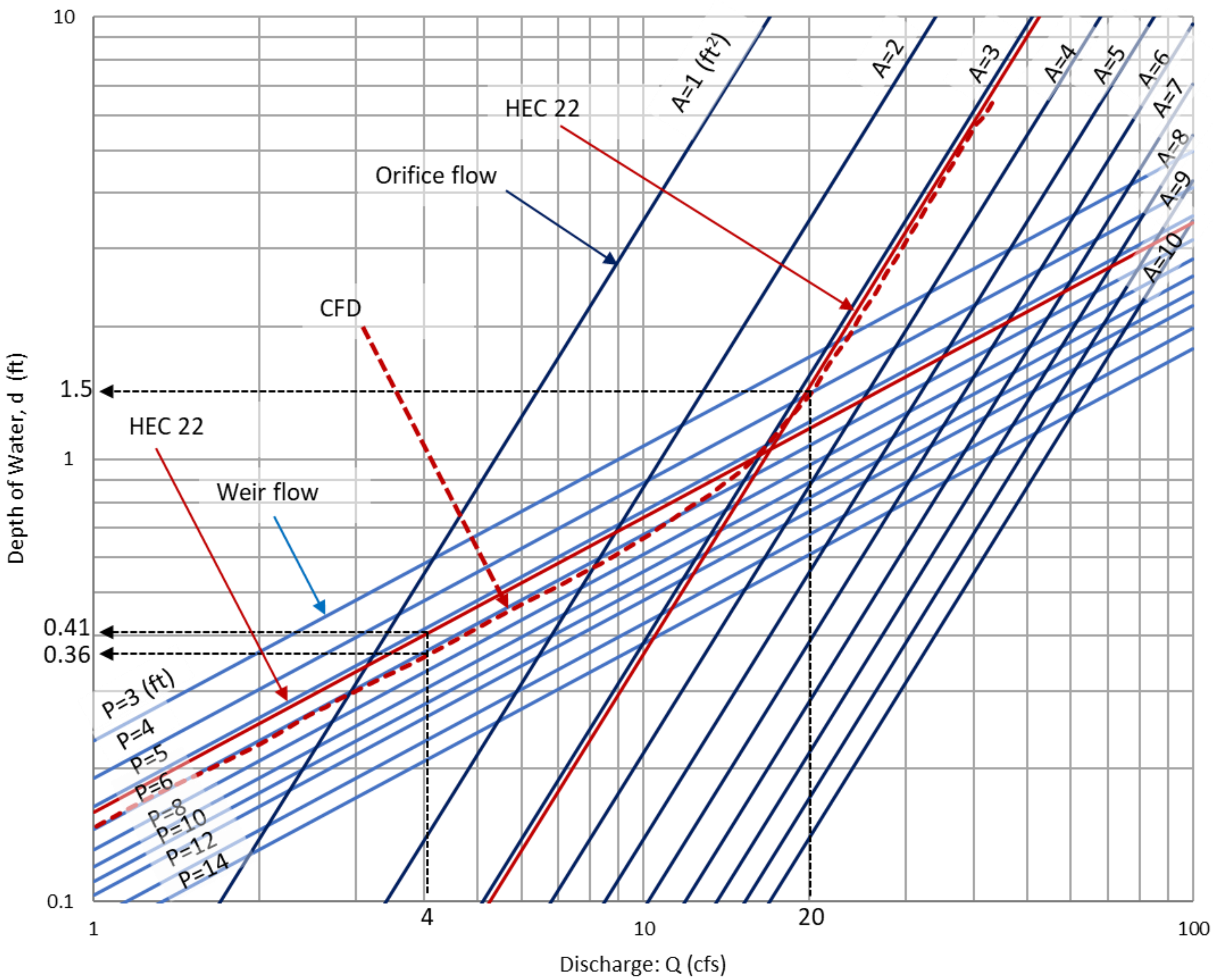
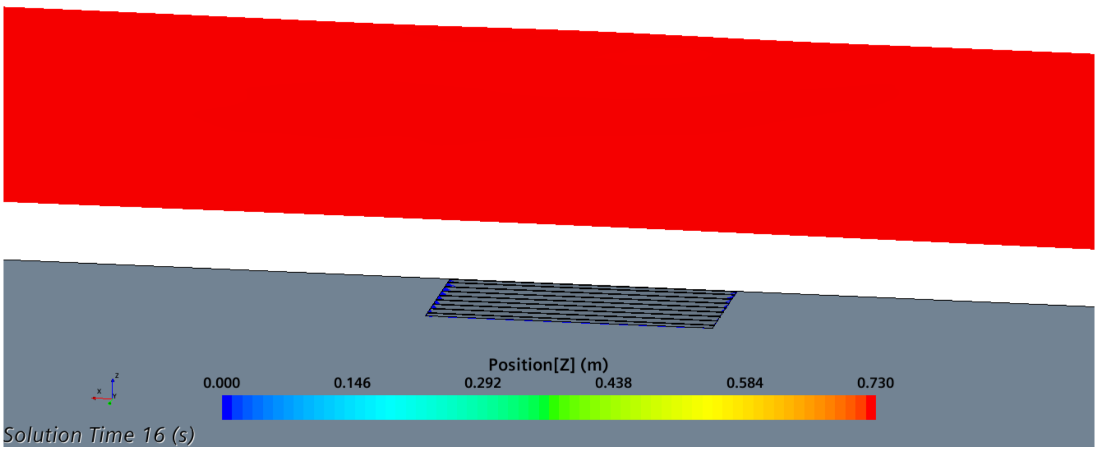


Figure 1. Chart 9B – grate inlet capacity in sump conditions (English units).

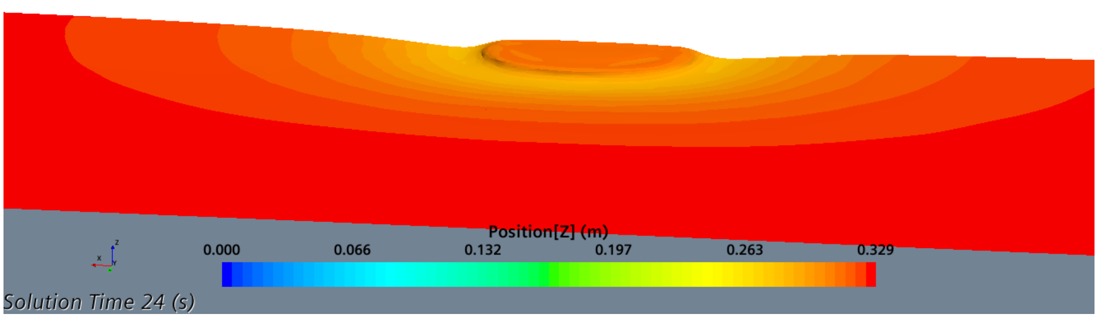
When using Chart 9 A or 9 B, one procedure for obtaining water depth from discharge when the flow may be in transition between orifice and weir flow is to use the greater depth from the orifice and weir flow lines for a given case. Two example cases are shown in Figure 1 for a P50 grate. The P50 grate has length 2.6 ft and width 1.3 ft which gives a grate area of 3.4 ft^2. The open area ratio is 0.9, leading to an open area of 3.1 ft^2. The perimeter of the grate is 5.2 ft. The solid red lines in Figure 1 are the orifice and weir flow lines for this specific geometry. The dashed red line plots the CFD flow results for this geometry. Two example flow rates are marked in the plot, Q=4 cfs, and Q=20 cfs. At a discharge of 20 cfs, the orifice flow line is above the weir flow line, indicating an orifice flow condition, and the water depth at that discharge can be read via the dashed black line from 20 cfs as 1.5 ft. Within the accuracy that can be read from the chart, the 1.5 ft depth matches the CFD result for this condition. At a discharge of 4 cfs, the weir flow line is above the orifice flow line, and the depth can be read as 0.41 ft from the HEC 22 weir flow line for this geometry. For 4 cfs, the CFD result is 0.36, underpredicting the HEC 22 value by about 12 percent. The dashed red CFD result line is nearly straight for both weir and orifice flow conditions very close to the intersection point of weir and orifice flow lines for this case. Using the maximum of the weir and orifice flow values in the transition between flow regimes appears to give a result of sufficient engineering accuracy.

Two types of CFD simulations were developed. In the first approach, water can drain from the initially filled tank only through the grate, there is no refill of the domain and conditions are transient with the water level dropping with time. This model allows for representing a continuous relationship between the water depth above the grate and discharge. At each time step, the mass flow rate through the grate and average water depth is recorded. Figure 2 shows three screenshots of the simulation results: when the flow is in the orifice (at 16 seconds of simulated time), transition (at 24 seconds), and weir (at 37 seconds) flow regimes.

(a) Orifice flow condition above a P50 grate:



(b) Transition flow condition above a P50 grate:



(c) Weir flow condition above a P50 grate:

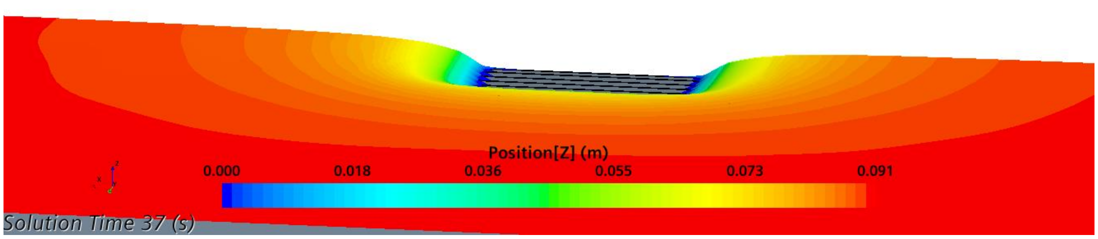


Figure 2. Orifice, transition, and weir flow regime above P50 grate simulated in STAR-CCM+

In the second modeling approach, water can drain from the initially filled tank only through the grate, and the tank is continuously refilled during the simulation to maintain the target depth. The simulation is run until a steady state flow through the grate is reached. In one simulation, discharge is established for one water depth, therefore multiple runs must be performed with changing target ponding depth to find a fit between the data points and in result, a relation between the discharge and water depth.

The results from the two CFD models and HEC 22 equations are compared in Figure 3 for the example of a P50 grate. P50 is a grate with vertical bars, whose open area is constant along the height of the bars. The red line is a solution to HEC 22 equations for weir and orifice flow, the data points from the transient CFD simulation form the green line, and the blue points are results from the steady state CFD simulations.

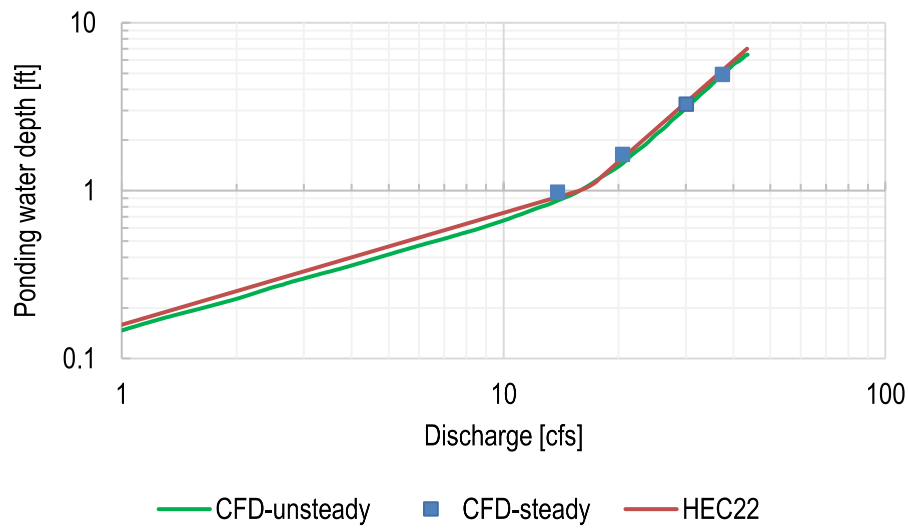


Figure 3. Ponding water depth vs. discharge according to HEC 22 equations and CFD steady and transient simulations for a P50 grate

Transient CFD models were also developed for a curved vane grate with varying dimensions. Figure 4 shows a comparison between the CFD model results and the HEC 22 estimate for a 2 ft by 4 ft grate. In this case, the two estimates give comparable relationships between the ponding depth and discharge through the grate for the weir flow regime. When the flow is in the orifice flow regime, the CFD model gives greater values of the flow rate than HEC 22 equations for a given water depth. CFD model results show that curved vane grates may perform better in sump condition than HEC 22 indicates. The opening ratios given in Chart 9 in HEC 22 may be too conservative for grates with non-vertical bars. To confirm this finding, flume experiments will be performed at the Turner-Fairbank laboratory.

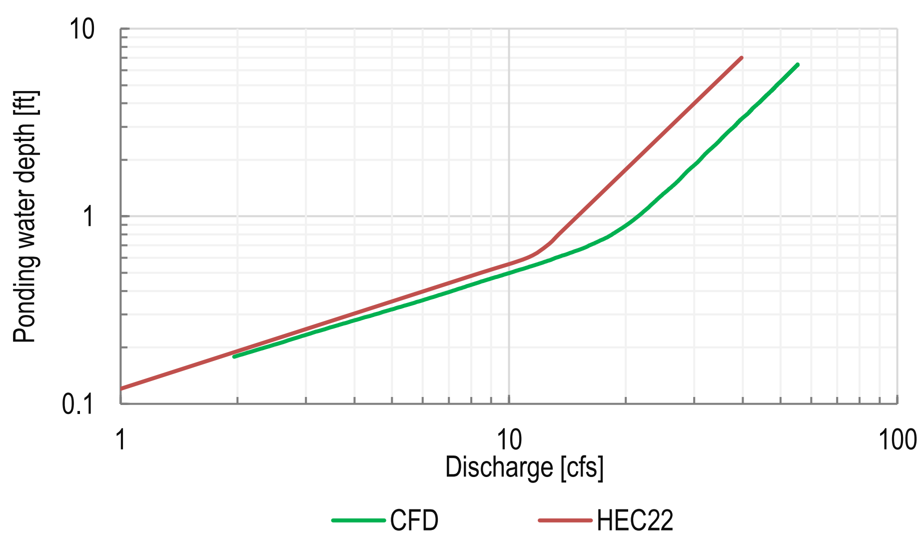


Figure 4. Ponding water depth vs. discharge according to HEC 22 equations and CFD transient simulations for a curved vane grate

References

[1] www.openfoam.org
[2] https://www.plm.automation.siemens.com/global/en/products/simcenter/STAR-CCM.html
[3] Hydraulic Engineering Circular No. 22, Third Edition, URBAN DRAINAGE DESIGN MANUAL, FHWA-NHI-10-009, September 2009

2: Computational Mechanics Research Support

Argonne staff ran nationwide videoconferences every other Thursday that were open to state Department of Transportation staff and university researchers supported by the Federal Highway Administration or state DOTs. The videoconferences provide a venue to discuss approaches and issues related to hydraulics modeling projects. Topics during this reporting period included, but were not limited to:

- new methodologies of scour modeling,
- approaches to modeling and mitigating hydroplaning risk,
- hydraulic analysis of catch basins.

3: Computing Support

Routine cluster maintenance including software and hardware upgrades, security patching against cyber threats, and development of custom tools to increase users' productivity. Currently working on upgrading the TRACC clusters to support the latest scientific and engineering software utilizing industry's best practice guidelines in open-source software and virtualization.

Anticipated work next quarter:

1: Computational Mechanics Research on a Variety of Projects

- development of a new methodology for riverbed scour,
- hydraulic analysis of a catch basins,

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.