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(279)*		**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+ 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: Development of an application to estimate riverbed scour around bridge piers and abutments – NextSCOUR

The advanced high performance computing cluster operated by Argonne National Laboratory for USDOT has been used over the past 18 months to develop an application that is able to calculate 3D scour patterns around complex bridge piers and abutments based on various erosion models and erosion shear stress measurements from experiments.

The high-level control application has been written in Python, and utilizes the well-supported OpenFOAM Version 8 application to iteratively solve for the flow conditions and flow interaction with an eroding riverbed using a K-epsilon turbulence model. The erosion rates are calculated based on wall shear stress results from the CFD application, which are averaged over a time period to smoothen local variations in the flow, primarily eddy shedding from piers and the other bluff bodies in the flow. To cut down the number of CFD solutions that have to be repetitively solved for each change in the riverbed geometry, much of the erosion is based on a finer representation of the erosion processed outside of the CFD application. This procedure consists of iterations between small erosion steps using the distribution of wall shear stresses from the previous CFD run, and a sand slide algorithm that lets material flow downhill based on a specified angle of repose while preserving mass balance. The intention in the coming months is to add sediment transport and a redeposition model to the software application, which has been prototyped using the commercial StarCCM+ application available to the team.

The goal is to make this software application widely available to researchers, which is possible because it is purely based on open-source software applications such as Python, OpenFOAM, OpenSCAD, and OpenMPI.

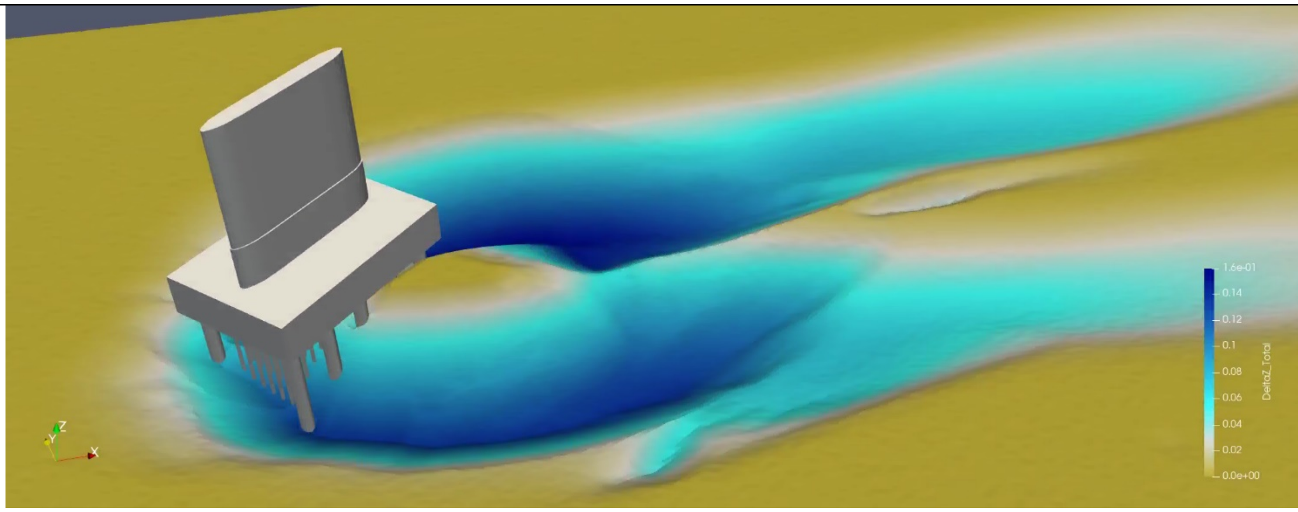
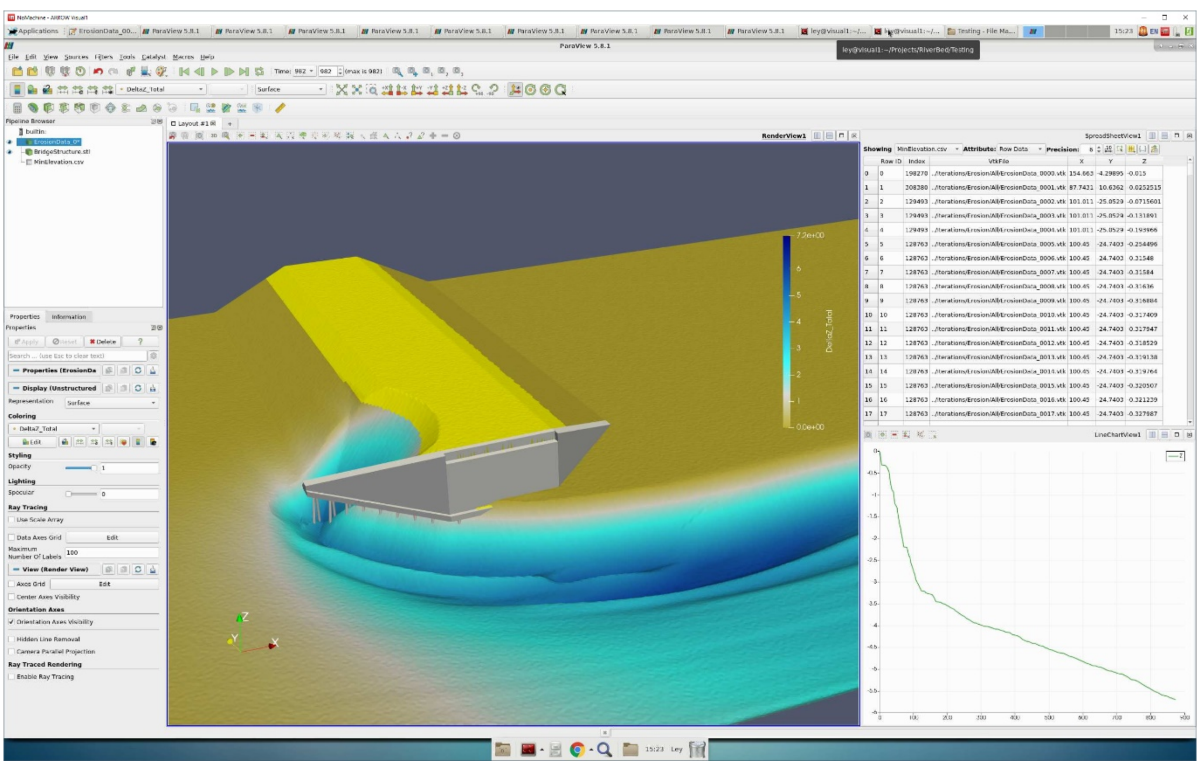


Figure 1: 3D CFD scour bed development using NextSCOUR on TRACC's HPC cluster

Figure 1 shows the shape of the riverbed based on a flume experiment with a complex pier, towards the end of running the 3D computational model. At the beginning of the simulation, the circular and H-shaped piles are all embedded under a flat layer of sand. An initial CFD run of this geometry using the exact flume flow conditions leads to a well-defined distribution of wall shear stresses around the pier model, which are then applied to the geometry of the riverbed based on erosion functions. This effectively changes the bottom of the CFD model domain, and a new mesh is automatically generated based on the newly deformed riverbed. OpenFOAM allows for mapping the previous flow solution onto the new volume mesh of a slightly changed river bed geometry as a starting point for the next CFD simulation of the flow based on the new geometry, cutting down on the computational effort.

Bed displacements in any one cycle between the CFD computation and the Python program that displaces the bed are limited to the scale of the computational cell size in the volume mesh. Under this constraint, the scour cycling process is repeated around 100 times to reach equilibrium scour, until the riverbed does not change its shape in any significant way any longer. The model can run on rather fine meshing resolutions, but needs an HPC system to do so. The development



of each computing mesh takes time, as does the solution of the flow field based on the new geometries. The algorithm is also being used on multi-core desktop machines at lower resolutions, and is quite scalable.

Figure 2: Visualization of results in ParaView, in this case the analysis of an abutment in full scale geometry

Figure 2 shows the example of a wing wall abutment under severe flood conditions. Using CFD models, runs can be configured for various geometries for comparison of outcomes. For example, the CFD model can be initially configured for flume conditions (inlet velocities, water levels, D50 sand grain diameters, and so on). Results can then be validated against physical models in flume experiments. We expect that this will lead to a calibration of the model, and to a greater understanding of the sensitivity of the scour to the various conditions, as it is easy to replicate runs for a matrix of small variations of the input parameters. For example, the scour depth can be determined as a function of variations in inlet velocity and grain size, by running multiple cases simultaneously on a scalable HPC system.

After analyzing the flume conditions and calibrating the models, the CFD approach allows for upscaling of the experimental setup to real world, full size geometries. It is obviously not possible to run experiments at full scale, but results from the CFD cases can be used to confidently estimate real world scour depths, and can be compared to results from current engineering practices.

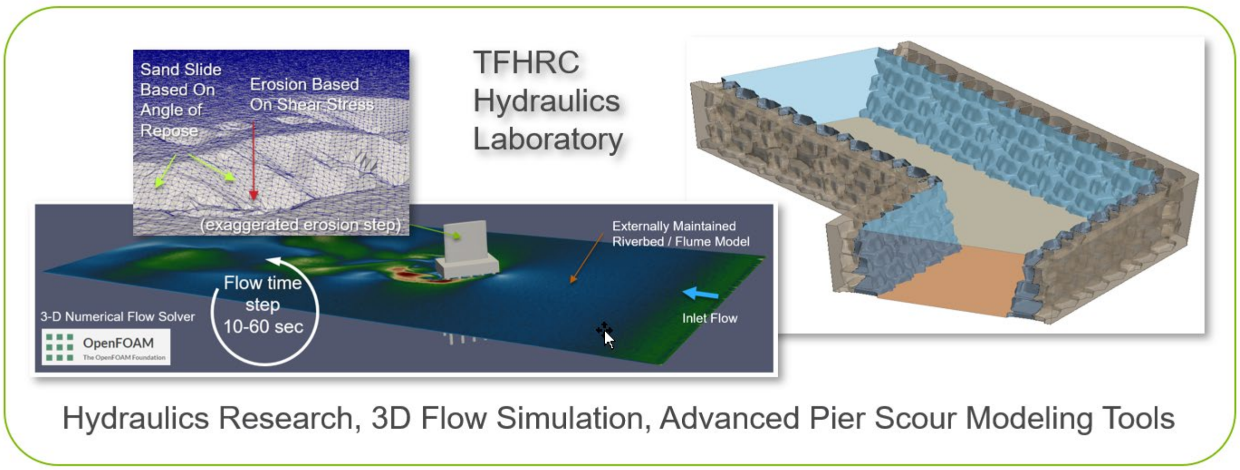


Figure 3: Conceptual illustration of the NextSCOUR algorithm and integration with TFHRC Hydraulics Laboratory activities

The lab capabilities at TFHRC are well suited to perform a series of experiments that the CFD models can be calibrated against. Researchers from the TFHRC Hydraulics Laboratory are in the process of setting up scaled down scour experiments using 3D-printed bridge components, while having access to TRACC's HPC systems to run the numerical models in direct comparison. Remote access includes fast remote desktop access, as well as other advanced network configurations to make access to the HPC system effective from both TFHRC offices and home locations. This configuration proved highly effective over the past year when the research team had to work remotely from home, and productivity was basically not affected by the current teleworking arrangements.

2: Computational Mechanics Research Support

Argonne Transportation Research and Analysis Computing Center (TRACC) computational mechanics 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's productivity. Argonne TRACC is 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.

The TRACC clusters are being replaced with a single integrated platform that is currently going into operation. This new system is based on a set of brand-new administrative machines and a new 480TB parallel file system, and much improved remote access capabilities. Computing hardware has been transitioned from the previous clusters into this new integrated environment.

Anticipated work next quarter:

1: Computational Mechanics Research on a Variety of Projects

- continued development of a new methodology for riverbed scour
- continued hydraulic analysis of a catch basins
- continued analysis of water film thickness on pavements

2: Computational Mechanics Research Support

This work will continue.

Task 3: Computing Support

A more detailed overview of the HPC cluster reconfiguration will be provided as part of the next quarterly report.

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

None.