Mississippi River Port Dynamics at West Memphis

Andy McCoy, Computational Fluid Dynamics Business Class Lead, HDR, Des Moines, Iowa, andrew.mccoy@hdrinc.com
Mark Forest, Floodplain Practice Leader, HDR, Reno, Nevada, mark.forest@hdrinc.com
Gary Brunner, Senior Water Resources Technical Advisor, HDR, Sacramento, California, gary.brunner@hdrinc.com
Preston Snyder, Chief, River Engineering Section
USACE, Memphis, TN, preston.a.snyder@usace.army.mil

Holly Enlow, USACE, Memphis, TN, holly.k.enlow@usace.army.mil

Extended Abstract

Introduction

The City of West Memphis, Arkansas owns a port located along the Mississippi River designed to support industrial transport and commerce in the region (oil and gas, fertilizer, construction, agriculture) (Figure 1 and Figure 2). Due to site specific hydrodynamics, conditions become very difficult for barge operators to navigate into and out of the port safely above a river flow of 1,000,000 cfs. During the years 2020 and 2021 the port was closed more than six months each year. The City of West Memphis, Arkansas, performed a planning study to investigate the underlying cause of the un-safe conditions, recommend short and long-term solutions, and test the efficacy of the solutions.

This abstract describes the development and results of modeling tools to a computational fluid dynamics (CFD) model that considered the unique conditions that exist due to the natural river channel bathymetry and site features. Port improvements were conceptualized and evaluated.

This following were documented:

- The riverine context for bank improvements
- The use of the 2D model to screen potential hydraulic alternatives and to inform the CFD boundary conditions
- Confirmation that the HEC-RAS model and the CFD model were reasonably reproducing known velocity conditions by comparing acoustic doppler current profiler (ADCP) information
- Existing conditions, velocities, and flow patterns near the docking area
- The results of design alternatives to improve flow patterns near the docking area

Figure 1. Dock configuration at low water with barge docked (flow direction arrows in blue)

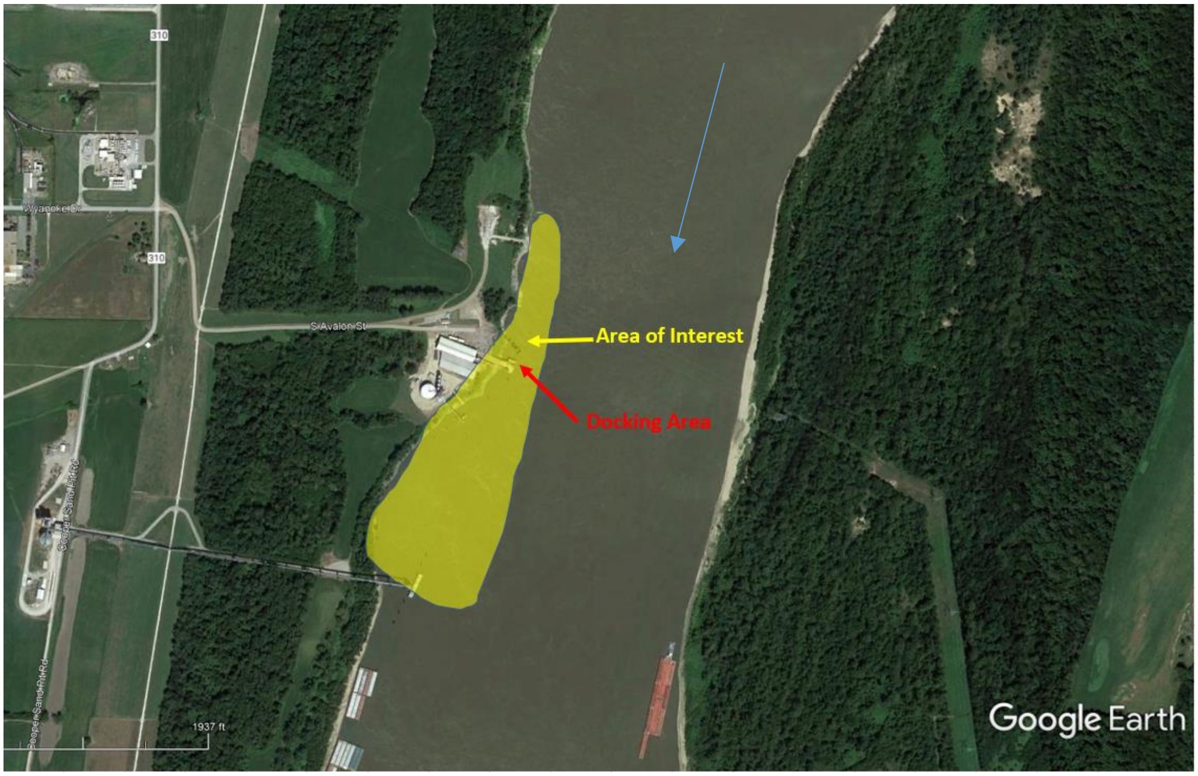

Figure 2. Area of interest – docking area (West Memphis, AR) (Flow Direction in blue)

Riverine Context

Understanding the context in which potential improvements would be made is essential. Significant changes to bank location have taken place over the past 50 years. The port improvement study has been performed at the same time as a USACE bank repair and stabilization project in the reach which has provided some unique opportunities to take advantage of and enhance of each organizations mission. Any modification to the project will require coordination and compliance with floodplain regulations, local and federal levee approvals, cooperation with USACE bank repair and stabilization program as well as the USACE Navigation project.

Model Development and Validation

A fully two dimensional (2D) model was developed for the project reach using the US Army Corp's Engineers' HEC-RAS Version 6.2. HEC-RAS has the capability to model flow in two dimensions using a depth averaged velocity. A model base grid cell sizing of 60 feet was chosen to allow for sufficient detail to capture flood wave dynamics with reasonable runtimes. Refinement regions with smaller grid sizes were added to: increase the details as needed near the port; to better capture the behaviors at the region where the flow separations occur; and to capture changes in roughness associated with river bank and port features. Breaklines were added at hydraulically significant high features and the flow separation regions.

FLOW-3D is a commercially available, three-dimensional (3D), CFD modeling application developed and supported by Flow Science, Inc (Flow Science 2021). It has been used to evaluate hydraulic performance and characteristics for a variety of natural and engineering flows, including river groynes (Choufu et al 2019) and circular pile groups (Deaneshfaraz et al 2021).

A model-fitted mesh was developed for the site. A known discharge was applied to the CFD model's upstream boundary. The upstream water surface elevation was solved as naturally occurring, sub-critical boundary. A known water surface elevation was applied to the downstream boundary. In total, the model domain encompasses an area approximately 2.5 by 7 miles, with a 160-foot thickness. In total, the number of elements in the final mesh is nearly 15,000,000. An upstream and downstream area are modeled with shallow water equations.

The 2D HEC RAS model and the FLOW-3D model were validated to the extent possible with Aerial Photography (Figure 3) and USACE ADCP (Figure 4 and Figure 5) velocity data for several different flow rates. This provided insight into how well each model performs as well as understanding limitations associated with each method.

Figure 3. Google image on April 22, 2015, with flow of 1,000,000 cfs compared to HEC-RAS 2D model result with flow at 1,000,000 cfs.

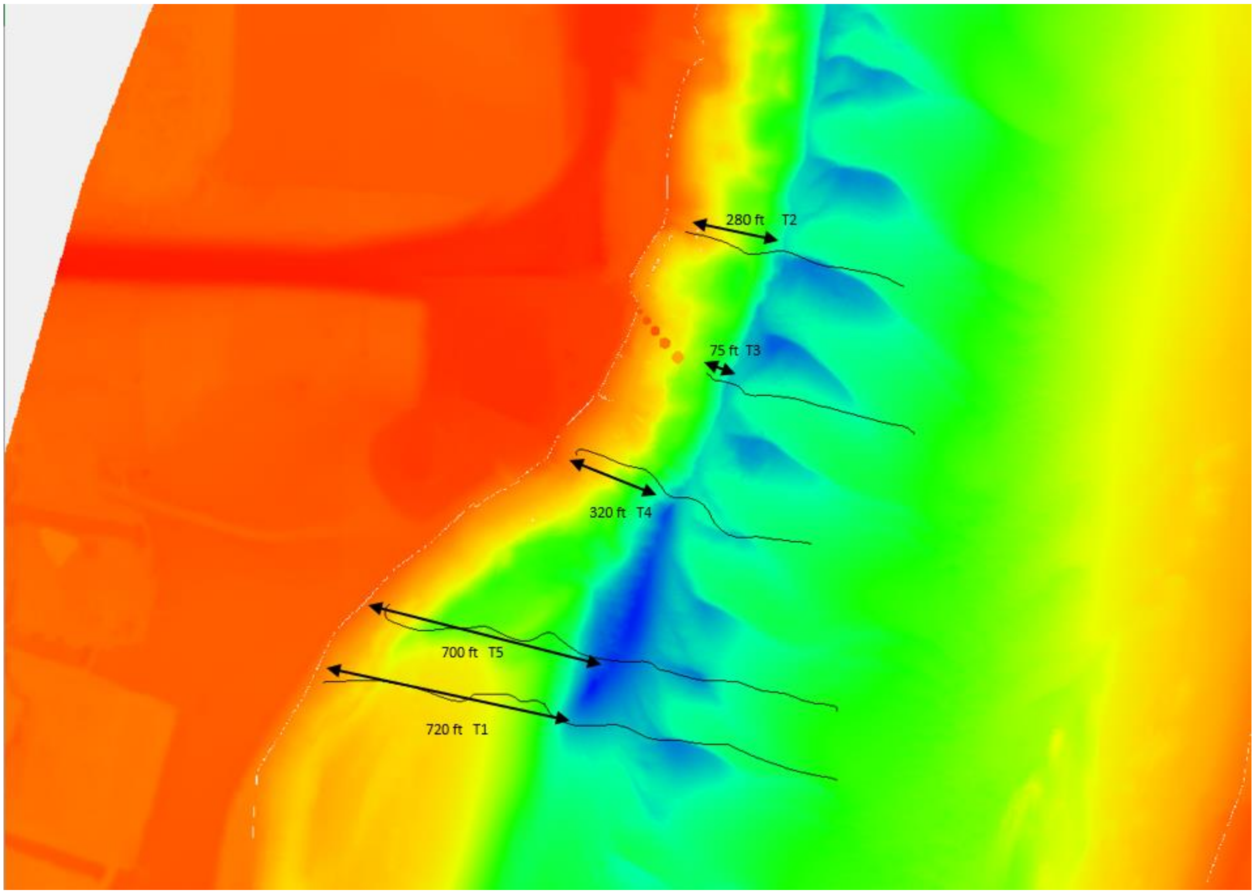

Figure 4. ADCP transects (984,000 cfs)

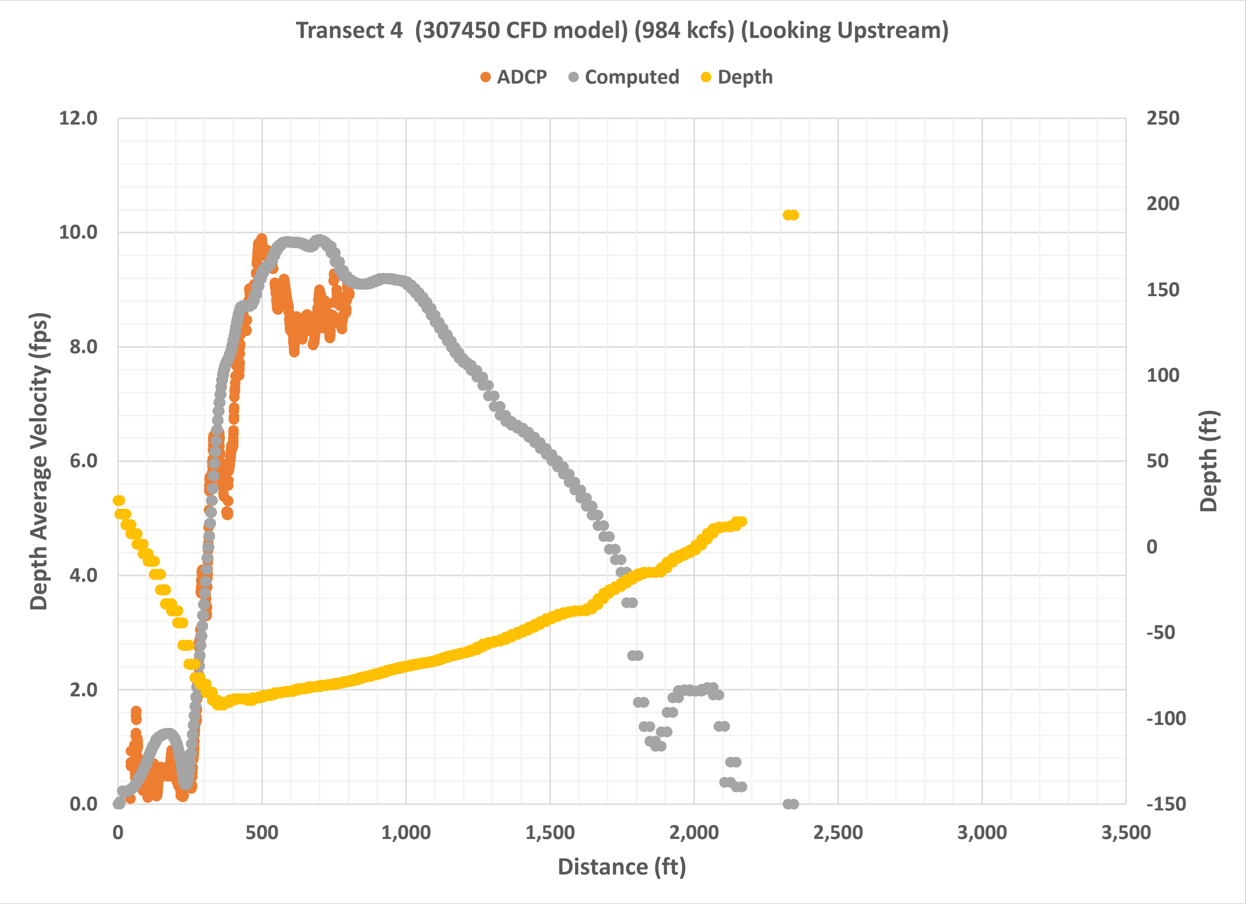

Figure 5. Transect 4 – ADCP compared to computed CFD results (984,000 cfs)
(the free surface is at 204.35 ft, and negative depth is the distance below the surface)

Modeling to Screen Alternatives

An initial group of more than 12 hydraulic alternatives, ranging from spur-dikes to protective dolphin removal was narrowed down through HEC-RAS 2D modeling and a smaller group of the most effective solutions were modeled using the CFD model.

Given what was understood about the hydrodynamic conditions at the dock, and what is feasible within the dock's ownership constraints, barge operations, and USACE's mission, it was determined to evaluate two design alternatives hydraulically further with the CFD model:

- Alternative A – Remove the protective dolphins and add a floating debris deflector (Figure 4-4 and Figure 4-5).
- Alternative D – Remove the protective dolphins and add a floating debris deflector. This alternative's floating debris deflector is longer and more parallel with the shoreline (Figure 4-6 and Figure 4-7).

For both alternatives, the thought was that removing the protective dolphins would reduce flow unsteadiness in the dock's vicinity and by removing the protective dolphins the blockage upstream of the dock would be removed allowing more upstream flow through the dock area, suppressing the strength of the lateral velocities that docking barges encounter. Figure 6 shows the velocity magnitude for existing conditions.

The hydraulic evaluation focused on time series of lateral velocities at points in the area of interest over nearly 30 minutes at 984,000 cfs. The location of each monitoring point is shown in Figure 7. Figure 8 and Figure 9 show the configuration for alternative A and D, respectively. Figure 4-10 shows an individual time series, and Table 1 shows the averaged over the time series at each point.

Specifically, points 14 and 25 are located at the area where it has been reported that strong lateral velocities can create dangerous conditions (above 1,000,000 cfs) for docking barges. As shown in Figure 8 and in Table 1, both alternatives A and D reduce the lateral velocities in this area. The lateral velocities are also reduced in the docking area, as is evident by inspecting points 12 and 13. Alternative D is also slightly better than Alternative A in this respect.

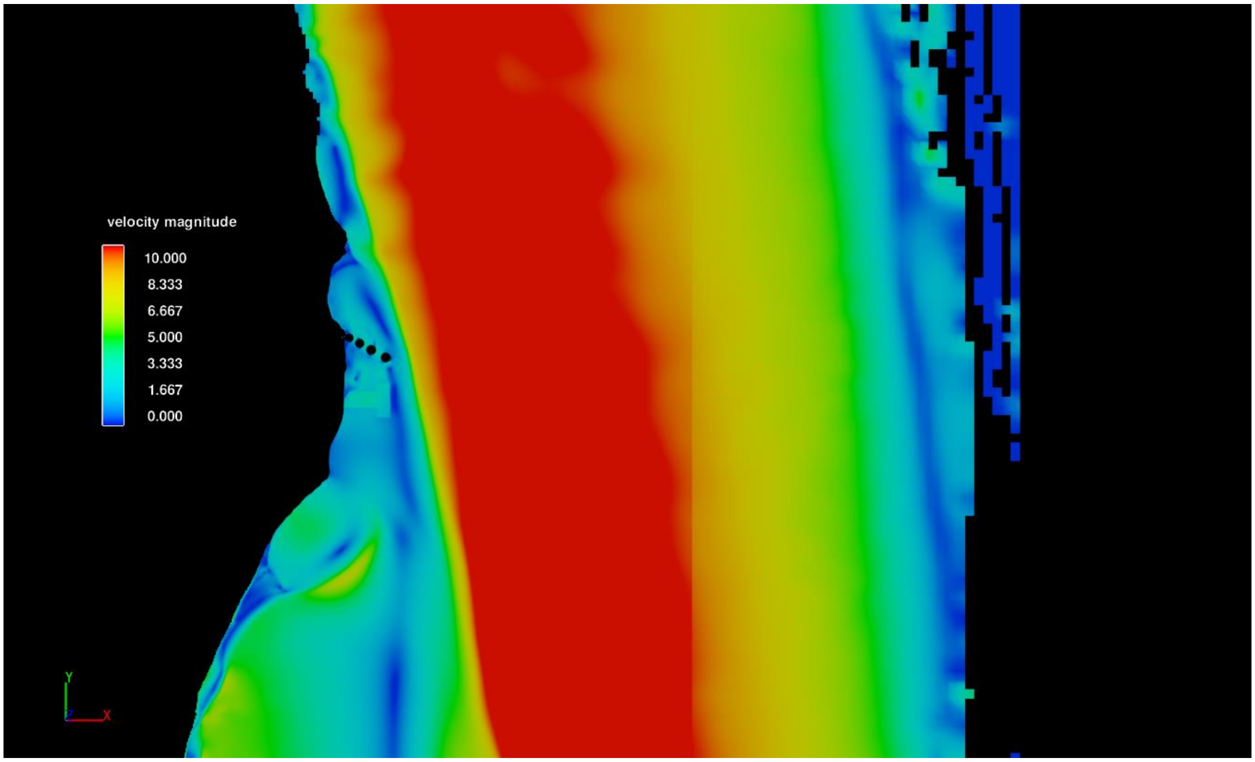

Figure 6. Velocity magnitudes from existing conditions model in fps (984,000 cfs)

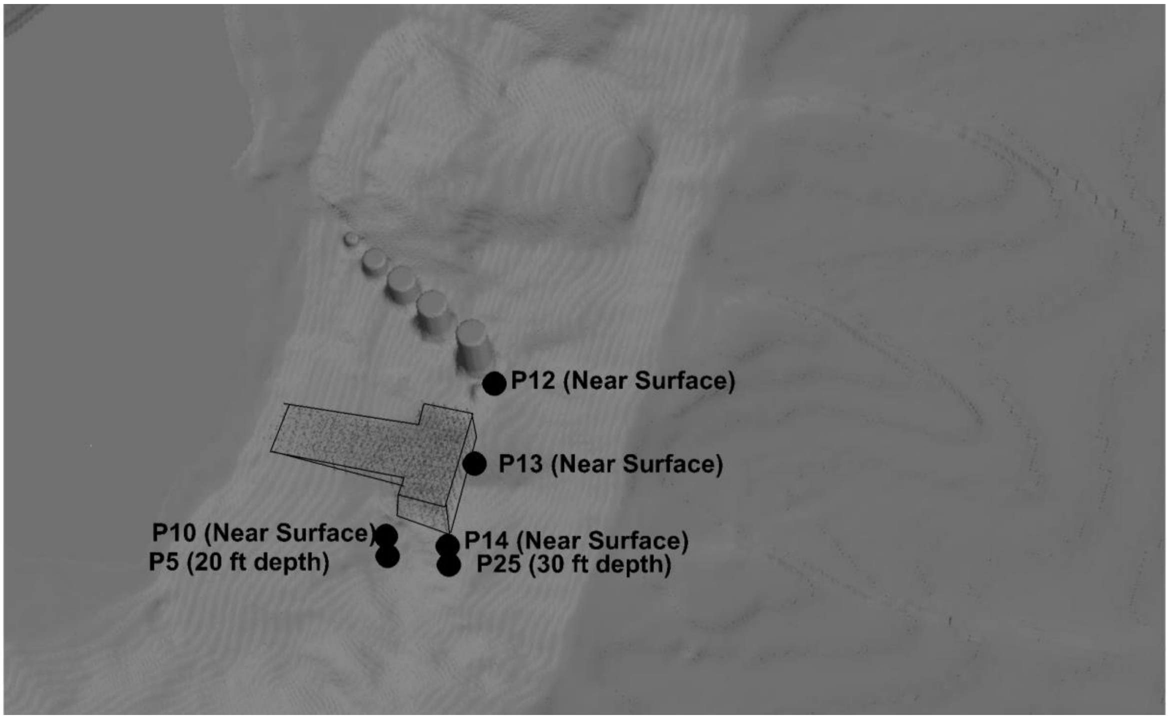

Figure 7. Area of interest (near dock), as modeled, existing conditions, and monitoring point locations.

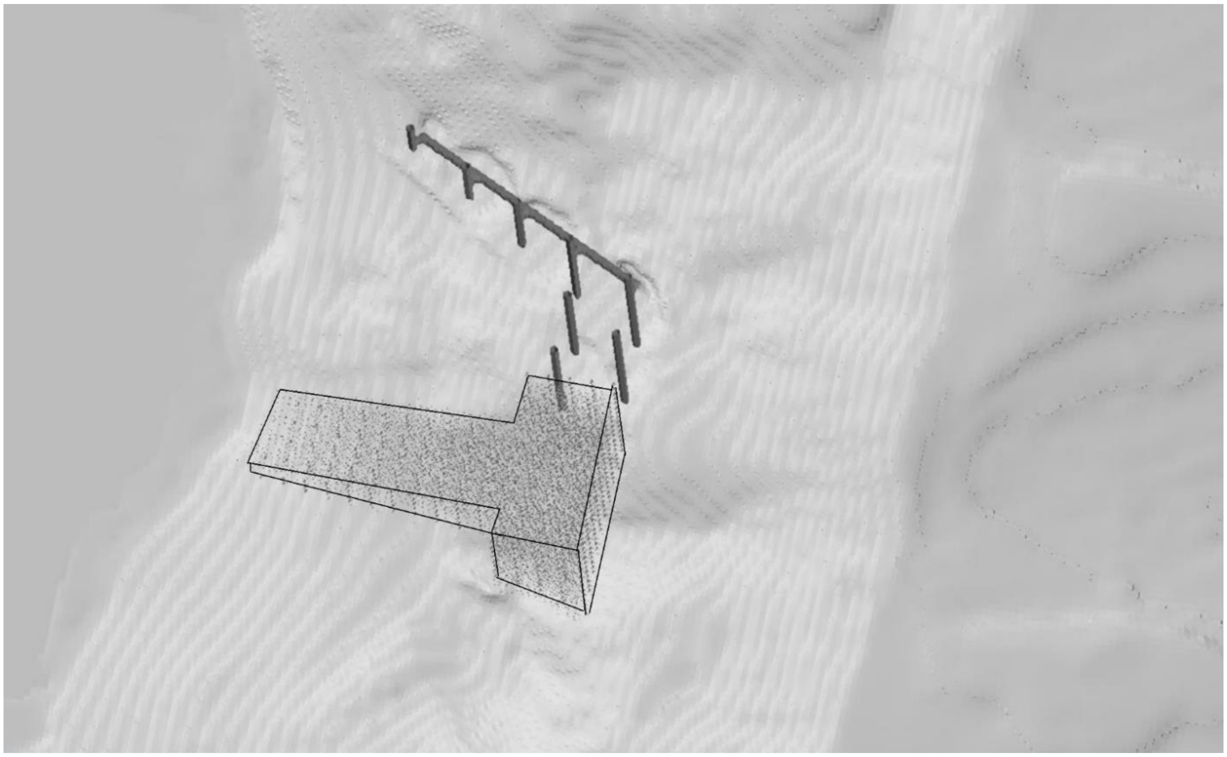

Figure 8. Alternative A: Removal of protective dolphins and floating debris boom, as modeled

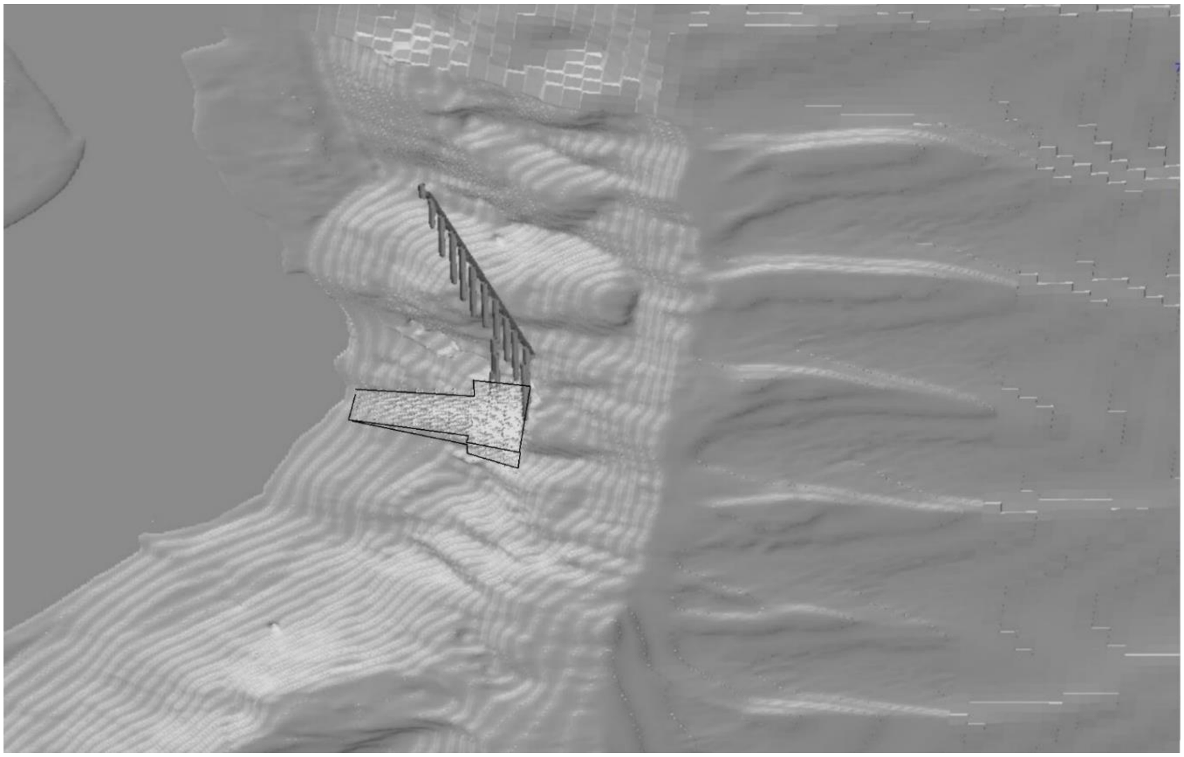

Figure 9. Alternative D: Removal of protective dolphins and floating debris boom, as modeled

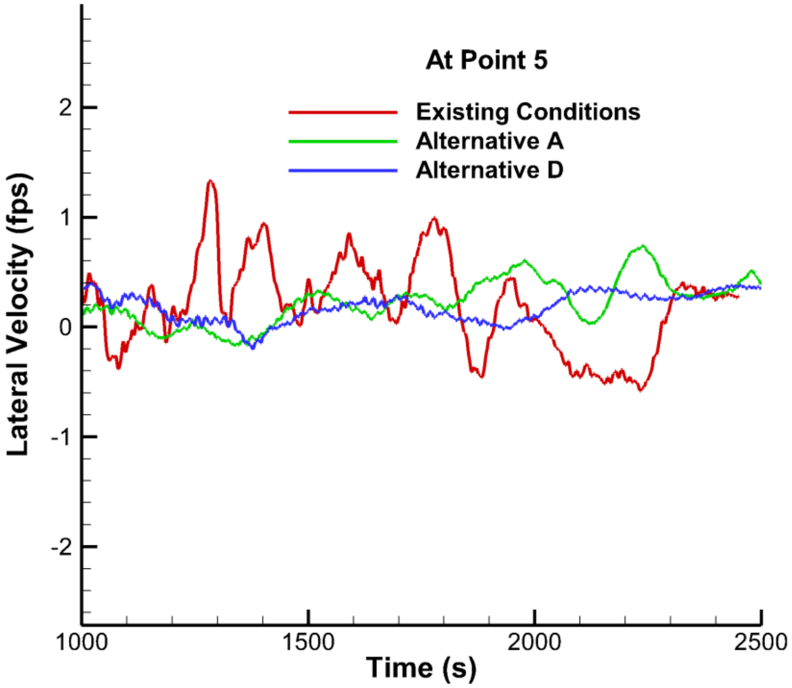

Figure 10. Lateral velocity time series at Point 5

Table 1. Average Lateral Velocities at Point Locations (fps)

Condition	Point 10	Point 5	Point 14	Point 25	Point13	Point 12
Existing	0.19	0.19	0.60	0.46	0.64	0.85
Alt A	0.19	0.17	0.33	0.28	0.34	0.71
Alt D	0.25	0.22	0.28	0.25	0.30	0.46

Summary

The objective of this analysis was to investigate the velocity patterns and flow dynamics at the port location using a numerical model. The development of a CFD model was documented and determined to reasonably replicate ADCP results in the port's vicinity. Improvements at the port that reduced lateral velocities, making the port safer for docking at high flows, were conceptualized and evaluated. Ultimately this study has leveraged the available state of the art technology to solve a practical problem in very complex conditions. The technical evaluation has showcased bathymetric data collection, sedimentation considerations, leveraging reach scale 1D models for development of boundary conditions, a HEC-RAS 2D model for concept development and evaluation, and a FLOW-3D CFD model to further investigate existing conditions and alternatives.

References

Choufu, Liang, Saeed Abbasi, Hanif Pourshahbaz, Poorya Taghvaei, and Samkele Tfwala. 2019. Investigation of flow, erosion, and sedimentation pattern around varied groynes under different hydraulic and geometric conditions: A numerical study, Water, 11.2; 235.

Daneshfaraz, Rasoul, Amir Ghaderi, Maryam Sattariyan, Babak Alinejad, Mahdi Majedi Asl, and Silvia Di Francesco. (2021). Investigation of local scouring around hydrodynamic and circular pile groups under the influence of river material harvesting pits, Water, 13.6; 2192.

Flow Science, Inc. 2021. User Manual: FLOW-3D HYDRO Documentation, Release 2022R1. Albaquerque: Flow Science, Inc.