Three-Dimensional CFD Analysis of
Construction Design Alternatives for an
ERDC Coastal and Hydraulics Laboratory
Flow Accelerator

Nuclear Science and Engineering Division
About Argonne National Laboratory
Argonne is a U.S. Department of Energy laboratory managed by UChicago Argonne, LLC under contract DE-AC02-06CH11357. The Laboratory’s main facility is outside Chicago, at 9700 South Cass Avenue, Argonne, Illinois 60439. For information about Argonne and its pioneering science and technology programs, see www.anl.gov.

DOCUMENT AVAILABILITY


Reports not in digital format may be purchased by the public from the National Technical Information Service (NTIS):
U.S. Department of Commerce
National Technical Information Service
5301 Shawnee Rd
Alexandra, VA 22312
www.ntis.gov
Phone: (800) 553-NTIS (6847) or (703) 605-6000
Fax: (703) 605-6900
Email: orders@ntis.gov

Reports not in digital format are available to DOE and DOE contractors from the Office of Scientific and Technical Information (OSTI):
U.S. Department of Energy
Office of Scientific and Technical Information
P.O. Box 62
Oak Ridge, TN 37831-0062
www.osti.gov
Phone: (865) 576-8401
Fax: (865) 576-5728
Email: reports@osti.gov

Disclaimer
This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor UChicago Argonne, LLC, nor any of their employees or officers, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of document authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof, Argonne National Laboratory, or UChicago Argonne, LLC.
Three-Dimensional CFD Analysis of Construction Design Alternatives for an ERDC Coastal and Hydraulics Laboratory Flow Accelerator

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

Nuclear Science and Engineering Division, Argonne National Laboratory

September 2020
# Table of Contents

1. Introduction and Objectives of the Study ................................................................. 1
2. Physical Testing of Honeycombs .................................................................................. 2
3. TFHRC Flume CFD Modeling Details ........................................................................... 9
   3.1 Perforated Distribution Pipe ..................................................................................... 9
   3.2 Walls of the Flow Accelerator ................................................................................ 10
   3.3 Honeycombs ........................................................................................................... 13
   3.4 Wire Screens .......................................................................................................... 14
4. Flow Quality Assessment Methods ............................................................................... 16
5. Flow Simulation in the TFHRC Flume Model ............................................................... 17
6. STAR-CCM+ and OpenFOAM Code-to-Code Comparison ............................................ 22
7. ERDC Flume Design .................................................................................................... 28
   7.1 Variations in the Design of the Distribution Pipe .................................................... 32
      7.1.1 Varying Porosity of the Original Design. Meshed-out Geometry of the Pipes .... 32
      7.1.2 Porous Baffle Modeling .................................................................................... 40
   7.2 Influence of the Transition Section Wall Geometry on the Flow in the Channel ...... 44
7.3 ERDC Flume Model with Varying Honeycombs and Screens ...................................... 48
   7.3.1 Use of One Honeycomb as in TFHRC Flume Model ........................................... 49
   7.3.2 Honeycomb Built of 5-inch-long PVC Pipes Ø 0.5 Inch ..................................... 51
   7.3.3 Honeycomb Built of 20-in-long PVC Pipes Ø 2”, ............................................... 55
7.4 Influence of the Addition of the Inlet Pipe to the Flume Model .................................. 58
7.5 ERDC Flume Model with Varying Flow Height ......................................................... 61
8. Summary and Recommendations .................................................................................. 65
9. Acknowledgements ...................................................................................................... 66
10. References .................................................................................................................. 67
11. Appendix ...................................................................................................................... 68
**List of Figures**

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Schematic drawing of the flow accelerator at TFHRC Hydraulics Laboratory</td>
</tr>
<tr>
<td>2.1</td>
<td>Experimental setup for the honeycomb flume testing</td>
</tr>
<tr>
<td>2.2</td>
<td>Water level in the flume downstream of a honeycomb in (a) low velocity flow, (b) high velocity flow</td>
</tr>
<tr>
<td>2.3</td>
<td>Water surface elevations at various measurement locations without the honeycomb and with the honeycomb</td>
</tr>
<tr>
<td>2.4</td>
<td>Head loss across the honeycomb: obtained in the experiment and calculated from Equation (1)</td>
</tr>
<tr>
<td>3.1</td>
<td>The geometry of the inlet and distribution pipes in TFHRC flow accelerator</td>
</tr>
<tr>
<td>3.2</td>
<td>The geometry of the back wall of the TFHRC flow accelerator and the standpipe</td>
</tr>
<tr>
<td>3.3</td>
<td>The geometry of the lateral converging walls of the TFHRC flow accelerator for the 3-ft and 6-ft wide test channel</td>
</tr>
<tr>
<td>3.4</td>
<td>The geometry of the vertical converging bottom wall of the TFHRC flow accelerator</td>
</tr>
<tr>
<td>3.5</td>
<td>The dimensions of the TFHRC flume honeycomb</td>
</tr>
<tr>
<td>3.6</td>
<td>Wire screens used in the TFHRC flume (a) view of the three frames and (b) a close-up view of the mesh with a honeycomb in the background</td>
</tr>
<tr>
<td>5.1</td>
<td>CFD model of the TFHRC hydraulic flume</td>
</tr>
<tr>
<td>5.2</td>
<td>Volume mesh on a cross section (a) along the flume, (b) a close-up view of the head section, and (c) across the channel</td>
</tr>
<tr>
<td>5.3</td>
<td>Water surface elevation relative to the bottom of the rectangular channel. Locations of the ‘plane section 1’ downstream of the converging section of the flow accelerator and ‘plane section 2’ in the testing zone</td>
</tr>
<tr>
<td>5.4</td>
<td>Contour plots of the $V_X$ velocity component on planes 1 and 2</td>
</tr>
<tr>
<td>5.5</td>
<td>Contour plots of the $V_Y$ velocity component on planes 1 and 2</td>
</tr>
<tr>
<td>5.6</td>
<td>Contour plots of the $V_Z$ velocity component on planes 1 and 2</td>
</tr>
<tr>
<td>5.7</td>
<td>Streamlines of water velocity colored with its magnitude in m/s. The range was clipped at 1.5 m/s for clarity of the figure. A view of (a) the full model, (b) close-up view of the head section</td>
</tr>
<tr>
<td>6.1</td>
<td>Mesh refinement in the vicinity of the (a) standpipe and (b) honeycomb in the OpenFOAM simulation</td>
</tr>
<tr>
<td>6.2</td>
<td>Details of the flow around the perforated pipe in the OpenFOAM simulation</td>
</tr>
</tbody>
</table>
Figure 6.3. Details of the flow downstream of the honeycomb in the OpenFOAM simulation. ...

Figure 6.4. Water surface elevation obtained with (a) STAR-CCM+, (b) OpenFOAM. Note that Z=0 m at the bottom of the rectangular channel. .................................................. 26

Figure 6.5. Velocity magnitude on five planes across the channel obtained with (a) STAR-CCM+, (b) OpenFOAM. The magnitude range was cut off at 2 m/s. ........................................................... 27

Figure 6.6. Streamlines of water velocity colored with its magnitude obtained with (a) STAR-CCM+, (b) OpenFOAM. The magnitude range was cut off at 2 m/s. ........................................ 27

Figure 7.1. Dimensions of the scaled up TFHRC flume model. ................................................. 29

Figure 7.2. Coarse mesh of the base ERDC flume model (a) and obtained water velocity magnitude distribution (b). .................................................................................................................. 31

Figure 7.3. Refined mesh of the base ERDC flume model (a) and obtained water velocity magnitude distribution(b). ............................................................................................................. 32

Figure 7.4. Geometry of the distribution pipe with circular perforations (a) diameter 1" (porosity 9%), and (b) diameter 2" (porosity 35%). ................................................................. 33

Figure 7.5. Example discretization of the CFD domain in the vicinity of the perforated pipe with porosity 35%. ....................................................................................................................... 34

Figure 7.6. Water surface elevation, (a) close-up view of the water entering the head section through the perforations in the standpipe with 9% porosity, (b) overall view of the flow accelerator. ................................................................................................................... 35

Figure 7.7. Volume fraction of water on a plane section along the center line of the flume accelerator for the model of a perforated pipe with porosity 9%. ........................................ 36

Figure 7.8. Contour plot of velocity for the threshold of volume fraction of water >= 0.5 on a plane section along the center line of the flume accelerator for the model of a perforated pipe with porosity 9%. .................................................................................................................. 36

Figure 7.9. Water surface elevation, (a) close-up view of the water entering the head section through the perforations in the standpipe with 35% porosity, (b) overall view of the flow accelerator. ................................................................................................................... 37

Figure 7.10. Velocity contour plot on a plane section along the center line of the flume accelerator for the model of a perforated pipe with porosity 35%. ........................................ 38

Figure 7.11. Contour plots of the velocity magnitude on three planes in the downstream section of the transition section, (a) perforated pipe with 1-inch-wide perforations, (b) perforated pipe with 2-inch-wide perforations. ........................................................................................................... 39

Figure 7.12. Water surface elevation, (a) close-up view of the water entering the head section through the perforations in the standpipe with 35% porosity modeled as a porous baffle interface, (b) overall view of the flow accelerator. ........................................................................................................... 41

Figure 7.13. Comparison of solutions obtained with the model with (a) meshed-out perforated pipe, (b) porous baffle interface model. Velocity magnitude (m/s) is plotted on a vertical plane section along the center line of the flow accelerator. Position Z (m) is plotted on the water surface. The adopted range of values is kept the same for both models. ........................................ 42
Figure 7.14: Contour plots of the velocity magnitude across the channel obtained from models with (a) meshed-out geometry of the perforated pipe, and (b) pipe modeled as a porous baffle interface. ................................................................. 43

Figure 7.15. Contour plots of the velocity in the main direction, $V_X$ and secondary flow velocity, $V_{YZ}$, across the channel obtained from models with (a) meshed-out geometry of the perforated pipe, and (b) pipe modeled as a porous baffle interface. The color bar value range is kept the same for all plots. ................................................................. 44

Figure 7.16. Geometry of the simplified flume model without the head section and with the original contraction section. ........................................................................................................ 45

Figure 7.17. Geometry of the simplified flume model with an elongated contraction section .... 45

Figure 7.18. A comparison of the straight channel lengths between the model with original and elongated contraction section ................................................................................................... 46

Figure 7.19. Contour plots of the main velocity component $V_X$ in the flume model with (a) original, and (b) 3 times longer contraction section. ................................................................. 47

Figure 7.20. Contour plots of the secondary velocity $V_{YZ}$ in the flume model with (a) original, and (b) 3 times longer contraction section ...................................................................................... 47

Figure 7.21. Geometry of the accelerator with the flow straighteners: perforated pipe, honeycomb, and wire screen. ........................................................................................................... 48

Figure 7.22. Water level established in the flume model with one 3-inch-long honeycomb Ø 9.5 mm. ........................................................................................................................................ 49

Figure 7.23. Contour plots of (a) velocity magnitude, (b) velocity in the main flow direction, and (c) secondary flow velocity on plane sections located 14 m away from the center of the inlet pipe in the flume model with one 3-inch-long honeycomb Ø 0.5" ........................................................................ 51

Figure 7.24. Water level established in the flume model with one 5-in-long honeycomb Ø 0.5". ........................................................................................................................................ 52

Figure 7.25. Streamlines of velocity in the flume model with a 5-inch-long honeycomb Ø 0.5". 52

Figure 7.26. Contour plots of velocity on plane sections in the flow accelerator in the flume model with a 5-inch-long honeycomb Ø 0.5", (a) velocity magnitude, (b) main flow direction velocity $V_X$, (c) secondary velocity $V_{YZ}$ .................................................................................. 53

Figure 7.27. Contour plots of velocity on plane sections in the test zone in the flume model with a 5-inch-long honeycomb Ø 0.5", (a) velocity magnitude, (b) main flow direction velocity $V_X$, (c) secondary velocity $V_{YZ}$ ........................................................................ 54

Figure 7.28: Water level established in the flume model with one 20 in long honeycomb Ø 2”. 54

Figure 7.29. Streamlines of velocity in the flume model with a 20-inch-long honeycomb Ø 2”.. 55

Figure 7.30: Contour plots of velocity on plane sections in the flow accelerator in the flume model with a 20-inch-long honeycomb Ø 2”, (a) velocity magnitude, (b) main flow direction velocity $V_X$, (c) secondary velocity $V_{YZ}$ .................................................................................. 56
List of Tables

Table 2-1: The measurements of water depth upstream and downstream of a honeycomb for a range of flow velocities........................................................................................................................................... 6

Table 5.1. Measures of the flow uniformity in the TFHRC flume model. ........................................21

Table 6.1. Boundary conditions used in the OpenFOAM model....................................................23

Table 7.1. Case set matrix .............................................................................................................28

Table 7-2: Uniformity for models of perforated pipes on a YZ plane at X=14 m. .........................40

Table 7-3: Uniformity measures for the simplified flume model..................................................48

Table 7-4. Typical dimensions of industrial PVC pipes – schedule 40 [10]..................................49

Table 7-5. Uniformity measures for various honeycomb types. ....................................................58

Table 7-6: Uniformity measures for models with or without an inlet pipe.................................61

Table 7-7: Uniformity measures for varying water height. ............................................................64
1 Introduction and Objectives of the Study

The U.S. Army Engineer Research and Development Center, ERDC, Coastal and Hydraulics Laboratory in Vicksburg Mississippi is planning to build a large flume with a 10-foot-wide flow channel. Three-dimensional Computational Fluid Dynamics (CFD) analysis was used in the design of the flume entry and flow accelerator sections that feed into the 10-foot-wide channel. The design is based on the plans of the existing fiberglass flume inlet at the Turner-Fairbank Highway Research Center (TFHRC) J. Sterling Jones Hydraulics Research Laboratory in McLean Virginia. The TFHRC flume inlet feeds water to a 6-foot-wide flume channel, and therefore a scale up factor of 10/6 in the width of the inlet section of the TFHRC flume would meet the width requirement of the ERDC flume channel. The CFD analysis was performed using STAR-CCM+ commercial CFD software. Additional analysis was done with the open-source CFD software, OpenFOAM, to cross-check the analysis of one of the cases with a code-to-code comparison.

A schematic drawing of the flume inlet and flow accelerator sections of the flume at TFHRC is shown in Figure 1.1. The flow enters the structure through a PVC pipe with a 24-inch inner diameter (“water inlet” marked with an arrow) into a perforated distribution pipe in the reservoir. The distribution pipe is a SDR26 PVC Waste Pipe (outer diameter 28 inches, inner diameter 26 inches) that spans the inlet region from the bottom to the top of the accelerator and is capped on the top to eliminate spillage of water. The perforations are 1 inch in diameter and there are 25 holes around the perimeter and 18 holes from top to bottom (overall 450 perforations). This way of injecting water into the flume inlet reservoir turns the flow from the inlet pipe 90 degrees into the horizontal direction, injects it around the circumference of the pipe, reduces the size of turbulent eddies, and produces a more uniform flow at the entry of the flume channel. The shape of the back wall of the accelerator is defined by two symmetric spiral functions. The wall meets the inlet cylindrical perforated pipe at the centerline of the flume. This shape makes it possible for the water (a) to split symmetrically into two streams to feed the left and right sides of the flume, and (b) to flow through the perforations on the entire circumference of the pipe nearly uniformly. Therefore, the flow momentum is distributed over an entire flume flow cross section, and a significantly higher velocity flow in the middle of the channel, which would form otherwise, is avoided. The spiral shape of the walls is followed by a straight section, which ends with three

Figure 1.1: Schematic drawing of the flow accelerator at TFHRC Hydraulics Laboratory.
frames, where honeycomb sections can be installed, each 3-inches-thick, with an 8-inch gap between them. Currently, one honeycomb is installed in the downstream frame, and it efficiently reduces the lateral turbulence in the test section. The frames also allow for installation of mesh screens whose goal is to additionally straighten the flow and reduce the axial turbulence. Three mesh screens with wire diameter of 0.25 mm and a mesh opening 0.75 mm are present on the downstream side of the frames. The honeycomb and screens filter out large eddies before the flow gets into the converging part of the accelerator. In the converging section, the walls converge laterally (on both sides) and vertically (from the bottom), with the contraction curves chosen such that the velocity streamlines of the flow follow the shape of the walls with no visible separation and flow that is adequately uniform at the entry to the flume channel. The converging part of the accelerator has an exchangeable 3-foot-wide and 6-foot-wide exit, which allows for performing physical tests in a wider or narrower test channel. The entire structure has an ability to tilt to model flow in channels with different slopes in the direction of the flow. The discharge in the TFHRC flume is usually about 30 cfs which gives water depth between 0.10 m and 0.20 m.

The discharge planned for the ERDC flume is about 60 cfs which is about twice that of the TFHRC flume. There are also plans for pumps that can deliver up to 80 cfs. To run the ERDC flume with a discharge of 60 cfs, but the same velocity (same water volume flux) as the TFHRC flume, would mean that the flow cross section area of the ERDC flume would need to be twice that of the TFHRC flume. Knowing that the ERDC flume channel width is 10/6 times the TRHRC flume channel width, meeting this condition requires the ERDC flume channel height be at least $2 \times \frac{6}{10} = \frac{6}{5}$ times the TFHRC flume channel height. Planned operating conditions for the ERDC flume led to a proposed flume channel height that exceeds this $\frac{6}{5}$ ratio, as described in the following sections of the report.

The primary objectives of this study were to use CFD to study the scale-up options for the TFHRC flume head section that would yield a good, well-functioning design for the planned ERDC flume head section. The main goal was to achieve a flow leaving the head section that is as uniform as possible over a cross section in the upstream section of the flume. In this process, alternatives for the inlet and flow distribution pipe were investigated. The hole diameter and spacing in the flow distribution pipe were analyzed for their respective effects on the flow distribution. An alternative of using slots to distribute flow from the inlet pipe was checked to see whether that might yield a more uniform flow downstream. It did not. The converging contour and length of the converging section were modeled and checked for adequate performance in the scaled-up head section. Variations in the placement, size and position of the honeycomb flow straighteners and screens were modeled and optimized. The effect of the length and diameter of tubes in the honeycomb on flow straightening and water surface height in the flume were investigated. The results of these investigations were used to obtain a scaled-up head section design for the ERDC flume.

## 2 Physical Testing of Honeycombs

Honeycombs are an essential part of the flume structure due to their role in straightening the flow by reducing the lateral components of the fluctuating velocity. Because the flow entering the honeycomb is forced to follow the path through the channels, the angle of the primary flow velocity vector leaving the honeycomb is smaller than $\arctan \frac{d}{L}$, where $d$ is the channel diameter and $L$ the channel length. It is recommended to use tubes of length equal to or more than ten diameters [1], so that the flow can become fully turbulent inside the tubes and the honeycomb exit velocity vector angle with respect to the axis is limited to less than about 6 degrees. It is also recommended that the last honeycomb ends 30-40 channel diameters before the contraction [2], which is a distance long enough for the small-scale turbulence generated in the honeycomb channels to dissipate.

Three-Dimensional CFD Analysis of Construction Design Alternatives for an ERDC Coastal and Hydraulics Laboratory Flow Accelerator
The honeycombs can be included in a CFD model with a fully meshed-out geometry of all channels, which tends to be very time and resource-consuming, or alternatively as a numerically much more efficient porous volume with the corresponding hydraulic resistance of the honeycomb. The porous resistance parameters of a honeycomb can be established from simple relations for pipe flow resistance because the honeycomb is a collection of pipes conveying the water in parallel. The head loss, \(dh\), over the length of a honeycomb is calculated from the loss for a single pipe [4]:

\[
dH = f_D \frac{L v^2}{2gD},
\]

where \(f_D = f_D(Re)\) is the Darcy coefficient as a function of Reynolds number, \(L\) is the length of the honeycomb, \(v\) is the pipe velocity, \(g\) is the gravitational acceleration, and \(D\) is the inner diameter of a pipe.

The Darcy coefficient can be obtained for example from the Moody diagram or calculated from the Colebrook-White equation [4] which is as follows:

\[
\frac{1}{\sqrt{f_D}} = -2 \log_{10} \left( \frac{k_s}{3.71 D_H} + \frac{2.51}{Re \sqrt{f_D}} \right),
\]

where \(k_s\) is the equivalent roughness height of the pipe surface, \(D_H\) is the hydraulic diameter \((D_H = D)\), \(Re\) is the Reynolds number, \(Re = \frac{Dv}{\nu}\), and the kinematic water viscosity is \(\nu = 10^{-6}\) m\(^2\)/s. For simplicity, it was assumed that the pipe inner surface is smooth i.e., the roughness height is zero, which gives:

\[
\frac{1}{\sqrt{f_D}} = -2 \log_{10} \left( \frac{2.51}{Re \sqrt{f_D}} \right),
\]

This equation does not account for entry/exit losses; however, these losses are generally about an order of magnitude lower than the pipe loss and can be neglected. To test the accuracy of this assumption, a polycarbonate honeycomb [3] was tested in a series of full-scale flume tests conducted in a force balance flume at the J. Sterling Jones Hydraulics Laboratory of FHWA. The experimental setup is presented in Figure 2.1. The flow direction and the position of the honeycomb in the flume are marked. Initially, tests were performed without the honeycomb in the flume. For each target velocity, the actual velocity was measured with UDV (Ultrasonic Doppler Velocimetry), and water depth measurements were taken.
Figure 2.1 Experimental setup for the honeycomb flume testing.
Figure 2.2. Water level in the flume downstream of a honeycomb in (a) low velocity flow, (b) high velocity flow.
The dimensions of the honeycomb under consideration were: 380 mm (14.96 in) in length, 390 mm (15.35 in) in width, with a thickness of 38.1 mm (1.5 in). The honeycomb cell size is 6.7 mm (17/64 in), with a wall thickness of 1.9 mm (0.075 in). The tested conditions covered a range of flow velocities, from 0.2 m/s to 1 m/s. Figure 2.2 shows a close-up view of the downstream side of the honeycomb during testing in (a) low velocity flow, and in (b) high velocity flow. At a low approach velocity, the water surface is smooth downstream of the honeycomb, however for the higher velocity flow, a sinusoidal wave forms with clearly visible local minima and maxima. During each of the tests, the downstream and upstream water depth was recorded in multiple positions, averaged, and the difference in the mean water levels was calculated.

Table 2-1 combines the water level measurements and the differences between the downstream and upstream locations obtained as an average from the experimental measurements and calculated from Equation (1). To show the differences in the measured elevation, several plots were prepared.

Figure 2.3 shows the measured water surface elevation in the physical tests, without and with the honeycomb as a function of pipe flow velocity. If there is no obstruction in the flow, the water elevation does not change significantly up to equivalent pipe velocity in the honeycomb tubes of 2.16 m/s and increases for higher velocities. When the honeycomb is present in the flume, the water level starts increasing in value at much lower velocity, 1.065 m/s. The water level upstream of the honeycomb is very similar for the three considered locations, independent of the flow velocity, whereas the downstream the water surface is almost level for flow velocity up to 1.065 m/s and for higher values the amplitudes in water surface elevation increase. Interestingly, the maximum water level downstream of the honeycomb is almost the same as the water level upstream.

<table>
<thead>
<tr>
<th>Test number</th>
<th>Target velocity (mm/s)</th>
<th>Velocity UDV (mm/s)</th>
<th>Flow rate (l/s)</th>
<th>Without honeycomb</th>
<th>With honeycomb</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Water depth (mm)</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Upstream Position 1</td>
<td>Position 2</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Position 3</td>
<td>Position 4 - minimum</td>
</tr>
<tr>
<td>1</td>
<td>200</td>
<td>193.6</td>
<td>14.7</td>
<td>149.0</td>
<td>149.3</td>
</tr>
<tr>
<td>2</td>
<td>300</td>
<td>312.7</td>
<td>19.2</td>
<td>149.6</td>
<td>152.3</td>
</tr>
<tr>
<td>3</td>
<td>400</td>
<td>394.7</td>
<td>26.0</td>
<td>150.8</td>
<td>154.2</td>
</tr>
<tr>
<td>4</td>
<td>500</td>
<td>497.4</td>
<td>32.8</td>
<td>150.4</td>
<td>157.6</td>
</tr>
<tr>
<td>5</td>
<td>600</td>
<td>614.1</td>
<td>35.9</td>
<td>150.8</td>
<td>162.3</td>
</tr>
<tr>
<td>6</td>
<td>700</td>
<td>701.8</td>
<td>39.7</td>
<td>149.9</td>
<td>166.7</td>
</tr>
<tr>
<td>7</td>
<td>800</td>
<td>801.1</td>
<td>44.4</td>
<td>150.3</td>
<td>170.7</td>
</tr>
<tr>
<td>8</td>
<td>900</td>
<td>895.8</td>
<td>50.3</td>
<td>155.1</td>
<td>181.0</td>
</tr>
<tr>
<td>9</td>
<td>1000</td>
<td>990.5</td>
<td>58.0</td>
<td>171.2</td>
<td>198.5</td>
</tr>
</tbody>
</table>
Table 2-1 cont’d

<table>
<thead>
<tr>
<th>Test #</th>
<th>Average Pos. 1&amp;2&amp;3 (I)</th>
<th>Average Pos. 4&amp;5 (II)</th>
<th>dH [mm] (I-II)</th>
<th>dH [mm] Eq. (1)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>149.3</td>
<td>148.48</td>
<td>0.9</td>
<td>1.7</td>
</tr>
<tr>
<td>2</td>
<td>152.4</td>
<td>150.2</td>
<td>2.2</td>
<td>4.1</td>
</tr>
<tr>
<td>3</td>
<td>154.2</td>
<td>148.65</td>
<td>5.5</td>
<td>6.3</td>
</tr>
<tr>
<td>4</td>
<td>157.6</td>
<td>148.91</td>
<td>8.7</td>
<td>9.6</td>
</tr>
<tr>
<td>5</td>
<td>162.8</td>
<td>150.93</td>
<td>11.8</td>
<td>14.1</td>
</tr>
<tr>
<td>6</td>
<td>166.7</td>
<td>152</td>
<td>14.7</td>
<td>17.7</td>
</tr>
<tr>
<td>7</td>
<td>170.0</td>
<td>153.1</td>
<td>16.9</td>
<td>22.1</td>
</tr>
<tr>
<td>8</td>
<td>180.1</td>
<td>153.705</td>
<td>26.4</td>
<td>26.5</td>
</tr>
<tr>
<td>9</td>
<td>198.2</td>
<td>169.81</td>
<td>28.4</td>
<td>30.9</td>
</tr>
</tbody>
</table>

Figure 2.3. Water surface elevations at various measurement locations without the honeycomb and with the honeycomb.
Figure 2.4 shows a plot of the head loss vs. pipe flow velocity in the honeycomb tubes. The $dH$ was calculated as difference between the mean upstream elevation and the minimum, mean and maximum downstream water elevation. The three calculations give very similar values for lower flow velocities, but the difference between them increases significantly with increasing velocity and reaches up to 200%. Nevertheless, the mean measured loss is very close to that given by Equation (1), and therefore it can be concluded that the theoretical Equation (1) can give a good approximation of the experimental measurements, and it can be used to set the axial loss parameter in a porous media model of this size honeycomb in a CFD model.

![Figure 2.4. Head loss across the honeycomb: obtained in the experiment and calculated from Equation (1).](image-url)
3 TFHRC Flume CFD Modeling Details

A CFD model of the TFHRC Flume was built to create a reference case for the ERDC flume design. The TFHRC flume has been in operation for a few years now and it has been proven that the design is working as expected. The flow simulation using the TFHRC CFD flume model is described in Section 5. Once built and tested, the TFHRC flume model was scaled up to the ERDC design size and various design options were investigated using the scaled-up model. Details of the options that were investigated and the tests that were performed can be found in Section 7.

In this section, all parts of the TFHRC flow accelerator model that have an influence on the flow conditions are described, along with their shape, dimensions, and other relevant characteristics. The CFD models of those components used in the study are also presented.

3.1 Perforated Distribution Pipe

The flow enters the flow accelerator through the holes in the perforated PVC pipe with a 26-inch inner diameter and 1-inch wall thickness. The circular perforations are 1 inch in diameter, and they are distributed uniformly in vertical (18 holes) and circumferential (25 holes) directions.

A fully meshed out model of the distribution pipe with holes was developed and tested as presented in Sections 6 and 7.1. To save computational time and resources, the distribution pipe was modeled in most of the CFD simulations as a porous baffle. The porosity of the pipe, meaning the ratio of void to total area of the cylinder is equal to $\chi = \frac{A_h}{A_c} = 0.09$. In STAR-CCM+ the pressure drop across a porous baffle is characterized with the following relation [7]:

$$\Delta p = -\rho(\alpha v_n + \beta)v_n,$$  \hspace{1cm} (4)

where $\rho$ is the density of water, $\alpha$ is the porous inertial resistance, $\beta$ is porous viscous resistance, and $v_n$ is velocity normal to the interface.

Based on [1] and [6] the porous resistance is calculated from Darcy-Weisbach equation:

$$\Delta p = 0.5k\rho v^2,$$  \hspace{1cm} (5)

where it was assumed that the porous viscous resistance is negligible, and therefore $\beta = 0$. From a comparison of equations (1) and (2), the porous inertial resistance is:

$$\alpha = 0.5k,$$  \hspace{1cm} (6)

and coefficient $k$ is taken from the Handbook of Hydraulic Resistance [6] for a perforated plate:

$$k = \frac{1}{\chi^2}[0.707(1-\chi)^{0.375} + 1 - \chi]^2,$$  \hspace{1cm} (7)

Substituting void fraction $\chi = 0.09$ yields:

$$\alpha = 163.$$  \hspace{1cm} (8)
3.2 Walls of the Flow Accelerator

The geometry of the walls of the flow accelerator play an important role in the distribution of the flow in the test zone of the flume. Their shape is designed to help to distribute the flow coming from the inlet pipe more uniformly within the accelerator, and the test channel.

The shape of a half of the symmetric spiral back walls is presented in Figure 3.2 (blue line). They are defined with the following parametric functions $X(t)$, $Y(t)$ and have units of meters:

\[
X(t) = -(0.457 + 0.733t) \cos t, \\
Y(t) = \pm (0.457 + 0.733t) \sin t, \\
\]

(9)

under the assumption that the point $(X,Y) = (0,0)$ is in the center of the standpipe (orange line), and $t \in [0, 1.94]$ rad.
Figure 3.2. The geometry of the back wall of the TFHRC flow accelerator and the standpipe.

The section of the accelerator downstream of the honeycomb is built as converging in the vertical and in-plane directions. Because the accelerator is meant to transition to two different widths of the straight channel, ~0.9 m (3 feet) and ~1.8 m (6 feet), two shape functions were designed for the in-plane contraction, $f_a$ and $f_b$:

$$f_{a,b} = \begin{cases} 
-ax^3 + 0.5W_1, & x \in [0, X_h] \\
(b(L-x))^3 - Y_h + 0.5W_1, & x \in [X_h, L] 
\end{cases} \quad (10)$$

where the length of the transition is $L = 4.023$ m, the location of the inflection point is $(X_h, Y_h) = (1.308$ m, $0.5(W_1 - W_2))$, the width of the transition at $x = 0$ is $W_1 = 3.505$ m. Also, for the 3-foot-wide channel:

$$W_2 = W_{a2} = 0.914 \text{ m},$$
$$a = 1.88 \times 10^{-10}, \quad b = 4.37 \times 10^{-11}, \quad (11)$$

and for the 6-foot-wide channel:

$$W_2 = W_{b2} = 1.829 \text{ m},$$
$$a = 2.88 \times 10^{-10}, \quad b = 1.55 \times 10^{-10}. \quad (12)$$
\[ a = 1.22 \times 10^{-10}, \quad b = 2.82 \times 10^{-11}. \]

The function representing the shape in the vertical direction is the same for both lateral contraction curves:

\[
f_{vc} = \begin{cases} 
    cx^3, & x \in [0, X_v] \\
    -c(L_2 - x)^3 + (H_1 - H_2), & x \in [X_v, L_2] \\
    H_1 - H_2, & x \in [L_2, L] 
\end{cases}
\]  

(13)

where the coefficient is \( c = 1.15 \times 10^{-10} \), the height of the transition at \( x = 0 \) is \( H_1 = 1.241 \) m, and at \( x = L_2 = 2.8 \) m is \( H_2 = 0.610 \) m, and the coordinates of the inflection point are \( (X_v, Y_v) = (1.4 \) m, 0.316 m).

The geometry of the lateral walls is presented in Figure 3.3 and the geometry of the vertical walls in Figure 3.4.

**Figure 3.3.** The geometry of the lateral converging walls of the TFHRC flow accelerator for the 3-ft and 6-ft wide test channel.
Figure 3.4. The geometry of the vertical converging bottom wall of the TFHRC flow accelerator.

3.3 Honeycombs

A manufactured PC2 polycarbonate honeycomb [3] was used in the TFHRC flow accelerator. The honeycomb is built from 9.5 mm (0.375 in) inner diameter tubes, with wall thickness 0.3 mm (0.012 in), which gives the porosity equal to $\chi = 0.8$. Its extent in flow direction is 95 mm (3.7 in).

The honeycomb was represented in the CFD model with a porous volume with a specified porosity and inertial and viscous porous resistance tensors. The diagonal components of the inertial and viscous resistance in the directions perpendicular to the main flow direction, i.e., across the channel and vertical, are in fact infinitely large because there is no flow in these two directions across the honeycomb. In the CFD model, these two components should be set to be several orders of magnitude larger than the component in the main flow direction. The off-diagonal components of the tensors are equal zero.

In STAR-CCM+ software, the porous body force in a porous region is defined as:

$$ f_p = -(P_i|v| + P_v)v. $$

(14)

The user needs to specify the components of two tensors: porous viscous resistance $P_v$, and porous inertial resistance $P_i$. To establish their values, the porous resistance of the honeycomb in the primary flow (axial) direction is calculated from the Darcy-White equation for one pipe as it was presented in equation (1). The pressure loss $\Delta p$ along the length of a tube is:

$$ \frac{\Delta p}{L} = \frac{f_d}{2D} \rho v^2, $$

(15)

where $L$ is the length of a tube, $D$ is the inner diameter of a tube, $v$ is the pipe velocity, $v = \frac{v_{appr}}{\chi}$, $v_{appr}$ is the approach velocity, $f_d$ is the friction coefficient calculated from the Colebrook-White formula [3]. The resistance for the off-axis directions is set to a large value, five orders of magnitude larger than in the flow direction [7], to force the flow to conform to the axial direction of the honeycomb tubes, but at the same time to avoid the numerical instability that may occur if a larger value is used.
Three-dimensional CFD Analysis of Construction Design Alternatives for an ERDC Coastal and Hydraulics Laboratory Flow Accelerator

3.4 Wire Screens

Three wire screens can be installed in metal frames in the TFHRC flume to additionally straighten the flow. They are made of 0.25 mm diameter cylindrical wires, with 0.75 mm wire spacing. The porosity of this screen is equal $\chi = \frac{A_h}{A_c} = 0.56$.

In the CFD model the screens were modeled as porous baffle interfaces [7] according to equations (4) and (5) presented in Chapter 3.1., with the difference that the porous inertial resistance was calculated as for a screen [6]:

$$\alpha = 0.5k, \quad k = 1.3(1 - \chi) + \left(\frac{1}{\chi} - 1\right)^2 = 1.2, \quad \alpha = 0.6.$$  \hspace{1cm} (16)

Figure 3.6 shows the wire screens used in the TFHRC flume where (a) is a view of the three frames and (b) is a close-up view of the mesh with a honeycomb in the background and two rulers for reference.
Figure 3.6. Wire screens used in the TFHRC flume (a) view of the three frames and (b) a close-up view of the mesh with a honeycomb in the background.
4 Flow Quality Assessment Methods

The primary goal of the ERDC head section design is that the flow approaching the test section of the flume channel is adequately uniform, the secondary flow is minimized, and the velocity profile is close to the fully developed distribution.

The uniformity of the flow is assessed in two ways at several positions along the channel:

1. by computing a normalized area weighted velocity deviation from the mean velocity over a cross section normal to the primary flow direction, and
2. by computing a ratio of the absolute value of velocity components perpendicular to the main flow direction to velocity magnitude.

The deviation of the main flow direction velocity component is defined as:

$$\varphi_x = \frac{1}{VA} \int |V_x - \bar{V}|dA$$  \hfill (17)

where $A$ is the area of the cross section, $V_x$ is the component of velocity in the main flow direction, and $\bar{V}$ is the area weighted mean velocity normal to the cross section given by:

$$\bar{V} = \frac{1}{A} \int V \cdot n \, dA$$  \hfill (18)

where $V$ is the velocity vector of the flow at a point on the cross section and $n$ is a unit normal vector at a point on the cross section.

The ratios of velocity components are as follows:

$$r_y = \frac{|V_y|}{V_x}, \quad r_z = \frac{|V_z|}{V_x}.$$  \hfill (19)

The normalized velocity deviation times 100% gives the average percent deviation of the velocity from the mean over the cross section and approaches zero when the velocity $V_x$ is uniform over a flume cross section. On the other hand, the deviation will increase when the flow develops in the channel.

The analysis of the ratio of the absolute value of velocity components perpendicular to the main flow direction and $V_x$ will show if there is any significant secondary flow present in the cross-sections through a test zone or zone of interest. The primary design goal of the study is to minimize the velocity deviation as well as the ratios at the outlet of the head section flow accelerator and within the flume channel farther downstream.
5 Flow Simulation in the TFHRC Flume Model

A model of the TFHRC flume, provided by TFHRC researchers, was set up and a simulation of the flow was performed in the STAR-CCM+ CFD/CSM software. The geometry of the model with all its parts, the boundary conditions used, and an example water surface are shown in Figure 5.1.

The geometry of the computational domain covers the volume of a section of the standpipe, the flow accelerator with a straight channel and the following parts: the perforated pipe, honeycomb and three wire screens.

The Eulerian two-phase solver was selected with Volume of Fluid (VOF) model to solve for the water – air interface. It is assumed that the water surface is defined at a volume fraction of water equal to 0.5.

The initial conditions for the simulation specify zero water content; the domain is filled only with air. At the start of the simulation, water enters the domain through a circular surface (a horizontal cross-section through the standpipe) to which a velocity inlet condition is assigned. Water flows through the perforated pipe (modeled as a porous baffle interface, mesh screens (porous baffle interfaces), a honeycomb (porous region) and a section of the straight channel of the flume. The flow leaves the model through a vertical surface at the end of the channel, where a pressure outlet boundary condition is specified. The water surface elevation is controlled at the pressure outlet boundary with the use of a triangular static pressure distribution on this surface with a zero-pressure value assigned at the vertical coordinate of the expected water surface elevation. The top surface of the domain is divided into two sections. The section above the inlet, up to the downstream edge of the honeycomb is modeled as a no-slip wall boundary condition because in the actual flume, this part of the accelerator is covered with a lid to avoid spillage of water during operation. The remaining part is open, and so it was modeled as a pressure boundary condition with pressure equal to the atmospheric pressure. The simulation is run until a steady state is achieved.

![Figure 5.1. CFD model of the TFHRC hydraulic flume.](image)

During physical testing, models are positioned in the downstream section of the straight channel. It is important to know what the computed flow conditions in the representative location of the model are, therefore, a so-called Derived part, a vertical plane section was defined there. This test section, ‘plane section-2’, was highlighted in Figure 5.1 in pink. It was used in the post-processing to analyze the distribution of the velocity components according to the formulas presented in...
Section 4. Additionally, a second plane was defined at the beginning of the straight section, with the same goal (see Figure 5.3).

A polyhedral volume cell mesher was selected to discretize the CFD domain. A uniform cell target size of 0.03 m was used, which results in the total number of cells of about 1.2 million. Figure 5.2 shows the mesh on planes along and across the flume. The mesh is relatively coarse, without any refinement in the vicinity of the expected water surface. This results in a rather “choppy” representation of the water surface but allows for a good approximation of the flow conditions in the body of water. The simulation runs fast and does not require significant computational resources.

Figure 5.2. Volume mesh on a cross section (a) along the flume, (b) a close-up view of the head section, and (c) across the channel.

Figure 5.3 shows the computed water surface (VOF of water = 0.5) with a contour plot of its elevation relative to the bottom of the rectangular channel. In this model, the inlet velocity is set up to 2.0 m/s, which results in a flow rate of 580 kg/s (20.5 cfs) through the inlet pipe. At the pressure outlet, hydrostatic pressure was applied with a zero-value reference pressure at 0.21 m from the bottom of the straight channel. The flow coming into the domain through the inlet pipe reaches the lid on the distribution pipe and subjects the top surface to a pressure of 5.9 kPa (0.86 psi). The level of water then drops across the thickness of the distribution pipe and then drops again due to the honeycomb resistance and losses in the contraction section.
Figure 5.3. Water surface elevation relative to the bottom of the rectangular channel. Locations of the ‘plane section 1’ downstream of the converging section of the flow accelerator and ‘plane section 2’ in the testing zone.

Figure 5.4, Figure 5.5, and Figure 5.6 show the contour plots of the water velocity components on two cross-section planes located (1) downstream of the converging section of the flow accelerator, and (2) in the test zone of the flume.

On plane section 1, at the exit from the flow accelerator, the velocity profile does not show a significant wall effect, but an almost symmetric secondary flow is present: the component in the main flow direction reaches maximum magnitude in two locations, on the right and left side of the centerline and the other two components change signs indicating a secondary flow vortex. Nevertheless, the magnitudes of the secondary velocity components are two orders of magnitude smaller than the axial component.

On plane section 2, located in the test section of the channel, the distribution of the velocity component in the main flow direction is close to symmetrical with respect to the center line of the channel and a wall effect is clearly visible. The entrance length for flow in a pipe is usually about 40 times the hydraulic diameter [8], which is 27.3 m in this case, more than the length of the entire flume (~17 m). Therefore, the velocity profile at this location falls in between a uniform value and a fully developed profile with a boundary layer forming within ~ 0.1 m from the walls. The other two components of velocity are close to zero within the water volume. Fluctuations in magnitude are present only at the water – air interface, with maximum values up to 3.6 % of the mean velocity magnitude of 1.32 m/s.
Figure 5.4. Contour plots of the $V_x$ velocity component on planes 1 and 2.

Figure 5.5. Contour plots of the $V_y$ velocity component on planes 1 and 2.

Figure 5.6. Contour plots of the $V_z$ velocity component on planes 1 and 2.

Figure 5.7 shows the streamlines of water velocity within the water mass, colored with its velocity magnitude. The flow shoots straight up through the inlet pipe and turns through the perforations of the standpipe. Once in the head section, it follows the shape of the back wall and forms two, one on each side, large vortices. The influence of the screens and honeycomb on the flow is noticeable; they straighten the spiraling flow and feed a more uniform flow to the contracting section. There is some separation of the flow downstream of the contraction, but overall, the downstream flow is almost uniform. The flow develops in the rectangular channel, slowing down near the wall and speeding up in the center of the channel.

The deviation of the velocity component $V_x$, as defined by equation (17), as well as the ratios of lateral and vertical components to $V_x$, calculated using equation (19), are collected in Table 5.1. The value of $\varphi_x$ for section 1 is equal 4.5%, and for section 2 is about 7 percent, and this variation is primarily the result of the developing boundary layers at the flume walls. For comparison, the value of the normalized deviation for a fully developed velocity profile is equal 0.12. The two
ratios, $r_y$ and $r_z$, have low values at section 2 indicating that the off-axis velocity components of less than half percent of the primary flow direction values.

(a)

![Streamlines of water velocity colored with its magnitude in m/s. The range was clipped at 1.5 m/s for clarity of the figure. A view of (a) the full model, (b) close-up view of the head section.]

(b)

Figure 5.7. Streamlines of water velocity colored with its magnitude in m/s. The range was clipped at 1.5 m/s for clarity of the figure. A view of (a) the full model, (b) close-up view of the head section.

Table 5.1. Measures of the flow uniformity in the TFHRC flume model.

<table>
<thead>
<tr>
<th>location</th>
<th>$\varphi_x$</th>
<th>$r_y$</th>
<th>$r_z$</th>
</tr>
</thead>
<tbody>
<tr>
<td>plane section 1</td>
<td>0.024</td>
<td>0.0094</td>
<td>0.0076</td>
</tr>
<tr>
<td>plane section 2</td>
<td>0.053</td>
<td>0.0042</td>
<td>0.0045</td>
</tr>
</tbody>
</table>
6 STAR-CCM+ and OpenFOAM Code-to-Code Comparison

An additional model of the TFHRC flume was developed to conduct a code-to-code comparison between STAR-CCM+ and OpenFOAM 6.0 [9] as a cross-check of the CFD models. In this case, the geometry of the perforated standpipe was meshed out. The geometry of the honeycomb was modified. The tubes forming the honeycomb used in the TFHRC flume are very small (9 mm = 0.35 inch in diameter) compared to the dimensions of the flume and representing them in the CFD model would require a huge number of additional volume cells that would result in a prohibitively large computational model. Therefore, a honeycomb with the same porosity, a larger tube diameter (2.5 cm = 1 inch) and tube length 30 centimeters, which yields the expected pressure drop as the actual honeycomb, was introduced to the model. The wire mesh was not included to simplify the analysis.

The computational domain in STAR-CCM+ was discretized with polyhedral cells using the Polyhedral Mesher with Prism Layer Mesher, and all solver settings as described previously.

In OpenFOAM, the ‘snappyHexMesh’ mesher was used, which produces primarily hexahedral volume cells. Prism layers were added to obtain a better resolution of the velocity profile close to the walls, and the mesh was refined in the sections of the flume where the air-water interface is expected. The resulting computational mesh is built of ~25 million cells. Close-up views of the mesh refinement around the standpipe and honeycomb in the OpenFOAM are shown in Figure 6.1. The unsteady RANS solver with a realizable k-epsilon turbulence model was selected along with the ‘interFoam’ solver to account for a two-phase flow, which corresponds closely with the VOF approach available in STAR-CCM+. Boundary conditions used in the OpenFOAM model are presented in Table 6.1.
Table 6.1. Boundary conditions used in the OpenFOAM model.

<table>
<thead>
<tr>
<th></th>
<th>$P_{rgh}$ (Pa)</th>
<th>$u$ (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inlet</strong></td>
<td>fixedFluxPressure, value = 0.0</td>
<td>uniformFixedValue, at 1.989 m/s*</td>
</tr>
<tr>
<td><strong>Outlet</strong></td>
<td>totalPressure, $p_0 = 0.0$**</td>
<td>pressureInletOutletVelocity, value = 0.0</td>
</tr>
<tr>
<td><strong>Top</strong></td>
<td>totalPressure, $p_0 = 0.0$**</td>
<td>pressureInletOutletVelocity, value = 0.0</td>
</tr>
<tr>
<td><strong>Tank Walls</strong></td>
<td>fixedFluxPressure, value = 0.0</td>
<td>noSlip</td>
</tr>
<tr>
<td><strong>Standpipe</strong></td>
<td>fixedFluxPressure, value = 0.0</td>
<td>noSlip</td>
</tr>
<tr>
<td><strong>Honeycomb</strong></td>
<td>fixedFluxPressure, value = 0.0</td>
<td>noSlip</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>$k$ (m$^2$/s$^2$)</th>
<th>$\epsilon$ (m$^2$/s$^3$)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inlet</strong></td>
<td>turbulentIntensityKineticEnergyInlet, intensity = 0.05</td>
<td>turbulentMixingLengthDissipationRateInlet, mixingLength = 0.25</td>
</tr>
<tr>
<td><strong>Outlet</strong></td>
<td>turbulentIntensityKineticEnergyInlet, intensity = 0.05</td>
<td>turbulentMixingLengthDissipationRateInlet, mixingLength = 0.25</td>
</tr>
<tr>
<td><strong>Top</strong></td>
<td>kqRWallFunction, value = 0.1</td>
<td>epsilonWallFunction, value = 0.1</td>
</tr>
<tr>
<td><strong>Tank Walls</strong></td>
<td>kqRWallFunction, value = 0.1</td>
<td>epsilonWallFunction, value = 0.1</td>
</tr>
<tr>
<td><strong>Standpipe</strong></td>
<td>kqRWallFunction, value = 0.1</td>
<td>epsilonWallFunction, value = 0.1</td>
</tr>
<tr>
<td><strong>Honeycomb</strong></td>
<td>kqRWallFunction, value = 0.1</td>
<td>epsilonWallFunction, value = 0.1</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>$\alpha$.water</th>
<th>$\nuTilda$ (m$^2$/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inlet</strong></td>
<td>fixedValue, value = 1.0</td>
<td>inletOutlet, inletValue = 0.0</td>
</tr>
<tr>
<td><strong>Outlet</strong></td>
<td>inletOutlet, inletValue = 0.0</td>
<td>zeroGradient</td>
</tr>
<tr>
<td><strong>Top</strong></td>
<td>inletOutlet, inletValue = 0.0</td>
<td>zeroGradient</td>
</tr>
<tr>
<td><strong>Tank Walls</strong></td>
<td>zeroGradient</td>
<td>zeroGradient</td>
</tr>
<tr>
<td><strong>Standpipe</strong></td>
<td>zeroGradient</td>
<td>zeroGradient</td>
</tr>
<tr>
<td><strong>Honeycomb</strong></td>
<td>zeroGradient</td>
<td>zeroGradient</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>$\nu$ (m$^2$/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inlet</strong></td>
<td>calculated</td>
</tr>
<tr>
<td><strong>Outlet</strong></td>
<td>calculated</td>
</tr>
<tr>
<td><strong>Top</strong></td>
<td>calculated</td>
</tr>
<tr>
<td><strong>Tank Walls</strong></td>
<td>nutkWallFunction</td>
</tr>
<tr>
<td><strong>Standpipe</strong></td>
<td>nutkWallFunction</td>
</tr>
<tr>
<td><strong>Honeycomb</strong></td>
<td>nutkWallFunction</td>
</tr>
</tbody>
</table>

* Inlet velocity is specified as a table to increase the vertical vector component from zero to target value over a 5-second time interval.
** Reference height, $h_{Ref}$, is specified as a field file in the ‘constant’ folder with a single value of 0.21 meters.
Figure 6.1. Mesh refinement in the vicinity of the (a) standpipe and (b) honeycomb in the OpenFOAM simulation.
Details of the water surface around the perforated pipe and downstream of the honeycomb obtained from the OpenFOAM simulation are presented in Figure 6.2 and Figure 6.3. The water elevation was plotted for STAR-CCM+ and OpenFOAM in Figure 6.4. Despite the differences in the type of the mesh and solver settings, the solution is very similar. Both codes estimate a higher water elevation in the head section, that drops when the flow reaches the contraction section of the flume. The elevation at the testing zone is equal to 0.21 m, as expected. The velocity magnitude on five planes across the channel obtained with (a) STAR-CCM+, (b) OpenFOAM is presented in Figure 6.5. The magnitude range was cut off at 2 m/s for clarity of the plots. The velocity magnitude profile was presented for the volume where volume fraction of water is equal to or greater than 0.5. The isosurface $\text{VOF}=0.5$ is smoother for the OpenFOAM model, due to the hexahedral mesh used; the polyhedral mesher used in STR-CCM+ results in a more nonuniform surface, with the amplitude in water elevation limited by the cell size. Nevertheless, the velocity profiles across both models are in good agreement. Figure 6.6 shows streamlines of velocity colored with its magnitude. The streamlines represent a complex spiraling flow in the head section that is straightened by the honeycomb. The contraction of the flume does not introduce significant flow separation or large-scale turbulence in the channel. This behavior of the flow is captured in both software applications.
Figure 6.3. Details of the flow downstream of the honeycomb in the OpenFOAM simulation.

(a)

(b)

Figure 6.4. Water surface elevation obtained with (a) STAR-CCM+, (b) OpenFOAM. Note that Z=0 m at the bottom of the rectangular channel.
Figure 6.5. Velocity magnitude on five planes across the channel obtained with (a) STAR-CCM+, (b) OpenFOAM. The magnitude range was cut off at 2 m/s.

Figure 6.6. Streamlines of water velocity colored with its magnitude obtained with (a) STAR-CCM+, (b) OpenFOAM. The magnitude range was cut off at 2 m/s.
7 ERDC Flume Design

The researchers at ERDC are in need of a 10-foot-wide flume channel to perform experiments in a wide variety of flow conditions with a discharge into the flume of about 1.7 m$^3$/s (60 cfs), with the possibility of an increase up to 2.26-2.4 m$^3$/s (80-85 cfs). The new design of the flume inlet needs to accommodate the higher flow rates and retain the uniformity of the flow in the channel, i.e., the velocity of the flow in the test section should have very low components in the two directions perpendicular to the main flow direction.

The available standpipe in the ERDC laboratory has the inner diameter of 0.91 m (36 in). To obtain a flow rate of 1.7 m$^3$/s (60 cfs) in the flume, the average flow velocity in the pipe should be equal 2.6 m/s (8.5 ft/s), which can be accommodated in the laboratory.

Three sets of flow conditions were identified by the ERDC researchers as representative of envisioned experimental conditions and therefore they were chosen as input values to the CFD model (see Table 7.1). They span conditions from shallow to deeper flows at different flume slopes and will allow for comprehensive analysis of the effectiveness of the proposed design.

<table>
<thead>
<tr>
<th>Case set matrix.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Case number</strong></td>
</tr>
<tr>
<td>Flow rate, Q</td>
</tr>
<tr>
<td>Water height in the channel, h</td>
</tr>
<tr>
<td>Channel width, w</td>
</tr>
<tr>
<td>Hydraulic diameter, D$_H$</td>
</tr>
<tr>
<td>Slope, S (m/m) or (ft/ft)</td>
</tr>
<tr>
<td>Manning number, n (-) (glass)</td>
</tr>
<tr>
<td>Average flow velocity in the channel, v</td>
</tr>
<tr>
<td>Froude number in the channel, Fr (-)</td>
</tr>
<tr>
<td>Reynolds number, Re (-)</td>
</tr>
<tr>
<td>Ratio of flow development length to hydraulic diameter [L/D$_H$]</td>
</tr>
<tr>
<td>Flow development length, L</td>
</tr>
</tbody>
</table>

Several design options were developed and assessed for the planned ERDC flume with the use of CFD modeling. As the first design candidate, a scaled-up TFHRC flume geometry based on the 6-foot-wide channel was assessed. The scaling factor was equal to 10/6 for all three dimensions to accommodate the requirement of a 10-foot-wide channel. Additionally, the flume height was increased to 1.2 m, to accommodate a water height of 0.84 m, required in case number 1 in Table 7.1. Without this addition, the flume is 1.0 m tall, which could lead to an unwanted potential
spillage of water when the flume is in operation. Also, the increase in the flume height decreases the pressure exerted by water on the lid on top of the distribution pipe. The resistance characteristics of the inner parts of the TFHRC flume, i.e., the honeycomb and screens, were kept unchanged, only the diameter of the inlet pipe was modified from 24 inches to 36 inches as per ERDC requirements. Figure 7.1 shows the geometry and dimensions of the scaled-up flume model.

![Figure 7.1. Dimensions of the scaled up TFHRC flume model.](image)

Once this base model was developed, several modifications were applied to the perforated pipe and flow straighteners. The shape and size of the perforations in the distribution pipe, as well its position in relation to the flow, the number of wire screens, the number of honeycombs, as well as the diameter and length of the tubes that form them were all varied in the CFD analysis to evaluate the effect on the flow at the ERDC flume scale. The results of the various models were analyzed to assess which combination gives the most uniform flow pattern in the testing zone of the flume.

In the next step, to check the design of the converging section of the head section, the model was simplified to include only the converging part of the accelerator and the straight rectangular channel. The downstream surface of the honeycomb was transformed into an inlet surface with assigned uniform velocity, simulating a uniform flow out of the honeycomb. Two cases were compared: (1) with the scaled-up geometry of the converging part unchanged, and (2) with the length of the converging section stretched to three times the length in the main flow direction. This modification, even though unrealistic, makes it possible to assess if under the assumption of a ‘perfect’ flow coming from the honeycomb and screen combination, flow separation points occur in the original geometry, and if so, do they disappear if the slopes of the wall curvatures are reduced to minimum under the existing size restrictions, resulting from the size of the building that hosts the laboratory.

The computational domain in the base case was discretized with the use of the polyhedral mesher in combination with the prism layer mesher. Because the size of the ERDC flume model is significantly bigger than the TFHRC flume model, modeling with the size of volume cells (0.03 m) as the TFHRC model would result in a prohibitively large problem to solve. Instead, a coarser
mesh, with target size 0.08 m, was used in the bigger regions, such as the head section and transition section, and a 0.05 m target cell size was used in the honeycombs and pipe, which resulted in 0.8 million cells in total. A test simulation showed that the water-air interface was not resolved with a required accuracy, and therefore the mesh was modified by making it finer, with target size 0.03 m, in the vicinity of the expected height of the water surface, the interface between phases. This volume mesh consisted of about 3.4 million cells and gave a better approximation of the water surface, without overly increasing the computational time. In both cases, a prism layer was defined on the walls, with the near wall layer thickness 0.014 m that assures wall Y+ values were above 50, which means that the velocity wall functions can be used by the solver to obtain an adequately accurate wall shear stress distribution. A comparison of the two discretizations and resulting velocity magnitude field is shown in Figure 7.2 and Figure 7.3. The coarse mesh gives a similar mean water surface elevation as the finer mesh, but the surface is not as smooth.

The time step size was chosen based on the CFL condition. The mesh size in the main flow direction was 0.03 m, and maximum velocity was 2.6 m/s, therefore, to obtain Courant number less than one, it is sufficient to use a time step of 0.01 seconds. The time needed to obtain 10 seconds of simulated time for the coarse mesh model was 2 hours 40 minutes, and for the fine mesh model was 5 hours 30 minutes, when run on 160 cores. The simulations were run until mass balance was achieved i.e., mass flow rate through the outlet was equal to the mass flow rate at the inlet, which took approximately 250 seconds of simulated time.

The various tested modifications to the models resulted in variations of geometry and therefore the computational mesh, but the above-described general characteristics remained unchanged. The modifications to the geometry, especially the addition of a flume channel, increased the residence time, and therefore the time needed to obtain a converged solution.
Figure 7.2. Coarse mesh of the base ERDC flume model (a) and obtained water velocity magnitude distribution (b).
7.1 Variations in the Design of the Distribution Pipe

A set of simulations testing variations in the design of the inlet distribution pipe was run with the base ERDC model without any flow straighteners. This way the influence of the various shapes and sizes of the perforations on the flow distribution in the flume accelerator could be analyzed without the influence of other design choices.

7.1.1 Varying Porosity of the Original Design. Meshed-out Geometry of the Pipes

Two variations of the size of the circular perforations were tested in the computational study, 1-inch and 2-inch diameter holes with the same distribution of the perforations, which results in a 9% and 35% porosity, respectively. The geometry of the pipe with holes was introduced to the ERDC flume model and meshed out. Meshing out the perforations makes it possible to test these options directly but makes the run very long and computationally expensive. Figure 7.4 shows the geometry of the pipes and Figure 7.5 illustrates the discretization of the CFD domain in the vicinity of the perforated pipe with a very fine grid on the pipe surface, fine grid in the area where the water surface is expected to form, and a coarse grid in the remainder of the water volume.

Figure 7.3. Refined mesh of the base ERDC flume model (a) and obtained water velocity magnitude distribution(b).
Figure 7.4. Geometry of the distribution pipe with circular perforations (a) diameter 1” (porosity 9%), and (b) diameter 2” (porosity 35%).
Figure 7.5. Example discretization of the CFD domain in the vicinity of the perforated pipe with porosity 35%.

Figure 7.6 shows the water surface elevation, (a) a close-up view of the water entering the head section through the perforations in the standpipe, and (b) an overall view of the flow accelerator, obtained from the model with the pipe with 1 in perforations. The modeling approach used in the study, Volume of Fluid, is a numerical technique for tracking and locating the free surface between immiscible fluids. Each cell contains a volume fraction of water, \( \alpha_w \) and air \( \alpha_a \), and \( \alpha_w + \alpha_a = 1 \). The water surface is computed as an isosurface of volume fraction of water equal 0.5. Water flowing from the top rows of perforations in the pipe in Figure 7.6 seems to disappear in midair. In fact, the stream of water turns into a water spray, with \( \alpha_w < 0.5 \) in the cells which was not captured in the figure. The volume fraction of water, from 0 to 1 was illustrated in Figure 7.7 on a plane along the centerline of the flume. The contour plot of velocity for the threshold of volume fraction of water \( \geq 0.5 \) on a plane section along the center line of the flume accelerator is presented in Figure 7.8.

If the perforations have a larger diameter, the water streams in the top section of the standpipe are clearly visible, as shown in Figure 7.9. Water shoots out of the perforations and hits the water surface in the head section, causing the formation of a surface wave with a large amplitude. The velocity field obtained in this simulation is shown in Figure 7.10.
Figure 7.6. Water surface elevation, (a) close-up view of the water entering the head section through the perforations in the standpipe with 9% porosity, (b) overall view of the flow accelerator.
Figure 7.7. Volume fraction of water on a plane section along the center line of the flume accelerator for the model of a perforated pipe with porosity 9%.

Figure 7.8. Contour plot of velocity for the threshold of volume fraction of water $\geq 0.5$ on a plane section along the center line of the flume accelerator for the model of a perforated pipe with porosity 9%.
Figure 7.9. Water surface elevation, (a) close-up view of the water entering the head section through the perforations in the standpipe with 35% porosity, (b) overall view of the flow accelerator.
In the model with a standpipe with 1-inch-wide holes, the flow is more evenly distributed and is ~3 times higher in magnitude along the vertical direction than in the model with a standpipe with 2-inch-wide holes. Bigger perforations produce a wave on the water surface with bigger amplitude and spreading out further from the pipe than if the perforations are smaller. On the other hand, the pressure from the water jet entering the structure on the pipe cap is approximately 4 times greater for a pipe with smaller holes (454 Pa = 0.06 psi) than bigger holes (106 Pa = 0.015 psi).

The contour plots of the velocity magnitude on three planes close to the outlet of the domain are shown in Figure 7.11 for the pipe with (a) 1-inch and (b) 2-inch perforations. The high velocity flow in the rectangular channel for case (a) is close to the side walls and for case (b) the high velocity flow is close to the center of the channel, which is preferable.

In conclusion, the standpipe with 2-inch perforations was selected for further testing, because this model gives a more uniform velocity profile in the downstream section of the accelerator, and because the pressure on the pipe lid is much lower than for the alternative design.
Figure 7.11. Contour plots of the velocity magnitude on three planes in the downstream section of the transition section, (a) perforated pipe with 1-inch-wide perforations, (b) perforated pipe with 2-inch-wide perforations.
7.1.2 Porous Baffle Modeling

The models with meshed out geometry of the perforated pipe take a significant amount of computational time because they require a very fine grid around the pipes, especially in the perforations, to properly resolve the velocity profile. The computational grid may reach approximately 20 million cells and it takes a few days to reach a converged solution. The pipe can be modeled as a zero-thickness porous baffle interface instead, as described in Section 3.1, which does not require such a high level of grid refinement around the pipe. With a porous baffle model, the domain is meshed out with less than 4 million volume cells and therefore the computational time decreases significantly. The porous baffle model was checked against the meshed-out geometry of the perforated pipe to allow for further investigation of computational models with less computer resources and within the project schedule.

Figure 7.12 shows the water surface elevation with a close-up view of the water entering the head section through the standpipe modeled as a porous baffle interface, and an overall view of the flow accelerator. Even though the resulting water surface is missing the details of the flow obtained in the simulation with the meshed-out pipe, which was shown in Figure 7.9, it is very similar to it. The similarities can be also seen in Figure 7.13 and Figure 7.14, where various results from the two models are shown side by side. Figure 7.13 compares the velocity magnitude (m/s) on a vertical plane section along the center line of the flow accelerator and the position Z (m) on the water surface. The plotted range of values is kept the same for both models for clarity of the figure. Figure 7.14 shows the contour plots of the velocity magnitude across the rectangular channel obtained from these models. The two velocity distributions are very similar, with the porous baffle model giving a more uniform and symmetric profile. The model with the porous baffle gives lower magnitudes of the flow velocity, but the difference is only ~6 % relative to the meshed-out geometry, which is not significant for this application. Additionally, the main flow and secondary flow velocity components were presented in contour plots in Figure 7.15. The color bar value range was kept the same for all four plots to show that the secondary flow velocity magnitude is much smaller than the velocity magnitude in the main flow direction.

The uniformity of the flow is assessed by computing a normalized area weighted velocity deviation from the mean velocity, \( \varphi_x \), and by computing a ratio of the absolute value of velocity components perpendicular to the main flow direction to velocity magnitude, \( r_y \) and \( r_z \). The results of the calculations are presented in Table 7-2.

<table>
<thead>
<tr>
<th></th>
<th>perforated pipe with 1” circular holes, porosity 9%</th>
<th>perforated pipe with 2” circular holes, porosity 35%</th>
<th>perforated pipe modeled as porous baffle interface, porosity 35%</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \varphi_x )</td>
<td>0.054</td>
<td>0.033</td>
<td>0.034</td>
</tr>
<tr>
<td>( r_y )</td>
<td>0.086</td>
<td>0.0123</td>
<td>0.0125</td>
</tr>
<tr>
<td>( r_z )</td>
<td>0.088</td>
<td>0.0081</td>
<td>0.0067</td>
</tr>
</tbody>
</table>
Figure 7.12. Water surface elevation, (a) close-up view of the water entering the head section through the perforations in the standpipe with 35% porosity modeled as a porous baffle interface, (b) overall view of the flow accelerator.
Figure 7.13. Comparison of solutions obtained with the model with (a) meshed-out perforated pipe, (b) porous baffle interface model. Velocity magnitude (m/s) is plotted on a vertical plane section along the center line of the flow accelerator. Position Z (m) is plotted on the water surface. The adopted range of values is kept the same for both models.
Figure 7.14: Contour plots of the velocity magnitude across the channel obtained from models with (a) meshed-out geometry of the perforated pipe, and (b) pipe modeled as a porous baffle interface.
Figure 7.15. Contour plots of the velocity in the main direction, $V_X$, and secondary flow velocity, $V_{YZ}$, across the channel obtained from models with (a) meshed-out geometry of the perforated pipe, and (b) pipe modeled as a porous baffle interface. The color bar value range is kept the same for all plots.

7.2 Influence of the Transition Section Wall Geometry on the Flow in the Channel

The inlet pipe and the head section upstream of the accelerator were removed from the model to analyze the influence of the shape of the transition section on the flow conditions in the rectangular channel. For this analysis, an ideal flow in the form of a uniform velocity with no off-axis components was applied on the downstream surface of the first honeycomb. This eliminates the effects of the upstream flow distribution and straightening structures from transition section flow analysis.

Figure 7.16 shows the geometry of a simplified flume model for transition section testing. The results obtained from this simulation were compared against those obtained from a model of the flume with a very long gradual contraction section. The contraction length for this model was three times longer than the original design. The geometry of the elongated model is shown in Figure 7.17. The goal in developing this model was to assess if it is possible to eliminate the secondary flow entirely by making the shape of the transition smoother and eliminating any separation points. The elongation is extreme on purpose: it cannot be longer under the limitation
on the max flume length, and so the optimum geometry should lay in between the original and this one. Figure 7.18 illustrates the decrease in the straight channel length in the modified model.

**Figure 7.16.** Geometry of the simplified flume model without the head section and with the original contraction section.

**Figure 7.17.** Geometry of the simplified flume model with an elongated contraction section
Contour plots of velocity components are compared for the two models in Figure 7.19 and Figure 7.20. The elongated contraction section flume model gives a more uniform flow in the test zone, than the original design, where there is a high velocity flow in the center of the channel. The secondary flow, although much smaller in magnitude as compared to the main flow direction, persists. The differences between the two models are not significant, which leads to a conclusion that the original design is sufficient and extending the transition to some intermediate length is not needed because it would not significantly improve the flow. The characteristics of flow uniformity are presented in Table 7-3.
Figure 7.19. Contour plots of the main velocity component $V_X$ in the flume model with (a) original, and (b) 3 times longer contraction section.

Figure 7.20. Contour plots of the secondary velocity $V_{YZ}$ in the flume model with (a) original, and (b) 3 times longer contraction section.
Table 7-3: Uniformity measures for the simplified flume model.

<table>
<thead>
<tr>
<th>X (m)</th>
<th>original contraction geometry</th>
<th>3 times longer contraction section</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>14</td>
<td>35</td>
</tr>
<tr>
<td>$\varphi_x$</td>
<td>0.038</td>
<td>0.059</td>
</tr>
<tr>
<td>$r_y$</td>
<td>0.011</td>
<td>0.011</td>
</tr>
<tr>
<td>$r_z$</td>
<td>0.011</td>
<td>0.011</td>
</tr>
</tbody>
</table>

7.3 ERDC Flume Model with Varying Honeycombs and Screens

Previous sections provide an analysis of the influence of the distribution pipe and the shape of the contraction section on the water flow in the flume. It was established that a 35% porous distribution pipe with circular perforations works best out of the tested options, and the pipe can be represented with a porous baffle interface with appropriate resistance characteristics; moreover, the shape of the scaled-up contraction originally found in the TFHRC flume is sufficient to give an almost uniform flow in the test zone. This section covers an analysis of other flow straighteners, honeycombs, and wire screens to additionally straighten the flow.

Three honeycombs with varying resistance were analyzed:

(1) a manufactured honeycomb as used in the TFHRC flume, with cell inner diameter 0.37 inch (9.5 mm) and length 3 inch (76 mm),

(2) a honeycomb made of $\frac{1}{2}$” PVC pipes, with inner diameter 0.60 in (15.3 mm) and length 10 inch (254 mm), and

(3) a honeycomb made of 2” PVC pipes, with inner diameter 2.05 in (52.0 mm) and length 20 inch.

These honeycombs vary in opening diameter, which changes the porosity, and pipe length, which influences the development length of the flow in the pipes; all of which will vary the head loss downstream the honeycomb. Moreover, in each of the cases, a model of a wire screen was introduced to the model, with formulation as described in Section 3.4. Figure 7.21 shows the location of the honeycomb and wire screen in the flow accelerator.

Figure 7.21. Geometry of the accelerator with the flow straighteners: perforated pipe, honeycomb, and wire screen.
Table 7-4. Typical dimensions of industrial PVC pipes – schedule 40 [10].

<table>
<thead>
<tr>
<th>Nominal Pipe Size</th>
<th>Outer diameter (in)</th>
<th>Average inner diameter (in)</th>
<th>Minimum wall thickness (in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/8&quot;</td>
<td>0.405</td>
<td>0.249</td>
<td>0.068</td>
</tr>
<tr>
<td>1/4&quot;</td>
<td>0.54</td>
<td>0.344</td>
<td>0.088</td>
</tr>
<tr>
<td>3/8&quot;</td>
<td>0.675</td>
<td>0.473</td>
<td>0.091</td>
</tr>
<tr>
<td>1/2&quot;</td>
<td>0.84</td>
<td>0.602</td>
<td>0.109</td>
</tr>
<tr>
<td>3/4&quot;</td>
<td>1.05</td>
<td>0.804</td>
<td>0.113</td>
</tr>
<tr>
<td>1&quot;</td>
<td>1.315</td>
<td>1.029</td>
<td>0.133</td>
</tr>
<tr>
<td>1-1/4&quot;</td>
<td>1.66</td>
<td>1.36</td>
<td>1.14</td>
</tr>
<tr>
<td>1-1/2&quot;</td>
<td>1.9</td>
<td>1.59</td>
<td>0.145</td>
</tr>
<tr>
<td>2&quot;</td>
<td>2.375</td>
<td>2.047</td>
<td>0.154</td>
</tr>
</tbody>
</table>

7.3.1 Use of One Honeycomb as in TFHRC Flume Model

The first analyzed honeycomb is manufactured from 9.5-mm-inner diameter, 3-inch (76 mm) long tubes. The water level obtained in the simulation with this flow straightener is presented in Figure 7.22. The head loss calculated from equations (1) and (3) is ~0.3 mm at approach velocity 0.26 m/s recorded on a plane upstream of the honeycomb. This small difference in water elevation could not be captured by the discretization. The velocity magnitude and velocity in the channel direction look rather uniform in Figure 7.23(a) and (b), and the maximum value is close to the magnitudes obtained from the previous models, but the lateral velocity fluctuations are not eliminated, and a rotational secondary flow is visible in Figure 7.23(c).

Figure 7.22. Water level established in the flume model with one 3-inch-long honeycomb Ø 9.5 mm.
Three-Dimensional CFD Analysis of Construction Design Alternatives for an ERDC Coastal and Hydraulics Laboratory Flow Accelerator
Figure 7.23. Contour plots of (a) velocity magnitude, (b) velocity in the main flow direction, and (c) secondary flow velocity on plane sections located 14 m away from the center of the inlet pipe in the flume model with one 3-inch-long honeycomb Ø 0.5”.

7.3.2 Honeycomb Built of 5-inch-long PVC Pipes Ø 0.5 Inch

The properties of the honeycomb region in the CFD model were adjusted to represent a flow straightener made from 5-in-long, 0.5-inch-diameter pipes. The water level obtained in this simulation is presented in Figure 7.24. The expected head loss is ~3 mm, and so with a mesh size of 3 cm, the loss is too small to be captured by the model. Nevertheless, the lateral turbulence is reduced which can be seen in the velocity streamline pattern in Figure 7.25, where the spiraling flow that is present in the head section is straighten out and slows down downstream of the honeycomb.

Figure 7.26 and Figure 7.27 show contour plots of velocity on plane sections in the flow accelerator and in the test section of the rectangular channel, with (a) the velocity magnitude, (b) the main flow direction velocity $V_X$, and (c) the secondary velocity $V_{YZ}$. The secondary flow that is clearly visible in the accelerator section, becomes less pronounced as the flow develops in the channel. In the test section, the secondary flow has the highest magnitude close to the walls at the free surface, which should not have a significant influence on the required conditions of the physical testing.
Figure 7.24. Water level established in the flume model with one 5-in-long honeycomb Ø 0.5”.

Figure 7.25. Streamlines of velocity in the flume model with a 5-inch-long honeycomb Ø 0.5”.
Figure 7.26. Contour plots of velocity on plane sections in the flow accelerator in the flume model with a 5-inch-long honeycomb Ø 0.5”, (a) velocity magnitude, (b) main flow direction velocity $V_X$, (c) secondary velocity $V_{YZ}$. 
Figure 7.27. Contour plots of velocity on plane sections in the test zone in the flume model with a 5-inch-long honeycomb Ø 0.5”, (a) velocity magnitude, (b) main flow direction velocity $V_X$, (c) secondary velocity $V_{YZ}$. 
7.3.3 Honeycomb Built of 20-in-long PVC Pipes Ø 2”

Lastly, a honeycomb made of 2 inch in diameter, 20-inch-long pipes was tested in the CFD model. The water level established in this flume model is presented in Figure 7.28. As previously, the honeycomb does not cause significant head loss at the expected approach volume flow; the calculated difference in free surface elevation is 1.5 mm. The influence of the straightening function of the honeycomb can be seen in the velocity streamline pattern. The streamlines of water velocity colored with their magnitude are shown in Figure 7.29, the contour plots of velocity components in the accelerator are shown in Figure 7.30, and in the plots for the test section are shown in Figure 7.31.

This simulation gave the lowest deviation of the velocity component in the main flow direction as well as the lowest ratios of the secondary to main velocity components, calculated according to equations (17) and (19). A comparison of the flow uniformity analysis can be found in Table 7-5.

Figure 7.28: Water level established in the flume model with one 20 in long honeycomb Ø 2”.

Figure 7.29. Streamlines of velocity in the flume model with a 20-inch-long honeycomb Ø 2”.
Figure 7.30: Contour plots of velocity on plane sections in the flow accelerator in the flume model with a 20-inch-long honeycomb Ø 2”, (a) velocity magnitude, (b) main flow direction velocity $V_X$, (c) secondary velocity $V_{YZ}$. 
Figure 7.31. Contour plots of velocity on plane sections in the test zone in the flume model with a 20-inch-long honeycomb Ø 2", (a) velocity magnitude, (b) main flow direction velocity $V_X$, (c) secondary velocity $V_{YZ}$.
Table 7-5. Uniformity measures for various honeycomb types.

<table>
<thead>
<tr>
<th></th>
<th>1 honeycomb as in TFHRC flume</th>
<th>1 honeycomb inner tube diameter 0.5 in, length 5 in, 1 screen</th>
<th>1 honeycomb inner tube diameter 2 in, length 20 in, 1 screen</th>
</tr>
</thead>
<tbody>
<tr>
<td>X (m)</td>
<td>14</td>
<td>14</td>
<td>23</td>
</tr>
<tr>
<td>$\varphi_x$</td>
<td>0.033</td>
<td>0.032</td>
<td>0.052</td>
</tr>
<tr>
<td>$r_y$</td>
<td>0.028</td>
<td>0.054</td>
<td>0.022</td>
</tr>
<tr>
<td>$r_z$</td>
<td>0.017</td>
<td>0.031</td>
<td>0.018</td>
</tr>
</tbody>
</table>

7.4 Influence of the Addition of the Inlet Pipe to the Flume Model

The CFD models presented in the previous section have been assigned a uniform velocity distribution at the inlet boundary, with magnitude derived from the flow rate. This assumption represents idealized conditions, which may differ from the actual conditions in the laboratory. In this test, water was fed from the pump to the flume through a system of pipes that ends with a long horizontal pipe with a 90-degree-bend close to the inlet at the head of the flow accelerator as shown in Figure 7.32. The pipe is approximately 40 hydraulic diameters long, which allows the flow to fully develop. The bend introduces a spiral secondary flow in the entry pipe, which is not dissipated over the short vertical section of the pipe connected to the head section of the flume. The pipe bend near the inlet was added to the model and its influence on the flow in the flume was analyzed. The CFD model used in this part of the study is the same as described in Section 7.3.3 with the difference of the additional pipe with a bend. The inlet boundary surface was moved to the free end on the pipe, and a fully developed velocity profile was applied on this surface. Figure 7.32 illustrates the resulting water surface in the flume. A comparison with Figure 7.28 shows that in this case water does not reach the top of the perforated pipe, and therefore does not induce any pressure on the cap. The addition of the pipe bend reduces the uniformity of the flow in the channel, but the change is not large. The velocity streamlines in Figure 7.33 are slightly less straight, and the contour plots of velocity on plane sections in Figure 7.34 and Figure 7.35 show a more pronounced secondary flow. The results of the uniformity calculations are combined in Table 7-6.

![Figure 7.32: Water level established in the flume model with one honeycomb Ø 2”, 20” long and an inlet pipe.](image)

Three-Dimensional CFD Analysis of Construction Design Alternatives for an ERDC Coastal and Hydraulics Laboratory Flow Accelerator
Figure 7.33: Top view of streamlines of velocity colored its magnitude in the flume model with one 20-inch-long honeycomb Ø 2” and an inlet pipe.
Figure 7.34: Contour plots of velocity on plane sections in the flow accelerator in the flume model with a 20-inch-long honeycomb Ø 2" and an inlet pipe, (a) velocity magnitude, (b) main flow direction velocity $V_X$, (c) secondary velocity $V_{YZ}$. 
Figure 7.35: Contour plots of velocity on plane sections in the test zone in the flume model with a 20-inch-long honeycomb Ø 2” and an inlet pipe, (a) velocity magnitude, (b) main flow direction velocity $V_X$, (c) secondary velocity $V_{YZ}$.

Table 7-6: Uniformity measures for models with or without an inlet pipe.

<table>
<thead>
<tr>
<th></th>
<th>without inlet pipe</th>
<th>with inlet pipe</th>
</tr>
</thead>
<tbody>
<tr>
<td>$X$ (m)</td>
<td>14 23</td>
<td>14 23</td>
</tr>
<tr>
<td>$\varphi_X$</td>
<td>0.033 0.050</td>
<td>0.035 0.057</td>
</tr>
<tr>
<td>$r_Y$</td>
<td>0.026 0.017</td>
<td>0.033 0.023</td>
</tr>
<tr>
<td>$r_Z$</td>
<td>0.020 0.017</td>
<td>0.021 0.017</td>
</tr>
</tbody>
</table>

7.5 ERDC Flume Model with Varying Flow Height

The case set matrix presented in Table 7.1 combines three flow cases, with the same flow rate, but differing slope, and therefore flow velocity and water depth. The first case was a reference case, used to test various parameters of the flow accelerator design. The results were presented in the previous sections. In this section, cases numbered 2 and 3 in the table, which correspond to water depth 0.39 m and 0.12 m respectively, are analyzed for the selected design.

Figure 7.36 presents the flow velocity in the channel at water depth 0.39 m, and Figure 7.37 presents the flow velocity at water depth 0.12 m. In both instances the uniformity of the flow improved as compared to the previously described flow conditions i.e., for water depth 0.84 m. Table 7-7 shows that the lower the water level, the more the ratio of $V_Y$ and $V_Z$ velocity components to the main velocity component decreases. As the hydraulic diameter decreases with shallower water, the boundary layer at the flume walls forms over a shorter distance, as shown in Table 7.1. Consequently, the deviation of $V_X$ in the selected cross-sections increases with decreasing water depth due to the faster developing flow.
Figure 7.36. Color plots of velocity in the test section of the flume model with one honeycomb Ø 2” and water height 0.39 m.
Figure 7.37. Contour plots of velocity in the test section of the flume model with one 20-inch-long honeycomb Ø 2” and water height 0.12 m.
Table 7-7: Uniformity measures for varying water height.

<table>
<thead>
<tr>
<th>X (m)</th>
<th>h_w=0.84 m</th>
<th>h_w=0.39 m</th>
<th>h_w=0.12 m</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>14</td>
<td>23</td>
<td>14</td>
</tr>
<tr>
<td>φ_x</td>
<td>0.033</td>
<td>0.050</td>
<td>0.045</td>
</tr>
<tr>
<td>r_y</td>
<td>0.026</td>
<td>0.017</td>
<td>0.017</td>
</tr>
<tr>
<td>r_z</td>
<td>0.020</td>
<td>0.017</td>
<td>0.008</td>
</tr>
</tbody>
</table>
8 Summary and Recommendations

The U.S. Army Engineer Research and Development Center, ERDC, in Vicksburg Mississippi is planning to build a large flume with a 10-foot-wide flow channel. Water inlet and accelerator sections are needed upstream of the flume channel that will produce an adequately uniform flow going into the channel from water pumped into the inlet section. This study used CFD to analyze the design alternatives for the inlet and accelerator sections that are based on a smaller existing flume at the Turner-Fairbank Highway Research Center. The flume channel width of the ERDC flume is to be 5/3 times that of the TFHRC flume (10 ft vs. 6 ft). That scale-up condition is fixed by ERDC. The maximum height of the ERDC flume inlet section, from the top to the lowest point, which may be below the height of the flume channel bed, is 95 inches, and that constraint is imposed by the height of the building that will be housing the new flume at ERDC. The design alternatives for inlet and accelerator sections that were investigated included options for the inlet flow distribution pipe, options for the contour and length of the converging section, and options for honeycomb and screen flow straighteners. Drawings of the flume can be found in the Appendix.

The results of the CFD analysis of the various options for the ERDC flume inlet and accelerator section designs are given in the following recommendations:

- A straight geometric scale-up of the TFHRC flume inlet and accelerator sections in three dimensions by the factor 5/3 should be combined with extending the top of the side walls and other height related components upward, giving a head section height of 2.2 m (87 in), and a flume channel height of 1.2 m (47 in). This upward extension is near the maximum that can be accommodated in the building that will be housing the ERCD flume and would allow for a maximum range of operating discharge conditions and water depth.

- A single, straight, vertical, cylindrical, inlet flow distribution pipe with a 36-inch inner diameter with circular, 2-inch in diameter, perforations in 30 rings and 37 columns evenly spaced around the circumference performs well, and it was the best of the options tested.

- A single honeycomb flow straightener with screen on the downstream end consisting of 2-inch inner diameter Schedule 40 PVC tubes 20 inches in length, performed best among the tested options. The recommended position for the honeycomb is 3.15 m (124 in) from the center of the inlet pipe.

- A straight section downstream of the honeycomb, but before the transition section was included in the design. Its length, 0.7 m (28 in), is less than recommended in literature (30 – 40 cell diameters) [2], but is enough to eliminate potential flow separation in the contraction. This decision was communicated to ERDC and was approved.
9 Acknowledgements

The funding for this project came from the Hydraulics Research Program at the Turner-Fairbank Highway Research Center, through Interagency Agreement Number DTFH61-14-X-300002 between DOT and DOE, and the work was performed under DOE's contract with UChicago Argonne, LLC, contract no. DE-AC02-06-CH11357.

The authors of this report want to thank the researchers from J. Sterling Jones Hydraulics Laboratory of FHWA, involved in the design of the TFHRC flume accelerator, as well as those involved in the experiments performed on a sample honeycomb. Their work helped significantly in carrying out this research.
10 References

[3] Plascore PC2 honeycomb specifications sheet
11 Appendix

This section provides the information on the recommended design of the ERDC flume.

The shape of the symmetric spiral back walls of the head section is defined with the following parametric functions $X(t) \text{[m]}, Y(t) \text{[m]}$:

\[
X(t) = -(0.764 + 1.224t) \cos t, \\
Y(t) = \pm(0.764 + 1.224t) \sin t,
\]

under the assumption that the point $(X, Y) = (0,0)$ is in the center of the standpipe, and $t \in [0, 1.94] \text{ rad}$, see Figure 11.1.

![Figure 11.1: The recommended geometry of the back walls of the head section.](image)
The functions defining the shape of the transition section walls, \( f_h \), are:

\[
f_h = \begin{cases} 
-a x^3 + 0.5 W_1, & x \in [0, X_h] \\
b(L - x)^3 - Y_h + 0.5 W_1, & x \in [X_h, L]
\end{cases}
\]  

where the length of the transition is \( L = 6.719 \) m, the location of the inflection point is \((X_h, Y_h) = (2.184 \) m, \( 0.5(W_1 - W_2))\), the width of the transition at \( x = 0 \) is \( W_1 = 5.854 \) m, and:

\[
W_2 = 3.054 \text{ m}, \\
a = 4.37 \times 10^{-11}, b = 1.01 \times 10^{-11}.
\]  

(22)

The function defining the shape of the wall in the vertical direction is:

\[
f_{vc} = \begin{cases} 
 c x^3, & x \in [0, X_v] \\
-c(L_2 - x)^3 + H, & x \in [X_v, L_2] \\
H, & x \in [L_2, L]
\end{cases}
\]  

(23)

where the coefficient is \( c = 4.13 \times 10^{-11} \), the height \( H = 1.055 \) m, and the coordinates of the inflection point are \((X_v, Y_v) = (2.338 \) m, \( 0.527 \) m).

The geometry of the lateral walls is presented in Figure 11.2 and the geometry of the vertical walls in Figure 11.3. Figure 11.4 shows the dimensions of the accelerator with a short channel, used in the computations.

![Figure 11.2. The recommended geometry of the lateral converging walls.](image)
Figure 11.3. The geometry of the vertical converging walls of the TFHRC flow accelerator.

Figure 11.4. Dimensions of the flume model with additional straight channel section, (a) side view, (b) top view.