Plan view details of the baffle configurations are shown in Figure 3.

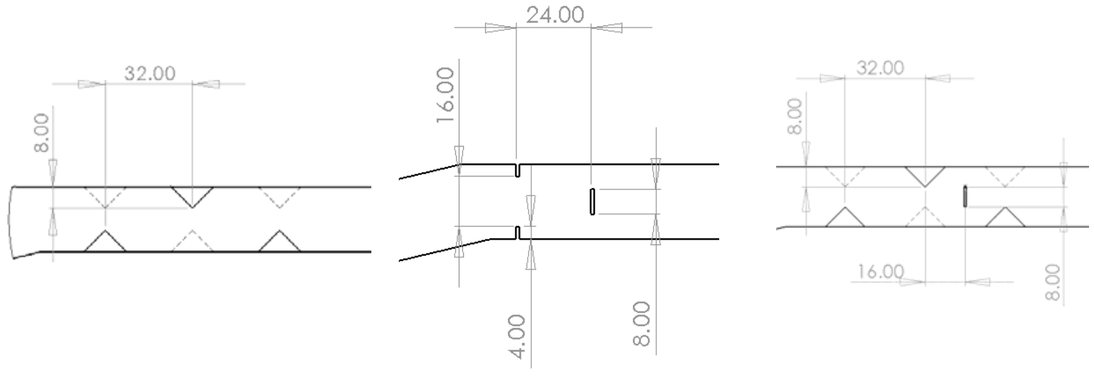


Figure 3. Plan View Details of Baffle Configuration Alternatives

Analysis Method

ANSYS CFX, Version 12, CFD software was used for the simulation. The fluid domain was modeled using an homogeneous volume of fluid approach. Turbulence was modeled using the ANSYS shear stress transport model. Boundary conditions used for the analysis are listed in Table 1.

Table 1. Boundary Conditions

Boundary Region	Boundary Type
Solid walls	Zero slip wall, roughness = 0.003 ft
Top wall	Free slip surface
Inlets	31240 lb/s (502 cfs) in each channel
Outlet	62840 lb/s (1004 cfs)

This is a preview. Click here to purchase the full publication.

Results

Figure 4 shows the results of the various mixing structure configurations as color contours with red representing the fluid from Inlet 1 and blue representing the fluid from Inlet 2.

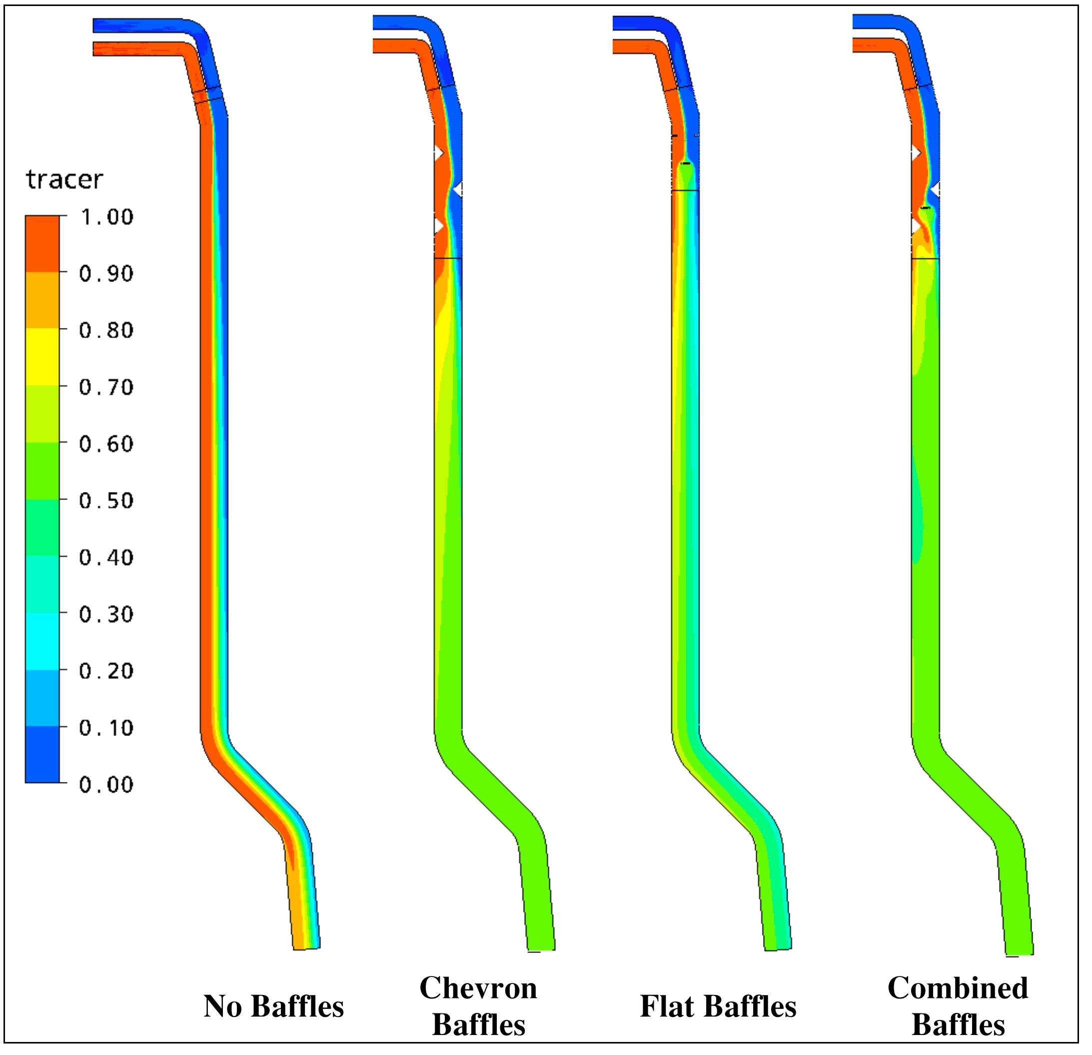


Figure 4. Color Contours of Mixing Characteristics

Cross sections of tracer concentration for each alternative are shown in Figure 5. The cross sections are located downstream of the mixing structures. Velocity vectors have been added to the cross sections to show the cross stream velocity components contributing to the mixing in the channel.

This is a preview. Click here to purchase the full publication.

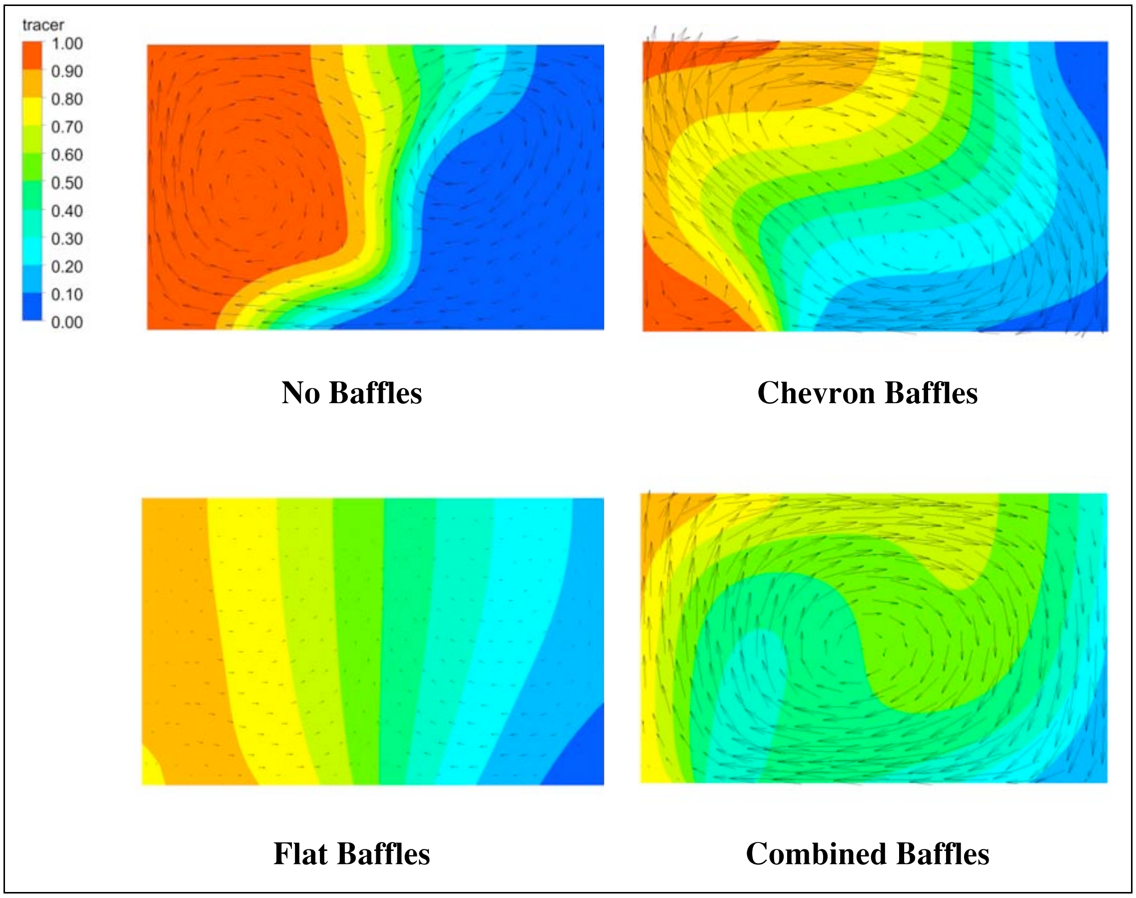
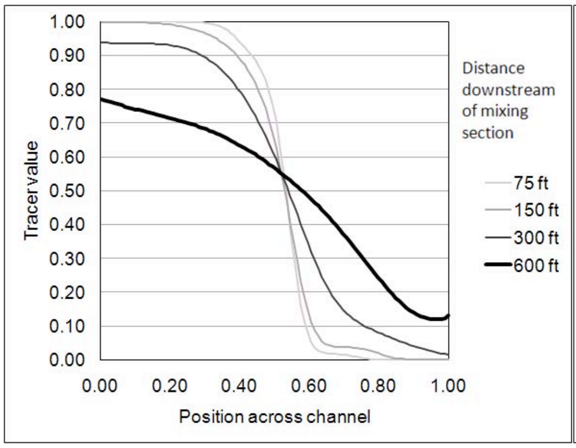


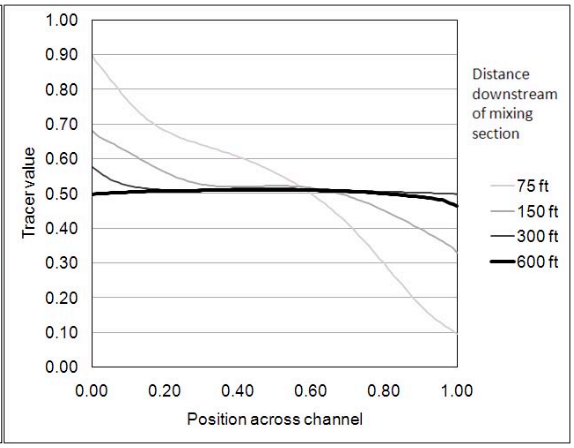
Figure 5. Cross Sections of Tracer Concentration

Profiles of tracer concentration were developed at four locations for each of the four configurations. Data were collected at a height of 10 ft above the channel bottom. The cross-stream profiles of tracer at locations along the model are shown in Figure 6. As can be seen in Figure 6, the Combined Baffle configuration mixed the quickest and the Chevron Baffles mixed second quickest. The Flat Baffles mixed at the slowest rate of any of the baffle configurations. This appears to be partly a result of smaller cross stream velocity components contributing to the mixing in the channel as can be seen in Figure 5.

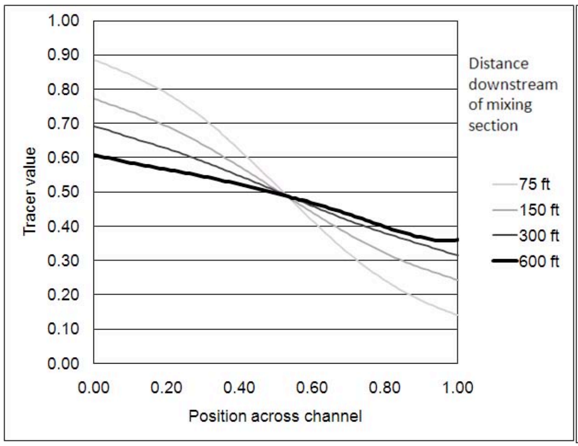
This is a preview. Click here to purchase the full publication.



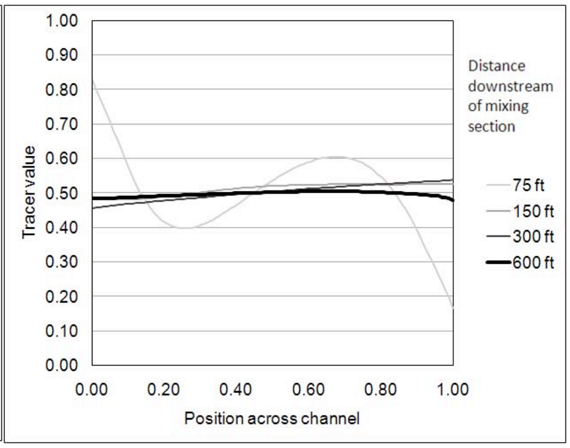
No baffles



Chevron Baffles



Flat Baffles

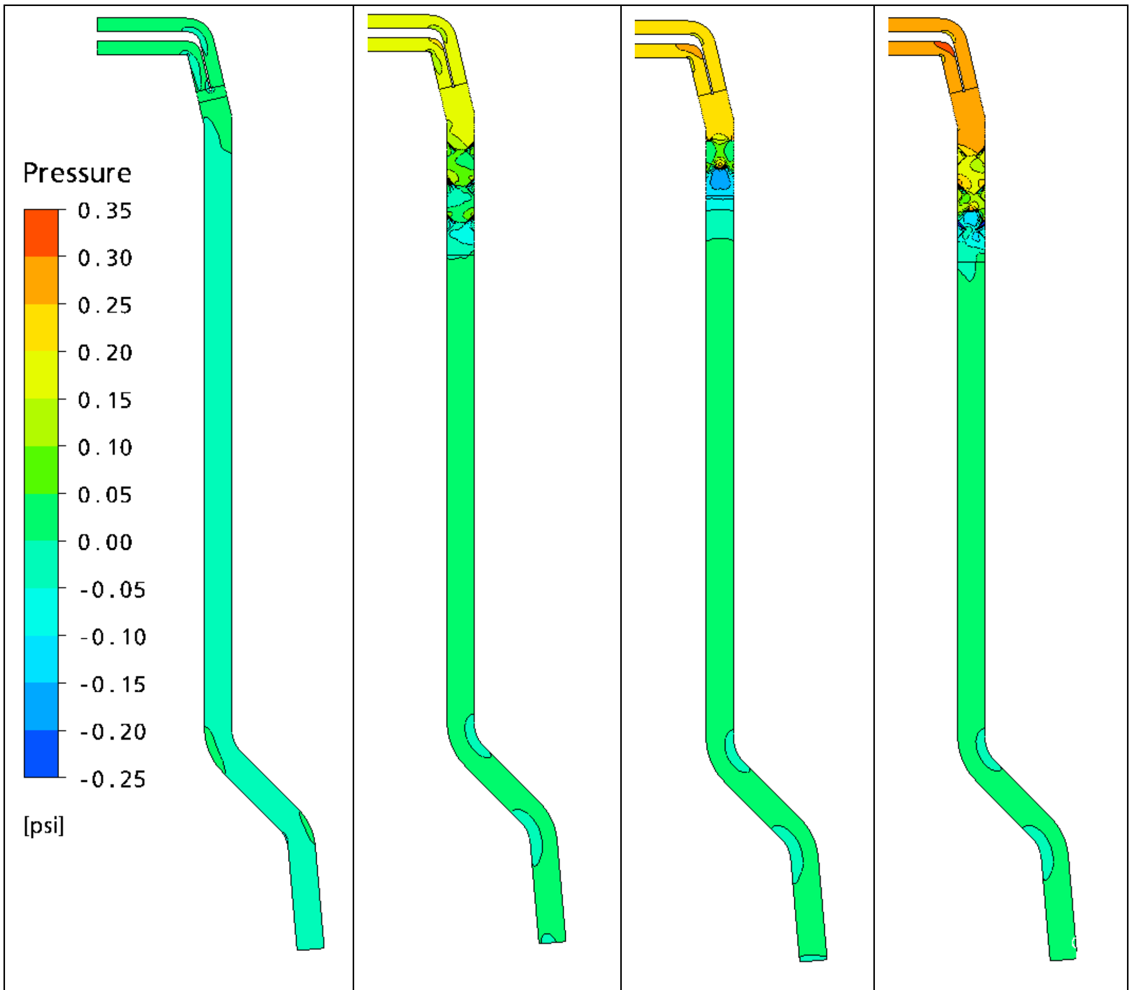


Combined Baffles

Figure 6. Tracer Profiles at Four Locations along the Mixing Channel Length

A relative pressure drop across the mixing structure section was determined by comparing the pressure on the bottom of the channel upstream and downstream of the mixing structure region. The results in psi and ft are shown in Figure 7. While the Combined Baffle configuration exhibited the best mixing it also has the highest pressure drop. The Chevron Baffles exhibited better mixing than the Flat Baffles in addition to having a lower pressure drop. Qualitatively, the best mixing per pressure drop appears to be the Chevron Baffles alternative.

This is a preview. Click here to purchase the full publication.



0.01 psi	0.15 psi	0.22 psi	0.27 psi
0.02 ft	0.35 ft	0.51 ft	0.62 ft
No Baffles	**Chevron Baffles**	**Flat Baffles**	**Combined Baffles**

Figure 7. Headloss across the Mixing Structure

The above analysis was performed with a fairly fine tetrahedral mesh. Additional analysis was performed on a slightly different combining channel and chevron baffle arrangement to evaluate the impact of different meshing strategies on the results. A plan view of the mixing channel layout used for the meshing strategies evaluation is shown in Figure 8.

This is a preview. Click here to purchase the full publication.

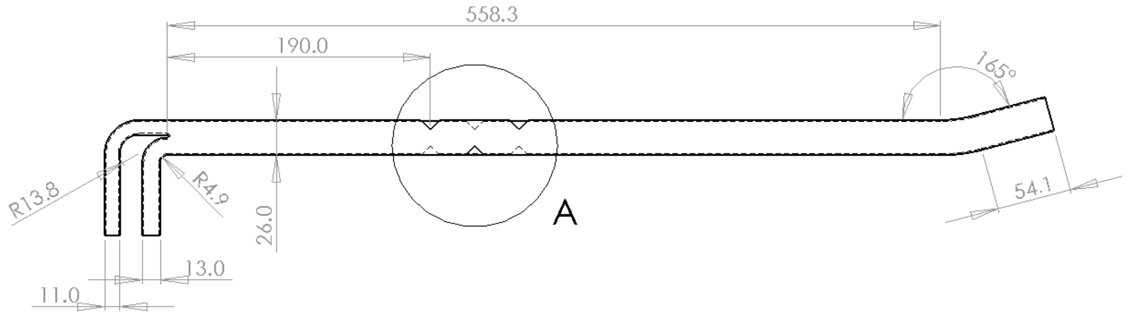


Figure 8. Plan View of Mixing Channel Layout Used for Meshing Strategy Evaluation

Detail A (from Figure 8) of the chevron baffles used for the meshing strategies evaluation is shown in Figure 9.

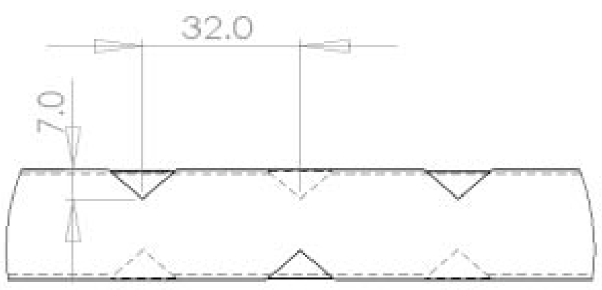


Figure 9. Detail A of Mixing Channel Layout Used for Meshing Strategy Evaluation

A tetrahedral fine mesh, tetrahedral coarse mesh, and a hexahedral mesh were compared for this analysis. The tetrahedral fine mesh has approximately five times more nodes than the tetrahedral coarse mesh. The tetrahedral fine mesh and the hexahedral mesh have a similar number of nodes. Tracer contours comparing the three meshing strategies are shown in Figure 10. Figure 10 illustrates that the meshing strategies can significantly impact the mixing results. Pressure contours comparing the three meshing strategies are shown in Figure 11. The pressure drops using the tetrahedral fine and coarse meshes were both approximately 0.07 psi. The pressure drop using the hexahedral mesh was approximately 0.08 psi. For this short reach, pressure drop did not appear to be sensitive to the meshing strategy.

This is a preview. Click here to purchase the full publication.

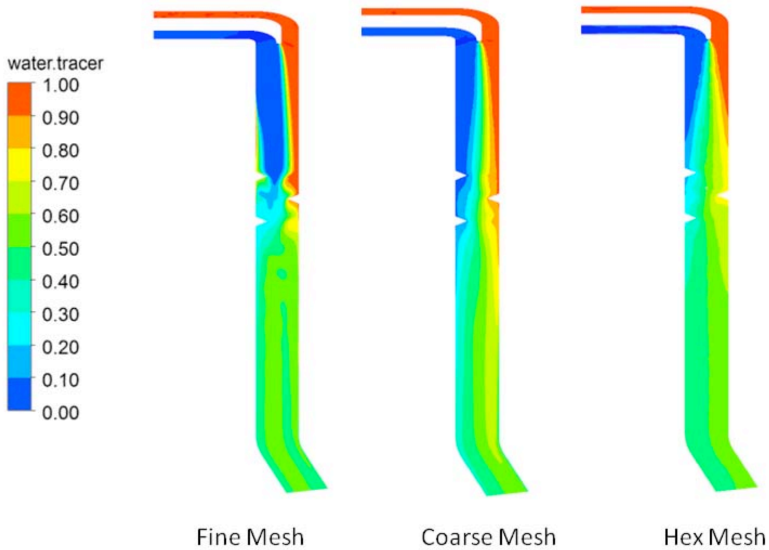


Figure 10. Tracer Contours Comparing Meshing Strategies

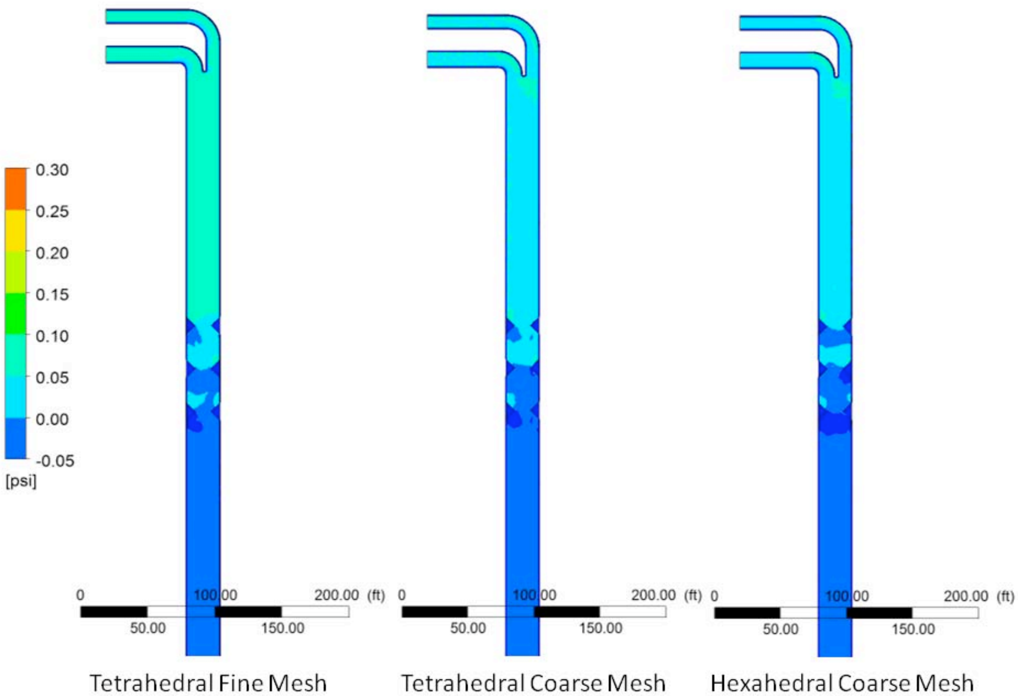


Figure 11. Pressure Contours Comparing Meshing Strategies

This is a preview. Click here to purchase the full publication.

Conclusions

The CFD analysis was an effective design tool for comparing the complex flow patterns associated with passive mixing structures. It provided valuable design guidance for selecting the most appropriate mixing structure considering both mixing characteristics and headloss. It was concluded from the CFD analysis that installation of the alternating chevron baffles provided the best mixing while controlling headloss. The mixing results were sensitive to meshing strategies indicating that care should be used when selecting the meshing strategy.

This is a preview. Click here to purchase the full publication.

FORMED SUCTION INLET DESIGN FOR MIXED FLOW PUMP APPLICATIONS

By

David Werth[1] and Yifan Zheng[2]

The use of formed suction inlets (FSI) on large vertical pumps has been common in the flood control field for many years. The US Army Corps of Engineers Type 10 inlet is a common example that has been used extensively throughout the south eastern United States for flood control as well as on numerous projects as part of the broader Everglades Restoration Project. The inlet is gaining popularity in higher head applications such as circulating water system for power plants.

Extensive design development and research in the performance of the Type 10 Inlet has been undertaken over the years. However, nearly all of this research has focused on Type 10 inlets for flood control applications which typically utilize axial flow pumps. Recently the Type 10 has been used or considered for a wider variety of projects including cooling water applications which typically utilize a radial or mixed flow pump. This type of pump typically has a smaller throat diameter than an equivalent capacity axial flow pump. Given that the design of the FSI is based on the throat diameter there is some question regarding the impact of the smaller resulting FSI design and corresponding increase in velocity.

In several recent applications of Type 10 inlets installed on mixed flow pumps the inlet was designed for a larger pump impeller inlet diameter to increase the size of the FSI and reduce the inlet velocity. A larger than normal reducing cone was then used to transition into the smaller pump throat. This paper presents a comparison of inlet conditions for a variety of mixed flow and flood control applications which were evaluated during physical model studies. Several different "inlet eye diameter" to flow rate or velocity ratios are investigated with an attempt to identify the ideal ratio to use for determining an appropriate diameter for the FSI design. These results can be used by design engineers when considering FSI for mixed flow pump applications and reduce the likelihood of inlet related problems associated with increased inlet velocities.

[1] David Werth, Ph.D., P.E. Principal, Clemson Engineering Hydraulics, Inc. Anderson, SC 29625, e-mail: dwerth@ceh-lab.com, Ph. 864-231-9585
[2] Yifan Zheng, P.E., M. ASCE, Principal Hydraulic Engineer, Bechtel Power Corporation, e-mail: yzheng@bechtel.com, Ph. 301-228-7660

This is a preview. Click here to purchase the full publication.

Introduction

The authors have presented several recent papers regarding the design and performance characteristics of formed suction inlets (FSI). These papers have discussed formed inlet submergence (Werth & Zheng, 2007), optimized intake design for formed inlets (Zheng& Werth, 2008), alternative FSI designs (Werth & Cheek, 2004) and comparisons of different FSI configurations (Werth et. al. 2009). Formed inlets have often been used in large flood control applications (they are often used to replace the typical pump bell on large vertical turbine pumps). Reasons for utilizing a FSI include that they are relatively insensitive to high cross flow conditions, eliminate sub-surface vortex activity, and may reduce the required minimum pump submergence to minimize surface vortex activity when appropriate surface vortex suppression measures are utilized.

Numerous configurations of formed inlets exist with design guidelines being published for several of them. The most familiar of these designs was developed by the U.S. Army Corps of Engineers and is referred to as the Type 10 FSI. Design guidelines for the minimum size of the inlet were published by the Corps (Fletcher and Oswald, 1992) and distributed widely in the 1998 Hydraulic Institute Standards (ANSI, 1998). Photos 1 and 2 show typical Type 10 formed inlets.

Photo 1 Typical Type 10 Formed Suction Inlets

While the Type 10 FSI is relatively well known other variations of inlets are also in service. Werth and Cheek (2004) presented guidelines for an alternative inlet which was intended for use under existing pumps with suction bells already installed. However, the scope of this paper will focus on the well known Type 10.

This is a preview. Click here to purchase the full publication.