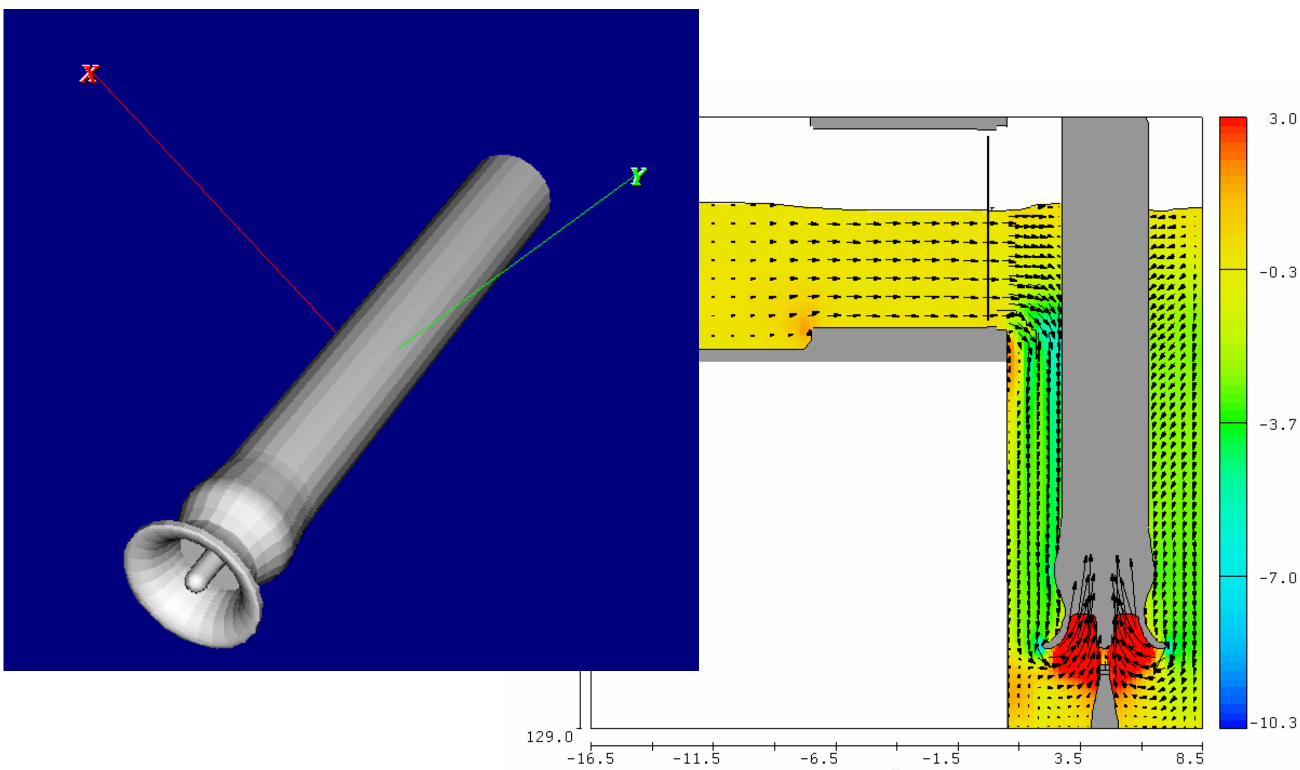
RECLAMATION

Managing Water in the West

Hydraulic Laboratory Report HL-2004-02

Investigation of the Proposed Lake Plant Pump Station

Lower Colorado River Authority



U.S. Department of the Interior
Bureau of Reclamation
Technical Service Center
Water Resources Research Laboratory
Denver, Colorado

December 2004

REPORT DOCUMENTATION PAGE		*Form Approved* *OMB No. 0704-0188*

1. REPORT DATE *(DD-MM-YYYY)*	2. REPORT TYPE	3. DATES COVERED *(From - To)*
10-12-2004	Technical	Jan - Dec 2004

4. TITLE AND SUBTITLE

Investigation of the Lake Plant Pump Station - Lower Colorado River Authority

5a. CONTRACT NUMBER

5b. GRANT NUMBER

5c. PROGRAM ELEMENT NUMBER

6. AUTHOR(S)

Higgs, James and K. Warren Frizell

5d. PROJECT NUMBER

5e. TASK NUMBER

5f. WORK UNIT NUMBER

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

U.S. Department of the Interior, Bureau of Reclamation
Water Resources Research Laboratory
PO Box 25007
Denver, CO 80225

8. PERFORMING ORGANIZATION REPORT NUMBER

HL-2004-02

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)

Lower Colorado River Authority
P.O. Box 220
Austin, TX 78767-0220

10. SPONSOR/MONITOR'S ACRONYM(S)

11. SPONSOR/MONITOR'S REPORT NUMBER(S)

12. DISTRIBUTION/AVAILABILITY STATEMENT

Available from National Technical Information Service, Operations Division, 5285 Port Royal Road, Springfield, Virginia 22161
http://www.ntis.gov

13. SUPPLEMENTARY NOTES

14. ABSTRACT

Investigations of the Lake Plant Pump Station were carried out by Reclamation for the Lower Colorado River Authority. A review of the plant design and layout was completed for compliance with the ANSI Hydraulic Institute Standards for Pumps. A 3-dimensional computational fluid dynamics (CFD) model of a single pump can was created using Flow Science, Inc.'s Flow-3D. The original design was modeled as well as a final design based on the HI review and results from the model run with the initial design. Final can design details are presented that will provide improved flow conditions into the pump bell, lessening the chance of future pump-performance problems.

15. SUBJECT TERMS

CFD modeling, Hydraulic Institute Standards for Pumps, pumping plant, canned pumps.

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT	b. ABSTRACT	c. THIS PAGE			Clifford A. Pugh
UL	UL	UL	SAR	28	19b. TELEPHONE NUMBER *(Include area code)* 303-445-2151

Hydraulic Laboratory Report HL-2004-02

Investigation of the Proposed Lake Plant Pump Station

Lower Colorado River Authority

James Higgs
K. Warren Frizell

U.S. Department of the Interior
Bureau of Reclamation
Technical Service Center
Water Resources Research Laboratory
Denver, Colorado

December 2004

Mission Statements

The mission of the Department of the Interior is to protect and provide access to our Nation's natural and cultural heritage and honor our trust responsibilities to Indian Tribes and our commitments to island communities.

The mission of the Bureau of Reclamation is to manage, develop, and protect water and related resources in an environmentally and economically sound manner in the interest of the American public.

Acknowledgments

These investigations were carried out with input and direction from Jim Dower, Project Manager with the Lower Colorado River Authority. The report was technically reviewed by Joseph Kubitschek of Reclamation's Water Resources Research Laboratory. Final peer review was completed by Clifford Pugh, Group Manager, Water Resources Research Laboratory.

Hydraulic Laboratory Reports

The Hydraulic Laboratory Report series is produced by the Bureau of Reclamation's Water Resources Research Laboratory (Mail Code D-8560), PO Box 25007, Denver, Colorado 80225-0007. At the time of publication, this report was also made available online at **http://www.usbr.gov/pmts/hydraulics_lab/pubs/HL/HL-2004-02.pdf**

Disclaimer

This work was conducted by Reclamation for the Lower Colorado River Authority, Memorandum of Understanding 7/29/2004.

TABLE OF CONTENTS

LIST OF FIGURES

INTRODUCTION

The Lower Colorado River Authority (LCRA) approached Reclamation requesting a time and cost estimate for a physical model study of the proposed Lake Plant Pump Station. LCRA is a utility responsible for managing the water supply and environment of the lower Colorado River basin and developing water and wastewater utilities in central Texas. A modified design of the Lake Plant Pump Station had been provided to LCRA [Figures 1 and 2], and they were well into the contracting and procurement process to initiate construction. After review of the estimate and schedule, LCRA acknowledged that budget and time did not permit the physical model study. Upon further discussions, a Memorandum of Understanding (MOU) [1] between The Lower Colorado River Authority (LCRA) and the Bureau of Reclamation authorized Reclamation to perform limited investigations of the proposed Lake Plant Pump Station. The investigations included:

- a peer review of the proposed design for compliance with the Hydraulic Institute Standards for Pumps;
- recommendations for modifications based on this review;
- a computational flow analysis of a single pump can, including analysis of the initial design and any recommended modifications.

The computational analysis included the original design plus a modified version that provides enhanced flow conditions approaching the pump bell.

CONCLUSIONS

The review of the initial pumping plant design found that the Hydraulics Institute Standards [2, 3] had basically been followed. Plant layout was adjusted slightly to improve flow conditions. Canned pumps are not covered comprehensively in the standards, however with the combination of this review and the computational flow analysis, the intent of the standards was met, with the exception of actually performing a physical model study. Important hydraulic criteria such as net positive suction head (NPSH) and minimum submergence levels were well within the published guidelines.

Computational flow studies showed the initial pump can design had asymmetric flow throughout the pump can and pump bell. These flow conditions may lead to performance issues, including submerged vortices, and possible adverse vibrations. The presence of either submerged vortices or significant vibrations can lead to reduced discharges, accelerated bearing wear, and possible impeller damage.

The recommended pump can design has rotated the horizontal vanes 22.5°, added a turning vane, and altered the height of the vertical vanes. These changes provide for more symmetrical velocities in the pump can and pump bell and will improve performance over the initial design.

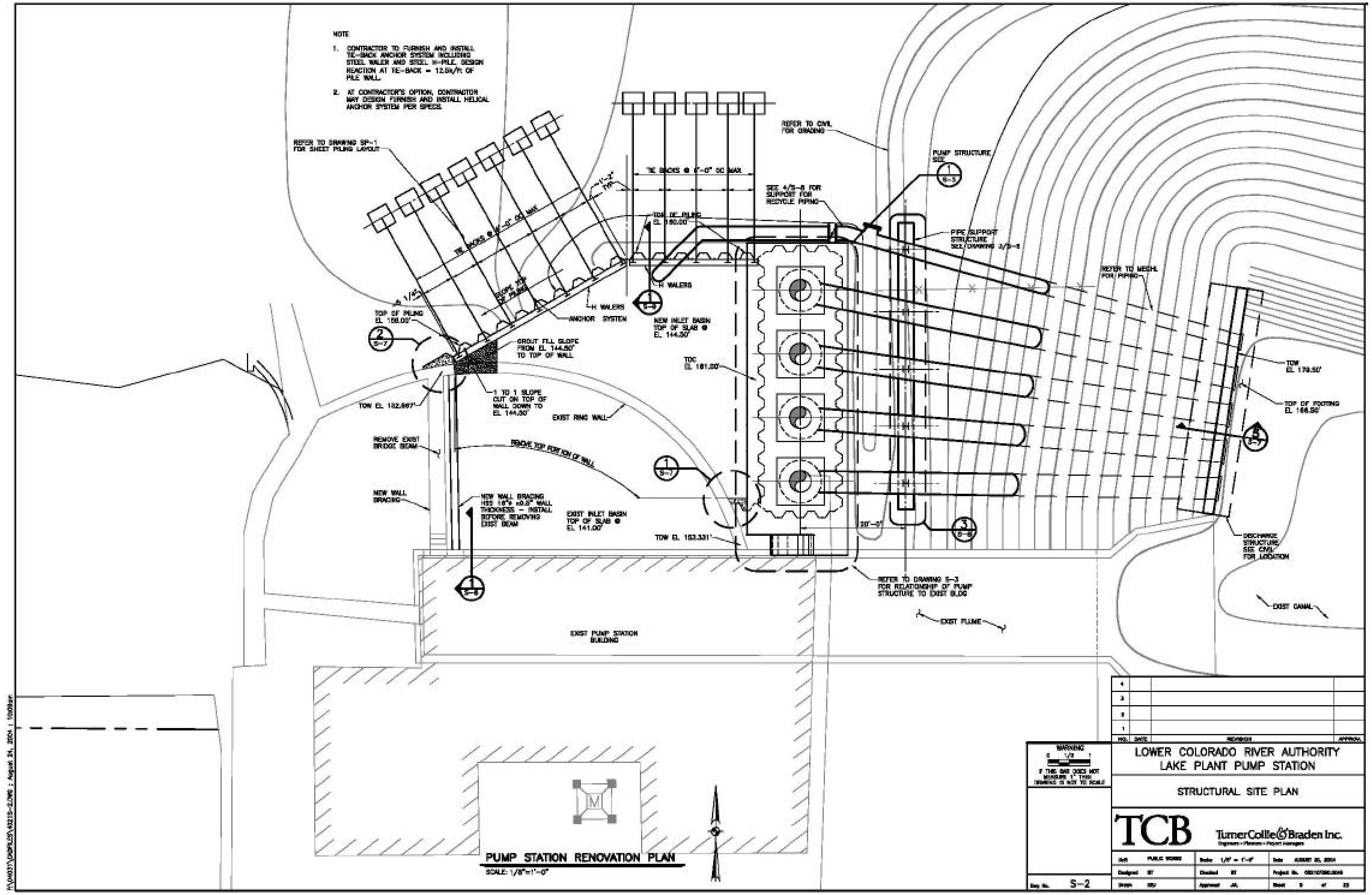


Figure 1. Site Plan for the Lake Plant Pump Station.

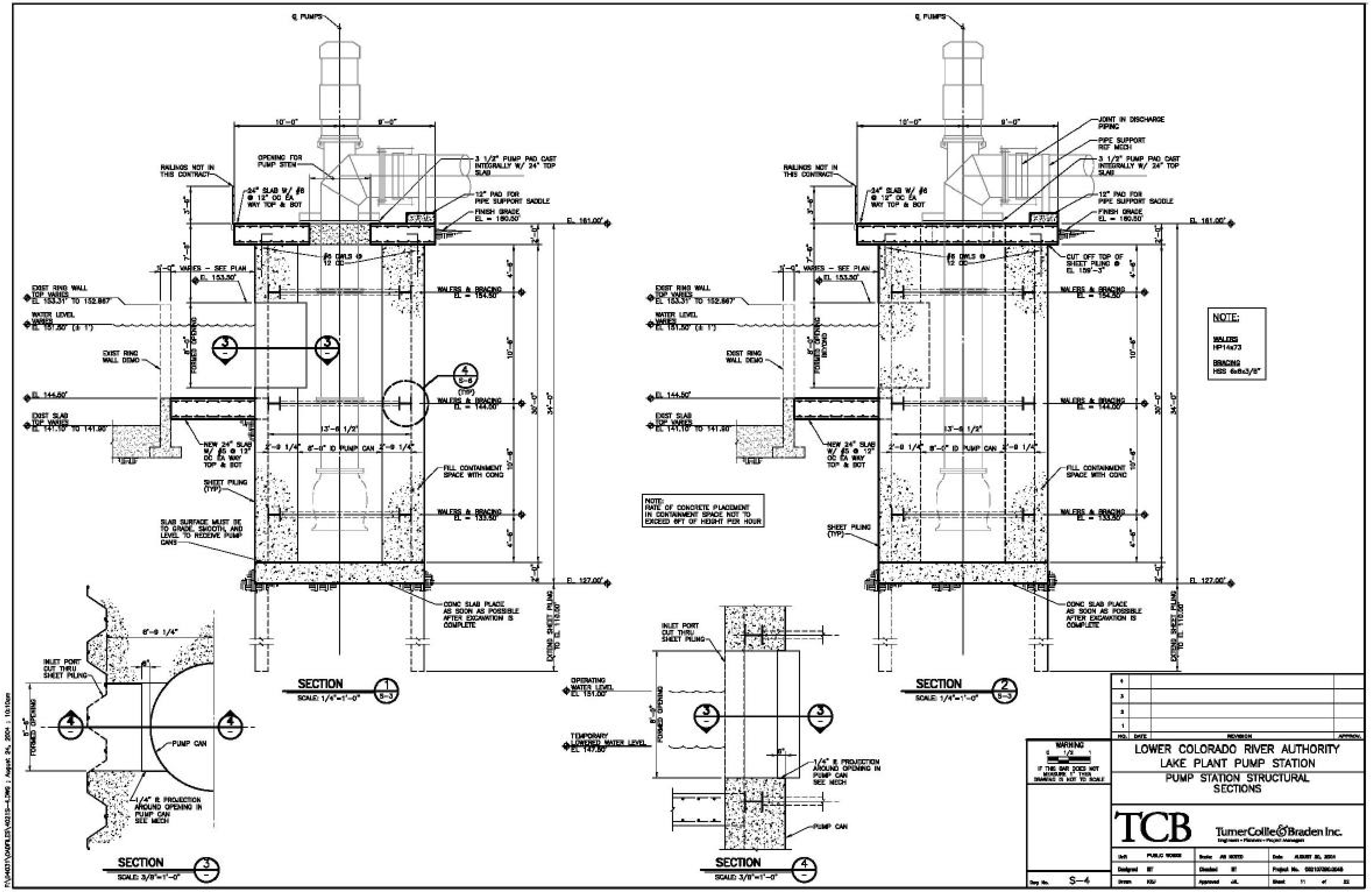


Figure 2. Section through a typical canned pump. Basically shows extents of the CFD model.

HYDRAULIC INSTITUTE STANDARDS REVIEW

The ANSI Hydraulic Institute Standards for Pumps (HI) is a compilation of tested and verified designs and guidelines for pumping plants and intake designs. Not every possible combination is found within the standards and hence they readily recommend performing additional studies (mainly physical hydraulic model studies) in order to verify the proper operation of a specific design.

Canned pumps, like those found in the Lake Plant Pump Station design are only briefly addressed in the standards, putting much of the onus on the pump/can manufacturer to assure proper operation. Generally a canned pump is a purchased unit, designed and tested by the pump manufacturer. In the case of the Lake Plant design, the cans are designed and built separate from the pumps. A canned pump is created by dropping a typical off-the-shelf pump inside the manufactured can. This was likely done for two reasons, cost savings and lack of available designs in the size range needed. Considering this design methodology and the size of the pumps, the HI standards recommend a physical model study to verify proper operation.

Since a physical model study was not performed, a review of the pertinent criteria that is available was performed along with a computational model. Dimensions of the pump can and recommended internal vanes were reviewed. The initial design had followed these basic criteria. The setting of the pump elevation was also reviewed, looking at the NPSH and minimum submergence criteria. The pump bell elevation was found to be well within the recommended setting and actually at quite a conservative level. Entrance velocities into the can and down the can approaching the pump bell were verified to be within the guidelines. The sump design when using canned pumps is not specifically addressed by the HI standards, so several minor modifications were suggested in an attempt to provide uniform flow across the width of the pump station, approaching the cans. While internal features to the cans, such as vertical and horizontal vanes were included in the design, the details and resulting performance of these vanes is not known. Without the benefit of a physical model study, a computational model was proposed.

CFD MODELING

Reclamation has performed several CFD models of proposed pump stations in the past, mostly at the feasibility level in order to assist in the layout of the plant. These studies have varied in the amount of detail but the results have been quite helpful in the early stages of a design. There are many steps required to develop an appropriate CFD (Computational Fluid Dynamics) model. These include development, refinement, and testing of the grid, boundary conditions, model extents, and obstacles (structures) for the CFD program.

CFD Program Description

The CFD program FLOW-3D Version 8.2 by Flow Science, Inc. [4], was used to model a single pump can for the Lake Plant Pump Station. FLOW-3D is a finite volume, free surface, transient flow modeling system that was developed to solve the governing Navier-Stokes equations, in three spatial dimensions.

The finite difference equations are based on an Eulerian mesh of non-uniform hexahedral control volumes using the Fractional Area/Volume (FAVOR™) method [5]. Free surfaces and material interfaces are defined by a fractional volume-of-fluid (VOF) function. FLOW-3D uses an orthogonal coordinate system as opposed to a body-fitted system.

Model Description

The computational model included a section of approach, a trash rack, the pump can, including internal vanes, and the pump bell and a section of the pump column. The approach or sump extended 21 ft upstream from the centerline of the pump, and was 12 ft wide. A rectangular channel (5.5 x 6.27 x 9 ft) connected the sump to the cylindrical pump can. The trash rack was modeled as a porous object 3-inch thick with an open volume fraction of 0.84 (i.e. percent open area). The pump can was generated with the inverse of a 3D solid object (stereolithography) generated using AutoCAD (Figure 3). The pump bell was also a 3D solid object generated using AutoCAD (Figure 4).

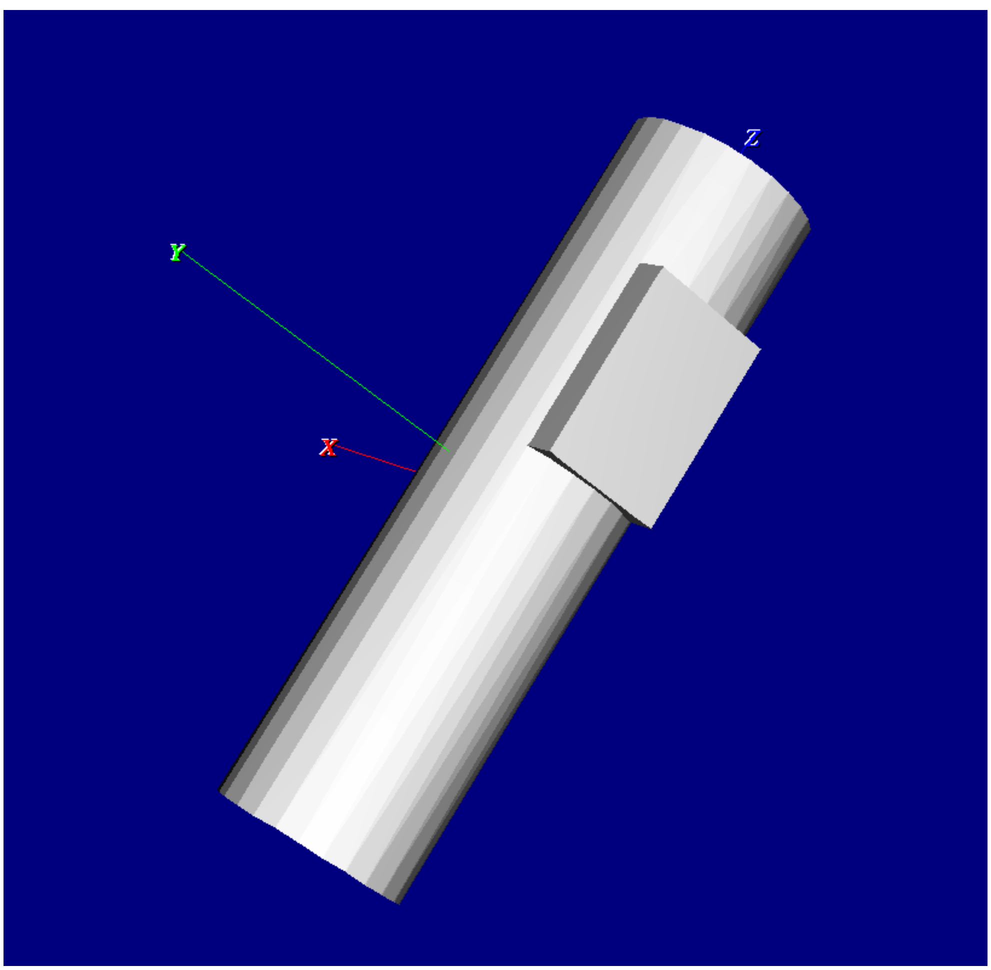


Figure 3. Pump can inverse object. The inverse of this stereolithography object was used to simulate the open volumes of the pump can.

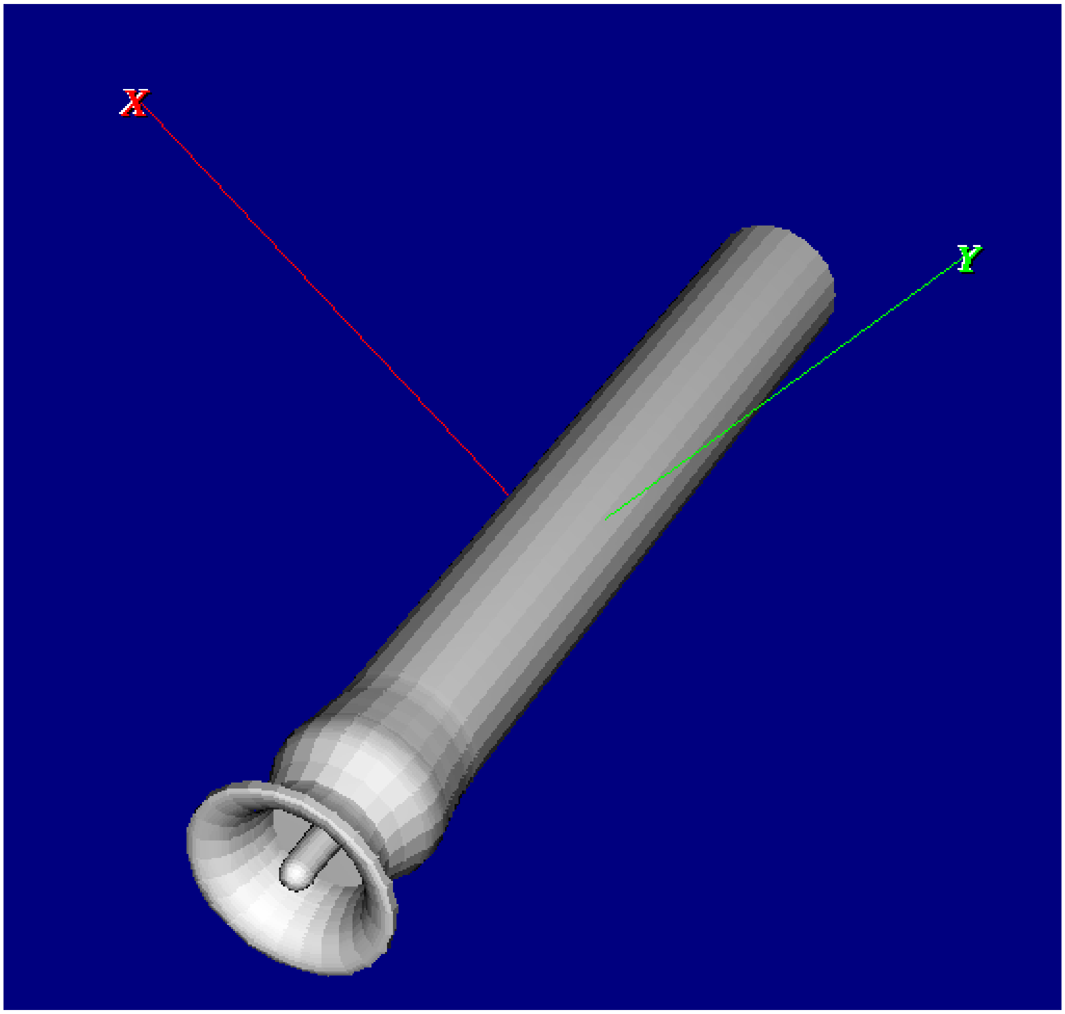


Figure 4. Pump object. This stereolithography object was used to simulate the pump. An obstacle "sink" was added on the plane inside the pump bell to model inflow of the pump. The lip of the pump bell was widened to be 2-3/4 inches rather than ½ inch to improve the models' performance and stability of the CFD process.

Mesh-Blocks

The CFD process used various cell configurations and spatial extents to optimize computation time. While smaller cell sizes develop more precise definition of obstacles and flows, they also increase the size of the computational domain, and decrease the time step (when the explicit option is used) of the simulation. Both of these typically increase computational time required for obtaining a quasi-steady state solution. Balancing the accuracy of the solution with the time and computational resources available is always a challenge.

Flow-3D can use multiple hexahedral shaped mesh-blocks, containing the non-uniform hexahedral control volumes. In Flow-3D V8.2, multiple mesh-blocks are allowed and can be nested (one inside another) or placed adjacent (one mesh-block can share a side with another mesh-block).

As is typical of this type of modeling, several mesh-block techniques were used to optimize performance for the reasons discussed above. The final model used two adjacent mesh-blocks, where a sparse mesh block defined the approach, and rectangular intake channel, while a high density mesh-block defined the pump can, pump piping and pump bell. Details of the mesh-block techniques used for the final simulations can be found in Appendix B.

Boundary Conditions

Boundary conditions applied to a CFD model simulate how the fluid acts/reacts at the sides of the mesh-blocks, which are the extents of the model.

Two sides of the upstream mesh-block pass through through the concrete and into the sump. These are the minimum and maximum Y values (approach streamwise left and right). One boundary is the sump, the minimum X (upstream). The models simulated the reservoir at these three boundaries using a static pressure boundary with a water elevation of 150.5 ft. The boundary for the maximum X (downstream) for the upstream mesh block, and the minimum X (upstream) for the downstream mesh block were defined as *mesh* since they shared a common plane and flow passed through them. Flow did not pass through any of the other boundaries, thus they were defined as a wall boundaries.

Other Options

The model used the Renormalized Group (RNG) option for viscosity, which is an advanced turbulence simulation technique. The RNG model uses equations similar to the more familiar k-*e* turbulence model. In the k-*e* model, equation constants are found empirically but are derived explicitly in the RNG model. Generally, the RNG model has wider applicability than the k-*e* turbulence model. In particular, the RNG model is known to describe low turbulence intensity flows and flows having strong shear regions more accurately.

The first-Order advection approximation was used for the momentum equation approximations, and the line implicit - successive over-relaxation option was chosen for pressure iterations.

RESULTS AND DISCUSSION

This study investigated the flow conditions throughout the whole model, but focused on

- Symmetry of velocities in the pump bell, both horizontal and vertical.
- Indications of rotational flow or vortices in the pump can and bell.
- Pressures in the pump bell.
- Symmetry of vertical velocities in the pump can

These parameters appear to encompass most of the performance issues that were of concern during the study.

Initial pump can design

The initial pump can design is shown in Appendix A – Initial Prototype Drawings.

The CFD simulation of the initial pump can design indicated poor down flow distribution in the pump can (Figure 5). This results in a highly asymmetric velocity distribution (Figure 6) and pressure distribution (Figure 7) in the pump bell. A significant cause of the problem was due to flow separation at the point where the flow enters the pump can. The area just below the inlet elevation (145.5 ft) shows a recirculation (figure 5) and if you look at sections taken at various elevations below this (Figure 8, Figure 9, and Figure 10), the asymmetries and circulation patterns are very apparent. Note elevations are given on the left axis of figure 5. Significant upwelling occurs at the front of the can, i.e. the side where the flow enters.

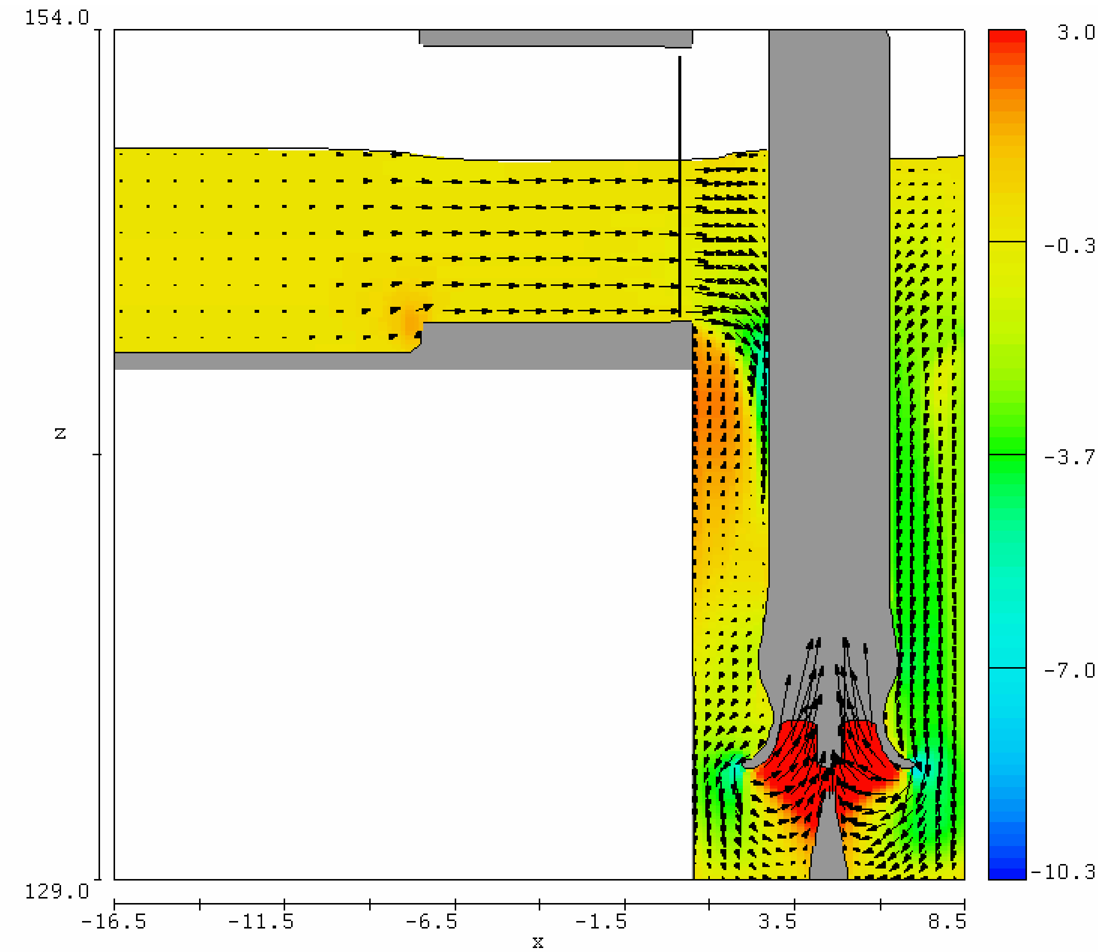


Figure 5. Initial design, profile of flow. Notice the upwelling to the left in the pump can and stronger down flow on the right, and the asymmetric approach at the 3 ft/s contour into the pump bell. Color contours show vertical velocities in ft/s. The longest vector represents 20.2 ft/s in the X-Z plane.

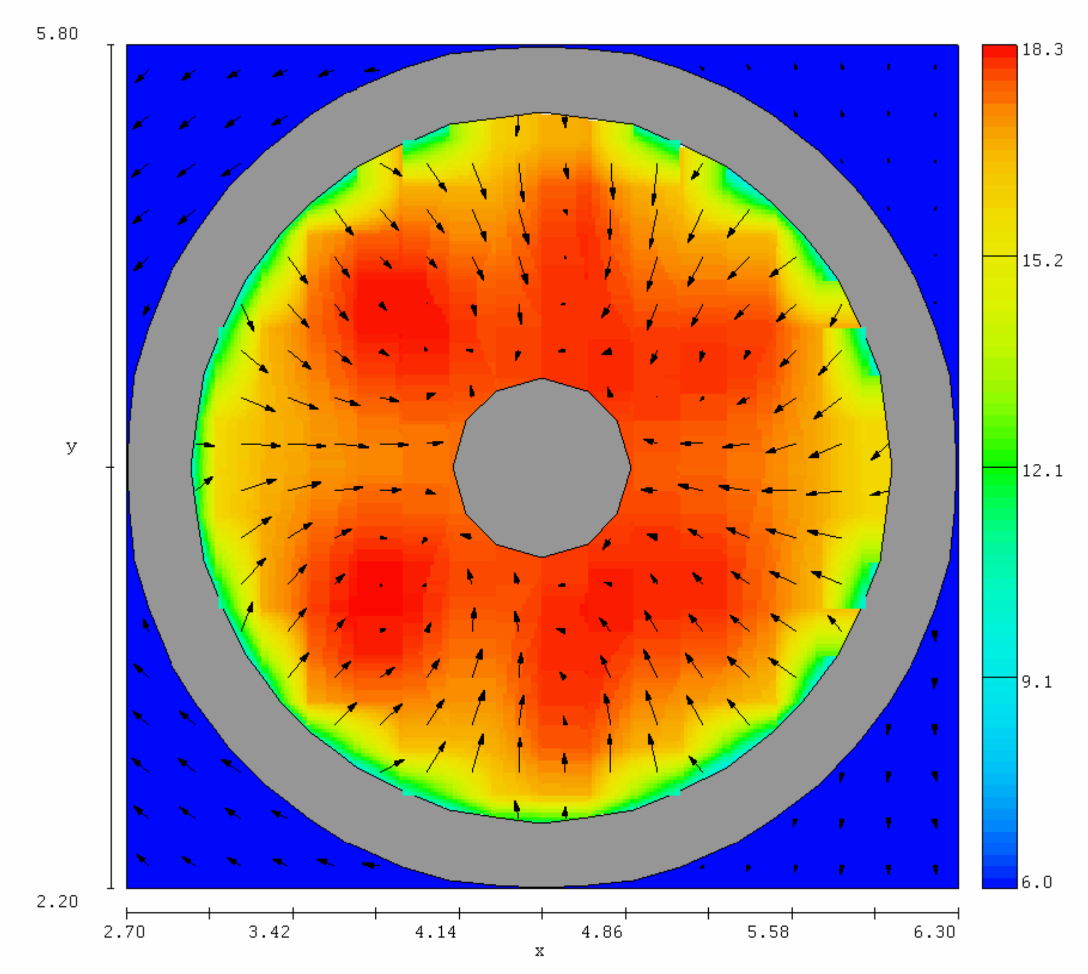


Figure 6. Initial design, pump intake velocities at z=133.1 ft, just inside the pump bell. The simulation displayed asymmetrical flow and possible vortices. Color contours show vertical velocities in ft/s, and limited between 6 and 18.3 ft/s to accentuate the asymetery. The longest vector represents 5.87 ft/s in the X-Y plane.

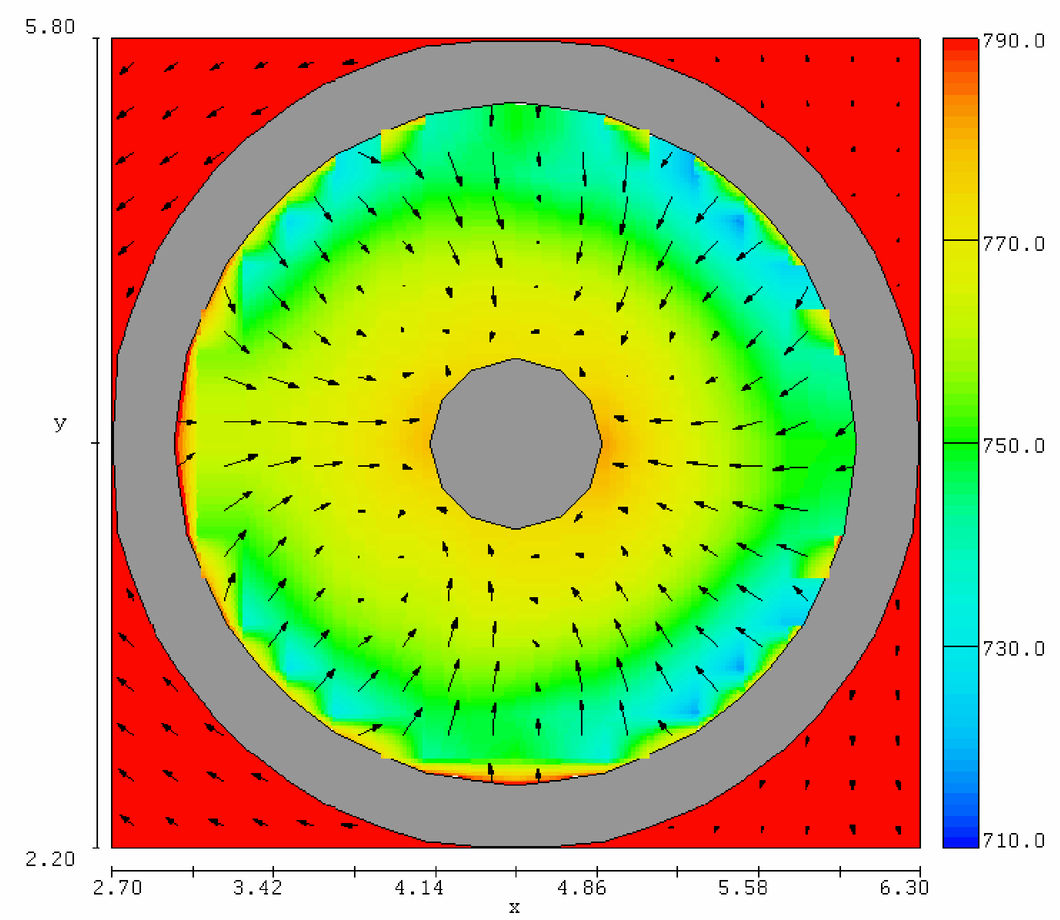


Figure 7. Initial design, pump intake pressures at z=133.1. The simulation displayed asymmetrical pressures. Color contours show pressure in lbs/ft^2, and limited between 710 and 790 lbs/ft^2 to accentuate the asymetery. The longest vector represents 5.87 ft/s in the X-Y plane.

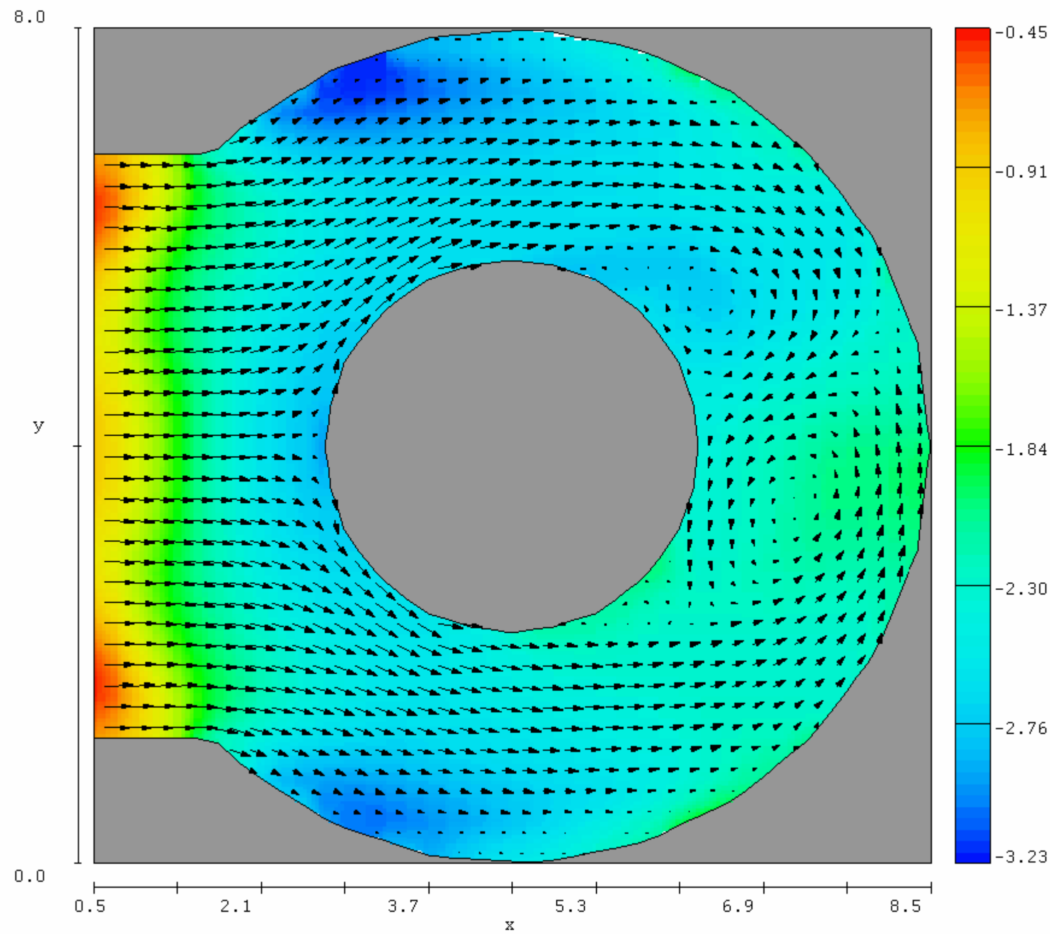


Figure 8. Initial design at Z=145.7 ft. Inflow appears to be concentrating to the right. Color contours show vertical velocities in ft/s, and displays the extreme vertical velocities. The longest vector represents 5.90 ft/s in the X-Y plane.

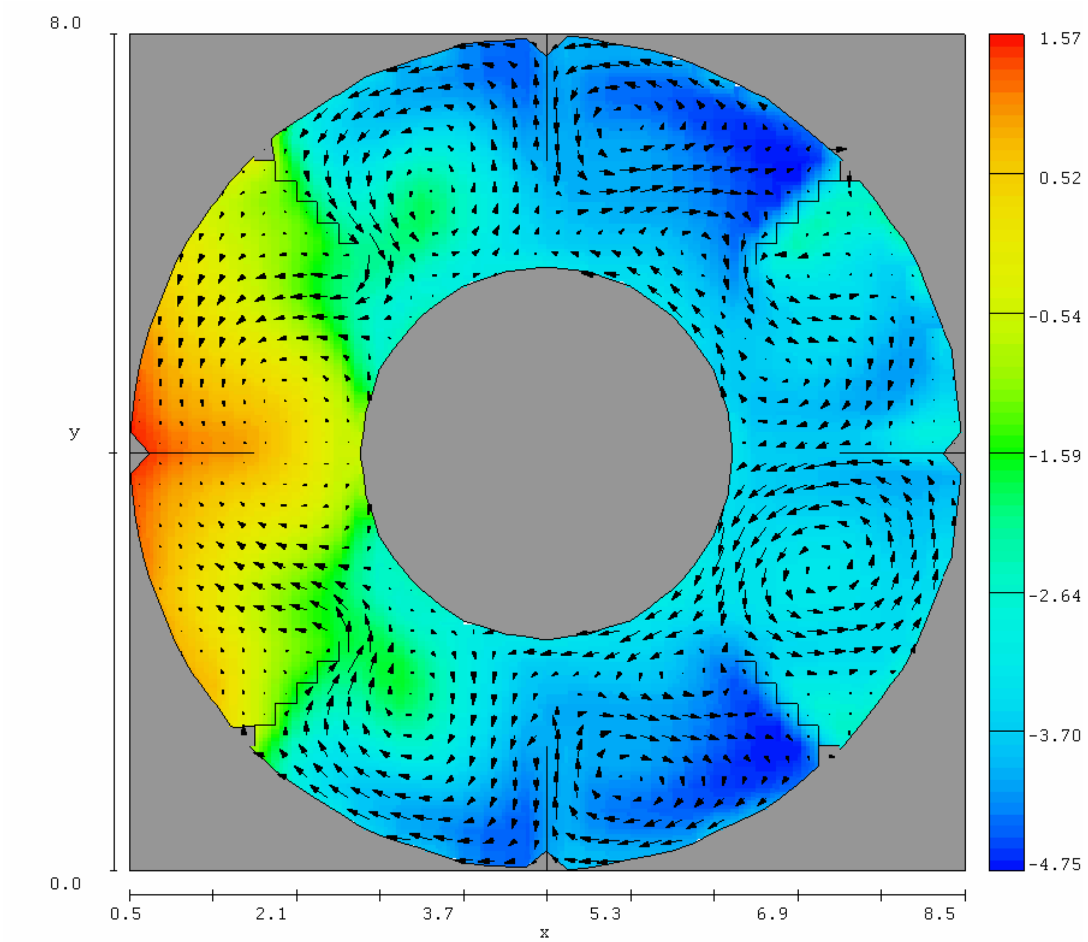


Figure 9. Initial design at Z=139.7. Significant upwelling appears on the left, while a considerable amount of flow is moving from the right to the left. Color contours show vertical velocities in ft/s, and displays the extreme vertical velocities. The longest vector represents 1.72 ft/s in the X-Y plane.

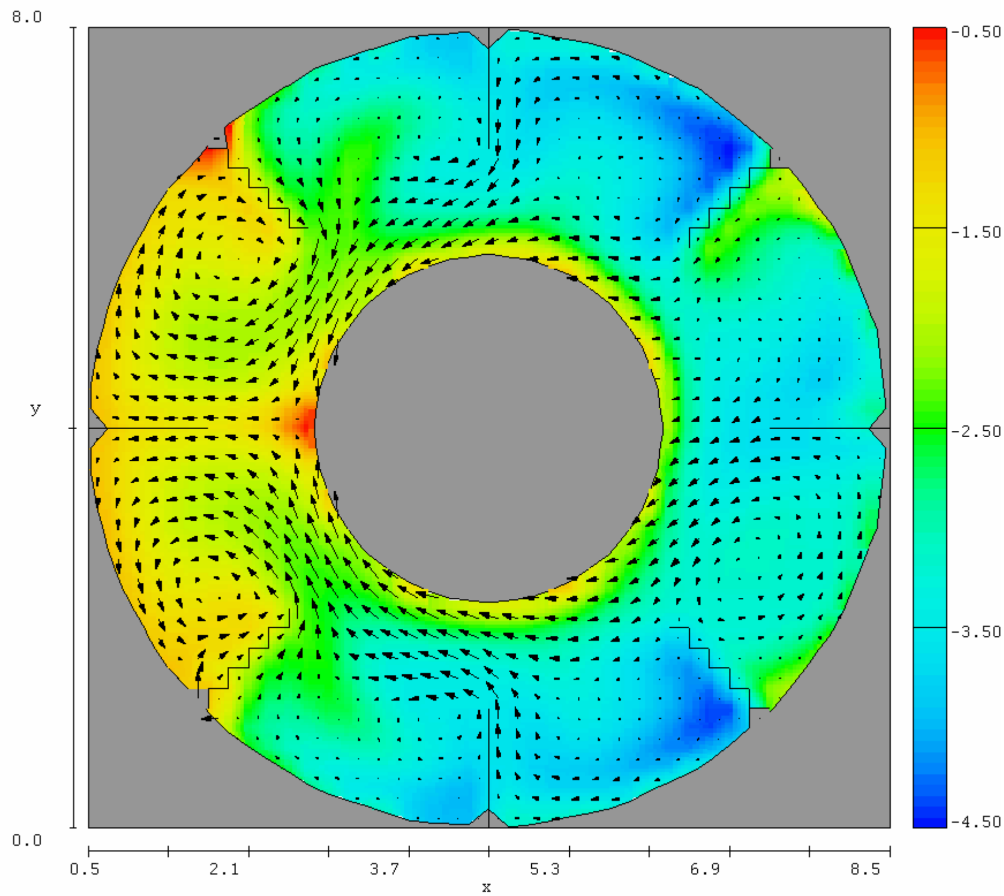


Figure 10. Initial design at Z=134.3. Significant upwelling appears on the left, while considerable flow is moving from the right to the left. The horizontal vanes disrupt the high horizontal velocities. Color contours show vertical velocities in ft/s, and display the near extreme vertical velocities. The longest vector represents 2.43 ft/s in the X-Y plane.

Recommended pump can design

To improve intake conditions, the study focused on an improving flow conditions in the pump can shortly above the impeller housing (Figure 10). This focus was based on the supposition that symmetrical flow conditions near elevation Z=134.3 ft would provide symmetrical flow conditions at the pump intake. This included achieving a more uniform vertical velocity and smaller horizontal velocities in that section. Improvements were made through the use of a turning vane (Figure 11) to improve down flow conditions near the entrance of the pump can, rotating the horizontal vanes 22.5°, and changing top elevations of the vertical vanes (Figure 12) to capture an adequate amount of downward flow between the vanes.

Through a series of trial-and-error investigations, six variations were simulated. Though limited by time constraints, the final simulation proved to be adequate. The symmetry of vertical velocities were greatly improved (Figure 13), velocities (Figure 14) and pressures (Figure 15) at the intake were nearly symmetrical, horizontal and vertical velocities were improved (Figure 16, Figure 17, and Figure 18). Sections were taken at the same locations as shown previously in the initial design.

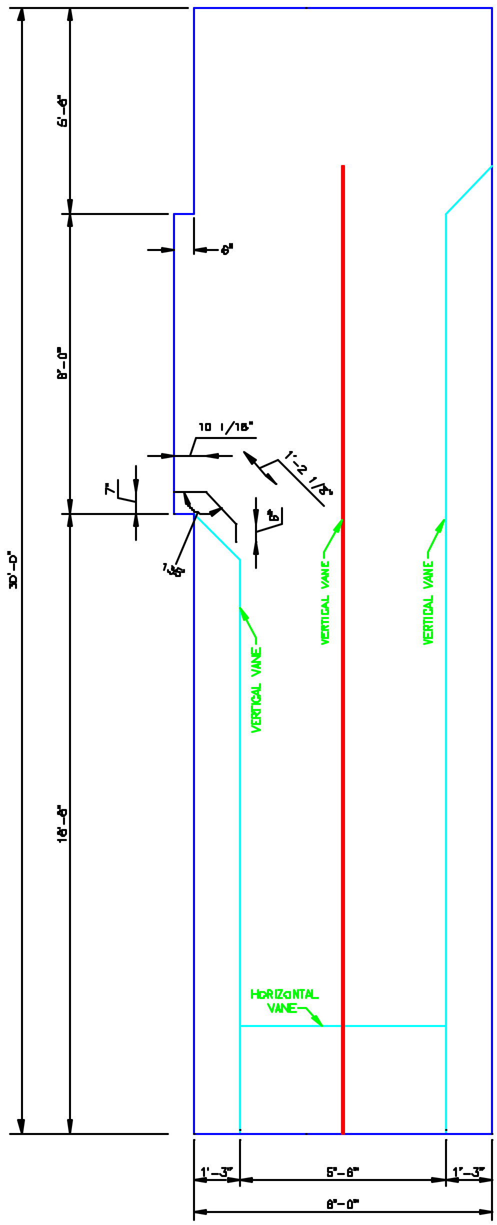
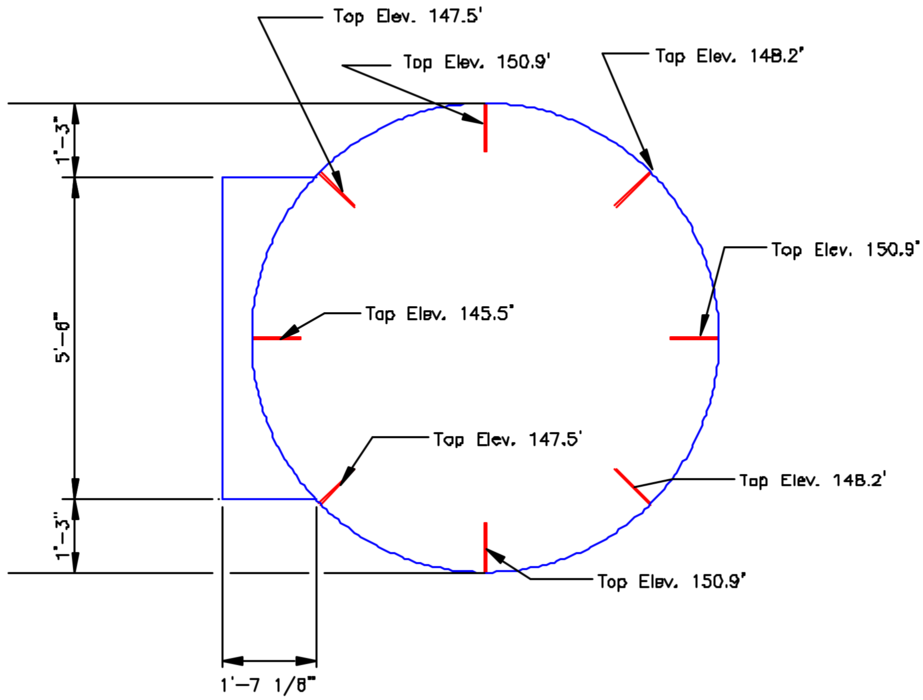


Figure 11. Turning vane details. The turning vane was used to reduce the amount of upwelling at the front of the can.



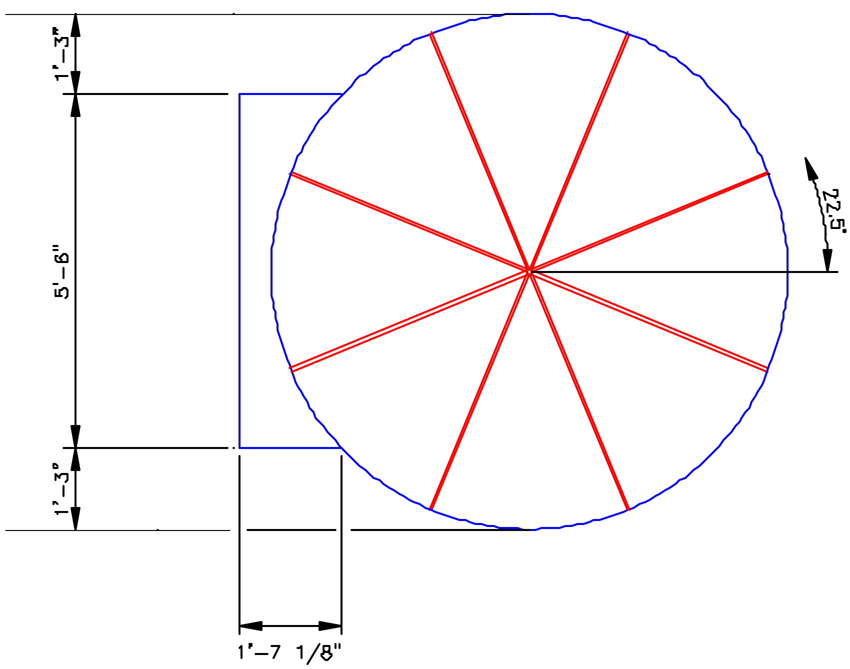


Figure 12. Details of the vertical vanes (top) and horizontal vane (bottom). The recommended design altered the top elevation of the vertical vanes and rotated the horizontal vanes by 22.5°.

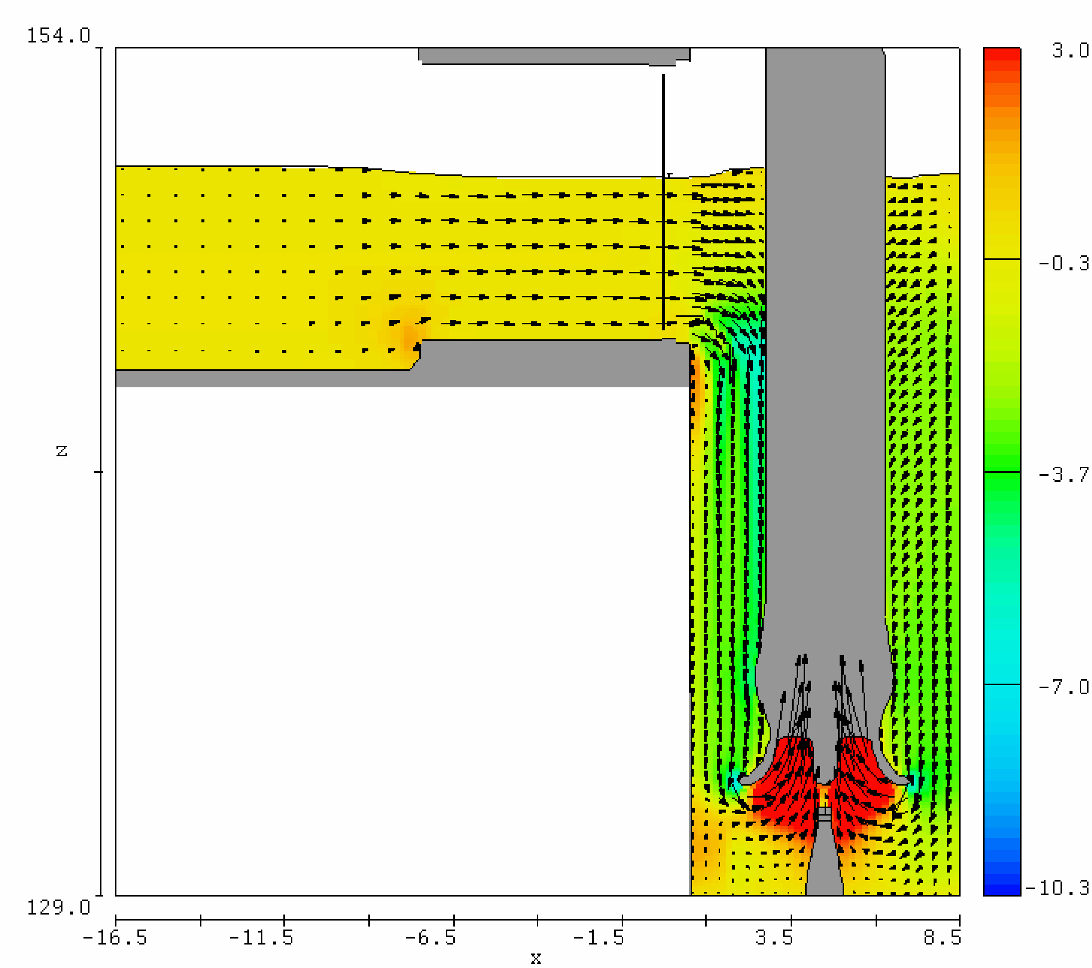


Figure 13. Recommended design, profile of flow. Notice the slight upwelling below the entrance of the pump can, but nearly equal down flow velocities through the rest of the can, and the nearly symmetric approach at the 3 ft/s contour into the pump bell. Color contours show vertical velocities in ft/s, and were limited for direct comparison to Figure 3. The longest vector represents 20.4 ft/s in the X-Z plane.

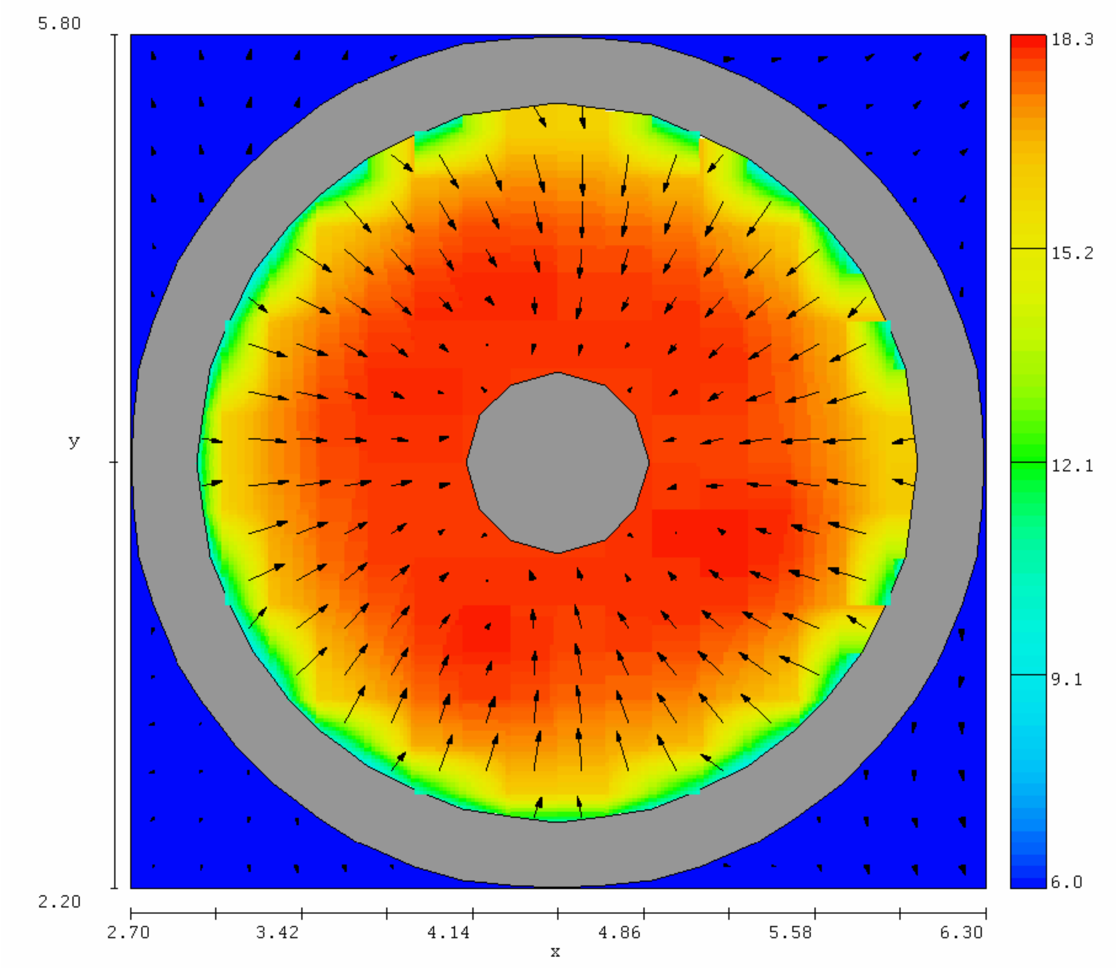


Figure 14. Recommended design, pump intake velocities at z=133.1. Color contours show vertical velocities in ft/s, and were limited for direct comparison to Figure 3. The longest vector represents 5.80 ft/s in the X-Z plane.

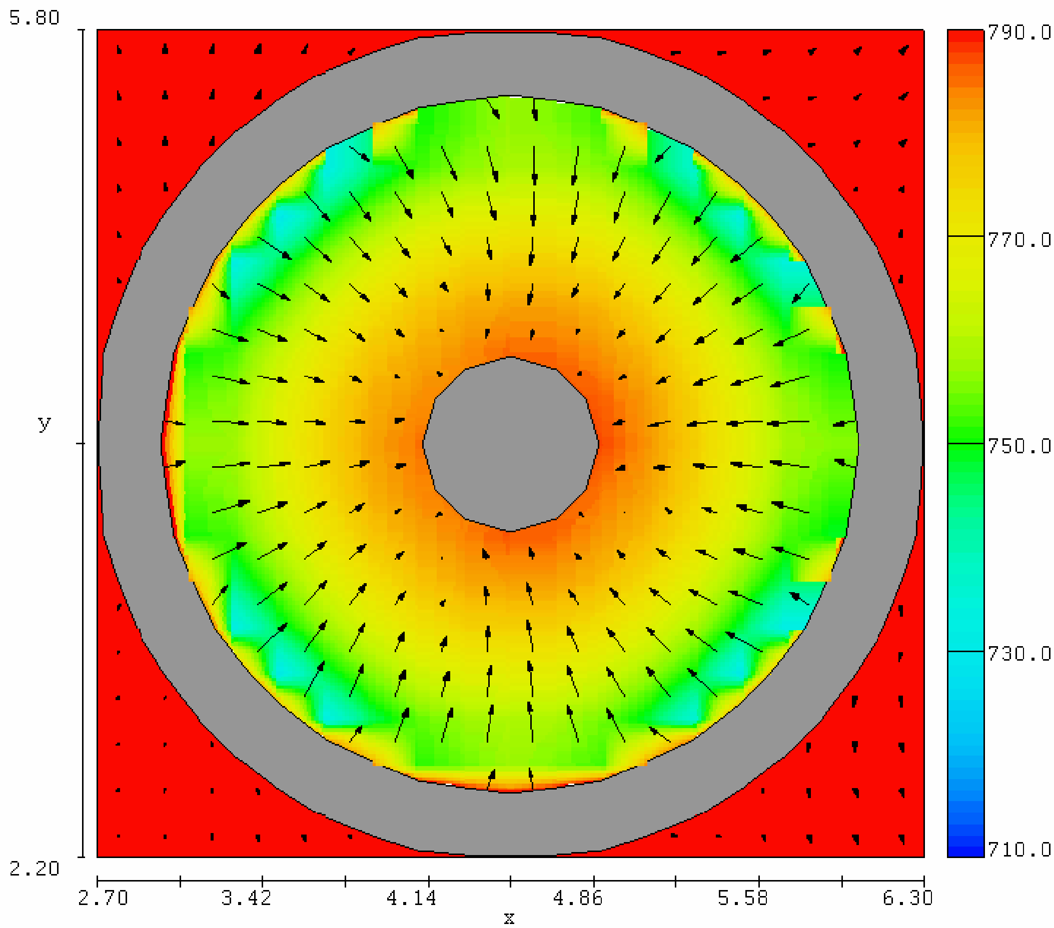


Figure 15. Recommended design, pump intake pressures at z=133.1. Color contours show pressure in lbs/ft^2, and were limited for direct comparison to Figure 7. The longest vector represents 5.80 ft/s in the X-Z plane.

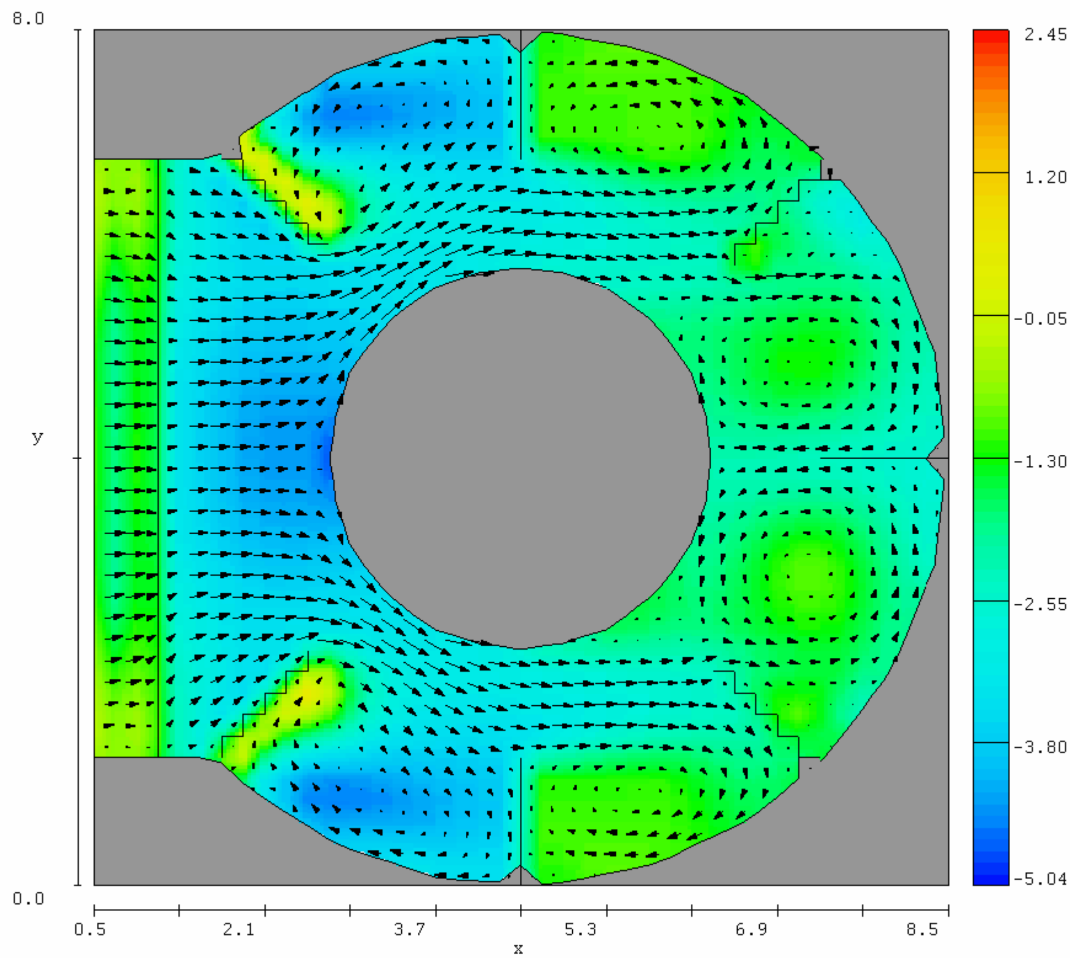


Figure 16. Recommended design at Z=145.8. Color contours show vertical velocities in ft/s, and were limited for direct comparison to Figure 8. The longest vector represents 5.65 ft/s in the X-Y plane.

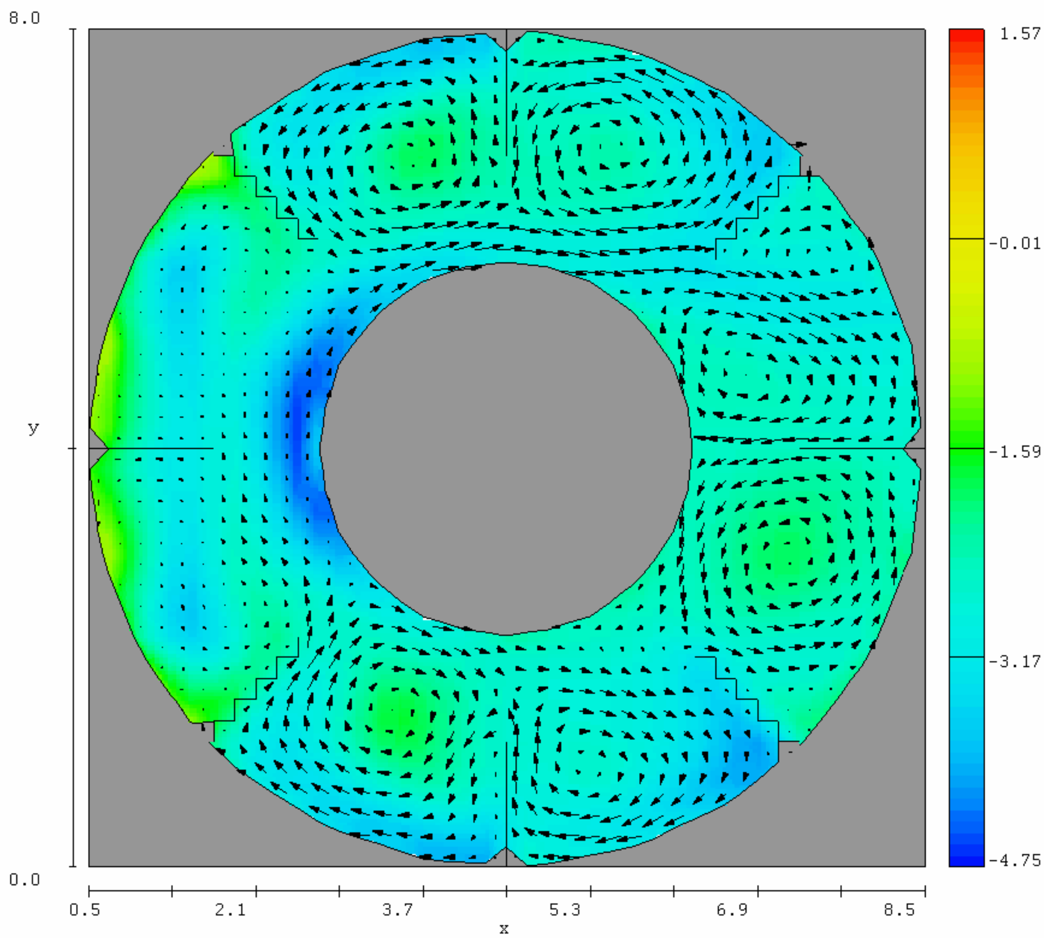


Figure 17. Recommended design at z=139.8. Color contours show vertical velocities in ft/s, and were limited for direct comparison to Figure 9. The longest vector represents 1.72 ft/s in the X-Y plane.

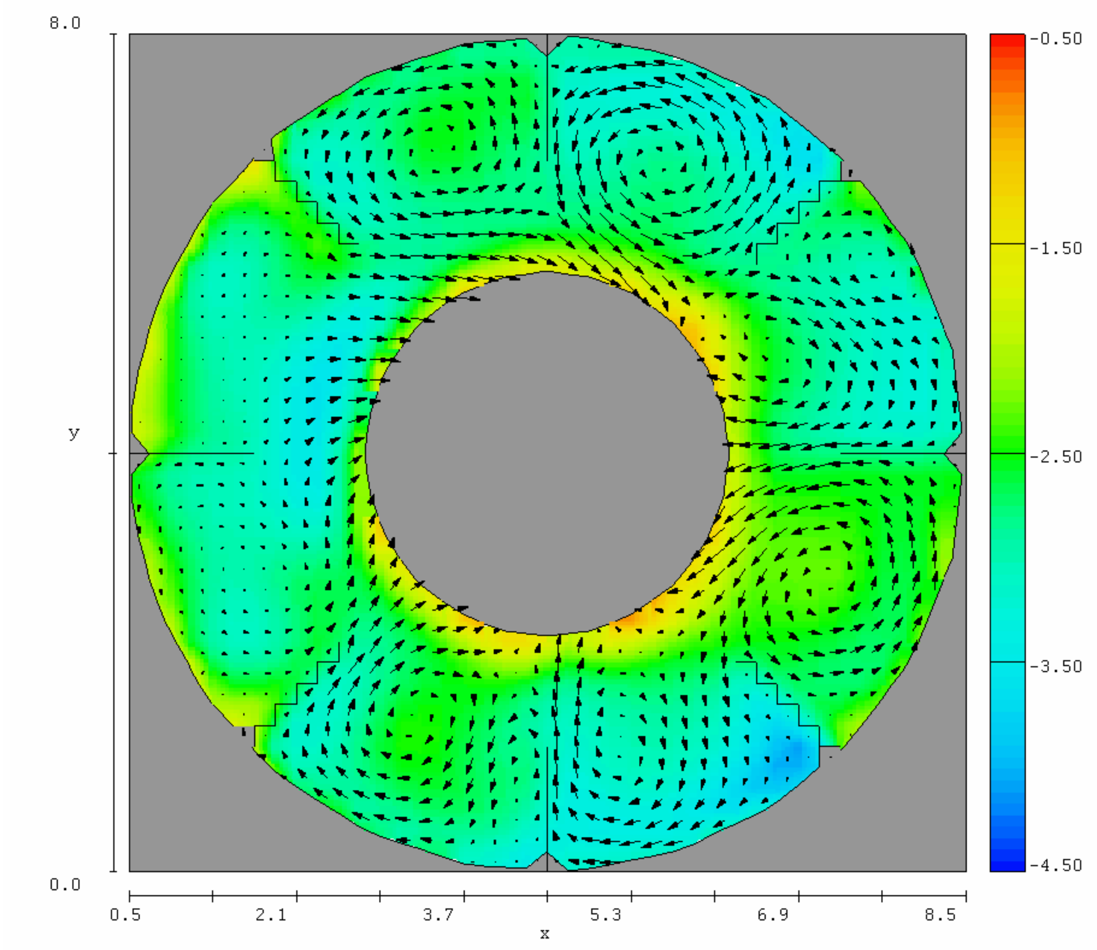


Figure 18. Recommended design at Z=134.3. Color contours show vertical velocities in ft/s, and were limited for direct comparison to Figure 10. The longest vector represents 1.04 ft/s in the X-Y plane.

REFERENCES

1. MEMORANDUM OF UNDERSTANDING BETWEEN U.S. Department of the Interior, Bureau of Reclamation And The Lower Colorado River Authority For Investigation of the Proposed Lake Plant Pumping Plant. July 29, 2004.
2. *ANSI/HI 2.1 - 2.5 - 2000* American National Standard for Vertical Pumps for Nomenclature, Definitions, Application and Operation.
3. *ANSI/HI 9.8 - 1998* American National Standard for Pump Intake Design
4. Flow Science Inc., FLOW-3D version 8.2 User's Manual. 2003.
5. J.M. Sicilian, "A FAVOR Based Moving Obstacle Treatment for FLOW-3D," Flow Science, Inc. Technical Note #24, April 1990 (FSI-90-TN24).

APPENDIX A – INITIAL PROTOTYPE DRAWINGS

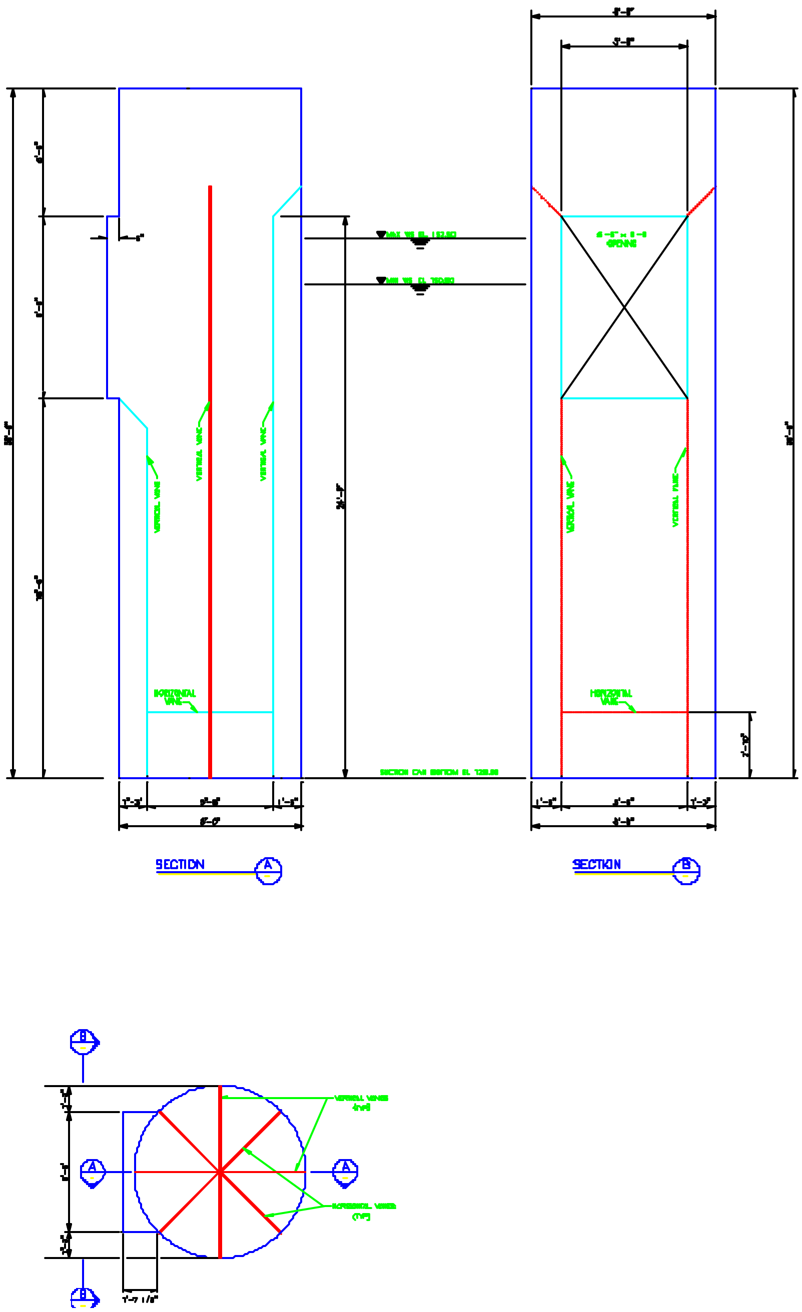


Figure 19. Original pump can design.

APPENDIX B – MESH-BLOCK DETAILS

The CFD process used various cell configurations and spatial extents to optimize computation time. While smaller cell sizes develop more precise definition of obstacles and flows, they also increase the size of the computational domain, and decrease the time step (when the explicit option is used) of the simulation. Both of these typically increase computational time required for obtaining a quasi-steady state. Balancing the accuracy of the solution with the time and computational resources available is always a challenge.

Flow-3D can use multiple mesh-blocks, which are hexahedral containing the non-uniform hexahedral control volumes. In Flow-3D V8.2, multiple mesh-blocks can be nested (one inside another) or adjacent (one mesh-block can share a side with another mesh-block).

As is typical of this type of modeling, several mesh-block techniques were used to optimize performance for the reasons discussed above. The final models used two adjacent mesh-blocks, where a sparse mesh block defined the sump and approach channel, upstream, while a high density mesh-block defined the pump can, pump piping and intake, downstream. The final spatial parameters are shown in Table 1, and the number of cells and cells spacing is shown in Table 2.

The typical X-Y composite mesh through elevation 145.7 feet is displayed in Figure 20. The typical X-Z composite mesh through the centerline of a pump can is displayed in Figure 21.

Table 1. Spatial parameters of the final models. The centerline of the pump was at X=4.5 and Y=4, and elevations directly correlate to the Z value.

	Upstream Mesh	**Downstream Mesh**
Minimum X (feet)	-16.5	0.5
Maximum X (feet)	0.5	8.5
Minimum Y (feet)	-2.0	0.0
Maximum Y (feet)	10.0	8.0
Minimum Z (feet)	144	129.0
Maximum Z (feet)	154	154

Table 2. Number of cells and typical cell sizes used in the final models.

	Upstream mesh-block		Downstream mesh-block	
	Number of cells	**Typical Cell Size (ft)**	**Number of cells**	**Typical Cell Size (ft)**
X-Direction	43	0.39535	40	0.2000
Y-Direction	31	0.38710	40	0.20000
Z-Direction	26	0.37698 ±0.06614	125	0.200305 ±0.002395
Total Cells	34,658		200,000	

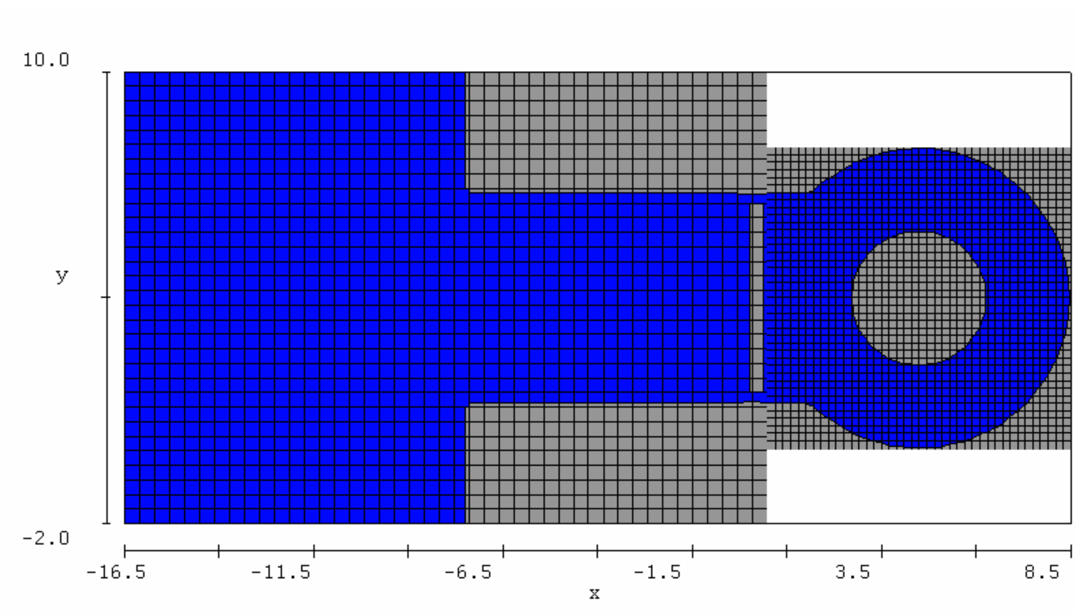


Figure 20. Typical X-Y composite mesh through elevation 145.7 feet. The cells of the upstream (left) mesh are roughly 0.4 feet on each side, with 43 cells in the X-direction and 31 cells in the Y-direction. The cells of the downstream (right) mesh are 0.20 feet on each side, with 40 cells in the X-direction and 40 cells in the Y-direction. The porous baffle that simulated a trash rack can be seen in the channel near the pump can.

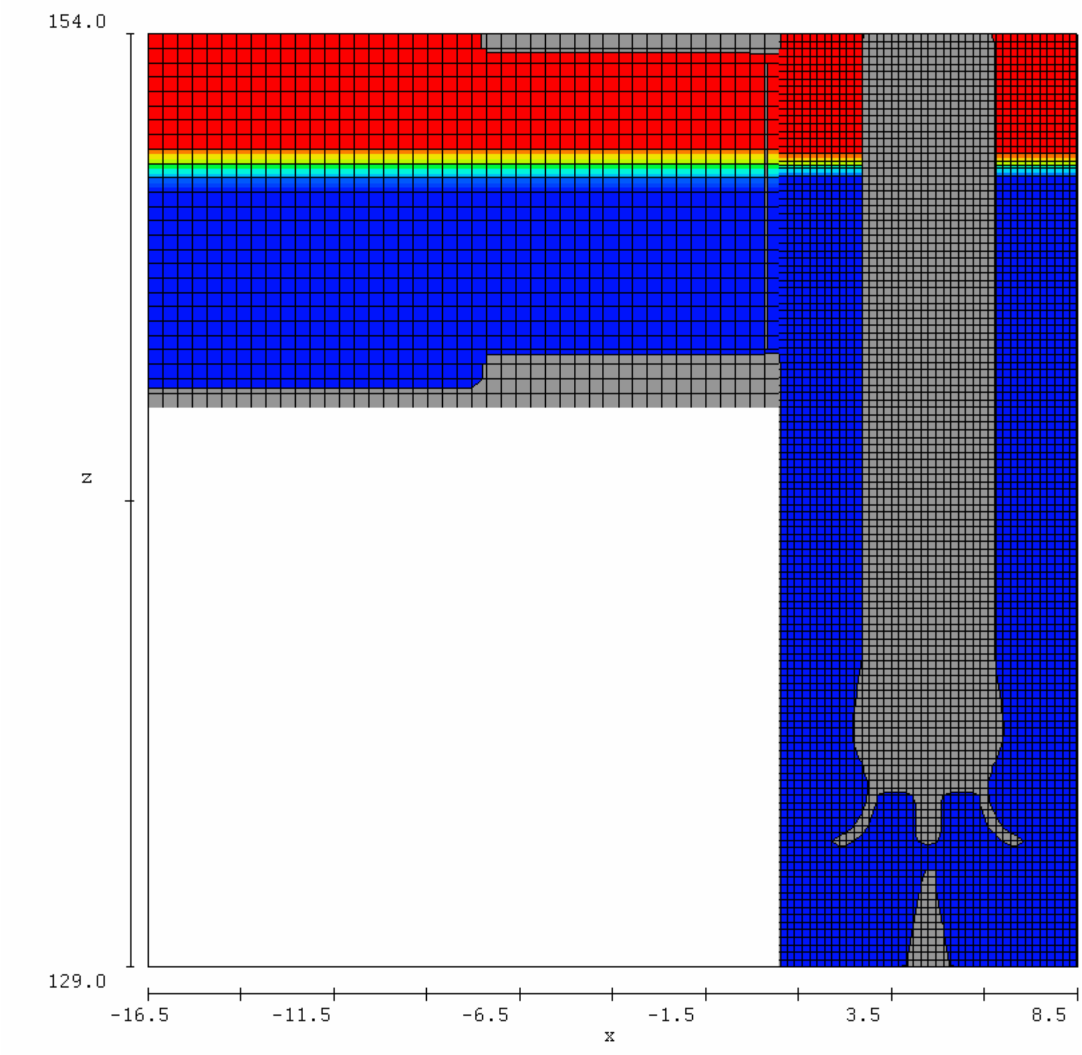


Figure 21. Typical X-Z composite mesh through the centerline of a pump can. The cells of the upstream (left) mesh are roughly 0.4 feet on each side, with 43 cells in the X-direction and 26 cells in the Z-direction. The cells of the downstream (right) mesh are roughly 0.20 feet on each side, with 40 cells in the X-direction and 125 cells in the Z-direction.