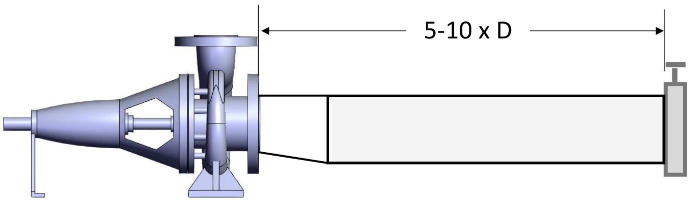
# Pump Inlet Manifold Design
# Space Constraints and Turbulence

## 1. Introduction

Pump inlet suction design is, in general, a well understood topic and standard engineering principles are usually employed. Installation manuals, textbooks, experimental publications [1-2], and 'rules-of-thumb' [3] for such designs are readily available. Although there may not be total agreement on exact values, there is general industry acceptance that certain aspects lead to good pump inlet design.




From the figure above, several principles or 'rules-of-thumb' are illustrated. A straight pipe length of 5-10 times the pipe diameter before the inlet is recommended and this should be free of obstacles including elbows, tees, and valves. The suction diameter should also be equal to, or larger than the pump inlet diameter. Flow velocities should also be kept below 2 m/s, but like many engineering problems, it can be a challenge to maintain these principles. Challenges relating to space are common and one such case is studied in this paper.

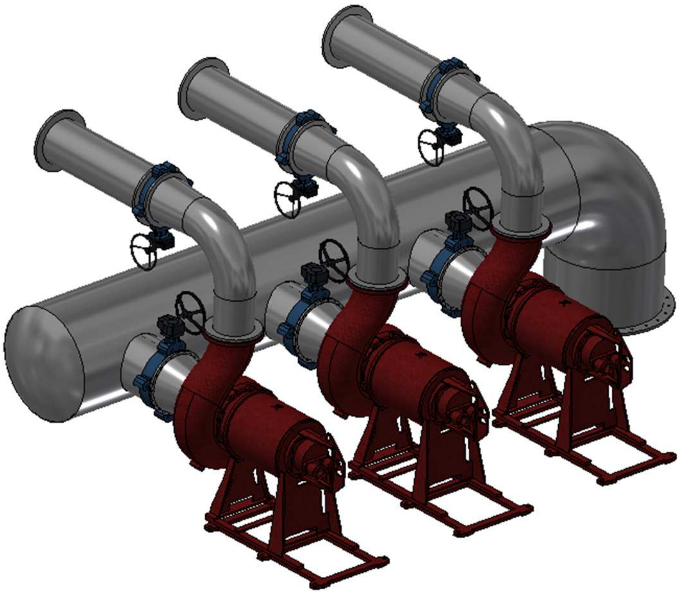
## 2. Design Standard

A standard that provides relevant guidance is ANSI/HI 9.6.6 *American National Standard for Rotodynamic Pumps for Pump Piping* [4]. From this standard the following requirements are noted:

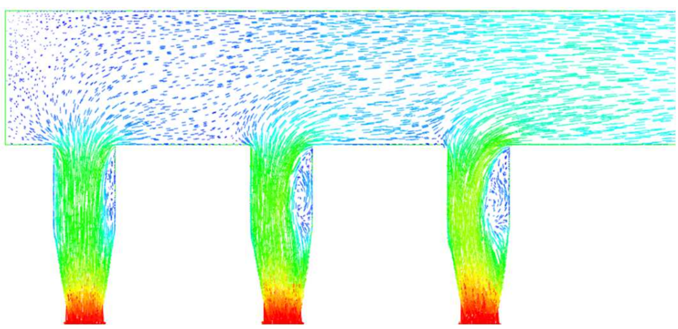
- Suction pipe to be as large as the pump suction nozzle, and valves should be one size larger .
- Maximum velocity at any point in the inlet piping is to be 2.4 m/s. Values above this should be evaluated for flow distribution, NPSH, noise, etc.
- Avoid swirl by having a pipe straight of appropriate length or  flow straightening device installed.
- Poor velocity distribution at the entrance to the pump can be evaluated using CFD.

## 3. Problem Definition

An irrigation facility, with increased water supply demands, led to a situation where a series of larger pumps were to be set up with inadequate inlet conditions due to the space constraints of an existing pump house. The proposed piping arrangement is shown in the figure below. The space restrictions resulted in the inlet pipes simply being a series of eccentric reducers after the valves and immediately off the supply line. Sharp corners also existed at the connection of the pump approaches with the main supply line, which suggested the formation of vortices downstream of these locations.

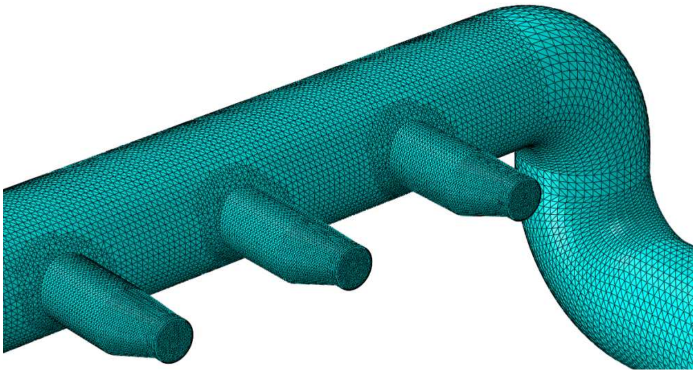


To confirm the inadequacy of the design, a Computational Fluid Dynamic (CFD) study was performed on the inlet portion of the system. The applied high flow caused large vortices to form and their proximity to the pumps was potential for catastrophic pump failure.



## 4. Flow Domain Modelling

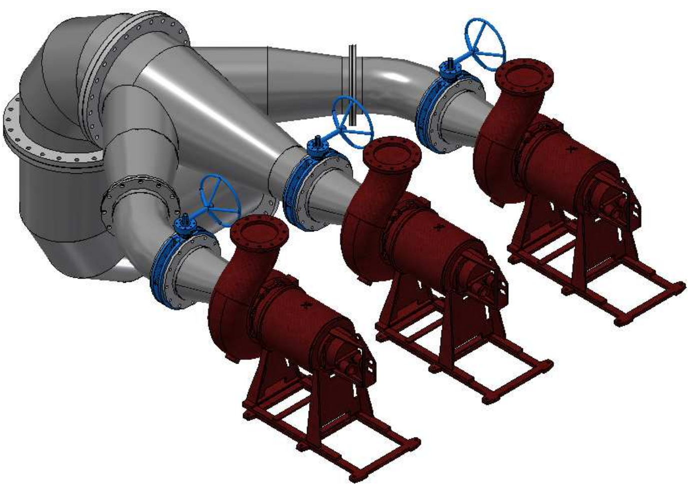
Abaqus CFD was used to model the steady-state flow conditions at the pump inlet. A one-equation Spalart-Allmaras turbulence model was used with fine mesh inflation to capture the sharp velocity gradients at the pipe walls.



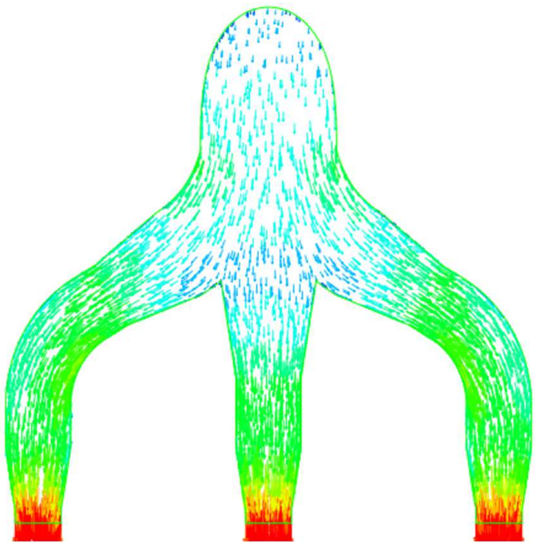
In boundary-layer flow, at high flow rates, large velocity gradients exist near the walls due to no-slip boundary conditions and cannot be avoided. These are often very thin regions and the size of dangerous vortex structures containing reverse flow are insignificant compared to the ones presented earlier. Adverse consequences arise when sharp bends or corners are inserted into the flow path as they lift off this turbulent layer from the boundary and transfer that to the freestream. The increased turbulent behaviour throughout the freestream raises the effective viscosity of the fluid leading to reduced efficiency, increased component wear, and overall poor performance.

## 5. Final Design

Based on the CFD analysis it was important to avoid all sharp corners and make any changes in flow direction as gradual as possible. Any changes near the pump inlets resulted in large vortices lingering near the entrances and any changes further upstream caused the turbulent boundary layers to mix with the freestream and worsen the quality of the inlet flow. After a number of iterations, a design was developed that divided the main supply flow, was free of large vortices, and minimised the transport of turbulence .



In assessing quality, the flow was first checked for large vortices which could damage the nearby pumps or valves. The vector velocity plots provided an indication of slow-moving recirculation zones and localised regions of acceleration causing turbulence. Streamlines, as shown above, were examined



for possible swirling and were also be used to track the transport of turbulence or particle matter. Streamlines can also be integrated to provide estimations of residence time or quality of mixing if desired. Another important check made of the CFD results was the inspection of pressures which can cause cavitation acting on the pipe surfaces. In the end, a non-standard design rationalised by engineering principles was developed and verified through CFD.

## 6. References

[1] Stephen, Christopher, et al. "Numerical flow prediction in inlet pipe of vertical inline pump." *Journal of Fluids Engineering* 140.5 (2018).

[2] Wilcox, David C. Turbulence modelling for CFD. Vol. 2. La Canada, CA: DCW industries, 1998.

[3] Mobley, R. Keith. *Plant engineer's handbook*. Elsevier, 2001.

[4] Hydraulics Institute 2009, *Rotodynamic Pumps for Pump Piping* (ANSI/HI 9.6.6), Hydraulic Institute, Parsippany.