CFD MODELLING TO INFORM PUMPSTATION DESIGN

Mike Summerhays and Wageed Kamish

Civil Infrastructure Group, AECOM New Zealand Limited

ABSTRACT

Computational Fluid Dynamics (CFD) is the science of predicting momentum mass and heat transport by numerically solving a set of fundamental governing equations in one, two and three dimensions. Because of the computational intensity associated with CFD simulations, application of the code has historically been limited to the academic environment. However, over the past two decades, the cost of computing power has decreased substantially while the processing speed has increased exponentially. These developments have now made the application of CFD in the commercial environment feasible. Many commercial software models are available for CFD modelling. These include, but are not limited to Fluent, CFX and STAR-CD and have different degrees of sophistication in terms of graphical user interfaces and numerical solution techniques.

In this study the commercial CFD code, Fluent, was applied in the design of three pump stations. Pump stations are usually designed using the heuristic method, but when the application of these methods becomes vague it may be necessary to employ CFD to reduce uncertainty in predicting performance efficiency. Typical heuristic design limits may include prediction of flow patterns on approach to the pump intakes, the minimum drawdown levels for safe operation, extent of air entrainment and the formation of potential dead spaces. Simulated outputs from the case studies showed that the CFD code provided a functional estimation of free surface and submerged vortex formation, excessive pre-swirl of flow entering the pump, non-uniform distribution of velocity on the impeller eye and entrainment of air bubbles which was then used during the design process to reduce operating costs and risk as well as improve efficiency.

KEYWORDS

Computational fluid dynamics, pump station, air entrainment

PRESENTER PROFILE

Mike Summerhays (BE, MIPENZ, CPEng, IntPE)

Mike Summerhays is a Technical Director at AECOM in New Zealand. Mike has over 40 years of experience working for central / local authorities and as a consulting engineer. He has been involved with a wide variety of projects including stormwater catchment planning, hydrological and hydraulic modelling, stormwater quality improvements and remedial options investigations.

Wageed Kamish (BSc Chem. Eng., MSc Civ. Eng., Pr. Eng.)

Wageed is a senior consultant at AECOM in New Zealand and has 19 years of experience in performing and managing water quality assessments, hydrodynamic and water quality modelling studies as well as hydrological modelling studies. More recently he has been involved in cleaner production projects as well as Computational Fluid Dynamics (CFD) modelling in support of detailed infrastructure designs.

FUNDAMENTALS OF CFD

1.1 HOW DOES CFD WORK

In all cases four steps are followed in configuring a CFD model for pump stations. These would include definition of the problem, geometry and mesh construction of the domain, specification of boundary conditions and physical models and then the solving of equations.

In principle CFD models solve the general conservation (transport) equation for mass, momentum, energy (Bird et. al. (1960)), by discretising the physical domain into finite volumes. Mathematically, the resulting partial differential equations are discretised into algebraic equations which are solved numerically to yield a solution field.

1.2 WHERE CAN IT BE APPLIED

Since CFD is a mathematical representation of reality and is performed entirely on a desktop basis, it is particularly useful in the:

- Conceptual studies of new designs
- Detailed product development
- Troubleshooting of existing designs and
- Redesigns

Traditionally and historically CFD has been applied in the automotive, chemical and aeronautical industries, but has recently been applied to flow-related problems in the civil engineering field.

1.3 ADVANTAGES OF CFD

The following advantages of CFD have been identified:

Low cost of implementation: The use of physical experiments and tests to obtain essential engineering data for design could be expensive. By comparison, computational simulations are relatively inexpensive and are not susceptible to equipment downtime during testing.

Speed: CFD simulations can be executed in a short period of time, ensuring quick turnaround and introducing engineering "data" into the design process at an early stage.

Simulate Real Conditions: Many flow and heat transfer processes cannot be easily tested with physical models. In these cases CFD provides the ability to theoretically simulate any physical condition.

Simulate Ideal Conditions: CFD allows greater control over the physical process and provides the ability to isolate specific phenomena for study. It also provides comprehensive information about the flow field in contrast to experiments which only permit data to be extracted at a limited number of locations in the system where gauges have been installed, e.g. pressure and temperature probes, heat flux gauges, etc. Additionally, numerical and graphical outputs are available.

1.4 LIMITATIONS OF CFD

The following limitations of CFD have been identified:

Physical Models: CFD solutions rely upon physical models of real world processes (e.g. turbulence, compressibility, chemistry, multiphase flow, etc.). As a result, the solutions that are obtained through CFD are only accurate as the physical models on which they are based.

Numerical Errors: Solving equations on a computer invariably introduces numerical errors and round-off errors will always exist, though they will be small in most cases. However, Round-off errors will tend to zero as the grid is refined.

2 APPLICATION 1 – MINIMUM WATER LEVEL FOR SAFE LEVEL OF OPERATION

2.1 Model preparation

The purpose of this study was to simulate the flow patterns and predicted performance of a water pump station which consisted of a number of chambers. Two flow configurations were imposed, simulating the use of either one or two of the extraction pumps. The investigation sought to examine the possible entrainment of air from the free surface due to vortex formation in the fluid at the free-surface level. The pump station layout is depicted in Figure 1.

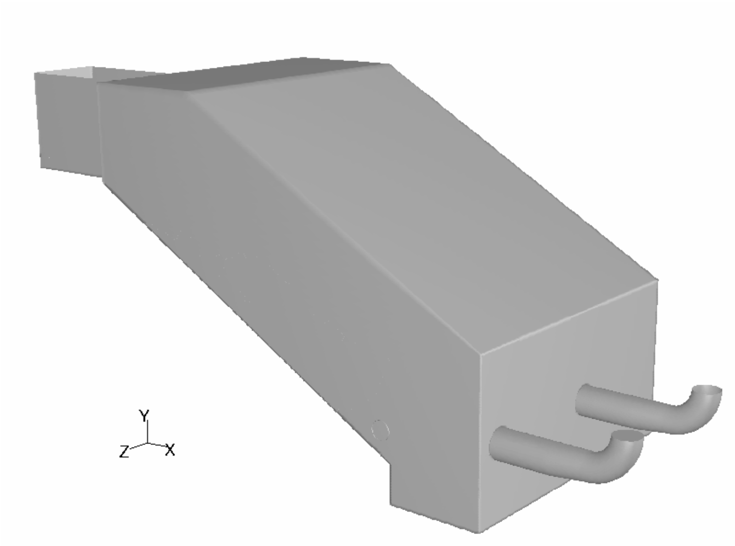

Figure 1: Geometry of chamber 1

The geometry was recreated in Fluent's pre-processor *Gambit* from existing layout drawings. The inlet duct was extended to reduce the effect of uniform boundary conditions and a large section of the chamber above the water was removed to reduce the model size, thereby reducing the time required for solution.

Figure 2 shows the location of the inlet and outlet boundaries to the chamber 1 domain. The blue face represents the outlets where a pressure outlet boundary condition was imposed. Each outlet has been named separately so that varying outlet configurations may be considered. The red face represents the velocity inlet boundary for the inflow of water. This boundary has a horizontal split that corresponds in height to the expected level of the free surface for a low inlet flow condition.

For the lower flow rate, only the lower section is used, while the upper section is blocked off. The green face represents an outflow (pressure outlet) boundary that allows for the displacement of air from the domain as water is introduced during the simulation. All the grey surfaces are taken as no-slip (i.e. zero velocity) walls. Figure 3 illustrates the formation of the mesh on the walls of the chamber. Predominantly hexahedral cells were utilised, however some tetrahedral cells were necessary to capture the geometry of the outlet pipes that protrude into the chamber. The geometry used approximately 700 000 hexahedral cells and 300 000 tetrahedral cells. More cells were used in areas of interest and some grid adaption (refinement) was performed during the simulation to improve the solution in areas with high gradients.

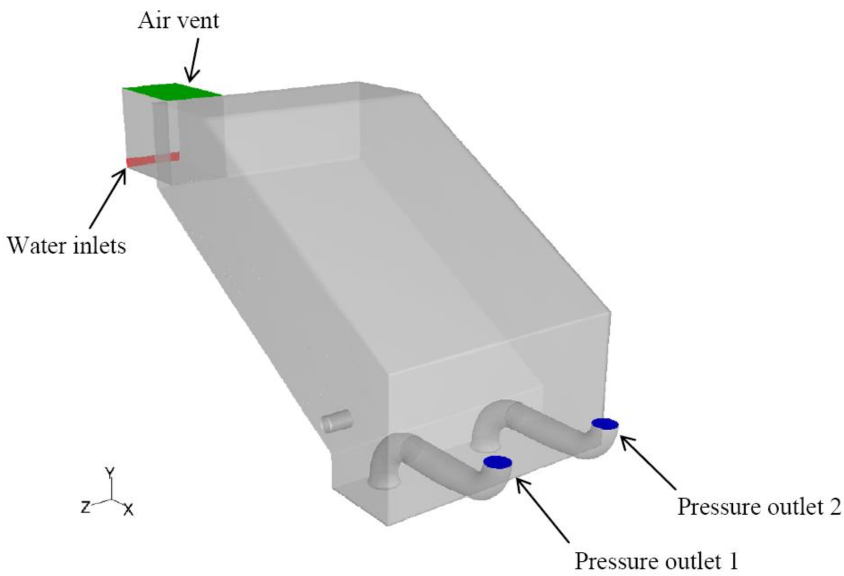

Figure 2: Boundary conditions for chamber 1

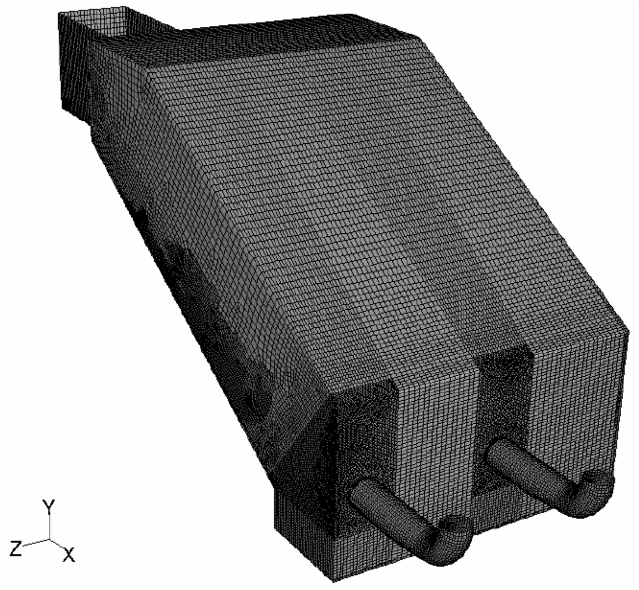

Figure 3: Domain mesh for chamber 1

Table 2: Input parameters for pump station model (application 2) shows the input parameters considered for the model.

Table 1: Input parameters

Input Parameter Description	Value
Density of water (kg/m^3)	998.2
Density of air (kg/m^3)	1.225
Water viscosity (kg/m.s)	0.001
Air viscosity (kg/m.s)	1.789e-5
Feed water flowrate [Low] (m^3/hr)	1,500
Feed water flowrate [High] (m^3/hr)	2,300

To allow for the tracking of the free surface, a multiphase model was used to weight the properties of the fluid in each cell according the volume fraction within the cell. This method is called the *Volume of Fluid* (VOF) method and is combined with the *Geo-Reconstruct* free-surface tracking scheme that provides a sharp well defined interface between the two phases.

The simulation made use of Wilcox's standard k-ω two equation turbulence model within an unsteady 3D solver framework. During the simulation water was introduced into the domain from the inlet and travelled down the inclined concrete surface to join with water already standing in the base of the chamber at the specified level. The pressure outlet then allowed the fluid to pass out of the domain, maintaining the set free-surface level. Once flow velocities behaved cyclically, the solver was terminated and the solution was regarded as converged. Solution times were in the order of two weeks on a single core processor (3.2 GHz).

2.2 SIMULATION OUTPUTS

Figure 4 below shows contours of static pressure on the walls of the chamber. The contours show the hydrostatic build-up of pressure in the negative-Y direction and suggest little fluctuation in pressure due to the flow.

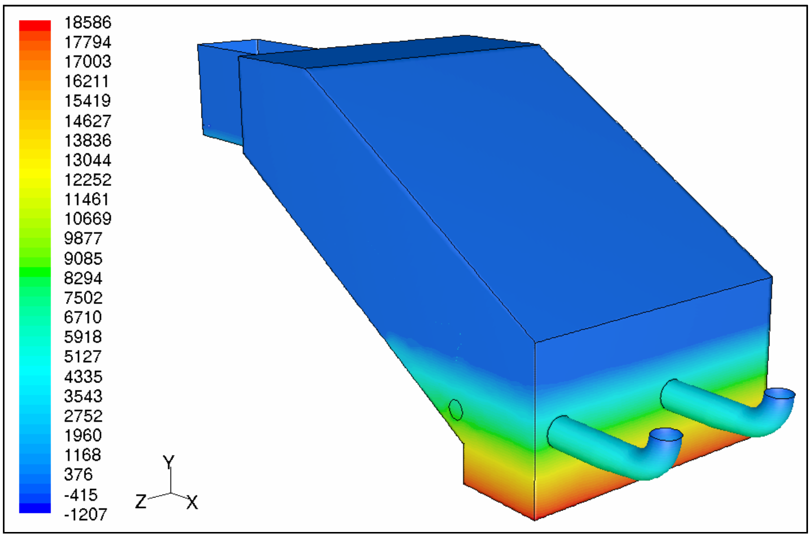

Figure 4: Contours of static pressure in chamber 1

Figure 5 shows contours of the volume fraction of water on the walls of chamber 1 for a flow rate of 2300m^3/hr. The figure shows some entrainment of air bubbles into the reservoir area where the incoming water first meets the standing water.

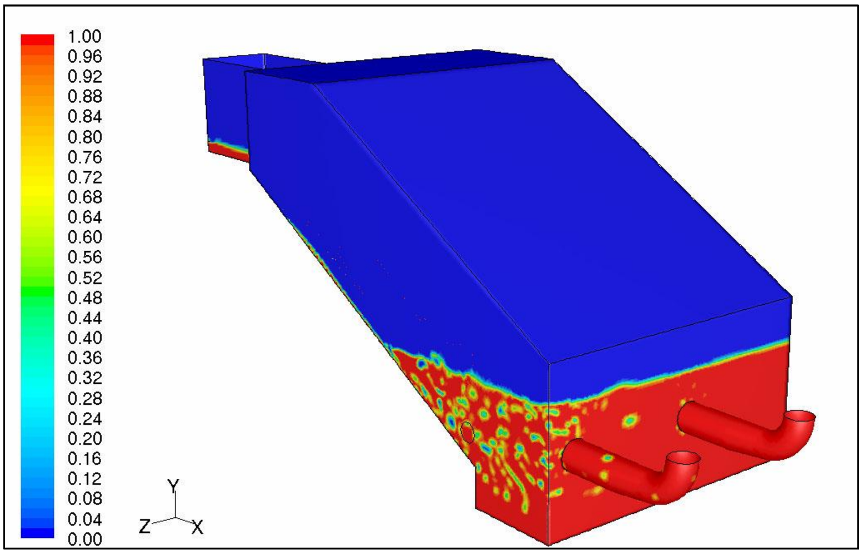

Figure 5: Volume fraction of water for chamber 1

Figure 6 below illustrates contours of velocity magnitude on the free surface for chamber 1. This suggests that the free surface is still relatively planar, without evidence of the formation of the free surface vortex indicating a safe operating condition for the chosen water depth.

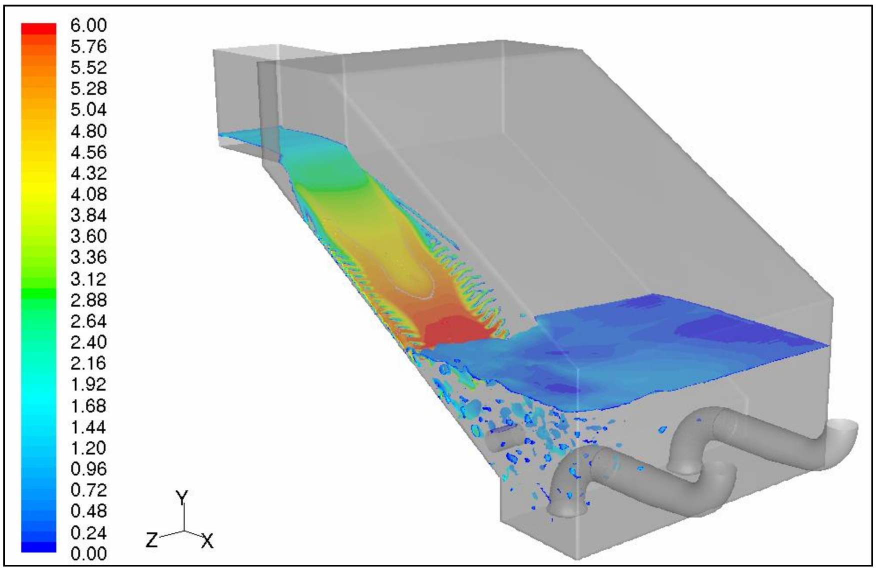

Figure 6: Contours of velocity magnitude (m/s) on the free surface in chamber 1

Figure 7 illustrates path lines coloured by velocity magnitude for chamber 1. A path line represents the path a weightless particle would follow, should it be introduced into the flow. The results indicate that although there is a large scale re-circulation of the water in the reservoir, vortices in the region of the outlets to the reservoir are weak. Consequently the low pressure core of the vortex is too weak to entrain any air from the free surface.

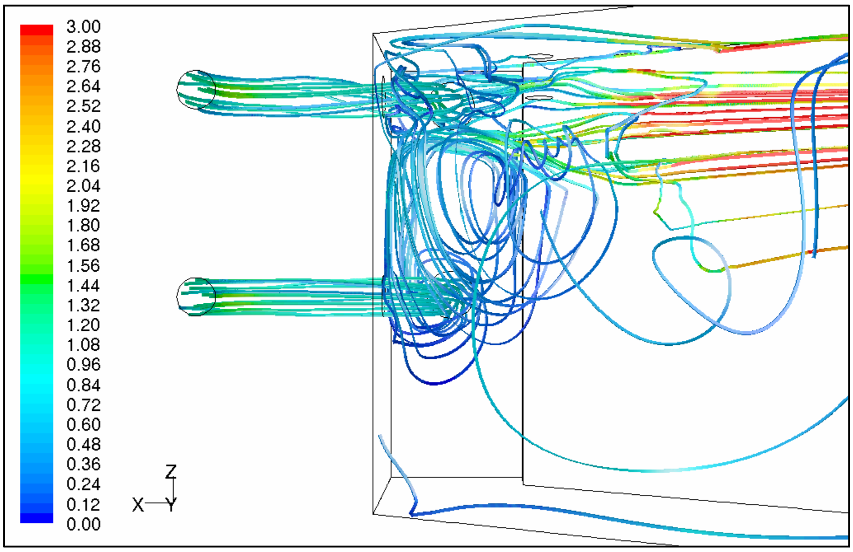

Figure 7: Path lines coloured by velocity magnitude in chamber 1

2.3 CONCLUSIONS

Based on the simulation outputs it is clear that the chosen fluid height, corresponding to the height of the pump axis, represents a safe working point for the pump station. Vortices formed in the region of the outlets of the reservoir were weak and it is anticipated that no air entrainment would occur. Other fluid levels may easily be examined using the same geometry to find the point at which air entrainment will eventually start occurring.

3 APPLICATION 2 – GEOMETRY EFFECTS ON INTAKE VELOCITY PATTERNS

A specific aim of the study was to determine the flow path lines and whether or not any significant air entrainment would occur at the minimum operating level (134 MSL) in the pump station. The air entrainment concern was particularly relevant, considering that a fine screen would be located close to the pump station inlet and the resultant headloss could possibly allow for the water level downstream of the screen to drop closer to the bottom of the pump station.

Specific concerns to be investigated included:

- Possible vortex formation and air entrainment at the minimum operating level taking into account the effect of the fine screen,
- Possible establishment of non-uniform velocity profiles on the approach to the pump intake,
- Minimum water level for "safe" operation.

The aforementioned concerns were investigated for two operating water levels in the pump station viz. 134 MSL and 135 MSL. Figure 8 shows a representation of the pump station.

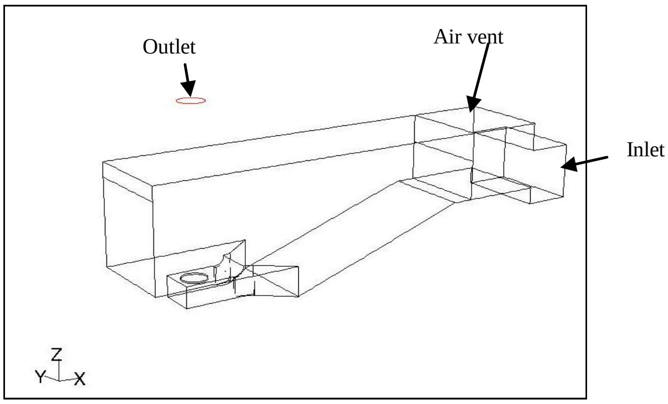

Figure 8: Pump station layout

3.1 MODEL PREPARATION

The geometry was created in AutoCad and imported into Fluent's pre-processor, Gambit. From Figure 8 it can be seen that the inlet and outlet conduits were extended in the model to reduce the effect of the imposed uniform boundary conditions.

The boundary conditions imposed at the required inlets and outlets were specified as "Velocity Inlets" and "Pressure Outlets" respectively. "Velocity Inlets" comprised the magnitude as well as the direction of flow across the inlet boundary while the "Pressure Outlets" were specified as the magnitude of the difference in height between the outlet and the water level in the pump station. For this configuration, a constant negative pressure of suitable magnitude to maintain the water level at the desired height was induced at the outlet, representing the suction head of the pump. The geometry consisted of approximately 880,000 cells.

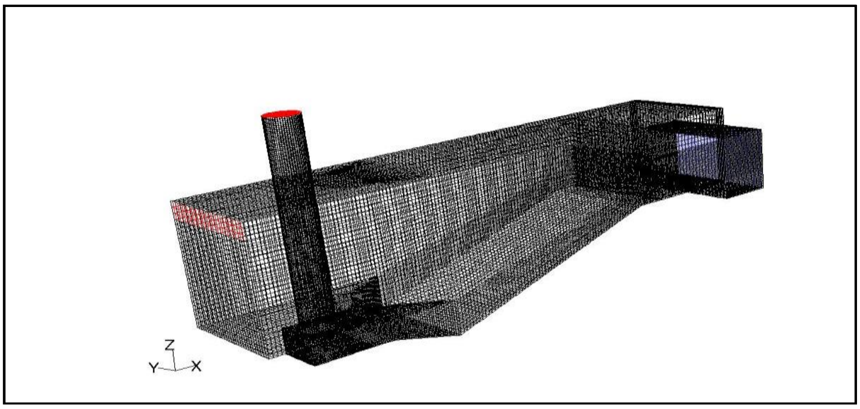

Figure 9: Formation of mesh on pump station walls

All walls were treated as "no-slip", that is zero velocity at the wall, boundary conditions while a two phase model, air / water, was used to describe the air / water interaction. Table 2: Input parameters for pump station model (application 2) indicates the input properties as well as feed-water flow rate for the fluids used in the model.

Table 2: Input parameters for pump station model (application 2)

Input Parameter Description	Value
Density of water (kg/m^3)	998.2
Density of air (kg/m^3)	1.225
Water viscosity (kg/m.s)	0.001
Air viscosity (kg/m.s)	1.789e-5
Feed-water flow rate (m^3/hr)	1,300

To allow for tracking of the free surface, a multi-phase model was used to weight the properties of the fluid in each cell according to the volume fraction. The VOF method and was combined with the *Geo-Reconstruct* free-surface tracking scheme that provides a sharp, well defined interface between the two phases. Pressure drop across the fine screen was modelled as a headloss, or a porous jump in Fluent.

The simulations made use of the standard k-ε turbulence model within an unsteady 3D solver framework and were approached in such a way that water from the inlet joins stagnant water within the main chamber, which is already at the specified operating level. The pressure outlet then allowed for the flow of water out of the domain, maintaining the set free surface level (representing the suction head on the pump). Once the flow velocities behaved cyclically the solver was terminated and the solution was regarded as converged.

3.2 SIMULATION OUTPUTS

Preliminary design information for the fine screen indicted that the spacing between the bars would be 25mm, with the width of the bars being 16mm. Based on this information it was decided, in consultation with the User Support of FLUENT in South Africa, that a relationship describing the pressure drop-velocity relationship should be used instead of modelling the screen as a geometric obstruction.

The headloss across the screen was calculated from Equation (1) assuming that the screen arrangement is as depicted in Figure .

$$\mathbf{h = B*(b/d)^{(4/3)}*v^2/2g*sin\ a}$$ (Koot, 1974) **(1)**

where,

$v^2/2g$	=	velocity head before the screen
d	=	distance between bars
b	=	width of bars
a	=	angle of Screen to Horizontal
B	=	depends on the form of the bar (2.42 for rectangular bars and 1.79 for round bars)

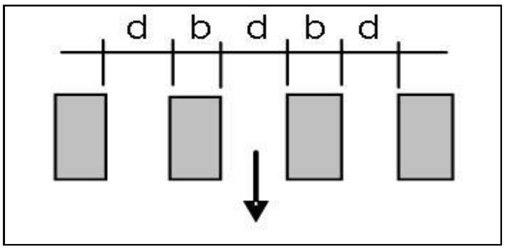

Figure 10: Arrangements of bars used in fine screen

For the lowest drawdown level (134 MSL), Figure 11 and Figure 12 shows the inlet flow patterns to the pump station. The illustrations show that the free surface is still relatively planar without any evidence that suggests the formation of free surface vortices. It should be noted that the average water level in the basin downstream of the screen is located at 134 MSL, corresponding to the proposed minimum drawdown level while just upstream of the screen the water level is slightly higher.

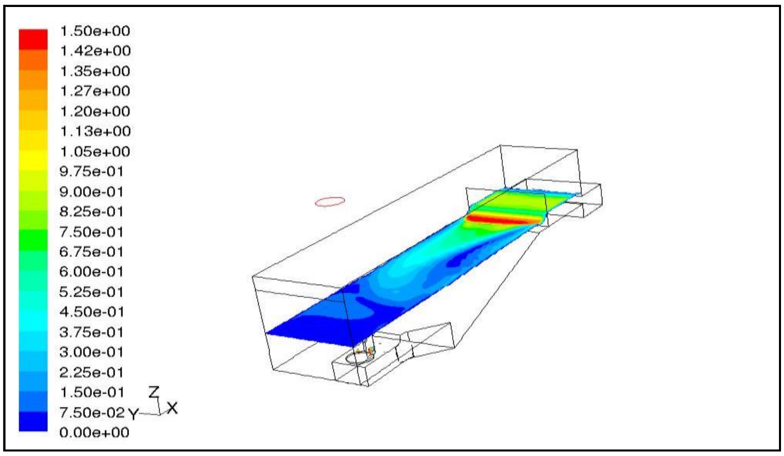

Figure 11: Contours of velocity vectors on the free surface (m/s)

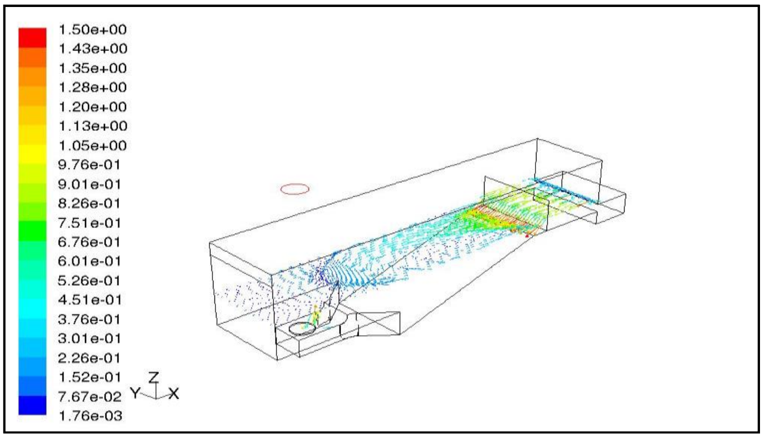

Figure 12: Velocity vectors on free surface - coloured by velocity magnitude (m/s)

It was noticed that some entrainment of air bubbles occurred immediately downstream of the screen. To quantify this a custom field function for calculating the average flow rate of air it was defined by multiplying the velocity in the outlet of the screen with the volume fraction of air and then monitoring was undertaken on the integral of the custom field function on the outlet. Based on this approach it was possible to quantify that the average volumetric flow of air at the outlet would be 0.001 m^3/s and would constitute the additional air volume added immediately downstream of the fine screen.

Figure 13 shows a relatively uniform water velocity on approach to the pump with no evidence of recirculation in the region.

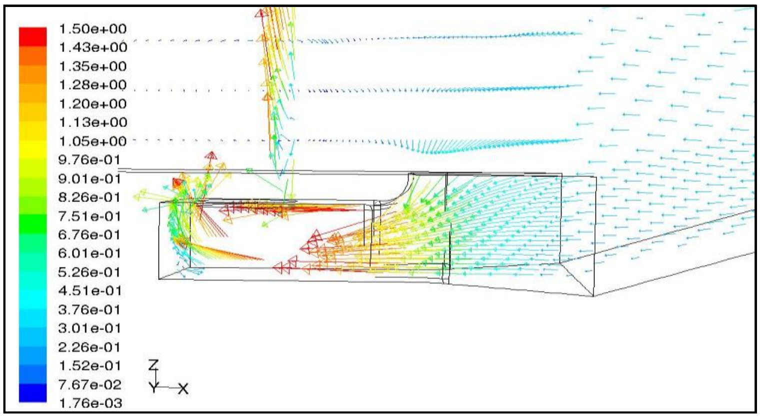

Figure 13 Velocity vectors at pump inlet - coloured by velocity magnitude (m/s)

Figure 14 illustrates the path lines in the region of the pump inlet and confirms that no recirculation should occur on approach to the pump intake.

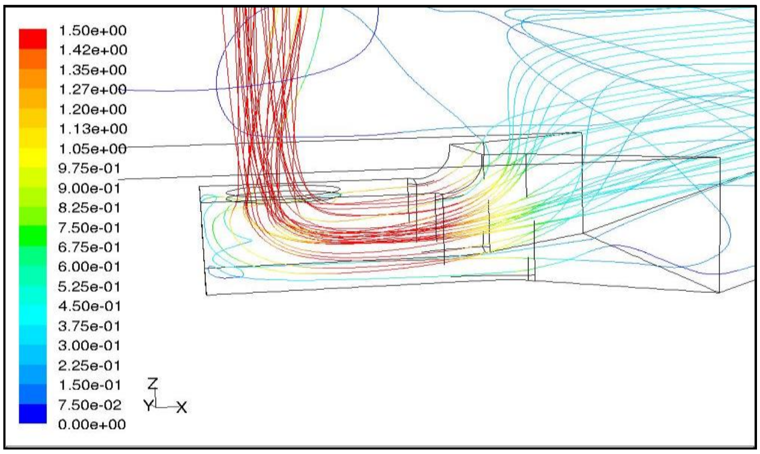

Figure 14 Path lines at pump inlet - coloured by velocity magnitude (m/s)

3.3 CONCLUSIONS

Based on the solutions obtained for the various scenario runs, it was concluded that the minimum operating level of 134 MSL downstream of the fine screen represents a "safe" operating level, which is not ideal, for the pump station with an average additional volumetric air flow rate of only 0.001 m^3/s being introduced. Taking into account that water would naturally contain 1%-3% air, the amount added due to entrainment would constitute an increase of less than 1% from the natural conditions. It was nevertheless recommended that the lowest operating level be checked to verify that the model prediction in terms of air entrainment was in fact valid.

It was also further concluded that the approach velocities to the pump intake was uniform and that no imbalance of load would be imposed on the pump.

4 APPLICATION 3 – REFINEMENT OF GEOMETRIC DESIGN

The overall objective of this application was to iteratively test the conceptual design of a sewage pump station before declaring the design final. The criteria for assessing the "goodness-of-design" were as follows:

- Occurrence of dead spaces,
- Uniformity of approach velocities at the pump intake.

Based the above criteria, changes were made to the geometry and the model re-run to test the efficiency of the changes. Using this approach it was possible to identify potential problem areas in the pump station design before it was built and commissioned.

4.1 INITIAL DESIGN

The layout of the initial design consisted of a single main chamber, fed from a stilling chamber. The connection between the two chambers was achieved by two rectangular conduits. The layout of the initial design is shown in Figure 15.

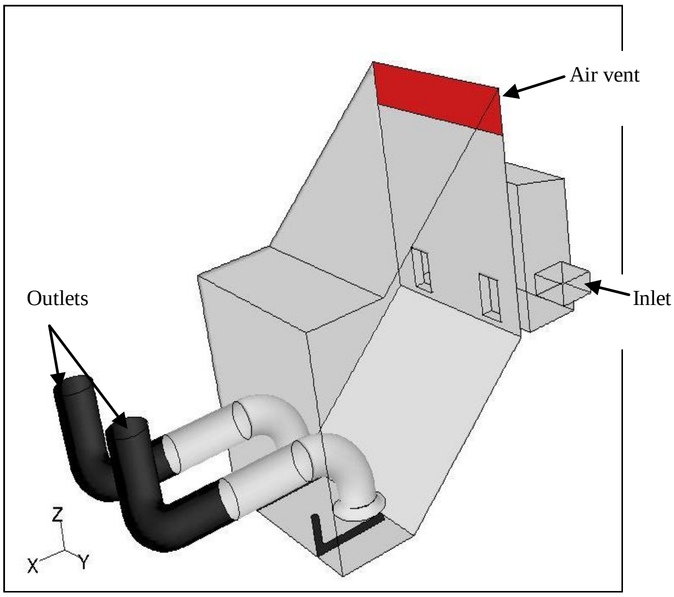

Figure 15: Geometry of initial pump station design

The boundary conditions imposed at the required inlets and outlets were specified as "Velocity Inlets" and "Pressure Outlets" respectively. "Velocity Inlets" comprised the magnitude as well as the direction of flow across the inlet boundary while the "Pressure Outlets" were specified as the magnitude of the difference in height between the outlet and the water level in the pump station. For this configuration the pressure outlets were located at the desired final water level with no need to induce a suction effect.

A significant portion of the geometry above the water level, occupied by air, was intentionally omitted to reduce the computational load on the model. The input parameters for the simulation and all other iterations of the geometry are shown in Table .

Table 3: Input parameters for pump station

Input Parameter Description	Value
Density of water (kg/m^3)	998.2
Density of air (kg/m^3)	1.225
Water viscosity (kg/m.s)	0.001
Air viscosity (kg/m.s)	1.789e-5
Feed-water flow rate (m^3/s)	900

4.1.1 SIMULATION OUTPUTS - INITIAL DESIGN

The velocity distributions on the free surface (see Figure 16) indicate a significantly large area of zero flow. This has the potential to result in settling of solid materials. It also indicates that the free surface is relatively planar with no evident vortex formation.

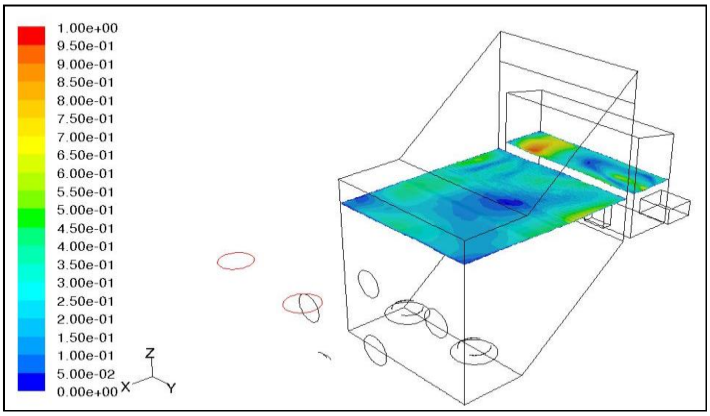

Figure 16: Velocity contours on the free surface (initial design)

The velocity contours at the pump intakes are shown in Figure 17**.** The figure indicates a uniform profile across each of the intakes. There is, however, a persistent area with zero flows which could lead the settling of solids.

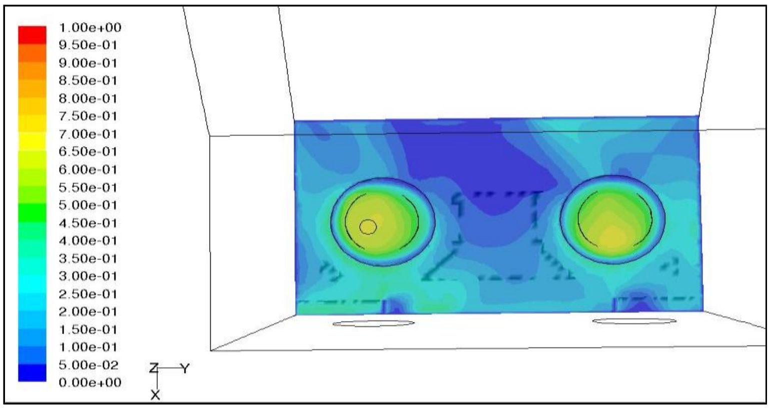

Figure 17: Velocity contours at the pump intakes (initial design)

The path lines of particles released from the inlet of the pump station are shown in Figure 18.

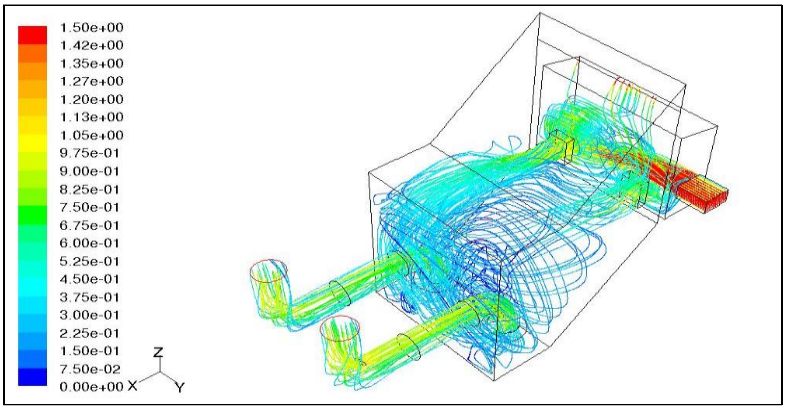

Figure 18: Path lines coloured by velocity magnitude (m/s) (initial design)

The path lines indicate the preferred path of water from the stilling chamber to the main chamber is via the conduit situated furthest from the inlet. This results in large scale recirculation in certain parts of the main chambers, especially in the region of the pump intakes, which was considered undesirable. Analysis of the potential flow patterns to be expected in the pump station prompted a redesign which will be discussed in the following section.

4.2 MODIFIED DESIGN

In recognition of the concerns raised during the flow simulation of the initial design the following changes were made to the pump station geometry:

- The main chamber was split into two to prevent the large scale recirculation. This was an attempt to increase the uniformity of flow on approach to the suction pipes,
- The baffles at the pump intake were raised to ensure uniform velocity distributions,
- The bottom corners of the main chambers were benched to reduce the probability of solids accumulation in those areas.

The modified layout of the pump station is shown in Figure 19. All input parameters were as before.

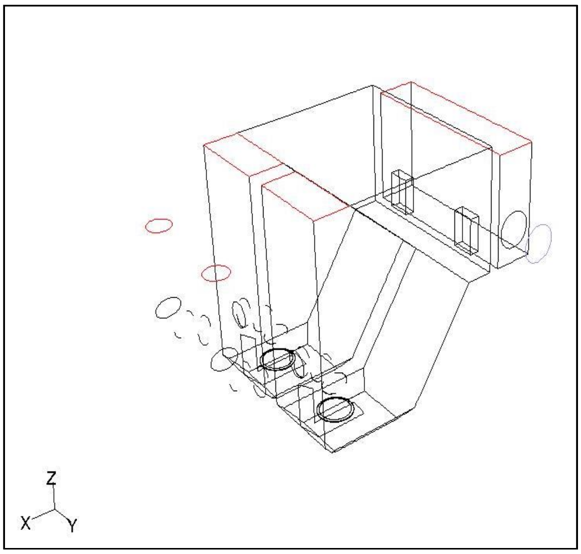

Figure 19 Geometry of modified pump station design

4.2.1 SIMULATION OUTPUTS MODIFIED DESIGN

The velocity distribution on the free surface of each of the main chambers is shown in Figure 20. Some areas of zero velocity are still visible and the water surface is still planar with no vortex formation.

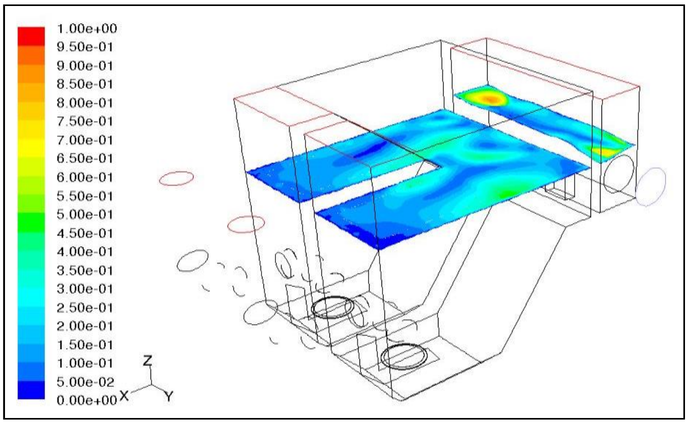

Figure 20: Velocity contours on the free surface of the main chamber (modified design)

The path lines for the modified designs are shown in Figure 21 and Figure 22.

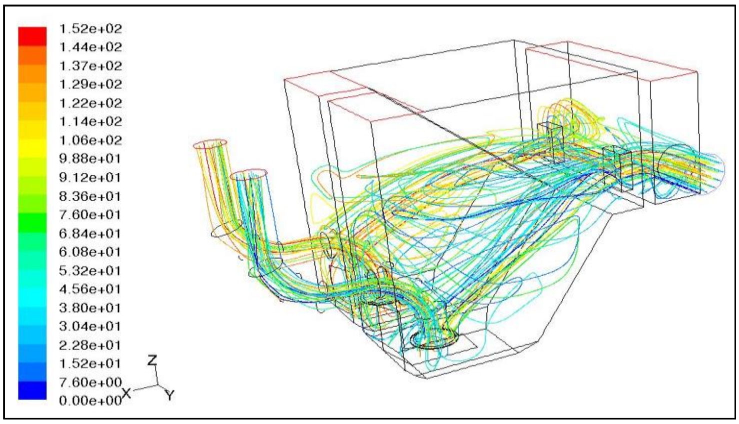

Figure 21 Path lines coloured by velocity magnitude (m/s) - modified design - profile

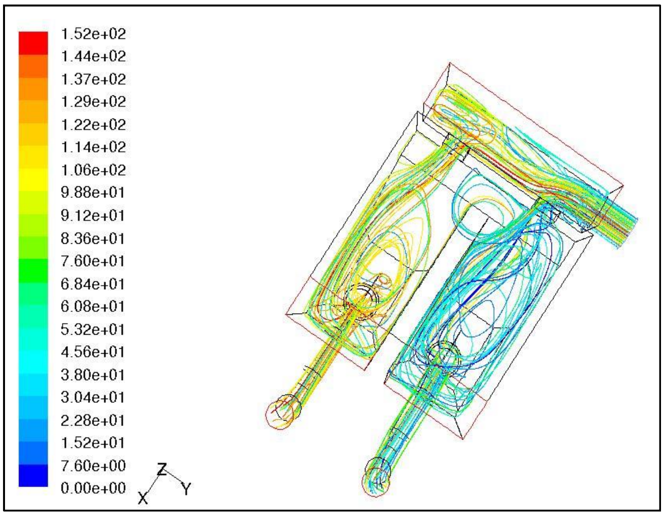

Figure 22 Path lines coloured by velocity magnitude (m/s) - modified design - plan

Although the path lines still indicate large scale recirculation, no vortices strong enough to entrain water were formed and the recirculation patterns were considered more desirable than those of the initial design.

The contours of velocities at the pump intakes are shown in Figure 23 which indicates that the velocities, on approach to the pump intake, are fairly uniform. It also shows that "dead spaces" still exist and that approach to the intake in each chamber is predominantly from the side furthest away from the inlet.

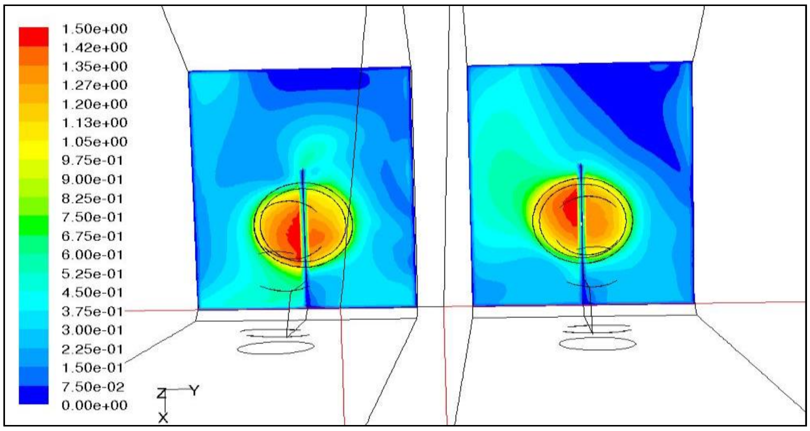

Figure 23 Velocity contours at the pump intakes (modified design)

5 CONCLUSIONS

Based on CFD applications presented in this paper the following conclusions can be drawn:

- CFD is an effective tool for evaluating the concerns relating to the designs of new pump stations because the overall flow patterns can be assessed before final designs are completed.
- Qualitative criteria for assessing pump station design such as:
  - Formation of free surface and submerged vortices,
  - Excessive pre-swirl of flow entering the pump and its variations with time,
  - Non-uniform distribution of velocity at the impeller eye and excessive variations in velocity with time,
  - Entrained air or gas bubbles, can be quantified by calculating the values of dimensionless variables (e.g. Froude or Reynolds numbers) to provide indications of the "goodness-of-design"
- Visual displays of flow patterns, pressure distributions and path lines allow for the assessment of design by visual inspection. Combined with the quantified dimensionless variables, design uncertainties can be eliminated at an early stage.

ACKNOWLEDGEMENTS

The authors would like to thank the Trans Caledon Tunnel Authority (TCTA) for funding of this work.

REFERENCES

ANSYS. 2009. http://www.fluent.com/software/fluent/index.htm

ANSYS. 2009. http://www.ansys.com/Products/cfx.asp

Bird, R.B., Stewart, W.E. and Lightfoot, E.W. 1960. *Transport Phenomena*. John Wiley & Sons, Inc.

CD-adapco. 2009. http://www.cd-adapco.com/

Koot, A.C.J. 1974. *Behandeling van Afvalwater.* Uitgeverij Waltman - Delft