CFD ANALYSIS OF A NEUTRALIZATION PIT SYSTEM

Work Executed

By

M/s Khodiyar CAD center

AHMEDABD, GUJARAT, INDIA

Table of Contents

1. Introduction

1.1 Introduction & Scope

Waste Water Treatment and Water Recycling Plants have become ubiquitous owing to their dearth of the fresh potable water resources and water contamination based on increased industrialization around the world. Growth in demand for recycled water is supported by high-tech factories from machinery, telecommunications and automotive industries housed in the concession area.

The scope of this document is to Compute the mixing Dynamics of the moving Fluids like “waste water” kept in compartment A (Size of 4.7m x 6m x 3.5m), with Sodium Hydroxide and Hydrochloric Acid kept in Compartment B(Size of 15m x 2.85m x 3m)and Compartment C (Size of 15m x 2.85m x 3m). Tank A is connected to tanks B and C using two penstocks of the size 0.6mx0.6mx0.3m. Tanks B & C are further driven with two eductors located in the center of the tanks B & C. A typical topology of the arrangement of the tanks is shown in the Fig.1The eductors have three circulators equi-spaced at 120 degrees from the axis of the input feed line.

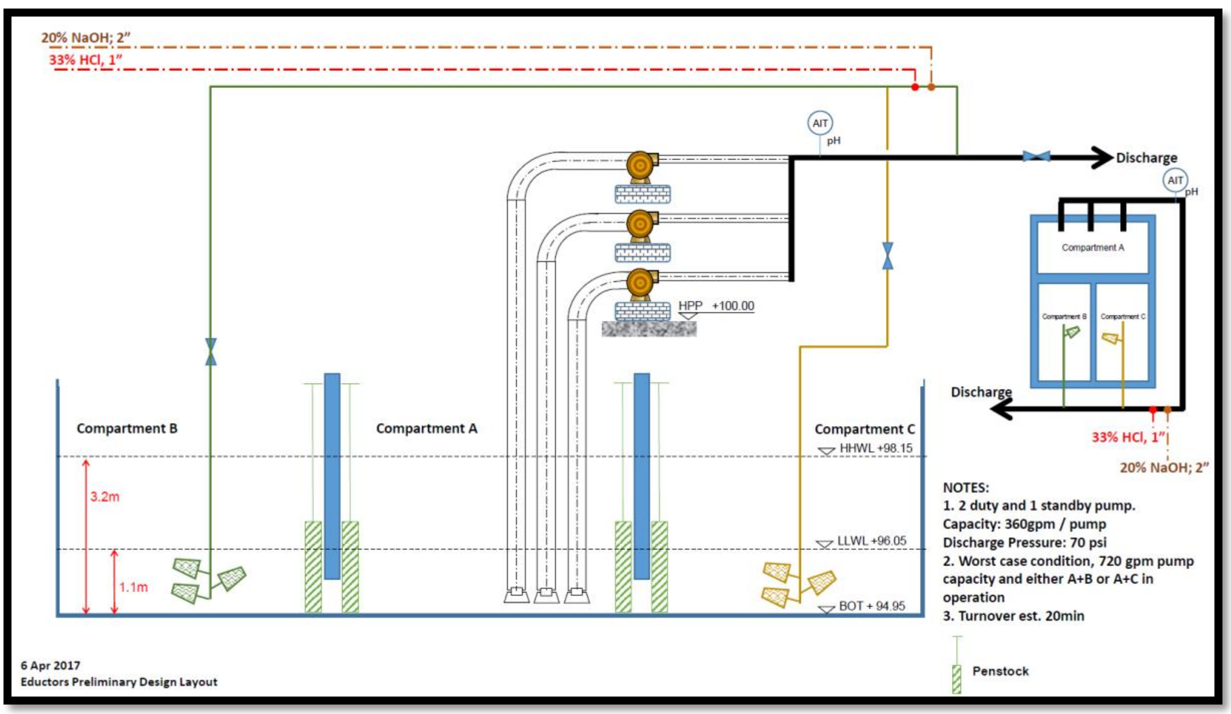


Fig.1. A typical Topology of the Tanks A, B & C of Neutralization Pit

To predict the mixing behavior of the moving fluids in the three compratments under the influence of the eductors and penstocks, it is proposed to perform a Computational Fluid Dynamics(CFD) analysis of the Neutralization pit system as per the 2D CAD drawings provided for the purpose of the analysis as shown in the Figure. 2 .

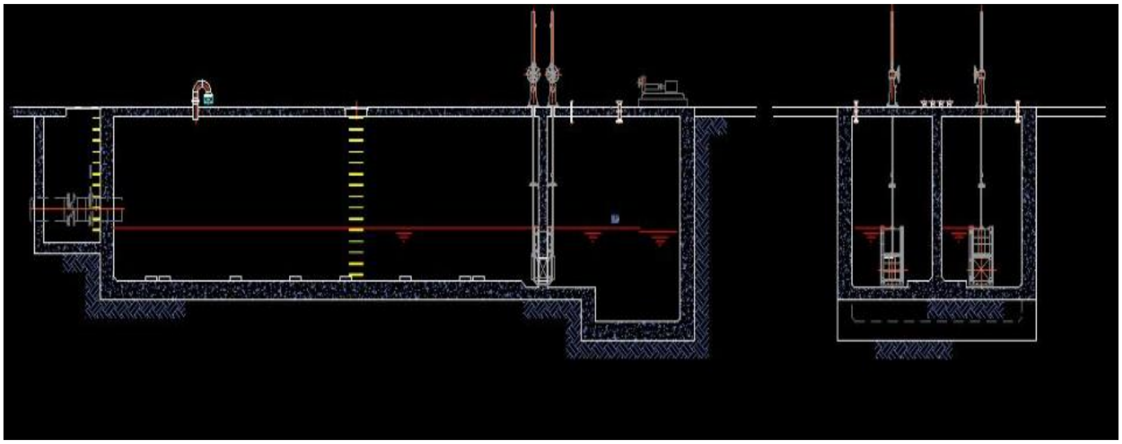


Fig.2 A 2D CAD drawing of the Neutralization Pit

1.2 Objective

The Primary Objective of this analysis study is to establish the flow dynamics and mixing phenomenon of waste water (tank-A) with NaOH+HCL+Water (tank-B&C).

The Analysis work performed here will not cover the chemical reactions involved in the process of Mixing and the

Analysis Study will be Modelled and performed for 20 minutes considering two the following two cases.

- Tank A+B/ A+C. (level H1)
- Tank A+B+C (level H3)

2. CAD & FE model

2.1 CAD Model

Based on the inputs given in the 2D drawing the a 3D CAD model is prepared for the purpose of the CFD analysis. The following figure 3, shows the details of the CAD model.

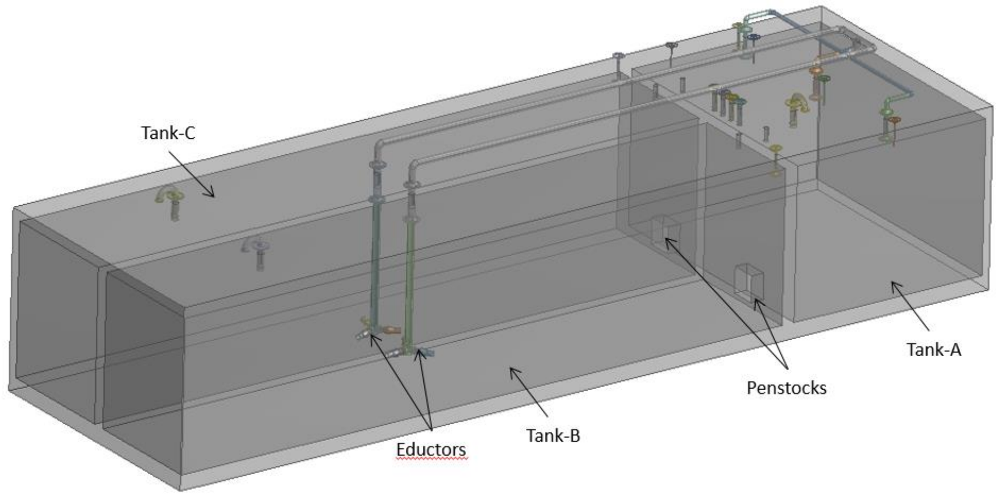


Fig.3 3D CAD model of the neutralization pit

2.2 FE Model

The FE model required for CFD is developed from CAD model. FE model is meshed with three dimensional unstructured hexahedral mesh elements to represent the whole geometry of the neutralization pit to the nearest approximation of the CAD model. These elements are proven for providing high quality solution with minimum number of cells. The fluids, tanks, eductors, feed pipes and the penstocks have been modelled in detail. All the critical elements in the system is given due consideration to represent the physics of the problem to best possible approximation so that the prediction represent the real mode of the neutralization pit. Figure 4 & 5 shows the finite element model of the representative neutralization pit.

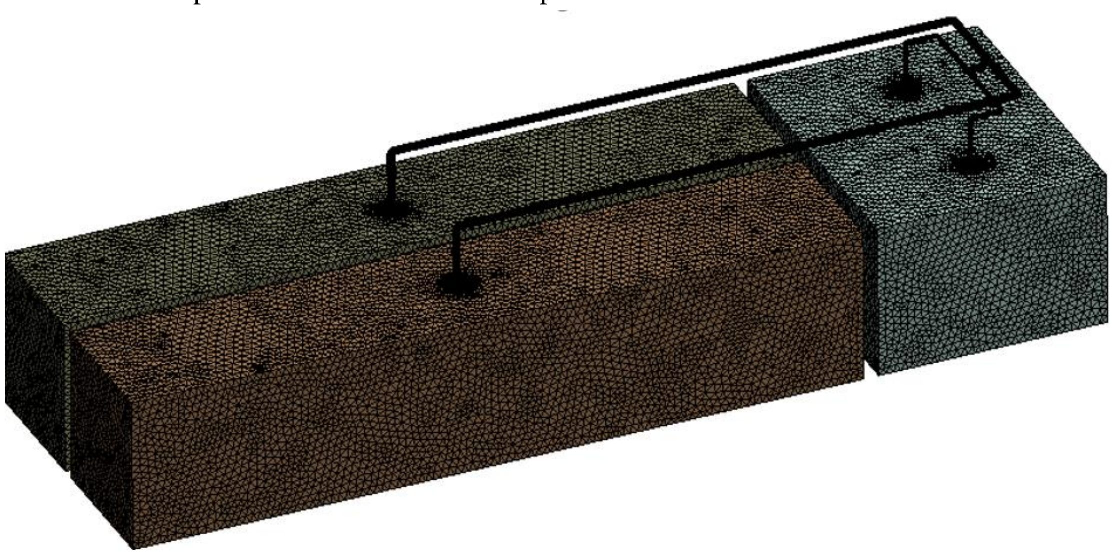


Fig.4 FE model of the Neutralization Pit

Fig.5 Zoomed FE model of the Eductors

The following table shows the detail of the number of elements used for modeling different media of the neutralization pit.

S.no	Domain	No. of Nodes	No of Elements
1	Fluid	206829	100162
2	Pump	1748	1443
3	Tank A	141001	30638
4	Tank B	250532	44620
5	Tank C	250532	44620
6	Educator	2045	1830
7	Circulators	2167	1728

Mesh Summary:
Total Number of Elements: 177634
Total Number of Nodes: 600110
Mesh quality: Skewness = 0.04 (avg.)
Aspect ratio = 1.05 (avg.)

3. Materials, Constraints & Assumptions for FE Simulation

3.1 Materials

The materials used for the analysis are listed below.

- Drain & Overflow
- Sodium Hydroxide
- Hydrochloric Acid
- Underground drainage
- Drain

3.2 Boundary Conditions & Constraints

CFD problem is idealized with a number of domains, like fluid, pump, tank A, tank B & tank C etc and their respective interfaces.

3.3 Assumptions made for the FE simulations

- The flow is the Neutralization pit is assumed to be a steady state flow.
- The effect of temperature change on the flow will be neglected assuming isothermal conditions. It is assumed that there are no strong body forces and relatively high flow rates owing to flow geometry.
- The Idealization uses standard wall functions and are assumed to effectively model the near-wall viscosity affected regions of turbulent flows.
- It is assumed that there are no slip boundaries at the wall of tubing, as all the fluid media involved in the analysis is flowing and is in continuous phase.
- The effect of wall roughness on the flow and shear stress will be neglected.
- All the solid materials used is analysis is assumed to be homogeneous and isotropic.
- The Fluids are assumed to be Non-Viscous and incompressible and follow the Bernoulli law.

4. Loads & Solutions

4.1 Loads and Load combinations

The provide solutions to the neutralization problem two load cases are considered for the analysis:

- **Case -1: Pump running 240 gpm x 1**
  Compartment A+B/A+C running water at H1 level (1.3m)(one pump start)
- **Case -2: Pump running 240 gpm x 1**
  compartment A+B+C running water at H2 level (3.2m) (two pumps running)

4.2 Solution:

The present analysis uses commercial CFD code, FLUENT, using control volume approach to solve the mass, momentum, energy balance equations created by domain discretization. The solver equations are solved by converting the complex partial differential equations into simple algebraic equations.

The κ-ε turbulence model with standard wall functions are used to solve the problem due to its proven accuracy in solving fluid mixture problems.

The Transient analysis of the problem is run for a cycle time of 20 minutes for each of the above case.

Analysis Results

Analysis results contain Velocity and Pressure profiles of the flow for the two load cases discussed earlier. The Eddy viscosity contour which is developed during the molecular diffusion due the momentum transfer is also presented in the results. The following figures show the analysis outputs (Fluid Flows) of the various cases.

4.2 Case-1 : Pump running 240 gpm x 1 (Compartment A+B/A+C running water at H1 level)

The Case 1 results are presented for time intervals of T=0, T=2 secs, T=6 secs, T= 10Secs, T= 40 secs and T= 20 mins, where flows significance is observed in the Figures 7 to 17. It may be noted the flow patterns at any time between T= 0 to T= 20 mins can be extracted from the present analysis.

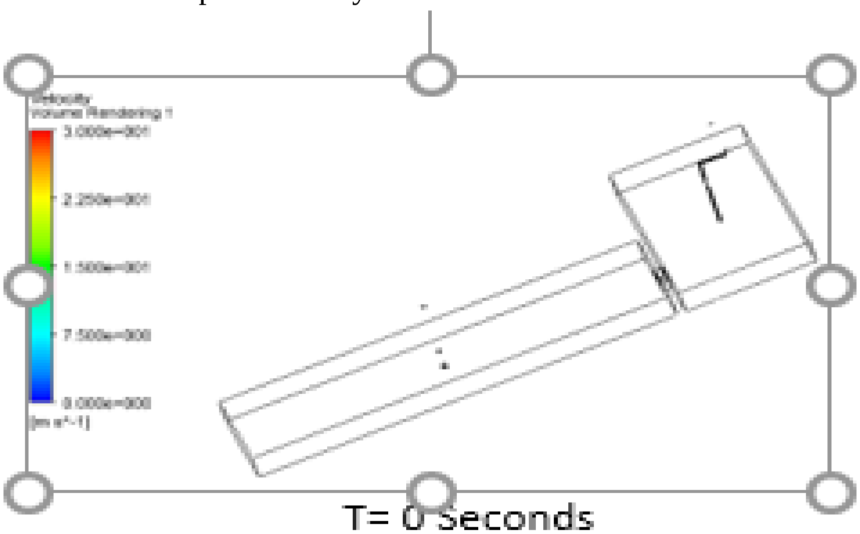


Figure 7 Velocity Profile for Case: 1 at T = 0 Secs.

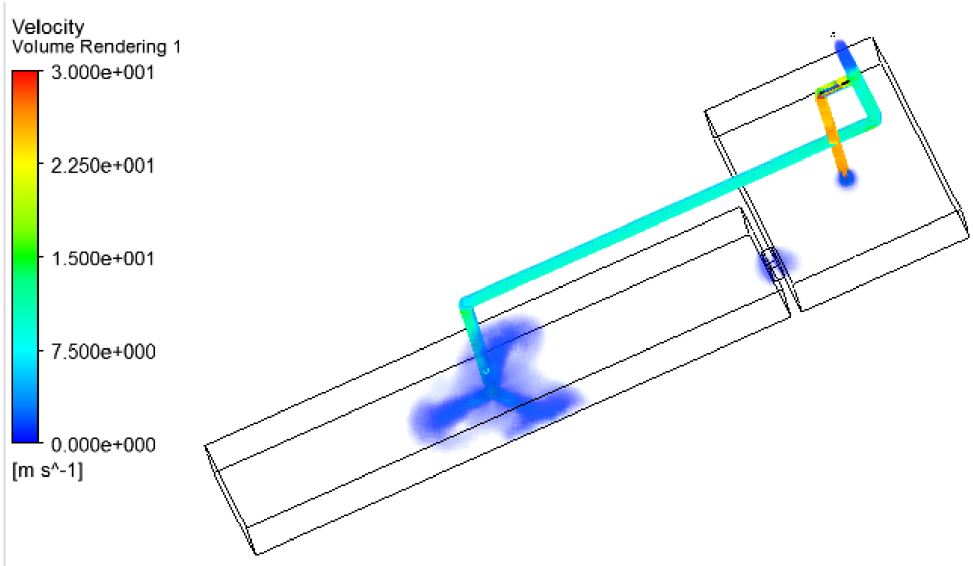


Figure 8 Velocity Profile for Case: 1 at T = 2 Secs.

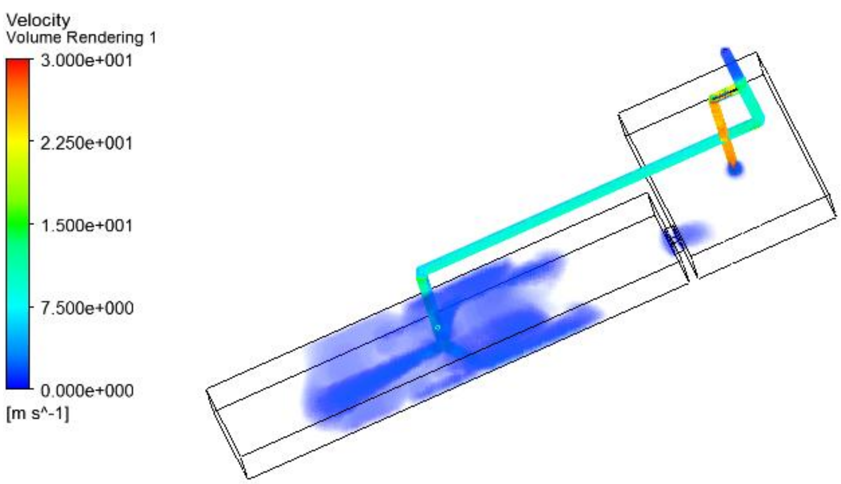


Figure 9 Velocity Profile for Case: 1 at T =6 Secs.

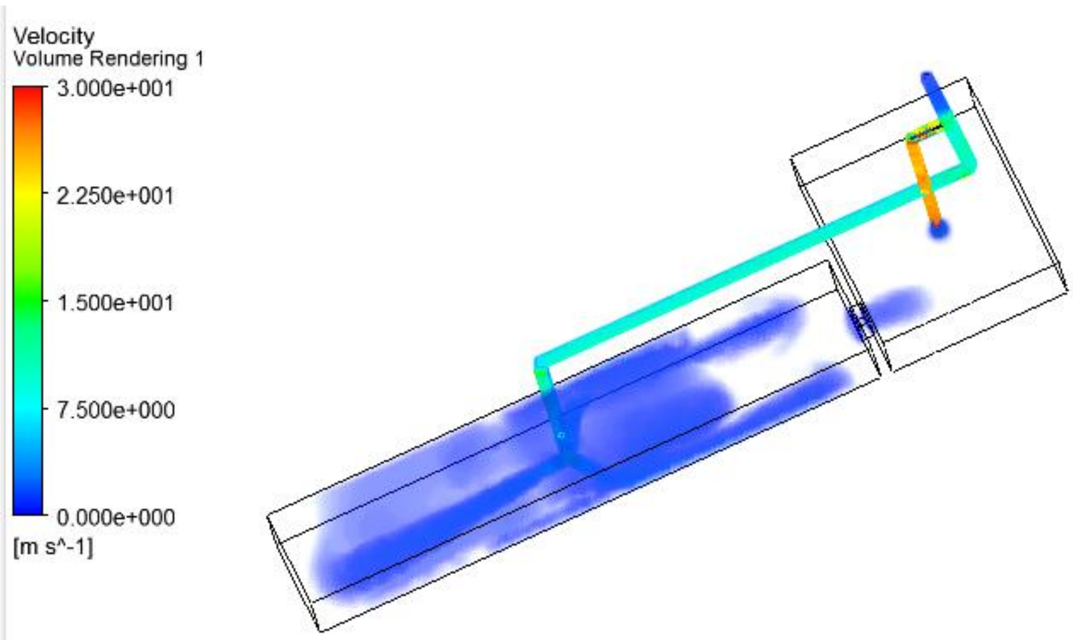


Figure 10 Velocity Profile for Case: 1 at T = 10 Secs.

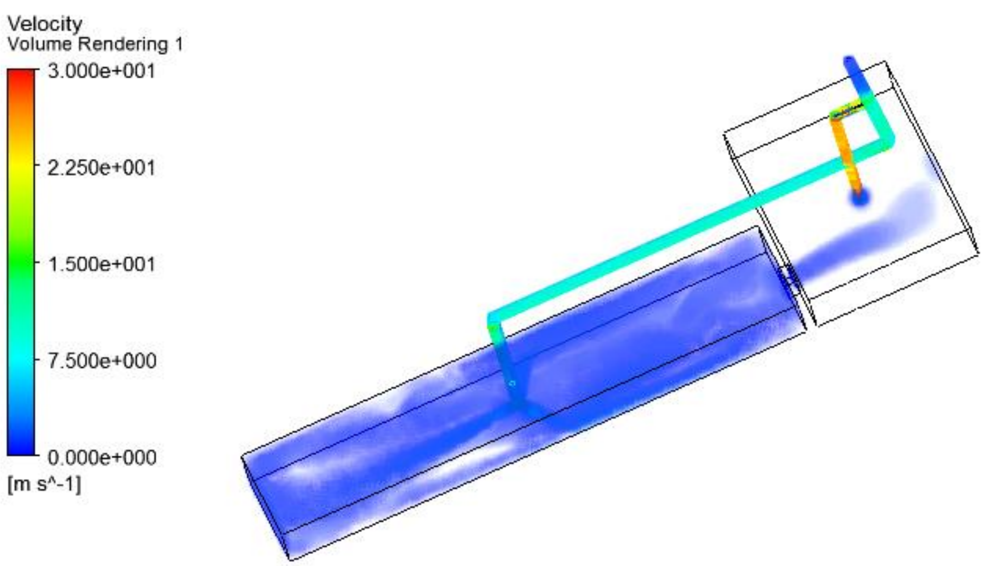


Figure 11 Velocity Profile for Case: 1 at T = 40 Secs.

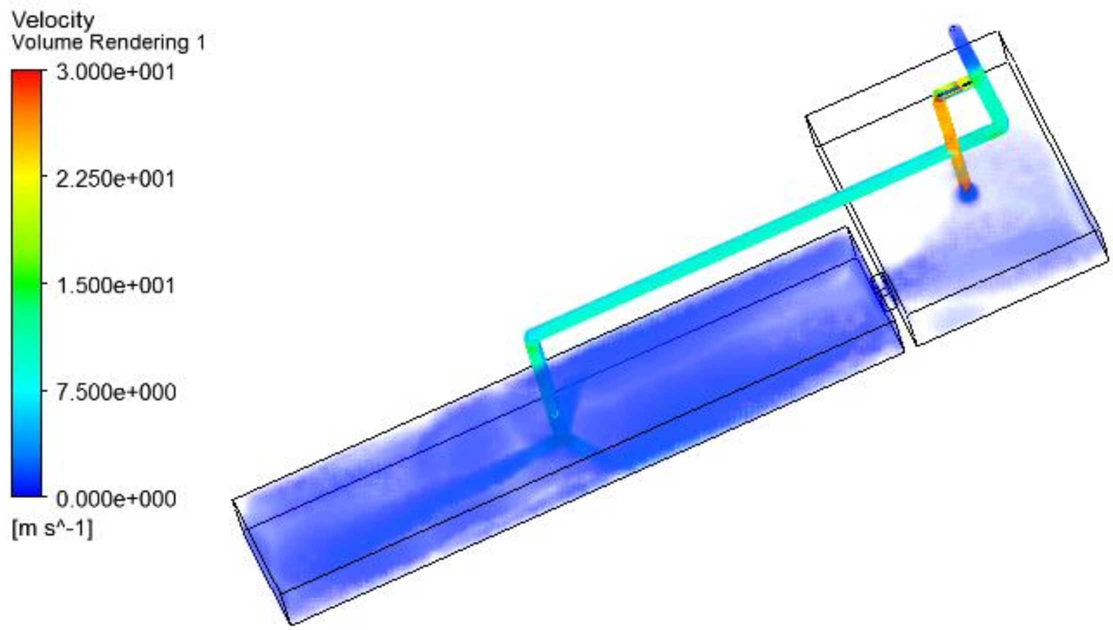


Figure 12 Velocity Profile for Case: 1 at T = 20 Mins.

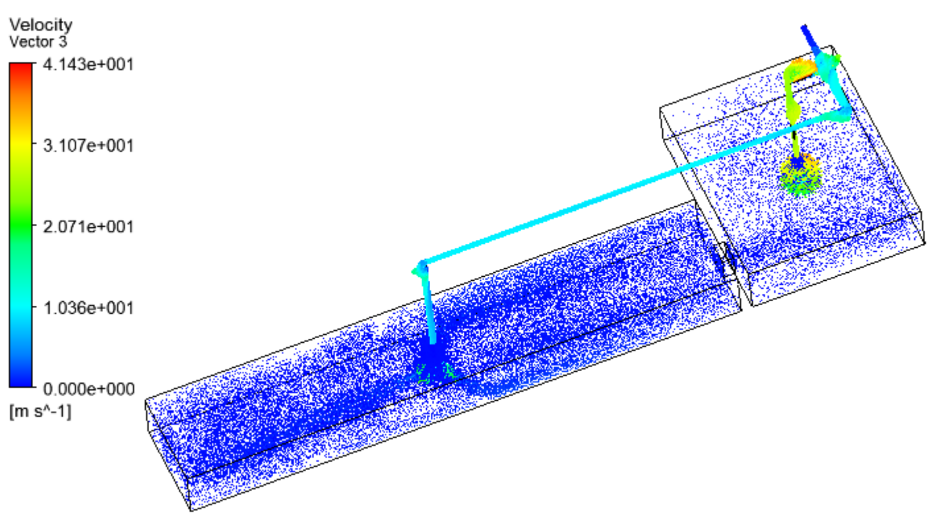


Figure 13 Velocity Vector Distribution

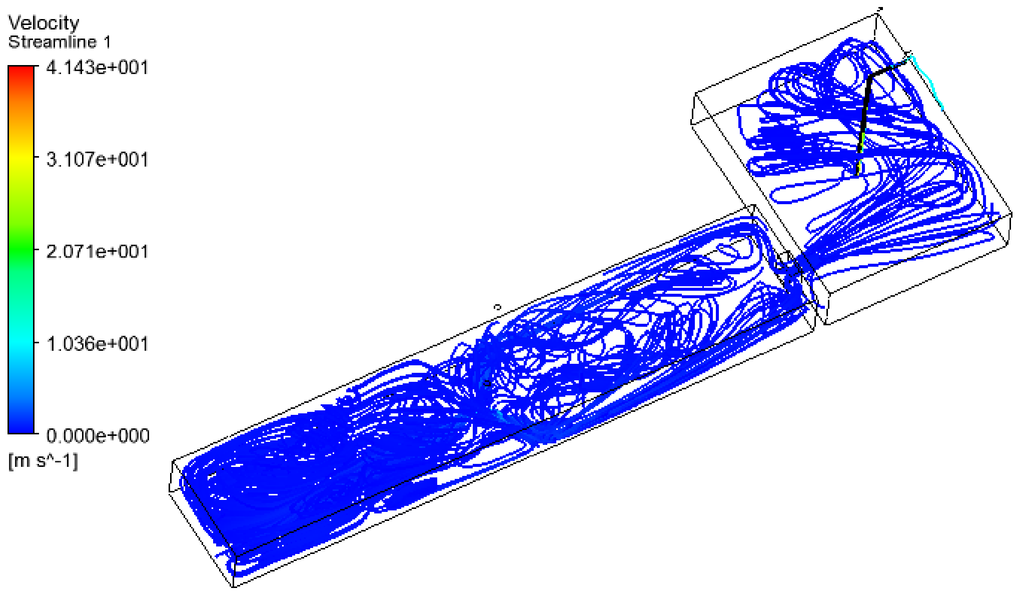


Figure 14. Distribution of Velocity Streamlines

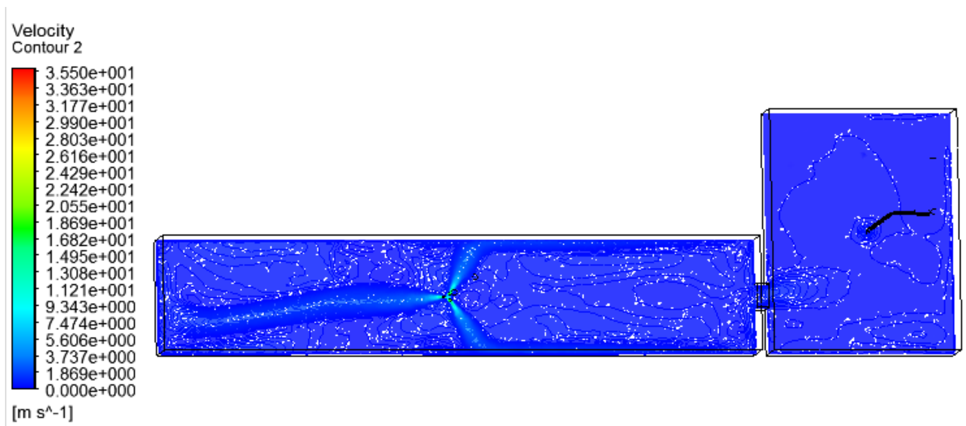


Figure 15. Velocity Contour distribution in overall Neutralization Pit(Plan)

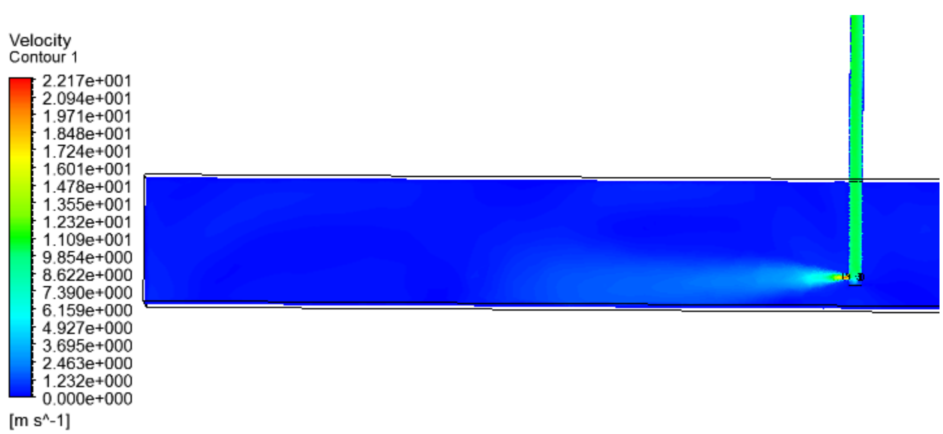


Figure 16. Velocity Contour distribution in the vicinity of educator (Elevation)

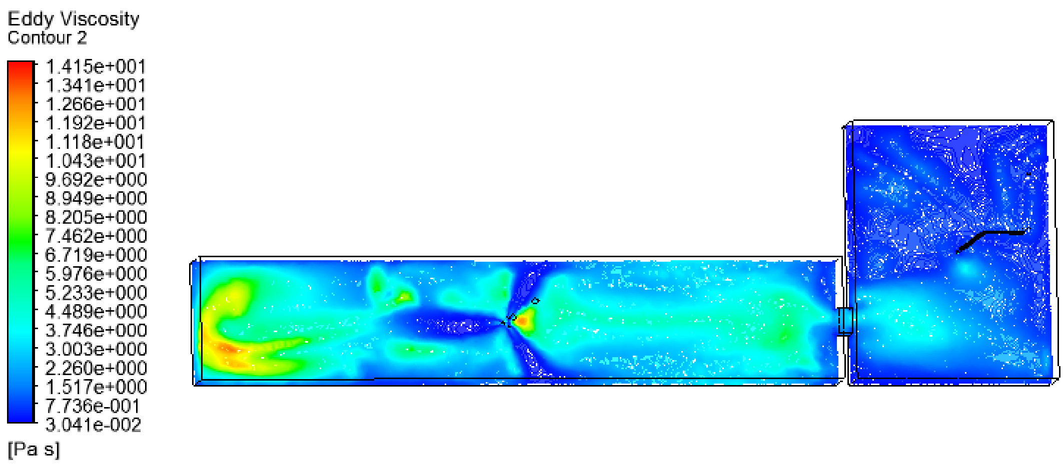
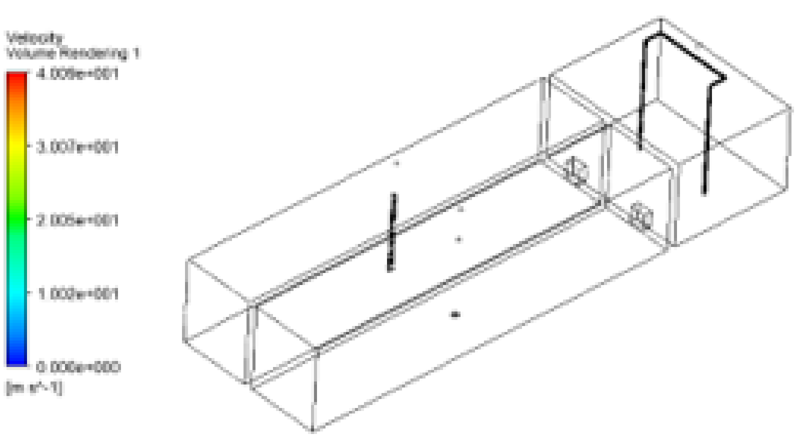


Figure 17. Eddy Viscosity contours in the Neutralization Pit

4.3 Case-2 : Pump running 240 gpm x 2: (compartment A+B+C running water at H2 level 3.2m, two pumps running)

The Case 2 results are presented for time intervals of T=0, T=0.5secs, T=30 secs, T= 80 secs and T= 20 mins, where flows significance is observed in the Figures 7 to 11. It may be noted the flow patterns at any time between T= 0 to T= 20 mins can be extracted from the present analysis.



T=0 Seconds

Figure 18 Velocity Profile for Case: 1 at T = 0 Secs.

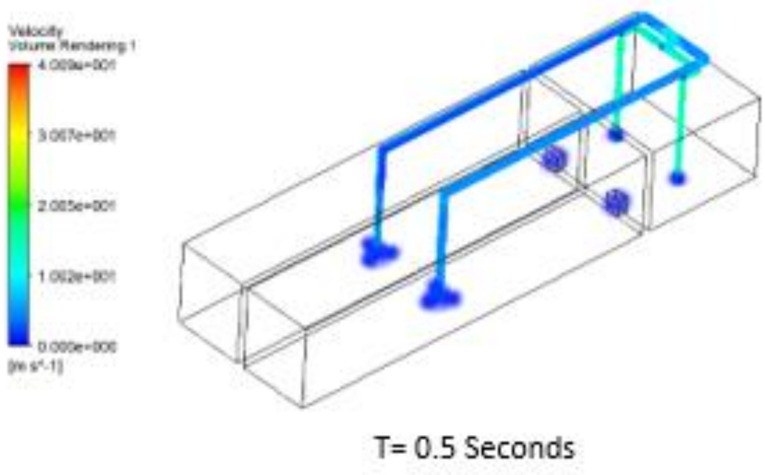


Figure 19 Velocity Profile for Case: 1 at T = 0.5 Secs.

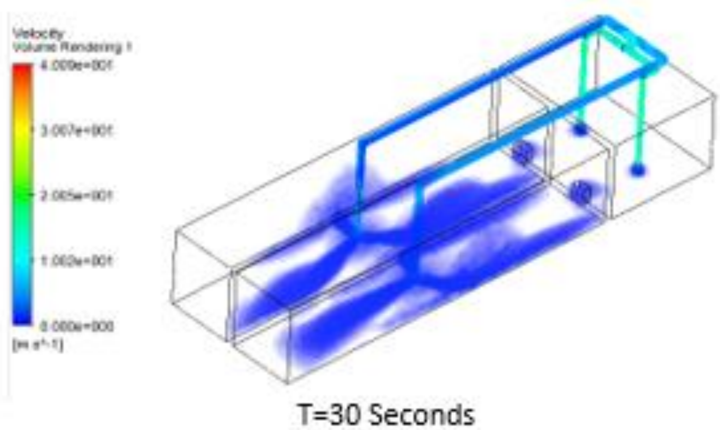


Figure 20 Velocity Profile for Case: 1 at T =30 Secs.

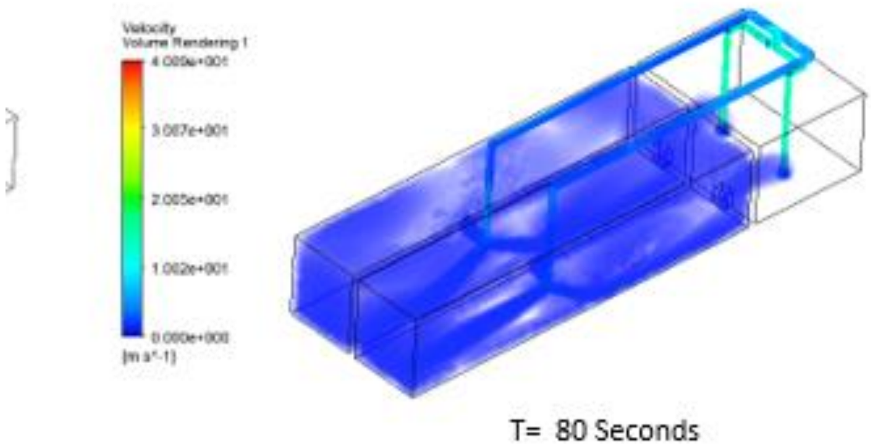


Figure 21 Velocity Profile for Case: 1 at T = 80 Secs.

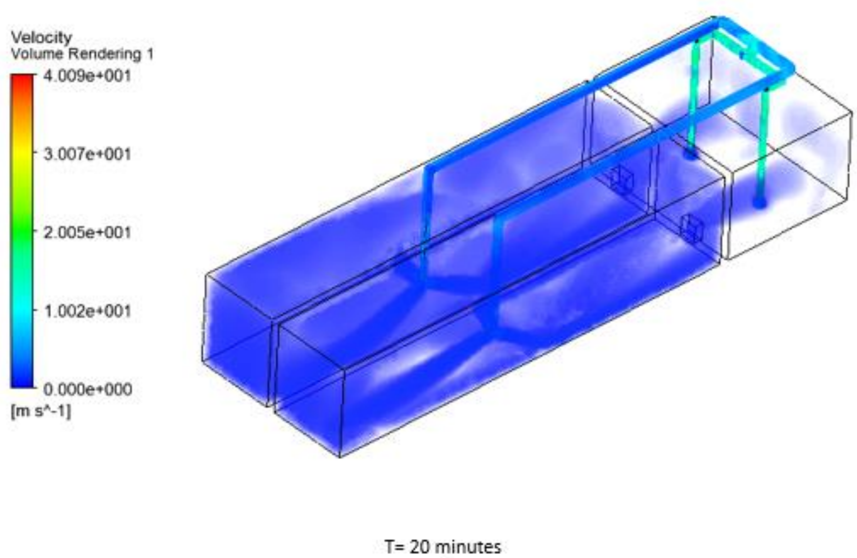


Figure 22 Velocity Profile for Case: 1 at T = 20 Mins.

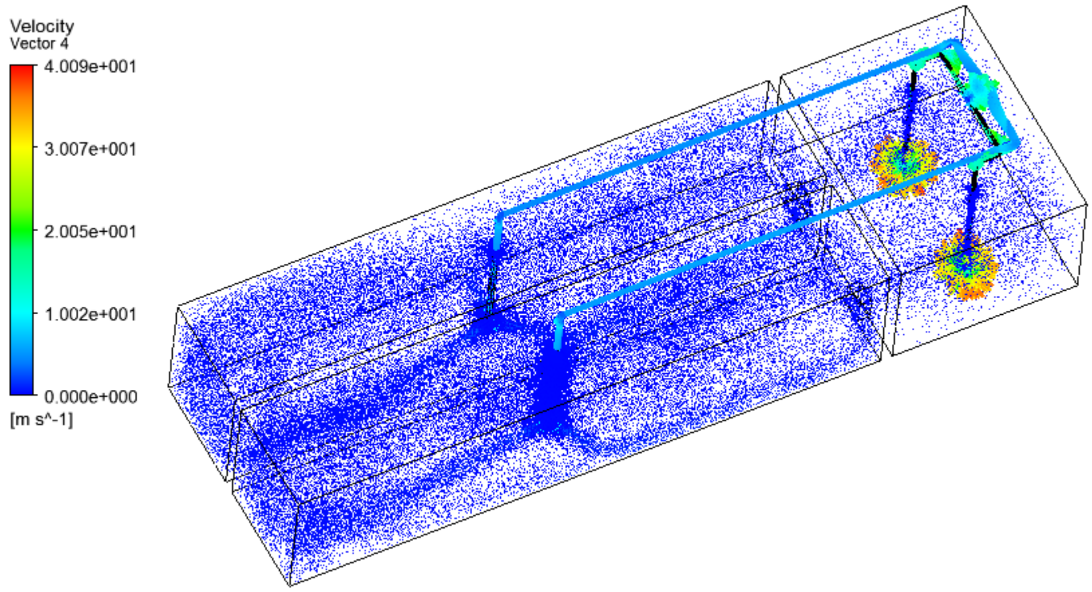


Figure 23 Velocity Vector Distribution

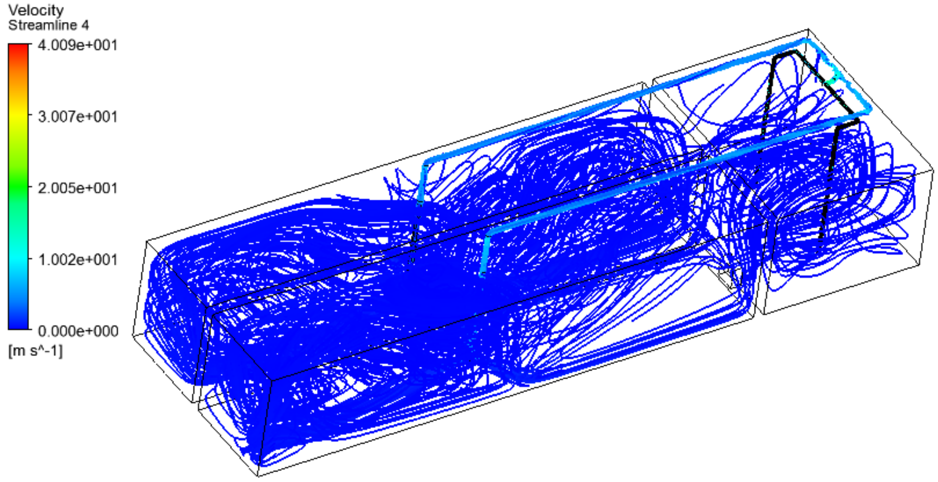


Figure 24. Distribution of Velocity Streamlines

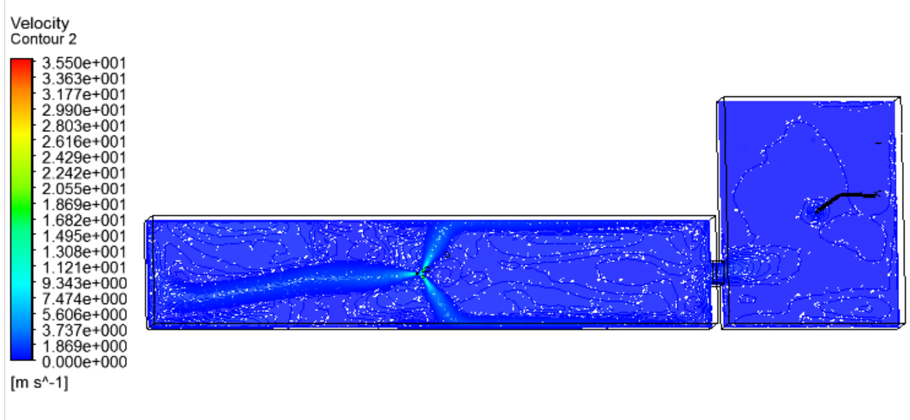


Figure 25. Velocity Contour distribution in overall Neutralization Pit(Plan)

5. Summary & Conclusions

- *The Neutralization pit of a water desalination plant is subjected to the Computational Fluid Dynamic Analysis in order to estimate the fluid flow behavior in the Tanks A, B and C.*
- *The Neutralization pit is completely modeled in CAD and Control Volume approach based idealization of the problem is done to predict the flow regime of the fluids under the influence of the two eductors built three circulators at 120 degrees in two tanks B & C*
- *To the Extent possible the Neutralization Tank is Completely represented mathematically, using Finite Element three dimensional unstructured hexahedral mesh elements to represent the whole geometry of the neutralization pit to the nearest approximation of the CAD model. The fluids, tanks, educators, feed pipes and the penstocks have been modelled in detail. in order to make the FE model as closer as possible to the realistic model.*
- *The results of the CFD analysis is presented and summarized for the two cases of Compartments A+B/A+C running water at H1 Level and Compartments A+B+C running water at H2 level. The Velocity, Pressures profiles for both the cases and the the eddy Viscosity is observed at the critical junctures of the neutralization pits.*
- *The κ-ε turbulence model with standard wall functions are used to solve the problem due to its proven accuracy in solving fluid mixture problems.*

- *It has been observed from the analysis results, that mass, momentum and energy are balanced thus proving the authenticity of the idealization and solution generated.*
- *The results show the transient behavior of the steady state turbulent flow at various time periods for both the case. It has been observed from the results that after 20 minutes there is perfect mixing in the neutralization tanks.*
- *The mixing mechanism strategy of A+B+C compartments with running after H2 level has displayed a more uniform missing through the neutralization pit much prior to the maximum time of 20 minutes for which the analysis is performed.*