DOE-FIU SCIENCE & TECHNOLOGY WORKFORCE DEVELOPMENT PROGRAM

STUDENT SUMMER INTERNSHIP TECHNICAL REPORT –SUMMER 2009

May 26, 2009 to July 31, 2009

NuVision Support: Demonstration of Power Fluidic™ Mixing Technology to Enhance Chemical Cleaning Operations in High Level Waste Tanks

Principal Investigators:

Edgard Espinosa (DOE Fellow Student)
Florida International University

Ethan King, PMP®
Project Manager
NuVision Engineering, Inc.

Florida International University Collaborators:

Leonel Lagos Ph.D., PMP®

Prepared for:

U.S. Department of Energy
Office of Environmental Management
Under Grant No. DE-FG01-05EW07033

ABSTRACT

At the Department of Energy (DOE) Savannah River Site, high level waste (HLW) tanks are being prepared for final closure. NuVision Engineering (NVE) Inc. has developed a technology called Power Fluidic™ Pulse Jet Mixers (PJM), which would aid in the waste removal process of HLW. Enhanced Chemical Cleaning (ECC) is the process that would be utilized for the removal of final~20% of the radioactive waste, called heal. After roughly 80% of the initial waste removed, posing fewer challenges, the remaining would be extracted through other methods. A demonstration of the technology was constructed at an 80% linear scale. The working model included the obstructions and features that are typically found inside high level waste tanks. The erected model included the upper and lower horizontal cooling coils and twelve drums representative of concrete columns. Through a drafted test plan, the demonstration completed a series of tests to determine whether the Power Fluidic™ PJM technology successfully met its objective. The constructed model was also replicated as a computational model. The computational model provides a theoretical baseline for the experimental tests conducted. The results obtained are highly dependent on the discretization and formulation of the computational fluid dynamic (CFD) model. Achieving accuracy was a high-priority, therefore, it is critical that the input data is addressed as best that describes the model. Results have been acquired for the earlier stages of the CFD analysis task breakdown. The velocity contours have been created, which show that the nozzles are creating a convective effect that is needed in order to introduce the acid to the fresh boundary layer. A mesh has been successfully completed for the task that follows. Further work is still required to complete the tasks.

TABLE OF CONTENTS

LIST OF FIGURES

1. INTRODUCTION

The Savannah River Site (SRS) is pursuing the use of a dilute-chemistry acid, which is used for the cleaning process at high level waste (HLW) tanks. The first intended tanks for the deployment of the Enhanced Chemical Cleaning (ECC) process are the Type I tanks in the SRS Area F tank farm. ECC purpose process is to reduce the oxalate loading collected at the bottom the Type I tanks. The driving element behind the technology is the circulating acid, which promotes a fresh boundary for the chemical agent reaction. However, the low-liquid level environment and geometry, refer to Figure 14 and 15 in Appendix A, inside the SRS tanks would pose a challenge in agitating the fluid. Being presented with this challenge, NuVision Engineering (NVE) Inc. developed the Power Fluidic™ Pulse Jet Mixers (PJM), and demonstrated that this technology would be a viable option for implementing ECC.

Power Fluidic™ PJM is comprised of a controller, a valve/jet pump skid, and a charge vessel. The compressed air acting across the jet pump can create either a high or low pressure. Through a series of cycles composed of suction, drive, and venting, the system generates a mixing effect inside the tank. The objective is to create the mixing effect to allow the dilute chemistry acid to react. The access point at which the jet penetrated the tanks is through two risers.

A demonstration of the technology was performed on July 22, 2009 in front of about 20 officials from Department of Energy (DOE) and SRS. The objective of the demonstration was to demonstrate the capability of the technology. It was investigated if a minimum speed of 0.2 ft/s would be achieved for the convective effect to occur, presenting a new boundary layer of the oxalic acid. During the date of demonstration, much enthusiasm was expressed on the technology. Power Fluidic™ very well would provide a low cost and reliable alternative for DOE and SRS. For this reason, the Power Fluidic™ PJM is being envisioned as the technology needed to tackle the HLW tank cleaning required at SRS.

2. EXECUTIVE SUMMARY

This research work has been supported by the DOE-FIU Science & Technology Workforce Development Initiative, an innovative program developed by the US Department of Energy's Environmental Management (DOE-EM) and Florida International University's Applied Research Center (FIU-ARC). During the summer of 2009, DOE Fellow (Edgard Espinosa) spent 10 weeks doing a summer internship at NuVision Engineering Inc., located in Mooresville, N.C., under the supervision and guidance of Erich Kesler, Ethan King, and Patrick Nevins. The intern's project was initiated on May 25, 2009, and continued through July 31, 2009, with the objective of aiding in the analysis of the system developed by NVE. The work analysis was accomplished using theories in the field of Computational Fluid Dynamics (CFD). Throughout the internship, the DOE Fellow used a software package called FLUENT™ along with a second software package called Gambit™. The DOE Fellows' previous knowledge on other CFD packages and fluid mechanics were significant in supporting Power Fluidic™ PJM. The DOE Fellows' duties during the internship included: drafting the executable arrangement that would approach the problem and arrive to deliverables, present findings of model, and modifying the CFD code to reflect additional tactics propose by the Project Manager. The DOE Fellow worked closely with lead engineers to further strengthen the Power Fluidic™ to be used on the SRS Tanks. The ECC cleaning process is a technique currently used to remove the 10% -20% waste (called the heal waste) remaining in the waste tank after much of the bulk waste is remove. The removal of the heal waste requires its own separate process due to the oxalate material formed. It's hard, tough characteristic make it difficult to remove, requiring to used a chemicals in the process. The CFD analysis completed during the internship was in support of constructed model that was presented to SRS officials on the demonstration date. However, the task of completing the model will continue after the summer internship is completed. A scope of work was commissioned to the DOE Fellow to further support NVE directly from Florida International University. This document specified the key points which pose as challenges to the analysis and how the DOE Fellows will address the issue.

3. RESEARCH DESCRIPTIONS

The efforts on the CFD model is anticipated to provide substantial analysis that can validate the performance of Power Fluidic™ along with the experimental data acquired through the test plan. The deliverables for this task is an analysis of the complete system. The following sections describe the must haves of the CFD model. The results obtained are highly dependent on the discretization and formulation of the CFD model. Achieving accuracy was a high-priority therefore, it is critical that the input data are addressed as specified.

A successful model which to authenticate the capabilities Power Fluidic™ Technology would require a strict guideline in the modeling. To ensure the proper approach and setup of the task, the mission is dissected into smaller tasks to arrive to the overall goal. In pursuing this logic, it would ensure the model is specified in details that can best be a representative of the environment in SRS tanks. The FLUENT™ CFD package software will be used to arrive to a solution to the technical challenges. The following are key points, identified by NVE, to achieve a successful model.

a. **Two phase analysis;**

b. **Open (free) surface boundary condition;**

c. **Account for energy lost to jets at breaking surface;**

d. **Mesh generation around obstacles (tank columns and cooling coils)**

e. **Model size and mesh size**

A successful model consists of five (5) subdivisions. Within the subdivisions, described below, the five key points mentioned above will be taken into consideration.

3.1 Tank Geometry

The CFD process will include the need to enhance the discretization to better capture the tank features. The features that will be meshed in the process will be the 60-ft diameter tank itself, along with a system of nested cooling coils located at a close proximity to the bottom of the tank. Appendix A contains Figure 12 and Figure 13 which are the schematics used to create the geometry in Gambit™. During the meshing of the cooling system, several problematic errors may occur. The discretizing elements must be efficiently assembled to avoid skewness. Skewness in the elements allow for the generation of errors during the numerical process. In addition, because of the fact the cooling coils being finely meshed, the growth of the element to the bottom of the tank must be a smooth transition as possible. This is due mostly because of the size difference between the surface of the pipe and the surface of the bottom of the tank. Additional geometry includes two vertical columns [Refer to Appendix A, Figures 14-15]. Figure 1 provides a good visual to provide a good understanding of the complexity of the obstructions inside the tank of the built model.

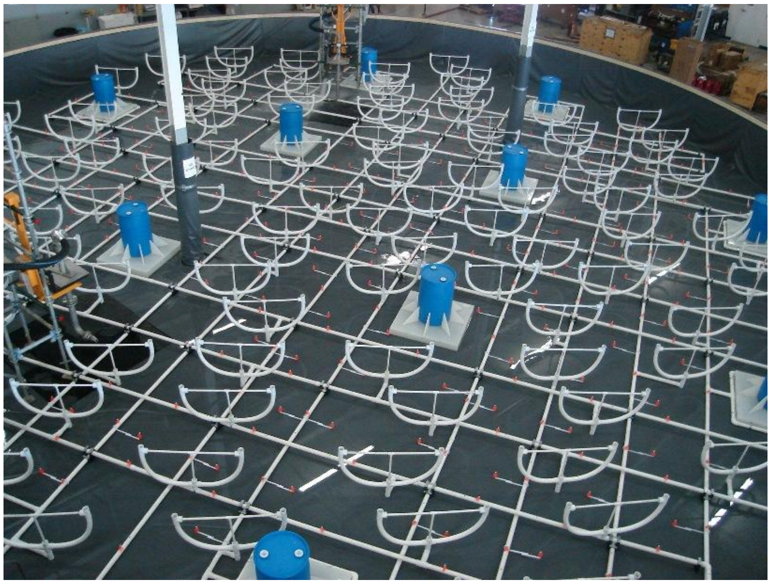

Figure 1. The demo model of the SRS tanks.

3.2 Fluid Properties

SRS provided new information regarding the range of data for fluid properties, including specific gravity and kinematic viscosity. This data will be entered in the FLUENT™ user graphic user interface to describe the fluid domain.

3.3 Scaling

The tanks at SRS will be represented at an 80% linear scale. This scale was chosen specifically because of the capabilities of the testing facility at NVE. The system and operating parameters were scaled to maintain the dimensionless groups in order to provide the realism of the model.

3.4 Steady State

Each cycle the system undergoes will consist of a total of 120 seconds. The cycle will consist of 10-20 seconds of high pressure (drive), a venting stage, and 90-100 seconds of low pressure (suction) intake at the nozzle (jet pump pairs). It is not understood yet how many cycles are required to achieve steady state; however, it would be investigated through the CFD model if this number can be determined. The nozzle will be formulated with a user defined function (UDF) that must express the velocity profile seen in Figure 2.

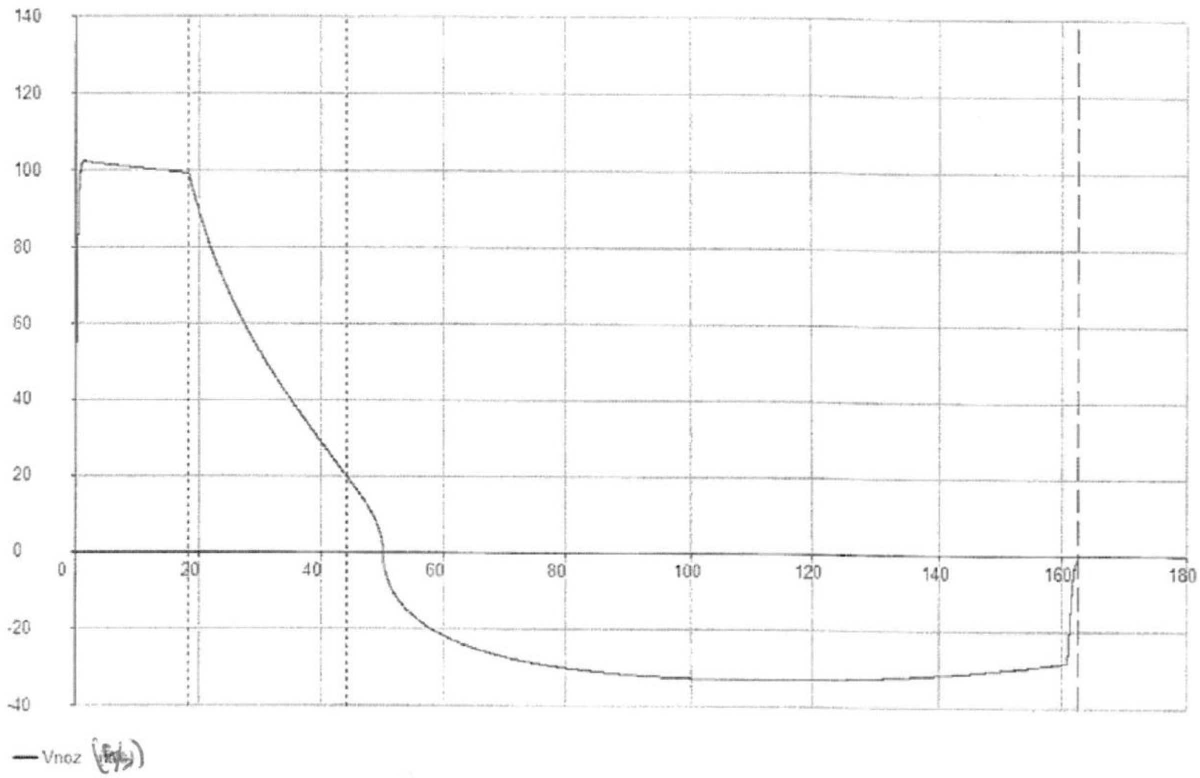

Figure 2 The velocity profile, duty cycle includes suction phase, drive phase, and venting.

3.5 Two-Phase System

The accuracy of the CFD model would be further enhanced by moving from a single-phase system to a two-phase system. In achieving the modeling of two-phase system, an open (free) surface boundary condition must be adhered to the model. The FLUENT™ software package has the capability of modeling open channels. Adjustments made to the setting will be made to adapt to the tank's environment. The possibility of cavitations at the nozzle exists highly due to the high speed at the nozzle. This will further need to be investigated to surely monitor for affects in the model. The plume breaking at the surface indicates a loss of energy. Measurement on the amount of energy loss at the surface, due to various liquid levels, high discharge pressure, etc., will be examine with the use of probe in the areas where the plume appears in the tank.

As already mentioned, the task is dissected into smaller cases. Each case will add to the overall model. Each sequential case will include previous results derived in the former cases.

Case I: Model the tank geometry, including the two nozzles located at the only access riser location.

The model will be at 80% linear scale of the actual measurements of the tanks at SRS. The geometry size that will replicate the built model established in the facilities of NVE. In actuality, the CFD model is modeling the demo being built at NVE facility. Included in the case is the placement of nozzle with appropriate boundary condition at the location of the riser access point.

Case II: Implementing the upper and lower cooling coils; geometrical feature which cause a disruption in the fluid path.

The geometry of the model will be enhancing further more to include the upper and lower coiling coils. The accuracy of the geometry would determine how much it affects in achieving the criteria of 0.2 ft/s, the major objective. With experienced University advisors on the subject, the meshed generation around tank geometry and obstruction found with be formulated to achieve the discretization of the model. The task will need to take consideration of model size which will highly affect number of elements associated with model. These efforts are made to improve accuracy in the solution. This case is composed of Case I and Case II.

Case III: Velocity Profile will provide simulation of Nozzle Flow at inlet.

The system operates on a series of pressure cycles, each elapsing 120 seconds. The charge vessel generates a vacuum build up for 90-100 seconds, then 10-20 seconds of high pressure discharge from the nozzle, ejecting the dilute chemistry acid and finalizing the cycle with a short vent. A velocity profile is generalized to emulate the actions of the duty cycle, suction, discharge, and venting. A User Defined Function will be written to for this purpose. This will provide the model with the essentials to act as the mock up. This case is composed of Case I, Case II, and Case III.

Case IV: Transformation to the Two-Phase System

To finalize the modeling, the final approach to simulate the events occurring in the demo tank, is to formulate the two-phase interface that is provide distinction inside the tank. In the previous cases (Case I – Case III) the environment was a single phase. However, to enhance the accuracy and realism of the model, the two phase system will need to me implemented. This should also include the free surface boundary condition at the point of interaction. Fluent has the capabilities to approach an environment with a free surface boundary condition. Fluent approaches the situation in manner similar to Open Channel Flows. Modification will be adjusted appropriately to describe the environment inside the modeled tanks.

Fluent has the ability to place measuring probe with the domain. The pressure at several locations will be measure including the pressure at the inlet of the nozzle. An analysis of pressure where the plume breaks at the surface and the pressure at the inlet will be conducted to measure the energy lost associated with the plume breaking at the surface.

4. RESULTS AND ANALYSIS

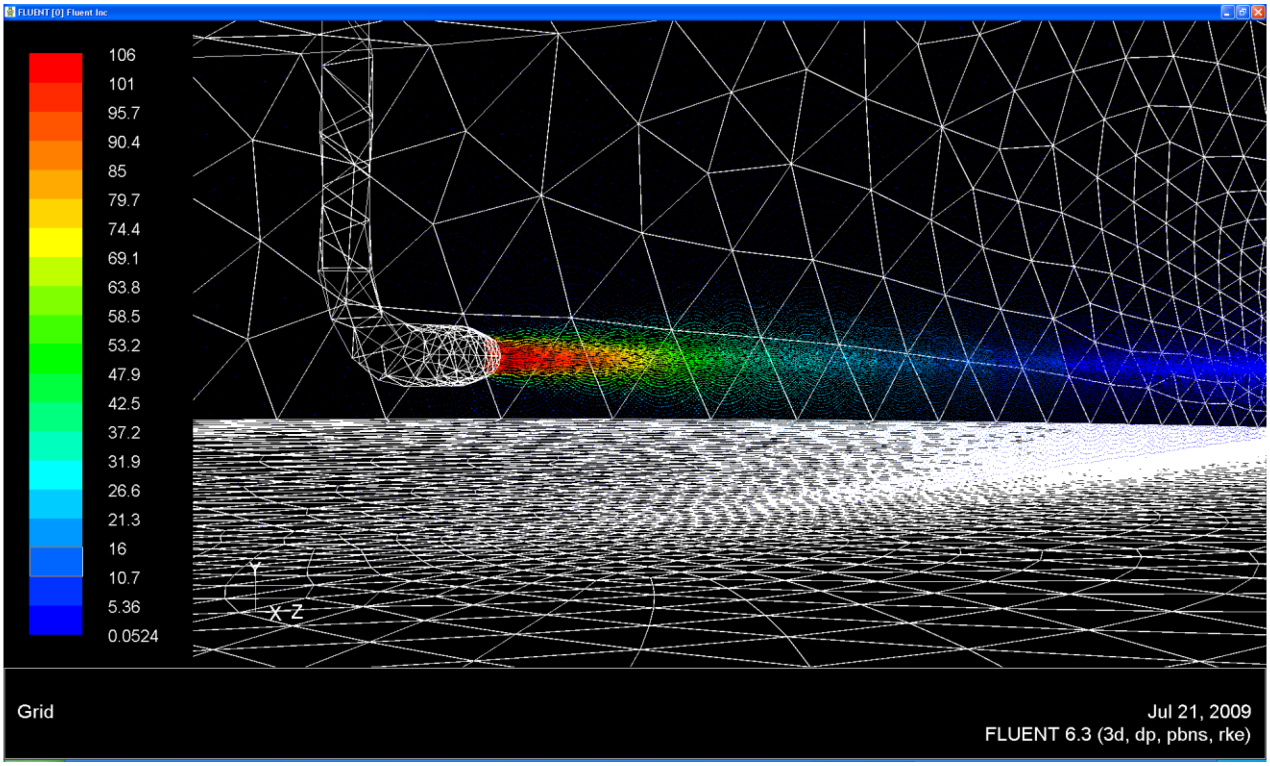

Figure 3. Discretization of the domain.

Case I: Model the tank geometry, including the two nozzles located at the only access riser location.

Figure 3 demonstrates the grid generated, which captures the features of the nozzle and tank walls. A critical area in this model is providing enough elements at the face of the nozzle (inlet) solely because it is the area in which the boundary condition has been specified. The boundary condition specified at the nozzle inlet was a velocity inlet. The total number of elements that were used to mesh this study (60ft. dia. tank, two nozzles) consisted of ~1.5 million elements. The scalar speed, which was specified, was 30.48 m/s (~100 ft/s.). The velocity profile provided, shown in Figure 2, was then used to determine the speed at which the fluid specified at the nozzles. The scale shown in Figure 3 is the overall scale to include the maximum velocity (ft/s) to the minimum velocity (ft/s) seen in the tank simulation.

To reach a convergence, the computational time was roughly three days the use of an appropriate turbulence model. Convergence however was reach in intervals of ~1500 iterations, strategically to avoid additional iterations during the process if trends were observed not to reach convergence, see Figure 4 for demonstration on preceding statement. Three sequence of running iterations were completed for the study involving only 60ft. dia. tank and two nozzles. The error which occurred during the first run was duly to the assumption of laminar flow from the nozzles. The error was clearly demonstrated in the calculation of the residuals, which explains the model tendency to converge. The set convergence criterion for the model was $10^{-4.}$ The iterative process in Figure 4 was not reaching the desire criteria.

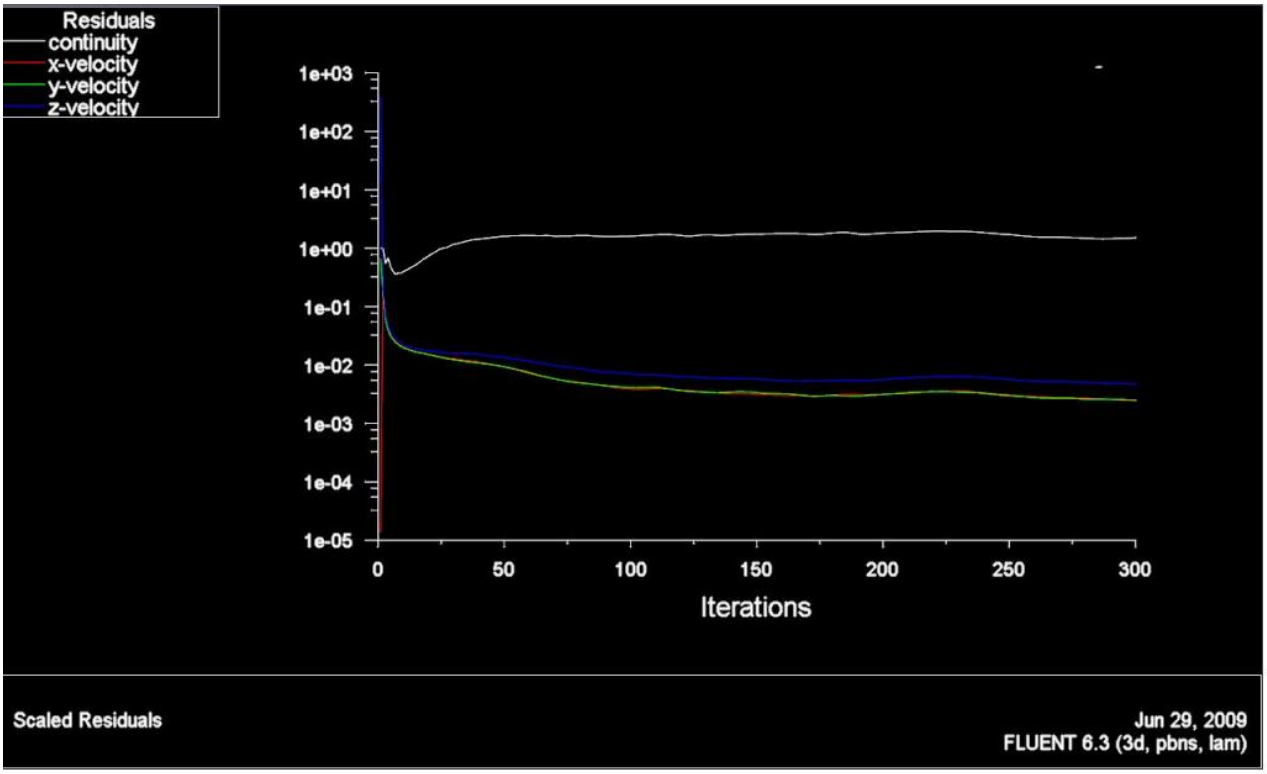

Figure 4 Residuals show not convergence in solution after 300 iterations.

The Reynolds Number on the flow was calculated to determine whether if this was situation that was occurring. A turbulence model was chosen to be implemented in the model's specification. The Reynolds Number provided evidence that this was a necessary component for the model. Turbulent flows are characterized by fluctuating velocity fields. These fluctuations mix transported quantities such as momentum, energy, and species concentration, and cause the transported quantities to fluctuate as well. Successful computation of turbulent flows requires consideration of the mesh generation. Numerical diffusion could steer the solution process into the wrong direction. During meshing process, area of interest should be finely mesh and avoid degenerate elements. A standard k- ε model was chosen, specifically the RNG k- ε model. The RNG k- ε model takes 10%-15% more computation time more than standard k- ε model due to the addition of an equation solved for in the process.

Another control feature that allowed convergence is the application of reducing the under-relaxation factors. The under-relaxation factor whose ratio is specified determines their degree of influence on the governing equations is: pressure, density, body forces, momentum, turbulent kinetic energy, turbulent dissipation rate, and turbulent rate. FLUENT™ has controlling feature allows the user to set ratios how much effect they have on the governing equations, which are flow and turbulence in this model. Under-relaxation is a useful device for nonlinear equations. It is often employed to avoid divergence in the iterative solution of nonlinear equations. Figure 5 demonstrates where tweaking with the ratio would be necessary. The continuity equation is not reaching the desire residual. The parameters (pressure, density, body forces, momentum) influence may be set to strong not allowing convergence.

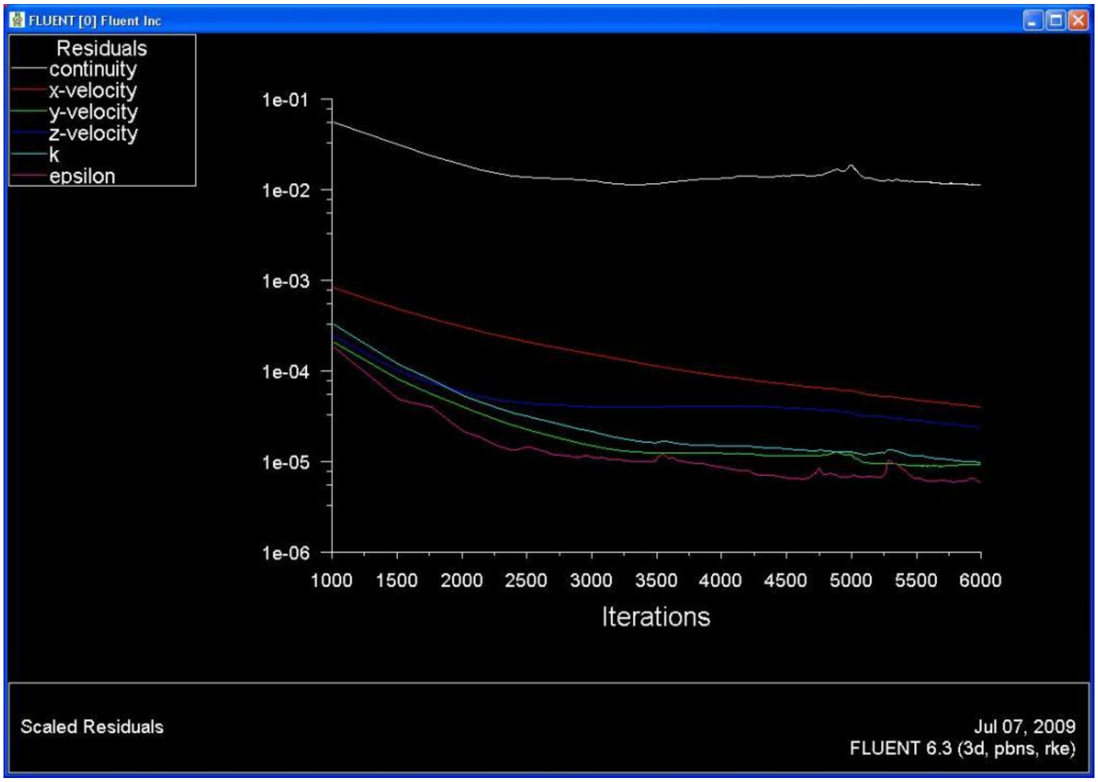

Figure 5 Convergence is approaching. Manipulation of the under-relaxation factors is necessary to aid full convergence.

Figure 5 has not reach convergence. At the start of the iterative process, lowering the ratio of each of the influencing parameters to:

Table 1 Under-relaxation ratios.

Parameter	Ratio
Pressure	0.3
Density	0.5
Body Forces	0.5
Momentum	0.5
Turbulent Kinetic Energy	0.8
Turbulent Dissipation Rate	0.8
Turbulent Rate	0.8

Changing the ratios to the preceding would allow a gradual convergence towards 10^{-4} in the residuals. However, it was strategized to gradual increase the ratios till it eventually reached 0.9, as recommended by the user guide, by every ~1500 iterations. The converging criteria set for the solution was 10^{-4}. Figure shows the smooth decreasing of the residual using the aid of gradual increased under-relaxation ratio nearly under 4000 iterations [Refer to Appendix A for computer computation for final iterations].

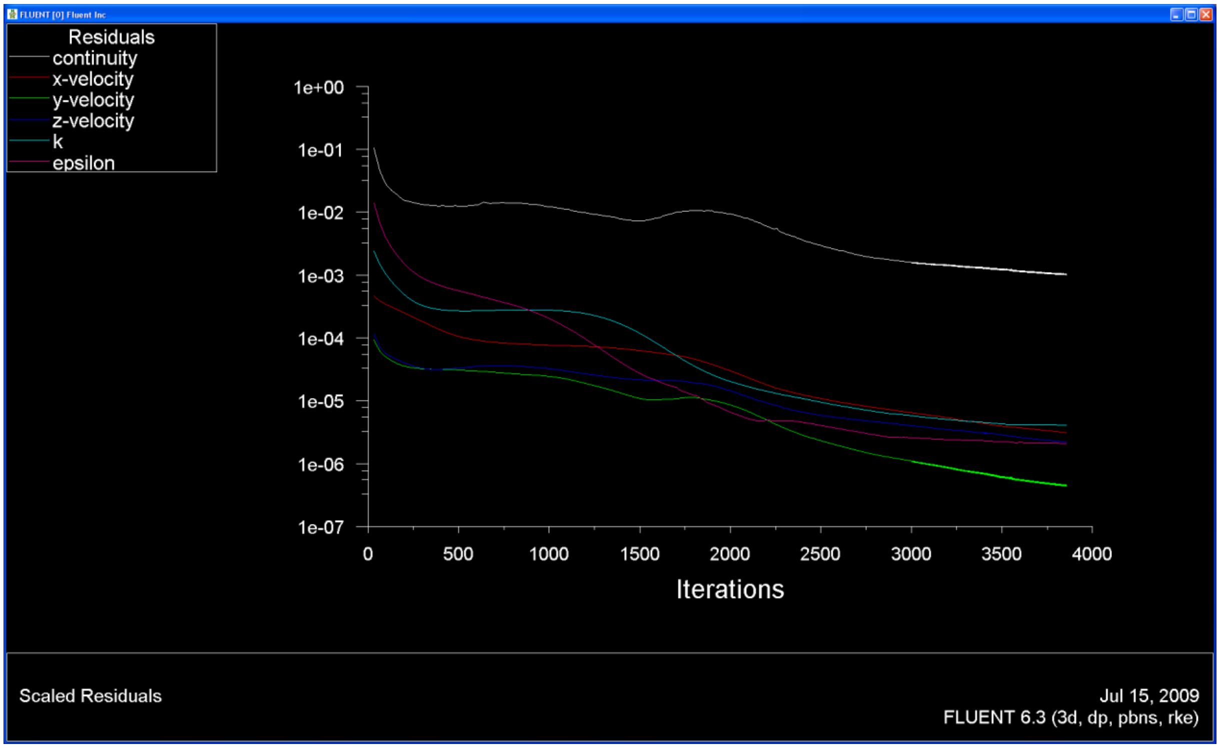

Figure 6. Final converged solution for case with only tank and two nozzles.

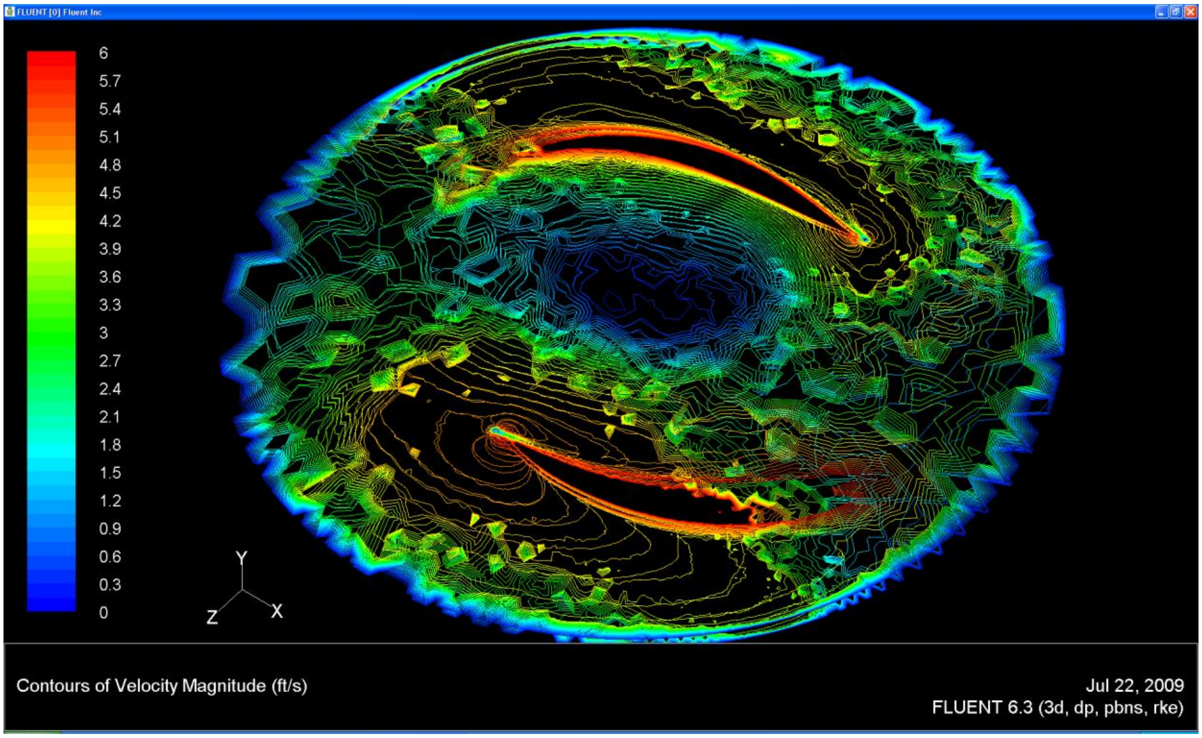

Figure 7. Flow patterns which in the tank. Contour plot at the bisecting horizontal plane at the nozzle mid orifice.

The fluid was expected to generate a circulation motion in the tank. As seen in Figure 7, the fluid being ejected out the nozzle is forced to rotate due to the accumulation of particles against the wall of the tank. It is because of the geometry of the tank that the circulation is possible. The scale on Figure 7 shows the velocity range for the contour plot of velocities up to 6ft/s. Due to the 100ft/s condition specified at the nozzle inlet, it is expected for area near the nozzle to demonstrate such speeds. Area reaching furthest from the nozzle is what is of interest in the study.

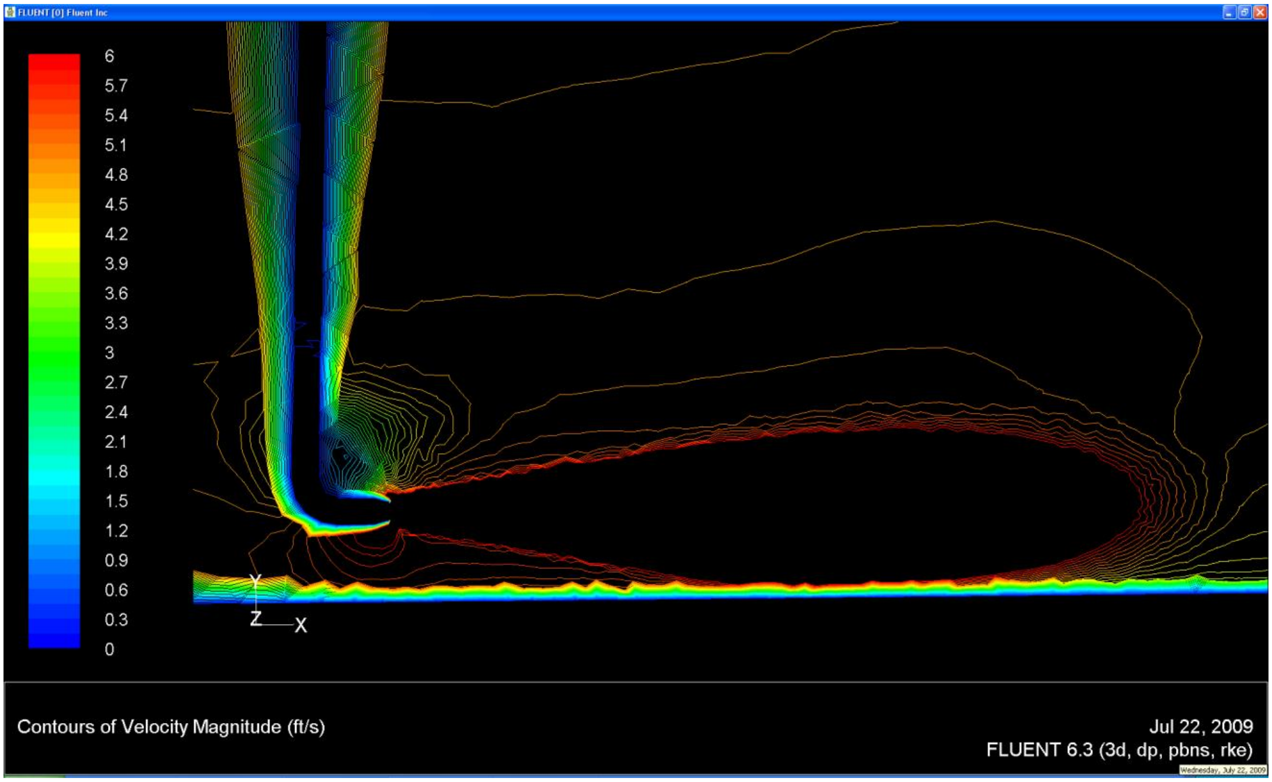

Figure 8 Examination of the plume.

The plume is the source of the energy and momentum, which propagates each fluid particle to cause motion. An examination of the outer band of the plume is seen in Figure 5. The scale adapted here is modified to shows velocities (ft/s) of the contour plot. Range of velocities in the contour plot greater than 6ft/s have been omitted to provide clarity in the plot. The figure shows that the minimum velocity in the plume is 6 ft/s and it reaches about 70% of the distance between the wall and the position of the nozzle. These bands provide us with great confidence that, in this case, the tank is completely filled (10ft). In the model scenario, only two feet of water will be in the tank, and the center of the nozzle will be 1ft above the bottom surface. The plume in Figure 8 is using an immense amount of energy to overcome the weight of the water above it.

Case II: Implementing the upper and lower cooling coils; geometrical feature which cause a disruption in the fluid path.

The upper and lower cooling coils was later incorporated in the model. Figure 9 demonstrates the final stages of the process. Five million elements were generated to mesh the tank's geometry, nozzles, and cooling system.

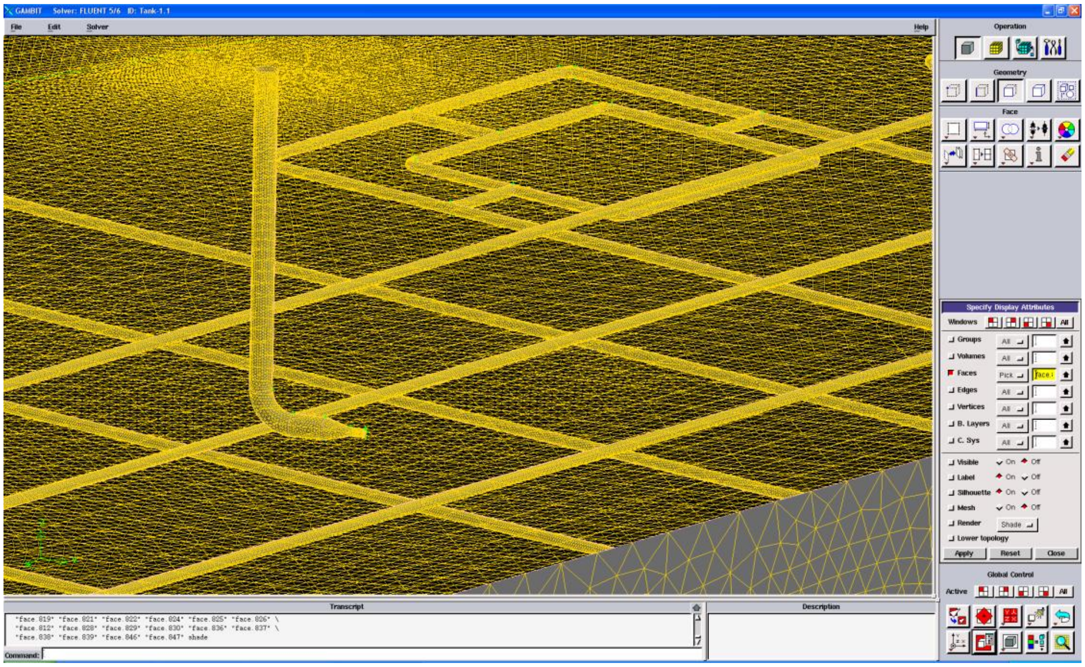

Figure 9. Meshed tank and horizontal coils.

Discretization is a key element in deriving accuracy in the CFD process. It is vital to capture the features of the tank. A large number of elements have been introduced, which requires a longer computational time. The system that was used was 64bit Windows OS. The system contained 4 processors each operating at 2.66GHz. An added bonus to the computer system was the 16GB of RAM. However, it was not enough to proceed with iterative process. The solver produced an error new to the user. The computer system was running low on memory not allowing the DOE Fellow to reach completeness for this task.

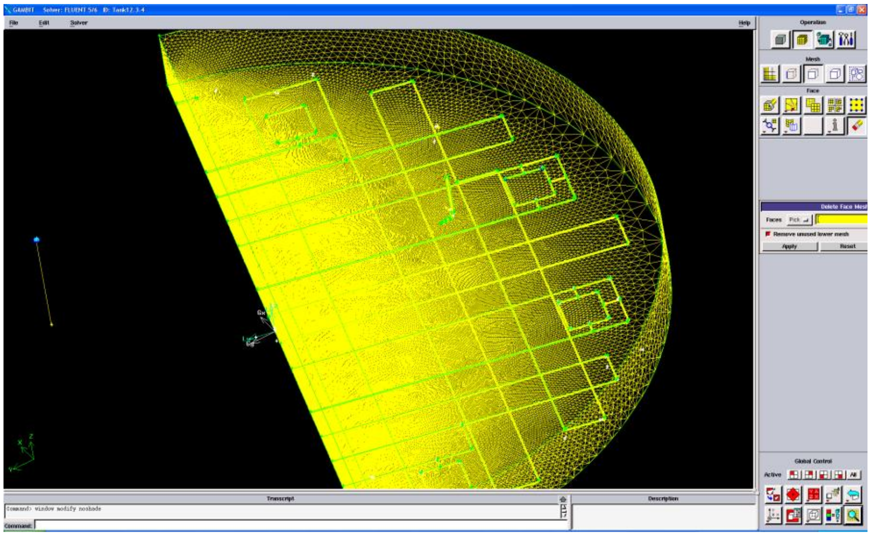

Figure 10 Reducing domain to half.

To cut down on the computation time, only half of the domain will be modeled and adding a periodic boundary condition at the surface of interaction between the haves will incorporate the effect that occurs in the modeled domain into the removed half.

5. CONCLUSION

The preliminary results that been acquired provide us with the confidence that the modeling steps that have been decided upon are favorable. However, it should be mentioned that the addition of many elements affects the computation time on incorporating the cooling coils in the modeling. The computer system that was used was not allowing for completeness in the following cases after Case 1. Now the challenge becomes reducing the meshing elements to a number that provides favorable results and that the computer would be able to compute. For this reason, before the model is completed, many coarse analyses would need to been reached to first before the final model is reached.

In terms of analyzing the system itself, after CFD validation of 0.2ft/s is achieved, consideration should be taken to analyze the effect of manipulating the orientation, rotating the nozzle, in which the fluid is ejected out of the nozzle. The system itself has the ability of being revolve. Perhaps after a series of numerous cycles, a new orientation should be introduced to enhance the eroding effect on the sludge.

Much of the work in CFD in still needed to be accomplished. A scope of work has been composition in accordance with NVE in efforts to continue to support the project. Future works include the optimization of jet nozzle to effect generate idea flow patterns to circulate the acid solution. During the investigation of enhancing the fluid mixing, consideration will be made to avoid the plume breaking at the surface.

6. REFERENCES

1. Computational Fluid Dynamics, CFD Online, http://www.cfd-online.com/.

2. Anderson, John D., Jr.; *Computational Fluid Dynamics: The Basic with Application*, McGraw-Hill, New York, 1995.

3. Patankar, Sushas V., *Numerical Heat Transfer and Fluid Flow*, Hemisphere Publishing Corporation, Washington, 1980.

Appendix A

```
3799 1.0225e-03 3.2272e-06 4.5201e-07 2.2725e-06 4.1185e-06 2.1087e-06 16:13:27 2232
3800 1.0709e-03 3.2229e-06 4.7719e-07 2.2807e-06 4.1274e-06 2.1085e-06 16:04:20 2231
iter continuity x-velocity y-velocity z-velocity          k    epsilon     time/iter
3801 1.0218e-03 3.2234e-06 4.5130e-07 2.2696e-06 4.1169e-06 2.1081e-06 16:11:49 2230
3802 1.0705e-03 3.2187e-06 4.7661e-07 2.2781e-06 4.1256e-06 2.1078e-06 16:02:51 2229
3803 1.0212e-03 3.2188e-06 4.5066e-07 2.2670e-06 4.1154e-06 2.1075e-06 16:10:28 2228
3804 1.0696e-03 3.2144e-06 4.7590e-07 2.2751e-06 4.1241e-06 2.1076e-06 16:01:36 2227
3805 1.0202e-03 3.2146e-06 4.4999e-07 2.2640e-06 4.1138e-06 2.1073e-06 16:09:17 2226
3806 1.0688e-03 3.2098e-06 4.7512e-07 2.2725e-06 4.1224e-06 2.1070e-06 16:00:29 2225
3807 1.0191e-03 3.2099e-06 4.4920e-07 2.2612e-06 4.1123e-06 2.1066e-06 16:08:12 2224
3808 1.0680e-03 3.2055e-06 4.7441e-07 2.2698e-06 4.1211e-06 2.1067e-06 15:59:28 2223
3809 1.0186e-03 3.2059e-06 4.4846e-07 2.2586e-06 4.1109e-06 2.1062e-06 16:07:12 2222
3810 1.0671e-03 3.2013e-06 4.7359e-07 2.2669e-06 4.1195e-06 2.1059e-06 16:05:54 2221
3811 1.0173e-03 3.2014e-06 4.4772e-07 2.2557e-06 4.1093e-06 2.1055e-06 16:04:47 2220
iter continuity x-velocity y-velocity z-velocity          k    epsilon     time/iter
3812 1.0657e-03 3.1966e-06 4.7293e-07 2.2643e-06 4.1180e-06 2.1053e-06 16:03:47 2219
3813 1.0162e-03 3.1968e-06 4.4706e-07 2.2531e-06 4.1074e-06 2.1052e-06 16:02:55 2218
3814 1.0647e-03 3.1921e-06 4.7193e-07 2.2617e-06 4.1165e-06 2.1051e-06 16:02:07 2217
3815 1.0159e-03 3.1927e-06 4.4627e-07 2.2504e-06 4.1065e-06 2.1044e-06 16:01:24 2216
3816 1.0645e-03 3.1881e-06 4.7160e-07 2.2586e-06 4.1148e-06 2.1044e-06 16:00:44 2215
3817 1.0153e-03 3.1882e-06 4.4581e-07 2.2477e-06 4.1046e-06 2.1041e-06 16:00:08 2214
3818 1.0630e-03 3.1842e-06 4.7072e-07 2.2556e-06 4.1134e-06 2.1038e-06 15:59:33 2213
3819 1.0138e-03 3.1846e-06 4.4511e-07 2.2443e-06 4.1029e-06 2.1034e-06 15:59:00 2212
3820 1.0628e-03 3.1796e-06 4.7014e-07 2.2534e-06 4.1116e-06 2.1035e-06 15:58:28 2211
3821 1.0135e-03 3.1803e-06 4.4442e-07 2.2420e-06 4.1014e-06 2.1030e-06 15:57:58 2210
3822 1.0624e-03 3.1751e-06 4.6936e-07 2.2509e-06 4.1104e-06 2.1027e-06 15:57:28 2209
iter continuity x-velocity y-velocity z-velocity          k    epsilon     time/iter
3823 1.0128e-03 3.1754e-06 4.4369e-07 2.2397e-06 4.0998e-06 2.1023e-06 15:56:59 2208
3824 1.0606e-03 3.1710e-06 4.6817e-07 2.2480e-06 4.1085e-06 2.1023e-06 15:56:31 2207
3825 1.0124e-03 3.1712e-06 4.4293e-07 2.2372e-06 4.0984e-06 2.1020e-06 15:56:03 2206
3826 1.0604e-03 3.1687e-06 4.6826e-07 2.2435e-06 4.1074e-06 2.1018e-06 15:55:36 2205
3827 1.0118e-03 3.1684e-06 4.4270e-07 2.2332e-06 4.0969e-06 2.1013e-06 15:55:09 2204
3828 1.0597e-03 3.1634e-06 4.6726e-07 2.2419e-06 4.1055e-06 2.1011e-06 15:54:42 2203
3829 1.0108e-03 3.1630e-06 4.4165e-07 2.2316e-06 4.0955e-06 2.1008e-06 15:54:15 2202
3830 1.0584e-03 3.1590e-06 4.6630e-07 2.2395e-06 4.1041e-06 2.1007e-06 15:53:48 2201
3831 1.0099e-03 3.1588e-06 4.4094e-07 2.2291e-06 4.0936e-06 2.1001e-06 15:53:22 2200
3832 1.0578e-03 3.1548e-06 4.6577e-07 2.2367e-06 4.1024e-06 2.1000e-06 15:52:56 2199
3833 1.0091e-03 3.1545e-06 4.4035e-07 2.2262e-06 4.0921e-06 2.0998e-06 15:52:29 2198
iter continuity x-velocity y-velocity z-velocity          k    epsilon     time/iter
3834 1.0571e-03 3.1508e-06 4.6511e-07 2.2340e-06 4.1011e-06 2.0996e-06 15:52:03 2197
3835 1.0087e-03 3.1506e-06 4.3970e-07 2.2235e-06 4.0902e-06 2.0990e-06 15:51:37 2196
3836 1.0560e-03 3.1462e-06 4.6425e-07 2.2315e-06 4.0990e-06 2.0987e-06 15:51:11 2195
3837 1.0078e-03 3.1460e-06 4.3894e-07 2.2210e-06 4.0889e-06 2.0986e-06 15:58:03 2194
3838 1.0550e-03 3.1421e-06 4.6348e-07 2.2288e-06 4.0975e-06 2.0983e-06 15:48:51 2193
3839 1.0071e-03 3.1421e-06 4.3826e-07 2.2182e-06 4.0871e-06 2.0976e-06 15:56:01 2192
3840 1.0542e-03 3.1383e-06 4.6283e-07 2.2256e-06 4.0958e-06 2.0976e-06 15:47:03 2191
3841 1.0060e-03 3.1386e-06 4.3770e-07 2.2149e-06 4.0855e-06 2.0974e-06 15:54:23 2190
3842 1.0537e-03 3.1350e-06 4.6242e-07 2.2224e-06 4.0942e-06 2.0970e-06 15:45:35 2189
3843 1.0052e-03 3.1347e-06 4.3710e-07 2.2119e-06 4.0837e-06 2.0964e-06 15:53:02 2188
3844 1.0529e-03 3.1301e-06 4.6167e-07 2.2203e-06 4.0925e-06 2.0963e-06 15:44:20 2187
iter continuity x-velocity y-velocity z-velocity          k    epsilon     time/iter
3845 1.0042e-03 3.1302e-06 4.3636e-07 2.2099e-06 4.0820e-06 2.0959e-06 15:51:52 2186
3846 1.0523e-03 3.1255e-06 4.6083e-07 2.2184e-06 4.0909e-06 2.0958e-06 15:43:13 2185
3847 1.0038e-03 3.1257e-06 4.3568e-07 2.2077e-06 4.0807e-06 2.0956e-06 15:50:48 2184
3848 1.0515e-03 3.1220e-06 4.6030e-07 2.2150e-06 4.0896e-06 2.0957e-06 15:42:12 2183
3849 1.0032e-03 3.1221e-06 4.3510e-07 2.2045e-06 4.0793e-06 2.0955e-06 15:49:48 2182
3850 1.0507e-03 3.1183e-06 4.5985e-07 2.2121e-06 4.0880e-06 2.0951e-06 15:41:14 2181
3851 1.0024e-03 3.1180e-06 4.3457e-07 2.2016e-06 4.0779e-06 2.0945e-06 15:48:51 2180
3852 1.0498e-03 3.1137e-06 4.5897e-07 2.2096e-06 4.0864e-06 2.0945e-06 15:40:19 2179
3853 1.0014e-03 3.1137e-06 4.3379e-07 2.1992e-06 4.0760e-06 2.0943e-06 15:47:55 2178
3854 1.0489e-03 3.1098e-06 4.5836e-07 2.2070e-06 4.0851e-06 2.0939e-06 15:46:40 2177
3855 1.0007e-03 3.1096e-06 4.3318e-07 2.1965e-06 4.0746e-06 2.0935e-06 15:45:34 2176
iter continuity x-velocity y-velocity z-velocity          k    epsilon     time/iter
3856 1.0481e-03 3.1056e-06 4.5773e-07 2.2044e-06 4.0834e-06 2.0934e-06 15:44:37 2175
! 3857 solution is converged
3857 9.9994e-04 3.1056e-06 4.3258e-07 2.1940e-06 4.0731e-06 2.0933e-06 15:43:45 2174
```

Figure 11. Screen shot of iterative process reaching convergence for after appropriate turbulence model was selected.

The column on the far left is the counter that of the iterations. This iteration is for the solution of the tank with the two nozzles in the simulation. It nearly reached 4000 iteration for convergence to be achieved. Other iterative runs were computer with much longer iterative runs, however incorporating a RNG k- ε model and manipulating the under-relaxation ratios, convergence was obtain. The second column before the far right side indicates the computation time each iteration was completed in the software internal clock. In this sample, the solution converged after 15 hours and 44 minutes after an interval set.

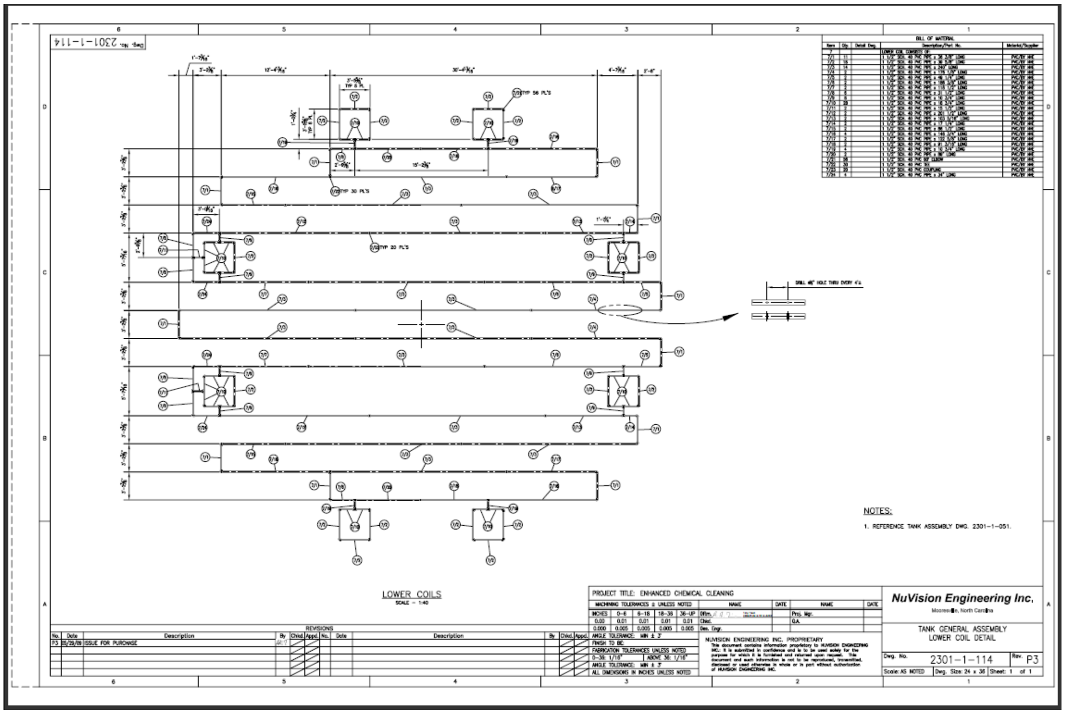

Figure 12. Lower cooling coils schematic.

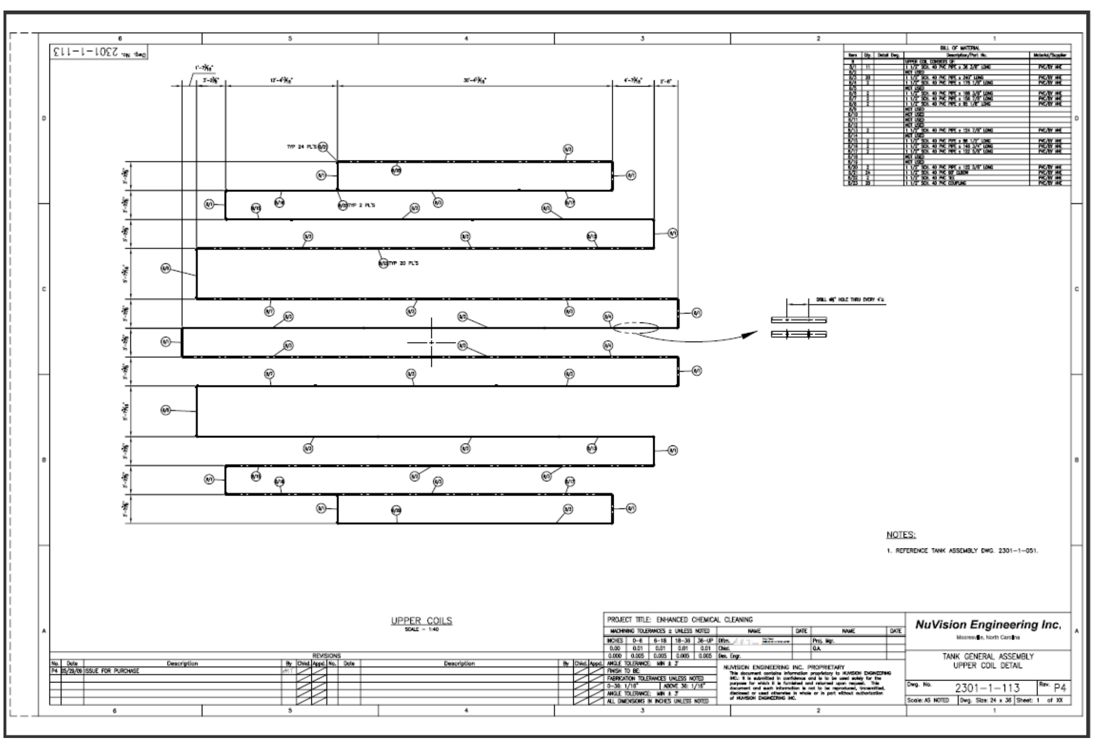

Figure 13. Upper cooling coils schematic.

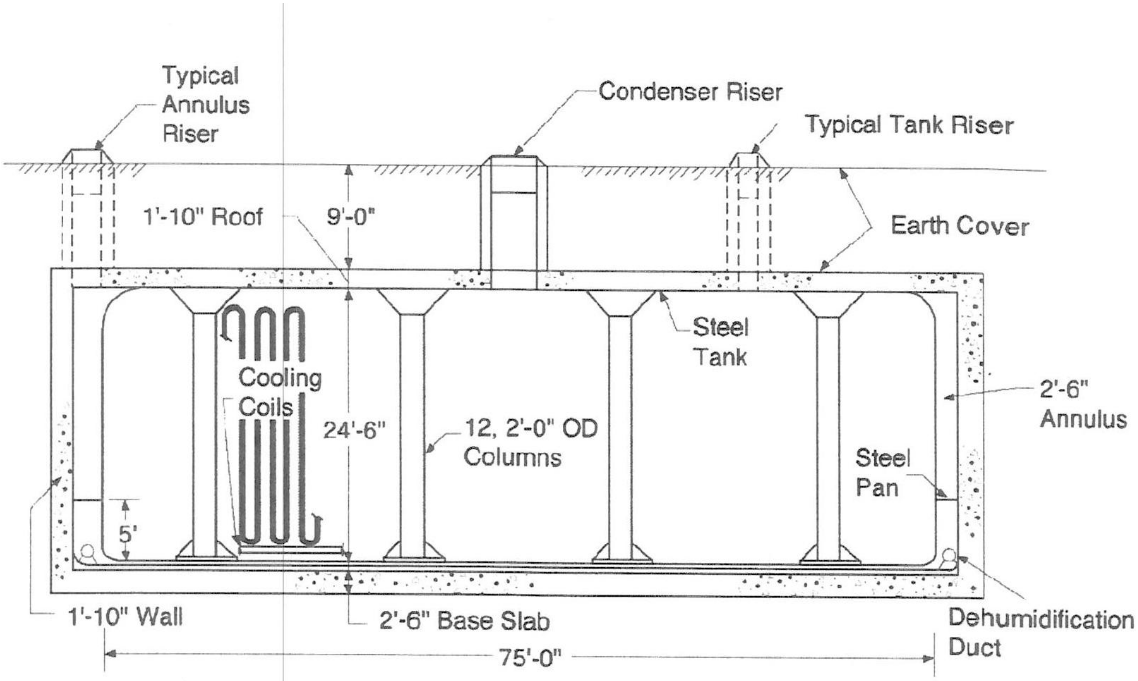

Figure 14. Typical SRS Type 1 tank.

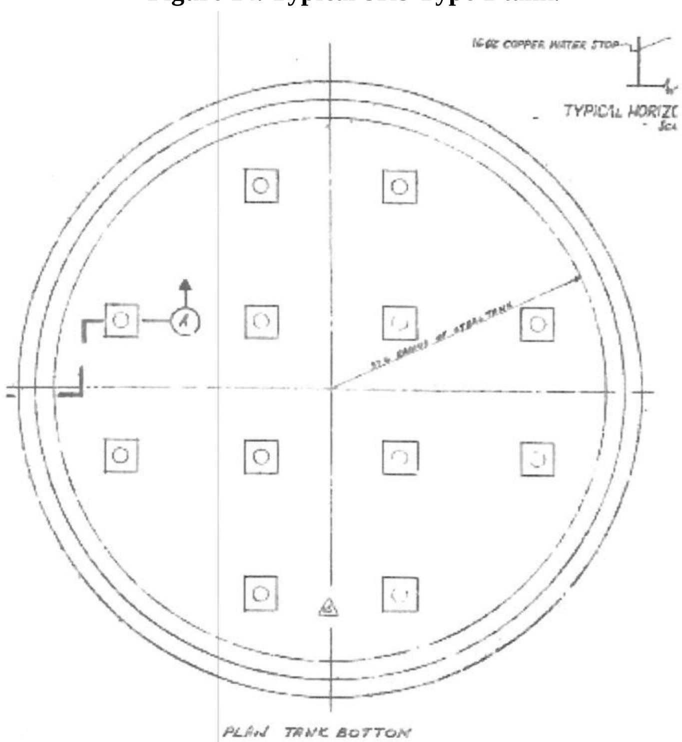

Figure 15. Typical SRS Type 1 tank, top view.