COMPUTATIONAL FLUID DYNAMICS ASSESSMENT OF EMERGENCY CORE COOLING SYSTEM CHECK VALVES AT COMANCHE PEAK

Brian M. Golchert
Westinghouse Electric
16W070 West 83rd St
Burr Ridge, IL 60527
Phone: 630-887-5239
golchebm@westinghouse.com

Thomas Loebig
Westinghouse Electric
1000 Westinghouse Drive
Cranberry Township, PA 16066
Phone : 412-374-4311
loebigtg@westinghouse.com

James Wyble III
Westinghouse Electric
Glen Rose, TX 76043
Phone : 254-897-5307

wyble3js@westinghouse.com

Andrea Lemons
Luminant-Comanche Peak Nuclear Power Plant
Glen Rose, TX 76043
Andrea.Lemons@Luminant.com

ABSTRACT

Both units at Comanche Peak have experienced chronic problems with check valves on part of their Emergency Core Cooling Systems (ECCS). These valves have seen ‘chatter’ and wear which has led to leakage from the ECCS accumulator through these valves. Recently, engineers at Comanche Peak took the pro-active approach to determine the root cause of this wear problem. Plant engineers, in consultation with Westinghouse, expected that the flow structure near and upstream of the check valves was the cause of the problem.

To determine the flow structures in a portion of the ECCS piping system, Westinghouse engineers performed a Computational Fluid Dynamics (CFD) assessment which was qualitatively used to examine the flow field in and around the check valve. A reasonable simulation was created and engineering judgment was used to analyze the flow system.

Analysis of the CFD results indicated that there is fairly significant swirl in the area of the check valve in addition to a pressure gradient across the check valve area. A CFD analysis with a new valve design using the original piping system showed that the flow instabilities with the new valve are markedly lower than with the original valve.

Comanche Peak has replaced the check valve on Unit 1. After six months of operation, this valve has performed well and shown no leakage. Comanche Peak plans to replace the check valve on Unit 2. This paper discusses the CFD analysis of the original check valve and the new check valve as well as the engineering judgment used to interpret the differences in the flow fields.

KEYWORDS
Engineering CFD assessment, flow induced problems, ECCS check valves

1. INTRODUCTION

Over the years, both units at Comanche Peak have experienced chronic problems with check valves on part of their Emergency Core Cooling Systems (ECCS). These valves have seen ‘chatter’ and wear which has led to leakage from the ECCS accumulator through these valves. Both plants have more than one ECCS train with slightly different piping configurations. Recently, engineers at Comanche Peak took the pro-active approach to determine the root cause of this wear problem and to see if there were cost-effective measures available to alleviate it. Plant engineers, in consultation with Westinghouse, expected that the flow structure near and upstream of the check valves was the cause of the problem with the check valves.

To assess the flow structure in and around the check valves, a CFD simulation of a portion of the D-loop Emergency Core Cooling System (ECCS) system for Comanche Peak Unit 1 was created. This model would be used to characterize the flow structure near the existing valves. Comanche Peak made the decision to investigate a possible replacement for the current check valve so a second simulation was created. To determine the flow structures in a significant segment of the ECCS piping system, CFD was qualitatively used to examine the flow field with the existing check valve and a potential replacement check valve for Comanche Peak Units 1 and 2. A ‘rigorous’ CFD analysis (grid sensitivity studies, parametric studies, etc.) was not performed. Instead, a reasonable simulation was created and engineering judgment was used to analyze the flow system to determine if the new check valve will experience a less deleterious flow field.

From the beginning, it was decided that this exercise would be an assessment rather than ‘analysis.’ As with most systems in a nuclear plant, providing ‘exact’ boundary conditions and flow rates was not practical. Also, most systems only have ‘bulk’ flow information and not local flow information to verify/validate a simulation. Given these restraints, it was decided from the onset that an engineering assessment using CFD was the most cost-effective method for making an informed decision on which valves would experience the least local flow variations. Engineering judgment was used to determine adequate meshes and boundary conditions. No attempt was made to compare any of the calculated flow results to onsite measurements.

To create the CFD simulations, nominal system operating parameters were used as inputs. The detailed geometry of the piping and the two valves (original check valve and the replacement check valve) were included in the simulation along with the throttling valve. In addition, the simulations were run transiently in order to qualitatively determine the temporal change in the flow field and to determine if the flow field with the new check valve is more stable than the flow field with the original check valve.

2. METHOD

A geometry of the D-loop for Comanche Peak Units 1 and 2 was created in SolidWorks. The existing piping configuration and check valve geometry were used for the model of the current system. A schematic of the original geometry is shown in Figure 1. For the projected new system, the existing geometry will be used with the potential new check valve (detailed geometry provided by the supplier). A schematic of the new geometry is shown in Figure 2. Figure 3 shows a cutaway of the proposed new check valve. The geometry for Unit 2 only has minor differences than the ones shown in Figures 1 and 2. These geometries were then imported into ICEM where the geometry was cleaned up for flow analysis. By ‘cleaned up,’ it is meant that the flow field was stopped at the face of the check valve. No metal was modeled. This is a simple piping system so there was not much cleanup involved. Specific plant drawings, plant information, and valve information used to create the geometry. Once the cleaned up geometries were completed, simple tetrahedron-mesh elements were built on the configuration and the meshes were exported into CFX. Tetrahedrons were used for the sake of expediency and schedule. Inputs provided by the plant were then used to determine boundary conditions and the code was run to convergence in steady-state (RMS values of pressure and momentum converged to 1.0E-04). The results from this simulation were post processed and analyzed.

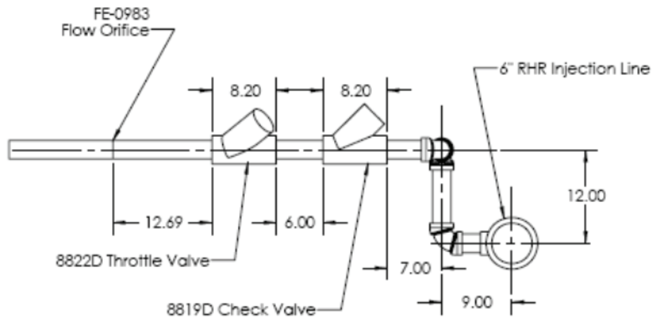

Figure 1: Schematics of piping system with original check valve for Unit 1 (units are in inches)

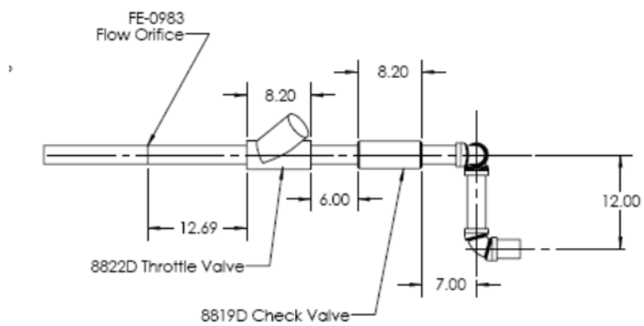

Figure 2: Schematic of piping system with proposed check valve for Unit 1 (units are in inches)

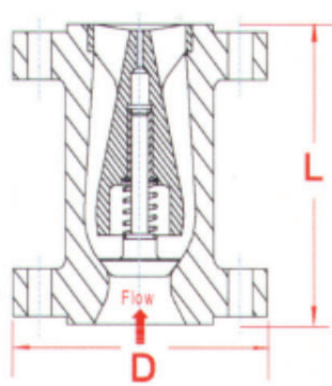

Figure 3: Geometry of new check valve

These simulations were then run as transients, using the steady-state results as initial conditions. The code was executed long enough to produce approximately 5-10 seconds of actual flow time. Due to the periodicity of the flow field, it was not necessary to run the transient any longer. Also, it should be noted that since only a portion of the piping system was being modeled and that pump pulsations were not included in the simulation, running the simulation until the periodicity appeared in the flow field was deemed sufficient. The results from the simulations were post processed and analyzed. The purpose of these transient simulations was to provide visual evidence as to the stability or instability of the flow field.

3. INPUT

From information provided by the plant, the following properties were used:

- Water at 100°F
- Pressure 1050 psi
- Flow rate of 600 $in^3/s = 0.3472\ ft^3/s$

These properties were used to determine inlet and boundary conditions for the CFX model. Note that prescribing the temperature and the reference pressure is sufficient for CFX to determine the local water density (i.e., water density is not an input).

From information provided by the plant, the piping is 2 inch schedule 160 with an inner diameter of 1.689 in = 0.1408 ft. This results in a cross sectional area of 0.0156 ft^2. Using the cross sectional area and the volumetric flow rate, one finds the inlet velocity to be 22.3 ft/s.

The 1050 psi pressure will be used as the reference pressure with zero relative pressure as an outlet boundary condition. No slip walls, an isothermal simulation, and the k-ε turbulence model with medium inlet intensity are the other inputs/boundary conditions used for the steady state cases. The k-ε turbulence model was chosen since the flow field is not dominated by many local flow vorticies. Except in the region near the check valves, the flow exhibits few recirculation zones. In as this was an applied engineering study, it was deemed appropriate to use the k-ε turbulence model.

4. UNIT 1 ANALYSIS

The computational model was built for the current Unit 1 configuration based on Figure 1. This model has ~640,000 tetrahedral cells. The grid was refined in regions of interest (such as in the check valve) but a significant effort was not made to create a very good mesh. As a check, a much larger model (~2,500,000 tetrahedral cells) was built. Since this work was done on a limited schedule, no effort was made create boundary layers. For the smaller mesh, the volume average cell size is 1.15E-6 ft^3. The largest cell volume is 2.13E-6 $ft^{3.}$

For both models, the transient case was run with high resolution advection and first order turbulence numerics. A maximum of 10 coefficient loops were used for each time step. Timesteps of 0.001 seconds were used and this resulted in an average Courant Number of 1.0. Both models provided the same relative flow distributions except that the detailed model more clearly showed the recirculation effects. By relative flow distribution, it is meant that the general flow field and velocity magnitude did not vary much (less than 5%) between the two grids.

The configuration modeled includes a portion of the downstream T junction. The focus of this analysis was on the flow field in the region of the check valves. Figure 4 shows the flow in the region around the check valve. There are portions of the piping system that will see high fluid velocities, mostly near the inlet orifice. The flow field in Unit 1 is much better behaved than the flow that was shown in Unit 2 (as detailed later in this paper).

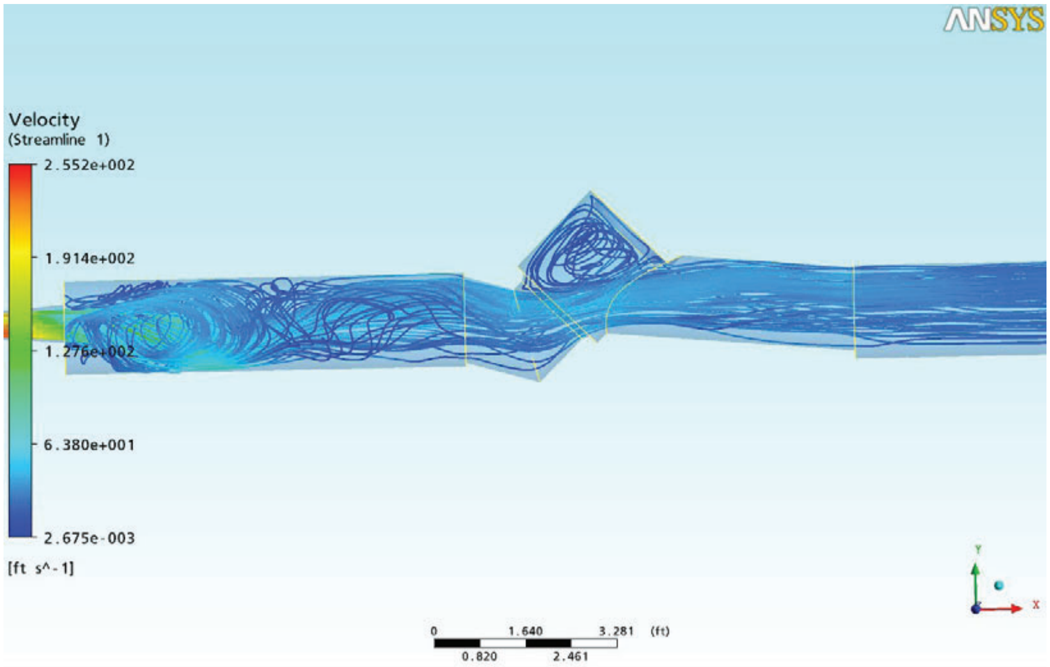

Figure 4: Velocity distribution and flow field in/near check valve

Because of the relative configuration of the valves, there is less vorticity going through Unit 1's check valve as opposed to Unit 2's check valve. However, there is still a distinct pressure gradient in the region of the original check valve (see Figures 5 and 6) which may eventually cause some damage to parts of the valve. Minimizing or smoothing the flow field (reducing swirl) and pressure distributions would increase the performance of the check valve. This minimization of the forces and swirl will reduce the wear on the check valve parts. Figures 5 and 6 show relative pressure distributions so no scale has been included.

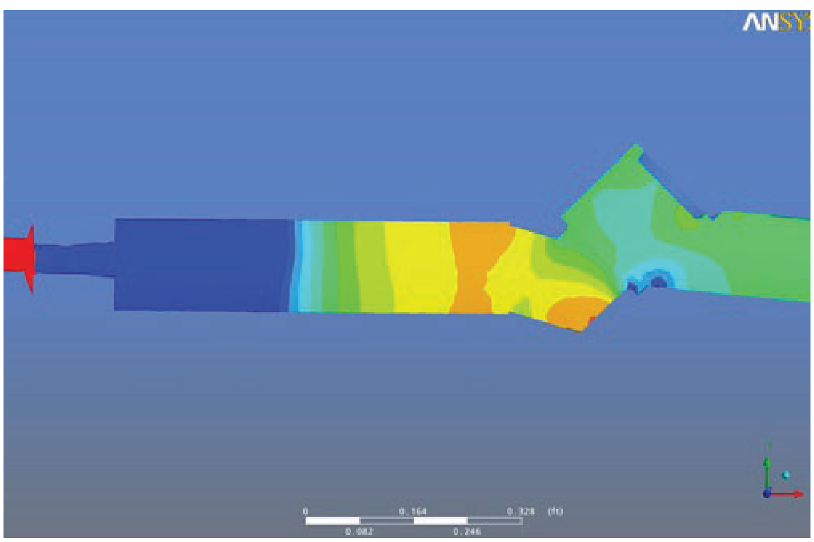

Figure 5: Pressure Distribution from the Side for Unit 1 Original Check Valve

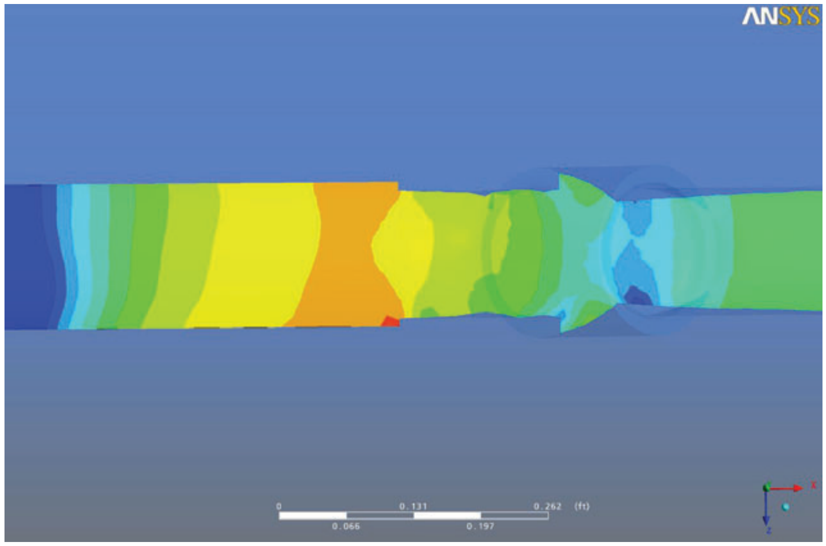

Figure 6: Pressure distribution from the top for Unit 1 original check valve

The computational model was built for the proposed Unit 1 configuration with the new check valve based on Figure 2. This model has ~490,000 tetrahedral cells. This model is slightly smaller than the previous model since the downstream T portion of the piping configuration was not included. Figure 7 shows the pressure distribution and the flow in the region around the check valve. The pressure and flow fields in Unit 1 with the new check valve are much better behaved than the flow with the original check valve.

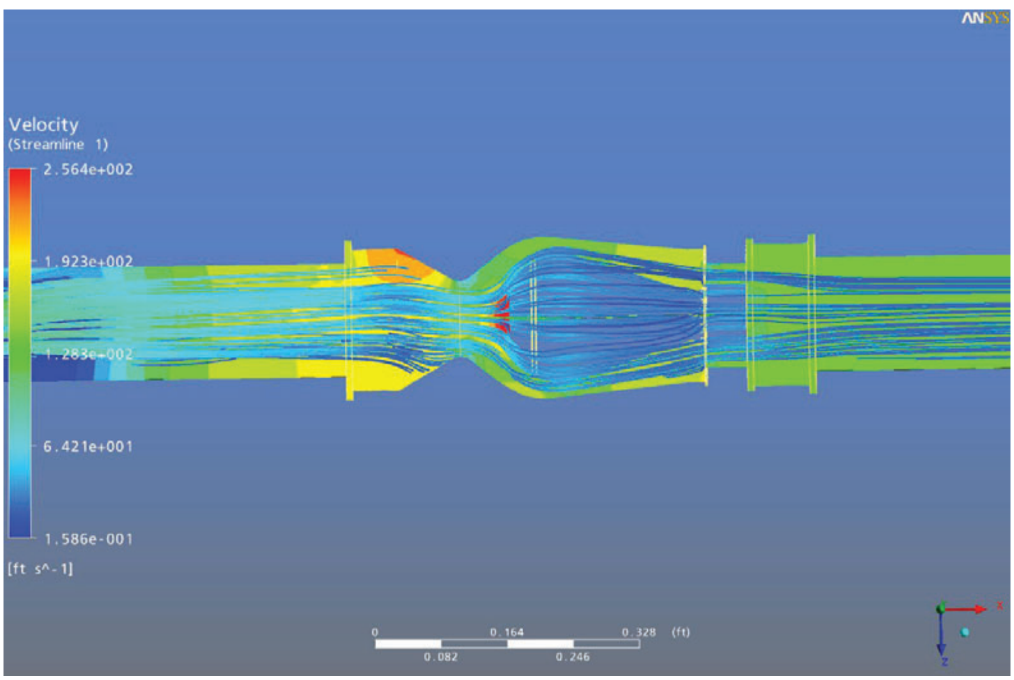

Figure 7: Flow field and pressure distribution for new check valve from the top

A closer examination of the pressure distribution in the check valve is shown in Figure 8. This view is looking in the direction of the flow. The pressure is higher in the center since this plane is at the tip of the check valve where the fluid begins to spread. Figure 8 shows that there is a slight asymmetry in the pressure field and this is expected since the flow field entering the check valve is not fully developed. The variation in this pressure distribution is much less than the variation of the pressure distribution of the original check valve.

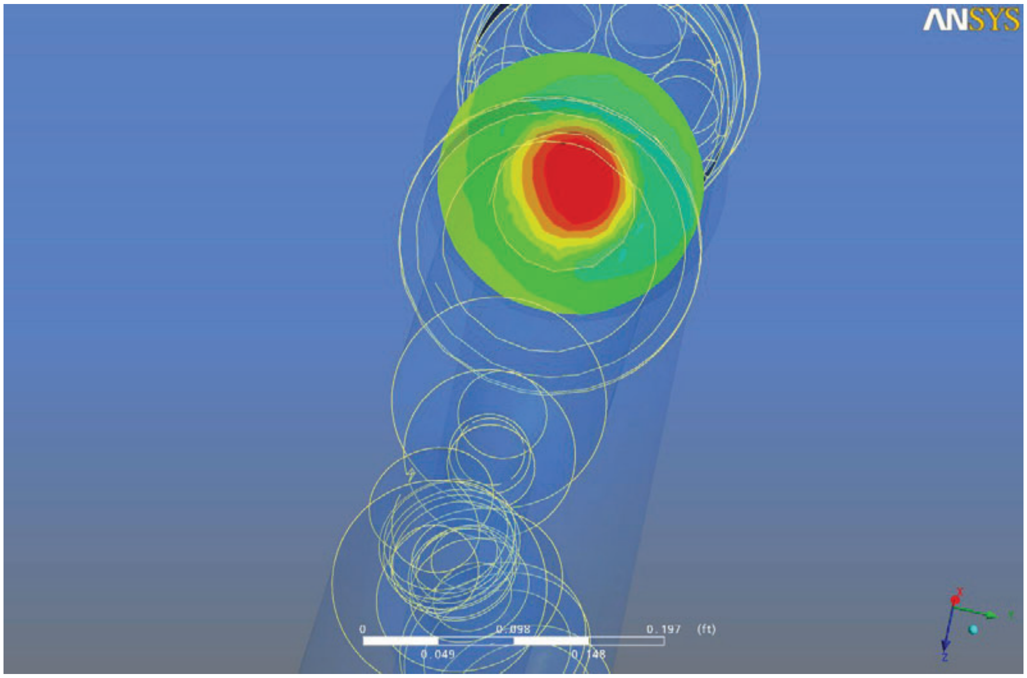

Figure 8: Pressure distribution across check valve

As a further check, the pressure distribution in the check valve was examined as one progresses downstream into the valve. All of the pressure distributions were relatively uniform. A severely non-uniform pressure distribution would impart an uneven force on the check valve which would lead to poor performance.

The simulation with the new check valve on Unit 1 was then run in a transient mode so that ~10 seconds of actual flow time was modeled. Care was taken so that the Root Mean Squared (RMS) Courant Number did not exceed 20. This implied very small computational time steps (~5.0E-03 s) which led to long computation times. Figures 9 and 10 show two pictures from different times during the transient evaluation. Both pictures have the maximum velocity capped at 100 ft/s in order to more graphically show the flow field.

As can be seen from the left hand portions of both pictures, the flow upstream of the check valve is highly turbulent and swirling. In Figure 9, the main flow is directed downward exiting from the orifice while in Figure 10, the main flow is directed upward. This swirling is a result of the geometry of the piping system. Regardless of the flow direction entering the check valve, the flow distribution inside the check valve remains relatively constant. This implies less motion on the check valve and better performance.

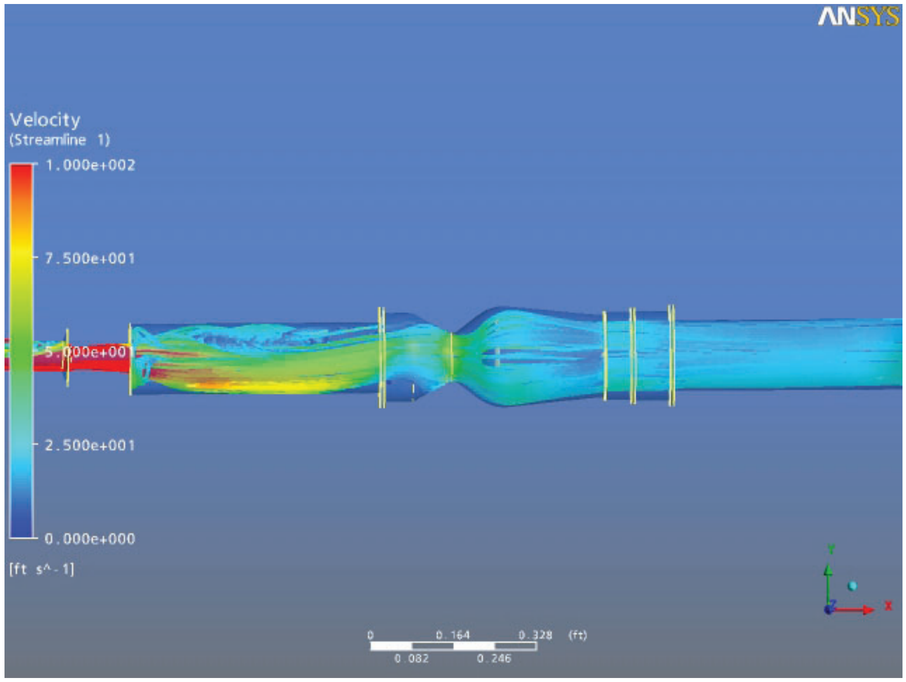

Figure 9: First time step picture from transient simulation for Unit 1

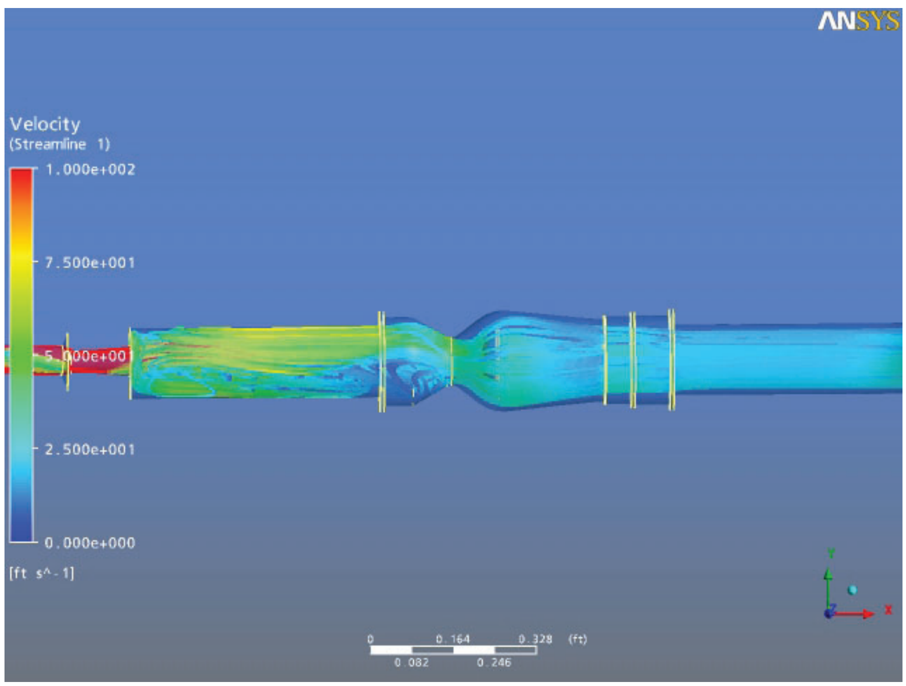

Figure 10: Second time step picture from transient simulation for Unit 1

5. Unit 2 Analysis

A similar analysis was performed for Unit 2. For this analysis the downstream geometry was extended slightly longer than the geometry for Unit 1. The results from the Unit 2 simulations show that replacing the check valve will reduce the potential for oscillations and valve chatter. Figure 11 shows the flow field for the original configuration (velocity was capped to highlight the effect of the flow field). Figures

12 and 13 show the pressure and flow field through the new check valve for Unit 2. The pressure and flow distributions inside of the check valve are significantly more uniform with the new check valve even though the same turbulent flow field upstream exists.

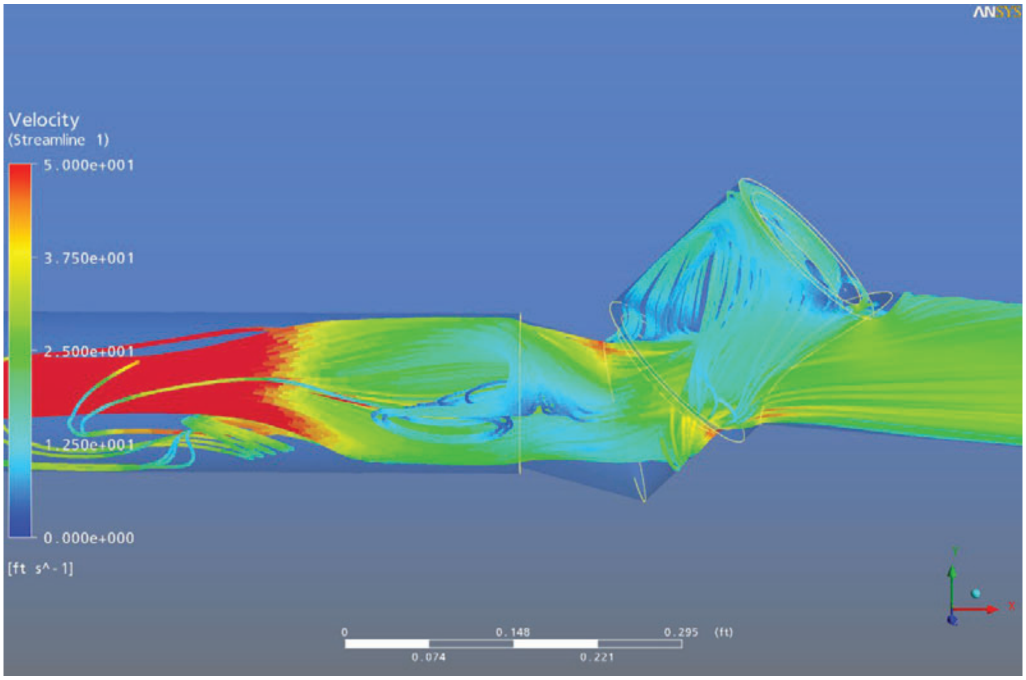

Figure 11: Flow field through existing Unit 2 check valve

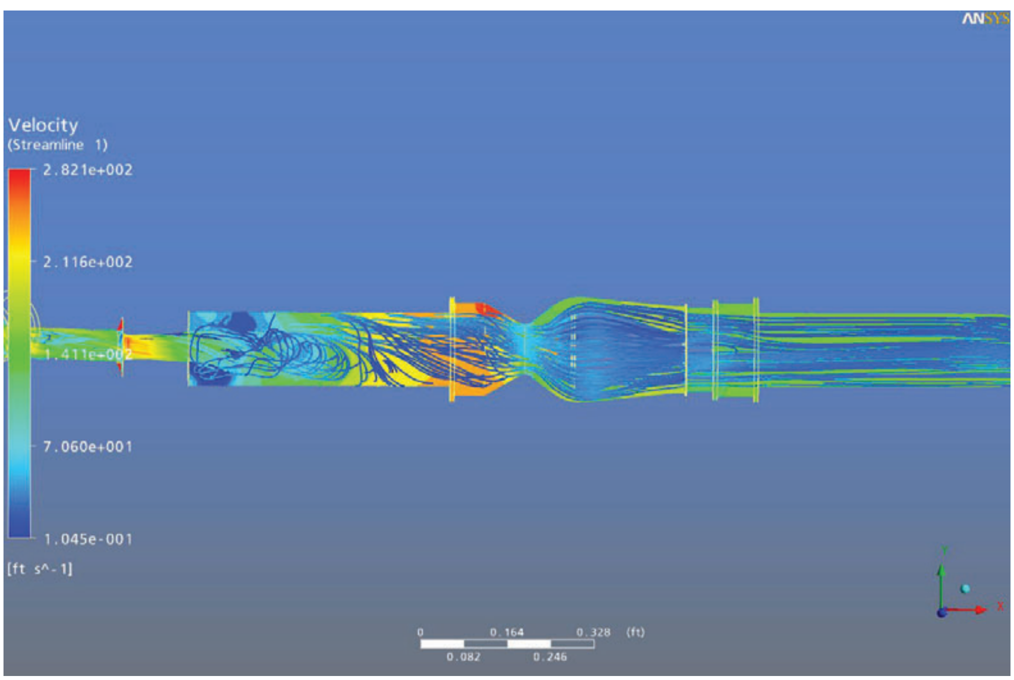

Figure 12: Pressure distribution and flow field for Unit 2 with new check valve (side view)

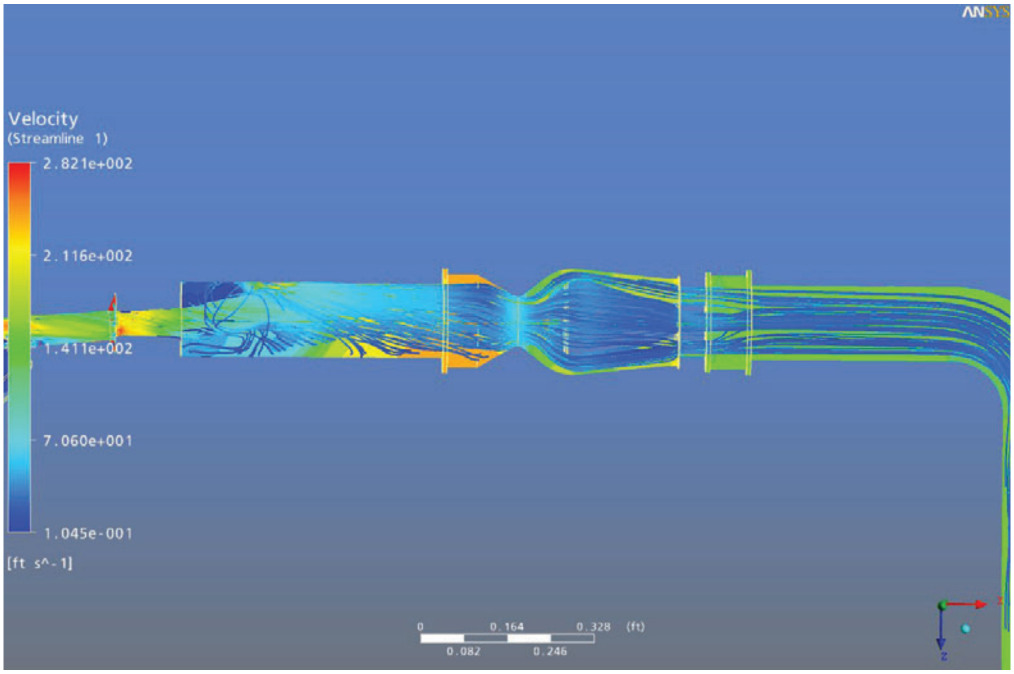

Figure 13: Pressure distribution and flow field for Unit 2 with new check valve (top view)

5. CONCLUSIONS

Analysis of the CFD results for both Units 1 and 2 indicate that there is a significant swirl in the area of the original check valve, in addition to a pressure gradient across the check valve area. These two facts could lead to local flow instabilities that may result in flow induced vibrations in the check-valve seat. The CFD results for both units with the new check valve in place indicate there are much fewer flow instabilities and a much more uniform pressure distribution across the new check valve.

The transient calculation for both units with the original check valve show that while the main flow field remains somewhat constant, there are portions of the flow field near the original check valve that significantly vary with time (the flow swirls). Replacement of the check valves in both units show reduced swirl and variation in the flow field. From these results, it is expected that the new check valves will have better performance than the existing check valves.

Based in part on this information, the plant replaced a check valve in Unit 1. After several months of operation, the plant found indication of leakage and plans to replace the check valves in Unit 2 as well.

ACKNOWLEDGMENTS

The authors would like to acknowledge the assistance and cooperation provided by the Luminant employees during this analysis. Their help was greatly appreciated.