CI TOC

Proceedings of
ASME TURBO EXPO 2002
June 3-6, 2002, Amsterdam, The Netherlands

GT-2002-30149

A NEW TAILPIPE DESIGN FOR GE FRAME TYPE GAS TURBINES TO SUBSTANTIALLY LOWER PRESSURE LOSSES

Richard Golomb
Cheng Fluid Systems
480 San Antonio Rd., Suite 120
Mountain View, CA 94040

Dr. Vivek Sahai
Cheng Fluid Systems
480 San Antonio Rd., Suite 120
Mountain View, CA 94040

Dr. Dah Yu Cheng
Cheng Fluid Systems
480 San Antonio Rd., Suite 120
Mountain View, CA 94040

ABSTRACT

Many GE frame gas turbines have a unique 90-degree tailpipe exhaust system that contains struts, diffusers, and turning vanes. As confirmed in a recent report by GE and other authors [1], it is known in the industry that this tailpipe design has large pressure losses. In this recent report a pressure loss as high as 60 inches of water (0.15 kgs/sqcm) was cited. Due to the flow separations they create, the report indicates that the struts can cause very high-pressure losses in the turbine. The report also states that these pressure losses can vary with different turbine load conditions. Cheng Fluid Systems and Cheng Power Systems have conducted a study aimed at substantially reducing these pressure losses. Flow control technology introduced to the refinery industry, i.e., the Cheng Rotation Vane (CRV) and the Large Angle Diffuser (LAD) can be used to mitigate the flow separation and turbulence that occurs in turns, bends, and large sudden expansions. Specifically the CRV addresses the flow separations in pipe turns, and the LAD addresses the flow problems that occur with large sudden expansion areas. The paper will introduce the past experience of the CRV and LAD, and will then use computer simulations to show the flow characteristics around a new design. First, the study meticulously goes through the entire GE exhaust system, starting with the redesign of the airfoil shape surrounding the struts. This new design has a larger angle of attack and minimizes the flow separations over a much wider operating range. Second, the pros and cons of the concentric turning vanes are studied and it is shown that they are more flow restrictive, rather than flow enhancing. Third, it is shown that the highly turbulent rectangular box type exhaust ducting design, substantially contributes to high noise levels and pressure losses. In this paper a completed design will be shown that incorporates a new airfoil shape for the struts, and by using CRV flow technology in combination with the LAD flow technology, the pressure recovery can be enhanced. If the pressure losses could be reduced by 40 inches of water (0.10 kgs/sqcm), the turbine efficiency could be increased by 5 %, and the power output could be increased by 6 %.

INTRODUCTION

With the current need for more efficient power generation cycles, the designs of many gas turbine exhaust systems are being examined in order to try and improve the turbines performance. An improvement in the exhaust system can lead to lower pressures at the turbine outlet. This lower pressure then translates into the gas turbines higher output, efficiency, and lower heat rate. It is the achievement of this goal that has led us to develop an improved design for the GE frame engine gas turbine exhaust system.

The current design of most GE Frame Type Gas Turbines have a unique 90 degree turn in their tailpipe exhaust system, that contains struts, turning vanes, diffusers, and a rectangular exhaust plenum. The schematic of this system showing the exhaust gas path is shown in Figure 1. After the gas exits the last stage of the turbine, it enters into an expanding diffuser area. The gas then passes over a series of struts. Inside the struts are posts that help transmit the weight load to the journal bearing. Downstream of the struts the gas continues to slow down in the diffuser, and then it makes a sharp 90-degree turn into the exhaust plenum, guided into this area by a series of concentric turning vanes. These turning vanes are used to help deflect the flow from the axial to the radial direction. The gas flow then rapidly expands into the rectangular shaped exhaust plenum, which is located immediately downstream of the turning vanes.

It is known in the industry that this type of tailpipe design causes separated flows, and flow turbulence, which can account for higher-pressure losses, high noise levels, and decreased turbine efficiency. As confirmed in a recent report by GE and other authors [1], pressure losses as high as 60 inches of water (0.15 kg/sqcm) were cited. The report indicated that the position and design of the struts cause very large pressure losses, due to the flow separations that they create. In a GE Frame Gas Turbine, as the swirling gas leaves the turbine and passes through and over the struts (these struts were not designed to account for any swirling flow), they separate and the flow turbulence increases. Downstream of these struts the gas then makes a sharp 90 degree turn out of the turbine, and the upper half of the gas is guided through concentric turning vanes into the exhaust plenum, while the lower half of the gas has to find its own flow path into the exhaust plenum. The report showed that these concentric turning vanes account for another pressure drop within the exhaust system, and they further increase the flow turbulence [1]. Finally, the exhaust gas as it passes through the turning vanes, and enters into the sudden expansion of the exhaust plenum, is subject to further separation, reverse flows, and increased turbulence. The report showed that these losses can vary widely under different turbine load conditions [1].

Cheng Fluid Systems and Cheng Power Systems have conducted a study aimed at substantially reducing these pressure losses and improving the exhaust system performance. This can be accomplished by using rotation vane technology and source and sink concepts to design an advanced diffuser. This flow turbulence control technology has been introduced to the refinery and chemical industry almost a decade ago [2]. The flow turbulence control technology can mitigate the flow separation and turbulence caused by turns, bends, and large sudden expansions, and can reduce the large pressure losses associated with these undesirable flow patterns [3].

From this previous study of the current GE tailpipe system design, four different areas were identified where flow improvements could be made. These areas are the strut areas, the 90-degree turn area, the concentric turning vane area, and the sudden expansion into the rectangular plenum area. All four of these areas in the current design produce turbulence in the flow that cause noise, vibration, and flow separation, all of which result in a great loss of total pressure. This high-pressure loss also reduces the expected turbine power output. By making certain design modifications and using the existing rotational transformation flow technology (CRV) and sudden expansion control technology (LAD), this excessive flow turbulence and pressure loss can be minimized, and the power output and efficiency of the turbine can be increased [3].

NOMENCLATURE

CRV	Cheng Rotation Vane
LAD	Large Angle Diffuser
ISO	International Standards Organization
Dh	Hydraulic Diameter (Dh=4A/P)
A	Cross Sectional flow Area
P	Wetted Perimeter
GE	General Electric
CFD	Computation Fluid Dynamics
STAR-CD	Software Used To Analyze The Strut Design
k-ε	Industry Standard Turbulence Model Used In STAR-CD Calculations

OBJECTIVE OF NEW DESIGN

To make a GE Frame Gas Turbine more efficient, an advanced combined cycle such as the “Advanced Cheng Cycle” can be used [4]. But high-pressure losses already associated with the current tailpipe design will increase even more; when an advanced combined cycle is used. Therefore to make an advanced combined cycle work at its highest efficiency, it is absolutely necessary to reduce and keep to a minimum the pressure losses occurring in the gas turbines exhaust system. Based upon a GE Frame cycle deck calculation, if the pressure losses are reduced by 40 inches of water (0.10 kgs/sqcm), the turbine efficiency can be increased by up to 5.8%, and the power output can be increased by up to 6.2% (at ISO conditions). To accomplish this, design improvements will have to be made to the struts, 90-degree turn, turning vanes, and sudden expansion areas of the exhaust tailpipe, to try and minimize these pressure losses, increase the flows, reduce the noise and vibration, and increase the turbines efficiency. The new design concepts are described below. They show how flow improvements can be made in the areas of the struts, 90-degree turn, turning vanes, and sudden expansion.

IMPROVEMENT OF FLOW IN THE STRUT REGION

The current strut design consists of a metal sheet covering a post and it has been shown that this design causes flow turbulence which leads to high pressure losses. The shape of the struts show an airfoil design with the following characteristics: a thickness to cord length ratio of 18%; the strut is positioned at a 15 degree angle with respect to the turbines axis; and the strut has a chord length to space between the tips ratio (known as solidity) of 1. (A high value of solidity increases the flow interaction between struts and thus creates a flow channel for the exhaust gas to go through.) If the solidity of the struts can be increased (which can be achieved by either increasing the chord length or decreasing the spacing between the struts), then the flow interaction between the struts can be increased and the angle at which "stall" occurs is delayed [5].

The computation fluid dynamics code STAR-CD software was used to analyze the strut design, in terms of effectiveness in controlling the different swirl angles, and minimizing the drag and flow separation effects. This code is used to calculate the flow field around the struts and in a large expansion, using the industry standard Reynolds number k-ε model. This turbulence model was chosen based on the wide experience the industry has had with this model and software.

As the flow exits the last stage of the turbine it has a swirling flow angle. The angle that the flow develops depends on the load conditions. For the Frame 9E gas turbine this angle can vary anywhere from -50° to 14° [1]. (Positive swirl angles are clockwise and negative swirl angles are counter clockwise.) The swirling flow also leads to a flow separation and turbulence at the surfaces of the struts.

As the gas flows from the turbine blade area at full speed and full load conditions (See Figure 2a), it passes through and over the current designed struts; the struts create a flow turbulence that is applied at right angles to the turbines axis. This flow turbulence leads to a flow separation from the outside walls of the struts. When this happens, the flow accelerates due to the reduction in the core-flow cross sectional area, and this causes a high-pressure drop and flow non-uniformities to occur. The CFD results also show that the current strut design removes some of the swirl from the flow.

Several airfoil designs [6] were considered to improve the flow in this section. Each airfoil design was analyzed to determine which of their geometric shapes had the maximum angle of attack and lift characteristics, before stall occurred. Figure 2b shows the flow through a set of newly designed struts. These struts have the same maximum thickness as the original struts; and are positioned at a 15-degree angle with respect to the turbines axis. The new struts are 1.5 times longer (in the axial direction) than the old design and they will fit in the current design area, thus increasing the solidity by 50%. With this new design, the flow separations off of the surfaces of the struts and their trailing edge are minimized. This new strut design also preserves the swirl in the flow and it will be shown in the next section that the swirl in the flow can also be beneficial in helping the exhaust gas navigate the 90-degree sharp turn into the exhaust plenum.

IMPROVEMENT OF FLOW AROUND THE 90-DEGREE TURN

When a fluid flows through an elbow or curved conduit, the flow stream next to the outer radius of the turn must travel a longer distance than the flow stream near the center of the turn. When this happens, the pressure distribution within the flow becomes uneven and the fluid separates into numerous secondary flows that separate, swirl, rotate, and contain accelerated and reverse flows. Thus, the velocity pattern at the exit of the elbow becomes turbulent. This is the reason why a curved conduit will produce a greater pressure loss in a turn, than a straight section of pipe.

Rotation vanes are specifically designed so that they impart a gyroscopic motion to the flowing fluid that counteracts the gyroscopic motion in the fluid that is produced by the 90-degree turn. The guideline for the gyroscopic rotation is designed to make all the flow stream lengths constant throughout the pipe turn. This will allow the fluid to negotiate the 90-degree turn and exit the turn with a flat uniform velocity profile. The benefits of having a flat uniform velocity flow profile at the exit of the 90-degree turn are; elimination of most of the pressure drop through the 90-degree turn; no fluid turbulence; less noise; less vibration; and no fluid separation. The fluid rotation vanes turn the 90-degree turn into the equivalent of a straight length of conduit.

Experimental observations have shown how a flow rotation can remove fluid separation caused by a 90-degree turn. Water and air were used to verify the performance of the rotation vanes. Figure 3 shows water flowing in a piping system with test dyes in the flow stream. This flow visualization test was conducted in a 6 inch diameter (15.24cm) elbow constructed of clear plastic. The flow in the figure is from left to right. Colored dyes were introduced through orifices upstream of the elbow entrance, so that the flow streamlines could be traced. Without rotation vanes there is fluid separation, turbulence, reduction of flow area, and a mixing of the dyes. Separation occurs off the inner and outer walls. To remove this flow turbulence and separation, a set of special designed rotation vanes are placed at the inlet of the elbow. With the rotation vanes there is a uniform flow at the end of the 90-degree turn without any inner mixing of the dyes and any flow turbulence. Rotation of the fluid has stopped at the exit of the elbow. Manometer measurements of the elbows wall static pressure distribution were also taken. The pressure readings showed an uneven pressure distribution when no rotation vanes were used, and that the inner radius of the elbow had a region of lower static pressure, while the outer radius of the elbow had a region of higher static pressure distribution. When rotation vanes were used the pressure was distributed evenly throughout the entire 90-degree turn. Figures 3 and 4 show the benefits of using vanes to pre-rotate the fluid. The rotation allows the innermost stream to be routed to the outer extreme at the elbow exit and at the same time the outermost stream is routed to the innermost layer. The pre-rotation results in equal length streamlines and removes the problem of fluid particle acceleration.

Figure 4 shows the measured cross-stream velocity profiles for an elbow in which air was used as the fluid in a 2-inch diameter (5.08cm) short radius elbow, at a Reynolds number of 100,000. Measurements of the flow were made at the exit of the elbow and at five additional positions each 0.25 diameters downstream from the proceeding position. For the case without rotation vanes, a separated flow can be seen at the inner wall and at 0.25 diameters downstream. A strong velocity gradient is also seen downstream of the elbow with the highest velocities near the outer wall. The velocity gradient diminishes slightly because of the mixing of the fluid, but is still evident at 1.25 diameters downstream of the elbow. The velocity profiles with the rotation vanes show that no separated flow exists and the flow at 1 diameter downstream of the elbow is nearly uniform. These same elbow flow results are obtained when the turning vane technology is used on the inside of a vertical pump at the pumps ninety degree turn discharge area. The turning vanes have the vertical pumps drive shaft going through its center, just as the drive shaft for the turbine would be positioned.

The struts will be redesigned around the turbines drive shaft, using the rotational vane concept to keep the positive swirling flow that exists at full load conditions. This flow rotation will neutralize the swirl produced by the 90-degree turn, resulting in a uniform, constant velocity, non-turbulent flow throughout the 90-degree turn.

REDESIGN OF THE CONCENTRIC TURNING VANES

Downstream of the struts the exhaust gas is guided by seven concentric turning vanes (which are part of the gas turbines exhaust framework), as it enters the sudden expansion of the exhaust tailpipe. These seven turning vanes are flow restrictive, in that they guide the gas from the back wall of the tailpipe exhaust ducting system and into the back end of the sudden expansion of the tailpipe exhaust ducting. This causes additional flow turbulence and pressure losses, as the turning vanes are not aligned with the flow. By removing all of the interior turning vanes and shaping the sharp inner corner of the exhaust tail pipe to act as an elbow, and repositioning and reshaping the outer most turning vane to conform to the flow path of the outer walls, the flow turbulence can be reduced by a large amount. This was confirmed by calculating the hydraulic diameter ($D_h = 4A/P$) (4 times the cross sectional flow area (A) divided by the wetted perimeter (P)) of the seven concentric turning vanes and the hydraulic diameter of the new design. The calculations showed that the effective flow area of the new design could be increased by 94%, when the interior vanes are removed and the corners are reshaped.

REDESIGN OF EXPANDING FLOW AREA INTO EXHAUST DUCT

As the exhaust gasses leave the turbine and enter the sudden expansion of the exhaust plenum, excessive flow separation and back mixing occur. Because the current design forces all of the exhaust gases to the back end of the tailpipe, this leads to large flow separations, reverse and accelerated flows, high flow turbulence, high pressure drops, and increased noise, as the gases reach the sudden expansion. To control this flow turbulence without excessive pressure losses, a maximum expansion angle of 15° must be maintained. In most ducting systems including the exhaust ducting for the frame gas turbine systems, this would require an extremely long length of space. Current sudden expansion control technology can correct this problem by using overlapping truncated cones, positioned so that their common focal point is in the center of the smaller originating area (the source). This design is based upon a mathematical point source method, which will cause the exhaust gas to act like it was being issued from a central point in a conical pattern of the desired expansion angle. Figure 5 shows computer simulation results using STAR-CD, showing flow turbulence and back mixing in a sudden expansion area without any cones, and a complete uniform, constant velocity, non-turbulent flow when cones are used. To achieve this will require that this point source concept be placed in the immediate expansion area after the 90-degree turn, before the rectangular exhaust plenum area. Experimental tests have shown that if the theoretical point source of the truncated cones is located within a solid body, it will still produce a uniform constant velocity at the exit of the cones. The resulting benefits of using a sudden expansion control design for this area of the tailpipe will be, a uniform flow of gas, with a constant velocity, and with maximum pressure recovery within the expansion.

CONCLUSIONS

This paper has analyzed the current GE Frame 7 exhaust design system. Four areas were identified where large pressure losses can occur. These are the strut areas, 90-degree turn area, concentric turning vane area, and the sudden expansion into the rectangular plenum area. Four changes to the design have been suggested to improve performance in these areas. By making these four changes in the tailpipe design of GE Frame Gas Turbines (see Figure 6), the flow turbulence can be minimized, while pressure losses can be recovered, and turbine efficiency can be increased. This can be accomplished by creating a new airfoil shape for the struts, which improves their aerodynamic characteristics, and by using rotational flow technology to minimize the flow separations and preserve the flow swirl as the gases enter into the 90 degree turn of the exhaust plenum. Also, by eliminating the interior concentric turning vanes and modifying the internal and external corners of the exhaust system, the flow will remain uniform as it makes the 90 degree turn and enters the exhaust plenum. By changing the outer diffuser walls to form an elbow, a flow path for both the upper half and lower half gases can be defined. This elbow will fit into the existing exhaust space and can be easily removed to allow for servicing. Finally, by modifying the transition ducting from the exhaust tailpipe to the exhaust ducting using the sudden expansion control technology, the gas flow can be kept uniform as it enters the large expansion of the rectangular exhaust duct. By making these changes in the exhaust tailpipe design, total pressure recovery will be enhanced and current pressure losses in the tailpipe can be reduced.

REFERENCES

1. Sultanian, B.K., Nago, S., Sakamoto, T., 1999, "Experimental and Three-Dimensional CFD Investigation in a Gas Turbine Exhaust System," In Transactions of ASME, Journal Of Engineering For Gas Turbines And Power, ASME Transactions, Volume 121, No.2, pg. 364-374.

2. Kosla, L., and Mutsakis, M., 1992, "New In-Pipe Flow Conditioner Cuts Fluid Problems," Chemical Engineering Magazine, September

3. Sahai, V., and Golomb, R., 2000, "Improve Condenser-Water Flow To Improve Cash Flow," Volume 144, No.4, Power Magazine, July/August, pg.62

4. Nelson, A., Vaezi, V., Cheng, D.Y., 2002, "A 50% Plus Mid Range Advanced Cheng Cycle," In IGTI Turbo Expo 2002, Amsterdam, The Netherlands, June 3-6.

5. Emery, J.C., Herrig, L.J., Erwin, J.R., Felix, A.R., 1958, "Systematic Two-Dimensional Cascade Tests of NACA 65-Series Compressor Blades at Low Speeds", NACA Technical Report 1386.

6. Jacobs, E.N., and Sherman, A., 1939, "Airfoil Section Characteristics as Affected by Variations of the Reynolds Number", NACA Technical Report 586

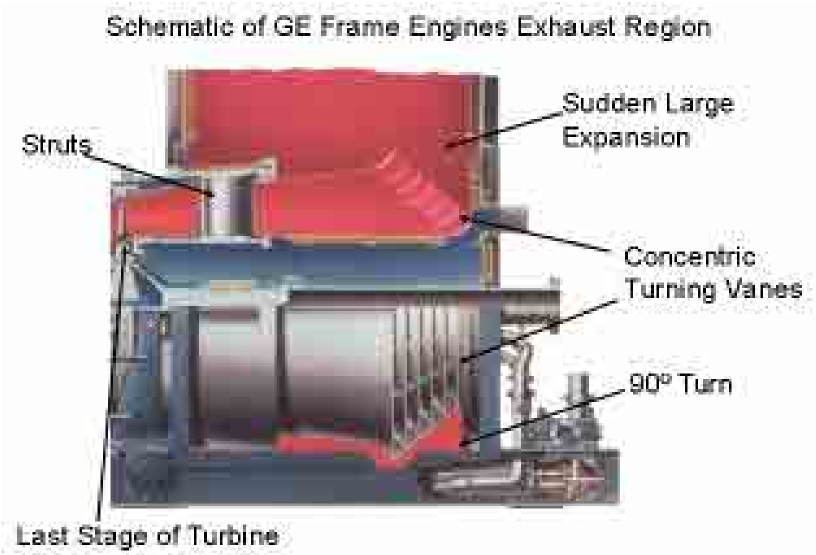

Figure 1. A Schematic of GE Frame Gas Turbine showing the exhaust gas path and areas where large pressure losses occur.

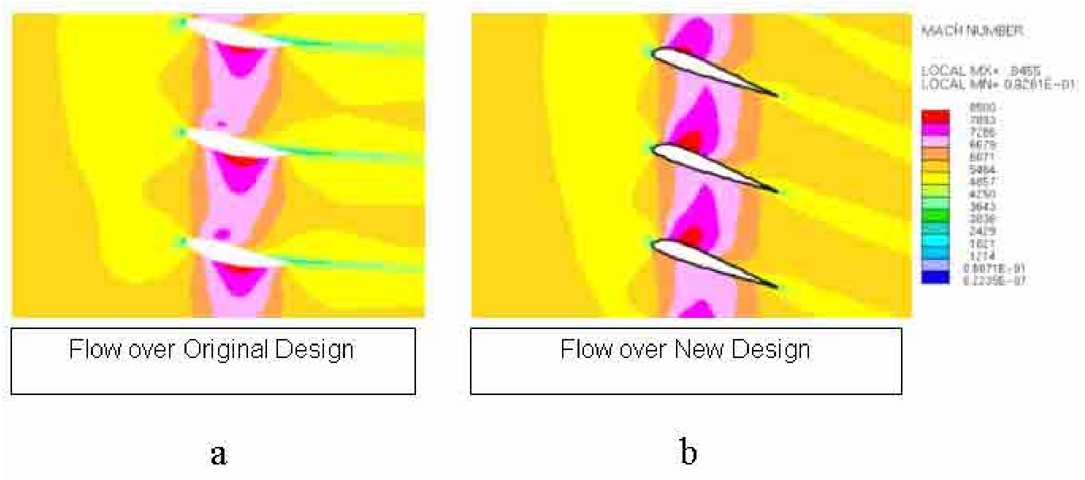

Figure 2. Differences in Flow Behavior between (a) the current strut design and (b) new strut design. The inlet conditions represent full speed full load operation. The turbulence caused by the current design leads to large pressure losses and removes the swirl that leads to large pressure losses downstream of the struts.

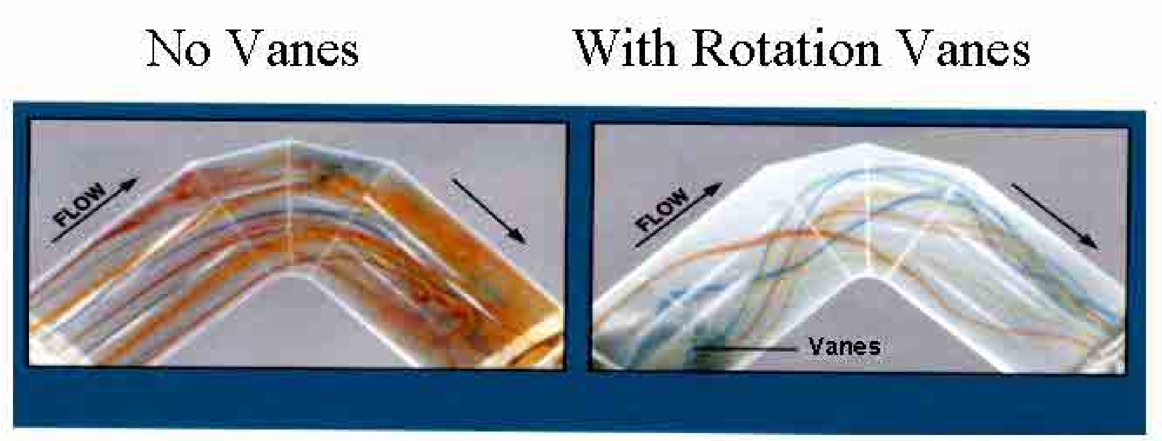

Figure 3. When water flows through an elbow, separation and turbulence occur. Using specially designed vanes to pre-rotate the fluid, the fluid negotiates the turn with all streamlines traveling in equal flow paths.

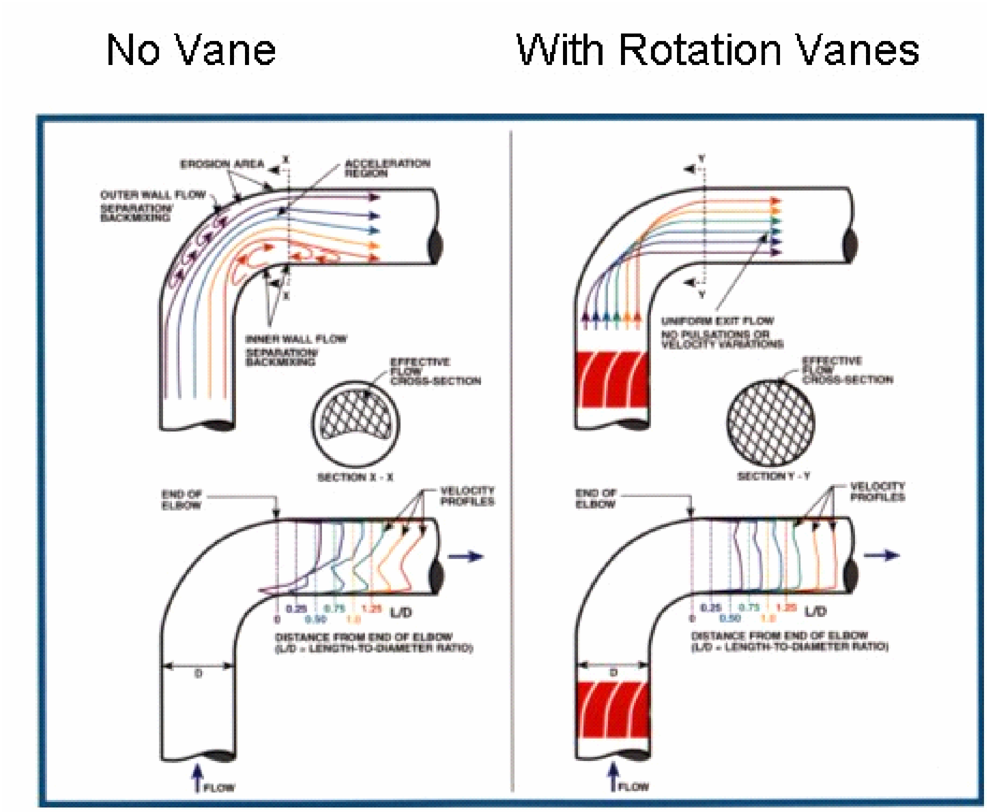

Figure 4. In a plain elbow with air flowing through, a skewed velocity profile results at the exit. With pre-rotation vanes, there is a flat velocity profile.

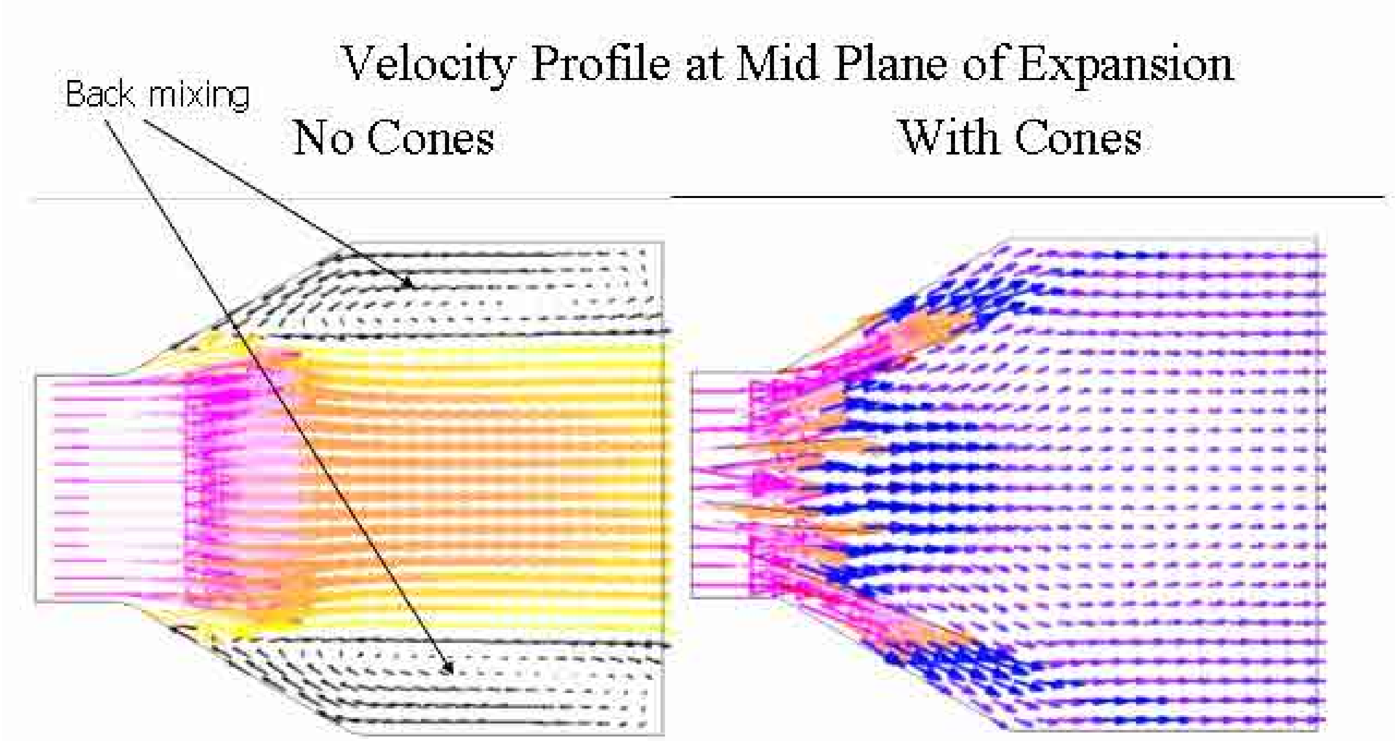

Figure 5. Without cones in a large expansion, significant back mixing occurs with a high velocity jet in the center. With overlapping concentric cones, the velocity profile through the expansion becomes uniform.

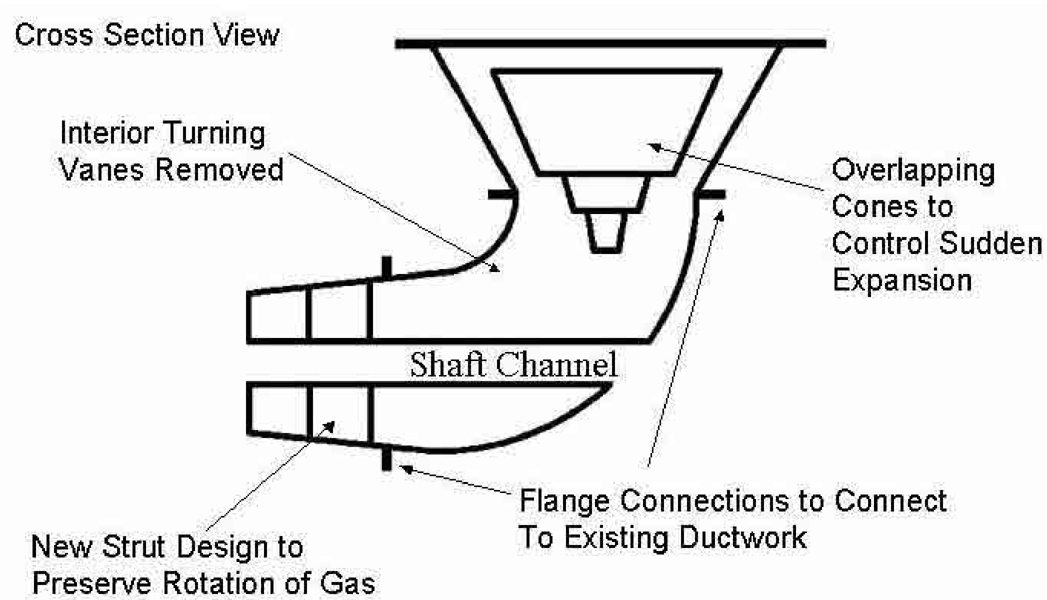

Figure 6. This is a schematic of a new tailpipe design that fits within the current space of the old design shown in Figure 1. The struts will be modified to the new shape shown in Figure 2. Downstream of the struts the wall and turning vanes will be replaced with an elbow.