CFD Analysis and Optimization of Industrial Turbine Exhaust Systems

Jiang Luo
Chris Twardochleb
Solar Turbines, Inc.
San Diego, CA 92101

ABSTRACT

Numerical analyses have been carried out for the 3-D flow field inside turbine exhaust diffuser-collector systems. The analyses were performed with a multi-block computational fluid dynamics (CFD) package which solves the Reynolds-averaged Navier-Stokes equations and the turbulence (k and ε) transport equations. Complex flow features, including the 3-D screw-type vortices inside the collector as well as the flow interaction near the diffuser-collector interface are revealed and examined. Their impact on the pressure recovery of the whole exhaust system was investigated through numerical simulations. It has been found that significant performance improvement can be obtained by managing the collector vortex size and strength and reducing the interaction loss. The effects of inlet flow conditions, including the velocity profile (blockage, etc) and the turbulence level on the pressure recovery are studied. The CFD analyses have been utilized to obtain a new design of diffuser-collector system with higher pressure recovery and lower manufacturing cost. Significant performance improvement of this new design over the existing design has been confirmed by the data. Comparison of rig test data and CFD predictions indicates that the 3-D CFD procedure has captured the major flow features and can be used as a valuable design tool.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>diffuser area</td>
</tr>
<tr>
<td>B1</td>
<td>boundary layer blockage</td>
</tr>
<tr>
<td>Cp</td>
<td>pressure recovery, ( \frac{P_{\text{local}} - \bar{P}}{P_{\text{t,1}} - \bar{P}} )</td>
</tr>
<tr>
<td>Cp1,4</td>
<td>pressure recovery, ( \frac{P_{\text{t,1}} - \bar{P}}{P_{\text{t,1}} - \bar{P}} )</td>
</tr>
<tr>
<td>H</td>
<td>turbulent kinetic energy</td>
</tr>
<tr>
<td>L</td>
<td>diffuser center line length</td>
</tr>
<tr>
<td>M</td>
<td>Mach number</td>
</tr>
<tr>
<td>P</td>
<td>total pressure</td>
</tr>
<tr>
<td>Pp</td>
<td>static pressure</td>
</tr>
<tr>
<td>δ</td>
<td>boundary layer thickness</td>
</tr>
<tr>
<td>ε</td>
<td>dissipation rate of turbulent kinetic energy</td>
</tr>
<tr>
<td>µ</td>
<td>molecular viscosity</td>
</tr>
<tr>
<td>µt</td>
<td>turbulent eddy viscosity</td>
</tr>
<tr>
<td>𝜀</td>
<td>total pressure loss, ( \frac{P_{\text{t,1}} - P_{\text{local}}}{P_{\text{t,1}}} )</td>
</tr>
<tr>
<td>𝜀2,4</td>
<td>overall total pressure loss, ( \frac{P_{\text{t,1}} - P_{\text{t,4}}}{P_{\text{t,1}}} )</td>
</tr>
</tbody>
</table>

Superscripts and subscripts
-  = mass-averaged value
1, 2 = diffuser inlet, diffuser exit
4 = collector exit

INTRODUCTION

The performance of industrial turbines can be significantly improved with an efficient exhaust diffuser. This is because higher pressure recovery of the exhaust system means lower pressure at the turbine exit (for a fixed ambient pressure), which means higher pressure ratio through the turbine stage and improved output power and efficiency. An industrial gas turbine exhaust system, which may include a diffuser and a collector, is often the largest engine component. Reducing the size of the exhaust system becomes very important, in order to obtain a compact engine design. However, the flow field inside a diffuser-collector system is highly three dimensional and complex, which may include regions of large flow separation/reattachment, effects of streamline curvature on turbulence, and unsteady separation (or vortex shedding) from struts inside the diffuser, swirling vortices and flow recirculation inside the collector. A thorough understanding and accurate prediction capability of flow features inside
the diffuser-collector are crucial for efficient exhaust system development.

It may be said that the design of turbomachinery is dominated by diffusion – the conversion of dynamic head into stream pressure (Wilson 1995). In most centrifugal compressors, the performance capabilities of both the rotor and the radial diffuser are limited by the diffusion capabilities of the flow channels. Much of the aerodynamic loss in turbomachinery is caused by boundary layer separation resulting from a certain degree of diffusion that is too large for the boundary layer to overcome. There have been extensive research efforts on the flow diffusion and various types of diffusers over the last several decades, e.g., the basic research by Kline et al. (1959) and Reneau et al. (1967), as well as the well-known correlation by Sovran and Klomp (1967). Many of them focused on the detailed mechanisms which govern the performance of common diffusers. Japikse (1984) presented a comprehensive summary of about 1,400 papers on diffuser research and development published before 1984. A lot of valuable information can be found in Japikse’s book. There are some recent efforts, such as the work on combustor diffusers by Little et al. (1997) and Hestermann et al. (1994).

Despite extensive research efforts on various diffusers over the last several decades (Japikse 1984), there have been only a few investigations into the complex flow fields inside industrial turbine exhaust systems. Japikse and Pamreen (1979) conducted a series of experimental tests for an automotive gas turbine curved diffuser with single or double discharge collectors. Their data provided a lot of insight to the diffuser-collector flow field. Senoo and Kawaguchi (1983) carried out detailed testing for annular curved-wall diffusers with different collector configurations. Attempts were made to retard the formation of screw type vortex inside the collector, which was found by the authors to decrease the overall pressure recovery.

Recently, Sultanian et al. (1998) carried out a comprehensive experimental and numerical study in a typical industrial turbine exhaust system. They concluded that the 3-D CFD code (STAR-CD) is capable of predicting gas turbine exhaust system flows, including the separation behind the struts and secondary flows inside the rectangular collector. While there is more data available than before, not all the essential physical features in an exhaust system have been revealed due to the complication created by the many geometric and flow parameters.

The objectives of the present paper are: 1) to gain further physical understanding of the flow features inside typical industrial gas turbine exhaust systems through extensive CFD simulations; 2) to develop a new exhaust system with optimized diffuser and collector shape with low manufacturing cost and high performance; 3) to carry out rig testing to validate the CFD predictions. The new exhaust system is intended to be used as a retrofit for Solar’s Saturn engine.

**COMPUTATIONAL PROCEDURE**

One of the leading commercial 3-D Navier-Stokes codes, STAR-CD, from Computational Dynamics Inc., was used in the current investigation. STAR-CD is a self-contained, fully integrated code system providing pre-processing, analysis and post-processing. Compared to other codes using multi-block structured grids, STAR-CD requires less memory to execute the computations, which were run on an SGI Power Challenge workstation and later on an Octane workstation at Solar Turbines.

The quality and size of computational mesh is crucial to the accuracy of predictions. Although STAR-CD can handle cells of tetrahedral, prism and pyramidal shapes which take less time to generate, hexahedral cells were used in this paper for better resolution of viscous layers. Generation of hexahedral mesh for complex geometries like the exhaust system is quite time-consuming and difficult due to the limitation of available grid generation packages. The multi-block mesh generation was finished using PROSTAR, STAR-CD's pre- and post-processing program. The geometry was either created in PROSTAR or defined by IGES files exported from CAD packages such as Pro/Engineer. The blocks are first built up one by one. About 10 blocks are generated for a typical diffuser-collector configuration. The mesh is then generated in each block and merged and compressed into the final mesh for the whole domain.

Typical meshes for exhaust systems are shown in Fig. 1. The meshes in Fig. 1 a) and c) are for the existing exhaust system of Saturn engine, which is serving as the baseline case. As stated above, one goal of this work is to obtain an improved design as a retrofit, which requires the new design to stay inside the 3-D envelope of the existing configuration. The meshes for the new design, which will be described in detail, are shown in Fig. 1 b) and d).
The effects of inlet flow swirl on the diffuser-collector performance have been well studied (e.g., Japikse and Pampreen 1979, Sultanian et al. 1998). The influence is not significant if the inlet swirl is low and the struts have well-designed shapes. The effects of inlet flow swirl are not included in the present analysis due to the low swirl angle at design conditions measured for the current Saturn engine. With zero inlet swirl, only half of the exhaust system needs to be modeled due to the geometric symmetry, which brings in 50% saving of CPU time. To further minimize the computational mesh size, the struts inside the diffuser are not modeled since their influence is very small under zero swirl anyway. Typically, the computational mesh has about 330,000 fluid cells, which is equivalent to 660,000 cells for the whole geometry. This grid size was found to be able to capture all the important flow features. The computed values of y+ (for near wall spacing of the mesh) are typically in the range of 50 to 200 for the wall surfaces.

Despite intense research efforts since the late 1960's (Launder 1989, Lakshminarayana 1996), turbulence modeling remains to be the most challenging subject in computational fluid dynamics. All the existing turbulence models are inexact representations of the complex turbulence phenomena and there is no perfect model. There are a number of turbulence models available in STAR-CD, including the standard k-ε model, the RNG k-ε model, and the quadratic nonlinear k-ε (denoted as NL k-ε hereafter) model. The RNG k-ε model (with wall functions near the surfaces) is selected as the default model in the present work due to its capability to capture separation and mean flow distortion on turbulence. The standard k-ε model is not favored since it cannot predict the effect of streamline curvature on turbulence (e.g., Luo and Lakshminarayana 1997), which is important in curved-wall diffusers.

Diffuser inlet flow velocity profiles, with the same aerodynamic blockage and the approximate pitch angle (i.e., radial angle) distribution as engine conditions, have been specified at the computational inlet. A separate subroutine has been written for STAR-CD to allow specification of proper profiles for both mean flow and turbulence properties (k and ε) at the computational inlet. At the computational exit (i.e., the collector exit), the averaged value of back pressure is specified to be the ambient atmosphere pressure. On the symmetry plane, the symmetry boundary condition was specified. On the walls, adiabatic boundary conditions are employed for the energy equations.

The system of 3-D Navier-Stokes equations and k and ε equations are solved using the SIMPLE method in STAR-CD. The second-order accurate QUICK scheme was utilized to discretize the convection terms in order to minimize the errors caused by high numerical dissipation such as in first order accurate upwind differencing scheme. The iterations are considered converged if the maximum residual for all equations is lower than 0.001.

**RIG TESTING TECHNIQUES**

The Diffuser Test Facility at Solar was used for the rig testing in the present work. This facility was constructed in recent years for the testing of new diffusers with different performance enhancing techniques. The test facility is configured to allow the airflow direction to be reversed, providing the capability of flowing air through the rig either under suction or pressure. Suction capability was required.
to allow testing of the half scale Laminated Object Models (LOM) of the existing and new designs of diffuser and collector, as these models begin to delaminate at the blower outlet temperature of 76.7 °C (170 °F) encountered when operating under pressure. Laminated object modeling was chosen as a cost-effective method of rapid prototyping the complex geometry contained in these models. The models were procured from outside vendors using the Pro/Engineer models developed at Solar Turbines on the basis of the CFD model. Model procurement typically requires approximately one month.

A schematic of a typical rig is shown in Fig. 2. Overall pressure recovery in the suction mode of operation was obtained by measuring the cell ambient pressure (inlet total pressure), diffuser inlet static pressure obtained from 24 static taps, and the collector outlet static pressure measured in the plenum downstream of the collector outlet.

The inlet boundary layer blockage was varied by altering the length of the constant area annular section directly upstream of the diffuser. A 1.5 inch (3.8 cm) and 10 inch (25.0 cm) length of ducting, with and without a boundary layer trip strip at the upstream end, was used to provide four different inlet conditions. Boundary layer surveys were conducted to determine the nature and extent of the boundary layer at inlet Mach numbers of 0.3 and 0.5.

![Fig. 2 Bell mouth and diffuser sections of a typical rig.](image)

**RESULTS AND DISCUSSION**

During the upgrade process of the Saturn exhaust system, both the diffuser and the collector have been redesigned. The redesign is a challenging task due to severe space restrictions. As stated earlier, the new design can not go beyond the envelope of the existing design. At the collector exit, the flange size, which is only 2.65 times the diffuser inlet area (A2/A1=2.65) also needs to be maintained since the new design is required to be interchangeable with the existing one.

**Analysis of the Existing Exhaust System**

The CFD effort started with an analysis of the existing Saturn exhaust diffuser-collector system. The geometry and computational mesh have been shown in Fig. 1 a) and c). Only half of the mesh is used in the computation due to the reasons described earlier. All the other geometric details have been included in the model.

The computational diffuser inlet flow conditions can be summarized as: M=0.45, T=840 °K (1050 °F), linear profile for pitch angle (from 0.0 degree at hub to 16.0 degree at shroud), 5/H = 0.1 at hub and 0.2 at shroud (blockage around 4.3%), and zero swirl. Using the diffuser inlet passage height as the reference length, the Reynolds number is 1.97x10^5. These values are close to the data measured from engine testing (Twardochleb et al. 1997) at the design condition except near the shroud endwall where there is large scatter in the data, mainly due to the tip clearance flow. These inlet boundary conditions are used in all the following CFD analyses except those for the half-size rig models.

Using the RNG k-ε model, the pressure recovery (Cp) from the inlet to the collector exit is predicted to be 0.451, being higher than the measured engine data Cp of 0.41 at zero inlet swirl (Twardochleb et al. 1997). Considering the complexity of the flow field inside the curved-wall diffuser and the collector with compound walls, the prediction can be regarded as in reasonable agreement with the data. The discrepancy maybe due to the struts not modeled in the computation, the inlet flow conditions and the limitation of the turbulence model.

The predicted Cp is found to drop significantly from the diffuser exit (Cp=0.683) to the collector exit flange (Cp=0.451). This is mainly due to the large loss of total pressure caused by the screw type vortex inside the collector, which will be shown in comparison to new designs later. The predicted total pressure loss coefficient (ζ) is only 0.0047 at the diffuser exit, which is only 16% of the overall loss (0.0291) at the collector exit. This suggests that the
emphasis be put on reducing the loss of total pressure in the collector.

**Design of the New Diffuser**

Examination of the computed flow field of the existing exhaust system indicated that the diffuser could be improved. A new diffuser was hence designed first. Due to the severe space restriction, the annular curved-wall diffuser type was maintained. However, both inner and outer walls' shapes have been redesigned through CFD optimization. More than 10 different geometries have been computed.

As shown in Fig. 3, the new diffuser is about 1.1 inch longer than the existing one. The outer wall curvature is smaller in the new one near the diffuser exit. The new diffuser allows smoother diffusion and turning of flow than the existing diffuser. The static pressure increases gradually along the inner and outer walls of the new diffuser, resulting in a smooth adverse pressure gradient. The predicted values of $C_p$ for the single diffusers (i.e. w/o collector), as well as other parameters, are listed in Table 1. The new diffuser $C_p$ has improved by about 5.5%.

Table 1 Parameters for the existing and new diffusers

<table>
<thead>
<tr>
<th>Parameter</th>
<th>$L/H$</th>
<th>$A_2/A_1$</th>
<th>$C_p1,2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Existing Diffuser</td>
<td>4.4</td>
<td>2.4</td>
<td>0.73</td>
</tr>
<tr>
<td>New Diffuser</td>
<td>4.8</td>
<td>2.5</td>
<td>0.77</td>
</tr>
</tbody>
</table>

![Existing Diffuser](image1)

![New Diffuser](image2)

Fig. 3 The new and existing diffusers

The above values of $C_p$ are obtained under the same diffuser inlet conditions as measured in the Saturn engine test. During the iteration process, it was found that the performance of curved-wall diffusers is very sensitive to the inlet flow conditions, particularly the flow pitch angle and turbulence level. This is because the flow has a strong tendency to separate from the outer (convex) curved wall due to the adverse pressure gradient as well as the damping of turbulence by the convex curvature (Bradshaw 1973). The flow can also separate from the outer wall due to the large adverse pressure gradient near the exit. Since engines often have different turbine exit flow conditions, the shape of a curved-wall diffuser may need to be designed on a case-by-case basis to achieve maximum performance.

**Collector Shape Optimization**

After the new diffuser has been designed, it is combined with various collector configurations to provide maximum performance, as well as to meet manufacturing requirements such as low cost, easy assembly and maintenance. To reduce the manufacturing cost, major features of a newly-designed exhaust collector for the Titan 130 gas turbine (Luo 1996, internal report) have been adopted. Instead of using compound curved surfaces as on the existing Saturn collector, planar surfaces are used for the new collector walls, which can be made from sheet metal, without the need for forming dies. Different views for the geometry and computational meshes of the new collector are shown in Fig. 1 b) and d).

One distinct feature of the new collector is the highly leaned front face, unlike the rectangular collectors which have been widely used (e.g., in GE’s design of Sultanian et al. 1998, Senoo and Kawaguchi 1983, and Japikse and Pampreen 1979). As will be demonstrated in the following, the leaned front face can substantially reduce the loss of total pressure inside the collector and increase the overall pressure recovery for the whole exhaust system.

The flow field inside an exhaust system is dominated by the flow diffusion, turning and 3-D vortices. The exhaust flow first undergoes diffusion and turning (from axial to radial) along the curved-wall diffuser, then discharges into the collector. The flow accumulates along the circumferential direction and moves towards the collector exit. Due to the strong turning, the flow inevitably rolls into two screw type vortices (one on each side) inside the collector and swirls towards the exit. This flow pattern can be seen in the streamline traces in Fig. 4. It is the screw-type vortices inside the collector that causes a large loss of total pressure, as observed earlier in the existing design.
One effective approach to reduce the total pressure loss caused by the screw-type vortex is to change the shape of collector, which will change the vortex size and strength. More specifically, the approach is to optimize the cross-sectional area of collector in the circumferential direction. Since the mass flow rate increases approximately linearly from zero at the bottom to the total mass rate at the top (collector exit) due to the flow accumulation, the collector cross-sectional area should also increase linearly in the circumferential direction. It should start from zero to match the increase of mass flow rate to maintain a uniform through flow velocity. A larger collector volume at the bottom will only provide more space for vortex formation.

With the design intention to keep a constant radius for the outer wall of the collector (e.g., front view in Fig. 15), the dimension of collector cross-section in the radial direction is almost constant since the diffuser ends at a constant radius. Therefore, the collector front face should not be parallel to the back face (as in the rectangular shape). Instead, the front face should be leaned away from the back face (see Fig. 1d) to enable the cross-section dimension in the axial direction have a linear increase, which results in the linear increase of the flow passage area. The benefit of the leaned front face of the collector was observed on the design of the exhaust system for the Titan 130 turbine engine.

In order to demonstrate the impact of collector cross-sectional area distribution on the performance of the whole system, a series of CFD simulations have been performed for collectors with different lean angles of the front face. With the same new diffuser described in the last section and the fixed location and size (diameter = 18.0 inch) of the collector circular exit flange, three exhaust system models were generated using PROSTAR with different lean angles for the collector front face. As seen in the symmetry section of the system in Fig. 5, the lean angles are 12.0, 16.5 and 19.0 degree (with respect to vertical direction). The collector dimensions in the radial direction are the same and are also kept inside the envelope of the existing collector.

The lean angle of 19.0 degrees is close to the largest possible angle without changing the overall geometry structure. This angle enables the collector cross-section area to increase linearly from a small value at the bottom to the collector exit area at the top. The lean angles of 16.5 and 12.0 degrees were selected to examine the influence of the distribution of the collector flow passage area in detail.

Computations were carried out to compare the three models using the diffuser inlet flow conditions described earlier. The computational grid size, numerical scheme, the flow properties and boundary conditions are kept the same to ensure a fair comparison. The only difference is the collector front face angle, i.e., the circumferential variation of the collector area. As seen clearly from Fig. 5, the diffuser discharge flow starts to roll into a vortex at the collector bottom. The size of this vortex grows with the collector cross-section area at the bottom. The 12-degree collector has the largest area at the bottom, which results in the largest loss of total pressure ($\xi$) and the lowest value of pressure recovery (Fig. 6). The location of the peak $\xi$ and the lowest $C_p$ is at the core of the vortex.

The above features can also be seen clearly at the section midway between the bottom and the top (i.e., the 9:00 o’clock position) in Fig. 7. The screw-type vortex has grown all the way from the bottom to this location in all three collector models. However, the vortex inside the 12-degree collector, being the largest at the bottom, has maintained the largest size (as seen in the contour of vorticity in Fig. 8a), resulting in the largest loss $\xi$ (Fig. 7a) and the lowest $C_p$ (Fig. 9a). There is a direct correspondence between the performance and the collector front face angle: the smaller the lean angle, the higher the total pressure loss and the lower the pressure recovery.
Fig. 5 Predicted contour of total pressure loss $\xi$ on the symmetry section (6:00 and 12:00 o'clock locations) of exhaust system models

Fig. 6 Predicted contour of pressure recovery $C_p$ on the symmetry section (6:00 and 12:00 o'clock locations) of exhaust system models
Fig. 7 Predicted contour of total pressure loss $\xi$ on the 9:00 o’clock section of exhaust system models

Fig. 8 Predicted contour of vorticity on the 9:00 o’clock section of exhaust system models

Fig. 9 Predicted contour of pressure recovery $C_p$ on the 9:00 o’clock section of exhaust system models
The mass-averaged values of \( C_p \) at the exits of diffusers and collectors are plotted in Fig. 10. It's evident that the overall \( C_p \) (i.e., \( C_p_{1.2} \)) drops with the decrease of lean angle. However, the \( C_p \) values at the diffuser exit (i.e., \( C_p_{1.4} \)) are almost the same for the three systems, indicating the difference in \( C_p_{1.4} \) is mainly caused by the difference in the collector shape. Similar observation can be made for the mass-averaged total pressure loss (\( \xi \)) in Fig. 11, where the overall loss reduces with the increase of the lean angle. The values of \( \xi \) are almost the same at the diffuser exit, and are a small portion of the overall loss. Most of the loss occurs inside the collector, caused by the circulating vortex flow.

Fig. 10 Predicted \( C_p \) (mass-averaged value) at the exits of diffusers and collectors.

Fig. 11 Predicted loss of total pressure \( \xi \) (mass-averaged value) at the exits of diffusers and collectors.

It's known that diffuser performance is strongly influenced by the inlet boundary layer thickness. Numerical simulations were carried out with a reduced level of inlet aerodynamic blockage: 2.9% \((\delta/\text{H}=0.1 \text{ at both hub and shroud})\) compared to the blockage of 4.3% at design conditions. As shown in Fig. 12, the lower blockage yields improvement in \( C_p \) for all the collectors. The trend of \( C_p \) versus lean angle holds the same for reduced inlet blockage.

Fig. 12 Effects of diffuser inlet boundary layer blockage on the pressure recovery of the exhaust systems.

One uncertain issue with CFD analyses of diffuser flows is the turbulence level at the diffuser inlet due to the lack of experimental data. In all the previous computations, the turbulence level at the diffuser inlet is specified to be high, due to the high turbulence fluctuations caused by upstream wakes, secondary flows and tip clearance flows. Specifically, values for the \( k \) and \( \varepsilon \) at the inlet were specified so that the turbulence eddy viscosity \( \mu_t \) is around 200 times the molecular viscosity \( \mu \). The effects of the inlet turbulence level are shown in Fig. 13, where high \( T_u \) means \( \mu_t = 200\mu \), medium \( T_u \) means \( \mu_t = 100\mu \), low \( T_u \) means \( \mu_t = 50\mu \). It's evident that the predicted diffuser performance is significantly affected by the inlet turbulence level. Nevertheless, the trend of \( C_p \) versus lean angle remains the same under different inlet turbulence levels.

Fig. 13 Effects of diffuser inlet turbulence level on the pressure recovery \( C_p \) of the exhaust systems.
All the computations shown in previous figures were performed with the RNG k-ε model, which is the default turbulence model in this paper. To assess the capability of turbulence models, the nonlinear k-ε model and the standard k-ε model have also been employed to analyze the current three collector models. The mass-averaged values of Cp from different k-ε models at the collector exit are plotted in Fig. 14, where all the Cp's are seen to increase with the decrease of the lean angle. At the same time, the mass-averaged loss t from each of the k-ε models at the collector exit was found to increase with the decrease of the lean angle.

![Fig. 14 Mass-averaged pressure recovery Cp of the exhaust systems predicted by different k-ε models.](image)

The standard k-ε model is seen to predict higher Cp and lower loss for all three collectors than the other two k-ε models, which have similar predictions. This is due to the standard k-ε model's inability to capture turbulence damping by convex curvature (as on diffuser curved wall and inside the collector vortex) and the problem of under-predicting the separation size. The standard k-ε model has probably over-predicted the performance. A detailed discussion on various turbulence models' performance for curved flows can be found in Luo and Lakshminarayana (1997). The solution accuracy is dependent upon the turbulence modeling. However, it should be noted that the predictions are self-consistent with each of the k-ε models.

The above analyses demonstrated that the collector cross-sectional area distribution along the circumferential direction is very important for the overall pressure recovery of the system. To avoid the unnecessary loss of total pressure, the rectangular shape (i.e., lean angle = 0 degree) should not be adopted for future collector design. The collector cross-section area should have a linear increase from the bottom to the top. In other words, the collector front face should lean away from the back face and it should have the largest possible lean angle to obtain the maximum pressure recovery.

**Design of New Diffuser-Collector System**

The final design of the new Saturn exhaust diffuser-collector system has integrated the new collector with the new diffuser. Manufacturing consideration has required the lean angle of the front face to be 18.0 degrees, slightly lower than the 19.0 degrees discussed above. Different views of the geometry can be found in Figs. 1b and 1d. Before comparing the new and existing designs, one critical performance enhancing technique, the diffuser extension with variable length, is presented first.

As seen in the contours of total pressure loss and vorticity in Figs. 5, 6 and 7, there is a strong interaction/mixing between the diffuser exit flow and the circulating collector vortex flow near the diffuser-collector interface (outer wall edge at diffuser exit). An important source of total pressure loss, the flow interaction/mixing is caused mainly by the difference in the two flows' direction. The edge of the diffuser outer wall and the collector wall has hence been designed to allow the two streams to merge at a small angle (see Fig. 17b and c).

The new diffuser (Table 1) was designed to have constant radius at the exit (i.e. same diffuser length). However, the static pressure at the diffuser exit is not uniform due to the collector. This has been known for some years (e.g. Japikse and Pampreen 1979). To accommodate this non-uniform exit pressure, the annular diffuser ideally should have variable area ratio in the circumferential direction. But that would increase the cost and was therefore not adopted for current redesign.
In order to reduce the flow mixing loss at the diffuser-collector interface, as well as to address the non-uniform exit pressure, the diffuser outer wall length has been extended almost radially into the collector. The extension, with front view shown in Fig. 15, has variable length circumferentially because of the non-uniform pressure. At the bottom, the diffuser is not extended. The extension length increases gradually from near the bottom to the collector exit.

Different extension lengths have been analyzed and assessed to achieve the best design. The computations were performed using the diffuser inlet profiles at the engine design conditions, as used before. The calculated flow fields with three different extension lengths, 2.0 inch (5.08 cm), 4.0 inch (10.16 cm) and 5.0 inch (12.7 cm), measured at the top or 12:00 o'clock location, Fig. 15) are compared. All the extensions have effectively reduced the loss by the interaction/mixing near the interface, with the 5.0 inch extension providing the highest improvement. Hence the final design of the new exhaust system includes the new diffuser with a 5.0 inch extension (Fig. 15), the collector with 18.0 degree lean angle for the front face and the diffuser-collector interface with minimal flow merging angle. Different views of the mesh for the new design can be found in Fig. 1.

<table>
<thead>
<tr>
<th>Table 2. Predicted performance of existing and new exhaust systems.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Cp</strong>&lt;sub&gt;1.2&lt;/sub&gt;</td>
</tr>
<tr>
<td>Existing system</td>
</tr>
<tr>
<td>New w/o extension</td>
</tr>
<tr>
<td>New system</td>
</tr>
</tbody>
</table>

The mass-averaged pressure recovery and loss of total pressure are listed in Table 2 and Fig. 16. It's obvious that the new design provides much better performance than the existing system. The predicted pressure recovery shows an improvement of 46% (from 0.451 to 0.658) over the existing system. The total pressure loss has been cut by about 40%. Since the values of Cp at the diffuser exit are relatively close, the new diffuser has not played the dominant role for the overall improvement. The improvement is mainly from the optimized collector shape, the diffuser extension and the diffuser-collector interface. The final new design also shows an improvement of 11% in Cp over the one without the diffuser extension. This finding is consistent with the experimental observation of Senoo and Kawaguchi (1983) in which they tested extensions with constant length.

Comparisons of the computed flow field details are shown in Figs. 17 and 18 (on the symmetry section) and Figs. 19, 20, 21 and 22 (on the 9:00 o'clock section). The improvement in reducing ξ and increasing Cp by the final new design over the existing one is evident in Figs. 17 and 18, respectively. Unlike the existing collector with compound walls, the new collector has planar surfaces, which can be made from flat sheet metal. This feature can be seen from the sections shown in Figs. 19-22.

The flow rolls into a big vortex inside the existing collector with almost circular cross-section, as seen in Figs. 20a and 21a. This vortex causes a heavy loss of total pressure. In contrast, the vortex is much smaller inside the new design (Fig. 20c and Fig. 21c). Comparing Fig. 20 b and c and Fig. 21 b and c, the diffuser extension has retarded the formation of the vortex and hence reduced the vortex strength (Fig. 20 b versus c), resulting in a lower loss of total pressure. Figure 22 shows the improvement in Cp brought by reducing the collector passage vortex size and strength and the total pressure loss.

<table>
<thead>
<tr>
<th>Table 3 Predicted performance of existing and new exhaust systems under the low inlet turbulence level.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Cp</strong>&lt;sub&gt;1.2&lt;/sub&gt;</td>
</tr>
<tr>
<td>Existing system</td>
</tr>
<tr>
<td>New system w/o extension</td>
</tr>
<tr>
<td>New system</td>
</tr>
</tbody>
</table>
Fig. 17 Predicted contour of total pressure loss on the symmetry section (6:00 and 12:00 o'clock locations)

Fig. 18 Predicted contour of pressure recovery on the symmetry section (6:00 and 12:00 o'clock locations)
Fig. 19 Predicted contour of total pressure loss at 9:00 o'clock section of exhaust systems

Fig. 20 Predicted contour of vorticity at 9:00 o'clock section of exhaust systems

Fig. 21 Velocity vector plots at 9:00 o'clock section of exhaust systems

Fig. 22 Contour of pressure recovery $C_p$ at 9:00 o'clock section of exhaust systems
In the above computations, the turbulence level at the diffuser inlet is specified to be high with $\mu_t = 200\mu$. As shown in Fig. 13, the inlet turbulence level has a significant impact on the predicted performance. The comparison of the three systems under a lower turbulence level (low Tu for $\mu_t = 50\mu$) is shown in Table 3 and Fig. 23. The Cp of the new design is now 39% (from 0.430 to 0.599) better than that of the existing design and the diffuser extension leads to an improvement of 13% in pressure recovery. This indicates that the improvement percentage of Cp obtained by the new design is not fixed and is affected by the inlet turbulence level. Nevertheless, the new design does show substantial increase in predicted Cp over the existing design under different inlet turbulence levels.

![Graph](image)

**Fig. 23 Predicted performance of the existing system, the new system and the new system without diffuser extension under different inlet turbulence levels.**

**Rig models for new and existing exhaust systems: comparison between CFD prediction and data**

To validate the CFD calculated performance, 0.46 scale LOM models have been procured to conduct rig tests at the Solar Diffuser Test Facility. The rig tests were carried out with cold air, while maintaining similar Reynolds and Mach numbers as in the engine. However, there is an additional transition duct (see Fig. 25) connecting the diffuser inlet with the bell mouth on the rig. So the diffuser inlet flow conditions are not exactly the same as that on the engine. The differences are primarily in the flow pitch angle distribution, the level of boundary layer blockage, and turbulence level.

In the rig test, the boundary layer blockage at the diffuser inlet was varied by altering the length of the constant-area duct upstream of the diffuser. The rig test conditions can be summarized in Table 4.

**Table 4 Measured boundary layer thickness and blockage at the inlet for Solar's Diffuser Facility**

<table>
<thead>
<tr>
<th>Conditions</th>
<th>#1</th>
<th>#2</th>
<th>#3</th>
<th>#4</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\delta_{hub}/H$</td>
<td>11%</td>
<td>30%</td>
<td>2.50%</td>
<td>15%</td>
</tr>
<tr>
<td>$\delta_{tip}/H$</td>
<td>21%</td>
<td>35%</td>
<td>7.50%</td>
<td>20%</td>
</tr>
<tr>
<td>Blockage</td>
<td>0.0496</td>
<td>0.0901</td>
<td>0.0140</td>
<td>0.0408</td>
</tr>
</tbody>
</table>

Where rig test conditions #1, #2, #3 and #4 are for tests with 10 inch (25.0 cm) long constant-area section without trip strip, 10 inch with trip strip, 1.5 inch (3.8 cm) without trip strip, and 1.5 inch with trip strip, respectively. It can be seen that the rig test conditions #1 and #4 have boundary layer blockage similar to the engine condition.

Extensive rig tests for the LOM models of both the new and the existing diffuser-collectors have been conducted at Solar. For the comparison of existing and new models, the following data are tabulated in Table 5 under the above four inlet conditions. The inlet total pressure in the Cp data is obtained by measuring the cell ambient pressure (i.e., the mainstream total pressure at the diffuser inlet).

**Table 5 CFD predictions and rig test data of pressure recovery Cp for existing and new diffuser-collectors.**

<table>
<thead>
<tr>
<th>Conditions</th>
<th>#1</th>
<th>#2</th>
<th>#3</th>
<th>#4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cp: Data-existing</td>
<td>0.374</td>
<td>0.383</td>
<td>0.411</td>
<td>0.462</td>
</tr>
<tr>
<td>Cp: Data-new</td>
<td>0.481</td>
<td>0.474</td>
<td>0.527</td>
<td>0.554</td>
</tr>
<tr>
<td>Cp: CFD-existing</td>
<td>0.389</td>
<td>0.396</td>
<td>0.434</td>
<td>0.448</td>
</tr>
<tr>
<td>Cp: CFD-new</td>
<td>0.506</td>
<td>0.503</td>
<td>0.568</td>
<td>0.583</td>
</tr>
</tbody>
</table>

Comparing the test data, it can be seen that the new diffuser-collector has provided improved pressure recovery over the existing one under all the four inlet conditions. The increase in Cp (ACp) ranges from 0.09 to 0.11 and the improvement percentage ranges from 19.2% to 28.6%. The measured values of pressure recovery are sensitive to diffuser inlet flow conditions. The pressure recovery is dependent not only on the boundary layer blockage, but also on the boundary layer status (laminar, transitional or turbulent) and the turbulence level. Even with similar blockage level, the pressure recovery under Condition #4 is much higher than that
under Condition #1. The boundary layers under Condition #1 may still be laminar or transitional, and hence are more likely to separate from the curved walls, resulting in lower pressure recovery. The boundary layers are probably fully turbulent under Condition #4.

For the existing diffuser-collector, the Cp of 0.374 at Condition #1 is 10% lower than the Cp of 0.41 measured in the engine, while the Cp of 0.462 at Condition #4 is 13% higher than 0.41. Since the averaged Cp of 0.418 (of Conditions #1 and #4) is actually close to the engine measurement, it may be assumed that the actual engine inlet condition is somewhere in the middle of rig test Conditions #1 and #4. Under this assumption, the new system would provide a pressure recovery of 0.518, which is the averaged value of #1 and #4. Then the new system would provide an improvement of about 24% from 0.418 to 0.518 under engine conditions.

Fig. 24 Predicted and measured values of Cp for the existing system and the new system under different rig testing conditions

To have a one-on-one comparison with the rig test data, CFD calculations have been performed for the LOM models of both existing and new Saturn diffuser-collectors. As can be seen in Fig. 25, the transition duct upstream of the diffuser inlet has been included in the computational model. Computations of both existing and new exhaust systems were performed for all the four diffuser inlet conditions as in Table 4. The flow is specified to be axial at the computational inlet, with the boundary layer blockage specified to be the same as that in each of the rig test conditions. To roughly simulate the turbulence triggered by the trip strip in the Conditions #2 and #4, a medium level of turbulence (\( \mu_t = 100\mu \)) was specified at the inlet. In Conditions #1 and #3 without the trip strip, a low turbulence level was specified with \( \mu_t = 20\mu \). The predictions for Cp values of both systems under the four inlet conditions are listed in Table 5 and are compared with the testing data in Fig. 24.

Fig. 25 Predicted contour of pressure recovery Cp on the symmetry section of the new exhaust system rig model under the inlet condition #1.

As seen from the Table 5 and Fig. 24, the predicted Cp values for the new system are higher than the predictions for the existing system under all the four inlet conditions. For both systems, the predictions are seen to follow the trend of the experimental data. The predicted improvement in Cp obtained by the new design under each condition is seen to be close to that in the measurement. The pressure recovery values are over-predicted by a factor ranging from about 3% to 8%. A discrepancy less than 10% can be considered as fairly reasonable since there are some approximations in the computations, e.g., the neglect of struts, the uncertainty in the specified turbulence for simulating the trip strip and the limitation of the turbulence model in curved and vortex flows. The CFD predictions of Cp tend to be higher than the measurements, as also seen in the computations by Sultanian et al. (1998) and Little et al. (1997).

With similar blockage, the calculated Cp values of both rig models are lower than the predictions under engine conditions (as presented in Tables 2 and 3). The difference in Cp is mainly brought by the difference in the flow pitch angle distribution at the diffuser inlet (throat region). While
the flow near the inner wall at the diffuser inlet is axial in the engine condition, the rig flow has significant radial velocity component, created by the upstream transition duct in the rig. This radial velocity near the inner wall causes the flow to separate downstream. There is also a difference in the flow pitch angle near the outer wall at the diffuser inlet. These factors may result in a higher recovery in the engine than that measured on the rig.

As discussed earlier, the diffuser inlet blockage in engine operation is close to that in the rig testing Conditions #1 and #4. The averages of the predicted Cp under Conditions #1 and #4 for the existing and new designs are 0.419 and 0.545. Hence, the calculated improvement is about 30%, which is higher than the measured improvement of 24% (from 0.418 to 0.518). This indicates that the performance improvement prediction by CFD should be viewed with caution. Little et al. (1997) made a similar observation on the CFD prediction accuracy for combustor diffuser systems.

It should be noted that predictions are very sensitive to the inlet conditions, including the turbulence level and boundary layer thickness. The predicted Cp of the new system under Condition #4 is 0.583, being much higher than value of 0.506 under Condition #1 which differs from #4 mainly in the level of turbulence. Since there are few experimental data obtained from engine testing that can be used to determine the turbulence level, the inlet turbulence level remains an uncertain parameter for CFD predictions. Nevertheless, the current code - STAR-CD is capable of predicting the trends of pressure recovery and the major flow structures very well.

CONCLUDING REMARKS

3-D Navier-Stokes analyses were performed for industrial turbine exhaust systems with different shapes of collector and diffuser. Complex flow features, including the 3-D screw-type vortices inside the collector as well as the flow mixing/interaction near the diffuser-collector interface, were revealed and examined in detail. Their large impact on the pressure recovery of the whole exhaust system was shown through numerical simulations.

It has been observed that most of the total pressure loss occurs inside the collector due to the screw-type vortex. A significant performance improvement has been obtained by reducing the collector vortex size, which is accomplished by a linear increase of the collector cross-sectional area in the circumferential direction. This optimal area distribution is realized by leaning the front face of the collector. The advantage of this type of collector over the widely-used rectangular collector was demonstrated by both the computations and rig test data.

A diffuser extension with variable length in the circumferential direction was found to reduce the flow mixing loss near the diffuser-collector interface. A new design which provides higher pressure recovery and lower manufacturing cost has been achieved. Rig test data has confirmed the significant performance improvement of this new design over the existing design.

The predictions were found to be very sensitive to the inlet flow conditions, including the velocity profiles (blockage and pitch angle) and the turbulence level. The analyses have over-predicted the pressure recovery of the rig models. The current CFD package is capable of capturing the complex flow features and predicting the trend; however, the magnitude of improvement should be viewed with caution. This is especially true for flows with streamline curvature and large-scale vortices, which can not be predicted well with state-of-the-art turbulence models.

ACKNOWLEDGEMENT

We would like to thank Solar's management for their permission to publish this work. Jason Schell, Archie French and Garret Schiff made important contribution to the diffuser rig and engine testing. We're grateful to Eli Razinsky, Tom Ragland and the Saturn uprate design team members for many technical discussions.

REFERENCES


