ABSTRACT

Both experimental and 3D CFD investigations are carried out in a scale model of an industrial gas turbine exhaust system to better understand its complex flow field and to validate CFD prediction capabilities for improved design applications. The model consists of an annular diffuser passage with struts, followed by turning vanes and a rectangular plenum with side exhaust. Precise measurements of total/static pressure and flow velocity distributions at the model inlet, strut outlet and model outlet are made using aerodynamic probes and locally a Laser Doppler Velocimeter (LDV). Numerical analyses of the model internal flow field are performed utilizing a three-dimensional Navier-Stokes (N-S) calculation method with the industry standard k-ε turbulence model. Both the experiments and computations are carried out for three load conditions: full speed no load (FSNL), full speed mid load (FSML, 57% load), and full speed full load (FSFL). Based on the overall comparison between the measurements and CFD predictions, this study concludes that the applied N-S method is capable of predicting complicated gas turbine exhaust system flows for design applications.

INTRODUCTION

Turbulent flow diffusion and pressure recovery processes occurring in a typical industrial gas turbine exhaust system are rather complex and three-dimensional (3D) in nature. With a growing trend towards higher efficiency combined-cycle (steam and gas turbine) power plants, both aerodynamic and thermodynamic management of gas turbine exhaust for improved performance is receiving its due attention in more recent designs. In most early designs, diffuser section was mainly targeted for most of the pressure recovery and almost all the remaining dynamic pressure was lost in the downstream ducting. For a given compressor discharge pressure and nearly ambient pressure at the inlet to the heat recovery steam generator (HRSG) unit in a combined cycle operation, higher pressure recovery in the exhaust system means lower pressure at the turbine outlet plane. This translates into higher overall pressure ratio across the turbine, giving higher power output and efficiency. A high-pressure-recovery exhaust system generally also favors a nearly uniform flow velocity (or dynamic pressure) and lower turbulence levels into the downstream silencer and HRSG units. It may be noted here that peaky velocity distribution and high turbulence level adversely impact silencer performance. An important feature of an industrial gas turbine exhaust system is that its inlet flow conditions such as mass flow rate and flow angle vary widely with turbine load under design and off-design operations. These considerations form the basis for the present experimental and CFD investigations into a scale model of GE’s MS9001E exhaust system, being one of the most complex designs in the existing product line.

The analysis of three-dimensional diffusion in an exhaust system is extremely complicated by the presence of too many geometric and inlet flow parameters. Available test data and analysis results are rather limited to guide practical designs, and are generally applicable for two-dimensional simple geometries. A number of mostly experimental and some analytical attempts have been made to improve industrial turbine exhaust system performance. Xingsu, Kunyuan and Zuomin (1981), for example, tested marine gas turbine exhaust volutes to select optimum geometric parameters. From the analysis of their experimental data, they also propose the basic design criteria of exhaust volutes. Mao, Wei and May (1987) used the design-of-experiments (DOE) approach to conduct limited model tests to achieve an optimized axial turbine exhaust hood configuration. Fleischer, Koerner, and Mann (1989) used scale model of a complete gas turbine exhaust system to test out suitable design of various flow-distributing devices for acceptable quality of exhaust velocity distribution. Desideri and Manfrida (1995) conducted extensive tests on the model of a gas turbine exhaust diffuser to map out detailed distributions of mean velocities, flow angles and turbulence...
Some of the early work in delineating the diffuser flow regimes are due to Kline and his coworkers at Stanford University (Kline, Abbott, and Fox, 1959; Fox and Kline, 1962; and Reneau, Johnston and Kline, 1967). They found that the maximum pressure recovery in a straight-walled two-dimensional diffuser with thin turbulent inlet boundary layer occurs when short-duration flow reversal propagates up and down the diffuser walls, a condition generally known as transitory stall. In practice, exhaust system diffusers are seldom designed for maximum pressure recovery under transitory stall condition because of a small margin to permanent wall boundary layer separation. The latter, if occurs, results in a marked deterioration in the diffuser pressure recovery performance. Ghose and Kline (1976) presents a method of predicting transitory stall in two dimensional diffusers. Experimental work of Howard, Henssler, and Thonmont-trump also provides details on various flow regimes and their relation to annular diffuser performance and so does the investigation by Takehira, et al. (1977). Iapikse and Papamast (1979) used a boundary layer calculation procedure to predict pressure recovery in an automotive gas turbine annular diffuser and compared with test data. Kanemoto, Toyokura, and Kurokawa (1982) report CFD validation quality data in an annular diffuser.

Test results of Kelnhofer and Derick (1971) indicate that pressure recovery normally increases when a rectangular diffuser is fitted with a straight constant-area tailpipe instead of a plenum chamber. Senoo and Kawaguchi (1983) found that a plenum chamber at the end of an annular curved diffuser reduced the pressure recovery. The design was improved with a bigger size plenum and by incorporating vanes to reduce the swirl.

Influence of inlet conditions on annular diffuser performance has been the subject of experimental studies by Wolf and Johnston (1969) and Adenubi (1975). The incoming swirl which varies significantly between design and off-design operating conditions has a dramatic influence on exhaust system performance. A moderate swirl level generally improves boundary layer separation at the diffuser outer wall and hence the diffuser performance. Beyond a critical swirl level, flow reversal, called vortex breakdown, at the inner radius occurs resulting in significant performance loss. The investigation by Srinath (1968) addresses in detail the issue of swirl effects on annular diffuser performance. Unsteady effects by way of vortex shedding behind struts in an annular diffuser form important design considerations especially under FSNL operating conditions. Fric et al. (1996) through their extensive experimental work report the benefits of using tapered struts to minimize the problem.

Kawagishi and Sakamoto (1992) used a commercial CFD code STAR-CD to successfully validate predictions against measurements in a steam turbine exhaust hood. Raab, Dirk and Hennecke (1996) used another commercial CFD code TASCflow to evolve an optimal aerodynamic design for the whole gas turbine diffusion system. The exhaust system flow fields considered in the present investigation, due to both geometry and inlet flow conditions, are much more complex than those considered in these two studies.

**Present Contribution.** In view of the joint nature of this technology development program, it was decided at the outset not to use any of the company proprietary in-house CFD codes for validation of N-S calculation method with an advanced industrial turbulence model. Instead, an approach to utilize available leading commercial technology for both high quality grid generation and 3D CFD simulation is used. The emphasis throughout this investigation has been on: (1) Accurate scale model of complex geometry of the selected gas turbine exhaust system, (2) Benchmark quality measurements to investigate detailed flow field in the exhaust system, (3) Understanding of the flow field under widely varied load (inlet) conditions, and (4) CFD method validation under these load conditions to confirm the usefulness of the method for design applications.

**EXPERIMENTAL METHOD**

**Experimental Facility.** The experimental program was planned to obtain detailed flow measurements for CFD validation in a 1/10 size scale model of the exhaust system of an industrial gas turbine (GE-MS9001E type). The model cross-section in the horizontal plane is shown in Fig. 1. It consists of an annular diffuser, 10 struts with an airfoil cross-section located inside the diffuser passage, axisymmetric turning vanes with 3-dimensional curved surfaces to deflect the flow from axial to radial direction, held together by 16 tie rods and stiffening plates, and the rectangular plenum. This exhaust system is a "side-exhaust" type where discharge from the last stage turbine flows axially in the annular diffuser, enters the rectangular plenum via turning vanes, and exhausts from the plenum at 90 degrees in the horizontal plane. It thus features a very complex flow field under all load conditions. The

**Figure 1 Test Model Configuration**
model geometry simulates the prototype design in detail. The inner and outer diameters at the model inlet section are 178 mm and 274 mm, respectively. The model outlet cross-section is 310 mm in width and 526 mm in height. At 160 mm upstream of the model inlet plane, 16 movable guide vanes are installed to generate desired inlet flow swirl for each load condition. The model was assembled in a high-speed air test stand. It is shown here in the photograph in Fig. 2. Compressed air (maximum pressure of about 3 atmosphere) was cooled in a heat exchanger and supplied to the test section through a control valve. The exhaust from the test section was discharged into the atmosphere through an outlet pipe and a stack. The flow diagram of the air test stand is described in Fig. 3.

Test Conditions. Tests were carried out for three load conditions: full speed no load (FSNL), full speed mid load (FSML), and full speed full load (FSFL). Main flow parameters for each of the three load conditions are given in Table 1, which shows a wide variation in both mass flow rate and inlet swirl angle from FSNL to FSFL. In the experiment, both the model inlet Mach number and the swirl angle, simulating the actual turbine outlet conditions, were adjusted by controlling the mass flow rate and the upstream vane angle. The outlet pressure was constant at the atmospheric value, and the model inlet and outlet pressure levels as well as the pressure ratio were almost the same as in the actual combined-cycle condition. Since this program mainly concerned an aerodynamic investigation and CFD validation, the inlet air temperature was not simulated to correspond to the turbine exhaust condition. In the scale model, the inlet flow Reynolds number, based on the strut chord length, was in the range 0.6-1.1x10^6 which are about 1/3-1/5 of the prototype values.

Measurement Instrumentation and Error Estimation. Aerodynamic measurements were done at four locations: model inlet, around struts, strut outlet and model outlet. At the model inlet and outlet planes, five-hole pressure probes with a head of 2.4 mm diameter were traversed to measure the distribution of total and static pressures, pitch angle and yaw angle. Pressure from each hole was detected by piezoresistive-type pressure transducers. The probes were calibrated in advance in an open-type wind tunnel under atmospheric condition. Total temperature probes were also traversed together with the five-hole pressure probes. At the strut outlet, four rake-type probes with total pressure and temperature heads of 3 mm diameter were circumferentially traversed simultaneously. In this arrangement, the probe heads were nominally fixed in axial direction, but could span about ±60 degrees laterally. A 5-Watt argon-ion 2D/3D Laser Doppler velocimeter (Aeromertics) was also used locally to measure velocity vectors around strut, since installing a probe in this measuring location would have significantly disturbed the flow field. Only 2D measurements in blade-to-blade region were made due to the limitation of the viewing window size on the tip wall. Pure water was sprayed upstream to create droplets of about 5 micron diameter for seeding in the LDV measurements. A hot film probe with a sensor of about 50 micron diameter and 1 mm length was traversed at the model inlet plane for measuring the turbulence intensity distribution, which formed a part of the inlet boundary condition specification in the calculations. Probes used for strut outlet and model outlet are shown in Fig. 4. Measurement procedures, except the LDV measurements, were programmed and automatically controlled on an HP-9000 engineering workstation.

The accuracy of measurements was checked by several methods. At the model outlet plane, velocities were measured by two different

<table>
<thead>
<tr>
<th>Load</th>
<th>Mass Flow Rate (kg/s)</th>
<th>Swirl Angle (deg)</th>
<th>Absolute Mach Number</th>
<th>Static Press (MPa)</th>
<th>Total Temp (deg.C)</th>
<th>Reynolds Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>FSNL</td>
<td>3.9</td>
<td>-50.8</td>
<td>0.452</td>
<td>0.107</td>
<td>43.1</td>
<td>0.64 x 10^6</td>
</tr>
<tr>
<td>FSML</td>
<td>5.8</td>
<td>-4.9</td>
<td>0.463</td>
<td>0.095</td>
<td>47.7</td>
<td>0.90 x 10^6</td>
</tr>
<tr>
<td>FSFL</td>
<td>6.8</td>
<td>13.6</td>
<td>0.567</td>
<td>0.09</td>
<td>44.3</td>
<td>1.17 x 10^6</td>
</tr>
</tbody>
</table>
methods: 1) using the total and static pressures by the pressure probe, and 2) LDV. Comparison shown in Fig. 5 indicates fairly good agreement between the two sets of velocity measurements. The integrated mass flow rate through the model inlet, strut outlet and the model outlet derived from the pressure-probe measurements were also compared with that measured by the orifice pressure-difference in the upstream lead pipe. The mass flow rate deviations from the orifice-based values were mostly within 3% in relative and 5% maximum. Considering the complex nature of the flow field through these measuring planes with a nonuniform flow distribution and separation, this accuracy is considered within an acceptable range. Estimated error in both total and static pressures is about 0.2%. With this error in pressure, the maximum relative errors in pressure-recovery and total pressure loss coefficients become 2% and 7%, respectively for FSNL, 4% and 9% for FSML, and 4% and 4% for FSFL.

COMPUTATIONAL METHOD

Strategy. At the outset of this joint program, a computational strategy for the validation of the state-of-the-art CFD technology was adopted and followed throughout the investigation. Three key elements of this strategy are: 1) Choice of a turbulence model, 2) Choice of computational grid, and 3) Choice of commercial N-S code. Each of the three elements are briefly discussed here.

Choice of turbulence model. In a recent study, Hirsch and Khodak (1995) compared different turbulence models: standard high-Reynolds-number k-ε model with wall functions, a low-Reynolds-number k-ε model, and an explicit non-linear algebraic Reynolds stress closure model (ASM) to predict an S-shaped diffusing duct flow. Although, ASM is found to give some improvements in predicting flow details, the overall flow parameters obtained with different turbulence models are close to each other. In an earlier study, Sultanian and Mongia (1986) found that the flow predictions using standard high-Reynolds-number k-ε model with wall functions compared very well with the measurements of Kanemoto, Toyokura and Kurokawa (1982) in an annular diffuser. In view of these and other investigations and overall accumulated experience with this model, it was decided to stay with the standard high-Reynolds-number k-ε model with wall functions, simply called the k-ε model, in the present investigation.

Choice of computational grid. The geometric complexity of the present computational domain favored the choice of an unstructured grid mainly from the speed of grid generation. However, from the considerations of computational accuracy and available solver speed, this choice was not made. Instead, it was decided to go for a multiblock high quality structured grid as discussed in the grid generation section below.

Choice of commercial N-S code. Based on a number of in-house benchmarking and other validation studies of different leading commercial CFD codes, STAR-CD was selected as the N-S solver in this investigation, since it required less memory to run the present model with 1.2 million cells on a highend HP workstation. This choice minimized the overall cost of CFD runs to simulate the three load conditions used in the experiments.

Grid Generation. Generation of high quality CFD grid in a complex computational domain remains to be one of the most challenging tasks in most applications in gas turbine design engineering. For generating multiblock structured grid in the scale model of the gas turbine exhaust system considered here, the native
grid generation capability of STAR-CD was found inadequate and
time-consuming. An evaluation of commercially available grid
generation technologies led to the superior gridding package
GridPro/az3000. Details on this grid generation technology may be
found in Eiseman (1995) and Cheng and Eiseman (1996). It suffices
to mention that the selection was made on the basis of three key
criteria: grid quality, speed of grid generation in a complex domain,
and compatibility with the chosen CFD solver.

The overall grid generation process is based upon a top-down
concept whereby a coarse wire frame is optimally mapped onto a
smooth, nearly orthogonal multiblock grid. The purpose of the wire
frame, which is a sequence of corners and linkages that form a coarse
unstructured quadrilateral (2D) and hexahedral (3D) grid, is to
define the pattern of grid points that cover the computational domain
under consideration. The wire frame corners are generally placed in
a loose fashion relative to the associated region boundaries and are
appropriately assigned to the corresponding parts of these boundaries.
With the glue of surface assignment, the wire frame represents a grid
pattern for the region called grid topology that represents the minimal
information needed to generate a multiblock grid. Organization for
this information is given by the Topology Input Language (TIL),
which is an object oriented language as discussed in Eiseman (1995).
The multiblock grids are then automatically generated by compiling
and running TIL codes. In this process, gridding on the bounding
surfaces and their common intersections are also generated
automatically.

The surface grid for the entire computational domain generated
using GridPro is shown in Fig. 6, which is showing lower half of the
model geometry only; the whole geometry, however, was used in the
flow simulations. In spite of very complex geometry of the exhaust
system, the figure shows that the grid lines are virtually
perpendicular to each other and transition of grid cells are smooth and
gradual. There are about 1.2 million computational cells in this
model. Clustering specifications have been used to ensure that first
nodes from the wall region fall within the logarithmic region for
proper applicability of the wall function boundary conditions used
here. Computed y+ values are found typically in the range of 30-150
for most wall surfaces.

Governing Equations. The governing conservation
equations of both the mean flow and the standard high-Reynolds-
number k-ε turbulence model to compute three-dimensional
compressible flow in this study are, for example, given in
Lakshminarayana (1966) with the turbulence model constants from
Launder and Spalding (1974).

Boundary Conditions. The 3D computational model used
here involves three types of boundaries: inflow (inlet plane),
outflow (outlet plane) and no flow (solid walls). Each type of
boundary condition is briefly discussed here.

Inflow (inlet Plane). Detailed measurements of the mean flow
and the turbulence field are used as inlet boundary conditions for all
the three load conditions. Flow quantities such as total pressure,
total temperature, flow angle, turbulent kinetic energy, turbulence
dissipation rate (obtained from local equilibrium with a suitable
length scale) and mass flow rate are specified at the inlet
computational cell faces.

Outflow (outlet Plane). The boundary conditions at the outlet
plane correspond to zero axial gradient for all the dependent
variables. Note in Fig. 6 that the computational model is extended
to facilitate such a boundary condition and to avoid any inflow at the
outflow boundary due to flow recirculation. During the
computations, however, the N-S solver modifies the outflow
quantities to conserve net air flow rate entering the computational
domain.
**Solid Walls.** For the solid wall boundary conditions, including all internal surfaces struts, turning vanes etc., the standard wall-functions as recommended by Launder and Spalding (1974) are used for all dependent variables. For the energy equation, adiabatic wall boundary conditions are used everywhere.

**Numerical Solution.** The finite difference (discretized) equations corresponding to each dependent variable of the conservation equations are obtained using the finite control volume approach of Patankar (1981). In each transport equation, diffusion is approximated by central differences and convection modeled with a nearly second order accurate QUICK scheme of Leonard (1979). This is done to primarily minimize false numerical diffusion errors in the computation. The N-S solver uses the semi-implicit-pressure-linked-equations (SIMPLE) approach of Patankar (1981) to obtain numerical solution of the resulting system of algebraic equations.

**RESULTS AND DISCUSSION**

Calculation results from 3D CFD simulations for the three load conditions are compared with the measurements in the diffuser strut region, strut outlet plane and model outlet plane. In addition, comparison between the calculated and measured overall performance of the exhaust system in terms of total pressure loss and static pressure recovery is also made. These results are discussed in the following sections.

**Flow Field in the Entire Model.** Figure 7 shows computed streamlines in the whole model from inlet to outlet, looking from top. This gives a good understanding of the overall flow characteristics. At FSNL, shown in Fig. 7(a), the inlet flow with a high swirl angle hits the struts and creates large separations in the inter-strut passage, suggesting a large pressure drop in this area. The largest velocities are observed around the strut outlet as shown in red color, accompanied with a sudden deceleration at downstream. Streamlines in the plenum also indicate to have large vortices. On the contrary, flow field in FSFL, shown in Fig. 7(b), the highest velocity is observed at the diffuser inlet, decelerating along the diffuser passage much more smoothly compared with FSNL. This conversion of dynamic pressure into flow static pressure (pressure recovery) takes place along the whole passage from the model inlet to the plenum outlet.

**Flow Field in the Diffuser Section and Around Struts.**

**Velocity vectors around strut.** Figure 8 shows velocity vectors around a diffuser strut at mid-span from both calculations and LDV measurements for both FSNL and FSFL. At FSNL, shown in Fig. 8(a), flow approaches the strut leading edge (suction side) with the swirl angle of -50 degrees. This leads to a large flow separation region covering the entire pressure surface, extending further downstream. Key features of these low-load-flow phenomena are well predicted by calculation; the center position of the secondary flow vortex from calculations almost coincides with that obtained from measurements. The velocity vectors in the passage upstream region, however, show some discrepancies. At FSFL, shown in Fig. 8(b), the incoming flow has negligible incidence angle with the strut leading edge, and it moderately turns to the axial direction in the passage without creating any large flow separation. Agreement between calculation and the LDV measurement is reasonable for both flow direction and velocity magnitude.

**Static pressure distribution on strut surface.** Figure 9 shows static pressure distribution on strut surface. At FSNL, shown in Fig. 9(a), the large inflow angle to the suction side causes a higher pressure in the front half than that on the pressure side. This
tendency is quite well predicted, though the calculated static pressure is slightly higher than measurement on pressure side. For FSML, shown in Fig. 9(b), the flow still has large incidence at the suction side leading to a high suction-side pressure at the front end. Calculations again agree fairly well with the measurements on both sides for FSML. For FSFL, shown in Fig. 9(c), pressure distributions on both sides assume the normal strut airfoil design condition, and the overall distribution is well predicted. Pressure level is also reasonably calculated although it is 1-2kPa higher than experiment for pressure side and 2-3kPa for suction side.

**Total pressure contour at strut outlet.** Figure 10 compares measured and calculated total-pressure contours at the strut outlet plane. At FSNL, shown in Fig. 10(a), outlet total pressure is high downstream of the suction side as the flow deflects to this side at low load, and low downstream of the separated reversed flow region earlier presented in Fig. 8(a). A comparison of calculation and measurement shows that this flow field is well predicted. For FSFL, shown in Fig. 10(b), the high pressure region corresponding to each inter-strut passage almost connects circumferentially with that from adjacent passage. This indicates that the flow velocities are fairly large even in the wakes, except in the root and tip regions.

**Circumferential distribution of total pressure at strut outlet.** Figure 11(a) shows that for FSNL the mid-span circumferential distribution of total pressure at the strut outlet plane features sharp peaks of high total pressure region and flat bottoms of low pressure regions. This behavior results from flow concentration toward a very narrow stream on the strut suction side and flow separation in the remaining region. Positions of these peaks are correctly predicted by calculations. The total pressure difference between the peak and bottom predicted by the calculation matches reasonably with the experiment. For FSML, shown in Fig. 11(b), flow expands to almost full pitch of inter-strut passage except in the wake region. The wider top and very narrow bottom are, therefore, observed in the circumferential distribution of total pressure unlike in the case of FSNL. The total pressure difference between the peak and bottom is again quite well predicted by the calculations. For FSFL, shown in Fig. 11(c), calculations do predict the experimentally observed narrow and sharp wakes. The test data, however, show larger unsymmetry in circumferential direction than calculations. This circumferential asymmetry is possibly caused by a 90-degree complex flow turning in the plenum. Some of the small wakes observed in measurements indicate that the traverse interval used in testing was still too large to capture some of the narrower wakes in FSFL.

![Experiment](a)FSNL

![Calculation](b)FSFL

Figure 8  Velocity Vectors Around Struts
Figure 9 Static Pressure Distributions on Strut Surface

Figure 10 Total Pressure Contours at Strut Outlet (view from downstream)

Figure 11 Circumferential Distributions of Total Pressure at Strut Outlet

(a) FSNL
(b) FSML
(c) FSFL

Downloaded from http://gasturbinespower.asmedigitalcollection.asme.org/GT/proceedings-pdf/GT1998/78620/V001T01A033/2410039/v001t01a033-98-gt-111.pdf by guest on 02 December 2021
Figure 12 Velocity Vectors in Model Outlet Plane (view from downstream)

(a) FSNL  
(b) FSFL

Figure 13 Total Pressure Contours at Model Outlet (view from downstream)

(a) FSNL  
(b) FSFL

Figure 14 Total Pressure Distribution at Model Outlet
Flow Field Exiting the Plenum.

Velocity vectors in the model outlet plane. Figure 12 shows velocity vectors in the model outlet plane for both FSNL and FSFL, comparing experiments (5-hole probe measurements) and calculations. The calculation well predicts the existence of two large counter-rotating vortices in upper and lower halves, including the rough location of their centers. The direction of vortices coincides with that of secondary flow generated by turning from diffuser to plenum outflow. Calculated velocity vectors in the region between the two counter-rotating vortices differ from the experimental data which exhibit greater symmetry between the upper and lower halves. For FSFL, shown in Fig. 12(b), the direction of calculated vectors in the region between the two vortices becomes nearly horizontal and the two vortices come closer to being symmetric. This reproduces the measured velocity pattern fairly well.

Total pressure distribution in the outlet plane. Figure 13 shows total-pressure contours at the model outlet plane at both FSNL and FSFL. At FSNL, shown in Fig. 13(a), the pattern of the total pressure distribution is roughly predicted. The high-pressure region is seen along the right endwall spanning from top to bottom. Weak and broad pressure peaks are seen in each upper and lower half in the low velocity regions from center to the left side. For FSFL, shown in Fig. 13(b), again flow pattern is roughly reproduced by calculation as in FSNL. In this flow condition, several pressure peaks are found along the right endwall with large and weak peaks in the upper and lower halves corresponding to the low velocity regions. The calculated range of total pressure variation is about 10kPa, which is the same as experiment, but the calculated mean level is 3-4kPa higher than experiment.

Figure 14 shows line plots of total pressure distribution at the model outlet plane at three different heights for both FSNL and FSFL. In FSNL, calculation predicts total pressure level 1-2kPa larger than experiment except a big peak observed in the right side at mid height. In FSFL, the discrepancy between calculation and experiment becomes larger to be 3-5kPa. The tendency that the model outlet pressures are calculated larger than measurements is seen more or less for all load conditions.

Total-Pressure Loss and Static-Pressure Drop/Recovery. Figure 15 describes the variation of mean total pressure loss and static pressure drop/recovery along the flow passage from the model inlet to strut outlet, and to model outlet, derived from both experiment and calculation. Definitions of the both parameters are:

Total pressure loss coefficient = \( \frac{P_{oi} - P_{ol}}{P_{oi} - P_{i}} \)

Static pressure drop coefficient = \( \frac{P_{i} - P_{li}}{P_{oi} - P_{i}} \)

where \( P_{oi} \) is total pressure and \( P_{i} \) is static pressure, and subscript 1 is at the model inlet and \( L \) indicates local mean value for each location.

For the total pressure loss distribution shown in Fig. 15(a), the following observations are made:

a) In FSNL, most of the total pressure loss occurs in the front part or diffuser passage from model inlet to strut outlet. This is clearly due to large separation around the strut.

b) In FSML and FSFL, the loss in the front part reduces significantly, and is smallest for FSFL. Loss in the back part (from strut outlet to model outlet) in FSML and FSFL is a little larger than in FSNL, which is considered due to increased velocities in this region. For the whole passage, total-pressure loss is the smallest for FSFL.

c) Comparing the calculation with experiment, predicted losses for the front part agree fairly well with experiment for all loads, while the differences for the back part become larger.

For the static pressure drop or recovery shown in Fig. 15(b), principal observations are:

Figure 15 Total Pressure Loss and Static Pressure Drop Distribution

Figure 16 Total Pressure Loss and Static Pressure Drop with Load
a) In FSNL, large static-pressure drop is observed in the front part, which indicates the diffuser does not function with full efficiency in this no load condition.

b) For FSNL and FSFL, large negative pressure drop or positive pressure recovery is obtained in the diffuser, suggesting a satisfactory diffuser performance. The entire pressure recovery is largest when the exhaust system operates under FSFL.

c) Calculated values again agree well with experiment for the front part, the comparison in back part shows larger discrepancies for each load.

The variation of the total pressure loss and static pressure drop with change in load condition from FSNL to FSFL is depicted in Fig. 16 for both the front part (from model inlet to strut outlet) and the whole passage (from model inlet to model outlet). Both the total pressure loss and static pressure drop decrease significantly near the mid load operating condition. A better agreement between calculation and experiment in the front part for all load conditions is also seen here. It is important to note here that even with larger discrepancy in the model exit plane, a certain tendency important to design application is clearly observed that the calculation always underpredicts total pressure losses and static pressure drops compared with measurements.

In a gas turbine exhaust system, most of the total pressure loss is associated with the secondary flows in the stalled regions. Relatively, only a small part of the overall loss can be attributed to friction in the wall boundary layers. It is therefore logical to assume that any further improvement in the near-wall treatment, such as using a two-layer model instead of the wall-functions, will only marginally improve predictions. However, capturing the flow behavior away from the wall using a more advanced turbulence model to accurately simulate effects of streamline curvature and turbulence anisotropy due to flow separation and widely varying swirl could further bring predictions closer to measurements. Taking into consideration the need for computational efficiency for industrial design applications, a Reynolds stress transport model that combines accuracy with speed is desirable. Based on a recent study by Hirsch and Khodak (1995) and an earlier study by Sultanian, Neitzel and Metzger (1986), ASM seems to be a promising turbulence model for further improvement in CFD predictions in the stalled regions of a gas turbine exhaust system where pressure loss is primarily due to secondary flows.

CONCLUSIONS

This paper presents the results of both experimental and 3D CFD investigations in a scale model of an industrial gas turbine exhaust system (GE-MS9001E type) to better understand its complex flow field and to validate CFD prediction capabilities. Principal conclusions of this study are as follows:

1. The flow field in the present exhaust system largely varies with the gas turbine operating load condition. The trends of such variation in total pressure loss and static pressure recovery as well as the local flow features are reasonably predicted by the 3D CFD calculations.

2. In a quantitative comparison, CFD predictions are found to compare well with the experiments for strut surface pressure and strut outlet total pressure in the front part (diffuser section) of the model. At the model outlet plane, pressure levels are calculated somewhat higher than experiment, indicating that secondary flow losses in regions of turning vanes and the plenum are not fully captured by the turbulence model.

3. Results indicate satisfactory prediction accuracy for total pressure loss and static pressure recovery in the diffuser section. For the entire exhaust system including the plenum, the CFD predictions of these values are found consistently somewhat less accurate under wide operating load variations from full-speed-no-load (FSNL) to full-speed-full-load (FSFL).

4. Overall, the applied CFD method offers a useful design engineering tool capable of predicting complex gas turbine exhaust system flows including the quantitative prediction of the total pressure loss and static pressure recovery.

ACKNOWLEDGMENT

Authors are grateful to Ron Wesorick, Alan Walker, Dincer Ozgur and Tom Taylor of General Electric Company and Takashi Ikeda, Shoichi Hisa, Tadashi Kobayashi and Tadashi Tanuma of Toshiba Corporation for their continued support and testing during the course of this joint technology program, and also to the two companies for their permission to publish this work.

REFERENCES


