THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS
345 E. 47 St., New York, N.Y. 10017

91-GT-62





# Viscous Flow Computations for Compressor/Combustor Diffuser Design to allow Air Extraction for IGCC Systems

**AJAY K. AGRAWAL and TAH-TEH YANG**
Department of Mechanical Engineering
Clemson University, Clemson, SC 29634-0921

**ABSTRACT**

A computational procedure based on the solution of fully elliptic Navier-Stokes equations on a body-fitted non-orthogonal grid was used to obtain flow fields in annular diffusers with a suction slot at the inner and outer walls. The turbulence effects were simulated by high Reynolds number form of the k-$\epsilon$ model. The calculation method was used to modify an industrial gas turbine (GE MS-7001F) compressor/combustor annular diffuser to allow extraction of compressed airflow for coal gasification in simplified IGCC Systems. The air for gasification was extracted through a suction slot on the outer wall of the diffuser which was curved to improve the overall performance and to avoid flow separation; both of these insured by providing accelerated flow through the suction slot and nearly constant wall pressure downstream of the slot. Suction slot and outer wall geometries to result in the above conditions were determined by a trial and error procedure. The diffuser's performance was further improved by extracting 6% of the compressed air through a slot at the inner wall, kept straight due to structural constraints. The resulting diffuser arrangement was relatively insensitive to the upstream disturbances.

**INTRODUCTION**

The integration of an air-blown coal gasifier and a power generating gas turbine-steam turbine, forming Integrated Gasification Combined Cycle (IGCC) system, can be simplified by extracting compressed air, required in the coal gasifier, from the gas turbine compressor/combustor diffuser. The mass flow rate through the gas turbine's expander might be unaffected since the extracted air returns to the combustor as a constituent of the low-Btu coal gas. With this approach an existing gas turbine, typically designed for medium and high-Btu fuels, can accommodate low-Btu coal gas if part of the compressed air is extracted and possibly if the combustor is redesigned.

A compressor/combustor diffuser in the gas turbine is used to decelerate the flow exiting the compressor before it enters the combustor. An ideal diffuser achieves the required velocity reduction with minimum loss in the total pressure and with uniform, stable flow conditions at its outlet. The diffuser must also be insensitive to changes in the inlet conditions. Extracting large amounts of air for coal gasification may adversely affect the performance of the diffuser. However, required air extraction can be achieved without adverse effects by properly contoured diffuser wall(s) and air extraction passages like those of Griffith diffusers. Extraction is an integral part of the Griffith diffuser where the desired velocity distribution may be preserved throughout by design [Yang and Nelson, 1979]. This type of diffuser allows bifurcation of the flow while maximizing pressure recovery. The latter could be significant for overall gas turbine efficiency.

Griffith diffuser, named so because the pressure distribution along the diffuser wall is taken from the high lift airfoils designed by Griffith (1942), exhibits a low pressure inlet region, a slightly favorable wall pressure up to a slot region, a sudden rise in pressure across this slot region, followed by a slightly favorable wall pressure region towards the exit of the diffuser. Boundary layer suction is used to achieve the pressure jump across the slot.

For well-defined potential cores Yang and coworkers (Nelson et al., 1975 and Nelson and Yang, 1977) used an inverse solution method to design axisymmetric curved wall diffusers. Excellent correlations between computed and measured flow fields were obtained. In 1985 Yang extended this early analysis to include rotational velocity profile at the diffuser inlet (Yang, 1985). However, agreement between analytical predictions and measured values of wall pressure distribution was unsatisfactory. Subsequently, a Navier-Stokes equation solver with thin-layer approximation was used for inverse design of axi-symmetric flow passages (Ntone and Yang, 1986). The viscous flow analysis by Ntone and Yang (1986) did not involve passages with bifurcation.

In the present analysis a fully elliptic Navier-Stokes solver for turbulent flow was applied, through trial and error, to modify the geometry of an industrial gas turbine compressor/combustor annular diffuser and to develop an air extraction scheme for simplified IGCC systems. Additional computations were performed on the resulting diffuser arrangement to determine effects of the upstream disturbances. The computed flow field, wall pressure distributions and exit velocity profiles are presented here. Experimental verification is scheduled for 1991-92.

Presented at the International Gas Turbine and Aeroengine Congress and Exposition
Orlando, FL June 3-6, 1991

## TECHNICAL APPROACH

Geometry of the annular diffuser with suction slots is depicted in Figure 1. The flow field in this passage was obtained numerically from conservation equations of mass and momentum. In formulating the problem the following assumptions were made:

a) the diffuser geometry is axi-symmetric, therefore, effects of three-dimensional flow distortion at the inlet to the diffuser were not modeled,
b) the flow is incompressible, a valid assumption since the maximum Mach number, at the exit of the extraction slot, was below 0.3, and
c) the flow is adiabatic.

### Governing Equations

Using cartesian tensor notation, governing equations for incompressible turbulent flow were expressed in time-averaged form as

$$\frac{\partial}{\partial x_j}(\rho V_j) = 0 \tag{1}$$

$$\frac{\partial}{\partial x_j}(\rho V_j V_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu_{eff}\frac{\partial V_i}{\partial x_j}\right] \tag{2}$$

The effective viscosity, $\mu_{eff}$ was defined by the relation

$$\mu_{eff} = \mu + \mu_t \tag{3}$$

The turbulent viscosity, $\mu_t$ in Eq. (3) was obtained from the standard k-$\epsilon$ model of turbulence.

### Boundary Conditions

__Inlet.__ At the inlet of the diffuser, axial velocity, deduced from measured data, was specified while the radial velocity was assumed to be zero. The inlet turbulence intensity was taken as 6 percent. Inlet turbulent energy dissipation was calculated from

$$\epsilon = [C_D^{0.75}\ k^{1.5}]/\ell \tag{4}$$

where the length scale, $\ell$ was 3 percent of the annular gap.

__Outlet.__ Constant static pressure was specified at the diffuser and suction slot outlets. The specified pressure at the outlet of the slot was determined iteratively to result in the desired extraction mass flow rate.

__Walls.__ The wall function approach for turbulent flow was employed. This means that the region between a wall and a near wall node, placed outside the viscous sublayer, was bridged by the logarithmic velocity profile. Turbulence in the near wall region was assumed to be in local equilibrium (Launder and Spalding, 1974).

### Grid Generation

To facilitate simple and accurate representation of boundary conditions, a generalized curvilinear coordinate system that conforms to the physical boundaries of the diffuser was used. However, flexibility to control grid point distribution in complex domains, such as the branching passage shown in Figure 1, is rather limited. Such configurations are especially suited to multi-domain flow computations where the physical domain is sub-divided into several domains. Governing equations are solved in each domain with boundary conditions provided by the solution in the neighboring subdomains(s) [Karki and Patankar, 1986, Leschziner and Dimitriadis, 1989]. With this approach grids in each region can be generated independently.

In the present study the grid was generated by sub-dividing the computational domain. However, unlike Karki and Patankar (1986) or Leschziner and Dimitriadis (1989), the governing equations were solved on a single computational domain embedded with blocked regions (Figure 4). The grid was generated in two steps. The initial grid, generated algebraically, was modified using a Poisson equations solver incorporating control functions. The modified grid was used only if it decreased the grid non-orthogonality. Although the flow solver was applicable for non-orthogonal grids, improved grid orthogonality enhanced convergence.

### Solution Procedure

The governing equations for each dependent variable could be reduced to a single general form as

$$\overline{\nabla}\ (\rho\ \overline{V}_\phi + \overline{J}_\phi) = S_\phi \tag{5}$$

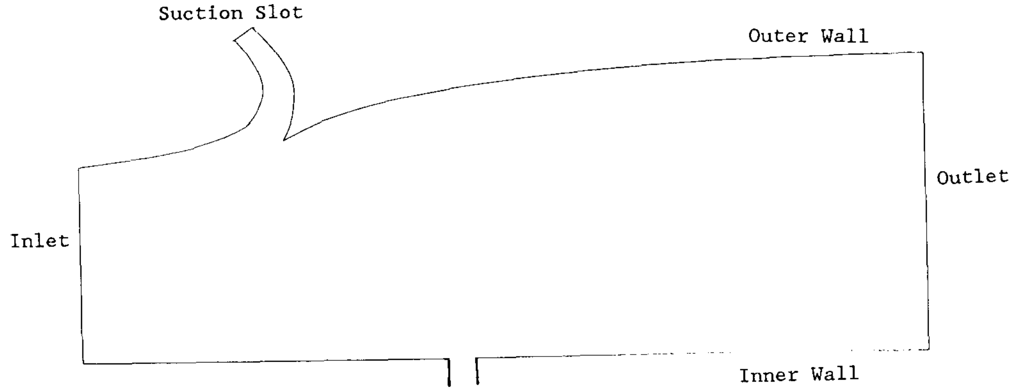


Figure 1. Compressor/combustor annular diffuser with suction slots.

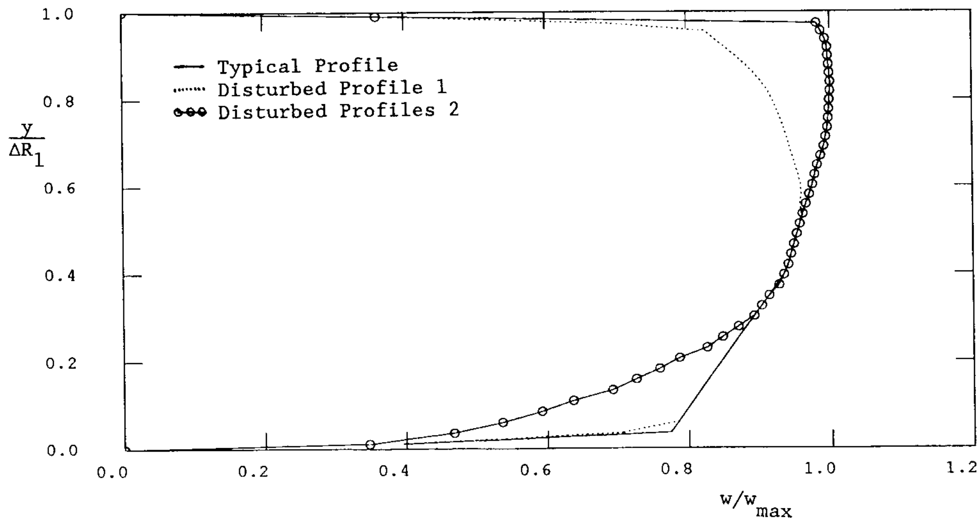


Figure 2. Axial velocity at the inlet to the diffuser.

where $\phi$ represents the dependent variable, $\bar{V}$ is the velocity vector, $\bar{J}_\phi$ is the diffusive flux vector and $S_\phi$ is the volumetric source of $\phi$. Integration of the generalized Eq. (5) results in the finite difference equations of the general form

$$a_P\phi_P = a_N\phi_N + a_S\phi_S + a_H\phi_H + a_L\phi_L + S_\phi \qquad (6)$$

where N, S, H and L represent neighbor points. Convection-diffusion terms in the transport equations were discretized using the hybrid scheme. The set of coupled non-linear equations were solved implicitly in an iterative manner. Coupling between velocity and pressure fields was provided by a variant of SIMPLE algorithm (Patankar, 1980) called SIMPLEST. Iterations were also required to fix static pressure at the exit of the slot to obtain specified extraction mass flow rate.

**RESULTS AND DISCUSSION**

In this section data and computed results for the existing straight wall diffuser are provided. Computed results for the curved wall diffuser with suction slot(s) follow.

**Straight Wall Diffuser.**

The straight wall diffuser considered for modification has an inlet radius ratio of 0.84, a length to inlet gap ratio of 4.2 and an outlet to inlet area ratio of 1.6. Approximate Reynolds number ($\rho\bar{w}\ D_h/\mu$) and mass averaged dynamic head at the inlet are $4\times10^6$ and 0.025 MPa, respectively. Axial velocity distribution at the inlet, deduced from measurements, is shown in Figure 2 (typical profile). For this profile the blocked area fraction B, defined as

$$B = 1 - \frac{1}{A}\int \left(\frac{w}{w_{max}}\right)\, dA \qquad (7)$$

where

$w_{max}$ - the maximum axial velocity at the inlet

was 0.09.

Computed flow field in the diffuser is shown in Figure 3. At the exit a block area fraction of 0.18 was obtained. The overall static pressure recovery coefficients, $C_P$ defined as

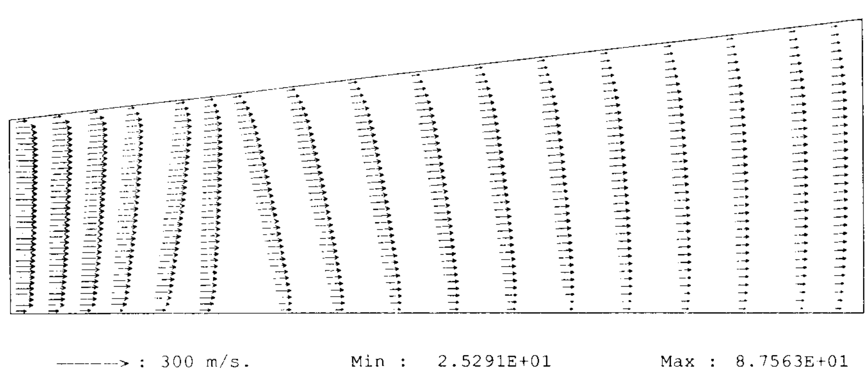


Figure 3. Flow field in the straight wall diffuser (no suction).

$$C_p = \frac{P_2 - P_1}{0.5\ \rho\ \bar{w}^2} \qquad (8)$$

where

$P_1$ - reference pressure

$P_2$ - local pressure

$\bar{w}$ - average axial velocity at the inlet

for the inner and outer walls were 0.46 and 0.89, respectively. The static pressure recovery coefficients at the inner and outer walls of the annulus were significantly different due to flow curvature in the vicinity of the diffuser inlet, as pointed out by Stevens and Williams (1980). Predicted $C_p$ values were as much as 10% higher than those measured by Stevens and Williams (1980).

**Curved Wall Diffuser.**

The computational grid for a typical geometry is shown in Figure 4 where details in regions impermeable to the fluid have been avoided. Standard deviation of about 25°, with the algebraic grid, was reduced to approximately 15° by modifying the grid using the Poisson equations solver. The average included angle for the resulting grid was close to 90°.

To assess numerical accuracy, calculations for a typical diffuser geometry were performed with 61 radial and 71 axial (61x71) grid points. 32% of the grid points were placed in the blocked region. A coarse 51x56 grid with 35% of the grid points in the blocked region was also used. Predicted static pressures and velocity fields from these two sets of computations differed by less than 1%. For identical geometry and boundary conditions, the mass flow rates through the diffuser and suction slot, as percent rate at the inlet, are summarized below:

| Grid Size | Main Diffuser | Suction Slot |
|---|---|---|
| 61x71 | 80.05 | 19.95 |
| 51x56 | 79.90 | 20.10 |

To ensure grid size convergence at conditions different from the test case, the finer 61x71 grid was used for all computations.

**Diffuser Geometry.** The axial location of the suction slot on the outer wall was determined from structural considerations of the GE MS-7001F gas turbine. Geometries of this suction slot and the diffuser outer wall were determined by trial and error to achieve nearly constant pressure downstream of the suction slot to eliminate the possibility of flow separation. During the trial and error procedure efforts were made to maximize the overall pressure recovery in the diffuser and minimize pressure drop through the suction slot. The inner wall of the diffuser was kept straight due to structural constraints.

Figure 5 shows static pressure recovery coefficient, $C_p$ (defined by Eq. 8) along the outer wall for two typical wall shapes (also shown in Fig. 5). Upstream of the slot the wall pressure decreased as the flow accelerated due to suction. Downstream of the slot the wall pressure remained nearly constant particularly for geometry 'B', which was considered for further investigation. For geometry 'B' the predicted flow field, in terms of the streamfunction distribution, is shown in Figure 6. Evidently the flow bifurcates without separation. At the diffuser exit a relatively high blocked area fraction of 0.21 was obtained. The flow structure close to the inner wall did not change appreciably when the geometry of the suction slot was altered.

**Suction at the Inner Wall.** Outer wall suction and curvature did not control boundary layer at the inner wall. To accomplish control, 6% of the compressed air was extracted through a slot perpendicular to the inner wall. In the gas turbine under consideration, this air could be routed for cooling the turbine rotor. Pressure distribution along the inner wall was affected considerably due to inner wall suction as shown in Figure 7. Suction at the inner wall had only a marginal effect on pressure distribution along the outer wall as shown in Figure 8. The outer wall pressure downstream of the suction slot remained nearly constant, thereby avoiding the possibility of flow separation due to upstream disturbance. Suction at the inner wall considerably improved axial velocity profile at the exit of the diffuser as seen in Figure 9. The exit blocked area fraction decreased to 0.14.

**Upstream Disturbance.** Due to upstream conditions two or three-dimensional disturbances can occur at the diffuser inlet. However, in the present study disturbance was characterized by a defect in the axial velocity at the diffuser inlet. Two types of upstream disturbances, one towards the outer wall and one towards the inner wall, were considered. The disturbed axial velocity profiles at the inlet are shown in Figure 2 along with the typical profile. As seen in Figures 10 and 11, the upstream disturbance did not affect the wall static pressure recovery coefficients downstream of the suction slots, especially at the outer wall. This is desirable for improving the operating range of the diffuser. Axial velocity profiles at the exit of the diffuser, with and without upstream disturbances, are shown in Figure 12. It is seen that the flow at the exit is affected by upstream disturbances, however it was far from separation.

## CONCLUDING REMARKS

Calculations show that 20% of the compressed air could be bifurcated through a suction slot on the outer wall without flow separation. Geometries of this suction slot and outer wall were determined to improve the overall performance of the diffuser and to avoid flow separation. However, due to adverse pressure gradient, the flow might be prone to separation near the inner wall. Attached flow could be insured throughout if the air for cooling the turbine rotor was extracted through a slot at the inner wall. The resulting diffuser arrangement was relatively insensitive to the upstream disturbances. Further improvements in the diffuser's performance may be anticipated if the structural constraint of a straight inner wall is relaxed, thereby allowing for a properly contoured inner wall.

## ACKNOWLEDGEMENT

This work was performed under the Morgantown Energy Technology Center, Department of Energy contract No. DE-AC21-89 MC26041 and with assistance from GE Gas Turbine Technology Department, Schnectady, New York. PHOENICS computational fluid dynamics code by CHAM of N.A. was used for computations.

## REFERENCES

Griffith, A.A., (1942), "Reducing Surface Friction between a Solid and a Fluid," Patent 578,763 (BR.), No. 11973.

Karki, K.C., and Patankar, S.V., (1986), "Use of Subdomains for Flow Computations in Complex Geometries," in Numerical Methods in Heat Transfer, eds. J.L.S. Chen and K. Vafai, ASME HTD vol. 62, pp. 7-12.

Launder, B.E., and Spalding, D.B., (1974), "The Numerical Computation of Turbulent Flows," Computer Methods in Applied Mechanics and Engineering, vol. 3, pp. 269-289.

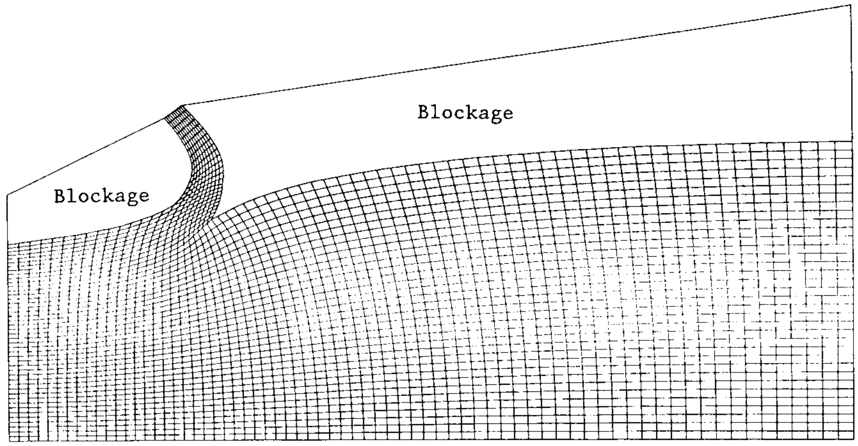


Figure 4. Computational grid

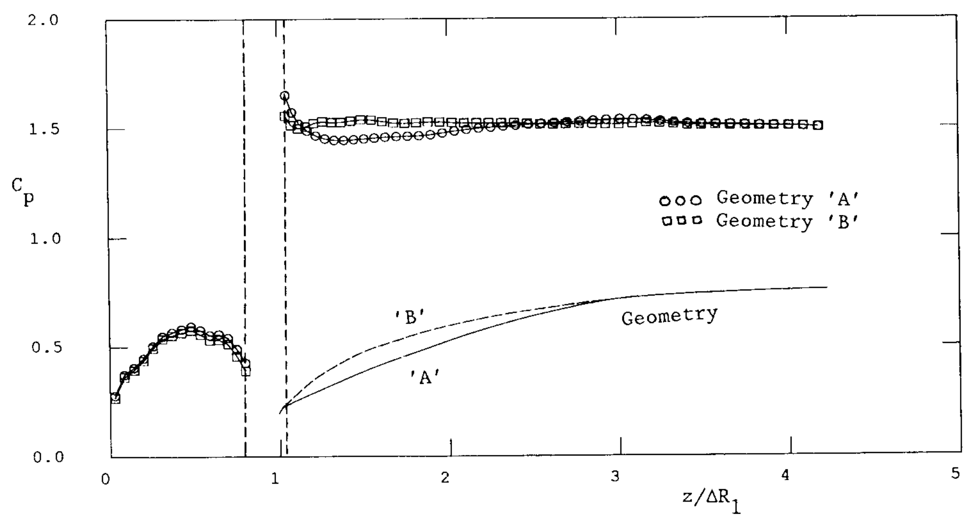


Figure 5. Static pressure recovery coefficient along the outer wall.

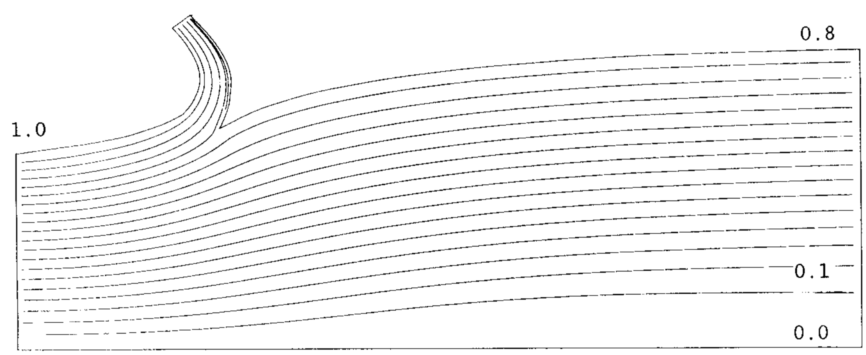


Figure 6. Streamfunction distribution in the diffuser with extraction at the outer wall.

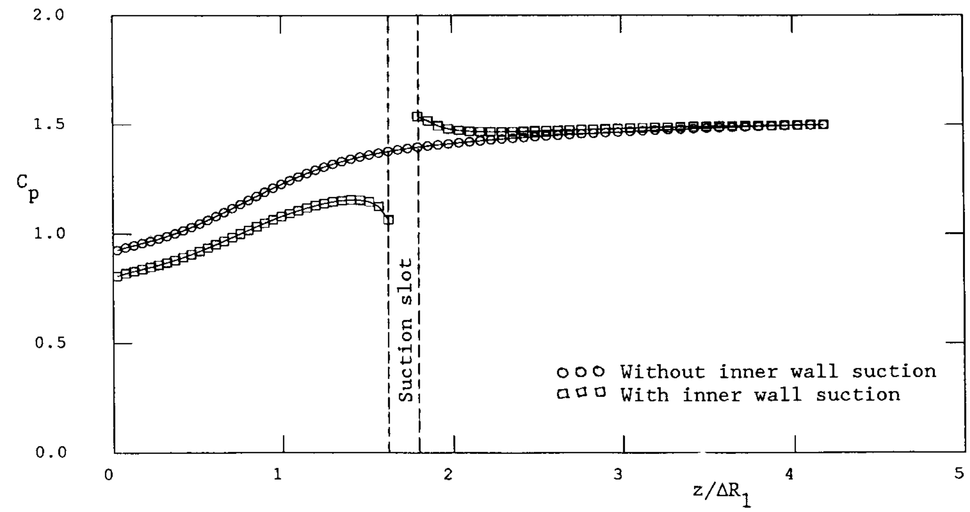


Figure 7. Static pressure recovery coefficient along the inner wall.

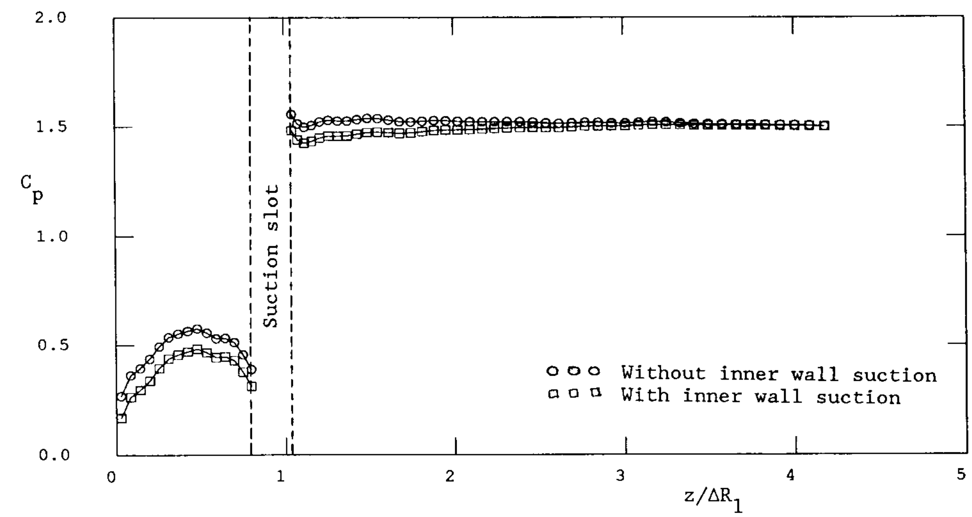


Figure 8. Static pressure recovery coefficient along the outer wall.

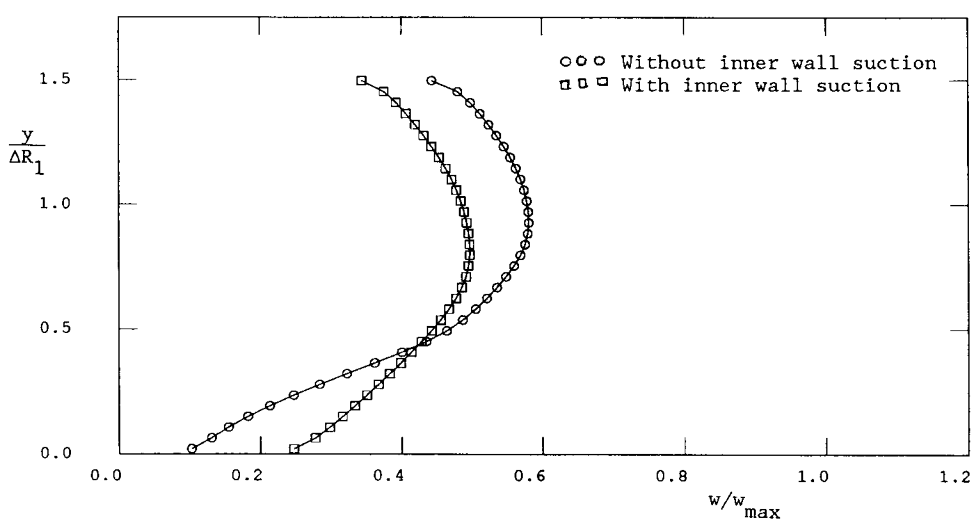


Figure 9. Axial velocity at the exit of the diffuser.

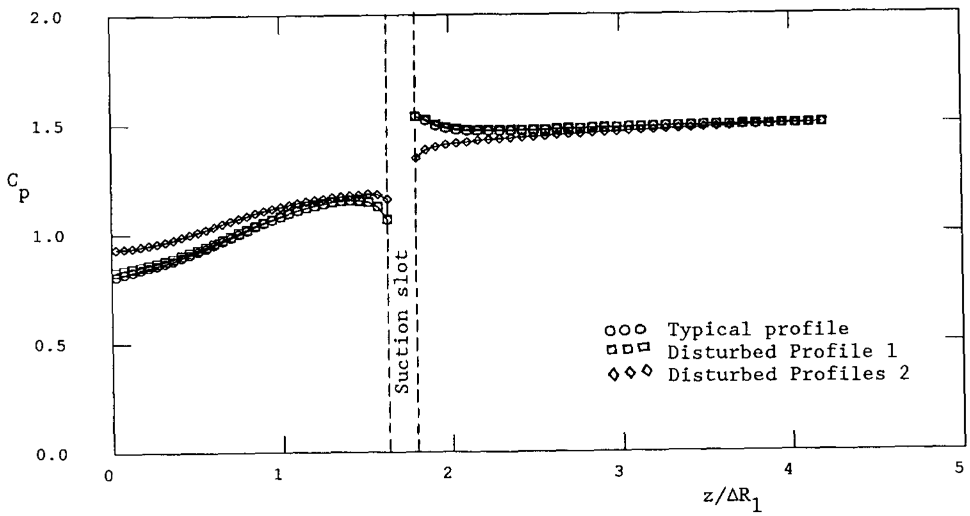


Figure 10. Static pressure recovery coefficient along the inner wall.

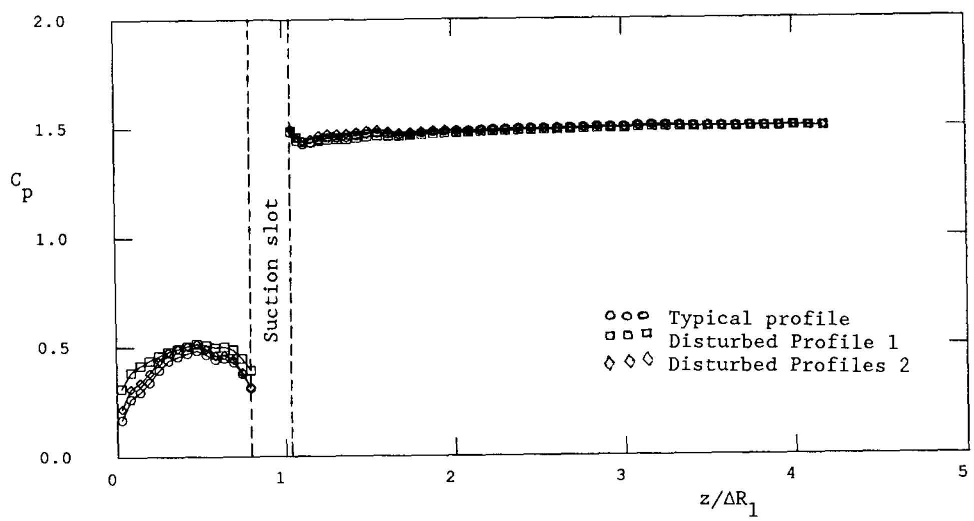


Figure 11. Static pressure recovery coefficient along the outer wall.

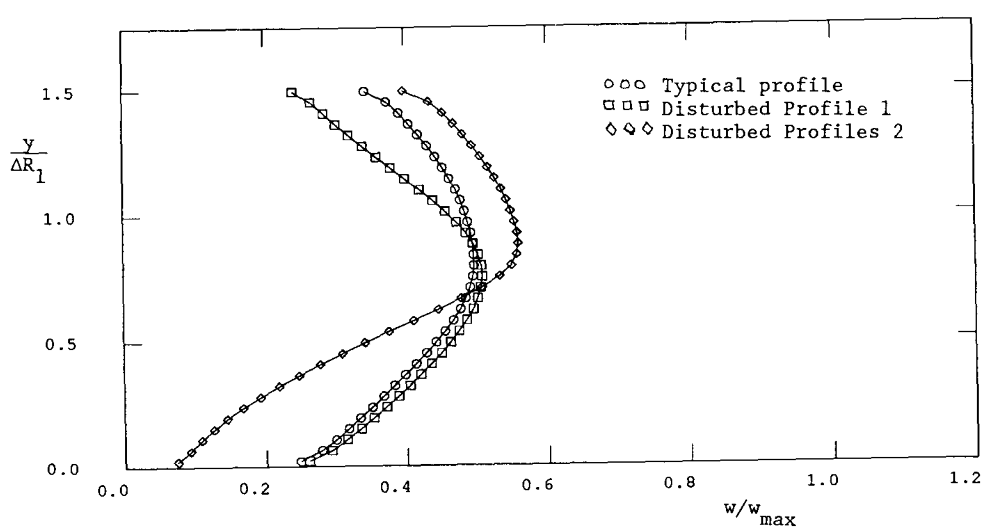


Figure 12. Axial velocity at the exit of the diffuser.

Leschziner, M.A., and Dimitridis, K.P., (1989), "Computation of Three-Dimensional Turbulent Flow in Non-Orthogonal Junctions by a Branch-Coupling Method," Computers and Fluids, vol. 17, No. 2, pp. 371-396.

Nelson, C.D., Yang, T., and Hudson, W.G., (1975), "The Design and Performance of Axially Symmetric Contoured Wall Diffusers Employing Suction Boundary Layer Control," Journal of Engineering for Power, vol. 97, pp. 125-130.

Nelson, C.D., and Yang, T., (1977), "Design of Branched and Unbranched Axially Symmetrical Ducts with Specified Pressure Distribution," AIAA Journal, vol. 15, pp. 1272-1277.

Ntone, F., and Yang, T., (1986), "Inverse Design of Axisymmetric Flow Passages using Compressible Viscous Flow Theory," Communications in Applied Numerical Methods, vol. 2, pp. 83-89.

Patankar, S.V., (1980), Numerical Heat Transfer and Fluid Flow, Hemisphere, Washington, D.C.

Stevens, S.J., and Williams, G.J., (1980), "The Influence of Inlet Conditions on the Performance of Annular Diffusers," Journal of Fluids Engineering, vol. 102, pp. 357-363.

Yang, Tah-teh, and Nelson, C.D., (1979), "Griffith Diffusers," Journal of Fluids Engineering, Vol.101, pp. 473-477.

Yang, T., (1985), "An Investigation of High Performance, Short, Thrust Augmenting Ejectors," Journal of Fluids Engineering, vol. 107, pp. 23-30.