Comparison of Heat Transfer Measurements with Computation for Turbulent Flow Around a 180 Degree Bend

D. L. BESSERMAN and S. TANRIKUT
Pratt & Whitney
Group Engineering and Technology
East Hartford, CT 06040

ABSTRACT

Results of detailed heat transfer measurements are presented for all four walls of a 180° 1:1 aspect ratio duct. Experiments using a transient heat transfer technique with liquid crystal thermography were conducted for turbulent flow over a Reynolds numbers range of 12,500-50,000. Computational results using a Navier-Stokes code are also presented to complement the experiments. Two near-wall shear-stress treatments (wall functions and the two layer wall integration method) were evaluated in conjunction with k-ε formulation of turbulence to assess their ability to predict high local gradients in heat transfer. Results showed that heat transfer on the convex and concave walls is a manifestation of the complex flow field created by the 180° bend. For the flat walls, the streamwise average Nusselt number increases to approximately two times the fully developed turbulent flow value. Ninety degrees into the bend, the importance of the cross-stream gradients is evident with the Nusselt number varying from approximately one to three times the fully developed turbulent flow value. The numerical predictions with two-layer wall integration k-ε turbulence model show very good agreement with the experimental data. These results reinforce the need to accurately predict local heat transfer rates in cooling passages of advanced turbine airfoils to enhance the durability of these components.

NOMENCLATURE

D Hydraulic diameter
g Gravitational constant
Gr Grashof number, \((g\beta(T_w - T_f)D^3)/\nu^2\)
h Heat transfer coefficient
k Thermal conductivity
m Mass flowrate
Re Reynolds number, \(mD/\mu/A\)
T Temperature
\(u_r\) Friction velocity \((\sqrt{u_T/\nu})\)
V Velocity
VM Mean coolant velocity
x Streamwise distance
y Coordinate normal to solid surface

INTRODUCTION

In order to increase the efficiency of gas turbine engines, current trends include increasing the combustor exit temperature as well as minimizing the coolant flow rate to the internal turbine blade passages. As a result, detailed knowledge of heat transfer associated with internal cooling passages is necessary to prevent local "hot spots" and achieve acceptable turbine airfoil metal temperatures and life.

A square cross-sectioned duct with a 180° bend, which is representative of a turbine blade internal coolant passage, has been studied (Wagner et al., 1989, Wagner et al., 1990, and Fan and Metzger, 1987) in both stationary and rotating environments. In these investigations, data were acquired with models in which a single copper element was used to measure the heat transfer for each side wall. Although the copper elements were segmented in the streamwise direction, the flat surface of the 180° bend was fabricated from either one or two copper segments. Therefore, both large streamwise and cross-stream gradients could not be resolved with these models. However, the importance of the secondary flow fields generated in the duct was demonstrated. In all cases the heat transfer coefficients were approximately a factor of two greater in the bend as compared to fully developed turbulent pipe flow.
More detailed heat transfer measurements have been made in 180° bends by Baughn et al. (1986) for circular ducts and by Johnson and Launder (1985) for square cross-sectioned ducts. Baughn et al. (1986) reported that the maximum heat transfer occurred 90° into the bend with the variation from the inside to the outside of the bend ranging from one to three times the turbulent pipe flow value. The classical double-cell secondary vortex which is created causes the slow moving fluid near the duct wall to move toward the inside of the bend and the cooler faster moving fluid near the plane of symmetry to "impinge" on the outside of the bend. In addition, streamline curvature tends to increase turbulent mixing on the outside of the bend. Therefore, the large increases in heat transfer on the outside of the bend were attributed to both the secondary flow and to increased turbulent mixing. Johnson and Launder (1985) also reported maximum heat transfer 90° into the bend where the heat transfer on the concave walls was 2.2 times larger than the convex walls for Re = 56,000. Larger variations between the concave and convex walls were reported for lower Reynolds numbers.

To date, computations have focused on predicting the flow field within ducts with a 180° bend. Monson et al. (1989) solved the Navier-Stokes equations for an incompressible fluid using a Prandtl mixing length zero-equation turbulence model. Comparisons between experimental measurements and the numerical solution indicate that the mean flow was well predicted in the first half of the bend since the flow field is mainly driven by the pressure field. However, the separation bubble which was measured on the inner wall near the end of the bend was not accurately predicted and the boundary layer thicknesses were overpredicted.

A series of modeling refinements were incorporated by Choi et al. (1989) in an effort to more accurately predict the flow. Replacement of wall functions by a fine grid used to resolve the buffer and viscous sub-layers and replacement of the standard k-ε turbulence model by an algebraic second-moment closure provided significant improvements in the agreement between predictions and measurements. However, Choi et al. (1989) suggest that accurate predictions of the primary flow may be dependent on the ability to predict secondary flow which may require the use of a turbulent transport model in the viscous sublayer.

Significant durability improvements in turbine airfoil designs beyond current state-of-the-art are dependent upon both an increased understanding of the heat transfer occurring within an internal cooling passage as well as the ability to accurately predict these heat transfer coefficients. Therefore, the current work included obtaining detailed heat transfer measurements in the turn region of a square cross-sectioned duct with a 180° bend as well as numerically predicting the heat transfer associated with this geometry. The numerical predictions incorporated a k-ε turbulence model with the wall shear stress computed via either wall functions or the two-layer wall integration method. It was the goal of this work to compare the experimental results with the computations to assess the predictive capability of the Navier-Stokes code in the high gradient regions.

**EXPERIMENTAL METHOD**

A schematic diagram of the geometry used in the current investigation is shown in Fig. 1. The apparatus was designed with 29.7mm (1.17in) radius of curvature on the convex wall. Since the concave wall has a smaller radius of curvature (27.4mm, 1.08in), the model has a flat surface at the 90° location. The channel features were machined from a nominally 25.4mm (1 in) thick sheet of clear acrylic. The square cross-section duct was formed by sandwiching this piece between two other sheets. Since the acrylic was not exactly 25.4mm thick, the resulting height and width of the channel were 24.4 and 25.4mm (0.96in and 1in), respectively. This yielded a hydraulic diameter of 25.1mm (0.99in).

![Fig. 1 Schematic Diagram of Square Cross-Sectioned Duct With 180° Turn.](image-url)

A transient technique described in Metzger and Larson (1986) was used to measure the heat transfer in the model shown in Fig. 1. Unlike copper model data which would yield average values for each element, this technique provided a means to obtain true local measurements. Heat transfer coefficients were obtained by solving the one-dimensional transient conduction equation for a semi-infinite wall subject to a convective heat transfer boundary condition. For each location in which the heat transfer coefficient was obtained, the time required for the wall to reach a specified temperature was recorded by observing the color change of liquid crystals applied to the surface of interest. An iterative solution scheme was employed to determine the heat transfer coefficients from these time-temperature pairs.

Inherent in the problem formulation is the assumption that the convective heat transfer coefficient is constant with time. Since Gr/Re<1 for these experiments, the results are independent of the wall-to-fluid temperature difference (Tg - Tf) and so the above assumption is believed to be valid. Due to the low thermal conductivity of acrylic (k = 0.19 W/m K) and the 25.4 mm wall thickness of the model, the semi-infinite wall assumption was valid. It was estimated that two-dimensional conduction effects within the acrylic affected the results by less than 3% (Vedula and Metzger, 1988).

A perforated plate and screen were placed in the small inlet plenum shown in Fig. 1. The velocity profile was surveyed three hydraulic diameters downstream of the screen, which indicated a variation of 20% between the maximum and average velocity at this
location. The distance from exit of the inlet plenum to the entrance to the turn was fourteen hydraulic diameters. As a result, the velocity profile entering the bend was partially developed which is representative of the flow conditions present in turbine airfoil internal passages.

The model was sprayed with 1.1°C (2°F) temperature band chiral nematic liquid crystals. A thin misting of black paint (thickness less than 0.025mm) was sprayed on top of the liquid crystals to provide a sharp contrast for viewing the color changes. The effect of conduction through this paint on the results was assumed to be negligible.

To initiate the test, heated air at 121°C (250°F) was diverted from a bypass line through the initially isothermal, 21°C (70°F), model. The time response of liquid crystal color changes were recorded on film tape. The air flow rate was measured with a venturi and the inlet and exit air temperatures at the channel centerline were recorded with thermocouples. At each streamwise location, the centerline temperature was linearly interpolated from these measured temperatures. The local mixed mean fluid temperature was assumed to be negligible.

Due to the plane of symmetry about the centerline of the duct, results are presented for only one of the two flat surfaces. Henceforth this surface shall be referred to as the top surface. In the turn region of the top surface, data were reduced at six degree intervals, in the streamwise direction, with seven equally spaced grid points across the width of the duct. This level of resolution provided a detailed mapping of the heat transfer coefficients which could be compared with the numerical predictions. The heat transfer coefficients on the concave and convex surfaces of the bend were determined at selected streamwise locations with five equally spaced data points across the width of these surfaces.

NUMERICAL PROCEDURE

The governing equations of continuity, momentum, and energy were solved using the Navier-Stokes code of Rhee (1986) for the same geometry used in the experiments. This is a pressure-based implicit procedure which solves the full Navier-Stokes equations in general coordinates, thus allowing use of body-fitted coordinate systems.

In Rhee's approach, the preliminary velocity field is first obtained from the momentum equations with a preliminary pressure field. Since this preliminary velocity field does not satisfy the continuity equation, pressure correction equations are solved to establish a new velocity and pressure field which does satisfy the continuity equation. The momentum and continuity equations are coupled through this pressure correction procedure. Then, the energy and turbulent scalar equations are solved in turn. To date, the procedure has been used in a wide variety of applications including two- and three-dimensional viscous and inviscid flows showing good comparison with experimental data (e.g., Rhee, 1986).

Two near-wall shear-stress treatments were evaluated in conjunction with the two-equation k-ε formulation of turbulence to assess their ability to predict high local gradients in heat transfer. In one case, the governing equations near the wall were solved by employing generalized wall functions which assume that the boundary layer velocity profile has the universal "law-of-the-wall" profile (Launder and Spalding, 1974). The turbulent kinetic energy (TKE) adjacent to the walls was computed from the TKE conservation equation. In the other case, the two layer wall integration method was used in which the governing equations are solved to the wall (Dash et al., 1983). Near the wall, the classical Van-Driest mixing length formulation was used. This region was patched with two-equation k-ε turbulence model at y+ = 50.

For the wall function simulation, a uniform grid of 30 x 30 was used for each cross-stream plane. The streamwise grid locations are shown in Fig. 2a. In order to resolve the viscous and buffer sub-layers, a non-uniform grid was employed for the two-layer wall integration simulation. The plane of symmetry in the duct was exploited in this case with a cross-stream grid of 59 x 30 as shown in Fig. 2b.

A no-slip boundary condition was applied to the surfaces of the duct for the momentum equations and a constant temperature boundary condition was used for the energy equation. A flat inlet velocity profile was specified for the computations presented below. Simulations with wall functions were performed with both a flat inlet velocity profile and using the velocity profile obtained from LDV measurements three hydraulic diameters downstream of the inlet plenum. These predictions indicated that the streamwise velocity profiles entering the bend were within 8% of the measurements for both inlet velocity profiles, within the experimental error of the measurements.

DISCUSSION OF RESULTS

To aid in the understanding of the duct heat transfer characteristics, the flow fields predicted within the duct are discussed first. Liquid crystal data, numerical predictions, and previous copper model data are then compared for the top, outer (concave) and inner (convex) surfaces of the duct.
Finally, detailed comparisons between the data and the predictions are highlighted with attention focused on the high gradient regions.

Flow Patterns
As flow approaches the bend, the velocity profile is characteristic of developing flow with a constant velocity in the core region of the flow. At the entrance of the bend, the peak velocity occurs near the inside of the bend which would be anticipated from inviscid flow theory. The predicted streamwise velocity profiles are shown in Fig. 3 for the plane of symmetry and parallel planes, at the 90° location. For both near wall shear-stress treatments, the maximum streamwise velocity has shifted toward the outside wall at the plane of symmetry (3a). However, the predictions in which the turbulent shear stress is integrated to the wall show a larger variation across the width of the duct. With increasing depth into the duct, the peak velocity shifts toward the inside (convex) wall.

These peaks in the streamwise velocity profile can be understood by observing the secondary flow field which develops within the duct at this location. Shown in Fig. 4, a classical single-cell vortex has developed in the region from the symmetry plane to the top surface of the duct. The secondary motion indicates that the flow migrates from the inner wall toward the outer wall near the symmetry plane. This motion results in an increase in the streamwise velocity near the outer (concave) wall and a corresponding decrease near the inner (convex) wall. Likewise, near the top wall, streamwise velocity increases near the inner (convex) wall due to the effect of the secondary flow. The non-uniform grid associated with the wall integration method was able to resolve small corner vortices (Fig. 4b).

As the flow progresses around the turn, the single-cell vortex is transformed into a more complex secondary flow structure (Fig. 5). In both simulations, two large circulation cells are generated at the 180° location in the top half of the duct (a total of four main circulation cells develop within the duct). However, a more complex flow structure is again predicted with the wall integration model.

Flow visualization tests, utilizing fluorescent dye, indicated the same overall flow features. Specifically, two and four recirculation cells were observed at the 90° and 180° locations, respectively. These results indicated that both sets of computations predicted the overall flow structure.

Fig. 3 Streamwise Velocity Profiles (A) Symmetry Plane (B) 25% Depth (C) 50% Depth (D) 75% Depth.

Fig. 4. Secondary Flow Patterns at 90° Plane (A) Wall Functions (B) Wall Integration.

Fig. 5. Secondary Flow Patterns at 180° Plane (A) Wall Functions (B) Wall Integration.
Heat Transfer Data

Liquid crystal data were taken for three coolant flowrates corresponding to a Re range of 12,500 - 50,000. In all cases, the highest heat transfer, on the top surface, occurred along the outside edge near the 90° location. With increasing Reynolds number, the peak shifted toward the entrance of the bend. However, the peak always occurred in the region where the outside wall is straight.

Since the current investigation used the same geometry as that used by Wagner et al. (1990), a direct comparison of the results can be made. In Fig. 6, the liquid crystal data, for the top surface, has been averaged across the width of the duct and is shown versus dimensionless streamwise distance (x/D). Copper model data (Wagner et al., 1990) are shown as straight line segments which correspond with the length of the copper segments. The Nusselt number for fully developed turbulent flow in a smooth tube with constant wall temperature is noted in the figure for reference (Kays and Crawford, 1980). The correlation used to obtain Nu∞ is

\[ \text{Nu}_\infty = 0.021 \text{ Pr}^{0.5} \text{ Re}^{0.8} \]

The liquid crystal data indicate a large increase in heat transfer from the 0° to 90° locations and a gradual decline in the second half of the turn. The copper model data indicate an increase in heat transfer throughout the turn which is consistent with the gradual decline in heat transfer obtained in the current study from the 90° to 180° locations. In order to make valid comparisons between the copper model data and the liquid crystal data, the liquid crystal results have been spatially averaged over regions corresponding to the copper model sections. In the turn region, Nu measured with the liquid crystal technique was lower than the copper model data by approximately 25% for Re = 12,500 and by 10% for Re = 50,000. Comparison of copper model data with the fully developed turbulent pipe flow value prior to the bend entrance indicate that for Re = 12,500 and Re = 50,000 Nu was approximately 20% higher than the correlations would predict. On the other hand, the liquid crystal data was lower than pipe flow by 5% for Re = 12,500 and was higher than pipe flow by 8% for Re = 50,000.

For Re = 25,000, the Nusselt number of the liquid crystal data, copper model data, and the fully developed pipe flow value were within 5% prior to the bend. As a result, excellent agreement was obtained for this case in the turn region with a difference of less than 3% between the current measurements and those of Wagner et al. (1990). Thus, the level of agreement between the liquid crystal and copper model data in the turn region is directly related to the agreement prior to the bend.

Predictions from the numerical simulations have been compared with the liquid crystal data in Fig. 7 for the top surface. Excellent agreement was obtained between the Nusselt number with wall functions and that with wall integration prior to the bend where the flow is characteristic of almost fully developed turbulent flow. In the turn region, both predictions exhibit the same characteristic Nu curve shape with streamwise location. The minimum heat transfer was predicted to occur where the flow enters the bend and can be attributed to the reduced heat transfer associated with the separation of flow from the concave (outer) wall. The maximum heat transfer was predicted at the 90° location with a secondary peak at the 180° location. The simulation which employed the wall integration turbulence model accurately predicted the streamwise averaged Nusselt numbers with less than 2% difference between the maximum values. However, the wall function simulations underpredict heat transfer in the high gradient regions associated with the entrance of the duct and the turn region. For this case, Nu was underpredicted by approximately 20% in the turn.

The spanwise averaged results for the outside (concave) wall indicate large variations with streamwise location around the turn (Fig. 8). At the 90° location, the flow impinges on the outside wall which results in the first peak in heat transfer. The subsequent reduction is attributed to separation at approximately 135°. The increase in heat transfer beyond the separation region is characteristic of the high heat transfer associated with reattachment of the flow. This figure also demonstrates that the simulation with wall functions predicted smaller gradients in heat transfer than either the predictions with wall integration or the liquid crystal measurements. In this case, copper model data of Wagner et al. (1990) provide accurate information regarding the average heat transfer for each copper segment but fail to provide information regarding the large variations which occur in the streamwise direction.

Fig. 6 Comparison of Spanwise Average Liquid Crystal and Copper Model Data (Wagner et al. 1990) for the Top Surface (A) Re=12,500, (B) Re=25,000, (C) Re=50,000.
Comparison of the liquid crystal data and wall integration predictions indicates some differences in the second half of the turn. The large variations across the width of the duct predicted with wall integration turbulence model can be understood by examining the secondary flow field at the 180° location (Fig. 5). Heat transfer near the outside wall is enhanced by the impingement of the flow from the recirculation cell near the corner. The subsequent reduction can be attributed to the relatively slow moving warm fluid associated with the interaction of the two recirculation cells near the outside wall. Heat transfer near the inside wall has been enhanced beyond the fully developed turbulent pipe flow value by the impingement and acceleration of the secondary flow on the top surface in this region. The secondary flow fields predicted by the simulations employing wall functions do not seem to affect the local heat transfer distribution. (Formulation of an anisotropic turbulence model may improve the local predictions in this region.)

CONCLUSIONS

Comparisons between predictions and measurements of the flow field and heat transfer within a duct with a 180° bend have yielded the following results and observations.
Legend

1. Both turbulence model near wall treatments adequately predict the overall and secondary flow features.
2. Liquid crystal data indicate that large streamwise and cross-stream gradients in heat transfer exist in the turn region of a duct with a 180° bend.
3. Although a coarser mesh can be used with wall functions (which would reduce computational time), these predictions failed to accurately predict the heat transfer in the high gradient regions.
4. Predictions which solve the governing equations to the wall show very good agreement with liquid crystal data.
5. Further analysis of the turbulence model is necessary to resolve the differences between the predictions and the measurements on the convex surface.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge Pratt & Whitney for permission to publish the results of this study. The plexiglas model used to acquire the liquid crystal data was borrowed from United Technologies Research Center. The authors gratefully acknowledge Scott Marquis and Glenn Bartkowski for their assistance in the data reduction. The authors are appreciative of the support and guidance provided by their colleagues at Pratt & Whitney and United Technologies Research Center.

REFERENCES


