REVIEW

Progress towards understanding and predicting heat transfer in the turbine gas path

Robert J. Simoneau and Frederick F. Simon
NASA Lewis Research Center, Cleveland, OH, USA

A new era is dawning in the ability to predict convection heat transfer in the turbine gas path. We feel that the technical community now has the capability to mount a major assault on this problem, which has eluded significant progress for a long time. In this paper we hope to make a case for this bold statement by reviewing the state of the art in three major and related areas, which we believe are indispensable to the understanding and accurate prediction of turbine gas path heat transfer: configuration-specific experiments, fundamental physics and model development, and code development. We begin our review with the configuration-specific experiments, whose data have provided the big picture and guided both the fundamental modeling research and the code development. Following that, we will examine key modeling efforts and comment on what will be needed to incorporate them into the codes. In this region we will concentrate on bypass transition, three-dimensional endwalls, and film cooling. We will then review progress and directions in the development of computer codes to predict turbine gas path heat transfer. Finally, we will cite examples and make observations on the more recent efforts to do all this work in a simultaneous, interactive, and more synergistic manner. We will conclude with an assessment of progress, suggestions for how to use the current state of the art, and recommendations for the future.

Keywords: heat transfer; computational fluid dynamics; modeling; turbine; transition; three-dimensional flow; film cooling

Introduction

A new era is dawning in the ability to predict convection heat transfer in the turbine gas path. We feel that the technical community now has the capability to mount a major assault on this problem, which has eluded significant progress for a long time. The complexity of the problem is illustrated in Figure 1. The flow physics include transitional flows, three-dimensional (3-D) intersections, film cooling, wakes, tip and hub leakage, strong crossflows, rotational forces, and others. Here, we will focus on only the first three: transition, 3-D endwalls, and film cooling. We will review the state of the art in three areas that we believe are indispensable to the understanding and accurate prediction of turbine gas path heat transfer: configuration-specific experiments, fundamental physics and model development, and code development, all of which are related as illustrated in Figure 2.

Address reprint requests to Dr. Simoneau at NASA Lewis Research Center, Cleveland, OH 44135, USA.

Received 3 April 1992; accepted 8 October 1992

Historically, experimental work and the modeling of the physics have preceded the complex computational predictions of the phenomena. This is particularly true with respect to heat transfer. We will follow this historical approach and begin our

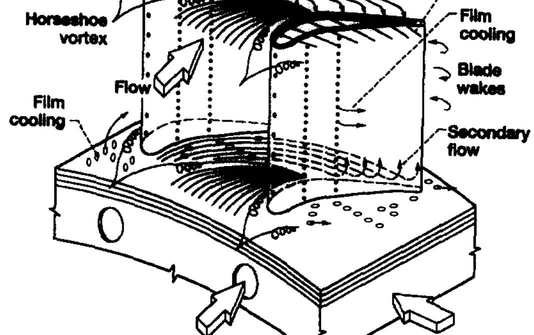

Figure 1 Complex flow phenomena in the turbine gas path

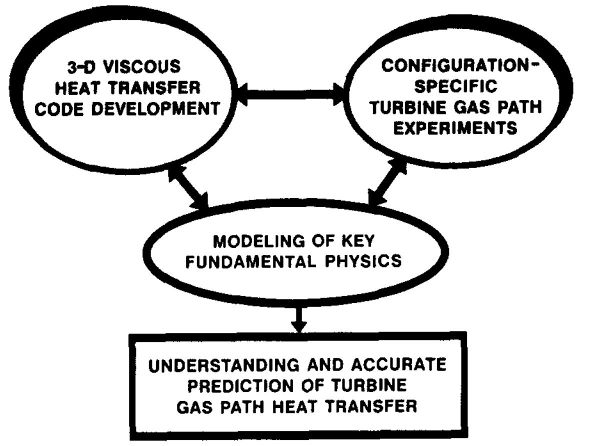

Figure 2 Research model for developing a turbine gas path heat transfer predictive capability

review with the configuration-specific experiments, whose data have provided the big picture and guided both the fundamental modeling research and the code development. Following that, we will examine key modeling efforts and comment on what will be needed to incorporate them into the codes. We will then review progress and directions in the development of computer codes to predict turbine gas path heat transfer. Finally, we will cite examples and make observations on the more recent efforts to do all this work in a simultaneous, interactive, and more synergistic manner. We will conclude with an assessment of progress, suggestions for how to use the current state of the art, and recommendations for the future.

These topics will not all be treated with equal depth. The experimental work will be reviewed more globally, zooming in on detail when it is important to illustrate key physics. The modeling, on the other hand, will be discussed in more detail than the other areas because we believe that the key to a successful predictive capability lies in the modeling. Closure models will always be needed to complete the large computer codes, and the success of these codes will depend on the accuracy, reliability, range of applicability, and computational efficiency of the models. The choice of models is often dictated by the user's need to emphasize one of these criteria over the others. Accordingly, we do not anticipate a clear choice of models emerging in the near future, and we have elected to explore a fairly wide range of models in two key turbine flow physics arenas: transition and 3-D endwall regions. We will also discuss modeling with regard to film cooling, albeit in less depth, since modeling in this area is less developed at present. Heat transfer code development is fairly new and will receive less in-depth discussion. Here the emphasis will be on what models are used and what the codes can do, rather than on the algorithmic details of the codes.

Configuration-specific experiments

Configuration-specific experiments provide a broad perspective, as well as considerable detail, on the important heat transfer physics in the turbine gas path. By configuration-specific we mean experiments in which the test article is an actual turbine blade or vane profile and passage. With rare exceptions, the experiments to be cited include heat transfer. To sort out this fairly substantial body of work, we will separate the research into two major categories: rotating and nonrotating. We will also treat one important hybrid, or subcategory —the use of upstream disturbance generators to simulate wake effects in an otherwise steady experiment.

Rotating rig experiments

The rotating rig experiments range from continuous running tests to ones with running times on the order of milliseconds. As a general rule, the long-duration tests are best for detailed aerodynamic measurements and the short-duration tests are best for heat transfer. Recent advances in instrumentation have blurred this distinction. However, as far as the existing literature is concerned, it is rare to find both detailed aerodynamic and heat transfer data in the same experiment. Choosing between realistic operating conditions and detail is another frequent compromise. In this review the focus will be on detail, since the emphasis is on establishing a reliable predictive capability; however, one must not lose sight of the ultimate goal of predicting the behavior of the engine.

The most comprehensive research on rotating rig heat transfer has come from Dunn and coworkers (Dunn and Stoddard 1979; Dunn and Hause 1982; Dunn 1984; Dunn et al. 1984a, 1984b; Dunn and Chupp 1988; Dunn et al. 1989; Dunn 1990) at Calspan, and Dring, Blair, and coworkers (Dring et al. 1980, 1982, 1986; Blair et al. 1989a, 1989b, 1992b) at United Technologies Research Center (UTRC).

The Calspan experiments are conducted in a shock-tunnel facility, using real turbine hardware with run times of about 30 msec. They are excellent for several reasons. First, heat flux can be measured directly and with very high time resolution by using the transient thin-film heat flux gage. Second, short of a real engine, they offer the closest simulation to the real world. The main limitation of these experiments is that the very short run times provide only skimpy aerodynamic data. However, the time-resolved heat flux data itself contains important aerodynamic information. Figure 3 (Dunn et al. 1989) shows a phase-locked ensemble average of the time-resolved midspan heat flux data on the suction surface. This complex behavior, when averaged out, must be modeled properly in the predictive schemes. As an example of the complex physics that must be modeled, note the heat flux peaks and valleys that occur with each blade passing. The peaks are at a level comparable to fully turbulent heat transfer, while the valleys correspond to a laminar level.

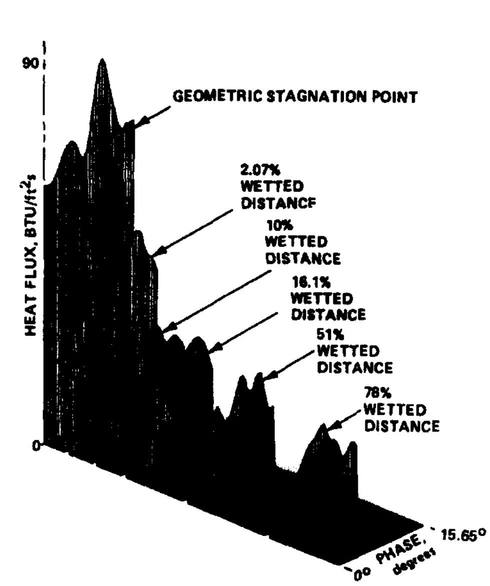

Figure 3 Phase-resolved contour plot of heat-flux distribution on actual turbine hardware suction surface, operating in a shock tunnel (Dunn et al. 1989). Used with permission from the American Society of Mechanical Engineers

By contrast, the UTRC experiments are run in a large-scale (1.5-m diameter), steady-running machine. Since the same 1-1/2 stage turbine has been used for almost all of the machine's 15-year history, a very comprehensive database has been established. The operating medium of room-temperature air at low speed offers excellent opportunity for detailed measurements. The aerodynamic measurements by Sharma et al. (1983) and the heat transfer measurements by Blair (1992b) are excellent examples of this detail. The midspan heat transfer data taken in this turbine by Blair et al. (1989a, 1989b) are shown in Figure 4. These data summarize just about everything we know about heat transfer down the midspan plane of a turbine. The data were acquired at two inlet turbulence intensity levels, 0.5 and 9.8 percent. As might be expected, the first vane row shows a dramatic effect of inlet turbulence. (The reference for comparison throughout Figure 4 is to a boundary-layer code.) The low inlet turbulence data exhibit almost completely laminar behavior, while the high turbulence data appear to be experiencing transitional behavior. There is some controversy among researchers as to whether grid-generated turbulence properly models a combustor outlet; however, the trend shown in Figure 4a is believed to be correct. The data on the pressure surface were higher than turbulent prediction level. A Reynolds number effect was reported. This is an area where further analysis/modeling is needed. The rotor data (Figure 4b) still show some free-stream turbulence effects and some transitional behavior, but a much reduced spread from that on the first vane. This has also been reported many times by Dunn. By the second vane row (Figure 4c), all inlet turbulence effects appear washed out; however, the transitional behavior still appears. The very high heat transfer towards the suction side trailing edge was a surprise, and remains unexplained. Research is under way to investigate this.

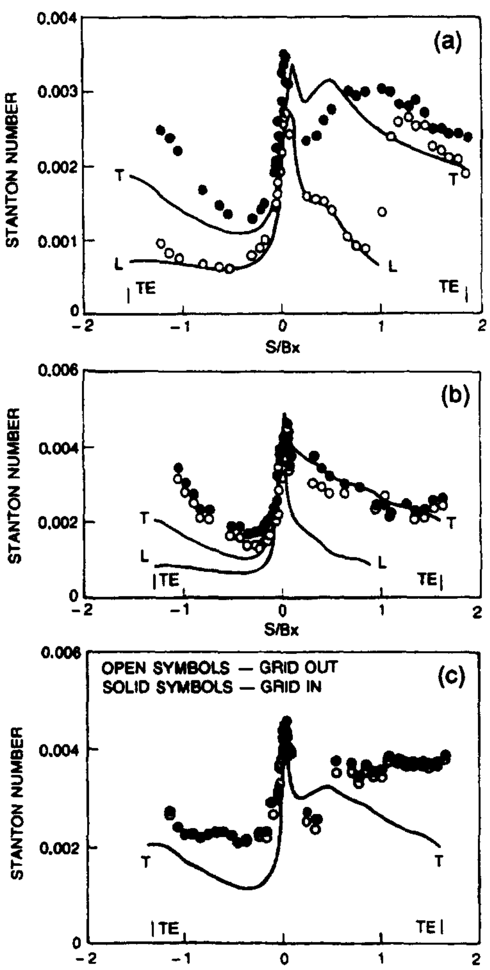

Figure 4 Midspan heat transfer distributions in a large, low-speed turbine, subject to high and low free-stream turbulence (Blair et al. 1989a, 1989b). (a) First stator; (b) rotor; (c) second stator. Used with permission from the American Society of Mechanical Engineers

While the UTRC experiments offer more detail, they are less realistic than the Calspan experiments. In our opinion there is no perfect experiment—each has its strengths and weaknesses. The two taken together, however, offer a commanding look into turbine heat transfer.

Other strong entries into the rotating experiment area are the short-duration (300 to 1000 msec) experiments. These fall into two classes: the isentropic light piston tunnel (ILPT), pioneered by the Oxford University team of Schultz and coworkers (Jones et al. 1978; Schultz et al. 1980), and the blowdown tunnels, pioneered by the Massachusetts Institute of Technology (MIT) team of Epstein and coworkers (Guenette et al. 1989). These facilities have the same transient heat flux measurement advantage as the shock tunnel, and they have longer run times. Coupled with new particle image velocimetry (PIV) techniques, they potentially offer the best of both worlds. If one were starting with a clean sheet of paper, these would probably be the rotating facilities of choice. This is exactly the case with the U.S. Air Force (Haldeman et al. 1991), following the MIT model. We can expect to see important contributions from these facilities in the future. The midspan heat transfer reported by Guenette et al. (1989) from the MIT transonic turbine tends to agree with that of Figures 3 and 4. To date, the ILPT facilities have been used only for nonrotating and simulated wake experiments to be discussed later, but plans for development of a full turbine stage at the Defense Research Agency (DRA) are under way (Harasgama and Wedlake 1991).

Steady-running, elevated-temperature facilities, which use actual turbine hardware, have long been used by NASA and the aircraft engine companies for performance and detailed aerodynamic data. These facilities have not been used much for heat transfer because steady-state heat transfer is so difficult to measure, especially on a rotor. This may change, however, as new heat flux gages, such as reported by Guenette et al. (1989), Mancuso and Diller (1991), and Ching and O'Brien (1991), are perfected. Roelke et al. (1991) have a program underway to measure heat transfer in a cooled radial turbine.

While in general this review is limited to work that includes heat transfer measurements, some rotating experiments with only aerodynamic data are important. Hodson (1984) made boundary-layer velocity profile measurements with hot wires in the rotating turbine blade passage. To our knowledge, these are the only such measurements and should prove very valuable for code validation. Addison and Hodson (1990a, 1990b) also measured unsteady surface fluctuations, using hot films on the surface, to gain insight into transition in a turbine. Binder et al. (1985, 1987) have made detailed laser measurements in a cold air turbine, which also should prove very valuable for code validation. Getting the right flow physics is, after all, the first requirement of a good heat transfer predictive capability. Of course, many of the heat transfer experiments, such as Dunn (1990), included unsteady pressure measurements.

Wake simulation experiments

A hybrid between the rotating and nonrotating experiments is the wake simulation experiments. In wake simulation facilities, either a rotating spoked wheel (Ashworth et al. 1985; O'Brien et al. 1986; O'Brien and Capp 1989; O'Brien 1990; Dullenkopf et al. 1991) or a rotating squirrel cage of bars (Pfeil et al. 1982; Bayley and Priddy 1981; Priddy and Bayley 1988; Schoberei 1991) is used to produce wakes that impinge on a downstream test surface. This is where the Oxford ILPT has received extensive use. An added contribution of the turbine experiment of Guenette et al. (1989) was to verify the validity of the rotating-bar simulation. They used exactly the same test blades as the Oxford group.

At Oxford a linear cascade of transonic turbine blades was heavily instrumented with thin-film transient heat flux gages. The cascade was placed in the ILPT, and a spoked-wheel wake generator was rotated in front of it. Schlieren pictures were also taken. The progress of the wakes and effects on heat transfer were extensively documented. Doorly and Oldfield (1985b) created a summary figure of the process (Figure 5), which shows the progress of the wake through the cascade. As with the Calspan experiments, the heat flux levels jump back and forth between laminar and turbulent levels. Both the shock and the wake produce a distinct effect. Doorly (1988) examined the data carefully and concluded that the primary effect was a premature transition from laminar to turbulent flow. He then proposed a simple model that showed good first-order agreement with data and could prove useful in making design decisions.

O'Brien and coworkers (O'Brien et al. 1986; O'Brien and Capp 1989; O'Brien 1990) concentrated on the stagnation region. Here again, an examination of the data in terms of a simple model may prove useful. O'Brien (1990) inserted the local unsteadiness into a standard correlation for stagnation point heat transfer in the form of an instantaneous "turbulence intensity" and found good correlation with the unsteady heat transfer data (Figure 6). Also, O'Brien et al. (1986) used both

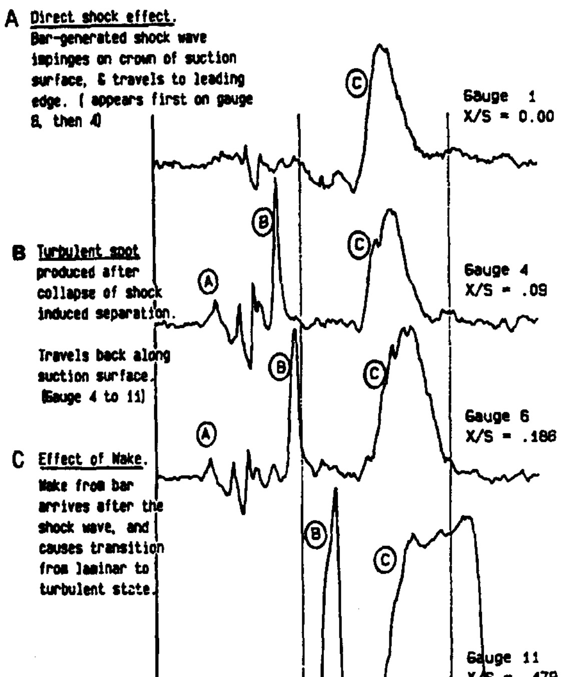

Figure 5 Summary of the progression of effects of passing wakes and shocks on heat transfer from a turbine airfoil suction surface (Doorly and Oldfield 1985b). Used with permission from the American Society of Mechanical Engineers

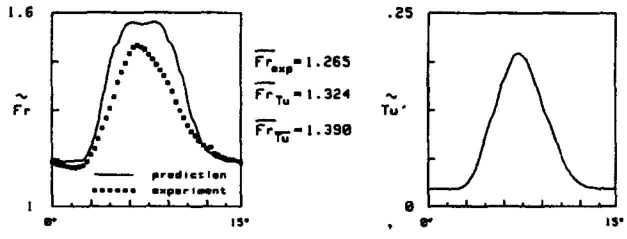

Figure 6 Results of using local unsteadiness as a variable to predict stagnation point heat transfer behind a passing wake (O'Brien 1990)

the transient heat flux technique and a traditional steady-state technique in the same experiment with excellent agreement, helping to validate use of the transient technique. These bar-generated wake simulation experiments are relatively inexpensive, are easy to run, and should produce valuable information.

In general, the rotating experiments and the wake simulation experiments have concentrated on airfoil midspan heat transfer. The Calspan experiments usually had some instrumentation near the hub and tip of the blades and vanes and also in the vane endwall region, with a few points on the blade platform. These data are in general too sparse to establish 3-D contours, but will be very valuable in calibrating 3-D codes. Recently, Blair (1992b) reported the results of a new experiment in the UTRC large, low-speed rotating rig, using both a dense array of thermocouples and liquid crystal thermography to map the full 3-D rotor blade and endwall heat transfer. The report includes a computational analysis and a comparison to the Graziani et al. (1980) data.

Cascade experiments

The older and much more comprehensive body of literature on turbine configuration-specific heat transfer is the cascade experiments. These go back more than 30 years and have provided much of what can be described as the conventional wisdom in turbine gas path heat transfer. Instead of trying to survey it all, we have chosen to include a representative (historical and international) sampling in the References section and in Table 1. We will comment on just a few specifics that are relevant to the theme of this paper. A few of the very early works are still worthy of note and should not be overlooked. Both Wilson and Pope (1954) and Turner (1971) provide excellent detail on the midspan flow physics and heat transfer. Dyban and Kurosh (1968) ran their experiment in both a cascade and an air turbine. While the details of their experiment are sketchy, they were among the first to cite the differences between cascade and rotating rig results, showing heat transfer increases of 50 to 100 percent in the rotating rig. Sieverding (1985) has reviewed the secondary flow in turbine passages, and this paper should be consulted.

A particularly important set of cascade data is the work of Langston et al. (1977) and Graziani et al. (1980). These two experiments have provided a comprehensive turbine cascade aerodynamic and heat transfer database. Furthermore, the blade profile is the same as the midspan cross section of the UTRC large, low-speed rig, which allows a comparison between cascade and rotating rig data (Blair 1992b). Figure 7 indicates the heat transfer detail available and the important trends.

Another valuable database is that of Hippensteele and Russell (1988) and Boyle and Russell (1990). These experiments covered a wide Reynolds-number range and identified an important trend in endwall data. At low Reynolds number, the

Table 1 Summary of turbine cascade experiments

Authors	Date	Cascade type	2-D/3-D endwall data	Film cooling	Gas temperature (K)	Turbulence level (%)	Reynolds number* $\times 10^{-5}$	Aero detail
Arts and Bourguignon	1990	Linear	2-D	Yes	370	1.0–6.0	15–22.5	Medium
Arts and De Rouvroit	1990	Linear	2-D	No	420	1.0–6.0	5–20	Medium
Bario et al.	1990	Linear	EW	Yes	Room	Unknown	3.3	High
Boyle	1990	Linear	EW	No	Room	2.0	0.7–5	High
Camci and Arts	1990	Linear	2-D	Yes	370	0.8–5.2	7.5–9.7	Medium
Camci and Arts	1991	Linear	2-D	Yes	370	0.8–5.2	9.0–10.7	Medium
Consigny and Richards	1982	Linear	2-D	No	420	0.8–5.2	2.3–9.4	Low
Dyban and Kurosh	1968	Linear	2-D	No	Room	—	7	Low
Gladden and Yeh	1992	Annular	2-D	No	1200	—	—	Low
Goldstein and Chen	1985	Linear	3-D	Yes	Room	Unknown	Unknown	Low
Graziani et al.	1980	Linear	3-D/EW	No	Room	1.0	5.5	High
Langston et al.	1977	Linear	3-D/EW	No	Room	1.0	5.5	High
Haller and Camus	1984	Linear	2-D	Yes	Room	0.5	8.5	High
Harasgama and Wedlake	1991	Annular	3-D/EW	No	450	6.5	17–52	High
Hippensteele and Russell	1988	Linear	EW	No	Room	1.4	0.8–4.0	High
Jones et al.	1978	Linear	2-D	No	440	2.0	4.2–12.6	Low
Nealy et al.	1984	Linear	2-D	No	811	6.5–8.3	5–25	Medium
Nicholson et al.	1984	Linear	2-D	No	430	0.2–4	11.1	High
Nirmalan and Hylton	1990	Linear	2-D	Yes	700	6.5	15.5–25	Medium
Schultz et al.	1980	Linear	2-D	No	430	1–4	4–12	Low
Takeishi et al.	1990	Annular	3-D/EW	Yes	Room	2.7	6.1	High
Turner	1971	Linear	2-D	No	360	0.5–5.9	10	Medium
Wedlake et al.	1989	Annular	3-D/EW	No	432	6.9	17.5–35	High
Wilson and Pope	1954	Linear	2-D	No	Room	Varied	1.8–7	Medium
York et al.	1984	Linear	EW	No	850	7.9–9.4	5–28	High

*Reynolds number as reported by authors—definitions will vary.

Figure 7 Heat transfer (Stanton number) distribution on the endwall and airfoil surfaces in a turbine cascade (Graziani et al. 1980). Used with permission from the American Society of Mechanical Engineers

lines of constant Stanton number tend to follow the streamlines, while at high Reynolds number they tend to span the passage. These results, coupled with the visual evidence provided by Gaugler and Russell (1984), make an impressive, if not very surprising, case that endwall heat transfer is truly 3-D. Boundary-layer analyses have little potential for success in this region. This will be discussed further in the Fundamental Physics and Model Development section and the Code Development section below, and the data will be shown there.

Most of the cascade work, including that just discussed, has been performed in linear cascades, where, although the flows are 3-D, the geometry is two-dimensional (2-D). It is sometimes desirable to get information with more complex blade shapes and where stronger radial gradients exist. Thus, although more difficult, annular cascade experiments are valuable. Recent studies by Wedlake et al. (1989) and Harasgama and Wedlake (1991) are excellent examples. The results, shown in Figure 8, are particularly significant, because the cascade is a highly 3-D modern design and is part of a larger program to make measurements in a full turbine stage.

Finally, it should be noted that the majority of the data have been acquired at or near room temperature. Very few "engine environment" data have been acquired and reported. The work of York et al. (1984) and Nealy et al. (1984) at Allison Gas Turbine and of Gladden and Yeh (1992) at NASA are exceptions. Comprehensive 3-D viscous heat transfer/flow codes coupled with industry test experience should bridge that gap. It is the role of the more detailed albeit less realistic, experiments to validate the codes and models.

Film-cooling experiments

Film cooling is a very important subject in the turbine heat transfer arena. The introduction of film cooling in the 1950s was responsible for a major improvement in overall turbine performance. Unfortunately, for purposes of this review, the

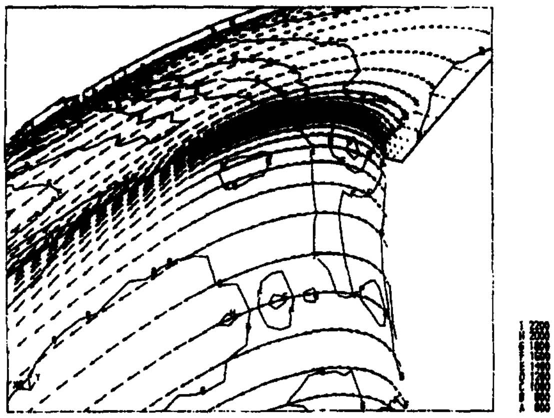

Figure 8 Endwall and vane suction surface heat transfer contours and velocity vectors in a full annular cascade of modern airfoil design (Harasgama and Wedlake 1991). Used with permission from the American Society of Mechanical Engineers

subject is so vast and so application specific that we can only scratch the surface. Nowhere in turbine heat transfer is there a more empirical data-based predictive capability than in film cooling. Furthermore, most of the specific film cooling details are held quite closely by the aircraft engine manufacturers. However, because it is so important to turbine performance, some methodology must be established to include film cooling in any turbine heat transfer predictive capability. Accordingly, we will attempt to review at least some of the key experimental/modeling/code efforts associated with film cooling.

Very little film cooling work has been done in the rotating frame of reference. One notable exception is the work of Dring et al. (1980), where the data were compared with stationary results. They were found to agree quite well on the suction surface, but the effectiveness tended to decay faster on the pressure surface.

Most of the film-cooling experiments have been conducted either on flat plates or in turbine cascades. We will concentrate on the cascade experiments in this section and defer the more basic flat-plate work to the next section, and then only as it supports the modeling. The film-cooling cascade experiments selected are all included in Table 1. They are only recent representative efforts, rather than a comprehensive listing. The reader is directed to them for further references. As with the previous section, most of the experiments are at low temperature. One notable exception is the work of Nirmalan and Hylton (1990), which was conducted at high temperature in a more real engine environment.

There is a fundamental question associated with film cooling: Is film cooling more an aerodynamic or a heat transfer problem? The answer, of course, is that it is both. However, the designer is frequently more interested in the distribution of the coolant and the aerodynamic penalties associated with it than with the actual heat transfer in the coolant region. For that reason, some of the references included in Table 1 are more focused on the aerodynamics than on the heat transfer. For example, Haller and Camus (1984), Bario et al. (1990), and Camci and Arts (1991) all devote considerable effort to the aerodynamics. Bario et al. (1990) have made detailed boundary-layer measurements of the jet trajectories for endwall film coolant caught in the secondary flows.

Film-cooling heat transfer results are presented in a variety of ways. The two most popular are (1) as a ratio of cooled to noncooled heat transfer and (2) as an effectiveness defined as a temperature ratio $(T_o - T_w)/(T_o - T_c)$, where T_o is the hot gas stream temperature, T_w is the wall temperature, and T_c is the coolant temperature. Effectiveness is much easier to measure and is frequently all the designer needs to know. Heat transfer is harder to measure, but is ultimately what is needed to predict thermal stress and durability. A very good example of effectiveness distribution is the recent work of Takeishi et al. (1990), shown in Figures 9a and 9b for a turbine vane suction and endwall surfaces, respectively. The results shown in these figures are quite consistent with what one would expect to find. Along the midspan region of the vane, the two rows of film-cooling holes are quite effective for a substantial distance along the vane chord. On the other hand, the effectiveness drops off dramatically on the vane surface near the endwall (Figure 9a) and it is quite low in the endwall region itself (Figure 9b). This, of course, is where the secondary flows and horseshoe vortexes are dominant, and they are sweeping the coolant away. This points out the difficulty with the flat-plate work and also with much of the cascade work, which has concentrated on the midspan region. This also relates to the earlier question of the relative importance of the coolant aerodynamics and the resulting heat transfer. The coolant needs to get to the right places, before the heat transfer question becomes the primary focus. Of course, the designer will work diligently to get the coolant to the right places and then the heat transfer again becomes important. Typical heat transfer results, such as those taken from Arts and Bourguignon (1990) for the pressure side of a nozzle vane, are shown in Figure 10. For the low blowing rate, the results show a substantial reduction in heat transfer immediately downstream of the two rows of holes and then a gradual increase to some more or less asymptotic level. At the higher blowing rates, there is first an increase, then the decrease. The authors report that the maximum effectiveness is in a region from about 15 to 50 hole diameters downstream of the injection, depending on the blowing rate, as shown in Figure 10.

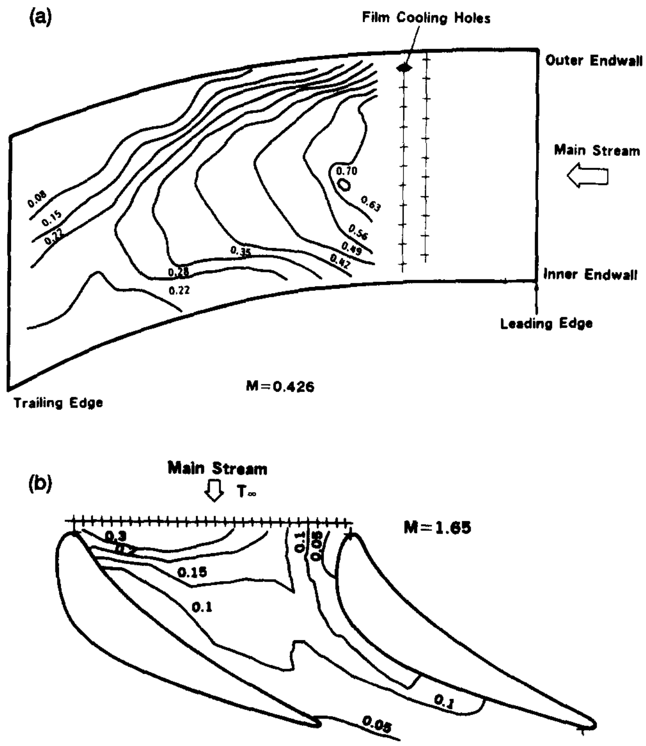

Figure 9 Film cooling effectiveness distribution in a turbine cascade (Takeishi et al. 1990). (a) Vane suction surface; (b) endwall region. Used with permission from the American Society of Mechanical Engineers

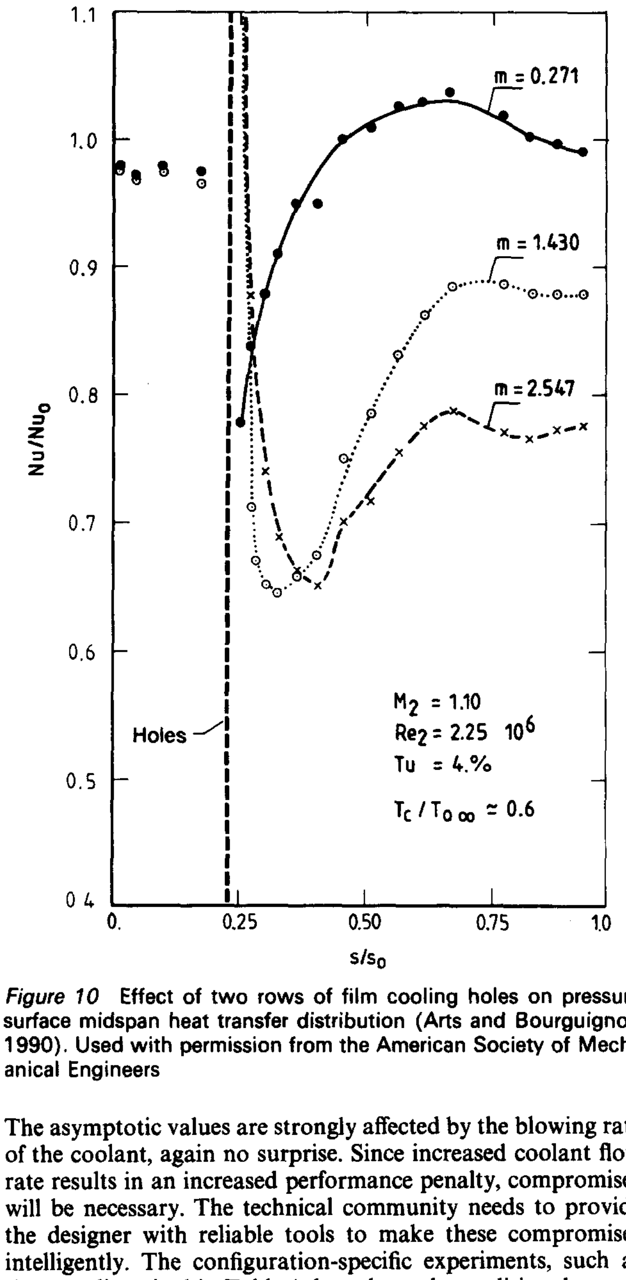

Figure 10 Effect of two rows of film cooling holes on pressure surface midspan heat transfer distribution (Arts and Bourguignon 1990). Used with permission from the American Society of Mechanical Engineers

The asymptotic values are strongly affected by the blowing rate of the coolant, again no surprise. Since increased coolant flow rate results in an increased performance penalty, compromises will be necessary. The technical community needs to provide the designer with reliable tools to make these compromises intelligently. The configuration-specific experiments, such as the sampling cited in Table 1, have been the traditional source of such guidance. Improved modeling and, ultimately, advanced computer codes will be needed. These will be discussed in the next sections.

Fundamental physics and model development

The complexity of flow on a turbine blade, as influenced by geometry, local conditions of acceleration or deceleration, endwall conditions, body forces, and the unsteady nature of the flow, requires an isolation of the various physical phenomena for experimental study and modeling purposes. It is common practice to divide a turbine blade into component parts to study and model the basic physics. For example, NASA Lewis Research Center is investigating the horseshoe vortex at the intersection of the plate and endwall, the effect of free-stream turbulence on the stagnation region heat transfer, the effect of rotor wakes, and transition from a laminar to a turbulent boundary layer. Since it would not be possible to cover all the fundamental physics and model-development research, this section will detail the progress in the areas of transition, endwall heat transfer, and film cooling. We think that these are among the most critical areas in the prediction of convection heat transfer in the turbine gas path.

Bypass transition region

The bypass transition terminology used here is that of Morkovin (1978), who suggested that for laminar-to-turbulent boundary-layer transition, when large disturbances exist (as is the case in a turbine), linear stability mechanisms are bypassed and finite nonlinear instabilities occur. The result is the sudden appearance of turbulent spots without the disturbance growth predicted by linear stability theory. The transition region is very important to turbine gas path heat transfer for two reasons. First, the various competing forces in the turbine gas path, such as free-stream turbulence, wakes, curvature, and acceleration, frequently result in the flow field lying between a laminar and turbulent flow. Significant portions of the turbine airfoil surface can be in transition flow. Second, the heat transfer levels associated with turbulent flow can be easily three times those of laminar flow at the same Reynolds number. Accordingly, the predictive uncertainties and the appropriate models associated with transitional flows are very important in the study of turbine gas path heat transfer. Excellent review articles of laminar-to-turbulent transition applicable to turbine heat transfer may be found in papers by Narasimha (1985), Mayle (1991), and Volino and Simon (1991).

The use of the time-averaged Navier–Stokes equations in the numerical codes requires the use of turbulence models. These models are based on the physics of the flow and generally perform well for fully turbulent conditions with varying degrees of success. As Volino and Simon (1991) point out, turbulence models for the prediction of transitional flows have been less than successful. Volino and Simon, reviewing the modeling efforts with one- and two-equation models, indicated a need to include more physics in the models. Because of the anisotropic nature of transition, they suggest a potential advantage in the use of a Reynolds stress model and list the experimental quantities required for developing turbulence models that will capture the physics of bypass transition. The following is a consideration of some models with the potential to simulate the mechanisms in bypass transition and how experimental and numerical experiments are aiding their development. The turbulence models to be considered are k-ε models, intermittency models, multiscale models, and Reynolds stress models.

k-ε models. In general, k-ε turbulence models used in numerical codes appear to simulate the transition governed by the transport and production of turbulence in the boundary layer. These models require a proper location of the initial profiles of k and ε to correctly predict the location of transition onset. Results with these models generally produce an underprediction of transition length (Rodi and Scheuerer 1985b). Using empirical correlations, Schmidt and Patankar (1988) modified the turbulent production term of the differential turbulent kinetic equation to make predictions consistent with experiment, as shown in Figure 11. This work, and the work of Stephens and Crawford (1990), demonstrate the sensitivity of the prediction of the onset of transition to the location at which calculations are started. Simon and Stephens (1991) suggest that, for free-stream turbulence levels less than 5 percent, calculations begin at the critical Reynolds number for linear stability growth. Furthermore, models designed for and calibrated with fully developed turbulent flows may not be adequate. Such has been found to be the case for the K.Y. Chien turbulence model, which produces a premature start of transition (Stephens and Crawford 1990) due to a damping function

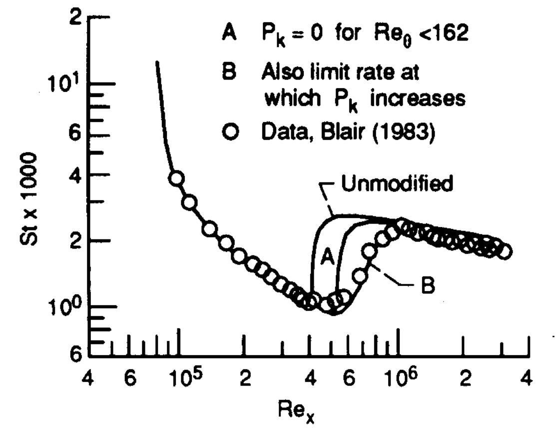

Figure 11 Effect of production term modifications to the k-ε model in the transition region (Schmidt and Patankar 1988)

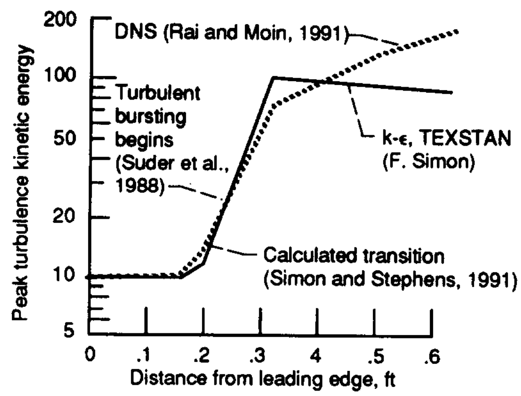

Figure 12 Comparison of computed disturbance energy from computations based on a k-ε model and on direct numerical simulation (DNS) for bypass transition (Simon and Stephens 1991; Rai and Moin 1991)

dependence on the boundary-layer normal distance. In the work of Stephens and Crawford, it was determined that the two-equation turbulence models that perform best in predicting transition have a damping function that is dependent on the turbulent Reynolds number. Improved models have a damping function dependence on both the turbulent Reynolds number and the boundary-layer normal distance (Chen 1990). In general, k-ε models do a respectable job of simulating bypass transition. Improvements to these models will require greater knowledge of the physics of transition. Experiments and direct numerical simulations (DNS) are being performed to achieve this end.

The experiments of Suder et al. (1988) produced some valuable insights into the mechanisms of bypass transition. Single hot-wire measurements of a boundary layer on a flat plate undergoing transition indicate that turbulence spots begin at a maximum boundary-layer turbulence level of 3.5 percent, regardless of the path (high or low free-stream turbulence) to transition. The intermittency at this point, according to Suder's data, is on the order of 0.03. Using Suder's criterion of 3.5 percent on the experimental results of Kuan and Wang (1989) results in a transition momentum Reynolds number of 249 compared to an experimental value of 297 for zero intermittency. Sharma et al. (1982) simulated the flow on the convex side of an airfoil with a pressure gradient distribution on a flat plate. He determined that the onset of transition, as defined by an intermittency of 0.1, occurs when the streamwise boundary-layer turbulence level reached a threshold value of three times the friction velocity. The data of Suder confirm this conclusion for an intermittency of 0.1.

In Figure 12, the boundary-layer disturbance growth, as measured by a DNS calculation by Rai and Moin (1991), and the k-ε calculated maximum turbulence kinetic energy in the boundary layer are presented for the case of zero pressure gradient and 2.6-percent free-stream turbulence. Suder's criterion when applied to Figure 12 produces a transition value consistent with the numerical results of Simon and Stephens (1991). They use the criterion of minimum coefficient of friction for the onset of transition and assume this to occur at zero intermittency. Figure 12 shows how the k-ε model captures the nonlinear growth that produces the first sign of turbulent spot formation. This example points out the power of numerical experiments to increase understanding of transition physics and to aid in the testing and development of transition models. The database produced from numerical experiments such as that of Rai and Moin (1991) will be invaluable.

Intermittency models. The experimental work of Sohn et al. (1991), Kuan and Wang (1989), Arnal et al. (1978), Kim and Simon (1991), and Kim et al. (1989) all demonstrate, via a conditional sampling approach, that the transition region cannot be described by a combination of Blasius and fully turbulent boundary-layer profiles. This puts into question the approach of transition modeling based on intermittency weighting of the "nonturbulent" and turbulent spot zones first proposed by Dhawan and Narasimha (1958). The results of Sohn et al. indicate that the nonturbulent zone starts with a Blasius velocity profile at low intermittency and deviates from this profile with increasing intermittency. Conversely, the turbulent spot zone becomes more like a fully developed turbulent velocity profile as the intermittency increases. These effects are shown in Figure 13. These results suggest that conditioned average techniques rather than global time averages in turbulence models would better describe the physics of transition. The use of conditioned averaging techniques for the turbulent spots and the laminar-like fluid surrounding the spots in a k-ε

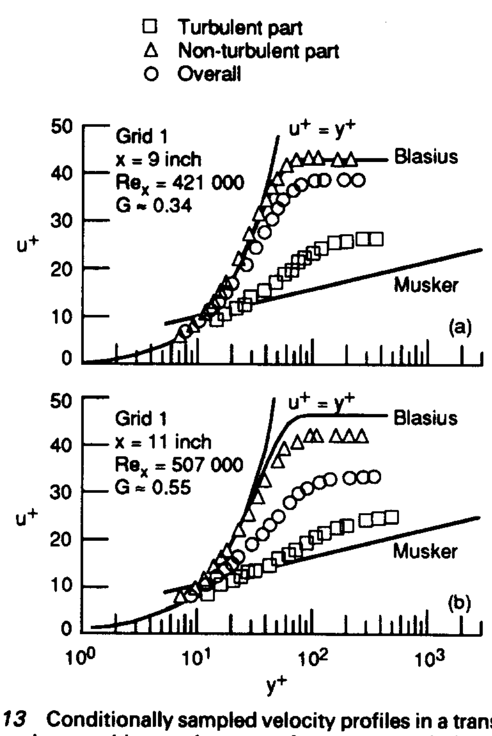

Figure 13 Conditionally sampled velocity profiles in a transitioning boundary layer, subject to 1 percent free-stream turbulence (Sohn et al. 1991). (a) Intermittency, 0.34; (b) intermittency, 0.55

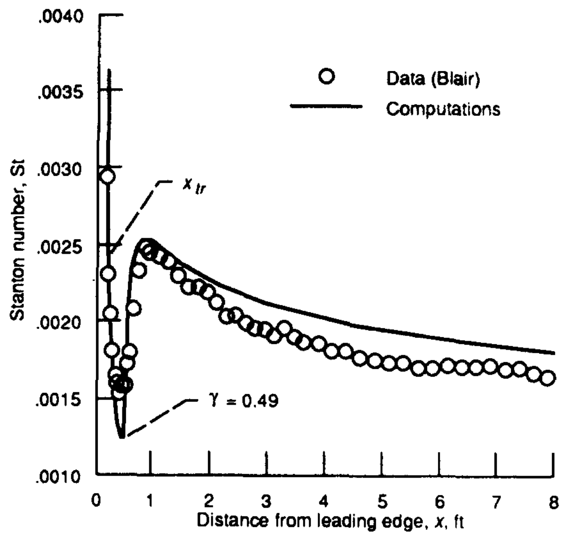

Figure 14 Comparison of intermittency-based k-ε model prediction to the transitioning heat transfer data of Blair (1983) at a free-stream turbulence of 2.8 percent (Simon and Stephens 1991)

model by Vancoille and Dick (1988) resulted in good agreement between experiment and prediction of the turbulence intensity profiles. Simon and Stephens (1991) derived conditioned energy equations and used a modified Vancoille and Dick approach to show its potential in predicting bypass transition heat transfer, as shown in Figure 14.

Multiscale models. The recent work of Volino and Simon (1991) applied quadrant and octant analysis to experimental data to analyze the differences in structure between turbulent and transitional flows. They found differences in transition and fully turbulent flows, which they believe are due to incomplete development of the turbulence in the transition region. They state that transitional flows are dominated by large-scale eddies because of the incomplete development of the cascade of energy from large to small scales. They indicate that this suggests the need for incorporating additional scales into transition models such as the standard k-ε model. Such a modified k-ε model would, for example, have a k equation for the large scales and one for the small scales. This is consistent with the experimental results of Blair (1983), whose power-spectral-density measurements indicated that the ratio of turbulent dissipation to production increased through transition. He used this information to explain the high turbulent kinetic energy (TKE) values near transition onset, which may be caused by an incomplete development of energy cascade. The development and implementation of a multiscale model for transition would appear to have much merit. Such a model might also capture the effect of length scale on transition that Suder et al. (1988) and others have found experimentally.

Reynolds stress models. The laboratory experiments of Sohn et al. (1991) and Sharma et al. (1982) and the numerical experiments of Rai and Moin (1991) indicate that a large level of anisotropy exists in a transitioning boundary layer, as seen in Figure 15. This suggests the use of a Reynolds stress type of turbulence model for a more exact accounting of all the stresses. The k-ε model assumes isotropic turbulence and is therefore limited in providing information on boundary-layer stresses. In the field of transition modeling, Reynolds stress modeling is a new area of work, and there is little to report to date. Savill (1992) is the only researcher we know who has successfully used a low-Reynolds-number Reynolds stress transport model for predicting transition. The onset of transition is well predicted in the range of free-stream turbulence from 1 to 6 percent. However, there remains a need to improve the modeling for transition length. While such an advanced model accounts for all the stresses, it should be noted that the stress in the streamwise direction has the greatest growth during transition. This may partially explain the success of the k-ε models.

At this juncture of transition models, it appears that the k-ε and q-ω models are the most practical for effective bypass transition calculations. However, use of these models requires proper attention to the transition region. Conditioned averaged techniques and a greater knowledge of the growth of nonlinear disturbances in the boundary layer will be required. Such information will permit the proper implementation of the initial profiles for k and ε.

Three-dimensional endwall region

The 3-D endwall region, formed by the intersection of the blade/vane with the hub, is an important heat transfer region. At this intersection a "horseshoe" vortex is formed. The result is a secondary flow that greatly influences the heat transfer at the endwall and at the vane suction surface.

The flow visualization work of Gaugler and Russell (1980) using neutrally buoyant, helium-filled bubbles documents well the nature and path of the vortex flow. These authors superposed their visual results on the data of York et al. (1984) in the same geometry cascade, as shown in Figure 16. The high Stanton number gradients near the entrance to the cascade appear to be along the vortex path. The experimental documentation of heat transfer by workers such as Blair (1974), Graziani et al. (1980) (cf. Figure 7), and Goldstein and Spores (1988) also shows that the result of endwall secondary flows is unwanted heat transfer augmentation at the endwall and airfoil suction surface. The effect of such secondary flows on heat transfer can be expected to be even more pronounced on modern gas turbines because the smaller blade aspect ratio permits a greater portion of the blade to be influenced by secondary flows. Such a complex heat transfer phenomenon is best studied by methods that allow for detail surface mapping of the heat transfer, as has been done by using the liquid crystal technique. Hippensteele and Russell (1988) used this technique to find a Reynolds number effect on the passage endwall heat transfer pattern and an increase in the Stanton number with an increase in the Reynolds number. This work was extended by Boyle and Russell (1990) to include the effect of the inlet boundary layer. It was determined in this effort that the pattern

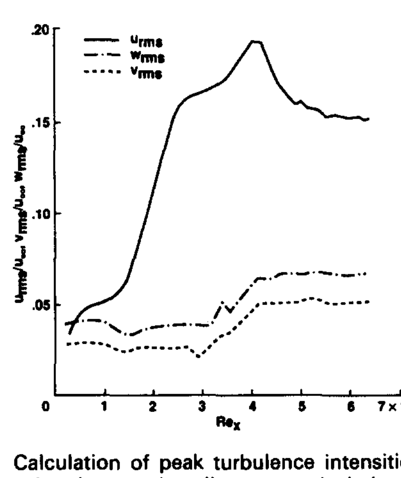

Figure 15 Calculation of peak turbulence intensities in a transitioning boundary layer, using direct numerical simulation (DNS) methods (Rai and Moin 1991).

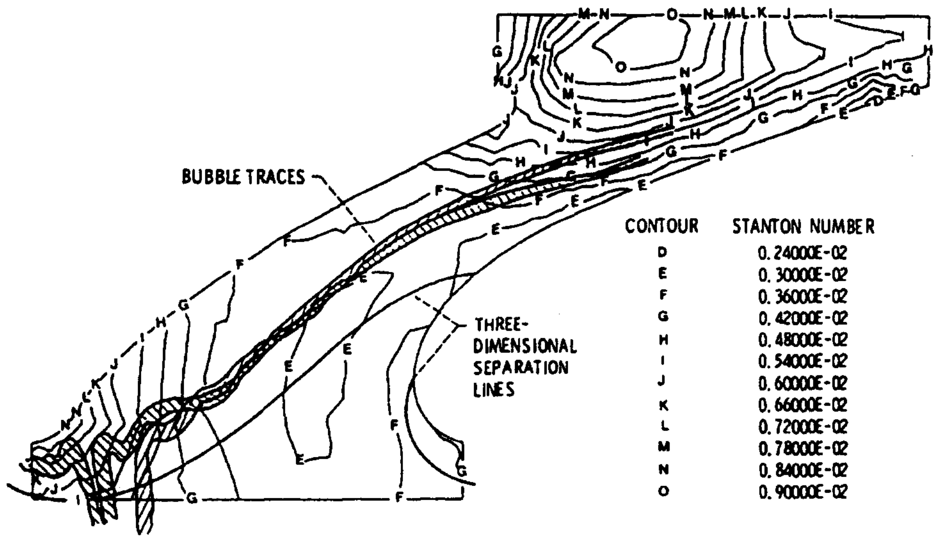

Figure 16 Superposition of flow visualization results on endwall heat transfer data of York et al. (1984) (Gaugler and Russell 1984).

of the Stanton number contour lines was not affected by changes in the thickness of the inlet boundary layer. However, the thickness of the boundary layer did affect the level of the Stanton number. It is clear that the complex nature of the endwall heat transfer requires a greater understanding of the flow physics for the development of turbulence models that can be used in 3-D numerical codes.

Unlike the situation in the preceding section (transition region), there are few "simple" heat transfer experiments to capture the basic endwall mechanisms, which need to be modeled. There are some fluid mechanics experiments, but little heat transfer. One notable experiment is the work of Abrahamson and Eaton (1991). In this study, a 2-D boundary layer was made 3-D by introducing a wedge obstruction and making measurements of the heat transfer. The heat transfer results of Abrahamson and Eaton are consistent with the near-wall turbulence measurements of Anderson and Eaton (1989), who noted that the turbulent shear stresses were suppressed. Abrahamson and Eaton hypothesize on the basis of their experimental results that the near wall region for their 3-D boundary layer was dominated by 2-D transport mechanisms. This insight has some value in conjunction with the more extensive research conducted in cascades to investigate basic endwall heat transfer mechanisms. Thus, the discussion of these cascade experiments that began in the previous section continues, now with the focus on modeling.

Also, in this section the modeling emphasis is different. Here the focus is on capturing the key physics in a 3-D flow field. Thus, rather than proceed from the simplest to the most complex model as done previously, we will start the discussion with the modeling that has the potential to capture 3-D anisotropic turbulence effects.

Reynolds stress models. Bradshaw (1972) recommends the Reynolds stress turbulence model for complex 3-D flows rather than models based on the eddy-viscosity principle (e.g., k-ε or Baldwin–Lomax). He states that "the simple behavior of eddy viscosity and mixing length in simple thin shear layers is not maintained in more complicated cases like three dimensional flow, multiple shear layers or flows with significant extra rates of strain." This view that Reynolds stress transport models have the greatest potential for complex flows is also held by many researchers involved in the development of turbulence models, including Launder et al. (1975), Donaldson (1971), Hanjalić and Launder (1972), and Lakshminarayana (1986). However, applications of second-order turbulence closure models have been mainly limited to momentum transport processes with little work in applications related to heat transfer. Their applicability for heat transfer requires further investigation.

Launder and Samaraweera (1979) included turbulent heat flux transport equations as part of a second-order turbulence closure approach to predicting heat transfer in free shears and boundary layers. However, in the near-wall region where molecular transport becomes more important, they used a wall function approach that required experimental information. It would be better and more consistent to have a low-Reynolds-number version of the Reynolds stress turbulence model, as in the case of two-equation turbulence models, to predict the near-wall momentum and thermal behavior. Such a low Reynolds number model was developed by Hanjalić and Launder (1976), although this early model did not use the full set of transport equations like the model of Shih and Mansour (1990). This newer model, when compared with the DNS results of Kim et al. (1987), performs very well in predicting the behavior of the near wall turbulence and the flow field. As indicated previously, little work has been done in using the second-order approach for heat transfer predictions, and no work appears to have been done in the endwall region using this model. Perhaps this reflects the additional level of difficulty in obtaining accurate numerical predictions. Despite the concerns about anisotropy, to date those who have applied turbulence models for the calculation of endwall heat transfer rates have used models based on the eddy-viscosity concept.

k-ε models. The k-ε or two-equation turbulence model assumes the turbulence to be isotropic. Despite the k-ε model's inconsistency with the anisotropic nature of the endwall turbulence, it has been found useful for 3-D numerical analyses. Hah (1989) used a 3-D Navier–Stokes code with a low-Reynolds-number version of a two-equation model to predict the heat transfer near the endwall of a turbine blade row. This permitted him to include effects of turbulence and transition from a laminar to turbulent boundary layer using a turbulence model of Chien (1982). As noted previously in the Bypass Transition section, this model produces transitions too soon, a result that Hah reported. Hah reports good comparisons of his numerical predictions with the experimental data (Figure 17a) of Graziani et al. (1980) for the endwall of a turbine blade cascade. This

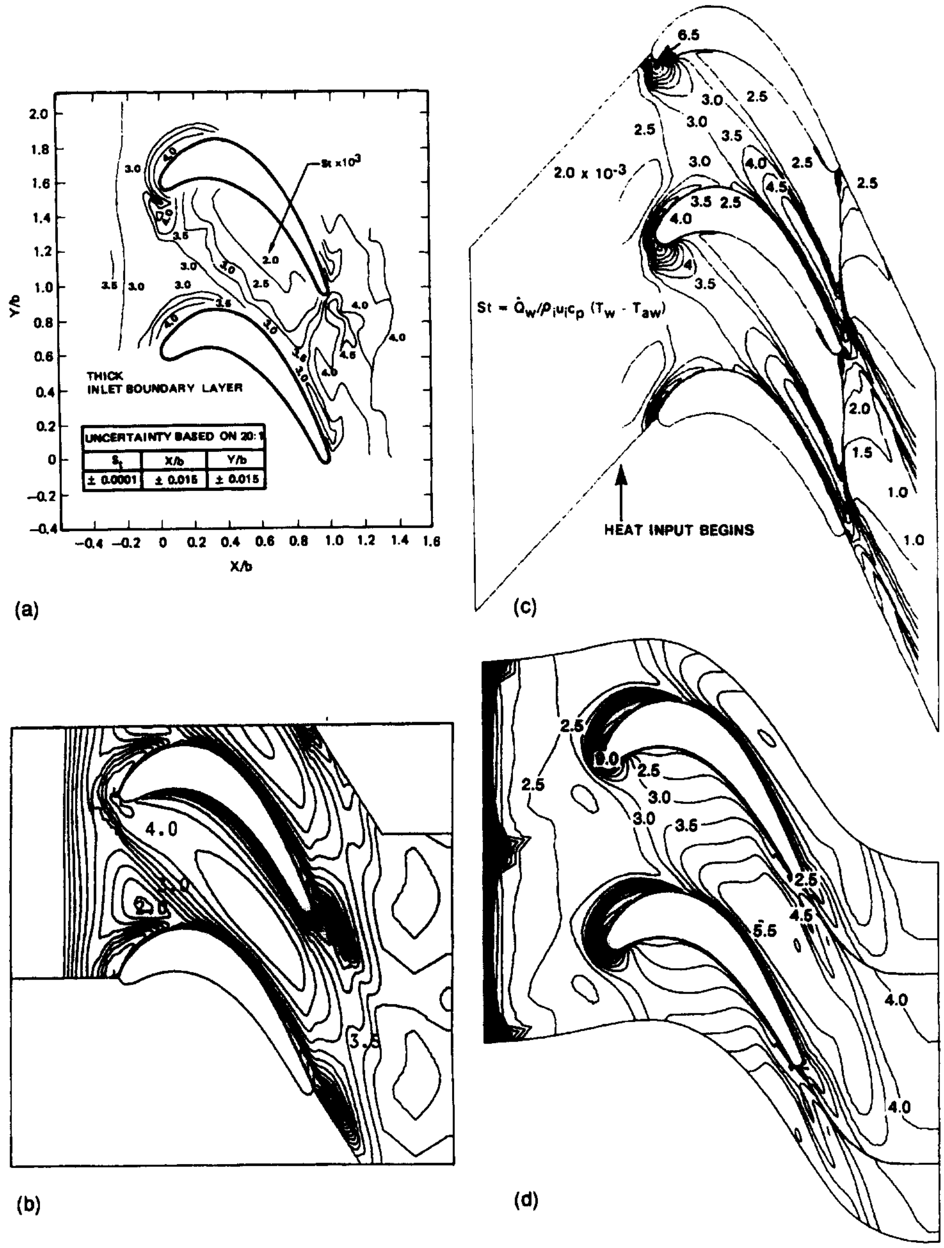

Figure 17 Comparison of the use of various turbulence models in full 3-D Navier–Stokes calculations of endwall heat transfer data of a turbine blade cascade. (a) Experimental data (Graziani et al. 1980). Used with permission from the American Society of Mechanical Engineers; (b) k-ε model (Hah 1989). Copyright © 1989 American Institute of Aeronautics and Astronautics. Used with permission; (c) q-ω model (Choi and Knight 1988). Copyright ©1988 American Institute of Aeronautics and Astronautics. Used with permission; (d) Baldwin–Lomax mixing-length model (Ameri and Arnone 1991)

is shown in Figure 17b. Choi and Knight (1988) used this experimental database to compare with their 3-D numerical code predictions. Their code uses the q-ω two-equation turbulence model. Agreement with the experiment was found to be good, as shown in Figure 17c. The possible reasons for the apparent success of low-Reynolds-number, two-equation turbulence models may be found in some ideas expressed by Launder et al. (1984). Launder states that "velocity gradients normal to a rigid boundary are so steep that, with the exception of flow near spinning surfaces, the low-Reynolds-number region can usually be regarded as in simple shear." He further states that streamwise convective transport effects are of secondary significance, suggesting that Boussinesq viscosity models will do the job of turbulence modeling as one nears the wall. He goes on to explain how, in what may be a compensating error, the calculation of an incorrect length scale can result in the correct overall heat and momentum transfer resistance. This occurs even though the predicted boundary layer profiles may not correspond exactly to the data. He recommends that improved near-wall length scale predictions be incorporated. This recommendation is being addressed by such efforts as that of Shih and Mansour (1990).

Mixing-length models. The above comments help to explain the apparent success of 3-D Navier–Stokes codes that use mixing-length models. Such models cannot account for the history and transport effects of turbulence. Chima and Yokota (1988) present results of a 3-D Navier–Stokes code that uses the Baldwin–Lomax model for a horseshoe vortex created on a flat plate by a circular cylinder normal to the plate. Comparisons with experiment show excellent agreement for flow and pressure contours. This approach is extended by Ameri and Arnone (1991) to include heat transfer for the endwall of a linear turbine cascade. The objective of this work was to explore the ability of the Baldwin–Lomax model in a 3-D Navier–Stokes code to predict heat transfer. Numerical results were compared to the data of Graziani et al. (1980) and were found to be reasonably good, especially in the endwall region. This is partly attributed to the fully turbulent nature of the flow in that region, as shown in Figure 17d. Since mixing-length models require corrections for pressure gradient, modifications were recommended for the model to improve the heat transfer results in the regions with large pressure gradient. The model also does not comprehend transition behavior on the blade and requires a transition criterion to be invoked. While the comparisons of the approach of Hah (k-ε model), Choi and Knight (q-ω model), and Ameri and Arnone (Baldwin–Lomax) with the experimental data of Graziani (Figure 17) reveal individual differences, the encouraging result is that they all capture the right trends. To give additional credibility to the use of a mixing-length approach, it is worth noting that Anderson and Eaton (1989) used the Rotta (1979) T-mixing-length model to get reasonable predictions of the experimental shear stresses. This model permits an anisotropic eddy viscosity.

"Wall-function" type modeling. Launder et al. (1984) describe wall function methods by which the inner, low-Reynolds-number, layer may be described by one-dimensional (1-D) near-wall solutions. Such methods have long been in use and would obviate the need to compute a numerical solution all the way to the wall, where high velocity and temperature gradients require about 50 percent of the grid points. However, the assumptions made in developing these traditional 1-D wall functions may not be applicable to complex 3-D flows.

A better approach is to develop solutions for 3-D near wall layers which reflect the physics of such flows. Work of this nature was done by Goldberg and Reshotko (1984) and Degani and Walker (1989, 1990). Goldberg and Reshotko performed a 3-D asymptotic analysis for high-Reynolds-number turbulent boundary layers and developed solutions for 3-D law-of-the-wall and law-of-the-wake models for the pressure-driven part of the turbomachinery endwall flow. The results compare well with experimental velocity values and their direction in the wall region. They concluded that the flow in the wall layer is 2-D and in the direction of the wall shear stress. An asymptotic expansion approach was used by Walker et al. (1986) to develop wall functions for 2-D turbulent flows, taking into consideration the coherent structure of the near wall flow. This approach was extended to 3-D flows for cases of no heat transfer and heat transfer by Degani and Walker (1989, 1990). In their asymptotic analysis, terms were included that took into account the effect of pressure gradient. This resulted in the conclusion that second-order effects of pressure gradient are not, in general, negligible and will cause the velocity and shear stress profile to skew significantly. The authors state that this is more in line with the experimental evidence. This is contrary to the conclusions of Goldberg and Reshotko. In their application of the asymptotic approach for predicting heat transfer, Degani and Walker (1990) used profiles developed by Walker et al. (1986) for the inner layer velocity and enthalpy. They compared their computed results of turbulent boundary layer in a plane of symmetry for a cylinder normal to a flat plate with the computations to the wall of a 3-D Navier–Stokes code that uses a Cebeci–Smith eddy viscosity model with constant Prandtl number. While there is little difference in the results of the two approaches, we believe that their wall function method will give more realistic results for heat transfer with pressure gradients.

This survey of turbulence models for endwall heat transfer code calculations leads us to conclude that for practical, accurate calculations at the end wall, the best choice for the present is a low-Reynolds-number, k-ε or q-ω model.

Film cooling

The protection of turbine blades from the effects of the hot gas stream generally requires the use of film cooling. The practical application for turbine blades is the injection of air through rows of holes inclined in the direction of flow (discrete film cooling) to create a blanket of coolant. Extensive experimental and analytical work has been performed to develop a predictive capability based on the experimental results. A good introduction to the understanding of the 3-D nature of discrete film cooling is given by Goldstein (1971). Eckert (1984) presents experimental correlation techniques that use the temperature superposition approach for predicting film cooling for a single hole or several rows of holes and for "full-coverage film cooling." This is a good example of the traditional computational approach to film cooling, which continues to be an effective way to use configuration-specific experiments for design purposes. A more general computational approach that has broader implications for design requires a numerical solution of the basic flow equations. These solutions need to comprehend 3-D flow and the elliptic reverse-flow region in the vicinity of the hole, created by the interaction of the main flow and discrete jets. This suggests that 3-D elliptic numerical calculations are required in the vicinity of the hole. Farther from the hole, the flow becomes more 2-D after the jets have bent over to the wall and grown together and the spanwise variations have disappeared; here, simpler analyses may suffice. Numerical film cooling codes need to comprehend many variables, such as the injection angle, hole spacing, velocity and density ratio of the jet and the mainstream, the state and dimensions of the oncoming boundary layer, surface curvature, pressure gradient, and free-stream turbulence. The following is a brief survey of how well existing film cooling models are meeting these requirements.

Mixing-length model. One of the early numerical codes for film cooling calculations was a modification of the STAN5 2-D boundary-layer program of Crawford and Kays (1976). The modification used a full-coverage film cooling model developed by Crawford et al. (1980), and the code was named STANCOOL. Three-dimensional lateral mixing was accounted for by an augmentation of the mixing-length model. The authors reported some success with this model, although it was not a surprise to find that the model did not accurately describe the near-hole region where 3-D effects are important. Schonung and Rodi (1987) state that the model does not account for a blockage effect that creates a fluid velocity above the injected jet that can be higher than the free-stream velocity. Harasgama and Burton (1991) used STANCOOL to predict film cooling at the endwall of a turbine annular cascade and determined that the 3-D nature of the flow at the endwall prevents STANCOOL from providing accurate Stanton number results. The experience with a 2-D model has pointed out the need for a model that better comprehends 3-D flow and the elliptic nature of the flow in the near-hole region.

One approach that uses a mixing-length model and accounts for 3-D effects in a simplified manner is the modeling performed by Kulisa et al. (1991). They applied spatial averaging in the direction of a row of jets to obtain a set of 2-D equations with 3-D source terms. The 3-D behavior of the jet is analyzed by an integral approach requiring additional closure relationships for the jet/free-stream interaction. The use of a mixing-length model did not permit a proper accounting of the high turbulence at the jet orifice. The method compared well with a film-cooled flat plate but appears to have limited application and requires improvements to the model.

k-ε models. The k-ε models used in numerical codes to predict film cooling are of two types. The simpler models assume isotropic eddy viscosity. A more complex model by Bergeles et al. (1978) incorporates an anisotropic version of the eddy viscosity and permits a more realistic modeling of the turbulent stresses. Bergeles et al. (1978) found the use of the isotropic k-ε model for film cooling to be inadequate. According to Demuren et al. (1986), this latter model causes a faster spreading of the injected jet in the lateral direction than the standard isotropic k-ε model. Laser Doppler velocimetry (LDV) measurements of Pietrzyk et al. (1989, 1990) give support to the application of eddy-viscosity models for film cooling. The work of the latter authors shows a correspondence between the measured value of turbulent shear stress and the velocity gradient.

Demuren et al. (1986) used the Bergeles anisotropic k-ε turbulence model in a 3-D, locally elliptic numerical method. It produced good comparisons of film-cooling effectiveness with 27 experimental test cases for low and high film-cooling blowing rates. Some difficulties were experienced in predicting film effectiveness for the near-hole region and for large spacing. Because this method required a great amount of computational investment, Schonung and Rodi (1987) used a 2-D boundary-layer code with a modification for the 3-D elliptic flow to predict film-cooling effectiveness and heat transfer. An injection model was used to account for the elliptic nature of the flow after injection and a dispersion model for the three-dimensionality of the flow. The dispersion terms describe the additional mixing by the lateral flow. The empirical constants required for the injection and dispersion models were determined from the 3-D elliptic calculations of Demuren et al. (1986), and an isotropic k-ε model was used. Haas et al. (1991) extended this approach to include the effect of the density difference between the hot stream and the coolant jet. The numerical results gave favorable comparisons with experimental results for a row of holes for flat plates, turbine blades, and a range of injection angles, spacings, blowing rates, and injection temperatures. It was determined that important curvature effects were not accounted for in this model.

Tafti and Yavuzkurt (1990) use a 2-D injection model with a low-Reynolds-number isotropic k-ε model, and account for the 3-D effects with an "entrainment fraction." The entrainment fraction is correlated to injection parameters by comparing predictions to experimental data for one row of holes. The intent of the work is to extend a previous predictive approach of Tafti and Yavuzkurt (1988) to multirow injection geometries and to injections into a laminar boundary layer, convex curved surfaces, and on a turbine blade. This is a simple yet comprehensive numerical approach that accurately predicts the spanwise average film-cooling effectiveness. Predictions are generally good as compared with experimental data except at the higher blowing rates where Stanton numbers are underpredicted.

Sathyamurthy and Patankar (1990, 1992) present a fully 3-D parabolic model that uses the Bergeles anisotropic turbulence model. The model neglects streamwise diffusion and reverse flow. It is reported to be as successful as the models that are partially parabolic and locally elliptic. This appears to be more so for a row of holes than for a single hole. Even for a single hole, there is excellent agreement for low blowing rates. As the blowing rate increases, elliptic effects become important and remain important for about eight diameters downstream, at which point the parabolic approach becomes more effective. In general, for a single row of holes, the 3-D locally elliptic method of Demuren et al. (1986) is not a great improvement over the fully 3-D parabolic approach. Both methods need improvements in the near hole region. The main advantage of the fully 3-D parabolic method is economy of computer resources.

A practical way to account for the flow differences between the near hole region and the region downstream from the hole is to apply an elliptic numerical approach to the near hole region and a partial parabolic method downstream of the hole. This was done by Dibelius and Wen (1989) and Dibelius et al. (1990) using an isotropic k-ε model. In general, there was good agreement between the calculations and experiments. Discrepancies between experiment and calculations in the separated, near hole region are attributed to the inability of the k-ε model to capture separated flow. An interesting result from the work of Dibelius and Wen is a criterion for determining when the jet is attached to the wall with no reverse flow, as shown in Figure 18. The figure shows that the right combination of injection angle and blowing rate will avoid a reverse flow region. Sathyamurthy and Patankar (1992) state that their 3-D parabolic approach gives good results for a shallow angle of 10° and a blowing ratio of 1. Figure 18 confirms the reason for the good performance of the parabolic approach.

Sathyamurthy and Patankar (1990) used their 3-D parabolic approach for the case of film cooling with lateral injection. The comparisons with experiment are very good, and it was determined that lateral injection can have blowing rates higher than the optimum for streamwise injection to produce a better coverage of film cooling than streamwise injection at comparable blowing rates. This is an example of the type of numerical analysis that needs to be performed on different film-cooling hole configurations or shapes. Shaped holes have shown potential to increase film-cooling effectiveness. An example of this is given in the experimental works of Papell (1984) and Simon and Ciancone (1985). Numerical analysis of the experimental results with shaped holes should result in increased understanding of the governing mechanisms at work in shaped holes and aid in their design.

Another aspect of discrete film-cooling analyses that needs more attention is the effect of free-stream turbulence on the film-cooling effectiveness. Experimental (Marek and Tacina 1975) and analytical (Simon 1986) studies with slot film cooling demonstrate the large degrading effect that free-stream turbulence can have on film-cooling performance.

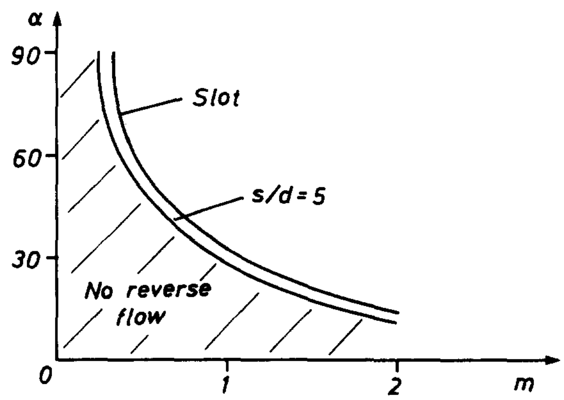

Figure 18 Criterion for determining when a film-cooling jet remains attached to the wall with no reversed flow (Dibelius and Wen 1989). Used with permission from Friedr. Vieweg & Sohn Verlagsgesellschaft

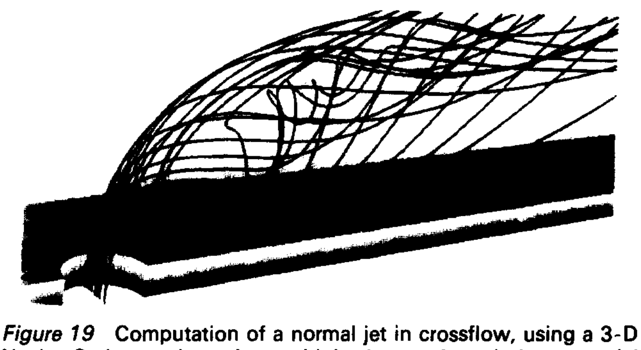

Figure 19 Computation of a normal jet in crossflow, using a 3-D Navier–Stokes code and a multiple-time-scale turbulence model (Kim and Benson 1992)

Finally, a possible approach to improving the modeling of the separated zone in the near hole region is to model the nonequilibrium nature of the turbulence with a multiple-time-scale model, as reported by Kim and Benson (1991, 1992). This model, performed for a normal jet in crossflow, produced excellent results of a primary vortex, a secondary vortex, and a horseshoe vortex formed from the interaction of the mainstream with the jet. An example of the flow complexities captured by the work of Kim and Benson is shown in Figure 19. The ability of the multiple-time-scale model to capture the complexity of the turbulence field about a jet makes it a prime candidate for future modeling efforts in film cooling.

For the present, it appears that 3-D or 2-D film cooling models that make provision for 3-D effects and use a k-ε type turbulence model in a semi-parabolic or parabolic numerical scheme produce results that are computationally practical and acceptable.

Code development

As stated earlier, convective heat transfer predictive capability in the turbine gas path has traditionally lagged behind the fluid mechanics predictive capability. This is still true. Although 3-D Navier–Stokes codes are beginning to enter the aerodesign decision-making process, they are not yet a part of the heat transfer design decisions.

For many years heat transfer design decisions were made on the basis of empirical correlations, coupled with industry proprietary test data. In the 1970s, with the introduction of the Patankar and Spalding (1972) method and the highly popular STAN5 (Crawford and Kays 1976) boundary-layer code, turbine gas path heat transfer prediction took on a new dimension. Presently, 3-D Navier–Stokes codes with heat transfer are beginning to emerge. We will discuss the boundary-layer methods and the Navier–Stokes codes and leave the empirical methods to history. Before leaving them, however, one comment is in order—simplicity is good. If you have a reliable correlation that covers the range of interest, then use it.

The use of advanced computational methods to analyze, and ultimately predict, turbine gas path heat transfer really began with the introduction of the STAN5 boundary-layer code by Crawford and Kays (1976). STAN5, based on the Patankar and Spalding (1972) method, is a highly modularized, finite-difference computer code that was designed to predict heat transfer through a flat-plate boundary layer. Over the years it has been continually modified and upgraded, particularly with respect to new turbulence/transition models. In addition to being a predictive tool, it has been a test bed for exploring new models (cf. Bypass Transition Region). Crawford (1985) has continued to maintain STAN5 in the form of a code called TEXSTAN. Members of the aeropropulsion industry have modified it and incorporated it into their design and analysis schemes. These codes are not often published, but one example can be found in a paper by Zerkle and Lounsbury (1989). This code allows for local free-stream velocity distributions, which can be supplied empirically or through coupling it with an aero analysis. One popular technique is to combine boundary-layer codes with the quasi-3-D Euler analyses of Katsanis and McNally (1977) in what has become known as the MTSB code described by Boyle et al. (1984). This code is particularly popular, because the Katsanis codes have been part of industry design systems for a long time and industry is comfortable with them. Although advanced modeling is beginning to enter predictive codes, the present standard for design and analysis is primarily algebraic mixing models and empirical transition models. The use of k-ε models in these codes is just beginning to emerge.

Another reason the boundary-layer codes are popular is that they are computationally efficient when compared with the Navier–Stokes codes. This is because they are parabolic and focus on a very small physical space in a large flow field. This in turn allows for the very high grid-resolution computations necessary for some heat transfer calculations. This is particularly true in the airfoil midspan region, where the flow is nearly 2-D and laminar-turbulent transition is one of the major flow features. Figure 20, taken from Zerkle and Lounsbury (1989), illustrates both the strength and weakness of such methods. The calculations, using a k-ε turbulence model, agree well with data over most of the region, but are very sensitive to free-stream turbulence in the transition region and miss the last two data points. When transition is less of an issue, the sensitivity to free-stream turbulence is less and the prediction is better. An obvious problem with these methods is starting the calculation on turbine airfoils, since it is basically a flat-plate calculation. Most codes have some starting scheme built into them. To examine this problem and to explore the use of Navier–Stokes analyses, Boyle (1990) used the thin-layer quasi-3D Navier–Stokes code of Chima and Yokota (1988) in an extensive investigation of various computational models, compared to several datasets. One example from Boyle's work, using the rotor data of Blair et al. (1989a) for comparison, is shown in Figure 21. The same type of behavior as was seen by Zerkle and Lounsbury (1989) with the boundary-layer code is seen here. In fact, one of the conclusions that can be drawn from Boyle's work is that the turbulence models developed in flat-plate experiments and with boundary-layer analyses can be extrapolated to the Navier–Stokes codes. They appear to work equally well, or equally poorly. This is important, because it suggests that the large body of work done on flat plates and simple 2-D channels remains valuable. It also suggests that

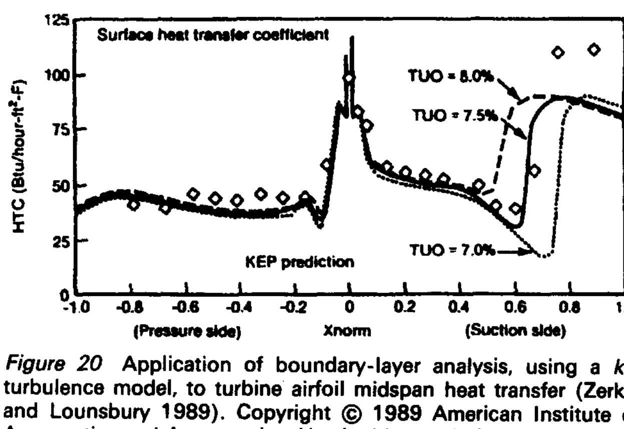

Figure 20 Application of boundary-layer analysis, using a k-ε turbulence model, to turbine airfoil midspan heat transfer (Zerkle and Lounsbury 1989). Copyright © 1989 American Institute of Aeronautics and Astronautics. Used with permission

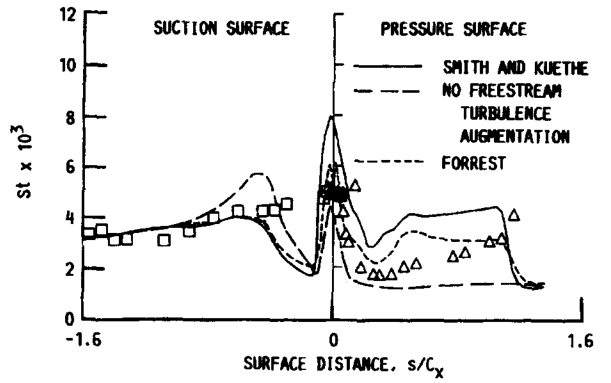

Figure 21 Comparison of various turbulence models developed from flat-plate data applied to turbine data of Blair et al. (1989a, 1989b), using a quasi-3-D Navier–Stokes code (Boyle 1990)

boundary-layer analyses can yield valuable insight, at least in the important midspan region. In the endwall region, not surprisingly, boundary-layer codes are much less successful. Since the experiments show that the flow in this region is truly 3-D and the outer flow field has major significance, one would expect this type of flow to be beyond the range of boundary-layer analyses. These analyses can be made to work, as per the example of Anderson (1987), but the effort to get the edge condition correct is so complex that one wonders if it is worth the effort. The incorporation of 3-D "wall functions" into the Navier–Stokes codes might be more practical.

Choi and Knight (1988) also showed good 3-D results on the blade surface, as shown in Figure 22. The subtle, but important, differences between experiment and computation are somewhat masked by the scale of the figure. The same transition sensitivity at midspan, as discussed earlier, is present in Figure 22. Choi and Knight (1991a) have explored the transition calculations in a recent paper. The focus here should be on the ability to capture the three-dimensionality. The endwall region is where the 3-D Navier-Stokes code will be most needed and where the investment will be most productive. Furthermore, transition is less of a factor in this region, lessening the need for some of the very high resolution turbulence models. This discussion was started in the preceding section and continues here. Probably the longest-term effort to develop 3-D Navier-Stokes codes *with heat transfer* has been by Knight and Choi (Knight and Choi 1987; Choi and Knight 1988, 1991a, 1991b), using the q-ω model, and by Hah (Hah 1984, 1989; Hah and Selva 1991), using the k-ε model. Both have compared their work to the Graziani et al. (1980) experiment with generally good results, as shown in Figure 17. A number of heat transfer codes have used the Baldwin–Lomax algebraic turbulence model, popular with the turbine aerodynamicists. The results of Ameri and Arnone (1991), using this model, are also shown in Figure 17. Dorney and Davis (1991) performed similar analyses using their 3-D Navier-Stokes code. In general, the results with Baldwin–Lomax are qualitatively good, but not as good as those using the two-equation models. In response to these shortcomings, Chima et al. (1993) have recently made a substantial revision to the Baldwin–Lomax turbulence model and have examined the Reynolds number dependence reported by Boyle and Russell (1990), as shown in Figure 23. The major trends are captured and the actual level of heat transfer predicted is quite good, especially in the high-Reynolds-number case.

A very popular 3-D Navier-Stokes code among the turbine aerodynamicists is the Dawes (1988) code; however, to date very little work has been done with it for heat transfer. Vogel (1991) has modified the Dawes code for heat transfer and has

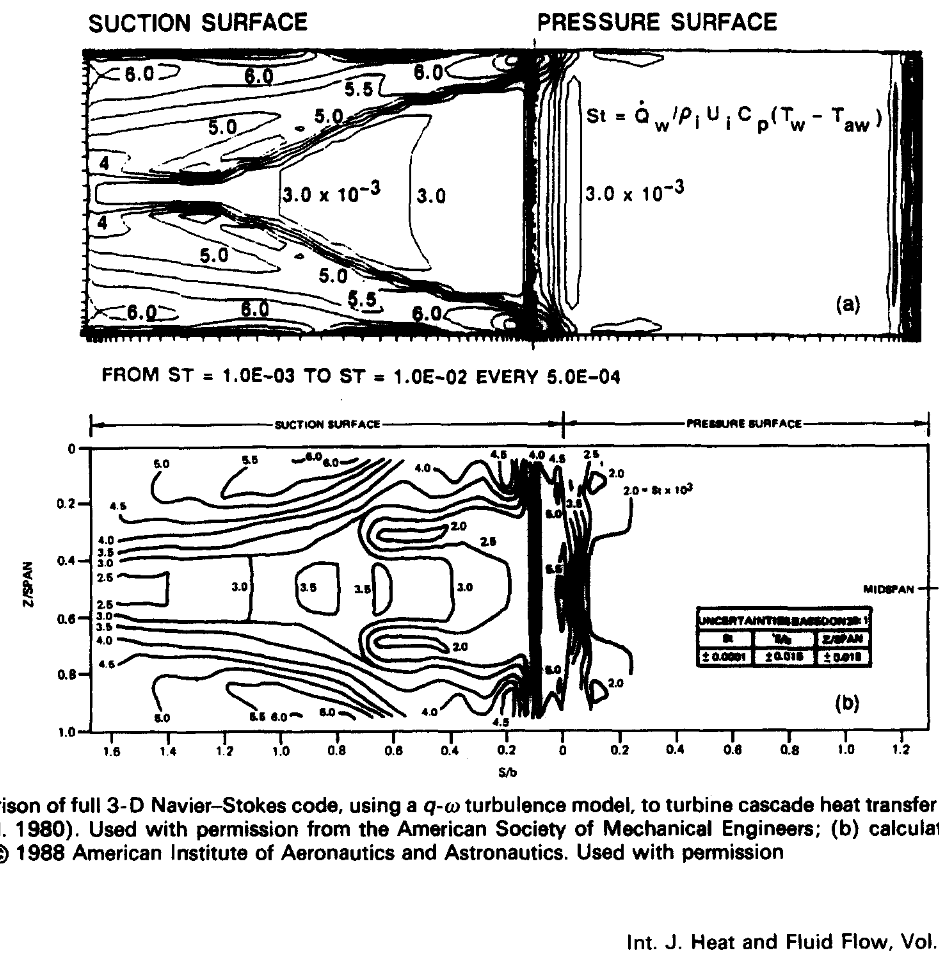

Figure 22 Comparison of full 3-D Navier–Stokes code, using a q-ω turbulence model, to turbine cascade heat transfer data. (a) Experimental data (Graziani et al. 1980). Used with permission from the American Society of Mechanical Engineers; (b) calculation (Choi and Knight 1988). Copyright © 1988 American Institute of Aeronautics and Astronautics. Used with permission

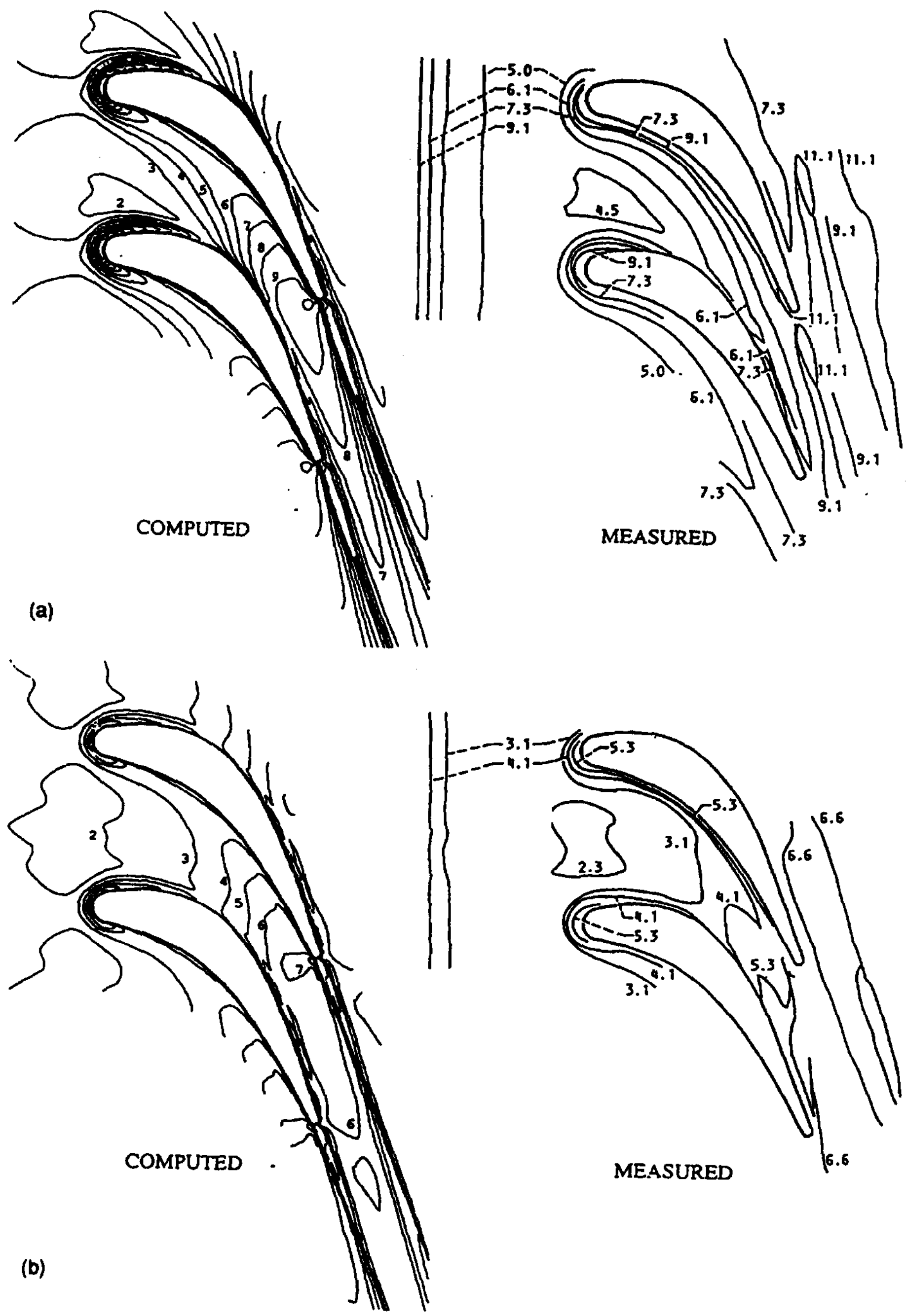

Figure 23 Comparison of full 3-D Navier–Stokes calculation, using a new modified Baldwin–Lomax mixing-length model, to endwall data of Boyle and Russell (1990) (Chima et al. 1993). (a) Low Reynolds number (78,000); (b) High Reynolds number (490,000)

included film cooling. Also, Dawes (1992) has recently added heat transfer to his code, albeit for a coolant side problem. These efforts should be followed for future progress.

It is encouraging, and maybe just a bit surprising, that the 3-D Navier–Stokes codes with simple models are doing so well. Two things seem to favor success. First, turbine flows are strongly pressure driven and, despite strong secondary flows, the main flow direction is almost always well established. Second, the ability to predict heat transfer accurately is greatly enhanced by getting the local free-stream total temperature correct. The codes seem to be able to do this well.

While the film cooling modeling effort is proceeding rapidly, the progress in incorporating advanced models into turbine flow codes that include film cooling has not been so dramatic. The industry standard today is a boundary-layer code, modified for film cooling, of the STANCOOL variety, which was discussed in some detail in the previous section on modeling. The full 3-D Navier–Stokes codes, as well as semi-elliptic versions of the parabolized codes, have been used to analyze individual holes or a small group of holes to study the local physics. This was also discussed in the previous section. To our knowledge, the only full 3-D Navier–Stokes code with film cooling is the work of Vogel (1991) and very recent work by Garg (1992). Vogel (1991) has incorporated film cooling into the Dawes code, mainly through the use of correlations and user-supplied input as to the location and nature of the film cooling parameters. This has a long way to go, but it is a beginning. Garg and Gaugler (1992), using the Chima and Yokota (1988) code and a Cebeci-Smith turbulence model, have shown some very promising results for the data of Nirmalan and Hylton (1990), as shown in Figure 24. The focus of this discussion applies to the prediction of heat transfer. If one is interested only in the aerodynamics of film cooling, Haller and Camus (1984) have demonstrated that Euler codes with the addition of source terms can be quite useful.

Up to this point every computational effort that has been discussed has been a steady-state analysis of an isolated blade row. Yet, recalling Figure 3, what we really have at work is a very complex time-dependent phenomenon. We have already cited long-standing evidence that the results in a turbine are different from the isolated blade row. Ultimately it is this unsteadiness that must be modeled and properly accounted for in the analyses. This can be done by using a time-accurate solution or by using a properly averaged time-averaged solution. At this time there are very few analyses of either type that treat heat transfer. Part of the work of Dring et al. (1986) included an unsteady boundary-layer analysis of their heat transfer data. Tran and Taulbee (1991) conducted an unsteady Euler/boundary-layer analysis of the Dunn (1990) unsteady heat transfer data with some success.

There are a number of fine efforts that attack the flow physics. Getting the flow physics correct, especially the total temperature distribution, is probably the major portion of the battle, so we will comment on these efforts. The time-accurate turbine flow field analyses have been led by the efforts of Rai (1989a, 1989b), Rai and Madavan (1990), and Rao and Delaney (1990). Rai modified the UTRC turbine geometry slightly to create four-rotor/three-stator periodicity and then compared the unsteady pressure calculations to the experiment of Dring et al. (1982) with quite favorable results. The results were also compared with the turbine exit total pressure contours measured by Butler et al. (1986) with qualitatively similar results. As part of a joint program between Allison Gas Turbine and Calspan, supported by the U.S. Air Force (Dunn 1990), a transonic turbine with an exact three-rotor/two-stator periodicity was constructed and tested. The results appear to support the time-accurate pressure analyses of Rao and Delaney (1990), although some questions exist. Their analysis of the blade unsteady pressure (Figure 25), if supported by continuing research, has important implications for future predictive capability. The results indicate that time-average of the unsteady behavior may not be the same as a steady calculation of an isolated blade row. Figure 25 is an early result that has been refined by Rao et al. (1992a, 1992b) and now includes heat transfer. The results are similar and the main conclusion is the same.

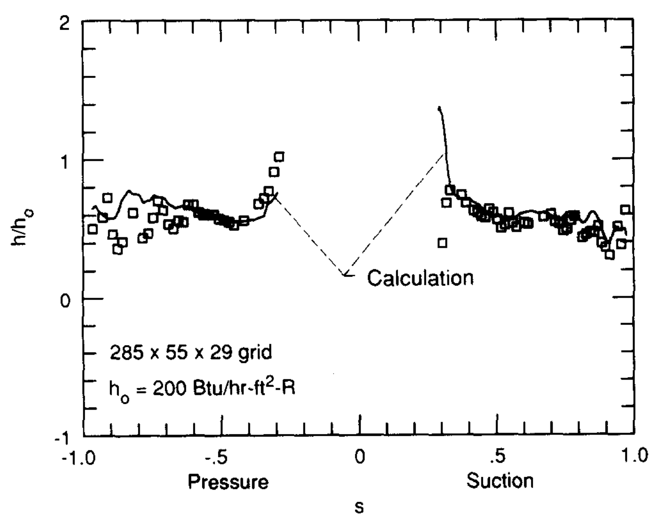

Figure 24 Comparison of Navier-Stokes calculations with the vane film-cooling data of Nirmalan and Hylton (1990) (Garg and Gaugler 1992)

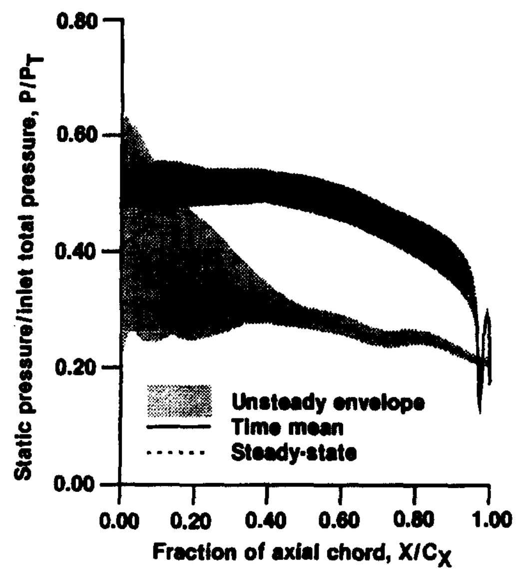

Figure 25 Rotor/stator interaction calculations, using a time-accurate Euler solver (Rao and Delaney 1990). Copyright © 1990 American Institute of Aeronautics and Astronautics. Used with permission

Another approach to understanding and modeling the unsteady flow physics of the turbine has been developed by Adamczyk et al. (1990). The idea set forth by Adamczyk (1985) is to break the flow field variables down into a quadruple decomposition. The instantaneous axial velocity, for example, contains an average term, a random unsteadiness term, a blade-passing unsteadiness term, and an unequal-blade-count unsteadiness term. Operations similar to Reynolds averaging are performed on the terms in these equations. Needless to say, this yields a huge number of terms to be modeled, and a program is under way to do that. Early results, modeling just a few terms (Adamczyk et al. 1990), are very encouraging.

In summary, the computational approach should be to use the level of sophistication needed to reliably predict the gas path region of interest. In the truly 3-D regions, such as endwalls, this probably means 3-D Navier-Stokes. Early results in these regions are encouraging. Along the midspan transition is a major issue. Thus, high resolution and sophisticated modeling will probably be needed. However, the computation may not necessarily need a 3-D Navier-Stokes code. A boundary-layer approach, zonally embedded in a bigger code (Euler or Navier-Stokes; quasi-3-D or full 3-D), may do well. Other regions, such as stagnation regions or film-cooling hole regions, may still find wall functions or correlations, embedded in the codes, to be the best approach.

Interactive model for developing predictive capability

We started this paper (see the Introduction) with a brief reference to a model for developing a convective heat transfer predictive capability in the turbine gas path (Figure 2). At this point we would like to expand on it. It is our opinion that the technical community has at its disposal very sophisticated computational and experimental tools with which to attack the very complex problems associated with the advanced design of modern aeropropulsion systems. (This can also be said of many other complex heat transfer systems.) It is also our opinion that in order to use these tools effectively to achieve an advanced capability, they must be used in a synergistic, interactive manner. This has not been the history. Traditionally, the research has been done in a more sequential manner in which the computational analyses have been compared to previously acquired data, or vice versa. The comparisons, cited earlier, of all the recent 3-D Navier–Stokes heat transfer computational analyses with the experiments of Langston et al. (1977) and Graziani et al. (1980) are excellent examples of this type of sequential approach at its best.

What will be needed in the future is an approach where the computational and experimental teams are working interactively, feeding the results of their work back and forth to make the most of both the experiment and the analysis. The modeling must be an integral part of this process. Unfortunately, at present there are very few heat transfer examples of this model at work, at least in the turbine heat transfer arena. The work reported by Guenette et al. (1989) is a good beginning in this type of approach.

In aerodynamics there are a number of fine examples of computational and experimental interaction, and the readers are directed to them. The work reported by Leylok and Wisler (1991) from research done in the General Electric large, low-speed axial compressor is a good example. A recent paper by Hathaway et al. (1992) is probably the best example of the model at work. Changes in the experiment design and procedure were made as a result of the analyses, and changes in the analyses were made as a result of the experiment. The effects of the one on the other were then evaluated. The work cited earlier of Dunn (1990), working with Rao and Delaney (1990) and Rao et al. (1992a, 1992b), is also a good example. So too is the tie between the research in the UTRC large, low-speed turbine (Dring et al. 1980, 1982; Blair et al. 1989a, 1989b, 1992b) and Rai et al. (Rai 1989a, 1989b; Rai and Dring 1990) and Adamczyk et al. (Adamczyk 1985; Adamczyk et al. 1990). More recently, the Rai code has been used by Rai and Dring (1990) and Dorney et al. (1990) to analyze simulated combustor generated inlet temperature distortions and hot-streak migrations through the same machine.

The ties to UTRC and the Calspan research are particularly noteworthy for this paper, because they deal with turbines and have a long tradition of heat transfer measurements. Future programs include plans for a more interactive research with the developing heat transfer codes. The work reported by Blair (1992b) shows the beginning of this effort. The work of Guenette et al. (1989) at MIT has already been mentioned and it is continuing. The turbine facility reported by Wedlake et al. (1989) and Harasgama and Wedlake (1991) for a turbine annular cascade at RAE is being expanded to a full stage and is expected to be more closely coupled to the heat transfer code development. A new and larger blowdown facility, built on the MIT model, has just been brought on line at Wright Patterson Air Force Base (Haldeman et al. 1991), with plans for substantial flow-field and heat transfer measurements on real turbine hardware, coupled with computational analyses.

Another effort is just beginning at NASA in a newly constructed transonic cascade facility, described by Verhoff et al. (1992). The machine was originally designed using the MTSB code (Boyle et al. 1984). Subsequently the design was analyzed using the 3-D Navier–Stokes codes of Ameri and Arnone (1991) and Chima et al. (1992). This led to design modifications. One example of the calculations from Ameri and Arnone (1991) is shown in Figure 26. The cascade is also being analyzed with

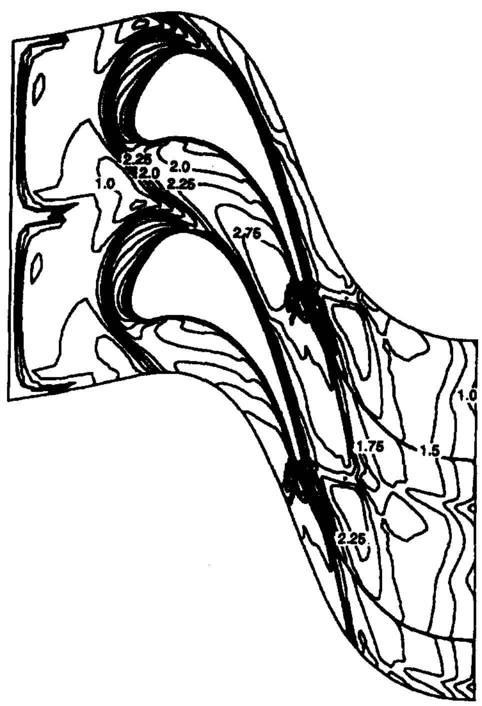

Figure 26 Calculations of endwall region heat transfer high-work, high-turning transonic turbine blade cascade, using 3-D Navier–Stokes code (Ameri and Arnone 1991).

the code of Hah and Selva (1991). The cascade is now operational, and the blade coordinates have been distributed to approximately two dozen researchers internationally. Plans include periodic workshops with the involved turbine heat transfer community to review and evaluate both the computational and experimental progress.

Concluding remarks

By examining the literature and drawing on our own experience, we have attempted to provide a critical examination of the state of the art in the prediction of convective heat transfer in the turbine gas path. Configuration-specific experiments, fundamental physics and model development, and heat transfer code development were all examined for their status and impact on establishing a capability. Although we cited a substantial body of literature, it was not our intention to perform a comprehensive literature survey, but rather to build a story and provide some insight.

From this experience we feel able to draw some conclusions about the present state of the art in turbine gas path convective heat transfer and to offer some suggestions for the future.

1. The state of the art in the prediction of turbine gas path convective heat transfer is advancing rapidly;
2. the present standard for turbine heat transfer prediction is a 2-D boundary-layer code embedded in a quasi-3-D Euler code:
   a. the turbulence modeling relies heavily on algebraic mixing-length models, with some use of k-ε modeling;
   b. transition predictions are largely empirical, based on models developed on flat plates;
3. a truly reliable 3-D predictive capability is not yet in hand; however, the tools to put such a capability in place are at hand;
   a. the necessary heat transfer codes are under development and showing good progress;

   b. the proper modeling efforts are under way and substantial progress is being made;
   c. there are numerous fine configuration-specific experiments, which are yielding very impressive turbine heat transfer results;
4. fundamental physics and model development is showing significant progress on several fronts:
   a. both laboratory and numerical experiments have yielded significant insight into the key physics, such as laminar-turbulent transition, associated with turbine heat transfer;
   b. new models, particularly k-ε models that have been specially tailored to a particular class of problems, are showing promise, at least for meaningful near term analyses;
   c. the evaluation of models, such as mixing-length and k-ε, in full 3-D codes is growing and offering valuable information. This trend is expected to continue;
5. progress is less dramatic, from a turbine gas path perspective, in the area of film cooling; however, good work is under way:
   a. the field of predicting film cooling in the turbine gas path is largely empirical, using specific hole test data referenced to flat-plate correlations;
   b. highly detailed analyses of single or small clusters of holes are under way and, coupled with fine experiments, are yielding good insight for model development;
   c. very little film cooling capability has worked its way into the 3-D viscous codes and, when it does, it is largely empirical. The current standard is the modification of a boundary layer code;
6. the stage is set for the integration of this research into a synergistic, interactive model to build a comprehensive 3-D turbine gas path heat transfer predictive capability.

At the present time, the recommendations for design analyses of turbine gas path convective heat transfer can be separated into two categories: midspan heat transfer and endwall heat transfer.

The midspan region can be defined as the middle 50 to 75 percent of the vane/blade span. It is the region where the endwall boundary layer has little or no influence. The flow is nearly 2-D or quasi-3-D. The flow can be characterized either by combination Euler/boundary-layer codes, such as MTSB, or by Navier–Stokes codes. Care must be taken to have very good modeling for laminar–turbulent transition in the turbine, where transition is subject to many external forces. Good prediction of transition will generally require both sophisticated turbulence modeling and high resolution near the walls. Thin-layer or parabolized solutions near the wall should work, since the flows are strongly pressure driven and nearly 2-D. In summary, the midspan region is one where the flow field should be relatively easy to characterize and the modeling is the issue.

In the endwall region, the flow field needs to be characterized well, and the full 3-D Navier–Stokes code is the recommended choice for computation. On the other hand, while the turbulence must be modeled, reasonable heat transfer results are not nearly so dependent on the modeling, as they are in transition. Even though the physics is very complex and simple models should not be expected to work, they do a fairly decent job. Mixing-length models, for example, can be made to work. The models of choice, however, are the two-equation (k-ε or g-ω) models. These form an effective compromise between capturing the physics and computational efficiency. They do a better job than the mixing-length models and definitely will help the turbine heat transfer engineer make good design decisions. In both regions for film cooling, the designer will still be relying heavily on the empirical data base for the present, although better flow-field definition will be a big help.

What about the future? Where should we be going in our research? First of all, it is our position that for steady-state heat transfer predictive capability, we are already into the future. All the right pieces are in place to make a major assault on turbine gas path heat transfer.

Of course, the list of recommendations for the future could be very long. We will resist the urge to provide laundry lists and just make a few suggestions. First, we strongly believe that experimental/modeling/computational synergism needs to become the accepted way of doing business. It is neither research efficient nor cost effective for everyone to do their own thing, and then try to fit it together. For example, the modeling efforts are getting more and more sophisticated, as are the computational efforts. However, the integration of models into codes frequently remains rather primitive. Code developers use the simplest models that will work in their codes, while modelers keep developing models without regard to how they will impact a huge computation of a complex problem. It should be emphasized that this operational model does not have to exist in one organization. The people who can contribute the best, wherever they are, should do the work—just not in isolation.

Second, the key has been, is now, and will continue to be understanding the fundamental physics and modeling it properly. The look to the future needs to emphasize the fundamentals in the more complex problems; that is, get off the flat plate!

Finally, the next real frontier, which we have already engaged, is the unsteady flow physics inherent in turbomachinery. Identifying the key unsteady flow physics and modeling it in very complex geometries is the agenda for the future and will surely occupy the rest of the 1990s.

Acknowledgments

The authors wish to acknowledge and thank the many contributors to this article. While the authors accept full responsibility for all statements, this paper truly is the collective work of the technical community, many of whom appear in the References. Specific contributions of figures are cited with the respective figure and are gratefully acknowledged.

References

Abrahamson, S. D. and Eaton, J. K. 1991. Heat transfer through a pressure-driven three-dimensional boundary layer. *J. Heat Transfer*, **113**, 355–362

Adamczyk, J. J. 1985. A model equation for simulating flows in multistage turbomachinery. ASME Paper 85-GT-226

Adamczyk, J. J., Celestina, M. L., Beach, T. A., and Barnett, M. 1990. Simulation of three-dimensional viscous flow within a multistage turbine. *J. Turbomach.*, **112**, 370–376

Addison, J. S. and Hodson, H. P. 1990a. Unsteady transition in an axial-flow turbine: Part 1—Measurements on the turbine rotor. *J. Turbomach.*, **112**, 206–214

Addison, J. S. and Hodson, H. P. 1990b. Unsteady transition in an axial flow turbine: Part 2—Cascade measurements and modeling. *J. Turbomach.*, **112**, 215–221

Ameri, A. A. and Arnone, A. 1991. Three dimensional Navier–Stokes analysis of turbine passage heat transfer. AIAA Paper 91-2241

Anderson, O. L. 1987. Calculation of three-dimensional boundary layers on rotating turbine blades. *J. Fluids Eng.*, **109**, 41–50

Anderson, S. and Eaton, J. K. 1989. Reynolds stress development in pressure driven three-dimensional turbulent boundary layers. *J. Fluid Mech.*, **202**, 263–294

Arnal, D., Juillen, J. C., and Michel, R. 1978. Experimental analysis and computation of the onset and development of the boundary layer transition. NASA TM-75325

Arnone, A., Liou, M. S., and Povinelli, L. A. 1991. Multigrid calculation of three-dimensional viscous cascade flows. AIAA Paper 91-3238

Arts, T. and Bourguignon, A. E. 1990a. Behavior of a coolant film with two rows of holes along the pressure side of a high-pressure nozzle guide vane. *J. Turbomach.*, **112**, 512–521

Arts, T. and De Rouvroit, M. L. 1990. Aero-thermal performance of a two-dimensional highly loaded transonic turbine nozzle guide vane. ASME Paper 90-GT-358

Ashworth, D. A., LaGraff, J. E., Schultz, D. L., and Grindrod, K. J. 1985. Unsteady aerodynamic and heat transfer processes in a transonic turbine stage. *J. Eng. Gas Turbines Power*, **107**, 1022–1030

Bario, F., Leboeuf, F., Onvani, A., and Seddini, A. 1990. Aerodynamics of cooling jets introduced in the secondary flow of a low-speed turbine cascade. *J. Turbomach.*, **112**, 539–546

Bayley, F. J. and Priddy, W. J. 1981. Effects of free-stream turbulence intensity and frequency on heat transfer to turbine blading. *J. Eng. Power*, **103**, 60–64

Bergeles, G., Gosman, A. D., and Launder, B. E. 1978. The turbulent jet in a cross stream at low injection rates: a three-dimensional numerical treatment. *Numer. Heat Transfer*, **1**, 217–242

Binder, A., Forster, W., Kruse, H., and Rogge, H. 1985. An experimental investigation into the effect of wakes on the unsteady turbine rotor flow. *J. Eng. Gas Turbines Power*, **107**, 458–466

Binder, A., Forster, W., Kruse, H., and Rogge, H. 1987. Unsteady flow interaction caused by stator secondary vortices in a turbine rotor. *J. Turbomach.*, **109**, 251–257

Blair, M. F. 1974. An experimental study of heat transfer and film cooling on large-scale turbine endwalls. *J. Heat Transfer*, **96**, 524–529

Blair, M. F., Dring, R. P., and Joslyn, H. D. 1989a. The effects of turbulence and stator/rotor interactions on turbine heat transfer: Part I—Design operating conditions. *J. Turbomach.*, **111**, 87–96

Blair, M. F., Dring, R. P., and Joslyn, H. D. 1989b. The effects of turbulence and stator/rotor interactions on turbine heat transfer: Part II—Effects of Reynolds number and incidence. *J. Turbomach.*, **111**, 97–103

Blair, M. F. 1983. Influence of free-stream turbulence on turbulent boundary layer heat transfer and mean profile development, Part I—Experimental data. *J. Heat Transfer*, **105**, 33–40

Blair, M. F. 1992a. Bypass-mode boundary layer transition in accelerating flows. Unpublished

Blair, M. F. 1992b. An experimental study of heat transfer in a large-scale turbine rotor passage. Paper to be presented at the 37th ASME International Gas Turbine and Aeroengine Congress and Exposition, Cologne, Germany

Boyle, R. J., Haas, J. E., and Katsanis, T. 1984. Comparison between measured turbine stage performance and the predicted performance using quasi-3D flow and boundary layer analyses. NASA TM-83640

Boyle, R. J. and Russell, L. M. 1990. Experimental determination of stator endwall heat transfer. *J. Turbomach.*, **112**, 547–558

Boyle, R. J. 1990. Navier–Stokes analysis of turbine blade heat transfer. *J. Turbomach.*, **113**, 392–403

Bradshaw, P. 1972. The understanding and prediction of turbulent flow. *Aeronaut. J.*, **76**, 403–418

Brown, A. and Martin, B. W. 1979. Heat transfer to turbine blades with special reference to the effects of mainstream turbulence. ASME Paper 79-GT-26

Butler, T. L., Sharma, O. P., Joslyn, H. D., and Dring, R. P. 1986. Redistribution of an inlet temperature distortion in an axial flow turbine stage. AIAA Paper 86-1468

Camci, C. and Arts, T. 1990. An experimental convective heat transfer investigation around a film-cooled gas turbine blade. *J. Turbomach.*, **112**, 497–503

Camci, C. and Arts, T. 1991. Effect of incidence on wall heating rates and aerodynamics on a film-cooled transonic turbine blade. *J. Turbomach.*, **113**, 493–501

Chen, T. H. 1990. An investigation of the low Reynolds number two-equation models for pipe flows and channel flows. M.S. thesis, Mechanical Engineering Department, The University of Texas, Austin, TX

Chien, K. Y. 1982. Prediction of channel and boundary-layer flows with a low-Reynolds-number turbulence model. *AIAA J.*, **20**, 33–38

Chima, R. V. and Yokota, J. W. 1988. Numerical analysis of three-dimensional viscous internal flows. NASA TM-100878

Chima, R. V., Giel, P. W., and Boyle, R. J. 1993. An algebraic turbulence model for three-dimensional viscous flows. AIAA Paper 93-0083

Ching, C. Y. and O'Brien, J. E. 1991. Unsteady heat flux in a cylinder stagnation region with high freestream turbulence. *Fundamental Experimental Measurements in Heat Transfer*, D. E. Beasley and J. L. S. Chen (eds.). ASME, New York, 57–66

Choi, D. and Knight, C. J. 1988. Computation of three-dimensional viscous cascade flows. *AIAA J.*, **26**, 1477–1482

Choi, D. and Knight, C. J. 1991a. Improved turbulence modeling for gas turbine application. AIAA Paper 91-2239

Choi, D. and Knight, C. J. 1991b. Aerodynamic and heat transfer analysis of the low aspect ratio turbine using a 3D Navier–Stokes code. AIAA Paper 91-2240

Civinskas, K. C., Boyle, R. J., and McConnaughey, H. V. 1988. Impact of ETO propellants on the aerothermodynamic analyses of propulsion components. AIAA Paper 88-3091

Colladay, R. S. and Russell, L. M. 1976. Streakline flow visualization of discrete hole film cooling for gas turbine applications. *J. Heat Transfer*, **98**, 245–265

Consigny, H. and Richards, B. E. 1982. Short duration measurements of heat-transfer rate to a gas turbine rotor blade. *J. Eng. Power*, **104**, 542–551

Crawford, M. E. and Kays, W. M. 1976. STAN5—A program for numerical computation of two-dimensional internal and external boundary layer flows. NASA CR-2742

Crawford, M. E., Kays, W. M., and Moffat, R. J. 1980. Full-coverage film cooling on flat, isothermal surfaces: A summary report on data and predictions. NASA CR-3219

Crawford, M. E. 1985. TEXSTAN Program, University of Texas, Austin, TX

Dawes, W. N. 1988. Development of a 3D Navier–Stokes solver for application to all types of turbomachinery. ASME Paper 88-GT-70

Dawes, W. N. 1992. The solution-adaptive numerical simulation of the 3D viscous flow in the serpentine coolant passage of a radial inflow turbine blade. ASME Paper 92-GT-193

Degani, A. T. and Walker, J. D. A. 1989. Asymptotic structure and similarity solutions for three-dimensional turbulent boundary layers. AIAA Paper 89-1863

Degani, A. T. and Walker, J. D. A. 1990. Computation of three-dimensional turbulent boundary layers with heat transfer in a plane of symmetry using embedded wall-layer functions. AIAA Paper 90-0307

Demuren, A. O., Rodi, W., and Schonung, B. 1986. Systematic study of film cooling with a three-dimensional calculation procedure. *J. Turbomach.*, **108**, 124–130

Dhawan, S. and Narasimha, R. 1958. Some properties of boundary layer flow during the transition from laminar to turbulent motion. *J. Fluid Mech.*, **3**, 418–436

Dibelius, G. H., Pitt, R., and Wen, B. 1990. Numerical prediction of film cooling effectiveness and the associated aerodynamic losses with a three-dimensional calculation procedure. ASME Paper 90-GT-226

Dibelius, G. H. and Wen, B. 1989. Numerical approximation of film cooling with a three-dimensional calculation procedure. *Finite Approximations in Fluid Mechanics II. DIG Priority Research Programme Results*, E. H. Hirschel, (Ed.). Braunschweig, Friedge, Vieweg, 65–79

Donaldson, C. D. 1971. A progress report on an attempt to construct an invariant model of turbulent shear flows. NASA CR-125904

Doorly, D. J. and Oldfield, M. L. G. 1985a. Simulation of wake passing in a stationary turbine rotor cascade. *J. Propulsion Power*, **1**, 316–318

Doorly, D. J. and Oldfield, M. L. G. 1985b. Simulation of the effects of shock-wave passing on a turbine rotor blade. *J. Eng. Gas Turbines Power*, **107**, 998–1006

Doorly, D. J. 1988. Modeling the unsteady flow in a turbine rotor passage. *J. Turbomach.*, **110**, 27–37

Dorney, D. J. and Davis, R. J. 1991. Navier–Stokes analysis of turbine blade heat transfer and performance. *AGARD 77th Symp. CFD Techniques for Propulsion Applications*, AGARD, 22-1–22-11

Dorney, D. J., Davis, R. L., Edwards, D. E., and Madavan, N. K. 1990. Unsteady analysis of hot streak migration in a turbine stage. *J. Propulsion Power*, **8**, 520–529

Dring, R. P., Blair, M. F., and Joslyn, H. D. 1980. An experimental investigation of film cooling on a turbine rotor blade. *J. Eng. Power*, **102**, 81–87

Dring, R. P., Joslyn, H. D., Hardin, L. W., and Wagner, J. H. 1982. Turbine rotor-stator interaction. *J. Eng. Power*, **104**, 729–742

Dring, R. P., Blair, M. F., Joslyn, H. D., Power, G. D., and Verdon, J. M. 1986. The effects of inlet turbulence and rotor/stator interaction on the aerodynamics and heat transfer of a large-scale rotating turbine model. I—Final report. NASA CR-4079

Dullenkopf, K., Schulz, A., and Wittig, S. 1991. The effect of incident wake conditions on the mean heat transfer of an airfoil. *J. Turbomach.*, **113**, 412–418

Dunn, M. G. and Stoddard, F. J. 1979. Measurement of heat-transfer rate to a gas turbine stator. *J. Eng. Power*, **101**, 275–280

Dunn, M. G. and Hause, A. 1982. Measurement of heat flux and pressure in a turbine stage. *J. Eng. Power*, **104**, 215–223

Dunn, M. G., Rae, W. J. and Holt, J. L. 1984a. Measurement and analyses of heat flux data in a turbine stage: Part I—Description of experimental apparatus and data analysis. *J. Eng. Gas Turbines Power*, **106**, 229–233

Dunn, M. G., Rae, W. J., and Holt, J. L. 1984b. Measurement and analyses of heat flux data in a turbine stage: Part II—Discussion of results and comparison with predictions. *J. Eng. Gas Turbines Power*, **106**, 234–240

Dunn, M. G. 1984. Turbine heat flux measurements: influence of slot injection on vane trailing edge heat transfer and influence of rotor on vane heat transfer. ASME Paper 84-GT-175

Dunn, M. G. and Chupp, R. E. 1988. Time-averaged heat-flux distributions and comparison with prediction for the Teledyne 702 HP turbine stage. *J. Turbomach.*, **110**, 51–56

Dunn, M. G., Seymour, P. J., Woodward, S. H., George, W. K., and Chupp, R. E. 1989. Phase-resolved heat-flux measurements on the blade of a full-scale rotating turbine. *J. Turbomach.*, **111**, 8–19

Dunn, M. G. 1990. Phase and time-resolved measurements of unsteady heat transfer and pressure in a full-stage rotating turbine. *J. Turbomach.*, **112**, 531–538

Dunn, M., Bennett, W., Delaney, R., and Rao, K. 1992. Investigation of unsteady flow through a transonic turbine stage: data/prediction comparison for time-averaged and phase-resolved pressure data. *J. Turbomach.*, **114**, 91–99

Dyban, Y. P. and Kurosh, V. D. 1968. Comparative study of the heat transfer of a nozzle blade profile in a wind tunnel and in an air turbine. NASA TT-F-16 060

Eckert, E. R. G. 1984. Analysis of film cooling and full-coverage film cooling of gas turbine blades. *J. Eng. Gas Turbines Power*, **106**, 206–213

Garg, V. K. and Gaugler, R. E. 1992. Heat transfer in film-cooled turbine blades. Paper submitted to the 38th ASME International Gas Turbine and Aeroengine Congress and Exposition, Cincinnati, OH

Gaugler, R. E. and Russell, L. M. 1980. Streamline flow visualization study of a horseshoe vortex in a large-scale, two-dimensional turbine stator cascade. NASA TM-79274

Gaugler, R. E. and Russell, L. M. 1984. Comparison of visualized turbine endwall secondary flows and measured heat transfer patterns. *J. Eng. Gas Turbines Power*, **106**, 168–172

Gladden, H. J. and Yeh, Y. C. 1992. Summary of experimental heat transfer results from the turbine hot section facility. To be published as NASA TP-3250

Goldberg, U. C. and Reshotko, E. 1984. Scaling and modeling of three-dimensional, end-wall, turbulent boundary layers. NASA CR-3792

Goldstein, R. J. 1971. *Film Cooling. Advances in Heat Transfer*, Vol. 7. Academic Press, New York, 321–379

Goldstein, R. J. and Chen, H. P. 1985. Film cooling on a gas turbine blade near the end wall. *J. Eng. Gas Turbines Power*, **107**, 117–122

Goldstein, R. J. and Spores, R. A. 1988. Turbulent transport on the region between adjacent turbine blades. *J. Heat Transfer*, **110**, 862–869

Graham, R. W. 1985. Transition in turbines. NASA CP-2386

Graziani, R. A., Blair, M. F., Taylor, J. R., and Mayle, R. E. 1980. An experimental study of endwall and airfoil surface heat transfer in a large scale turbine blade cascade. *J. Eng. Power*, **102**, 257–267

Guenette, G. R., Epstein, A. H., Giles, M. B., Haines, R., and Norton, R. J. G. 1989. Fully scaled transonic turbine rotor heat transfer measurements. *J. Turbomach.*, **111**, 1–7

Haas, W., Rodi, W., and Schonung, B. 1991. The influence of density difference between hot and coolant gas on film cooling by a row of holes: predictions and experiments. ASME Paper 91-GT-255

Hah, C. 1984. A Navier–Stokes analysis of three-dimensional turbulent flows inside turbine blade rows at design and off-design conditions. *J. Eng. Gas Turbines Power*, **106**, 421–429

Hah, C. 1989. Numerical study of three-dimensional flow and heat transfer near the endwall of a turbine blade row. AIAA Paper 89-1689

Hah, C. and Selva, R. J. 1991. Navier–Stokes analysis of flow and heat transfer inside high-pressure-ratio transonic turbine blade rows. *J. Propulsion Power*, **7**, 990–996

Haldeman, C. W. Jr., Dunn, M. G., Lotsof, J., MacArthur, C., and Cohrs, B. 1991. Uncertainty analysis of turbine aerodynamic performance—measurements in short duration test facilities. AIAA Paper 91-2131

Haller, B. R. and Camus, J.-J. 1984. Aerodynamic loss penalty produced by film cooling transonic turbine blades. *J. Eng. Gas Turbines Power*, **106**, 198–205

Hanjalić, K. and Launder, B. E. 1972. A Reynolds stress model of turbulence and its application to thin shear flows. *J. Fluid Mech.*, **52**, 609–638

Hanjalić, K. and Launder, B. E. 1976. Contribution towards a Reynolds-stress closure for low-Reynolds-number turbulence. *J. Fluid Mech.*, **74**, 593–610

Harasgama, S. P. and Burton, C. D. 1991a. Film cooling research on the endwall of a turbine nozzle guide vane in a short duration annular cascade. Part 2: Analysis and correlation of results. ASME Paper 91-GT-253

Harasgama, S. P. and Wedlake, E. T. 1991. Heat transfer and aerodynamics of a high rim speed turbine nozzle guide vane tested in the RAE isentropic light piston cascade (ILPC). *J. Turbomach.*, **113**, 384–391

Hathaway, M. D., Chriss, R. M., Wood, J. R., and Strazizar, A. J. 1992. Experimental and computational investigation of the NASA low-speed centrifugal compressor flow field. ASME Paper 92-GT-213

Hippensteele, S. A. and Russell, L. M. 1988. High-resolution liquid-crystal heat-transfer measurements on the end wall of a turbine passage with variations in Reynolds number. *ASME 1988 National Heat Transfer Conference*, H. R. Jacobs (Ed.), Vol. 3. ASME, New York, 443–453

Hodson, H. P. 1984. Boundary layer and loss measurements on the rotor of an axial-flow turbine. *J. Eng. Gas Turbines Power*, **106**, 391–399

Holland, M. J. and Thake, T. F. 1980. Rotor blade cooling in high pressure turbines. *J. Aircraft*, **17**, 412–418

Jones, T. V., Schultz, D. L., Oldfield, M. L. G., and Daniels, L. C. 1978. Measurement of the heat transfer rate to turbine blades and nozzle guide vanes in a transient cascade. *Proceedings of the 6th International Heat Transfer Conference*, Vol. 2. Hemisphere, Washington, DC, 73–78

Katsanis, T. and McNally, W. D. 1977. Revised FORTRAN program for calculating velocities and streamlines on the hub-shroud mid-channel stream surface of an axial-, radial-, or mixed-flow turbomachine or annular duct. Vol. I: User's manual. NASA TN D-8430

Kim, J., Moin, P., and Moser, R. 1987. Turbulence statistics in fully developed channel flow at low Reynolds number. *J. Fluid Mech.*, **177**, 133–166

Kim, J., Simon, T. W., and Kestoras, M. 1989. Fluid mechanics and heat transfer measurements in transitional boundary layers conditionally sampled on intermittency. *Heat Transfer in Convective Flows: Presented at the 1989 National Heat Transfer Conference*, R. K. Shah (Ed.). ASME, New York, 69–81

Kim, J. and Simon, T. W. 1991. Free-stream turbulence and concave curvature effects on heated, transitional boundary layers. NASA CR-187150

Kim, S.-W. and Benson, T. J. 1992. Fluid flow of a row of jets in crossflow—a numerical study. AIAA Paper 92-0534

Kim, S.-W. and Benson, T. J. 1991. Calculation of a circular jet in crossflow with a multiple-time-scale turbulence model. NASA TM-104343

Kiock, R., Lehthaus, F., Baines, N. C., and Sieverding, C. H. 1986. The transonic flow through a plane turbine cascade as measured in four European wind tunnels. *J. Eng. Gas Turbines Power*, **108**, 277–284

Knight, C. J. and Choi, D. 1987. Development of a viscous cascade code based on scalar implicit factorization. AIAA Paper 87-2150

Kuan, C. and Wang, T. 1989. Some intermittent behavior of transitional boundary layers, AIAA Paper 89-1890

Kulisa, P., Leboeuf, F., and Perrin, G. 1991. Computations of a wall boundary layer with discrete jet injections. ASME Paper 91-GT-143

Kurosh, V. D. and Epik, E. Y. 1970. The effect of turbulence on heat transfer in turbomachinery flow passages. *Heat Transfer Sov. Res.*, **2**, 134–138

Lakshminarayana, B. 1986. Turbulence modeling for complex shear flows. *AIAA J.*, **24**, 1900–1917

Langston, L. S., Nice, M. S., and Hooper, R. M. 1977. Three dimensional flow within a turbine cascade passage. *J. Eng. Power*, **99**, 21–28

Launder, B. E., Reece, G. J., and Rodi, W. 1975. Progress in the development of a Reynolds-stress turbulence closure. *J. Fluid Mech.*, **68**, 537–566

Launder, B. E. and Samaraweera, D. S. A. 1979. Application of a second-moment turbulence closure to heat and mass transport in thin shear flows. I: Two-dimensional transport. *Int. J. Heat Mass Transfer*, **22**, 1631–1643

Launder, B. E., Reynolds, W. C., and Rodi, W. 1984. *Turbulence Models and Their Applications. Vol. 2. Second Moment Closure-Methodology and Practice.* Editions Eyrolles, Paris, France

Leylok, J. H. and Wisler, D. C. 1991. Mixing in axial-flow compressors: conclusions drawn from three-dimensional Navier-Stokes analyses and experiments. *J. Turbomach.*, **113**, 139–160

Mancuso, T. and Diller, T. E. 1991. Time-resolved heat flux measurements in unsteady flow, fundamental experimental measurements. *Heat Transfer: Presented at the Winter Annual Meeting of ASME*, D. E. Beasley and J. L. S. Chen (Eds.). ASME, New York, 67–75

Marek, C. J. and Tacina, R. R. 1975. Effect of free stream turbulence on film cooling. NASA TN D-7958

Mayle, R. E. and Dullenkopf, K. 1990. A theory for wake-induced transition. *J. Turbomach.*, **112**, 188–195

Mayle, R. E. and Dullenkopf, K. 1991. More on the turbulent-strip theory for wake-induced transition. *J. Turbomach.*, **113**, 428–432

Morkovin, M. V. 1978. Instability transition to turbulence and predictability. AGARD Report AG-236

Narasimha, R. 1985. The laminar-turbulent transition zone in the boundary layer. *Prog. Aerospace Sci.*, **22**, 29–80

Nealy, D. A., Mihelc, M. S., Hylton, L. D., and Gladden, H. J. 1984. Measurements of heat transfer distribution over the surfaces of highly loaded turbine nozzle guide vanes. *J. Eng. Gas Turbines Power*, **106**, 149–158

Nicholson, J. H., Forest, A. E., Oldfield, M. L. G., and Schultz, D. 1984. Heat transfer optimized turbine rotor blades—an experimental study using transient techniques. *J. Eng. Gas Turbines Power*, **106**, 173–181

Nirmalan, N. V. and Hylton, L. D. 1990. An experimental study of turbine vane heat transfer with leading edge and downstream film cooling. *J. Turbomach.*, **112**, 477–487

O'Brien, J. E., Simoneau, R. J., LaGraff, J. E., and Morehouse, K. A. 1986. Unsteady heat transfer and direct comparison to steady-state measurements in a rotor-wake experiment. *Heat Transfer 1986: Proceedings of the Eighth International Heat Transfer Conference*, C. L. Tien, V. P. Carey, and J. K. Ferrell (Eds.), Vol. 3. Hemisphere, Washington, DC, 1249–1256

O'Brien, J. E. and Capp, S. P. 1989. Two-component phase-averaged turbulence statistics downstream of a rotating spoked-wheel wake generator. *J. Turbomach.*, **111**, 475–482

O'Brien, J. E. 1990. Effects of wake passing on stagnation region heat transfer. *J. Turbomach.*, **112**, 522–530

Papell, S. S. 1984. Vortex generating flow passage design for increased film-cooling effectiveness and surface coverage. NASA TM-83617

Patankar, S. V. and Spalding, D. B. 1972. A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. *Int. J. Heat Mass Transfer*, **15**, 1787–1806

Paxson, D. E. and Mayle, R. E. 1991. Laminar boundary layer interaction with an unsteady passing wake. *J. Turbomach.*, **113**, 419–427

Pfeil, H., Herbst, R., and Schroder, T. 1982. Investigation of the laminar-turbulent transition of boundary layers disturbed by wakes. ASME Paper 82-GT-124

Pietrzyk, J. R., Bogard, D. G., and Crawford, M. E. 1989. Hydrodynamic measurements of jets in crossflow for gas turbine film cooling applications. *J. Turbomach.*, **111**, 139–145

Pietrzyk, J. R., Bogard, D. G., and Crawford, M. E. 1990. Effects of density ratio on the hydrodynamics of film cooling. *J. Turbomach.*, **112**, 437–443

Priddy, W. J. and Bayley, F. J. 1988. Turbulence measurements in turbine blade passages and implications for heat transfer. *J. Turbomach.*, **110**, 73–79

Rai, M. M. 1989a. Three-dimensional Navier–Stokes simulations of turbine rotor-stator interaction; Part I: Methodology. *J. Propulsion Power*, **5**, 305–311

Rai, M. M. 1989b. Three-dimensional Navier–Stokes simulations of turbine rotor-stator interaction; Part II: Results. *J. Propulsion Power*, **5**, 312–319

Rai, M. M. and Dring, R. P. 1990. Navier–Stokes analysis of the redistribution of inlet temperature distortions in a turbine. *J. Propulsion Power*, **6**, 276–282

Rai, M. M. and Madavan, N. K. 1990. Multi-airfoil Navier–Stokes simulations of turbine rotor-stator interaction. *J. Turbomach.*, **112**, 377–384

Rai, M. M. and Moin, P. 1991. Direct numerical simulation of transition and turbulence in a spatially evolving boundary layer. AIAA Paper 91-1707

Rao, K. and Delaney, R. 1990. Investigation of unsteady flow through transonic turbine stage. Part I: Analysis. AIAA Paper 90-2408

Rao, K. V., Delaney, R. A., and Dunn, M. G. 1992a. Vane-blade interaction in a transonic turbine. Part I—Aerodynamics. AIAA Paper 92-3323

Rao, K. V., Delaney, R. A., and Dunn, M. G. 1992b. Vane-blade interaction in a transonic turbine. Part II—Heat transfer. AIAA Paper 92-3324

Rodi, W. and Scheuerer, G. 1985a. Calculation of heat transfer to convection-cooled gas turbine blades. *J. Eng. Gas Turbines Power*, **107**, 620–627

Rodi, W. and Scheurer, G. 1985b. Calculation of laminar-turbulent boundary layer transition on turbine blades. *Heat Transfer and Cooling in Gas Turbines.* AGARD-CP-390

Rotta, J. C. 1979. A family of turbulence models for three-dimensional boundary layers. *Turbulent Shear Flows* Vol. 1, F. Durst (Ed.). Springer, New York

Sathyamurthy, P. S. and Patankar, S. V. 1992. Film-cooling studies using a three-dimensional parabolic procedure. To be published

Sathyamurthy, P. S. and Patankar, S. V. 1990. Prediction of film cooling with lateral injection. *Heat Transfer in Turbulent Flow: Presented at AIAA/ASME—Thermophysics and Heat Transfer Conference*, R. S. Amano, M. E. Crawford, and N. K. Anand (Eds.). ASME, New York, 66–70

Savill, A. M. 1992. A low-Reynolds number second moment closure for predicting by-pass transition. *5th Biennial Colloquim on Computational Fluid Dynamics*, (Paper no. 3.5), University of Manchester Institute of Science and Technology Department of Mechanical Engineering, Manchester, UK

Schmidt, R. C. and Patankar, S. V. 1988. Two-equation low-Reynolds-number turbulence modeling of transitional boundary layer flow characteristic of gas turbine blades. NASA CR-4145

Schoberei, T. and Pardivala, D. 1991. Completion of the preliminary measurements on the research facility for investigating the effects of periodic unsteady inlet flow, pressure gradient and curvature on boundary layer transition, wake development and heat transfer, Internal Report PR-2, Texas A&M University, College Station, TX

Schonung, B. and Rodi, W. 1987. Prediction of film cooling by a row of holes with a two-dimensional boundary-layer procedure. *J. Turbomach.*, **109**, 579–590

Schultz, D. L., Oldfield, M. L. G., and Jones, T. V. 1980. Heat transfer rate and film cooling effectiveness measurements in a transient cascade, testing and measurement techniques in heat transfer and combustion, AGARD CP-281

Sharma, O. P., Wells, R. A., Schlinker, R. H., and Bailey, D. A. 1982. Boundary layer development on turbine airfoil suction surfaces. *ASME Trans.*, **104**, 698–706

Sharma, O. P., Butler, T. L., Joslyn, H. D., and Dring, R. P. 1983. An experimental investigation of the three-dimensional unsteady flow in an axial flow turbine. AIAA Paper 83-1170

Shih, T. H. and Mansour, N. N. 1990. Modeling of near-wall turbulence. NASA TM-103222

Sieverding, C. H. 1985. Recent progress in the understanding of basic aspects of secondary flows in turbine blade passages. *J. Eng. Gas Turbines Power*, **107**, 248–257

Simon, F. F. and Ciancone, M. L. 1985. Flow visualization study of the effect of injection hole geometry on an inclined jet in crossflow. NASA TM-86936

Simon, F. F. 1986. Jet model for slot film cooling with effect of free-stream and coolant turbulence. NASA TP-2655

Simon, F. F. and Stephens, C. A. 1991. Modeling of the heat transfer in bypass transitional boundary-layer flows. NASA TP-3170

Sohn, K. H. and Reshotko, E. 1991. Experimental study of boundary layer transition with elevated freestream turbulence on a heated flat plate. NASA CR-187068

Sohn, K. H., Reshotko, E., and Zaman, K. B. M. Q. 1991. Experimental study of boundary layer transition on a heated flat plate. NASA TM-103779

Stephens, C. A. and Crawford, M. E. 1990. An investigation into the numerical prediction of boundary layer transition using the K. Y. Chien turbulence model. NASA CR-185252

Suder, K. L., O'Brien, J. E., and Reshotko, E. 1988. Experimental study of bypass transition in a boundary layer. NASA TM-100913

Tafti, D. K. and Yavuzkurt, S. 1988. Prediction of the heat transfer characteristics for discrete hole film cooling—one row injection into a turbulent boundary layer. ASME HTD, 103, ASME Winter Annual Meeting, Chicago, IL, 45–52

Tafti, D. K. and Yavuzkurt, S. 1990. Prediction of heat transfer characteristics for discrete hole film cooling for turbine blade applications. *J. Turbomach.*, **112**, 504–511

Takeishi, K., Matsuura, M., Aoki, S., and Sato, T. 1990. An experimental study of heat transfer and film cooling on low aspect ratio turbine nozzles. *J. Turbomach.*, **112**, 488–496

Tran, L. T. and Taulbee, D. B. 1991. Prediction of unsteady rotor-surface pressure and heat transfer from wake passings. ASME Paper 91-GT-267

Turner, A. B. 1971. Local heat transfer measurements on a gas turbine blade. *J. Mech. Eng. Sci.*, **13**, 1–12

Vancoille, G. and Dick, E. 1988. A turbulence model for the numerical simulation of the transition zone in a boundary layer. *Int. J. Eng. Fluid Mech.*, **1**, 28–49

Verhoff, V. G., Camperchioli, W. P., and Lopez, I. 1992. Transonic turbine blade cascade testing blade facility. AIAA Paper 92-4034

Vogel, T. 1991. Computation of 3D viscous flow and heat transfer for the application to film cooled gas turbine blades. AGARD Computational Techniques for Turbomachinery, CP 510, *AGARD 77th Symp. CFD Techniques for Propulsion Appl.*, 7-1—7-8

Volino, R. J. and Simon, T. W. 1991. Bypass transition in boundary layers including curvature and favorable pressure gradient effects. NASA CR-187187

Walker, J. D. A., Scharnhorst, R. K., and Weigand, G. G. 1986. Wall layer models for the calculation of velocity and heat transfer in turbulent boundary layers. AIAA Paper 86-0213

Wedlake, E. T., Brooks, A. J., and Harasgama, S. P. 1989. Aerodynamic and heat transfer measurements on a transonic nozzle guide vane. *J. Turbomach.*, **111**, 36–42

Weinberg, B. C., Yang, R.-J., McDonald, H., and Shamroth, S. J. 1986. Calculations of two and three-dimensional transonic cascade flow fields using the Navier–Stokes equations. *J. Eng. Gas Turbines Power*, **108**, 93–102

Wilson, D. G. and Pope, J. A. 1954. Convective heat transfer to gas turbine blade surfaces. *Proc. Inst. Mech. Eng.*, **168**, 861–874

Wittig, S., Dullenkopf, K., Schulz, A., and Hestermann, R. 1987. Laser-doppler studies of the wake-effected flow field in a turbine cascade. *J. Turbomach.*, **109**, 170–176

York, R. E., Hylton, L. D., and Mihelc, M. S. 1984. An experimental investigation of endwall heat transfer and aerodynamics in a linear vane cascade. *J. Eng. Gas Turbines Power*, **106**, 159–167

Zerkle, R. D. and Lounsbury, R. J. 1989. The influence of freestream gradient on heat transfer to gas turbine airfoils. *J. Propulsion Power*, **5**, 82–88