Evaluation of Alternatives for Two-Dimensional Linear Cascade Facilities

This paper presents two low-cost alternatives for turbine blade surface heat transfer and fluid dynamics measurements. These models embody careful compromises between typical academic and full-scale turbomachinery experiments and represent a comprehensive strategy to develop experiments that can directly test shortcomings in current turbomachinery simulation tools. A full contextual history of the wide range of approaches to simulate turbine flow conditions is presented, along with a discussion of their deficiencies. Both models are simplifications of a linear cascade: the current standard for simulating two-dimensional turbine blade geometries. A single passage model is presented as a curved duct consisting of two half-blade geometries, carefully designed inlet and exit walls and inlet suction. This facility was determined to be best suited for heat transfer measurements where minimal surface conduction losses are necessary to allow accurate numerical model replication. A double passage model is defined as a single blade with two precisely designed outer walls, which is most appropriate for flow measurements. The design procedures necessary to achieve a desired flow condition are discussed.

[DOI: 10.1115/1.2985073]

1 Introduction

Large-scale efforts to compare simulations to experiments, as performed by Garg [1] and Haldeman and Dunn [2], demonstrate that many of the characteristic state-of-the-art modeling applications routinely fail when applied to modern turbine blade geometries. However, Dunn [3] pointed out that it is unclear if such failures are due to specific modeling issues, or deficiencies in the experimental measurements. There are several approaches presented in the open literature to experimentally simulate the flow field around a given gas turbine engine rotor or stator blade geometry. Turbomachinery testing facilities can, in general, be divided into two main subsets: nonrotating cascades (linear and annular) and rotating facilities. The first simplification of an actual turbine is a steady state annular rotating cascade, as demonstrated by Atassi et al. [4]. This approach is primarily used for compressor geometries as implemented by Schulz and Gallus [5] and Wisler et al. [6]. Blair [7] obtained highly resolved maps of the heat transfer coefficient without film cooling using an incompressible, steady, ambient temperature, large-scale turbine rotor passage.

Transient rotating annular cascades driven by either a large tank in blowdown mode (as illustrated by Abhari and Epstein [8]), a shock-tube (as pioneered by Dunn and Stoddard [9], Dunn [10], Dunn and Chupp [11], and Dunn et al. [12]), or an isentropic light piston (as demonstrated by Chana and Jones [13]) offer cost advantages when compared to steady state facilities as these have reduced flow requirements due to their ephemeral run times. Such tests often have a duration of a few hundred milliseconds requiring the use of complex transient measurement techniques to extract heat transfer and pressure data on test engine components. This approach only allows for low spatial resolution measurements of surface heat flux and pressure [3]. The heat transfer sensors in these studies, thin-film heat flux gauges, provide a high frequency response that allows time-accurate measurements of unsteady convective heat transfer rates. These sensors use a quasi-1D conduction model to calculate the surface heat flux. Dunn [14] and Epstein et al. [15] presented typical analyses used to extract time-resolved heat flux data from these gauges. This approach presents several problems that can make the measurements difficult to interpret and model. Mukerji et al. [16] demonstrated that such sensors can corrupt the heat transfer measurement by as much as 30%, by changing the thermal boundary condition on the blade, if the sensor substrate has a significantly lower thermal diffusivity than the surrounding material. The substrate causes a local temperature rise over the gauge, producing a wall temperature step. This has been termed as the “heat island effect” by Dunn et al. [17]. Corrections for this problem have been proposed by Moffatt et al. [18]. Furthermore, Diller [19] argued that the flow conditions in blade passages are highly sensitive to local perturbations: i.e., a poorly installed gauge can cause physical disruptions of the boundary layer, affecting the measurements by as much as 75% [20]. Such concerns effectively limit their usefulness to modeling efforts. Furthermore, the harsh conditions in these experiments cause these sensors to have a relatively high mortality rate. This behavior can be traced to the high inlet temperatures of these experiments that are on the order of 500 K and mechanical stresses due to rotation rates as fast as 10,000 rpm. Another issue with these experiments is the necessary lead time to develop the experimental apparatus. Typically, there is at least a 3–4 year evolution from “drawing board” to data collection [21]. These facts do not obviate the usefulness of rotating facilities, as they can match all relevant nondimensional parameters. However, it suggests that to obtain higher measurement fidelity and use measurement techniques that can be precisely replicated numerically (constant heat flux or isothermal surfaces) it is practical to simplify the flow field, especially if the blade midspan behavior is of primary interest. However, it is important to note that the primary trade-off of such an approach is the inability to match all relevant nondimensional parameters and lack of rotation. Never-
theless, as the objective of such experiments is to improve numerical modeling efforts, such compromises are often acceptable. A further simplification of the flow field is a nonrotating annular cascade, which consists either of a full annulus or a 60 deg sector, based on flow requirements. Martinez-Botas et al. [22] presented uncooled heat transfer results in an annular cascade driven by an isentropic light piston. Thermochromic liquid crystal paint was used to obtain spatially resolved measurements in this experiment. However, the time and expense required to build such a facility gives them no real advantage over linear cascades.

Linear cascades are considered an acceptable compromise to provide well-resolved data and well-defined conditions for both design and modeling improvement purposes. Baughn [23] and Guenette et al. [24] suggested that the flow around the center airfoil of a two-dimensional linear cascade presents nearly identical flow characteristics as those found along the midspan position of a blade in a rotating annular cascade. These facilities provide tremendous flexibility in investigating a variety of conditions, including reducing endwall losses by axisymmetric endwall contouring [25]; endwall and blade heat transfer [26,27]; incidence effects on film cooling performance [28]; and film cooling-generated aerodynamics losses [29]. They are amenable to optical fluid mechanics and heat transfer measurement techniques such as laser-Doppler anemometry (LDA) [30], infrared thermography [31], and thermochromic liquid crystals [32]. Additionally, linear cascades can be run in steady state or transient modes, the latter being more cost effective due to reduced flow requirements. A popular fashion of performing transient linear cascade tests is to use an isentropic light piston with ceramic test blades, as illustrated by Camci and Arts [28] and Sieverding et al. [33]. It is important to point out that depending on the choice of measurement technique the ability of a given facility to achieve various nondimensional numbers is often constrained. For example, thermochromic liquid crystals, which change temperature over a narrow temperature range (at most 10–15°C), limit the possible temperature range for a given facility. However, this measurement technique allows high-resolution temperature maps that can be directly compared against numerical simulations. The number of blades has been found to be important in ensuring that the desired flow conditions are achieved. To obtain a periodic flow field around the center measurement blade in a typical linear cascade, several “dummy” blades are required. As the typical mass flow rate through a passage in between two blades is approximately 1 kg/s, the requirement for so many passages places a substantial flow requirement for the facility. In the case of Giel et al. [34], the required flow rate was 26 kg/s at steady state conditions.

To further reduce the cost of performing heat transfer measurements on real turbine blade geometries, the restriction on the number of blades can be relaxed. However, this makes achieving a periodic flow condition more difficult. Abuaf et al. [35] proved this point with a transonic four-passage cascade that was used to collect heat transfer measurements. An examination of the mass flow rates through each passage revealed that the flow in this cascade was not periodic. Goldstein and Spores [36], Radomsky and Thole [37], and Pridy and Bayley [38] presented a further simplification, termed a double passage cascade. These models comprise of a single blade bounded by two shaped outer walls. Yet another simplification is a single passage model. This model consists of a single passage bounded by two walls, which are shaped by the blade geometry under examination. Blair [39] first utilized a single passage model to perform endwall heat transfer and film cooling measurements. Bailey [40], Chung and Simon [41], and Chung et al. [42] extended this approach to study airfoil aerodynamics. Buck and Prakash [43] combined a single passage model with a mass transfer analogy technique to perform film cooling measurements. It is important to note that both double and single passage models have been actively used with entirely subsonic flows. Thus, it was unclear how to extend these techniques to more modern blade geometries with significantly increased turning angle and supersonic flow conditions.

This paper discusses recent efforts to design double and single passage models to achieve periodic flow conditions for a transonic, highly cambered blade geometry. Such experimental facilities have the advantage of providing highly resolved fluid mechanics and heat transfer measurements at steady state conditions without massive flow supply requirements. Consequently, these models are considerably cheaper and provide the same data as a full linear cascade. Our objective is to use these facilities with well-defined flow and thermal boundary conditions to improve the numerical modeling of such flows. It is important to point out, however, that these models are intended for use for a specific inlet flow angle and pressure ratio. Consequently, it is unclear how useful these models are for significantly off-design conditions.

2 Overview of Passage Design Concepts

There are two well-accepted computational domains for nonrotating two-dimensional turbine blade geometries. Both these approaches simulate an infinite row of blades, as shown in Fig. 1(a). Incoming and departing streamlines have been included in this figure for discussion purposes. One approach is a single blade with periodic boundary conditions at midpitch, as shown in Fig. 1(b). The other uses two blade surfaces, the pressure surface of the upper blade and the suction surface of the lower blade with periodic boundaries leading up to and departing from these surfaces, as shown in Fig. 1(c). A variation of this approach uses one full blade and two blade surfaces, as demonstrated in Fig. 2.

The single and double passage experimental techniques mimic these approaches although the inlet and exit periodic boundary condition surfaces in Fig. 1(c) and Fig. 2 are replaced with walls. The ultimate design objective is to establish a flow field that
matches a two-dimensional infinite cascade flow condition. Kodzwa and Eaton [44] demonstrated that the single passage model is ideal for steady state heat transfer measurements with two well-insulated blade surfaces, access for optical surface heat transfer measurements, and reduced flow requirements. Conversely, the double passage model of Laskowski [45] is ideal for optical flow measurements as demonstrated by Vicharelli and Eaton [46]. The double passage approach was not deemed optimal for heat transfer measurements as the thermal losses within the instrumented center blade were determined to be unacceptable. Both approaches are useful for one operating condition and new walls are necessary for different run conditions. The aerodynamic design procedure incorporated 2D and 3D full-geometry simulations. The essence of the procedure is indicated as follows.

1. Perform infinite cascade simulation on two-dimensional blade geometry at the conditions of interest.
2. Obtain calculated streamlines from simulation to develop initial guesses for the shape of passage walls and suction rates, if appropriate. Additionally, obtain the surface pressure distribution, skin friction, and various flow field parameters around airfoil for comparative purposes.
3. Perform a numerical optimization on these shapes and suction rates to achieve an infinite cascade flow field.
4. Verify the design with experimental data.
5. Iterate between steps 3 and 4, if necessary.

Note that our approach assumes that computational fluid dynamics (CFD) can accurately predict the target pressure distribution for a given airfoil. Previous studies have indeed indicated that such an assumption is reasonable for transonic and subsonic airfoils with fully attached boundary layers [1,47]. Nevertheless, it is important to state that this assumption requires additional validation with varying airfoil geometries to establish its limitations. An immediate example would be airfoils with potential boundary layer separation. The Reynolds-averaged Navier–Stokes (RANS) equations for a compressible turbulent flow were solved using a commercial CFD package, STAR-CD [48]. In all the simulations for the aerodynamics design, a two-layer, two-equation $k$--$\varepsilon$ turbulence model was implemented. Table 1 summarizes the expected flow conditions for the given blade geometry at typical test conditions (from Buck [49]). The models are designed around an advanced first stage rotor blade geometry. This airfoil is highly cambered and operates at transonic conditions, with meanflow Mach numbers as high as 1.5. This geometry was used by Haldeman et al. [50] in full rotating aerodynamic and heat transfer tests. The model scale (1.3) was selected based on instrumentation concerns and flow supply limitations. The experiment was designed to run at ambient conditions with the model back pressure assumed to be at atmospheric pressure.

Figures 3 and 4 show a schematic of experimental single and double passage models with their salient features identified. The presented single passage model was designed to be placed on top of a plenum with flow passing upwards either from a plenum through a bellmouth (in the case of the single passage) or from a carefully designed nozzle (in the case of the double passage). The inlet duct length was chosen to be long enough such that there would be adequate clearance between the exhaust flow and the inlet and also to provide probe access to an inlet measurement station. The downstream duct length was chosen to be long enough such that the exhaust manifold shape would have minimal effect on the flow in the test region.

3 Infinite Cascade

Infinite cascade simulations were conducted to develop a baseline and provide target data to complete the passage model designs. Details on these viscous and inviscid simulations and methodology can be found in Refs. [44,45]. The following summarizes the results of these analyses. Figure 5 displays the calculated pressure distribution using these approaches, presented as the isentropic Mach number, $M_s$, versus the surface coordinate, $s/c_{blade}$. This parameter is a reformulation of the pressure distribution, using the inlet stagnation pressure to compute a Mach number as follows:

$$M_s = \sqrt{\frac{2}{\gamma - 1} \left[ \left( \frac{P_{0,inlet}}{P} \right)^{\gamma(\gamma-1)/\gamma} - 1 \right]} \quad (1)$$

The negative surface distance positions shown in this figure correspond to locations on the pressure side of the airfoil, while the positive surface positions correspond to the suction surface. Figure 6 provides a visual description of these surface coordinate positions, along with a definition of the axial location along the blade. Table 2 presents the computed stagnation point locations using inviscid calculations and various turbulence models. Of interest is the sensitivity of the standard $k$--$\varepsilon$ turbulence model to changes in the inlet turbulence condition. This sensitivity was removed when the modeling equations were modified to limit the production of turbulent kinetic energy, as proposed by Chen and

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Test condition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chord length, $c_{blade}$ (mm)</td>
<td>36.1</td>
</tr>
<tr>
<td>Leading edge diameter (mm)</td>
<td>$=6$</td>
</tr>
<tr>
<td>Airfoil pitch spacing (mm)</td>
<td>39.8</td>
</tr>
<tr>
<td>$\gamma_{blade} = c_{p}/c_{v}$</td>
<td>1.4</td>
</tr>
<tr>
<td>Inlet angle</td>
<td>29.2 deg</td>
</tr>
<tr>
<td>Exit angle</td>
<td>$-68.6$ deg</td>
</tr>
<tr>
<td>$P_{0,inlet}/P_{exit}$</td>
<td>2.57</td>
</tr>
<tr>
<td>$P_{0,inlet}$ (Pa)</td>
<td>$2.60 \times 10^5$</td>
</tr>
<tr>
<td>$T_{0,inlet}$ (K)</td>
<td>300</td>
</tr>
<tr>
<td>Approximate inlet Mach number</td>
<td>$=0.34$</td>
</tr>
<tr>
<td>Reynolds number, $Re = \rho u_{inlet} c_{blade} / \mu$</td>
<td>$6.62 \times 10^5$</td>
</tr>
</tbody>
</table>
Kim [51]. Figure 7(a) shows Mach number contours for the two-dimensional RANS calculation using this model. To design the shape of the inlet and outlet walls and the amount of suction, streamlines were extracted from this simulation. Figure 7(b) presents the streamlines in the initial orientation of the blade geometry, and the streamlines after the domain was rotated for implementation in the presented experimental facilities.

4 Single Passage Model

A heuristic-based numerical simulation approach was developed to design the inlet and exit wall shapes. Full details can be found in Ref. [44]. The design procedure involved the following steps.

1. Correctly position the stagnation point on the suction and pressure side surfaces by adjusting the shape of the inlet walls using streamlines from the RANS infinite cascade calculation to achieve satisfactory agreement with respect to the location of the stagnation point on the two measurement surfaces.

2. To decouple this from any effect from the shape of the exit passage, enforce periodic boundary conditions, instead of wall boundary conditions along the trailing edge surfaces of the domain.

3. Correctly position and set up the shock structure on the aft side of the airfoil geometry by shaping the exit duct walls again using streamlines from the RANS simulation produced from step 1.

4.1 Flow Model. The computational grid for the single passage was generated in a multiblock fashion utilizing an in-house structured iterative elliptic grid generator developed by Wu [52]. This approach ensures grid line orthogonality on all the boundaries of the domain. Three $H$-grid blocks were used in constructing the domain; two blocks were used for each bleed section and one for the main passage. The cell heights near the walls were stretched to achieve $y^+$ values ranging from 0.14 to 3.7. We performed a grid refinement study to determine the smallest possible grid size that could be run while maintaining satisfactory accuracy in $M_{\infty}$; this resulted in a mesh size of approximately 60,000 cells for the majority of results presented here. The largest two-dimensional grid during the design process contained approximately 150,000 cells. This study revealed that the oblique shock structure observed along the suction surface primary flow feature was highly sensitive to grid resolution. A stagnation boundary condition was specified at the inlet and constant pressure boundary conditions were implemented on the bleed exit boundaries.

4.2 Inlet Wall Design. To design the inlet walls, the exit surfaces were replaced with periodic boundary conditions that extended one-chord length ($c_{blade}$) in the axial direction downstream of the trailing edges of the airfoils. This approach equated the values of all flow variables along the boundaries. The ideal inlet wall shapes would generate a streamline one-displacement-thickness ($\delta_1$) away from the surface that would correspond to a streamline in the infinite cascade flow field. For this to occur, the boundary layer must remain attached for the bleed to function correctly. Hence, the chosen wall shape must have streamwise pressure gradients that ensure attached thin boundary layers. This goal was achieved by taking successive streamlines from the infinite cascade simulation and performing a two-dimensional RANS calculation to determine the predicted $M_{\infty}$ distribution. The streamlines were chosen consistent with the desired direction of movement for the predicted stagnation point. The bellmouth was redesigned using ovals with dimensions consistent with the chosen inlet streamlines.
Table 3 compares the agreement between the stagnation point locations on both surfaces using the previously described approach, an alternative approach developed by Buck and Prakash [43] and the infinite cascade. Figure 8 presents predicted Mach distributions for designs using the Buck and Prakash technique and the design practice discussed here. An examination of these distributions verified that the bleed conditions have a negligible ef-

---

**Table 2** Comparison of computed stagnation point locations using various turbulence models and conditions for infinite two-dimensional cascade

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>(M_{\infty}=0(x/c_{blade}))</th>
</tr>
</thead>
<tbody>
<tr>
<td>(k-e) standard (TI%=10%, (\ell/c_{blade}=0.028))</td>
<td>2.28 \times 10^{-3}</td>
</tr>
<tr>
<td>(k-e) standard (TI%=10%, (\ell/c_{blade}=0.028))</td>
<td>1.37 \times 10^{-3}</td>
</tr>
<tr>
<td>(k-e) Chen (TI%=10%, (\ell/c_{blade}=0.028))</td>
<td>2.76 \times 10^{-3}</td>
</tr>
<tr>
<td>(k-e) Chen (TI%=10%, (\ell/c_{blade}=0.028))</td>
<td>2.28 \times 10^{-3}</td>
</tr>
<tr>
<td>Inviscid calculation (NOVAK)</td>
<td>2.76 \times 10^{-3}</td>
</tr>
</tbody>
</table>

---

**Table 3** Comparison of computed stagnation point axial locations for new design versus infinite cascade and Buck and Prakash [43] design

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>(M_{\infty}=0(x/c_{blade}))</th>
</tr>
</thead>
<tbody>
<tr>
<td>(k-e) Chen (TI%=10%, (\ell/c_{blade}=0.28))</td>
<td>2.28 \times 10^{-3}</td>
</tr>
<tr>
<td>Suction side blade, (k-e) Chen (TI%=10%, (\ell/c_{blade}=0.28))</td>
<td>2.64 \times 10^{-3}</td>
</tr>
<tr>
<td>Pressure side blade, (k-e) Chen (TI%=10%, (\ell/c_{blade}=0.28))</td>
<td>2.64 \times 10^{-3}</td>
</tr>
<tr>
<td>Pressure side blade [43], (k-e) Chen (TI%=10%, (\ell/c_{blade}=0.28))</td>
<td>1.24 \times 10^{-3}</td>
</tr>
<tr>
<td>Pressure side blade [43], (k-e) Chen (TI%=10%, (\ell/c_{blade}=0.28))</td>
<td>3.71 \times 10^{-3}</td>
</tr>
</tbody>
</table>
fect on the downstream shock structure. The computed skin friction coefficients along the bleed walls indicated that these walls would produce no separation.

4.3 Exit Wall Design (Tailboards). The results presented by Kodzwa and Eaton [44] demonstrated the necessity of curved exit walls in achieving the desired shock structure. By extension from the design process for the inlet walls, the desired exit wall designs should be those that produce streamlines a $\delta_1$ away from the wall that closely follow those of an infinite cascade flow field. Furthermore, considering that the displacement thicknesses of the boundary layers on the exit walls are strongly coupled to the shock structure, it is virtually impossible to determine a priori the optimal wall shape. Thus some form of iterative scheme must be used to design the exit walls.

Figure 9 evinces the similarities in the shock structure between the infinite cascade flow field and one with periodic tailboards and properly designed bleeds. The maximum error between the flow fields was estimated to be approximately 8%. The error was defined using the following equation:

$$\epsilon_{\text{M}_\text{n}} = \left| \frac{\text{M}_\text{IC}}{\text{M}_\text{2DRANS}} - 1 \right|$$

where $\text{M}_\text{IC}$ is the computed Mach number from the infinite cascade simulation and $\text{M}_\text{2DRANS}$ is the computed Mach number from the single passage simulation.

Figure 10 displays computed trailing edge streamlines from the single passage computation with periodic exit boundaries. The design procedure for the exit wall is based on the postulate that the ideal wall shape consists of the closest streamline to the trailing edge, rotated to account for the growing boundary layer along the wall. The pivot point for this rotation was assumed to be at the trailing edge of each blade, as shown in Fig. 11. Based on previous results, it was conjectured that it was only necessary to adjust the pressure surface exit wall. Hence, this wall was rotated counterclockwise by a defined angle, $\phi_{ps}$. A rotation angle for the suction surface exit wall was also defined ($\phi_{ss}$). However, this was found to be unnecessary by exploratory simulations.

Figure 12 presents the evolution of the single passage $\text{M}_\text{n}$ dis-
tribution with increasing angle of rotation for the pressure side wall. When $\phi_{ps}=0$ deg, the interference of the boundary layer with the mainstream flow causes a strong initial shock to form, evidenced by the dramatic drop in the suction side $M_{is}$ distribution. As $\phi_{ps}$ increases, the $M_{is}$ distribution along the suction side blade wall approaches that of the infinite cascade simulation. The difference between these two results is minimized at a particular angle; in these simulations it was at $\phi_{ps}=0.3$ deg. Beyond this value, the oblique shocks continue to weaken, resulting in even faster flow over the suction side wall. These results also indicate that beyond an angle of approximately $\phi_{ps}=1.90$ deg, there is a limited change in the surface pressure distribution. In all cases, there was no noticeable effect on the location of the stagnation points due to the changing of the exit wall geometry.

The initial build for the single passage used a tailboard angle of $\phi_{ps}=0.3$ deg. This angle was predicted to give the best $M_{is}$ agreement. The computed flow field “error” for this geometry ($\epsilon_{MC} = |M_{IC}/M_{DRANS} − 1|$) demonstrated that high quality agreement with the infinite cascade $M_{is}$ distribution ensures a flow field that closely matches that found in an infinite cascade. This result is important as it suggests that precisely matching the desired $M_{is}$ distribution guarantees a match to the flow field conditions. Kodzwa and Eaton [44] experimentally determined, however, that such a model produced a strong normal shock in the passage. After successive experimental iterations, the implemented tailboard angle was $\phi_{ps}=1.90$ deg; this produced a pressure distribution that closely followed the infinite cascade simulation, which differed significantly from the prediction. This angle choice was heuristically described as a “safety margin” to account for the three-dimensional effects that resulted from thicker tailboard boundary layers than those predicted by the two-dimensional RANS design process or uncertainty in the manufacturing process. As the flow field is very sensitive to this angle, instead of using adjustable tailboards, we chose to machine this part out of a solid piece of Ren Shape.

### 4.4 Experimental Validation

To evaluate the flow conditions in the blade passage, two airfoil surfaces with closely spaced pressure taps were installed in a model constructed out of a low-thermal conductivity material (Ren Shape 450, a high-density polyurethane material) to minimize thermal losses as much as possible. The taps consisted of cross-drilled holes; the tap on the surface was a 0.62 mm diameter hole, drilled perpendicular to the local surface tangent and the cross-drilled port was a 1.6 mm diameter hole. A 1.59 mm diameter copper tube with an internal diameter of 0.88 mm was glued into the port. 1.59 mm diameter Tygon tubes were push-fit over the ends of the copper tubes and connected to a Scanivalve (Scanivalve Corporation #SSS-48C Mk III).

Figure 13 presents the measured $M_{is}$ distribution without a turbulence grid installed. For comparison, the desired pressure distribution for the given airfoil geometry and the computed $M_{is}$ distribution for the periodic single passage is included. The uncertainty of $M_{is}$ was computed to be $\pm 0.046$ ($P=0.95$). These results show close agreement with the desired pressure distribution for the given airfoil geometry. The sole location where there is a significant difference between prediction and experiment is at $s/c_{blade} = 1.0$. Grid refinement studies suggested that this difference was due to (i) inadequate grid resolution around the shock and (ii) our chosen numerical algorithm. However, it is important to note that this agreement is equivalent to that illustrated by Medic and Durbin for the transonic rotor airfoil used by Camci and Arts [28] ($\pm 10\%$). Thus we considered the agreement here to be acceptable.

Table 4 compares the mass flow rates for the two design results to their measured counterparts from the as-built model, as reported by Kodzwa and Eaton [44]. The estimated uncertainties for the mainstream and bleed mass flow rates were 1% and 6%, respectively. The substantial difference between the measured and predicted mass flow rates could be due to three-dimensional effects or deficiencies in our numerical models; however, further study would be necessary to confirm the exact cause.

Table 4 | Comparison of computed and experimentally measured (Kodzwa and Eaton [44]) bleed mass flow rates

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Design</th>
<th>Measured</th>
</tr>
</thead>
<tbody>
<tr>
<td>Suction side bleed ($m_{sb}$, kg/s)</td>
<td>$8.93 \times 10^{-2}$</td>
<td>$8.2 \times 10^{-2}$</td>
</tr>
<tr>
<td>Pressure side bleed ($m_{pb}$, kg/s)</td>
<td>$9.29 \times 10^{-2}$</td>
<td>$7.2 \times 10^{-2}$</td>
</tr>
<tr>
<td>Mainstream mass flow rate ($m$, kg/s)</td>
<td>0.616</td>
<td>0.670</td>
</tr>
</tbody>
</table>
heat transfer measurement data collected using this model. It is important to state that these experiments were conducted at near ambient conditions; consequently these data are not directly applicable to engine design. However, we believe that they are best suited for comparison to numerical models due to their well-defined thermal and flow boundary conditions.

5 Double Passage Model

In the double passage model, as no bleeds are present, it is necessary to design the outer walls to account for boundary growth from inlet to exit. The primary benefit of the double passage approach is that the difficulties associated with bleeds and exit walls are circumvented. The general procedure is as follows.

1. Select the outer wall streamlines from the infinite cascade simulation that generate minimal separation and a computed pressure distribution that is close to the infinite cascade solution.
2. Using an optimization procedure, refine these wall shapes to account for boundary layer growth along the wall.

5.1 Flow Model. Figure 14 presents the computational domain used to develop this model. The maximum value of $y^+$ for the nearest grid point to the blade was 0.9. The same inflow and outflow conditions as those used in the single passage model simulations were used. In order to keep the number of grid cells to a minimum, a two-layer $k$-$\varepsilon$ model was specified at the blade surface and wall functions were used for the passage walls. The results were monitored to ensure that the cell nearest to the wall was well within the viscous sublayer around the blade, and within the log-layer along the passage walls. Grid refinement studies showed that at least 70,000 cells were required for the simulations.

5.2 Outer Wall Design. Laskowski et al. [53] presents a detailed implementation of this approach for the previously described blade geometry. An inverse procedure is utilized to determine the wall shape that will give the same surface pressure and skin friction on the blade surface as that of the infinite cascade result. The essence of this approach is the computation of a cost function based on the blade surface pressure distribution and skin friction. The ultimate objective is to develop outer wall shapes such that streamlines a $\delta_1$ away from the wall closely follow those of an infinite cascade flow field. However, a critical issue is the selection of the initial wall shapes. The streamlines selected by Laskowski et al. [53] were approximated from the predicted stagnation streamlines from the infinite cascade simulation. Subsequent calculations using these outer walls resulted in pressure gradients along the upper wall large enough to produce a separation zone (as shown in Figure 15). Consequently, a penalty function was added to the cost function to ensure that the flow remained attached along both the passage walls. Alternative choices for streamlines for the initial set of outer walls were not explored to determine if this could be avoided. Figure 16(a) presents the final wall shapes for the double passage. The suction surface wall was found to require little modification from the initial guess, as the pressure gradients along this wall are favorable. This feature encouraged the development of thin boundary layers that minimized the flow field. Figure 16(b) compares the computed Mach number contours for the infinite cascade and the double passage model, demonstrating the flow field agreement between the numerically designed double passage and the infinite cascade condition.

5.3 Experimental Validation. To experimentally measure the flow conditions in the double passage, an instrumented airfoil with 17 pressure taps was installed in the model. The walls were fabricated from Plexiglas to allow the transmission of laser light for high-accuracy particle-image velocimetry (PIV) and LDA. This procedure allowed for measurement of the mean flow field and various turbulence correlations.

The taps consisted of cross-drilled holes. The tap on the surface was a 0.58 mm diameter hole, drilled perpendicular to the local surface tangent. These were connected to the Scanivalve. Figure 17 compares the measured and predicted $M_\infty$ distributions for the model against the infinite cascade simulation. This figure demonstrates that the numerical optimization procedure indeed developed a model that met the desired distribution. The measured mass flow rate through the system was 0.63 kg/s, which is slightly lower than the single passage. However, even after considerable numerical design effort, PIV measurements by Vicharelli and Eaton [46] indicated that the separation zone along the pressure side wall persisted in the completed model and affected the measured turbulence flow field. This result was observed in spite of the fact that the mean flow measurements around the blade were in relatively good agreement with their computed counterparts. This observation demonstrates...
that future facilities should specifically target elimination of this issue. We are currently in the process of performing high-resolution, boundary layer LDA measurements for comparison against numerical predictions. These data may be used to establish various boundary layer thickness parameters around the airfoil.

6 3-D Simulation Results

The models were developed using two-dimensional flow computations. Nevertheless, in the actual experimental facility, the flow is inherently three dimensional, in part due to the presence of flat endwalls that enclose the model. Researchers such as Langston [54] and Chung and Simon [41] have demonstrated that the boundary layers that develop along the endwalls cause highly complex three-dimensional flow features in between cascade blades. With this in mind, the aspect ratio of the model was chosen to be $A_S = 1.276$, where $A_S$ is defined as:

$$A_S = \frac{H_{\text{model}}}{AP}$$

$AP$ is the blade pitch spacing and $H_{\text{model}}$ is the passage height. This value was based on recommended practice as reported by Buck [49] and flow supply limitations. Ideally, this aspect ratio is large enough such that the three-dimensional effects are limited to the near-endwall regions. This design approach would result in a highly two-dimensional flow field over a wide-band encompassing the midspan region of the blade. Three-dimensional RANS simulations of the single and double passages were used to verify this assumption; further details can be found in Ref. [44,45]. Results from the single passage model calculations are only presented here, in the interests of brevity. However, the double passage model simulations exhibit practically identical characteristics.

The calculation domain included one-half of the channel width with a symmetry boundary condition at the channel centerline. A hyperbolic tangent grid stretching was used to resolve the endwall boundary layers. The computational grid contained approximately $2.6 \times 10^6$ cells, with $y^+$ values ranging from $1.3 \times 10^{-4} \leq y^+ \leq 3.0$.

The computed three-dimensional $M_{\infty}$ distribution at midspan of the model ($Z' = z/H_{\text{model}} = 0.0$) was found to closely agree with the two-dimensional simulation. This result demonstrated that the two-dimensional calculation is a good representation of the midspan flow conditions. Figure 18 compares the $M_{\infty}$ distribution at

![Fig. 16](image1)

![Fig. 17](image2)

![Fig. 18](image3)
the endwall ($Z' = -0.5$), at the centerline ($Z' = 0.0$), and at an exemplary intermediate location ($Z' = -0.375$). The effect of three-dimensionality is primarily observed on the suction side wall. This behavior was subsequently attributed to vortical structures that form from the endwall boundary layers. An examination of predicted $M_c$ distributions at various locations evinced that the three-dimensional nature of the flow is limited to region $-0.25 \leq Z' \leq -0.5$. It should be stated that these results are specific to this geometry and flow conditions; hence, it is difficult to extend these results to other facilities.

7 Conclusions

Two alternative approaches for achieving well-documented flow at conditions of interest to gas turbine designers have been presented. The design procedure and philosophies for transonic single and double passage models have been introduced and developed. Single passage models are advocated as a means of obtaining highly resolved heat transfer measurements with minimal surface conduction losses, whereas double passage models have been presented as a manner to obtain high-resolution fluid dynamics measurements around turbine airfoil geometries. Both approaches have been developed to produce a two-dimensional flow field that is identical to that in a two-dimensional infinite cascade. Pressure measurements from facilities built using these design procedures were used to verify this assertion. These facilities offer tremendous savings over linear cascades, as only one or two passages are utilized. This approach allows the use of steady state heat transfer measurement techniques that are more amenable to comparisons with numerical simulations. We have performed surface heat transfer and PIV measurements using both models. We are also in the process of performing detailed boundary layer measurements around the center blade in our double passage experiment. It should be stated that the mass flow requirements of these two facilities were approximately identical. This result is due to the fact that the single passage model has boundary layer bleed to limit viscous effects. Thus the cost of running and manufacturing both facilities is practically identical. It is important to point out that these models are designed for a specific inlet flow angle and pressure ratio; it is unclear how useful this approach would be for significantly off-design conditions. Furthermore, we have only validated these design philosophies for a transonic rotor blade geometry with fully attached boundary layers. Consequently, an important question that future efforts must answer is the utility of these models with significantly different airfoil geometries.

Acknowledgment

This research is sponsored by GE Aircraft Engines, through their University Strategic Alliance Program and by AFOSR Contract No. F49620-02-1-0284 monitored by T. Beutner. P.M.K. received support from the National Science Foundation via a three year graduate fellowship. The authors would like to express their sincere gratitude to D. Mukerji (at Stanford) and F. Buck (at GEAE) for their assistance and advice. Additionally, the authors would like to thank X. Wu and C. McNeil for their support in developing and executing the computational models. J. Hammer, L. Johal, and S. Sutton are gratefully acknowledged for their high level of expertise in manufacturing various parts in the experiments necessary to conduct this research.

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$c_p$</td>
<td>specific heat with constant pressure</td>
</tr>
<tr>
<td>$M$</td>
<td>Mach number</td>
</tr>
<tr>
<td>$P$</td>
<td>static pressure (Pa)</td>
</tr>
<tr>
<td>$P_0$</td>
<td>stagnation (total) pressure (Pa)</td>
</tr>
<tr>
<td>$T$</td>
<td>static temperature (K)</td>
</tr>
<tr>
<td>$T_0$</td>
<td>stagnation (total) temperature (K)</td>
</tr>
<tr>
<td>$TI%$</td>
<td>turbulence intensity</td>
</tr>
<tr>
<td>$\bar{u}_{\text{inlet}}$</td>
<td>area-averaged inlet velocity</td>
</tr>
<tr>
<td>$s_c$</td>
<td>distance relative to stagnation point along airfoil surface (mm)</td>
</tr>
<tr>
<td>$x$</td>
<td>axial distance relative to airfoil leading edge (mm)</td>
</tr>
<tr>
<td>$y^+$</td>
<td>dimensionless wall normal distance</td>
</tr>
<tr>
<td>$z$</td>
<td>spanwise surface coordinate relative to the centerline of the passage (mm)</td>
</tr>
<tr>
<td>$Z'$</td>
<td>dimensionless spanwise surface coordinate relative to the centerline of the passage</td>
</tr>
<tr>
<td>$\delta_1$</td>
<td>displacement boundary layer thickness (mm)</td>
</tr>
<tr>
<td>$\phi$</td>
<td>rotation angle (degrees)</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>ratio of specific heats ($c_p/c_v$)</td>
</tr>
<tr>
<td>$\rho$</td>
<td>density (kg/m$^3$)</td>
</tr>
<tr>
<td>$\mu$</td>
<td>kinematic viscosity (Pa·s)</td>
</tr>
</tbody>
</table>

Large subscripts/superscripts

2DRANS = refers to two-dimensional RANS simulation
$\tau$ = refers to dimensionless distance
$+$ = refers to dimensionless distance
blade = refers to blade dimensions
$c$ = refers to blade surface
IC = refers to infinite cascade simulation
is = refers to isentropic condition
model = model
$p$ = referring to constant pressure
plenum = plenum condition
$0$ = stagnation (total) condition
$rec$ = recovery condition
$v$ = refers to constant volume
ss = refers to suction surface
$ps$ = refers to pressure surface

References

649.
Numerical Study of the Heat Island Effect for Button-Type Heat Flux
Gages,” Proceedings of the 5th ASME/JSME Joint Thermal Engineering
culating the Heat Island Correction and Uncertainties for Button Gauges,”
pp. 279–368.
Gage in a Transonic Turbine Cascade,” Proceedings of the ASME Heat Tran-
ments in an Annular Cascade of Transonic Gas Turbine Blades Using the
Transient Liquid Crystal Technique,” ASME J. Turbomach., 117(3), pp. 423–
431.
Scaled Transonic Turbine Rotor Heat Transfer Measurements,” ASME J.
Turbomach., 102, pp. 1–7.
ARC Report and Memorandum.
“Endwall Heat Transfer Measurements in a Transonic Turbine Cascade,”
ASME Paper No. 96-GT-180.
“Blade Heat Transfer Measurements and Predictions in a Transonic Turbine
[28] Carni, C., and Arts, T., 1991, “Effect of Incidence on Wall Heating Rates and
Aerodynamics on a Film-Cooled Transonic Test Blade,” ASME J. Turbomach.,
Secondary Flow and Loss Fields Within a Linear Turbine Cascade,” ASME J.
Turbomach., 113(2), pp. 375–383.
Measurements in the Wake of a Compressor Cascade With Flow Separation,”
Proceedings of the ASME International Gas Turbine Institute, Atlanta, GA,
June 16–19, 6(A), pp. 91–103.
Investigation of an Innovative Cooling System (ICS) for High Temperature
Transonic Turbine Stages,” Proceedings of the ASME International Gas Tur-
bine Institute, Amsterdam, The Netherlands, June 3–6, 5(A), pp. 159–167.
iveness and Heat Transfer Distribution on a Gas Turbine Airfoil,” ASME J.
the Flow Field Downstream of a Turbine Trading Edge Cooled Nozzle Guide
Heat Transfer on Rotor Blades in a Transonic Turbine Cascade,” ASME J.
Turbomach., 126(3), pp. 191–212.
302–309.
255–262.
[38] Priddy, W., and Bayley, F., 1988, “Turbulence Measurements in Turbine Blade
73–79.
[40] Bailey, D., 1980, “Study of Mean and Turbulent-Velocity Fields in a Large-
Junction of a Gas Turbine: Application of a Boundary Layer Fence,”
ASME Paper No. 90-WAHT4-4.
Near the Blade/Endwall Junction of a Gas Turbine: Visualization in a Large-
Scale Cascade Simulator,” ASME Paper No. 91-GT-45.
Test Model to Obtain Turbine Airfoil Film Cooling Effectiveness Data,”
ASME Paper No. 95-GT-91.
in a Transonic Single Passage Model,” Technical Report No. TF 93, Stanford
University, Stanford CA, available at http://www.stanford.edu/group/fpc/
Publications/TF.html.
a Stationary and Rotating Serpentine Passage,” Ph.D. thesis, Stanford Uni-
versity, Stanford.
Passage Turbine Cascade,” Exp. Fluids, 40(6), pp. 897–917.
Dynamics, Melville, NY.
[50] Halderman, C., Dunn, M., Barter, J., Green, B., and Bergholz, R., 2005, “Aero-
dynamic and Heat-Flux Measurements With Predictions on a Modern One
and One-Half Stage High Pressure Transonic Turbine,” ASME J. Turbomach.,
2005, “Inverse Design of and Experimental Measurements in a Double Pas-