Low-Pressure Turbine Separation Control—Comparison With Experimental Data

Vijay K. Garg
University of Toledo, Toledo, Ohio
Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the Lead Center for NASA’s scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA’s institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- **TECHNICAL PUBLICATION.** Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA’s counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.

- **TECHNICAL MEMORANDUM.** Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.

- **CONTRACTOR REPORT.** Scientific and technical findings by NASA-sponsored contractors and grantees.

- **CONFERENCE PUBLICATION.** Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.

- **SPECIAL PUBLICATION.** Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.

- **TECHNICAL TRANSLATION.** English-language translations of foreign scientific and technical material pertinent to NASA’s mission.

Specialized services that complement the STI Program Office’s diverse offerings include creating custom thesauri, building customized data bases, organizing and publishing research results . . . even providing videos.

For more information about the NASA STI Program Office, see the following:


- E-mail your question via the Internet to help@sti.nasa.gov

- Fax your question to the NASA Access Help Desk at 301–621–0134

- Telephone the NASA Access Help Desk at 301–621–0390

- Write to:
  NASA Access Help Desk
  NASA Center for AeroSpace Information
  7121 Standard Drive
  Hanover, MD 21076
Low-Pressure Turbine Separation Control—Comparison With Experimental Data

Vijay K. Garg
University of Toledo, Toledo, Ohio

Prepared for the
Turbo Expo 2002
cosponsored by the American Society of Mechanical Engineers
and the International Gas Turbine Institute
Amsterdam, The Netherlands, June 3–6, 2002

Prepared under Contract NAS3–00180 and Cooperative Agreement NCC3–926

National Aeronautics and
Space Administration

Glenn Research Center

July 2002
Acknowledgments

The author wishes to thank Dr. Raymond Gaugler, Chief, Turbine Branch, at the NASA Glenn Research Center for his support of this work. Help from Dr. Richard Rivir of the Air Force Research Laboratory and Major Jeffrey Bons of the Air Force Institute of Technology regarding the experimental data is also gratefully acknowledged.

Available from
NASA Center for Aerospace Information
7121 Standard Drive
Hanover, MD 21076

Available electronically at http://gltrs.grc.nasa.gov/GLTRS
ABSTRACT

The present work details a computational study, using the Glenn-HT code, that analyzes the use of vortex generator jets (VGJs) to control separation on a low-pressure turbine (LPT) blade at low Reynolds numbers. The computational results are also compared with the experimental data of Bons et al. [1] for steady VGJs. It is found that the code determines the proper location of the separation point on the suction surface of the baseline blade (without any VGJ) for Reynolds numbers of 50,000 or less. Also, the code finds that the separated region on the suction surface of the blade vanishes with the use of VGJs. However, the separated region and the wake characteristics are not well predicted. The wake width is generally over-predicted while the wake depth is under-predicted.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>B</td>
<td>jet blowing ratio (= (\rho u)<em>{jet}/(\rho u)</em>{local} )</td>
</tr>
<tr>
<td>C_p</td>
<td>blade pressure coefficient (= (p_{o,in} - p_r)/0.5(\rho u^2)_{m} )</td>
</tr>
<tr>
<td>C_x</td>
<td>axial chord of the blade</td>
</tr>
<tr>
<td>d</td>
<td>injection hole diameter</td>
</tr>
<tr>
<td>k</td>
<td>turbulence kinetic energy</td>
</tr>
<tr>
<td>\ell</td>
<td>turbulence length scale</td>
</tr>
<tr>
<td>p</td>
<td>pressure</td>
</tr>
<tr>
<td>\Pr</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>Re</td>
<td>inlet Reynolds number (= C_x(\rho u/\mu)_{m} )</td>
</tr>
<tr>
<td>Tu</td>
<td>freestream turbulence intensity</td>
</tr>
<tr>
<td>u</td>
<td>streamwise mean velocity</td>
</tr>
<tr>
<td>v*</td>
<td>shear velocity</td>
</tr>
<tr>
<td>x,y</td>
<td>Cartesian coordinate system with origin at the blade leading edge</td>
</tr>
<tr>
<td>y*</td>
<td>distance in wall coordinates (= \rho y\nu* / \mu )</td>
</tr>
<tr>
<td>\alpha</td>
<td>thermal diffusivity</td>
</tr>
<tr>
<td>\Delta y</td>
<td>distance (from the wall) of the first point off the wall</td>
</tr>
<tr>
<td>\epsilon</td>
<td>turbulence dissipation rate</td>
</tr>
<tr>
<td>\gamma</td>
<td>wake loss coefficient (= (p_{o,in} - p_{w,ex})/(p_{o,ex} - p_{w,ex}) )</td>
</tr>
<tr>
<td>\mu</td>
<td>dynamic viscosity</td>
</tr>
<tr>
<td>\rho</td>
<td>density</td>
</tr>
<tr>
<td>\tau</td>
<td>shear stress</td>
</tr>
</tbody>
</table>

\[ \omega \] specific turbulence dissipation rate \(= \epsilon/k \)

Subscripts:

- ef effective value
- ex value at cascade exit
- in value at cascade inlet
- jet vortex generator jet conditions
- \ell laminar value
- local local blade mid-channel conditions
- o stagnation value
- s static value
- t turbulent value
- w value at the wall

INTRODUCTION

For the flow over an airfoil, there is an adverse pressure gradient region over a large part of the suction surface. This can cause the flow to separate on the suction surface, especially at low Reynolds numbers, in which case the boundary layer over the blade may be laminar. It is well known that a laminar boundary layer can withstand only a minor adverse pressure gradient before separation while a turbulent boundary layer can withstand a much stronger adverse pressure gradient region. Flow separation reduces lift and increases drag. It is thus undesirable and needs to be controlled.

Flow separation can be controlled by both passive and active techniques. The former employs small vortex generators (rectangular or delta shaped winglets) which are imbedded in a boundary layer ahead of a line of flow separation [2]. Spanwise arrays of vortex generators are often placed along a wing upstream of the flap hinge or inside the lip of a jet engine inlet diffuser. The longitudinal vortices generated in the boundary layer increase cross-stream mixing of streamwise momentum and suppress or eliminate separation. These fixed vortex generators have the advantages of simplicity, ruggedness and low cost. However, they add parasitic drag in flow situations where separation control is not needed, e.g., take-off and landing.
In one of the active techniques, small jets blown through holes in
the solid surface can generate longitudinal streamwise vortices in a
boundary layer. These vortices increase cross-stream mixing of
streamwise momentum. The surface holes are pitched an angle to the
surface (generally low pitch angle of 30-45°), and inclined relative to
the main flow direction (skew angle varying from 45-90°), much like
the compound-angled holes on the shower-head of film-cooled blades.
This method is called the vortex generator jet (VGJ) method. This
method was first examined by Wallis [3, 4], and by Wallis and Stuart
[5] primarily for the purpose of delaying shock-induced separation of
turbulent boundary layers. Ball [6] employed VGJs alone and together
with fixed generators for stall suppression in inlet diffusers of jet
engines. Henry and Pearcey [7] show clearly the development of a
single, dominant vortex for a VGJ at high skew angles. It has been
shown by Johnston and Nishi [8] that this vortex energizes the
separating boundary layer by bringing high momentum freestream fluid
down to the wall. Experimental data on the beneficial effects of VGJs
are available in [9].

The flow requirements for the jets in the VGJ method is a very
small fraction of the total jet engine flow, and compressor bleed air is
sufficient to power the system. The jets can be turned on and off as
desired. With jets off, there is no parasitic drag. The potential
applications of the VGJ method are numerous for external flow over
aircraft and missiles. Internal flows in diffusers can also be improved
and pressure losses reduced. The VGJ method can be used with
appropriate flow direction sensors and a feedback control system to
suppress surge and rotating stall in compressors of jet engines.

The operating Reynolds number for the low-pressure turbine in an
aircraft gas turbine engine can drop below 25,000 during high altitude
cruise. At such low Reynolds numbers, the boundary layers on the
LPT blades are largely laminar, making them highly susceptible to
flow separation near the aft portion of the blade suction surface, with
associated increase in losses and drop in performance. Altering the
blade shape to avoid this low Reynolds number separation problem is
not feasible since such a modification will likely impair the engine
operation at higher (design) Reynolds numbers. As such, an active
flow control strategy, such as the use of VGJs, has been tested
experimentally at the Air Force Research Laboratory recently [1, 10].
This paper aims to use the Glenn-HT code in order to analyze the use
of VGJs at low Reynolds numbers, and to compare the computational
predictions with the experimental data of Bons et al. [1].

ANALYSIS

The numerical simulation has been performed using the NASA
Glenn Research Center General Multi-Block Navier-Stokes Convective
Heat Transfer code, Glenn-HT. Briefly, the code, formerly known as
TRAF3D.MB [11], is an explicit, multigrid, cell-centered, finite
volume code with a $k$-ω turbulence model without any wall functions.
This is a general purpose flow solver designed for simulations of flows in
complicated geometries. The Navier-Stokes equations in a rotating
Cartesian coordinate system are mapped onto a general body-fitted
coordinate system using standard techniques. The multistage Runge-
Kutta scheme developed by Jameson et al. [12] is used to advance the
flow solution in time from an initial approximation to the steady state.
A spatially varying time step along with a CFL number of 4 is used to
speed convergence to the steady state. Eigenvalue-scaled artificial
dissipation and variable-coefficient implicit residual smoothing are used
along with a full-multigrid method. The overall accuracy of the code
is second order. No wall functions are used, thus avoiding any bias to
the complex three-dimensional flow structures near the blade or any
other surface. While the Glenn-HT code is basically a compressible
code, the present problem involves incompressible flow at rather low
Reynolds numbers. Thus, there were some questions regarding the use
of Glenn-HT code in such a situation. Needless to say, such questions
about getting converged solution from the Glenn-HT code for such a
case have been put to rest. How these results compare with experimental
data is another matter.

While the Glenn-HT code has the original $k$-$\omega$ model [13], the
shear stress transport (SST) model of Menter [14], and the $k$-$\varepsilon$
model of Wilcox [15] were implemented in it by Garg and Ameri [16].
The SST model encompasses both the $k$-$\omega$ and the $k$-$\varepsilon$ models, with the original $k$-$\omega$ model of Wilcox [13] activated in the near-wall region
and the standard $k$-$\varepsilon$ model [17] activated in the outer wake region and
in free shear layers. Moreover, the definition of eddy viscosity is
modified to account for the transport of the principal turbulent shear
stress. The reader is referred to Menter [14] for an elucidating
discussion of the SST model. More details on the relevant equations
and their implementation are available in Garg and Ameri [16]. The
SST model was used for the present computations.

It is assumed that the effective viscosity for turbulent flows can
be written as

$$
\mu_{\text{eff}} = \mu_{t} + \mu_{f} \quad (1)
$$

where the laminar viscosity $\mu_{l}$ is calculated using a power-law for its
dependence on temperature [18]. The turbulent viscosity $\mu_{t}$ is
computed using the SST model. The turbulent thermal diffusivity is
computed from

$$
\kappa_{t} = \frac{\mu_{t}}{\rho Pr_{t}} \quad (2)
$$

where a constant value of 0.9 is used for the turbulent Prandtl number,
$Pr_{t}$.

Boundary Conditions

At the main flow inlet boundary located at an axial distance equal
to the blade axial chord upstream of the blade leading edge, the total
temperature, total pressure, swirl, and meridional flow angle are
specified, and the upstream-running Riemann invariant based on the
total absolute velocity is calculated at the first interior point and
extrapolated to the inlet. The velocity components are then decoupled
algebraically, and the density is found from total temperature, total
pressure and total velocity using an isentropic relation. For the
turbulence model, the value of $k$ and $\omega$ is specified using the
experimental conditions, namely

$$
k = 1.5 (u_{\text{in}} T_{u_{\text{in}}})^2, \quad \omega = k^{1/2}/\kappa, \quad (3)
$$

where $T_{u_{\text{in}}}$ is the intensity of turbulence at the inlet (taken to be 0.01
as per experimental data for the PakB blade), $u_{\text{in}}$ is the absolute
velocity at inlet, and $\kappa$ is the integral length scale representing the size
of the energy containing eddies. This length scale needs to be assumed,
if not reported as part of the experimental conditions, as in the
present case. It was assumed to be 5% of the blade axial chord.

At the main flow exit plane located at an axial distance equal to
80% of the blade axial chord downstream of the blade trailing edge,
the static pressure is specified and the density and velocity components are extrapolated from the interior. At the solid surface of the blade, the no-slip condition is enforced, and temperature is specified as per experimental data. The boundary conditions for turbulence quantities on the walls are \( k = 0 \), and

\[
\omega = 100 \frac{\partial u}{\partial y_{wall}}
\]

for a hydraulically smooth surface. An upper limit is imposed on the value of \( \omega \) at the wall, as suggested by Menter [19] and found effective by Chima [20],

\[
(\omega_{\text{max}})_{\text{wall}} = \frac{800}{Re} \frac{v}{(\Delta y)^2}
\]

The grid around the blade extends to mid-way between two adjacent blades with periodic flow conditions in terms of cylindrical velocity components set on a dummy grid line outside this boundary. For a linear cascade (which is true for the experimental data), it is possible to consider only a slice of the real span for computational purposes with a periodic boundary condition at both ends of the computational span. For the basic blade with no VGJ, this slice can be of any reasonable width, but with VGJs, the width of the slice is one span-wise pitch of the holes.

The effect of VGJs has been incorporated in the form of appropriate boundary conditions at the hole locations on the blade surface. Each hole exit in its true oval shape is represented by 224 control volumes. Different velocity and temperature profiles for the injected fluid can be specified at the hole exits. For the present study, polynomial distribution [21] of velocity and temperature of the coolant at the hole exit was specified. The coolant angle was taken to be the same as the hole angle. Turbulent intensity at the hole exit was assumed to be 1% (same as for the freestream), while the turbulence length scale at the hole exit was taken to be the same as the hole diameter \( d \).

**EXPERIMENTAL DETAILS**

Measurements were made by Bons et al. [1] in an induction wind tunnel at the U.S. Air Force Research Laboratory. The linear cascade consisted of eight 0.88 m (34.5 in) span, 0.18 m (7 in) axial chord (C\(_a\)) blades plus two partial blade endwalls. The blades were fabricated from molded polyurethane resin. The 2-D blade shape studied is the Pratt & Whitney "PakB" research design, which is a Mach number scaled version of a typical highly loaded LPT blade design. The cascade had a solidity (axial chord to blade spacing) of 1.13, an inlet flow angle of 55° (measured from the plane of the cascade), and a design exit angle of 30°. Blades 4 and 6 were instrumented around both pressure and suction surfaces (in the center 0.2 m of their span) with forty 1 mm diameter static pressure taps. Uncertainties in the pressure measurement translated to an error of ±0.18 in the \( C_p \) data at \( Re = 100,000 \) [1].

Blade #5 of the cascade had the active separation control. It was manufactured with a hollow cavity running the span of the blade and covering the region from 40% to 90% axial chord. A needle valve located upstream of the feed port allows fine control of the mass flow rate into the blade cavity. Air exhausts from this valve into a 1.2 cm diameter capped copper tube running the span of the blade inside the cavity. Holes of 1.5 mm diameter and spaced every 2.54 cm along the copper tube produced an even distribution of air flow to the VGJs. The 1 mm diameter (\( d \)) cylindrical VGJ holes were drilled from the exterior surface of the blade, through the 4 mm thick wall, and into the cavity with a 30° pitch angle and a 90° skew angle. The VGJ holes have a length of \( 8d \) and are spaced every 10\( d \) along the center 0.46 m of this blade span. The row of VGJs was placed at 0.73 C\(_x\). This location is within the separated region on the baseline, uncontrolled blade at a Reynolds number of 50,000 [1]. The jet blowing ratio (\( B \)) was computed as the ratio of the jet exit velocity to the local freestream velocity as calculated from the local pressure coefficient. While the inlet Reynolds number was varied between 25,000 and 100,000, \( B \) was taken as 0, 1, 2 or 4.

Standard pitot and hot-wire probes mounted on a large Dantec 3-axis traverse located atop the tunnel facility were used to map a 0.6m × 0.6m planar cross-section of the flow domain. Wake velocity profiles were measured using a single 4µm diameter tungsten hot-wire. A single TSI sub-miniature hot-film probe was used for boundary layer profile measurements. The hot-film sensor diameter was 25µm and the sensing element length was 0.25mm. The error in the hot-wire and film probes was within ±2% at flow rates of interest. More details are available in [1].

**COMPUTATIONAL DETAILS**

The computational span extended over a part of the blade span with a periodic boundary condition at both ends. In the axial direction, the computational domain extended from the inlet plane located one axial chord upstream of the blade leading edge to the exit plane located 80% of the axial chord downstream of the blade trailing edge. Around the blade, the grid extends to mid-way between two adjacent blades with periodic boundary conditions. Figure 1 shows a spanwise section of the multi-block viscous grid around the blade. The viscous grid is obtained from an inviscid grid by clustering the grid near all the solid walls (blade here). The clustering is done in such a way as to ensure that in the viscous grid, the distance of any cell center adjacent to a solid wall, measured in wall units (\( y^+ \)), is less than half for the cases studied here, following Boyle and Giel [22]. The inviscid grid was generated using the commercial code GridPro/az3000 [23]. For computational accuracy the ratio of two adjacent grid sizes in any direction was kept within 0.8-1.25. As can be observed from Fig. 1, the grid quality is very good especially near the blade surface. Figure 1 also shows the grid over the blade span near and within the injection hole exits.

Initially, the grid for the blade with injection holes consists of 136 blocks but before the solver is used, it can be merged into just 10 blocks using the Method of Weakest Descent [24]. The final viscous grid consists of 160128 cells, formed by clustering near the blade from an inviscid grid with 92928 cells. There are 224 cells within the hole exit on the blade surface. The inviscid grid has 144 cells around the blade (for the O-grid around the blade), 28 cells in the blade-to-blade direction from the blade to the periodic boundary in-between the two blades, and 24 in the spanwise direction. After clustering, the number of cells in the blade-to-blade direction increases to 48. Two more grids were generated for a grid-independence study. One inviscid grid had 1.5 times the number of cells in each direction as compared to the basic grid described above. For the second grid, the basic inviscid grid was clustered near the blade with a grid spacing half of that for the basic viscous grid. All these variations of the basic grid yielded nearly the same values for the skin friction coefficient on the entire blade surface as the basic grid; any variations were within ±2%. The results presented here correspond to the basic grid shown in Fig. 1.
Figure 1 compares the computed boundary layer profiles over the blade suction surface for Re = 50,000 and B = 0. This figure shows the boundary layer profiles for two inlet Reynolds numbers and Tu = 1%. The comparison is very good on the pressure surface, but not so good on the suction surface. In the case of the present computations, the boundary layer on the pressure surface is essentially laminar, as evidenced in Figure 2. However, the boundary layer on the suction surface is turbulent, as shown in Figure 3. This figure compares the computed boundary layer profiles for Re = 25,000 and 100,000. The comparison is very good, especially at 75% chord. The boundary layer on the suction surface is not well resolved by the computations.

Figure 2 shows the computed boundary layer profiles for Re = 50,000 and B = 0. Clearly at this high Reynolds number, the boundary layer on the suction surface is turbulent. The comparison with the experimental data is very good, especially at 75% chord. The boundary layer on the suction surface is not well resolved by the computations.

Figure 3 compares the computed boundary layer profiles over the blade suction surface for Re = 50,000 and B = 0. This figure shows the boundary layer profiles for two inlet Reynolds numbers and Tu = 1%. The comparison is very good on the pressure surface, but not so good on the suction surface. In the case of the present computations, the boundary layer on the pressure surface is essentially laminar, as evidenced in Figure 2. However, the boundary layer on the suction surface is turbulent, as shown in Figure 3. This figure compares the computed boundary layer profiles for Re = 25,000 and 100,000. The comparison is very good, especially at 75% chord. The boundary layer on the suction surface is not well resolved by the computations.

Figure 4 shows the computed boundary layer profiles over the blade suction surface for Re = 50,000 and B = 0. This figure shows the boundary layer profiles for two inlet Reynolds numbers and Tu = 1%. The comparison is very good on the pressure surface, but not so good on the suction surface. In the case of the present computations, the boundary layer on the pressure surface is essentially laminar, as evidenced in Figure 2. However, the boundary layer on the suction surface is turbulent, as shown in Figure 3. This figure compares the computed boundary layer profiles for Re = 25,000 and 100,000. The comparison is very good, especially at 75% chord. The boundary layer on the suction surface is not well resolved by the computations.

Figure 5 compares the computed boundary layer profiles over the blade suction surface for Re = 50,000 and B = 0. This figure shows the boundary layer profiles for two inlet Reynolds numbers and Tu = 1%. The comparison is very good on the pressure surface, but not so good on the suction surface. In the case of the present computations, the boundary layer on the pressure surface is essentially laminar, as evidenced in Figure 2. However, the boundary layer on the suction surface is turbulent, as shown in Figure 3. This figure compares the computed boundary layer profiles for Re = 25,000 and 100,000. The comparison is very good, especially at 75% chord. The boundary layer on the suction surface is not well resolved by the computations.

Figure 6 compares the computed boundary layer profiles over the blade suction surface for Re = 50,000 and B = 0. Clearly at this high Reynolds number, the boundary layer on the suction surface is turbulent. The comparison with the experimental data is very good, especially at 75% chord. The boundary layer on the suction surface is not well resolved by the computations.

Figure 7 compares the computed boundary layer profiles over the blade suction surface for Re = 50,000 and B = 0. Clearly at this high Reynolds number, the boundary layer on the suction surface is turbulent. The comparison with the experimental data is very good, especially at 75% chord. The boundary layer on the suction surface is not well resolved by the computations.

Figure 8 compares the computed boundary layer profiles over the blade suction surface for Re = 50,000 and B = 0. Clearly at this high Reynolds number, the boundary layer on the suction surface is turbulent. The comparison with the experimental data is very good, especially at 75% chord. The boundary layer on the suction surface is not well resolved by the computations.

Figure 9 compares the computed boundary layer profiles over the blade suction surface for Re = 50,000 and B = 0. Clearly at this high Reynolds number, the boundary layer on the suction surface is turbulent. The comparison with the experimental data is very good, especially at 75% chord. The boundary layer on the suction surface is not well resolved by the computations.

Figure 10 compares the computed boundary layer profiles over the blade suction surface for Re = 50,000 and B = 0. Clearly at this high Reynolds number, the boundary layer on the suction surface is turbulent. The comparison with the experimental data is very good, especially at 75% chord. The boundary layer on the suction surface is not well resolved by the computations.
experimental data show steeper profiles than the computed results at all the stations, similar to the results in Fig. 7 for Re = 100,000 and no VGJ. Bons et al. [1] attribute the doubly-inflected (but attached) velocity profile at 79% axial chord station to the ultimate penetration depth of the vortex jets into the freestream. The computed results do not show any such anomaly.

Figure 11 compares the computed velocity profiles in the wake of 0.62 axial chord downstream from the blade trailing edge for an inlet Reynolds number of 50,000 against the experimental data of Bons et al. [1] with and without VGJs. We may note that the data corresponding to B = 0 are also available in Fig. 8; they are included here to compare with the data for B ≠ 0. The computed wake with VGJs on (B ≠ 0) is shallower and somewhat narrower than that without any VGJ (B = 0), as one would expect since the separation region vanishes when VGJs are on. However, the effect is not so dramatic, specially for the wake width, as for the experimental data. Experimentally the wake is very narrow when the VGJs are on compared to the case when they are not. The wake depth recovery, ratio of the wake depth for B ≠ 0 to that for B = 0, is about the same for both the computational and experimental data. Also, there is little difference between the profiles corresponding to B = 1 and B = 2 for both the computational and experimental data.

CONCLUSIONS

The Glenn-HT code was used to compute the flow over a low-pressure turbine blade at very low Reynolds numbers with and without the use of vortex generator jets. The results are compared with the experimental data of Bons et al. [1] for steady VGJs. It is found that the code determines the proper location of the separation point on the suction surface of the baseline blade (without any VGJ) for Reynolds numbers of 50,000 or less. Also, the code finds that the separated region on the suction surface of the blade vanishes with the use of VGJs. However, the separated region and the wake characteristics are not well predicted. The wake width is generally over-predicted while the wake depth is under-predicted. Thus, there is a need to improve the prediction of wake characteristics, especially in situations where large separated regions may exist upstream of the wake, as in the present case. For resolution of the unsteady effects due to flow separation and wake shedding, use of an unsteady code is recommended.

REFERENCES

Fig. 6 Comparison of boundary layer profiles of streamline velocity normalized by mid-channel velocity at three chordwise stations: 0.67, 0.73 and 0.79 axial chord on the suction side. Re = 50,000; Tu = 1%; B = 0.

Fig. 7 Comparison of boundary layer profiles of streamline velocity normalized by mid-channel velocity at three chordwise stations: 0.67, 0.73 and 0.79 axial chord on the suction side. Re = 100,000; Tu = 1%; B = 0.

Fig. 8 Comparison of wake velocity profile at 0.62 axial chord downstream from the blade trailing edge for two inlet Reynolds numbers. Tu = 1% and B = 0.

Fig. 9 Comparison of loss coefficient vs. inlet Reynolds number for Tu = 1%.

Fig. 10 Comparison of boundary layer profiles of streamline velocity normalized by mid-channel velocity at three chordwise stations: 0.67, 0.73 and 0.79 axial chord on the suction side. Re = 50,000; Tu = 1%; B = 2.

Fig. 11 Comparison of wake velocity profile at 0.62 axial chord downstream from the blade trailing edge for Re = 50,000; Tu = 1%; B = 0, 1, 2.
### Low-Pressure Turbine Separation Control—Comparison With Experimental Data

**Vijay K. Garg**

*University of Toledo, Toledo, Ohio 43606*


The present work details a computational study, using the Glenn HT code, that analyzes the use of vortex generator jets (VGJs) to control separation on a low-pressure turbine (LPT) blade at low Reynolds numbers. The computational results are also compared with the experimental data for steady VGJs. It is found that the code determines the proper location of the separation point on the suction surface of the baseline blade (without any VGJ) for Reynolds numbers of 50,000 or less. Also, the code finds that the separated region on the suction surface of the blade vanishes with the use of VGJs. However, the separated region and the wake characteristics are not well predicted. The wake width is generally over-predicted while the wake depth is under-predicted.