Int. Symp. on Heat Transfer in Gas Turbine Systems
August 9-14, 2009, Antalya, Turkey

EFFECTS OF STATOR/ROTOR LEAKAGE FLOW AND AXISYMMETRIC CONTOURING ON ENDWALL ADIABATIC EFFECTIVENESS

Ryan Erickson and Terrence Simon
Mechanical Engineering Department
University of Minnesota, Minneapolis, MN 55455, USA
(*Corresponding author: tsimon@me.umn.edu)

INTRODUCTION

The high pressure turbine stage possesses a wealth of complexities for designers to consider. One of recent interest is that of leakage (or purge) flow ejection at the stator-to-rotor interface. At this interface, a small cavity is designed into the engine to provide clearance between the rotor disk and the stator. The dimensions of this cavity can vary as a result of transient operating conditions. The ideal cavity is as small as practical. The cavity must be protected from hot gas ingression. Current designs meter flow bled from the high pressure compressor into the cavity using labyrinth seals, as shown in Figure 1. This flow, used to prevent ingression of hot gas from the passage, also offers cooling. Its interaction with the hot gas path flow as it exits the disc cavity, is the topic of this study.

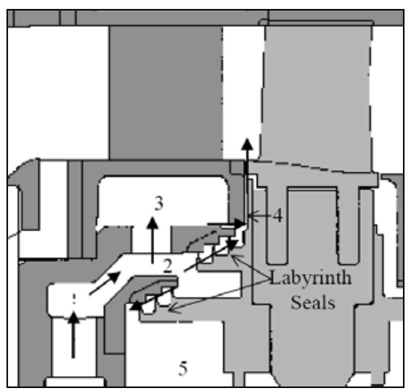

Figure 1. Representative engine cross-section showing location of labyrinth seals and disc cavity. Taken from Dénos et al. [2002].

Use of excess leakage flow offers an efficiency penalty since that compressor discharge air is not heated in the combustor or expanded in the vane. It also may cause passage aerodynamic losses as it mixes with the passage flow. Thus, though it may provide cooling, it must be used sparingly. Unfortunately, studies documented in the literature of leakage flow and its effects are geometry dependent. Various designs are present in the literature showing leakage flow being discharged radially, tangentially, or at angles between the two with respect to the mainstream flow. Most early studies in this area focused on the problem of parallel disks (one rotating) with a shroud, as described in a review by Owen [1988]. Many of the parallel disk studies focus on the problem of finding the minimum coolant flow rate for preventing ingression, such as that of Bhavnani et al. [1992]. Ko et al. [1992] and others investigated thermal transport at the disk cavity/mainstream interface with the parallel disk geometry. Such research identified recirculation zones within the cavity where ingested mainstream fluid could create thermal-stress problems. More recently, Cao et al. [2003] performed unsteady CFD simulations in a parallel disk geometry that did not include blades. They identified rotating flow features within the disk cavity that resulted in alternating cells of ingression and ejection. These cells were verified by pressure measurements in a two-stage turbine rig. They were found to not be strong functions of disk speed. The cells generally rotated at 90-97% of the rotor disk speed and were hypothesized to be driven by instabilities at the mainstream/disk cavity interface. This particular study is interesting as most unsteady ingression/ejection behavior was previously attributed to asymmetries in the annulus pressure field caused by stator wakes.

Several recent numerical/experimental studies from the von Karman Institute [e.g. Dénos et al. 2002, Paniagua et al. 2004, and Pau et al. 2008] have focused on the effects of leakage flow on the mainstream. Their geometry essentially matches the case previously discussed of a radial outflow disc cavity with no step between the stator rim and the rotor hub; however, it includes also the effects of blading, temperature ratio, Mach number, and operating Reynolds number. In general, they found that leakage flow increases secondary flow losses as most of it is entrained into the main passage vortex. This finding also suggests that leakage flow has cooling potential for the endwall, but only in the upstream portion of the passage.

Other disk cavity designs can be found in the open literature. These designs allow for leakage flow ejection which is not entirely radial. One, by Ong et al. [2006], incorporates a small downward step between the stator rim and a rounded rotor platform. They find that leakage flow enters the passage with less swirl than that of the main flow, effectively decreasing efficiency in downstream blade rows due to enhanced mixing by stronger secondary flows. They also cite the velocity difference of leakage and mainstream flows as a source of loss at the injection location due to mixing, and note an improvement in stage efficiency when injecting leakage flow with additional swirl. A similar computational study by Marini and Girgis [2007] looks at the effect of modifying the endwall leading edge shape on mainstream/leakage flow interaction. They compared one endwall contour that was recessed below the stator rim with one that was level with the rim. Results suggested that the recessed endwall design was more sensitive to changes in leakage flow rates and produced lower stage efficiency than that computed with the no-step counterpart. The above indicates that recent interest has led to important and informative studies, but much remains to be learned.

The current study investigates mainstream/leakage flow interaction comparing two new endwall contour designs. These designs have similar characteristics to some of those previously studied [e.g. Marini 2007] where there is no height change between the stator rim and the rotor endwall. They differ, however, in their endwall leading edge curvature. The idea behind varying the endwall curvature is that it may introduce leakage flow in a manner that leads to lower aerodynamic losses. Endwall and blade leading edge curvature have been used in some cases to successfully weaken the horseshoe vortex and passage secondary flows [see a review by Simon and Piggush, 2006]. Measurements are compared with unsteady CFD results. Due to complexities involved in making measurements within a turbine, a linear cascade is substituted. Though the cascade is not a perfect replication of the turbine geometry and flow, more detailed information may be obtained and important effects may be isolated and documented. In the present study, this is done with rotational and Mach number effects excluded so that the injection and secondary flow may be carefully documented. Engine Reynolds numbers are matched. The scale-up in geometry is 14.

TEST SECTION GEOMETRY AND MEASUREMENTS

The geometry for the present study is that of a modern power generation gas turbine. The cascade layout and dimensions are given in Figure 2 and Table 1, respectively. To simulate the disk cavity region, dimensions were extracted from engine drawings to create the leakage plenum shown in Figure 2 (right). The two endwall contours tested are also shown in this figure.

The cascade test section allows access for measurements to be made at any location throughout the passage as well as in the vicinity of the leakage flow-to-mainstream interface. Pressure tubes and transducers are used to characterize aerodynamic losses, while thermal anemometry is used to measure velocities, unsteadiness at the interface, and turbulence characteristics. Thermocouple probes are used to trace heated flow through the test section as well as extract from the data endwall adiabatic effectiveness values. Relevant data near the leakage-to-mainstream flow interface will be reported.

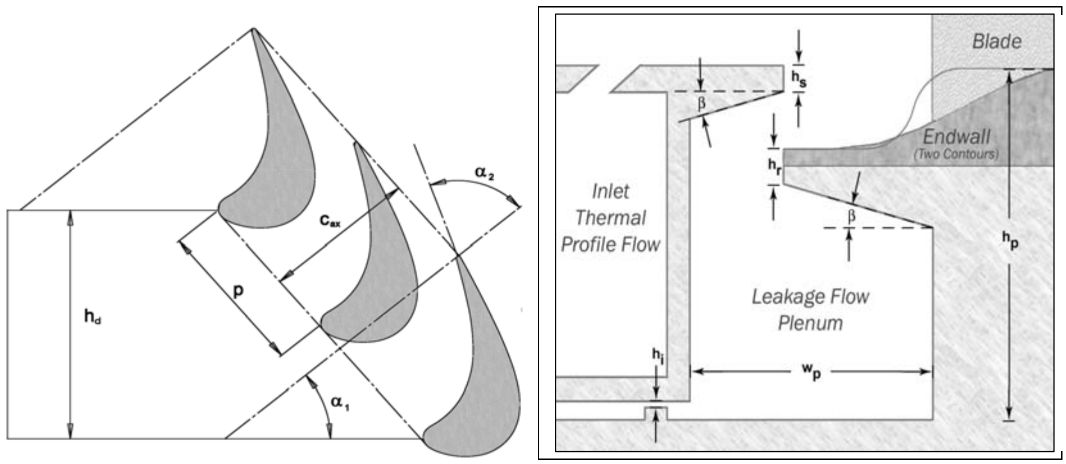

Figure 2. Cascade layout (Left) and Cross-section of leakage plenum including endwall contours (Right)

Table 1
Cascade and Leakage Plenum Dimensions

Cascade Dimensions	
Scale Factor:	14.06
Chord Length (c):	55.79 cm
Axial Chord (c_{ax}):	43.20 cm
Pitch (p):	42.10 cm
Blade Aspect Ratio:	1.20
Inlet Flow Angle (α_1):	40.0°
Outlet Flow Angle (α_2):	70.0°
Inlet Duct Height (h_d):	64.50 cm
Inlet Duct Width (w_d):	50.60 cm

Leakage Plenum Dimensions	
Stator Endwall Lip (h_s):	2.14 cm
Rotor Endwall Lip (h_r):	2.85 cm
Injection Slot (h_i):	0.50 cm
Plenum Height (h_p):	28.80 cm
Plenum Width (w_p):	21.60 cm
Chamfer Angles (β):	15.0°

COMPUTATIONAL METHODOLOGY

Also, results of a computational study are presented. The commercial CFD software FLUENT was used. A 3D model for each endwall contour design is created which replicates all features found in the linear cascade and leakage plenum geometries (Figure 3). The two grids contain roughly 2.8 x 10^6 cells each and are refined near boundary layers ($y^+ < 15$). All flow-related boundary conditions are prescribed based upon measurements from the linear cascade test section. Leakage flow rates tested include 0.5, 1.0, and 1.5% of the mainstream mass flow rate. The k-ω Shear Stress Transport model is used for turbulence calculations with turbulence intensities, dissipation rates, and length scales provided from cascade turbulence measurements. Mixing between the mainstream and leakage fluid is visible through study of static temperature vales over endwall-normal planes. The endwall surfaces are prescribed to be adiabatic, allowing for adiabatic effectiveness values to be calculated as well.

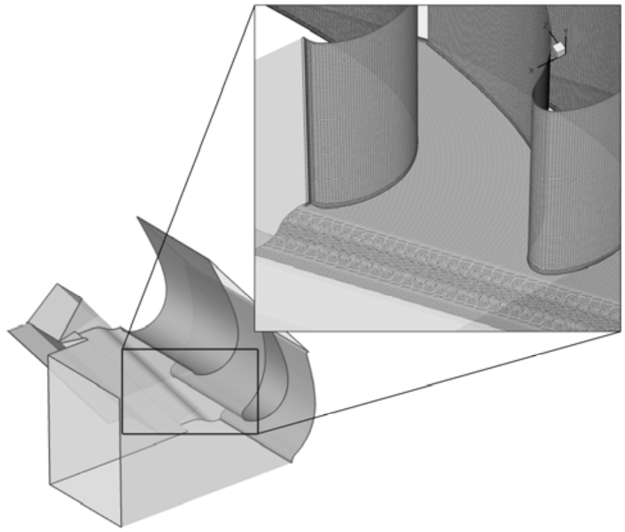

Figure 3. Computational domain showing close-up of surface grid density

RESULTS

Metrics to be compared between the two endwall designs include aerodynamic losses and adiabatic effectiveness distributions. Of particular interest regarding these designs is their susceptibility to unsteady behavior at the leakage-to-mainstream interface. This behavior has been viewed in the cascade test section using flow visualization and has no dependence on stator wakes or rotational conditions, as previously discussed in the literature. Time-averaged cascade measurements will be compared to computational results to gain a better understanding of behavior in the interface region and its effects on passage performance. Results of varying leakage flow rates will also be reported with an emphasis on each design's sensitivity to variations within the selected range.

REFERENCES

Bhavnani, S.H., Khodadadi, J.M., Goodling, J.S., and Waggott, J. [1992], An Experimental Study of Fluid Flow in Disk Cavities, *Trans. of ASME, J. of Turbomachinery*, Vol. 114, pp. 454-461.

Cao, C., Chew, J.W., Millington, P.R., and Hogg, S.I. [2003], Interaction of Rim Seal and Annulus Flows in an Axial Flow Turbine, *ASME Turbo Expo 2003,* GT2003-38368.

Dénos, R., Paniagua, G., [2002], Influence of the Hub Endwall Cavity Flow on the Time-Averaged and Time-Resolved Aero-Thermodynamics of an Axial HP Turbine Stage, *ASME Turbo Expo 2002*, GT2002-30185.

Ko, S.H., and Rhode, D.L. [1992], Thermal Details in a Rotor-Stator Cavity at Engine Conditions with a Mainstream, *Trans. of ASME, J. of Turbomachinery,* Vol. 114, pp. 446-453.

Marini, R., and Girgis, S. [2007], The Effect of Blade Leading Edge Platform Shape on Upstream Disk Cavity to Mainstream Flow Interaction of a High-Pressure Turbine Stage, *ASME Turbo Expo 2007,* GT2007-27429.

Ong, J.H.P., Miller, R.J. and Uchida, S. [2006], The Effect of Coolant Injection on the Endwall Flow of a High Pressure Turbine, *ASME Turbo Expo 2006*, GT2006-91060.

Owen, J.M. [1988], Air-Cooled Gas Turbine Disks: a Review of Recent Research, *Int. J. of Heat and Fluid Flow,* Vol. 9, No. 4, pp. 354-365.

Paniagua, G., Dénos, R., and Almeida, S. [2004], Effect of the Hub Endwall Cavity Flow on the Flow-Field of a Transonic High-Pressure Turbine, *ASME Turbo Expo 2004,* GT2004-53458.

Pau, M., Paniagua, G., Delhaye, D., de la Loma, A., and Ginibre, P. [2008], Aerothermal Impact of Stator-Rim Purge Flow and Rotor-Platform Film Cooling on a Transonic Turbine Stage, *ASME Turbo Expo 2008,* GT2008-51295.

Reid, K., Denton, J., Pullan, G., Curtis, E., Longley, J. [2005], The Interaction of Turbine Inter-Platform Leakage Flow with the Mainstream Flow, *ASME Turbo Expo 2005,* GT2005-68151.

Simon, T.W., and Piggush, J.D. [2006], Turbine Endwall Aerodynamics and Heat Transfer, *J. of Propulsion and Power,* Vol. 22, No. 2, pp. 301-312.