The Influence of Step and Fillet Shape on Nozzle Endwall Heat Transfer
JeongJu Kim, Heeyoon Chung, DongHo Rhee, HyungHee Cho

Abstract—There is a gap at combustor-turbine interface where leakage flow comes out to prevent hot gas ingestion into the gas turbine nozzle platform. The leakage flow protects the nozzle endwall surface from the hot gas coming from combustor exit. For controlling flow’s stream, the gap’s geometry is transformed by changing fillet radius size. During the operation, step configuration is occurred that was unintended between combustor-turbine platform interface caused by thermal expansion or mismatched assembly. In this study, CFD simulations were performed to investigate the effect of the fillet and step on heat transfer and film cooling effectiveness on the nozzle platform. The Reynolds-averaged Navier-stokes equation was solved with turbulence model, SST k-omega. With the fillet configuration, predicted film cooling effectiveness results indicated that fillet radius size influences to enhance film cooling effectiveness. Predicted film cooling effectiveness results at forward facing step configuration indicated that step height influences to enhance film cooling effectiveness. We suggested that designer change a combustor-turbine interface configuration which was varied by fillet radius size near endwall gap when there was a step at combustor-turbine interface. Gap shape was modified by increasing fillet radius size near nozzle endwall. Also, fillet radius and step height were interacted with the film cooling effectiveness and heat transfer on endwall surface.

Keywords—Gas turbine, film cooling effectiveness, endwall, fillet.

I. INTRODUCTION

FOR enhancing gas turbine thermal efficiency and output of power, turbine inlet temperature has increased. Combustion method has been developed to reduce NOx. Pre-mixed combustion method effected to reduce NOx; however, temperature on endwall has increased prior to past combustion method. Consequently, endwall increased heat loads on turbine components. Cooling methods consist of slot cooling and discrete hole for protecting endwall from hot gas stream. Slot is created at combustor-turbine interface gap for protecting endwall surface from hot gas. The interface gap is an area where cooling performance is improved.

This paper reports effects of step and fillet shape at combustor-turbine interface on endwall heat transfer. In addition, time-resolved vector and streamline predictions within the vane stagnation plane are presented. Film cooling effectiveness, Nusselt number, and Net Heat Flux Reduction are simulated by varying fillet radius size and step height size.

JeongJu Kim, Heeryoon Chung, and Hyung Hee Cho are with the Heat Transfer Lab, Yonsei University, 03722 South Korea (phone: (+82)-2-2123-7227; fax: (+82)-2-312-2159; e-mail: kimdanny55@gmail.com, justphy@yonsei.ac.kr, hhcho@yonsei.ac.kr).
Dong Ho Rhee is with Korea Aerospace Research Institute, Daejeon, 34133 South Korea (e-mail: rhee@kair.re.kr)

II. LITERATURE REVIEW

Several past studies have investigated the performance of purge flows from upstream gaps at combustor-turbine interface. Many studies have preceded film cooling from discrete holes and purge flow. Reference [1] measured secondary flows in a vane passage and combustor-turbine leakage flow and found cooling performance according to mass flow rate (MFR). Reference [2] found effect of leakage flows on endwall cooling.


Many studies [3]-[9] were performed about MFR on mid-passage gap and combustor-turbine interface gap. Only a few studies [10]-[12] have investigated about step configuration. The study reported in this paper seeks to understand film cooling effectiveness and heat transfer varying fillet radius size and step height size. In addition, the resulting film cooling effectiveness and heat transfer will be supported by time-resolved flow field predictions in the stagnation plane.
of the nozzle guide vane.

III. COMPUTATIONAL METHODS

Simulations of the combustor-turbine interface geometry were performed using the computational fluid dynamics software CFX [13]. A summary of the geometry was shown in Table I. The RANS equation was interpreted by SST k-omega turbulence model. The SST K-omega turbulence model [14] had shown reasonable prediction with experimental results in gas turbine [15], [16]. Transition turbulence model was Gamma Theta model modifies turbulent transport equations to simulate laminar, transition, and turbulence states in a fluid. The Numerical setup was described in Fig. 1. To predict the endwall heat transfer upstream of the platform, the computational grid was extended 1.77Cₚ upstream of the vane leading edge and the outflow boundary condition was extended 1.67Cₚ downstream of the trailing edge as shown in Fig. 1 (a).

![Fig. 1 Depictions of (a) computational domain, (b) the endwall grid](image)

### TABLE I

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Computational Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>x₀/₁₆th</td>
<td>6.2 × 10²</td>
</tr>
<tr>
<td>Lₐx</td>
<td>22.5</td>
</tr>
<tr>
<td>Cₐ</td>
<td>40</td>
</tr>
<tr>
<td>Pₐ/Cₐₜ</td>
<td>1.33</td>
</tr>
<tr>
<td>Sₐ/ₐₚ</td>
<td>0.98</td>
</tr>
<tr>
<td>W – Upstream slot width (mm)</td>
<td>0.025 × C</td>
</tr>
<tr>
<td>Slot length to width</td>
<td>1.9</td>
</tr>
<tr>
<td>Slot injection angle</td>
<td>90°</td>
</tr>
</tbody>
</table>

### TABLE II

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Computational Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Pₑth, Temperature 30.68 bar</td>
</tr>
<tr>
<td>Coolant</td>
<td>Tₑ, 837 K</td>
</tr>
<tr>
<td>Inlet</td>
<td>mₑ, MFR = 1%</td>
</tr>
<tr>
<td>Outlet</td>
<td>mₒₑthet, 0.3571 kg/s</td>
</tr>
</tbody>
</table>

A. CFD Mesh

The computational mesh is shown in Fig. 1 (b). The commercial grid was used by CFX Auto-mesh. The total number of elements in this geometry was around 5,000,000. The momentum, energy and turbulence equations were performed until the residual values of the computations converged. The convergence of residuals for x-momentum, y-momentum, and z-momentum were resolved to levels lower than 10⁻⁴. Area-averaged endwall temperature valued less than 0.1% over 500 iterations for satisfaction for convergence. After convergence, the mesh was adapted y⁺ values less than 1.

B. Boundary Condition

The rotational periodic boundary condition was applied to this domain and the outflow boundary condition is extended 1.67Cₚ downstream of the trailing edge. The boundary condition in this geometry was shown in Table II. In this study, the inlet temperature boundary condition had a temperature profile. The inlet temperature profile was referred to KARI (Korea Aerospace Research Institute).

![Fig. 2 Temperature inlet profile from combustor exit](image)

### TABLE III

<table>
<thead>
<tr>
<th>Fillet Radius (r/Cₚ) (%)</th>
<th>Step Height (h/Cₚ) (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2.2</td>
<td>2.2</td>
</tr>
<tr>
<td>4.4</td>
<td>4.4</td>
</tr>
<tr>
<td>6.6</td>
<td>6.6</td>
</tr>
</tbody>
</table>

The inlet temperature profile was calculated by combustor exit temperature distribution assuming pattern factor and RTDF. Fig. 2 shows temperature inlet profile. In this study, simulation was performed by using this inlet temperature profile.

IV. RESULTS AND DISCUSSION

The variables are shown in Table III. Steady 3D RANS predictions for endwall surface are performed where there is a slot at combustor-turbine interface. Film cooling effectiveness...
and heat transfer with slot geometry are simulated in this study. The predicted results are analyzed.

A. Film Cooling Effectiveness at Fillet Geometry

Efficiency for the numerical research was computed using definition in (1):

\[
\eta = \frac{T_{m} - T_{aw}}{T_{m} - T_{c}}
\]

(1)

\(T_m\) is a mainstream temperature, \(T_c\) is a coolant temperature, and \(T_{aw}\) is an adiabatic wall temperature on endwall surface.

In this study, film cooling effectiveness with fillet geometry and without fillet geometry is predicted on adiabatic condition. Predicted film cooling effectiveness contours are shown in Fig. 3. Film cooling effectiveness with fillet geometry appears to be more endwall cooling effects than without fillet geometry. Without fillet, coolant and hot stream are mixed because coolant flows upstream from slot. Coolant cannot protect on endwall surface that causes film cooling effectiveness is low on endwall without fillet. However, with fillet, fillet geometry controls the coolant direction to protect endwall flow so that film cooling effectiveness with fillet is higher than without fillet. Line plots of film cooling effectiveness augmentation in Fig. 4 were created by extracting data from Fig. 3. Film cooling effectiveness of each fillet radius sizes is made little difference. Fig. 3 shows that film cooling effectiveness is reduced around leading edge and passage between pressure side and suction side. One of the reasons for film cooling effectiveness is low is horseshoe vortex that is occurred by leading edge. Because of horseshoe vortex, fluid flows around leading which causes coolant cannot protect endwall surface from hot stream.

Another reason is passage vortex that leads fluids to move toward suction side as pressure difference between pressure side and suction side.

B. Film Cooling Effectiveness at Step & Fillet Geometry

Based on fillet geometry (\(r/C_x=6.6\%\)), film cooling effectiveness is predicted on step configuration. In Fig. 5, contours of film cooling effectiveness according to step height are shown. There is no big change; however, film cooling effectiveness is decreasing after passing the leading edge. Fig. 6 shows that line plots of film cooling effectiveness for four cases. The influence of film cooling effectiveness has increased until step height (\(\varepsilon/C_x=4.4\%\)) after then film cooling effectiveness has decreased. Step configuration causes pressure distribution change so that main stream flows differently comparing no step configuration. Main stream flows upstream because of step configuration. In case of step height (\(\varepsilon/C_x=4.4\%\)), main stream flows upstream and coolant flows along the fillet configuration. For relative pressure distribution, coolant is mixing with hot main stream less than no step configuration which causes enhancing film cooling effectiveness. However, in case of step height (\(\varepsilon/C_x=6.6\%\)), film cooling effectiveness is reduced compared to the step height (\(\varepsilon/C_x=4.4\%\)). Increasing step height causes to accelerate main stream velocity that coolant cannot be covered with endwall surface.

Fig. 3 Contours of film cooling effectiveness (a) Without fillet (b) With fillet (r/C_x=6.6\%)

Fig. 4 Lateral averaged film cooling effectiveness according to fillet radius size

Fig. 5, Contours of film cooling effectiveness according to step height
The trend of film cooling effectiveness was similar to changing fillet geometry. However, after step height (ε/Cx=6.6%), film cooling effectiveness had reduced. The tendency for upstream of coolant that coolant could not cover on endwall which effected to heat transfer on endwall. At step height (ε/Cx=6.6%), there was no hot gas injection into slot.

Overall, the steady RANS simulations showed film cooling effectiveness and heat transfer on step and fillet configuration. Film cooling effectiveness had increased at step geometry compared to no step geometry. To design at combustor-turbine interface, it was important to consider cooling performance and hot gas injection which effect to durability and fully gas turbine efficiency.

ACKNOWLEDGMENT

This study was supported by the aerospace research program (KA00157) of Korea Aerospace Research Institute (KARI) and the human resources development program (No. 20144030200560) of the Korean Institute of Energy Technology Evaluation and Planning (KETEP). Those programs are funded by the Korean government Ministry of Trade, Industry and Energy.

REFERENCES


**JeongJu Kim** received his B.S. degree from ChungAng University, Korea, in 2015. He is an integrated course candidate in Mechanical Engineering at Yonsei University. His current research interests are on the heat transfer in gas turbine.

**Heeyoon Chung** received his B.S. degree from Yonsei University, Korea, in 2013. He is an integrated course candidate in Mechanical Engineering at Yonsei University. His current research interests are on the heat transfer in gas turbine.

**DongHo Rhee** received his B.S. degree from Yonsei University, Korea, in 2007. He received Ph.D. (2013) from Yonsei University, Korea. Dr. Rhee is currently a researcher at the Korea Aerospace Research Institute, Deajeon, Korea

**HyungHee Cho** received his B.S. (1982) degree from Seoul National University, Korea. He received M.S. (1985) degree from Seoul National University and Ph.D. (1992) from Minnesota University, USA. Dr. Cho is currently a Professor at the school of Mechanical Engineering at Yonsei University in Seoul, Korea.