Elliptic Trailing Edge for a Turbine Blade: Aerodynamic and Aerothermal Effects

By Mahmoud Moustafa El-Gendi, Sang Wook Lee, Chang Yeol Joh, Geun Sik Lee, Chang Ho Son and Wui Jun Chung

1) School of Mechanical Engineering, University of Ulsan, Ulsan, South Korea
2) Mechanical Power and Energy Department, Faculty of Engineering, Minia University, Minia, Egypt

(Received April 24th, 2012)

Three-dimensional flow simulations through a turbine cascade are carried out for an exit isentropic Mach number of 0.79. The objective of the present study is to qualitatively and quantitatively clarify the effect of using an elliptic trailing edge (TE) instead of a circular one on the flow. Calculations are carried out using a parallel, turbulent, structured, single-block code and the delayed detached eddy simulation is employed for turbulence. We maintain the TE thickness (minor axis) and modify the other axis (major axis). Time-averaged pressure and heat transfer coefficient distributions along the blade are presented with more attention paid to the TE region. The results show that increasing the major-to-minor axis ratio decreases both the heat transfer coefficient and loss. Therefore, designers can adjust this ratio according to application needs.

Key Words: Turbine Flow, CFD, Trailing Edge, Base Pressure, Heat Transfer

Nomenclature

- a: major axis, normal to b
- ab0.5: a/b = 0.5 as shown in Fig. 1
- ab1: a/b = 1 as shown in Fig. 1
- ab2: a/b = 2 as shown in Fig. 1
- ARC: arc length along the blade from the stagnation point to the beginning of the TE on the pressure or suction side
- b: minor axis (=D)
- CC: cold case
- D: thickness of the TE (=7.43 mm)
- h_ref: a reference h for normalization (=1135 W/K/m²)
- h: heat transfer coefficient
- HC: hot case
- M: Mach number
- n: coordinate along the normal direction to the surface of the flat plate
- p: pressure
- PS: pressure side of the blade
- r: coordinate along the surface of the blade, originating from the stagnation point on the leading edge
- Re: Reynolds number
- S: coordinate along the surface of the blade, originating from the center of the TE as shown in Fig. 5
- SS: suction side of the blade
- T: temperature
- TE: trailing edge
- X: tangential axis to the camber line at the circular TE
- x: axial coordinate, originating at the stagnation point on the leading edge
- Y: transverse axis, normal to the X-axis
- y: pitch coordinate, normal to x

Subscripts

1: inlet
2: exit
is: isentropic
o: total
w: wall

1. Introduction

The main objective of turbomachinery designers is improving performance by increasing efficiency. Any reduction of efficiency caused by the flow is termed loss. The base pressure plays a vital role in trailing edge (TE) loss, which contributes to one-third of the total loss.1) Vito et al.2) devised a new inverse design method that uses a Navier-Stokes solver for flow analysis and a Euler solver for inverse design. Although this method is more accurate than previous methods that use a Euler solver for both, it excludes the TE because the pressure at the TE is affected by separated flow, not by the blade geometry. Therefore, we cannot use the inverse design method to improve the TE geometry. A common approach for this task is to undertake experimental or numerical experiments. Xu and Denton3) found that there is a direct proportionality between the TE thickness and TE loss. Herman et al.4) reached a similar conclusion, finding that the thicker the TE is, the higher the TE loss is. They also noticed that a round TE has larger TE loss than a sharp TE but less than that of a square TE, and approximately equivalent loss to that of a tapered TE. Heine mann and Bütefisch5) estimated the shedding frequency of several turbine cascades at different flow conditions. They noticed that the highest shedding frequency is obtained at the thinnest TE.
Cicatelli and Sieverding\textsuperscript{61} found a uniform pressure distribution at the TE of a turbine cascade at $M_{2,\text{in}} = 0.4$. Sondak and Dorney\textsuperscript{71} and Manna et al.\textsuperscript{83} numerically confirmed this distribution using the same cascade and flow conditions as Cicatelli and Sieverding.\textsuperscript{60} Sieverding et al.\textsuperscript{69} conducted experiments at $M_{2,\text{in}} = 0.79$ on a cascade with half the scale of that used in other research.\textsuperscript{6,8} They noticed that the TE pressure distribution is non-uniform. This non-uniformity decreases the average value of the base pressure and hence increases the TE loss.

Attaching a splitter plate on the TE can increase the base pressure.\textsuperscript{13} Layukallo et al.\textsuperscript{10} inserted tabs on a cylinder body to control the flow separation and reduce the drag coefficient. Matsuno et al.\textsuperscript{11} controlled a cylinder wake using a plasma actuator and optimized plasma actuator control parameters using the Taguchi method. El-Gendi et al.\textsuperscript{12} proposed a new method to control the vortex shedding using micro-tubes. On the other hand, the elliptical cylinder has a lower drag coefficient than that of the circular cylinder at the same frontal area, and the drag coefficient decreases with increasing major-to-minor axis ratio.\textsuperscript{13} Therefore, El-Gendi\textsuperscript{14} proposed an elliptic TE for a turbine cascade. However, El-Gendi\textsuperscript{14} did not investigate the heat transfer effect on this proposed TE.

In the present study, we perform a numerical simulation for the experimental results of Sieverding et al.\textsuperscript{9,15} and modify the TE geometry to the elliptic shape. We investigate the effect of this modification on the loss and heat transfer coefficient.

2. Numerical Method

2.1. Model employed in this study

We investigate two test cases in the present work: circular TE and elliptic TE. The turbine cascade with a circular TE (ab1) is the same as that used in Sieverding et al.\textsuperscript{9} For the elliptic TE case, we study two shapes: ab0.5, where the elliptic TE major axis is half the TE thickness, and ab2, where the elliptic TE major axis is twice the TE thickness. The circular TE and modified elliptic TE are shown in Fig. 1. To maintain the thickness of the elliptic TE to the same value as that of the circular one, the chord length of the elliptic cascade is modified. Table 1 shows a comparison between the dimensions of the circular and elliptic cascades.

2.2. Computational grid

The O-type grid is used in all cases to reduce the skew of the grid near the leading and trailing round edges. The grid has $413 \times 194 \times 50$ grid points; the total number of grid points is approximately $4.0 \times 10^6$. The minimum thickness of the grid cell on the blade is $0.002 \text{mm}$, which is equivalent to $y^+ \approx 1$. The grid spacing along the span direction is constant. The span length of the computational domain is $14.7 \text{mm}$, which is approximately twice the thickness of the TE and one-tenth that of the chord length.

2.3. Boundary conditions

For both geometric cases, we adopt two boundary condition cases: cold case (CC) and hot case (HC). The HC represents the case that exists in real jet engines where the flow is very hot and the blade temperature is approximately constant and lower than that of the flow using coolant. The CC represents the approximate case encountered in many wind tunnels and numerical simulations where the flow is approximate to the ambient temperature and there is no significant difference between the wall and flow temperature so that the blade is approximately isolated.

The boundary conditions of the test cases are shown in Table 2. The CC boundary conditions are exactly the same as those used by Sieverding et al.\textsuperscript{9} On the other hand, the HC has the same exit isentropic Mach number ($M_{2,\text{in}}$) and exit isentropic Reynolds number ($Re_{2,\text{in}}$), based on chord length for the ab1 case, as that of the CC. The inlet and exit boundary conditions are calculated using the method of characteristics. Along the solid wall, no slip boundary condition is implemented. A periodic condition is used in both pitch and span directions. In the span direction, the boundary conditions at the hub and tip are extrapolated from the interior and averaged. Then, this averaged value is imposed at hub and tip. The isothermal boundary condition is used along the wall in the HC, and the adiabatic boundary condition is used in the CC. Although the temperatures ($T_{\text{in}}$ and $T_{\text{at}}$) do not represent real jet engine temperatures, its
ratio \( T_{o1}/T_w \simeq 1.43 \) represents the temperature ratio in real jet engines.\(^{16}\)

### 2.4. Numerical scheme

Calculations are carried out using a parallel, structured, single-block code, where the Navier-Stokes equations are discretized using the cell vertex method. In this code, the lower upper symmetric Gauss Seidel (LUSGS) method is employed along with the dual time method in order to obtain time-accurate results for unsteady calculations. In addition, the Roe scheme with E-fix is used to calculate inviscid numerical fluxes, where the second-order accuracy is achieved using the MUSCL scheme with the Van Albada flux limiter. On the other hand, the viscous fluxes are calculated by central differencing.

The calculations are carried out on a eight-processor Linux cluster using five inner iterations. The time-averaged results took 10,000 iterations. Every iteration has a dimensionless time step, \( \Delta t^* = 105.42 \times 10^{-6} \), and a physical time step, \( \Delta t = 3.33 \times 10^{-7} \text{s} \), which is equivalent to \( \text{CFL} \simeq 100 \). We think that this number of iterations is sufficient because there are no significant differences in results when numbers are increased. The time-averaged results are also averaged along the span.

### 2.5. Turbulence model

As a turbulence model, the delayed detached eddy simulation (DDES)\(^{17}\) method is adopted in the present work. It is a hybrid scheme that functions as a Reynolds-Averaged Navier-Stokes (RANS) model near the wall and as a large eddy simulation (LES) model in the region away from the wall. Unlike RANS models, the DDES has the capability to provide more accurate results for unsteady flows. Unlike LES, the DDES does not require a very fine grid near the wall. To estimate the boundary layer region at which RANS is applied, DDES depends not only on the grid spacing but also on the flow showing that the DDES has an advantage over detached eddy simulation (DES).

### 3. Results and Discussion

#### 3.1. Cold case

##### 3.1.1. Three-dimensional effect

Although the problem is two-dimensional, we prefer to calculate it three-dimensional to accurately include the effect of turbulence.\(^{18-20}\) Figure 2 shows three-dimensional dimensionless vorticity isosurfaces near the circular TE. The resolved turbulence and three-dimensionality are clear thanks to the DDES turbulent model and three-dimensional simulation.

##### 3.1.2. Pressure distribution along the blade

Figure 3 shows the time-averaged dimensionless pressure distribution around the blade along with experimental data.\(^{9}\)

![Fig. 2. Dimensionless vorticity isosurfaces.](image)

![Fig. 3. Distributions of time-averaged dimensionless pressure along the blade; Exp. carried out by Sieverding et al.\(^{9}\)](image)

![Fig. 4. Boundary layers.](image)
The abscissa "r" represents a coordinate length along the surface of the blade. An absolute value of r means the distance from the stagnation point on the leading edge, and the sign of r indicates the side of the blade. That is, a positive value corresponds to the suction side (SS), and a negative value to the pressure side (PS).

The numerical result of the circular case shows good agreement with the experimental data. The pressure distributions of the elliptic cases are very close to that of the circular case except at \( r/\text{ARC} \approx 0.45 \), showing that our modification maintains the same blade load.

### 3.1.3. Boundary layers

Next, we compare the velocity distribution in boundary layers on the PS and SS, i.e., the distribution along the perpendicular direction to the surface on the PS at the point \( r/\text{ARC} = -0.94 \), and on the SS at the point \( r/\text{ARC} = 0.95 \).

Figure 4 shows the time-averaged velocity distributions of our computational results for both cases, and the experiment result\(^9\) for the \( ab_1 \) case. The computational results of the \( ab_1 \) case agree well with the experimental data.\(^9\) Similar to static pressure along the blade, the velocity distribution is almost identical for all cases studied.

### 3.1.4. Vortex shedding frequency

Spectral analysis is performed for the pressure fluctuations at midspan on the TE. In \( ab_1 \), there was a predominant vortex shedding frequency of 7.45 kHz, which shows reasonable agreement with an experimentally obtained frequency of 7.37\(^9\) or 7.6 kHz.\(^15\) On the other hand, the predominant vortex shedding frequency is 7.81 kHz for \( ab_0.5 \) and 11.27 kHz for \( ab_2 \). Therefore, the circular TE cascade has lower frequency than that of the elliptic one.

### 3.1.5. Time-averaged pressure in the TE region

The contours of time-averaged pressure normalized by the stagnation pressure at the inlet (\( p/p_o \)) are shown in Fig. 5. The abscissa "S" represents the length along the blade surface, where the negative sign refers to the direction from the TE center (\( S = 0 \)) toward the PS, and the positive sign toward the SS and \( D \) is the TE thickness. All cases studied have two pressure minima next to the beginning of the TE and \( ab_1 \) and \( ab_0.5 \) have an additional minimum region near \( S/D = 0 \).

Both time-averaged numerical and experimental\(^9\) pressure distributions normalized by the inlet total pressure, \( p/p_o \), along the TE surface are shown in Fig. 6. The experimental data and numerical results of the circular TE agree
qualitatively with small discrepancy remaining. We think that this discrepancy is caused by the inability of the numerical code including the turbulence model to predict the pressure distribution accurately in this separated flow region.

The numerical non-uniform pressure distribution of the circular TE along the TE surface confirms the experimental data.\(^9\) This non-uniform pressure distribution is remedied by the elliptic TE, \(ab_2\), as shown in Fig. 6. By comparing Figs. 5 and 6, it can be seen that in the \(ab_0.5\) case (Fig. 5(a)), the TE begins at \(S/D \approx \pm 0.65\) and the two pressure minima are located at \(S/D \approx \pm 0.6\). In the \(ab_1\) case (Fig. 5(b)), the trailing edge begins at \(S/D = \pm 0.75\) and two pressure minima are located at \(S/D \approx \pm 0.66\). In the \(ab_2\) case (Fig. 5(c)), the TE begins at \(S/D \approx \pm 1.2\), and the pressure minima are located at \(S/D \approx \pm 1\). The shear layers almost separate close to these two pressure minima. Therefore, increasing the \(a/b\) ratio increases the distance between the separation points and the beginning of the TE which delays the separation. In addition, it is observed that \(ab_2\) has a recovered pressure distribution and there is no minimum pressure near \(S/D = 0\).

3.1.6. Instantaneous dimensionless vorticity contours in the TE region

Figure 7 shows the instantaneous dimensionless vorticity contours behind the TE in all cases. The separated shear layers from both the PS and SS interact with each other. This interaction between the two separated shear layers forms the vortex shedding.\(^{21}\) Therefore, preventing this interaction for example, using a splitter plate, prevents the forming of the vortex shedding.\(^{22}\) As shown in Fig. 7, increasing the major-to-minor axis ratio delays this interaction. Thus, there is no minimum pressure region just behind the TE in the \(ab_2\) case as shown in Fig. 5(c). In addition, it is expected that this delay will alleviate the bad effect of vortex shedding by decreasing the loss resulting from the vortex shedding.

3.1.7. Entropy distribution

Since entropy represents loss,\(^1\) Fig. 8 shows the comparison of entropy distributions at \(X/D = 2.5\). The angle between the \(X\)-axis and the axial direction, \(x\)-axis, is 66°. The origin of the \(X\) and \(Y\) axes is taken at \(S/D = 0.0\) of the circular TE (Fig. 5(b)). Therefore, the \(Y/D\) location in the computational domain is the same for all cases studied. The entropy decreases as a result of increasing the \(a/b\) ratio. We estimate the average value of \((S - S_0)\) along \(Y/D\) for all cases using the trapezoidal rule. We found that the \(ab_2\) case
has the lowest average value of entropy. In addition, the average entropy value of the $ab1$ case is 2.3 times the $ab2$ case, and in the $ab0.5$ case, 2.4 times that of the $ab2$ case.

### 3.2. Hot case

#### 3.2.1. Heat transfer validation

Before the main calculations, we calculate test problems to validate our code for the HC.

We simulate the turbulent boundary layer on a heated flat plate. The calculated results are compared with the experimental data measured by Bell.\(^{23}\) The velocity and temperature in the uniform flow are 17.95 m/s and 20.3°C, respectively. Further, the wall temperature is 46.3°C. Figure 9 shows the velocity and static temperature distributions at a point 1.53 m from the leading edge. Both velocity and static temperature distributions show good agreement with the experimental data\(^{23}\) which validates our code.

To test the reliability of our results in the hot case, we simulate the flow around the Mark II blade whose TE is modified to a circular shape. This blade has been investigated numerically\(^{16,24}\) and experimentally.\(^{25,26}\) The boundary conditions used are the same as those used by Luo and Lakshminarayana.\(^{16}\)

The total inlet temperature ($T_{in} = 772$ K) and wall temperature ($T_w = 540$ K) are the same as those of the HC. Therefore their ratio ($T_{in}/T_w \approx 1.43$) is the same as that of the HC. The O-type grid is used in this validation test; the total number of grid points is the same as that used in the main calculations for the HC and CC. Figure 10 shows the distributions of the pressure and heat transfer coefficient, where LExp., LCFD and PCFD represent Luo and Lakshminarayana’s experimental results,\(^{16}\) their computational results\(^{16}\) and the current computational results, respectively. Pressure is normalized by the total pressure at the inlet ($P_{o1}$). In addition, the flow is tripped at $r/ARC \approx -0.09$ on the PS in the PCFD.

As shown in Fig. 10, the pressure distribution of the current computational result agrees reasonably well with that of the experimental result, LExp. For the velocity coefficient, the result of the PCFD falls within the scattering data of the LExp.\(^{16}\) and has better agreement with LExp.\(^{16}\) than Luo and Lakshminarayana’s computational result, LCFD.\(^{16}\)

On the whole, the computational results in this test problem found using the current code agree reasonably with the experimental data, which validates the current code for turbine cascade problems and increases the reliability of the HC results.

#### 3.2.2. Heat transfer coefficient

Figure 11 shows the heat coefficient along the blade surface. The heat transfer coefficient distributions of the elliptic cases are very close to that of the circular case. The distribution behavior of the heat transfer coefficient along the PS and SS are similar. The heat transfer coefficient has its high-
present calculation, the DDES and three-dimensional simulation were used to treat turbulence accurately through the turbine cascade. The present numerical results were compared with the experimental data of Sieverding et al., Bell, and Luo and Lakshminarayana, and showed good agreement. We proposed an elliptic TE instead of a circular TE as a new, practical shape for the TE geometry to remedy the non-uniform pressure distribution at the circular TE. The TE thickness (minor axis) was the same for all cases studied.

Both the loss and heat transfer coefficient decreased as a result of increasing the major-to-minor axis ratio. On the other hand, this axis ratio effect on other quantities such as blade load was negligible. Therefore, although the circular TE is the default shape for the TE, we do not think that it is always the best choice. Designers can optimize the gas turbine performance by choosing the TE shape that can withstand a specific temperature and estimate the efficiency. The temperature can then be changed and procedures repeated until the best efficiency is reached.

Acknowledgments

This work was supported by the 2012 Research Fund of the University of Ulsan.

References

1) Denton, J. D.: Loss Mechanisms in Turbomachines, ASME J. Turbomachinery, 115 (1993), pp. 621–656.
2) Vito, L., Braembussche, R. A. and Deconinck, H.: A Novel Two-Dimensional Viscous Inverse Design Method for Turbomachinery Blading, ASME J. Turbomachinery, 125 (2003), pp. 310–316.
3) Xu, L. and Denton, J. D.: The Base Pressure and Loss of a Family of Four Turbine Blades, ASME J. Turbomachinery, 110 (1988), pp. 9–17.
4) Herman, W., Prust, J. and Helon, R. M.: Effect of Trailing-Edge Geometry and Thickness on the Performance of Turbine Stator Blading, NASA TN D-6637, 1972.
5) Heinemann, J. and Büttrich, K. A.: Determination of the Vortex Shedding Frequency of Cascades with Different Trailing Edge Thicknesses, Tech. Rep., AGARD-CP-227, Unsteady Aerodynamics, Paper 11, 1997.
6) Cicali, G. and Sieverding, C. H.: The Effect of Vortex Shedding on the Unsteady Pressure Distribution around the Trailing Edge of a Turbine Blade, ASME J. Turbomachinery, 119 (1997), pp. 810–819.
7) Sondak, D. L. and Dorney, D. J.: Vortex Shedding in a Turbine Cascade, AIAA Paper 98-3573; AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, 34th, Cleveland, OH, 1998.
8) Manna, M., Mulas, M. and Cicali, G.: Vortex Shedding behind a Blunt Trailing Edge Turbine Blade, Int. J. Turbo Jet Engine, 14 (1997), pp. 145–157.
9) Sieverding, C. H., Richard, H. and Desse, J.: Turbine Blade Trailing Edge Flow Characteristics at High Subsonic Outlet Mach Number, ASME J. Turbomachinery, 125 (2003), pp. 298–309.
10) Layukalo, T., Hayashi, D. and Nakamura, Y.: Passive Separation Control on a Body at Transonic Speed, Trans. Jpn. Soc. Aeronaut. Space Sci., 44 (2002), pp. 229–237.
11) Matsuno, T., Ota, K., Kanatani, T. and Kawazoe, H.: Optimal and Robust Design of Plasma Actuator for the Control of a Cylinder Wake, Proceedings of KSAS-JSASS Joint International Symposium on Aerospace Engineering, 2008, pp. 115–119.
12) El-Gendi, M., Ibrahim, M., Mori, K. and Nakamura, Y.: Novel Flow Control Method of Vortex Shedding for a Turbine Blade, Trans. Jpn. Soc. Aeronaut. Space Sci., 53 (2010), pp. 122–129.
13) White, F. M.: *Fluid Mechanics*, 4th ed., McGraw-Hill, Seoul, 1999, pp. 457–459.
14) El-Gendi, M.: Proposed Trailing Edge Geometry for a Turbine Cascade, Proceedings of International Symposium on Marine Engineering and Technology, 2011, pp. 231–234.
15) Sieverding, C. H., Ottolia, D., Bagner, C., Comadore, A., Broackert, J. F. and Desse, J.: Unsteady Turbine Blade Wake Characteristics, *ASME J. Turbomachinery*, 126 (2004), pp. 551–559.
16) Luo, J. and Lakshminarayana, B.: Numerical Simulation of Turbine Blade Boundary Layer and Heat Transfer and Assessment of Turbulence Models, *ASME J. Turbomachinery*, 119 (1997), pp. 794–801.
17) O’Callaghan, J. J. and Kurosaka, M.: Vortex-Induced Energy Separation in Shear Flows, *AIAA J.*, 31 (1993), pp. 1157–1159.
18) Wilcox, D. C.: *Turbulence Modeling for CFD*, 3rd ed, DCW Industries, Inc., California, 2006, pp. 2–8.
19) Schmidt, S. and Thiele, F.: Detached Eddy Simulation of Flow around A-Airfoil, *Flow Turbulence Combust.*, 71 (2003), pp. 261–278.
20) Spalart, P. R.: Detached-Eddy Simulation, *Annu. Rev. Fluid Mech.*, 41 (2009), pp. 181–202.
21) Blevins, R. D.: *Flow-Induced Vibration*, Krieger Publishing Company, Malabar, Florida, 2001, pp. 43–55.
22) Kamphuis, J. W.: *Hydrodynamics around Cylindrical Structures*, World Scientific Publishing Co., Singapore, 2006, pp. 6–9.
23) Bell, J. B.: Boundary Layer on Heated Plate at Constant Pressure, Proceedings of Computation of Turbulent Boundary Layer, Vol. 2, AFOSR-Stanford, Stanford, 1968, pp. 282–296.
24) Rahman, F., Visser, J. A. and Morris, R. M.: Capturing Sudden Increase in Heat Transfer on the Suction Side of a Turbine Blade using a Navier-Stokes Solver, *ASME J. Turbomachinery*, 127 (2005), pp. 552–556.
25) Nealy, D. A., Mihelc, M. S., Hylton, L. D. and Gladden, H. J.: Measurements of Heat Transfer Distribution over the Surfaces of Highly Loaded Turbine Nozzle Guide Vanes, *ASME J. Turbomachinery*, 106 (1984), pp. 149–158.
26) Hylton, L. D., Mihelc, M. S., Turner, E. R., Nealy, D. A. and York, R. E.: Analytical and Experimental Evaluation of the Heat Transfer Distribution over the Surface of Turbine Vanes, NASA CR-168015, 1983.