Numerical and Experimental Studies on Separated Boundary Layers over Ultra-High Lift Low-Pressure Turbine Cascade Airfoils with Variable Solidity: Effects of Free-stream Turbulence

FUNAZAKI Ken-ichi, YAMADA Kazutoyo, CHIBA Yasuhiro and TANAKA Nozomi
Department of Mechanical Engineering
Iwate University
3-5 Ueda 4, Morioka, Iwate 020-8550, JAPAN

ABSTRACT
This paper deals with LES investigation, along with measurements, on the interaction between inlet freestream turbulence and boundary layers with separation bubble over ultra-high lift low-pressure turbine airfoils. The cross section of the test airfoils is typical for highly-loaded LP turbines for civil aeroengines. The solidity of the cascade can be reduced by increasing the airfoil pitch by at least 25%, while maintaining the throat in the blade-to-blade passage. Reynolds number examined is 57,000, based on chord length and averaged exit velocity. Free-stream turbulence is about 0.85% (no grid condition) and 2.1% (with grid condition).

Hot-wire probe measurements of the boundary layer are carried out to obtain time-averaged and time-resolved characteristics of the boundary layers under the influence of the freestream turbulence. A newly developed probe positioning tool, which is installed downstream of the cascade with minimal blockage, enables precise probe positioning along lines normal to the airfoil surface.

Numerical analysis based on high-resolution LES (Large-Eddy Simulation) is executed to enhance the understanding of the flow field around the Ultra-High Lift and High Lift LP turbine airfoils. Emphasis is placed on the relationship of inherent instability of the shear layer of the separation bubble and the free-stream turbulence. Standard Smagorinsky model is employed for subgrid scale modeling. The flow solver used is an in-house code that was originally developed by one of the authors as FVM (Finite Volume Method)-based fully implicit and time-accurate Reynolds-Averaged Navier-Stokes code. Homogeneous isotropic turbulence created with SNGR (Stochastic Noise Generation and Radiation) method using von Karman-Pao turbulent energy spectrum is applied in the present study for the emulation of inlet turbulence.

Nomenclature

- $C$: blade chord length
- $C_{ax}$: blade axial chord length
- $C_p$: static pressure coefficient
- $E(k_n)$: turbulent energy spectrum
- $H$: helicity
- $H_{12}$: shape factor
- $k_n$: wave number vector for n-th mode ($= (k_1', k_2', k_3')$)
- $k_n'$: unit vector normal to $k_n$ ($= (k_1', k_2', k_3')$)
- $L$: turbulence length scale
- $N$: number of modes
- $N_x, N_y$: data size, number of realizations
- $P_{01}, P_{02}$: inlet and outlet stagnation pressure
- $Re_x$: Reynolds numbers based on chord length and exit averaged velocity
- $Tu_{in}$: inlet turbulence intensity
- $U_{in}, U_{2}$: inlet and exit averaged velocities
- $x, x_s$: axial direction, distance along the blade surface
- $y, y_n$: tangential direction, normal direction to the surface
- $\delta_1, \delta_2$: displacement and momentum thickness using time-averaged velocity
- $\omega$: vorticity vector
- $\Delta^+_{x}$, $\Delta^+_{y}$: viscous wall unit based on streamwise grid spacing
- $\Delta^+_{x}$, $\Delta^+_{y}$: viscous wall unit based on heightwise grid spacing
- $\Delta^+_{z}$: viscous wall unit based on spanwise grid spacing

Subscript

- in, 2: inlet, outlet

Abbreviation

- FT: Freestream Turbulence
- HL: High Lift
- SNGR: Stochastic Noise Generation and Radiation
- UHL: Ultra -High Lift

1. INTRODUCTION
Great attention has been paid to the development of ultra high lift (UHL) low-pressure turbine (LPT) blades for aeroengines and a number of relevant studies have been made during the last decade experimentally or numerically, sometimes both. This research trend has been motivated by the expectation that the realization of such turbine blades enables drastic reduction of the blade count, eventually leading to lighter aeroengines with low total life cycle cost. A significant obstacle that has to be overcome or alleviated to develop UHL turbine blades is considerable loss increase due to flow separation on the blade suction surface, particularly at cruise low Reynolds number condition.
Effects of freestream turbulence (FT) upon the behavior of the separation bubble have also been investigated by many researchers. For example, Shyne et al. [1], Volino and Hultgren [2], Volino [3, 4] conducted detailed measurements of separated boundary layers influenced by free-stream turbulence with various turbulence intensities. Suzen et al. [5] examined the impact of elevated free-stream turbulence from the viewpoint of predicting its transitional characteristics or separation-induced bypass transition. RANS-type approaches are still popular in predicting this type of bypass transition. Suzen et al. [6] also conducted very comprehensive studies on transitional flows in LPT using P&W Pak-B airfoil under several Reynolds numbers and inlet freestream turbulence intensities. They collected a number of relevant experimental data and made detailed comparisons between those experimental data and the numerical results obtained by their RANS code using an intermittency transport equation to predict the bypass transition. It has turned out from the comparisons that their approach has a great capability of predicting transitional flows and it is very promising as design tool. On the other hand, despite its strict requirement on computational resources, LES approach [7] [8, 9] [10], as well as DNS approach [11], is now becoming a commonly used method to calculate unsteady three-dimensional flows around LPT airfoils containing large separation. Since LES approach is inherently an unsteady analysis and capable of time-accurate capturing of highly resolved vortical motions associated with the separated flows, it does not necessitate any modeling of transition, regardless of disturbance-induced or separation-induced one. Therefore, it is surely worth pursuing further possibility of LES approach in practical applications, along with accumulation of relevant experimental data through well-organized measurements.

This paper deals with LES-based numerical investigations as well as the experiment on the flow field around UHL and HL LPT cascades. The focus of the present study is on the effects of FT. In order to create inlet turbulent flow field as inlet boundary condition, SNGR (Stochastic Noise Generation and Radiation) method using von Karman-Pao turbulent energy spectrum is employed in this study. Hot-wire probe measurements are made over the blade suction surface to investigate unsteady behavior of the separated boundary layer, although only time-averaged data will appear in this paper.

2. Experimental Setup
2.1 Test Section and Cascade

The test apparatus in the present study was almost the same as that used in the previous study done by Funazaki et al. [12]. Figure 1 shows a cross-section of the test apparatus that contains the test linear cascade and turbulence grid. One may find the belt in this apparatus, which is because it was originally designed and manufactured for the investigation of wake-blade interaction using horizontal bars on the belt [12]. The bars were removed from the belt in this study. The linear cascade consisted of seven airfoils whose cross section was a typical profile of modern aeroengine LPT. Geometrical information on the airfoil and cascade is listed in Table 1. There were two instrumented brass airfoils in the cascade to measure static pressure distributions over the suction and pressure surfaces, respectively. Each of the airfoils, which are designated Blade #3 and Blade #4 in Figure 2, had 30 pressure holes of 0.5mm diameter on its suction or pressure surface, as shown in Figure 3. The rest of the airfoils in the cascade were made of aluminum. The pitch of the cascade was changeable by use of spacing plates with different sizes. The cascade could be slightly tilted back and forth in order to adjust incidence of the airfoils. Blade #4, being situated almost in the center of the duct, was the target airfoil whose aerodynamic performance and boundary layer development were examined in detail. Since the pitchwise periodicity of the cascade was very critical, particularly in the present configuration where the target airfoil was located between the instrumented airfoils, manipulation of the two adjusting plates shown in Figure 1 was repeated until satisfactory periodicity of the cascade exit flow field was achieved. The measured static pressure distributions, as will be shown later, provide a rough verification of the achievement of the pitchwise periodicity in the cascade, at least, for no-grid case.
The turbulence grid was installed about 640 mm upstream of the cascade, as shown in Figure 1. The grid was composed of wires of 2mm diameter with 16mm mesh width. The turbulence grid was not parallel to the cascade due to the mechanical constraint of the test apparatus, eventually causing more or less non-uniform turbulence intensity distribution along the pitchwise direction of the cascade. Again, it seems from the static pressure distributions that the pitchwise periodicity of the cascade also held under the elevated FT case. However, interpretation of the data for that case seems to require some caution. Besides, as mentioned above, the belts of the wake generator shown in Figure 1 were not actually used to produce incoming wakes, while they were utilized to generate a triggering pulse for the data acquisition.

### 2.2 Instruments

Inlet and outlet total pressures were measured by use of two miniature Pitot tubes, where both probes were placed on the midspan plane. The Pitot tube, penetrating through the blade-to-blade passage from the trailing edge side, measured inlet total pressure \( P_{01} \) at the place 72mm upstream of the leading edge of Blade #4. The other Pitot tube for the outlet pressure measurement was placed 15mm downstream of the trailing edge of the airfoils in the axial direction. The probe was aligned with the exit flow direction from the cascade, using a tuft as flow indicator. A PC-controlled traversing device moved the downstream Pitot tube over two pitches to calculate the averaged exit velocity.

Figure 2 also depicts the zone of the hot-wire measurement. The measurement zone extended from \( x/C_{ax} = 0.5 \) to the blade trailing edge in the streamwise direction and from \( y_{ax} = 0.2 \text{mm} \) to 10mm in the direction normal to the blade suction surface. A newly developed probe positioning machine, which was installed downstream of the cascade with minimal blockage, realized the automatic and precise probe positioning along the normal lines to the airfoil surface. The instantaneous velocity data acquired by a single hot-wire probe (Dantec 55P11) connected to Kanomax CTA (Constant Temperature Anemometer) unit were converted and stored in the PC with sampling frequency of 20kHz. The size of each of the realizations, \( N_j \), was \( 0.5 \times 50 \times \ldots \) word. From these velocity data, \( \tilde{u}_k \) \( (k=1,...,N_j) \) ensemble-averaged velocity \( \tilde{u} \) was calculated by the following equations.

\[
\tilde{u}(x, y_j; j\Delta t) = \frac{1}{N_j} \sum_{j=1}^{N_j} u_k(x, y_j; j\Delta t), \quad (1)
\]

\[
j = 0, \ldots, N_j - 1,
\]

where \( \Delta t \) was data sampling interval \( (\approx 50 \text{\( \mu \)}s) \), \( N_j \) was the number of the realizations used for ensemble averaging \( (\approx 100) \). The time-averaged velocity was calculated from the arithmetic average of \( \tilde{u} \). The outer edge of the boundary layer was defined to be the location where the time-averaged streamwise velocity reached 98% of the maximum velocity \( U_{\infty} \) attained within the measurement range. Time-averaged boundary layer integral parameters such as displacement thickness \( \delta \) or \( \delta_t \) were then calculated using the time-averaged velocity. The shape factor \( H_{12} \) was determined by

\[
H_{12} = \frac{\delta}{\delta_t}, \quad (2)
\]

### 2.3 Uncertainty Analysis

Uncertainty associated with the pneumatic measurement was governed by the accuracy of the pressure transducers. Most severe cases in terms of the measurement accuracy were for low-speed flow conditions \( (U_{\infty} = 4.9 \text{m/s}) \). As mentioned above, the accurate pressure transducer with \( \pm 0.5 \text{Pa} \) was used for these cases. The standard procedure of Kline and McClintock [13] determined that the uncertainty of the inlet velocity \( U_{\infty} \) was about \( \pm 1.7\% \). Uncertainty of the static pressure coefficient turned out to be about \( \pm 3.5\% \) around the peak region of the coefficient on the suction surface.

The uncertainty associated with the pneumatic measurement also determined the accuracy of the hot-wire probe measurements because the probe calibration relied on the velocity measured with the Pitot tube while any other errors such as the error due to the curve fitting remained small (less than 1%). Therefore the uncertainty of the hot-wire probe measurement was estimated to be about \( \pm 2\% \).

### 2.4 Test Conditions

This study examined the flow fields for two solidity cases with the fixed exit Reynolds number \( Re_e = 5.7 \times 10^4 \), where the Reynolds number was defined as follows.

\[
Re_e = \frac{C U_2^2}{V}, \quad (3)
\]

This relatively low Reynolds number was first chosen by Funazaki et al. [12] to explore a lower limit of operating Reynolds number by taking full advantage of wake passing over the airfoils. The tested solidities of the cascade were expressed in terms of relative reduction of the solidity from the original, and they were -14.2\% (designated \( S=15 \) hereafter) and -23.6\% (\( S=25 \)). The former will be referred to as HL (High Lift) condition and the latter will be referred to as UHL condition. The inlet turbulence intensity enhanced with the turbulence grid turned out to be 2.1\%, while the background turbulence level with no grid was about 0.85\%. The measurement was made about 30\% \( C_{ax} \) upstream of the Blade #4.
leading edge. Figure 3 demonstrates the acquired velocity data with grid and no grid conditions.

Figure 4 Inlet velocity fluctuations with and without turbulence grid

3. NUMERICAL SIMULATION

3.1 Flow Solver

The flow solver used in this study is an in-house LES code that was originally developed by one of the authors (Yamada) as a time-accurate three dimensional unsteady compressible RANS (Reynolds Averaged Navier-Stokes equations) solver [14]. It was a fully implicit code based on cell-centered Finite Volume Method using a high-resolution upwind scheme with MUSCL (Monotone Upstream-centered Schemes for Conservation Laws) interpolation. The code is able to deal with multi-blocked computational grid system taking advantage of MPI (Message Passing Interface). In order to make any intrusion of numerical viscosity into the analysis as small as possible, a fourth-order central compact scheme was implemented in place of the upwind scheme, with filtering of 10-th order accuracy for securing numerical stability. Three-point backward difference was used to approximate time-derivative and an inner iteration, so-called Newton iteration, was employed to attain second order accuracy in time. The viscous fluxes were determined by the second-order central difference along with the divergence theorem. The standard Smagorinsky model is employed in this code for subgrid scale turbulence modeling with van Driest dumping function, where Smagorinsky constant is 0.1. Application of Smagorinsky model based LES to the analysis of transitional boundary layer may be open to dispute, however, several examples have proved that reasonable results can be obtained for computational grid system with high quality and a large number of grid points [15].

3.2 Computational Domain and Grid System

Figure 5 shows the computational grid for S-25 solidity case. The grid system, containing more than 40 subdomains, basically consisted of H-type grid sub-systems. They were created by use of Gridgen Ver. 15 (Pointwise). Since the present study examined two solidity cases, the grid systems used for these two cases differed from each other. Detailed comparisons between the two grid systems are shown in Table 2. Note that the span of the computational domain was set to be 10% of the axial chord length, aiming to capture a highly resolved three-dimensional unsteady behaviors of the separation bubble on the blade suction surface with minimum allocation of the computer resources.

In addition to the grid point clustering near the surface, great care was given here to streamwise grid resolution over the region ranging from 55% - 60% C<sub>ax</sub> to the trailing edge where the separation bubble existed. In fact, high grid-resolution in the streamwise direction (streamwise small and uniform viscous wall unit Δξ<sup>+</sup>) was found to be vital to capture transitional nature of separation bubble, resulting in more accurate prediction of reattachment of the separation. The grid generation followed a guidance given by Matsuura and Kato [9], who stated that the minimum streamwise grid spacing should be less than 10 times minimum normal-to-wall grid spacing. In addition, large number of grid points were also allocated near the inlet boundary for minimizing the turbulence decay, otherwise the inlet turbulence quickly diminished before reaching the blade leading edge.

<table>
<thead>
<tr>
<th>Tested solidity case</th>
<th>Number of cells</th>
<th>Total</th>
<th>6.7millions</th>
<th>24millions</th>
</tr>
</thead>
<tbody>
<tr>
<td>S-15</td>
<td>Suction surface</td>
<td>511</td>
<td>1421</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Pressure surface</td>
<td>229</td>
<td>677</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Pitch direction</td>
<td>199</td>
<td>241</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Span direction</td>
<td>39</td>
<td>49</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Inside the boundary layer</td>
<td>40-50</td>
<td>50-60</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Minimum spacing on blade surface</td>
<td>4 x 10&lt;sup&gt;3&lt;/sup&gt;</td>
<td>4 x 10&lt;sup&gt;3&lt;/sup&gt;</td>
<td></td>
</tr>
<tr>
<td>Resolution (viscous wall units)</td>
<td>Flow direction</td>
<td>Δξ&lt;sup&gt;+&lt;/sup&gt;</td>
<td>Upstream of the separation point</td>
<td>&lt;11.0</td>
</tr>
<tr>
<td></td>
<td>Downstream of the separation point</td>
<td>&lt;7.8</td>
<td>&lt;2.75</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Spanwise direction</td>
<td>Δζ&lt;sup&gt;+&lt;/sup&gt;</td>
<td>&lt;6.41</td>
<td>&lt;5.10</td>
</tr>
<tr>
<td></td>
<td>Minimum spacing on the surface</td>
<td>Δη&lt;sup&gt;+&lt;/sup&gt;</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td></td>
<td>Inside the boundary layer</td>
<td>4.86</td>
<td>&lt;3.0</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Inlet and outlet boundary locations</td>
<td>Inlet boundary from blade L.E.</td>
<td>-50.4% C&lt;sub&gt;ax&lt;/sub&gt;</td>
<td>-37.7% C&lt;sub&gt;ax&lt;/sub&gt;</td>
</tr>
<tr>
<td></td>
<td>Outlet boundary from blade T.E.</td>
<td>118.9% C&lt;sub&gt;ax&lt;/sub&gt;</td>
<td>91.2% C&lt;sub&gt;ax&lt;/sub&gt;</td>
<td></td>
</tr>
</tbody>
</table>

Figure 5 Computational grid for S-25 solidity case, where every 10-th line is shown

3.3 Boundary Conditions

On the inlet boundary, all flow quantities, except inlet velocity magnitude, were specified on the basis of the experimental data for Re<sub>ax</sub> = 5.7 x 10<sup>4</sup>, while the mass flow rate was adjusted on the
outlet boundary to meet the equation of continuity. As mentioned above, the flow solver is for compressible flow analysis, therefore, it is very difficult for the solver to deal with completely the same flow conditions as the experiment because of the very low inlet velocity encountered in the experiment, making the problem stiff. To avoid this difficulty, the inlet velocity in the present calculation was set to be almost 10 times the actual velocity, while all geometries were accordingly scaled down by a factor of 10 so as to keep the Reynolds number unchanged. Non-slip condition was applied on the blade surface and the periodic boundary conditions were imposed on the rest of the boundaries except inlet and outlet boundaries. Implementation of each of the boundary conditions was made by use of fictitious cells attached to the boundaries. Besides, non-dimensional time step, normalized with the inlet sonic speed and the axial chord length, was 0.001. During the time-accurate flow analysis, Courant number remained around 3.

The present study dealt with two cases of inlet turbulence based on the experiment with and without the turbulence grid. To feed velocity fluctuation due to the turbulence into the flow field from the inlet as time-varying boundary condition. The procedure how to specify the inlet turbulence is given in the following.

3.4 Representation of Inlet Turbulence

3.4.1 SNGR Method

In order to mimic realistic inlet turbulent flow field, the present study first created homogeneous isotropic turbulence field by use of SNGR (Stochastic Noise Generation and Radiation) method [16] [17], which was then applied to the computational domain on a basis of Taylor’s frozen hypothesis.

According to the SNGR method, a turbulent velocity vector can be expressed by using random Fourier decomposition as follows:

\[ \mathbf{u}_n(x) = 2 \sum_{s=1}^{S} \tilde{u}_n \cos(\mathbf{k}_n \cdot x + \psi_s) \mathbf{k}'_n, \]  

where

\[ \tilde{u}_n = \sqrt{E(k_n) \Delta k_n}, \]

\[ \mathbf{k}_n = \{k_n,k_n,k_n\} = \{\cos \varphi_n \sin \theta_n, \sin \varphi_n \sin \theta_n, \cos \theta_n\}, \]  

\[ \mathbf{k}'_n = \{k'_n,k'_n,k'_n\} = \frac{1}{\sqrt{\Delta k_n}} \{k_2 - k_1, k_3 - k_1, k_3 - k_2\}, \]

and \( E(k_n) \) is turbulent energy spectrum, \( \psi_s \) is a phase lag. Two angles \( \varphi_n \) and \( \theta_n \) are azimuth angle and the supplement of elevation angle of the wave number vector \( \mathbf{k}_n \), respectively. The unit vector \( \mathbf{k}'_n \) for \( n \)-th mode was determined to satisfy the equation of continuity for incompressible flow, i.e., \( \mathbf{k}_n \cdot \mathbf{k}'_n = 0 \). The angular relationship of the vector \( \mathbf{k}_n \) in the wave number coordinate system, in conjunction with the relative position of vector \( \mathbf{k}'_n \) with respect to \( \mathbf{k}_n \) are depicted in Figure 6. All angles appearing in the above were random variables that were determined by use of probability density functions as shown in Table 3.

Assuming the isotropic turbulence for the inlet flow field, \( E(k_n) \) can be expressed using von Karman-Pao turbulent energy spectrum as follows:

\[ E(k) = \alpha \frac{2\tilde{\kappa}}{k_e} \frac{k/k_e}{[1 + (k/k_e)^2]^{5/2}} \exp \left[ -\rho_n \left| \psi - \rho_n \right| \right], \]

\[ \tilde{\kappa} = \int_0^\infty E(k) dk, \]

where \( \alpha \) is Kolmogorov constant (=1.45276), \( \tilde{\kappa} \) is total turbulence kinetic energy, \( k_e \) is Kolmogorov wave number. The parameter \( k_e \) determines the shape of the energy spectrum and is given by use of turbulence length scale \( L \) as follows;

\[ k_e = \frac{2\pi}{L}. \]  

Figure 6 Schematic of wave number vector \( \mathbf{k}_n \) and unit vector \( \mathbf{k}'_n \)

Table 3 Probability functions used to create random variables

| Prob(\( \varphi_n \)) = 1/2\( \pi \) | 0 \( \leq \varphi_n \leq 2\pi \) |
| Prob(\( \psi_s \)) = 1/2\( \pi \) | 0 \( \leq \psi_s \leq 2\pi \) |
| Prob(\( \theta_n \)) = 1/2 \( \sin \theta_n \) | 0 \( \leq \theta_n \leq \pi \) |
| Prob(\( \alpha_n \)) = 1/2\( \pi \) | 0 \( \leq \alpha_n \leq 2\pi \) |

3.4.2 Implementation of Turbulent Flow Field

Figure 7 illustrates the procedure how to apply the homogeneous turbulent velocity vector field to the computational domain to simulate inlet turbulence of any arbitrary intensity. The turbulence vectors were first created by the SGNR method inside the turbulent flow domain as shown in Figure 7. In this case, the total turbulence kinetic energy \( \tilde{\kappa} \) was given by the following expression,

\[ \tilde{\kappa} = \frac{3}{2} (\tilde{u}_n, \tilde{u}_n)^3, \]

where \( \tilde{u}_n \) was inlet turbulence intensity. The turbulent flow domain extended over one blade pitch and 10% axial chord length in the tangential and spanwise directions, respectively. Since the turbulent flow field given by SGNR did not have time-varying characteristics, a special treatment based on the idea of Taylor’s frozen hypothesis was employed here to change the one of the space coordinates into time coordinate. Once the homogeneous turbulent flow field was established, a plane was then selected inside the turbulent flow domain. This plane moved inside the domain towards the direction indicated in Figure 7 by \( \tilde{u}_n \Delta t \) for every computational time step. Then the turbulence velocity vectors on this moving plane cut were taken out and fed to the inlet boundary of the computational domain including the cascade as time-varying turbulence. After the moving plane reached one end of the turbulent flow domain, the sweeping was repeated from the other end. In this sense the turbulence velocity data created by the present approach were inherently featured with timewise cyclicity and not complete random data.

It is important here to make a comment on the merge and decay of the SNGR-based artificial turbulence initiating from the
inlet boundary, as well as on the procedure to cope with it in order to obtain a realistic flow condition. As clearly seen in Figure 7, in which the intensity of the turbulence was expressed in terms of the magnitude of vorticity, the inlet turbulence became weakened and seemingly merged into large vortical structures much faster than does in a real situation. This was because the numerical scheme and the density of the computational grid used in this calculation were not adequate enough to retain the whole energy spectrum of the initial turbulence. Since it was impossible to have any information a priori on change in the turbulence characteristics, some trial-and-error adjustment was exercised to assign a proper value of the inlet turbulence intensity $T_h$, so that the calculated turbulence intensity near the blade leading edge almost matched the measured value for the case using the turbulence grid (2.1%). The inlet turbulence intensity for the case with no turbulence grid was set to be zero only for the sake of simplicity, although non-zero turbulence intensity was observed in the experiment.

4. RESULTS
4.1 UHL Condition
4.1.1 Time-averaged characteristics

Figure 7 shows time-averaged distributions of static pressure coefficient $C_p$ on the blade surface calculated for the UHL condition with the turbulence grid and without the grid, in comparison with the experimental data. The static pressure coefficient was defined here as follows;

$$C_p(x) = \frac{P_{\infty} - p(x)}{\frac{1}{2} \rho U^2}$$

The LES results for no grid case exhibited plateau in $C_p$ extending from $x/C_a = 0.6$ to 0.75, followed by abrupt drop. The calculated $C_p$ almost agreed with the experimental data (open symbols), showing slight overestimation of the extent of the separation bubble, or moderate capability of predicting transitional behavior of the separation bubble leading to the reattachment. This discrepancy can be attributed to small but finite level of inlet FT observed in the experiment (see Figure 3). The LES results for with-grid condition attained good agreement with the measurement (solid symbols). In particular, the separated zone and the reattachment were adequately reproduced. There appeared a slight disagreement between the LES and the experiments near the leading edge suction side, implying that a slight change in incidence might have happened due to the inlet FT in the experiment.

Figure 7 Calculated static pressure distributions on the blade surface with and without turbulence grid, compared with the experimental data for S-25 solidity case

Figure 8 Time-averaged velocity profiles calculated for no-grid (upper) and with-grid (lower) conditions, compared with the experimental data for S-25 solidity case (solid line : LES / symbol : Exp)

Figures 8, 9 and 10 demonstrate the calculated time-mean boundary layer characteristics, i.e., velocity profiles, momentum thickness and shape factor distributions, respectively, all compared with the experiments. As observed in Figure 7, the LES results for with-grid condition yielded much better agreement with the measurements, while the LES calculations for no grid condition significantly overestimated the size of the separation bubble and only provided unsatisfactory results particularly in terms of velocity profiles around the separated region. Figure 9 depicts the momentum thicknesses. Since the change in increasing rate of momentum thickness can be a measure to spot the onset point of boundary layer transition, one may think that the transition for the with-grid condition initiated around $x/C_a = 0.7$. The LES
analysis made a reasonable prediction of the FT-affected momentum thickness distribution. It also seems interesting that the calculated momentum thickness for with-grid condition became smaller than that for no grid condition, indicating a possibility for loss reduction by use of moderate freestream turbulence intensity. Unfortunately, such a phenomenon was not observed in the experiment, i.e., the measured momentum thickness for the grid case was always larger than or at best nearly equal to that for the no grid case. Figure 10 shows the calculated and measured shape factor distributions. The LES analysis for the no grid case overestimated the shape factor, while the LES predicted the peak positions of the measured shape factors for both FT conditions.

Pitchwise distributions of stagnation pressure loss coefficient $Y_p$, defined by Eq. (11), are plotted in Figure 11.

$$Y_p(y) = \frac{P_{01} - P_{02}(y)}{\frac{1}{2} \rho U^2},$$

(11)

where the outlet stagnation pressure $P_{02}(y)$ was measured 15% $C_{ax}$ downstream of the blade trailing edge in the axial direction.

(It should be mentioned that the experimental data for the with-grid conditions is now being measured and not available at this moment). The LES analyses reasonably agreed with the experimental data for the no grid case, whereas the measured peak value of the loss coefficient was somewhat smaller than that of LES analysis. For UHL condition, it does not seem that the enhanced FT did not had a considerable impact on the calculated loss distribution. The peak of the loss in the blade wake slightly shifted to the pressure side due to the elevated FT, while the wake width slightly reduced keeping the maximum value of the loss almost the same.

4.1.2 Time-resolved flow fields

Figure 12 illustrates vorticity magnitude contours around the blade subjected to the FT. This figure clearly indicates that the inlet turbulence having isotropic vortical structure gradually experienced streamwise elongation as it was convected in the blade-to-blade passage. Quasi-periodic vortex shedding was also observed on the suction side of the blade; however, it is not clear from this figure whether the inlet FT might have any impact on the separation bubble and subsequent vortex shedding. Figure 13 provides a direct comparison between no grid and with-grid results, where the reattachment point was determined as the location where time-averaged wall shear stress changed its value from negative to positive. This comparison indicates that the FT severely affected the separation bubble so that earlier reattachment occurred and the vortex shedding became rather periodic, compared to the calculation for the no grid condition.

It also seems very useful to see the inception of the interaction between the incident FT and the front portion of the blade, especially in relation to FT-induced bypass transition. Figure 14 shows flow visualization near the blade leading edge using iso-value surface of the second invariant of the strain-rate tensor, so-called Q value, colored by non-dimensional helicity $H$, where $H$ was defined as

$$H = \frac{\mathbf{\omega} \cdot \mathbf{\omega}}{\left| \mathbf{\omega} \right|},$$

(12)

Note that $H$ is a measure not only to extract streamwise vortical structure from the flow field but also to detect its rotation direction. Inspection on the vortex-like structures with attention to their colors revealed that several elongated vortices, some of which
appeared to be counter-rotating each other, emerged on the suction surface near the leading edge after the inlet turbulence interacted with the blade leading edge. These vortical structures gradually decayed as they were swept downstream, seemingly leaving only slight traces behind. This means that no strong evidence to FT-induced bypass transition was found in the present analyses.

![Figure 12](image12.png)

Figure 12 A snapshot of interaction between the inlet turbulence and the blade, visualized by means of vorticity magnitude

![Figure 13](image13.png)

Figure 13 Snapshots of rolling up of the shear layers of separation bubbles on the suction side of the blade, visualized by means of spanwise vorticity (left: no grid / right: with grid)

4.2 HL Condition

4.2.1 Time-averaged characteristics

Figure 15 shows time-averaged distributions of static pressure coefficient calculated for the HL conditions, with the experimental data. It is clear that the aerodynamic load for the HL condition was reduced in comparison with that for the UHL condition. The calculated results for no grid case became flat on the suction side from $x/C_{ax} = 0.6$ to 0.83, thereafter dropped shortly. The agreement between the calculated values and the experimental data was fairly good except for the separated zone, which started to reattach from $x/C_{ax} = 0.8$ in the experiment. The elevated FT affected the separation bubble so that the extent of the separation bubble became smaller than that of no grid case. It appears that the effect of the enhanced FT on the separation bubble was rather noticeable for the HL condition in comparison with that for the UHL condition. The LES analysis also supported this finding.

![Figure 14](image14.png)

Figure 14 Q-value-based visualization of time-resolved flow field over the suction surface near the leading edge for S-25 solidity case, where iso-value surfaces with $Q=30$ are extracted

![Figure 15](image15.png)

Figure 15 Calculated static pressure distributions on the blade surface with and without turbulence grid, compared with the experimental data for S-15 solidity case
Time-averaged velocity profiles for the S-15 cases are shown in Figure 16. It is clear that the LES analyses made rather poor estimations of the velocity profiles around the separation bubble even for the elevated FT case, which was somewhat in contrast to Figure 8. The size of the separation bubble was overestimated, especially the height of the separation bubble was not properly predicted. This was probably because of inappropriateness of the grid system used for S-15 case, as well as the introduced FT. Another attempt using much larger number of grid points is now being scheduled.

The calculated boundary layer integral parameters are shown in Figures 17 and 18, along with the experimental data. Although quantitative discrepancies are confirmed between the LES analyses and the measurements, the calculations seem to have succeeded in predicting the time-mean characteristics of the separation bubble, such as the peak value position or the response to inlet FT. As mentioned above, somewhat favorable effect, in other words, separation bubble suppressing effect of the FT was observed through the measurements in a relatively clear manner in Figure 18. Time-averaged stagnation pressure loss profiles shown in Figure 19 provide an evidence for this favorable effect due to the inlet FT in the HL condition, however, this is not the case for the UHL condition.

5. CONCLUSIONS

This study conducted detailed hot-wire probe measurements of the boundary layer on the Ultra-High lift and High Lift low-pressure turbine airfoils. Large Eddy Simulation was also carried out using the experimental data and information. Inlet freestream turbulence was created by SGNR method and utilized in LES analyses as inlet boundary condition. The main focus of the study was on the investigation of the effects of freestream turbulence on the behavior of the separation bubble. Important
findings through this study are as follows;

1. The flow solver, which was developed in this study on the basis of LES approach, successfully resolved flow structures in detail and properly predicted the flow field subjected to the freestream turbulence, in terms of static pressure on the blade surface or velocity profile. However, it has also turned out that further improvement is strongly needed for the numerical method and grid system used in this study.

2. Significant effects of the inlet freestream turbulence were identified not only in the experiments but also in the calculations in terms of the size of the separation bubble, and vortex shedding process.

3. Few clues in connection with FT-induced bypass transition were found in this study. This was partially due to improprieness of the methodology for creating a realistic turbulence in this study.

REFERENCES


